



GE FANUC 21 CONCEPT 55 MILL TEACHER GUIDE

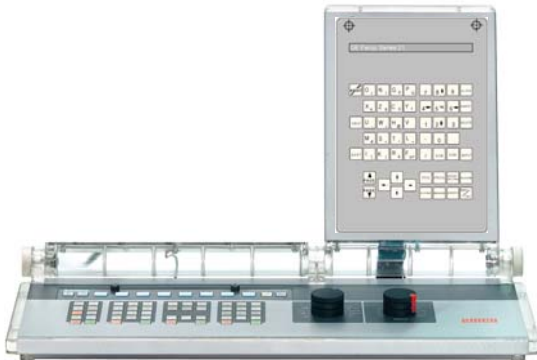
11/1/07 Version 2
Made by EMCO
Authored by Chad Hawk

Training Index

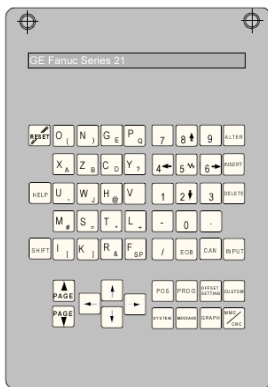
Control Keyboard.....	Pg 1
• Fanuc 21 Control	
• Machine Control	
Fanuc 21 Screen	Pg 2
Fanuc 21 Keys	Pg 3
• Data Input Keys	
• Change Keys	
Cursor & Page Keys	Pg 4
• Function Keys (Display Keys)	
• Soft Key Module	
Machine Keys	Pg 5
• Machine Function Keys	
Direction Keys	Pg 6
• Spindle Override Keys	
• Accessory Functions	
Mode Dial	Pg 7
• Feed Override Dial	
Referencing the Machine	Pg 8
Work Shift Description (Picture)	Pg 9
Work Shift (How to do Z Work Shift)	Pg 10
Work Shift (How to do X Work Shift)	Pg 12
• Manually starting the Spindle	
Work Shift (How to do Y Work Shift)	Pg 13
Tool Offset Description (Picture)	Pg 15
Tool Offset (How to do Tool Offsets)	Pg 16

Program Training & Tool listing.....	Pg 17
Inserting a New Program	Pg 18
• Calling a Existing Program up	
• Insert a word	
• Insert a End of Block	
Delete a Program	Pg 19
• Delete all Programs	
• Delete a word	
• Delete a Block	
Cancel word	Pg 20
• Alter a word	
• Search for number Block	
• Search for word	
G Codes	Pg 21
M Codes	Pg 22
• Used Addresses	
Program 1 (Contour Out Side)	Pg 24
2D simulation (Setup)	Pg 25
Input & Output the Programs & offsets thru the Fanuc Software.....	Pg 27
Running a Program	Pg 28
Program 2 (C & R)	Pg 29
Program 3 (Drilling)	Pg 30
Program 4 (R's)....	Pg 31
Program 5 (I&J)	Pg 32
I & J Test.....	Pg 33
Program 6 (Main Program) using M98 (sub programs)	Pg 34
Program 7 (Main for Pocket Milling)	Pg 35
Program 8 (Sub program for program 7)	Pg 36
Quiz	Pg 37
Appendix	

Machine Components



EMCO Control Keyboard

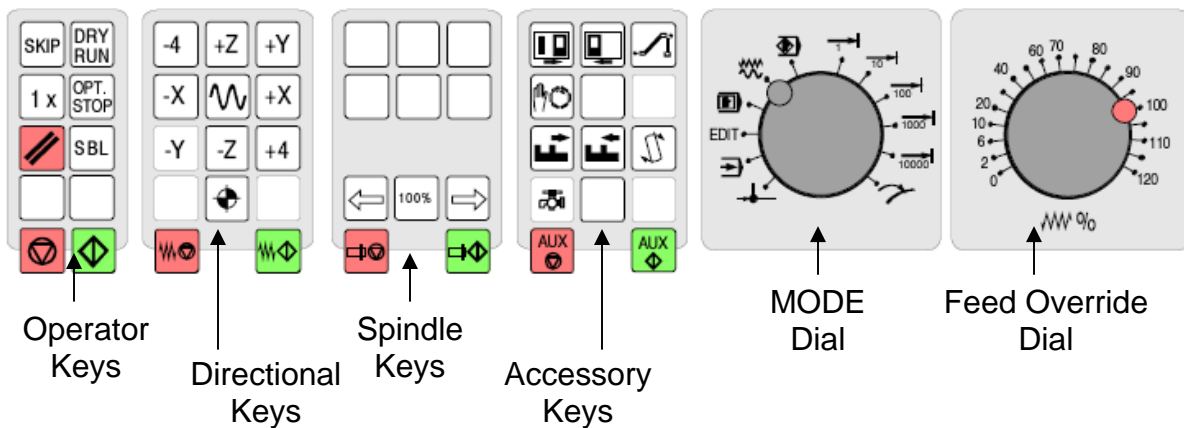


Fanuc 21 Keypad

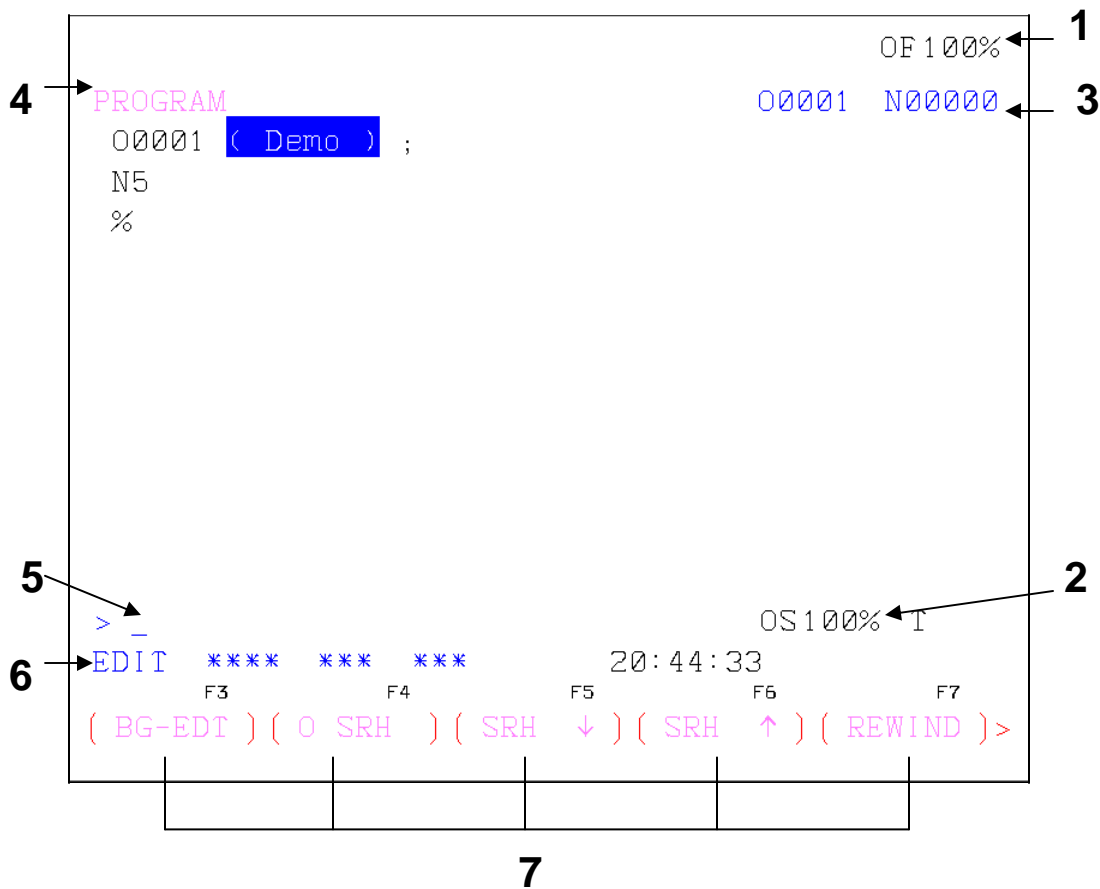


Fanuc 21 Soft Keys

EMCO Machine Control



The Fanuc 21 Screen



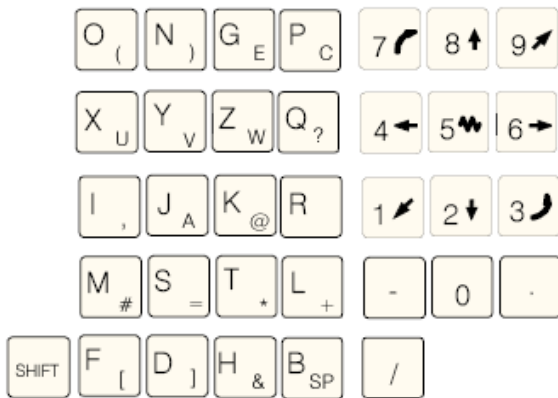
1. **Displays of Feed**
2. **Spindle Speed override**
3. **Display of Program and Number block**
4. **Display of active Screen**
5. **Entry line**
6. **Display of active Mode**
7. **Display of Soft key Functions**

FANUC 21 KEYS



RESET = cancels most alarms, resets program, interrupts programs

DATA INPUT KEYS



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

CHANGE KEYS



ALTER = alter word (replace word)



INSRT = insert word, create new program



DELETE = deletes word / block or programs



INPUT = input offsets / words or numbers



CAN = deletes entries in the address one by one



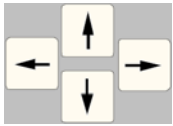
EOB = end of block

CURSOR & PAGE KEYS



Page Up = pages up in a program or additional screens

Page Down = pages down in a program or additional screens



Cursor up = moves up one line or to left in the screen

Cursor left = moves left in the screen

Cursor right = moves right in the screen

Cursor down = moves down one line or to the right in the screen, search function, and calls up programs

FUNCTION KEYS (DISPLAY KEYS)



POS = displays actual, relative, machine positions



PROG = displays program, library page



OFFSET/ SETTINGS = displays offset, work shifts pages



SYSTEM = displays parameters, diagnostic pages; use page up or down for optional pages

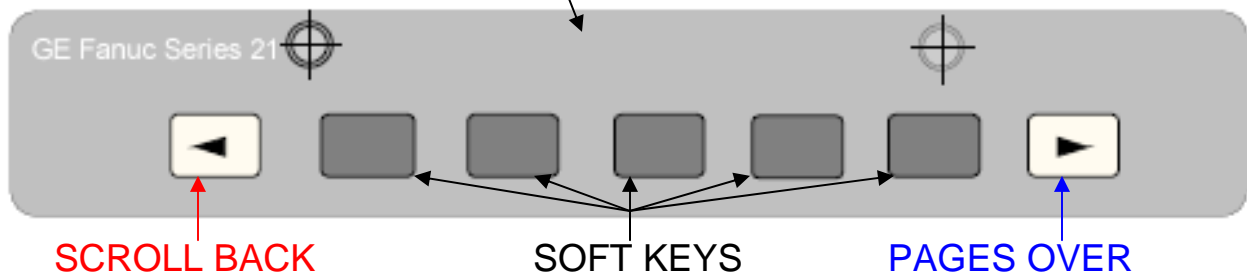


MESSAGE = displays operator & alarm messages



GRAPH = displays 2-d graph simulation

SOFT KEYS



EMCO MACHINE KEYS



= Operator Keys



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



= Press for test run without spindle on and rapids only (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



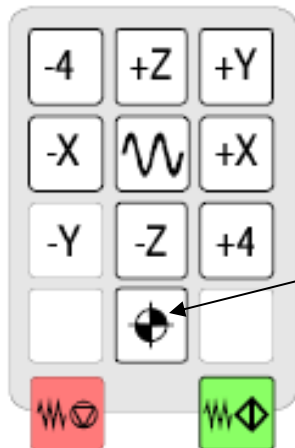
= (Cycle stop) program hold, feed hold



= (Cycle start) program start

Note: Skip, Dry Run, Optional Stop, and Single Block will show at the top of the screen when pressed. When pressed again they will disappear and turn off.

DIRECTION KEYS



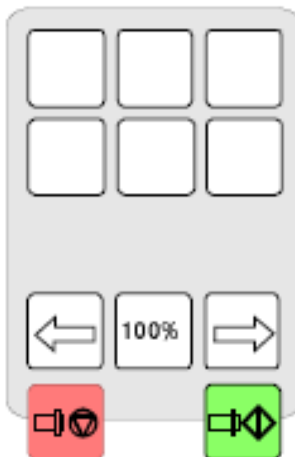
These keys control axis directional movements

+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)

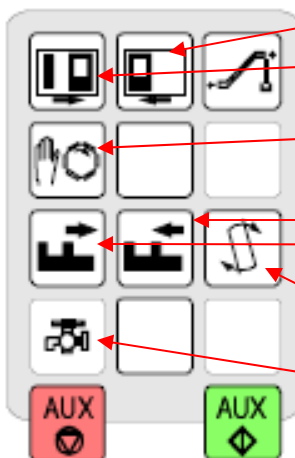
Arrow key pointing left decrease the Spindle speed (50% low)

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 releasing Tool in spindle (Door Open and hold the Tool)

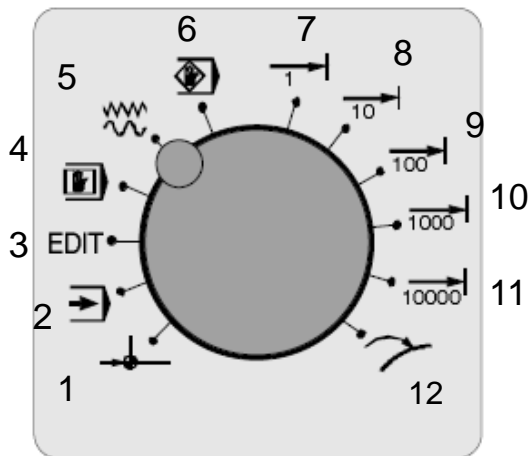
Press once coolant on

Press again coolant off

Press auxiliary drives on (Green)

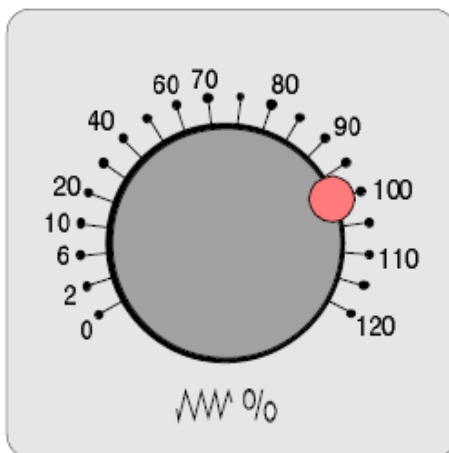
Press auxiliary drives off (Red)

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) SIEMEN MODE (Not used on Fanuc)
- (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
- (12) SIEMEN MODE (Not used on Fanuc)

FEED OVERRIDE DIAL

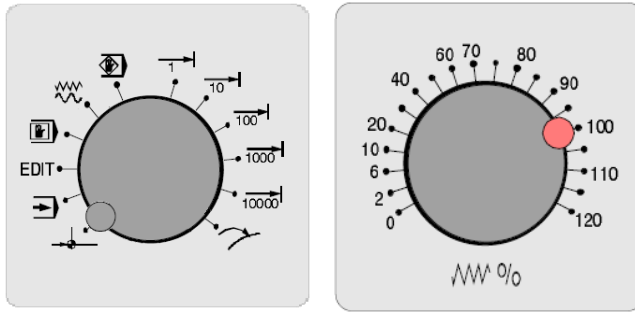


Controls feed for jogging in the X,Y, Z Axis.
Overrides from 0% to 120% of the
programmed feed rate or the rapid rate

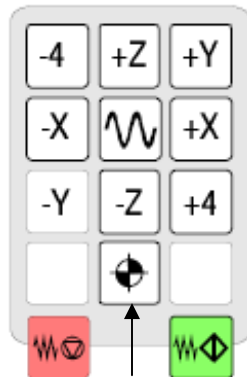
Turning the Machine On/Entering Fanuc Software

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 Reference all button below or follow steps 4-6
4. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
5. Press the X- (arrow pointing left) this references the X axis
6. Press the Y- (arrow pointing left) this references the Y axis

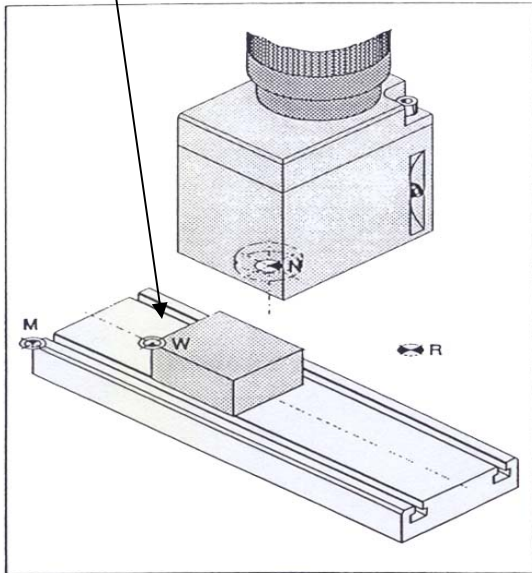


Reference all axis

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".

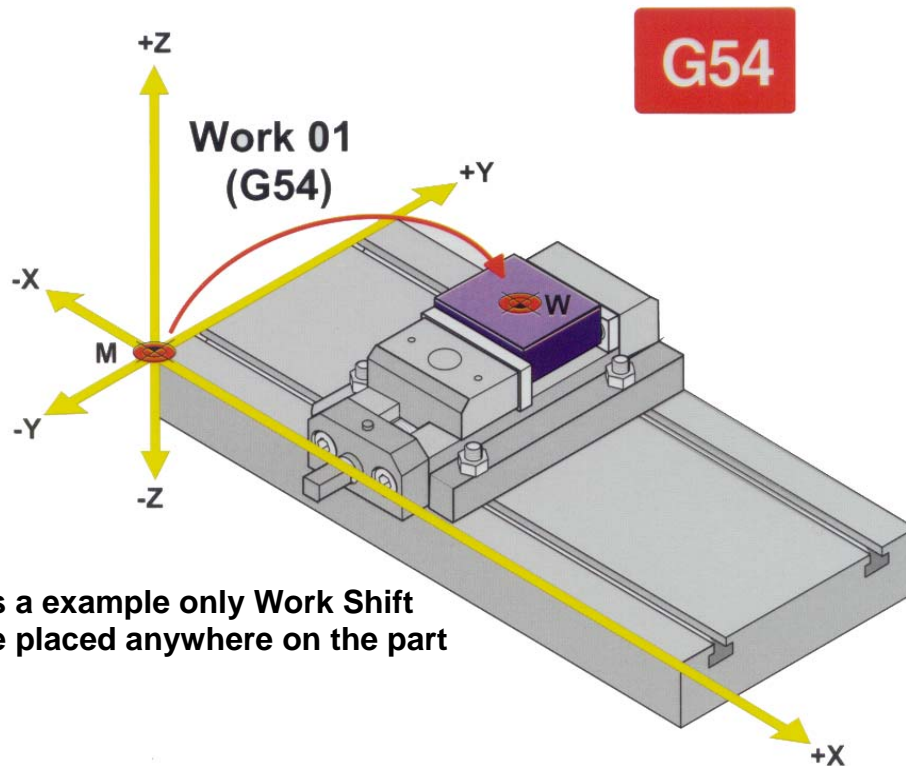
Required after every power failure.

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.



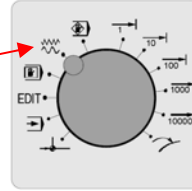
This is a example only Work Shift can be placed anywhere on the part

Work Shift:

Note: There are 2 main ways of doing this Education way or **Industry way**. Step 1 thru 3 is for the Education way; skip these steps if you are setting up **Industry way**; go to step 4.

1. To place a collet tool holder without tool in the spindle

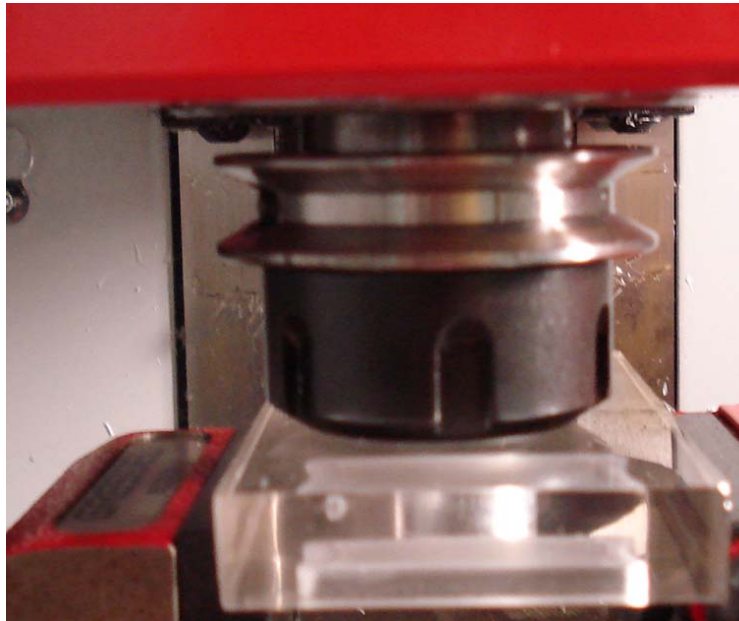
2. Rotate Mode Dial to Jog



3. Jog the 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 Collet Holder Cap and Work Piece



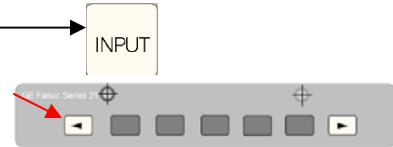
Note: Machine 0 for Z is the spindle nose touching the top of the machine bed.

4. Press the OFFSET/SETT button

- Press the W.SHIFT Soft key (Gray Button)

5. 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

- If there is a value then cursor to each one (Example 2) and type 0 and press the input button



- Press the left arrow on the soft keys

Note: Industry way skip Steps 6 thru 10

6. Press the OFFSET Soft key (Gray Button)

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

7. Press the left arrow on the soft keys



8. Press the W.SHFT Soft key (Gray Button)

9. Cursor down to 01 (G54) location so Z is highlighted (Example 2)

10. Press Input button






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

Example 1				Example 2			
OF100%				OF100%			
OFFSET			00001 N00000	WORK COORDINATES			00001 N00000
NO.	DATA	NO.	DATA	NO.	DATA	NO.	DATA
001	0.0000	009	0.0000	00 X	0.0000	02 X	0.0000
002	0.0000	010	0.0000	(EXT) Y	0.0000	(G55) Y	0.0000
003	0.0000	011	0.0000	Z	0.0000	Z	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	01 X	0.0000	03 X	0.0000
008	0.0000	016	0.0000	(G54) Y	0.0000	(G56) Y	0.0000
ACTUAL POSITION (RELATIVE)							
X	0.0000	Y	0.0000				
Z	0.0000						
OS100% I				OS100% I			
JOG F3 F4 F5 F6 F7 20:46:16				JOG F3 F4 F5 F6 F7 20:49:22			
(OFFSET) (SETING) (W.SHFT) () (OPRT)				(OFFSET) (SETING) (W.SHFT) () (OPRT)			

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

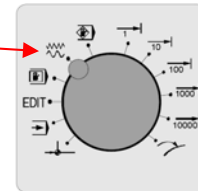
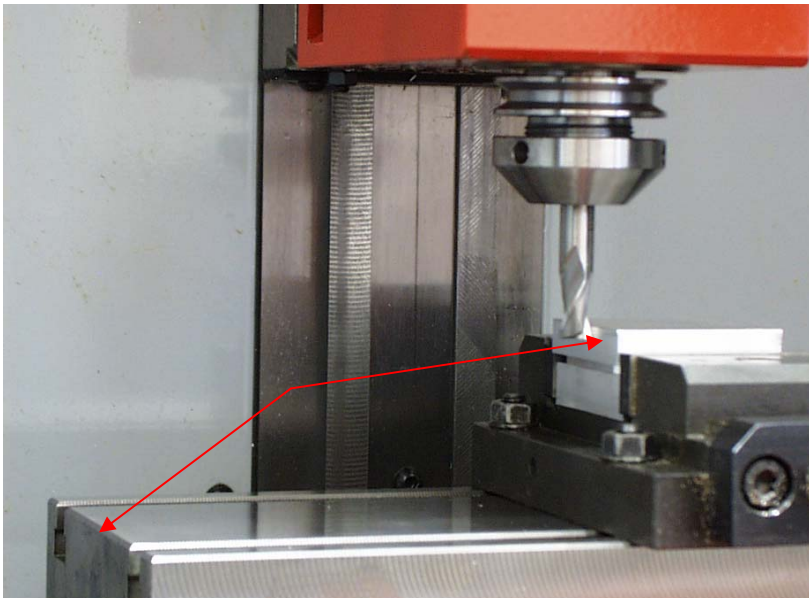
12. Place Endmill or Edge finder in the spindle

Optional: To scratch the material to set offsets. If you want to manually touch off continue to step 13.

- Move MODE Dial to MDI
- Press the PROGRAM display button  until top left of the screen shows **PROGRAM (MDI)**
- Type S1000 M3 then Press Input button 
S= Spindle Speed M3= Spindle on Clockwise
- Then press CYCLE START  (**Door must be closed**)

13. Rotate 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
SETTING

- If the top left of the screen doesn't show OFFSET then press the OFFSET Soft key (Gray button)
- Type the value from the Actual Position Relative X
- Press the left arrow on the soft keys



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

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

16. Press Input button

INPUT

F6
(+INPUT)

17. Type in the radius for the Tool and press the +input soft key

Note: (Version 14.12 software and up only) If old version add radius of tool and Step 16's value together, type in, and press input button

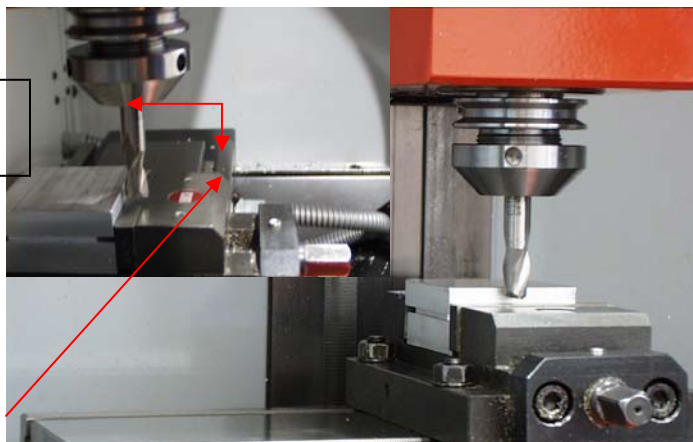
Example 1				Example 2			
OFFSET		OF 100%		WORK COORDINATES		OF 100%	
NO.	DATA	NO.	DATA	NO.	DATA	NO.	DATA
001	0.0000	009	0.0000	00 X	0.0000	02 X	0.0000
002	0.0000	010	0.0000	(EXT) Y	0.0000	(G55) Y	0.0000
003	0.0000	011	0.0000	Z	0.0000	Z	0.0000
004	0.0000	012	0.0000				
005	0.0000	013	0.0000				
006	0.0000	014	0.0000				
007	0.0000	015	0.0000				
008	0.0000	016	0.0000				
ACTUAL POSITION (RELATIVE)							
X	0.0000	Y	0.0000	01 X	0.0000	03 X	0.0000
Z	0.0000			(G54) Y	0.0000	(G56) Y	0.0000
				Z	0.0000	Z	0.0000
> JOG **** * 20:46:16 OS100% T				> JOG **** * 20:47:33 OS100% T			
{ OFFSET } { SETING } { W.SHFT } { } { (OPRT) }				{ OFFSET } { SETING } { W.SHFT } { } { (OPRT) }			

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

+Z

19. Jog the Tool to the Front of the Work Piece & touch using the Direction keys.

Side
View



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

20. If the screen is at OFFSET or WORK COORDINATES then go to the bullets if not then press the OFFSET/SETT button

OFFSET
SETTING

- If the top left of the screen doesn't show OFFSET then press the OFFSET Soft key (Gray button)
- Type the value from the Actual Position Relative Y

- Press the left arrow on the soft keys



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

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

23. Press Input button

INPUT

F6
(+INPUT)

24. Type in the radius for the Tool and press the +input soft key

Note: (Version 14.12 software and up only) If old version add radius of tool and Step 23's value together, type in, and press input button

Example 1

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		

00001 N00000

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 0.0000	(G56)	Y 0.0000
	Z 0.0000	Z	0.0000

00001 N00000

OS100% I

> _ JOG **** * 20:46:16

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

OS100% I

> _ JOG **** * 20:48:33

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

Example 2

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 0.0000	(G56)	Y 0.0000
	Z 0.0000	Z	0.0000

00001 N00000

OS100% I

> _ JOG **** * 20:48:33

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

OS100% I

> _ JOG **** * 20:46:16

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

Example 1

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		

00001 N00000

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 0.0000	(G56)	Y 0.0000
	Z 0.0000	Z	0.0000

00001 N00000

OS100% I

> _ JOG **** * 20:46:16

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

OS100% I

> _ JOG **** * 20:48:33

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

Example 1

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		

00001 N00000

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 0.0000	(G56)	Y 0.0000
	Z 0.0000	Z	0.0000

00001 N00000

OS100% I

> _ JOG **** * 20:46:16

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

OS100% I

> _ JOG **** * 20:48:33

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

Example 2

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 0.0000	(G56)	Y 0.0000
	Z 0.0000	Z	0.0000

00001 N00000

OS100% I

> _ JOG **** * 20:48:33

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

OS100% I

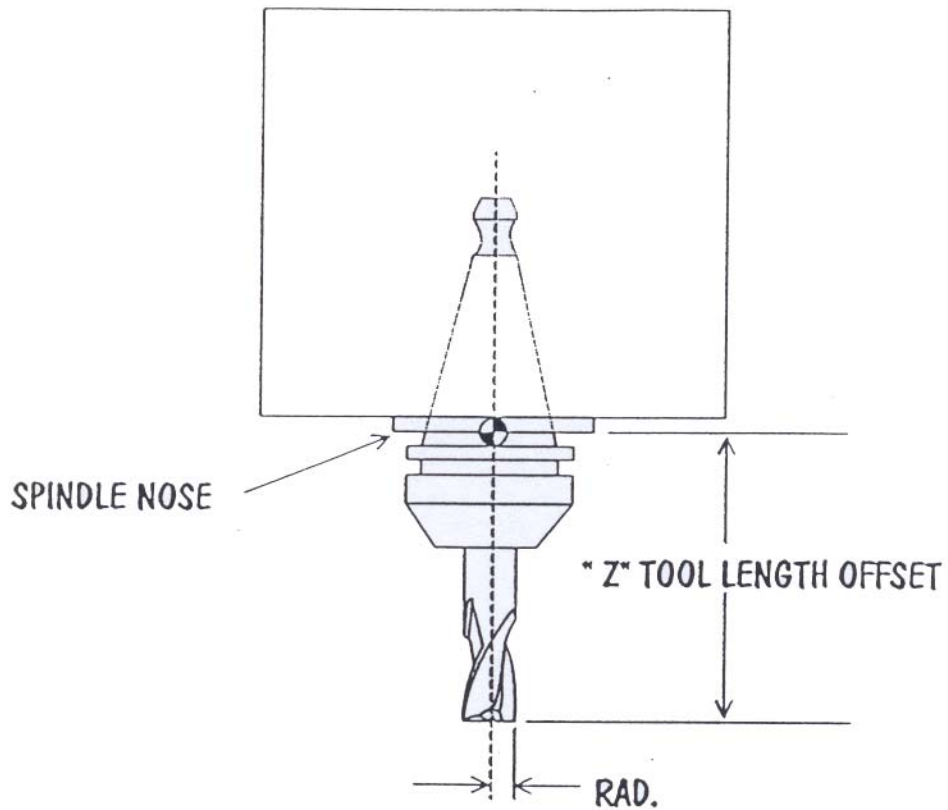
> _ JOG **** * 20:46:16

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

25. Jog the Tool up above the Work Piece using

+Z

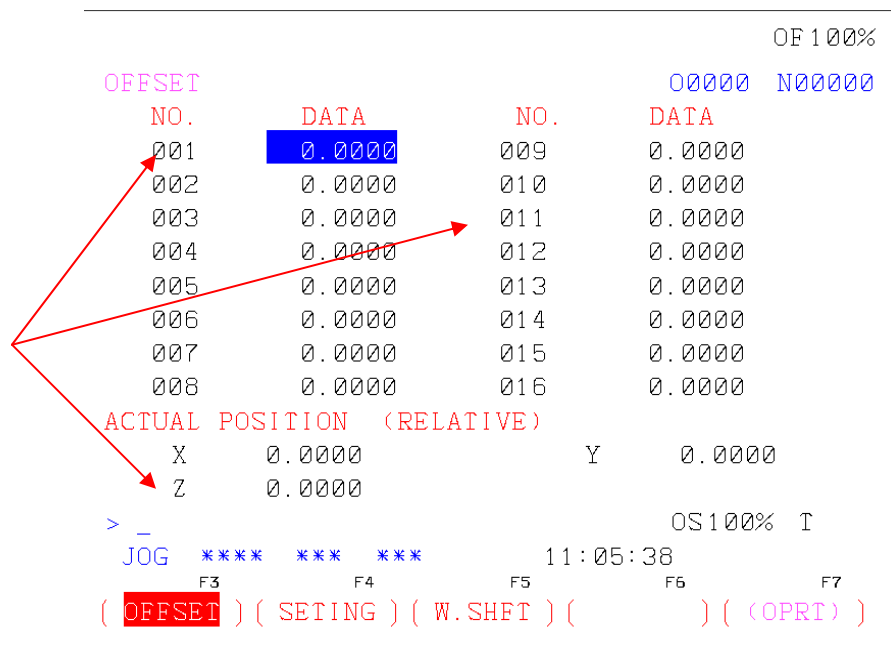
TOOL OFFSET



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
{ 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 + 010 = 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: **Industry way** will have a large value for all tools set and the tools are independent of each in relationship to the work piece

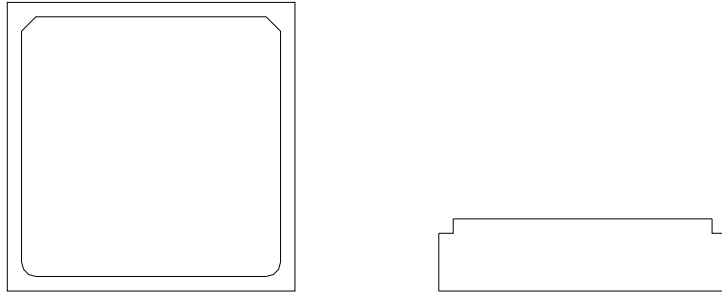


NOTE: When you use a T the H = Height

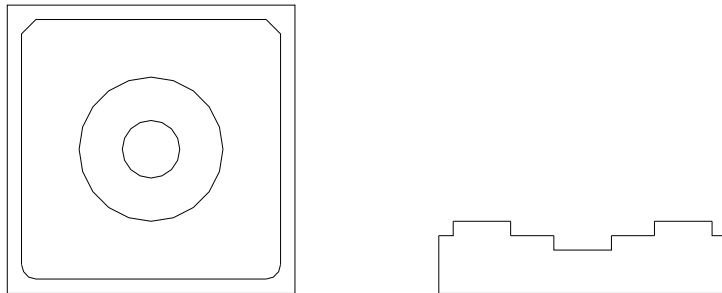
When you use a G41 or G42 the H = Radius

Program Training

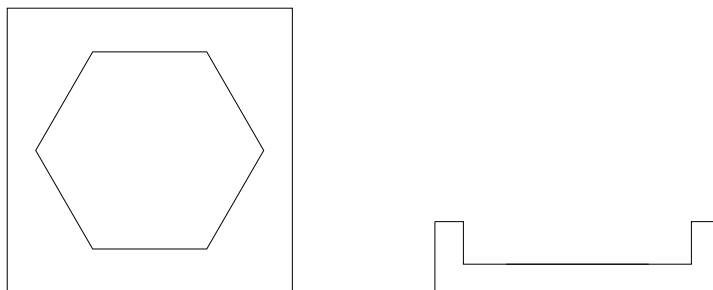
Program O0001



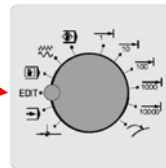
Program O0006



Program O0007



Change the Mode Dial to Edit
Press the Program Button
To do functions on pages 21 - 23



- **INSERT A NEW PROGRAM**

1. Press letter o then a program number between 1- 8999
2. Press insert button



Example: Q0001 OR Q1

- **CALL A EXISTING PROGRAM UP**

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



- **INSERT A WORD**

1. Press letter then number
2. Press insert button



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



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

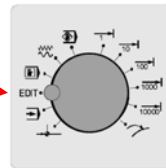
Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT

USE INPUT FOR ALL OTHER SCREENS AND MODES.

Change the Mode Dial to Edit
Press the Program Button
To do functions on pages 21 - 23

PROG



- **DELETE A PROGRAM**

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

DELETE

Example: O0001 OR O1

- **DELETE ALL PROGRAMS**

1. Press letter o plus the – & 9999
2. Press delete button

DELETE

Example: O – 9999

- **DELETE A WORD**

1. Highlight the Word
2. Press delete button

DELETE

- **DELETE A BLOCK OR LINE NUMBER**

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

DELETE

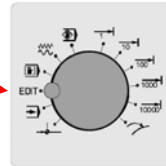
- **CANCEL MISTYPED WORD (Backspace)**

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

Change the Mode Dial to Edit
Press the Program Button
To do functions on pages 21 - 23



- **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

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

Survey commands M-CODES: Mostly used

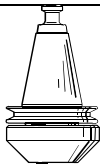

M00	Programmed stop, unconditional
M01	Programmed stop, conditional
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
M05	Spindle OFF
M06	Tool Index
M25	Open clamping vice
M26	Close clamping vice
M30	Main program end with new start of program
M71	Blow-off ON
M72	Blow-off OFF
M98	Subroutine call-up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

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

Used Addresses

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

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

F1Z 010	<u>Collet holder</u>	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	<u>Slot end mill, HSS</u>	

Program screen & Edit mode

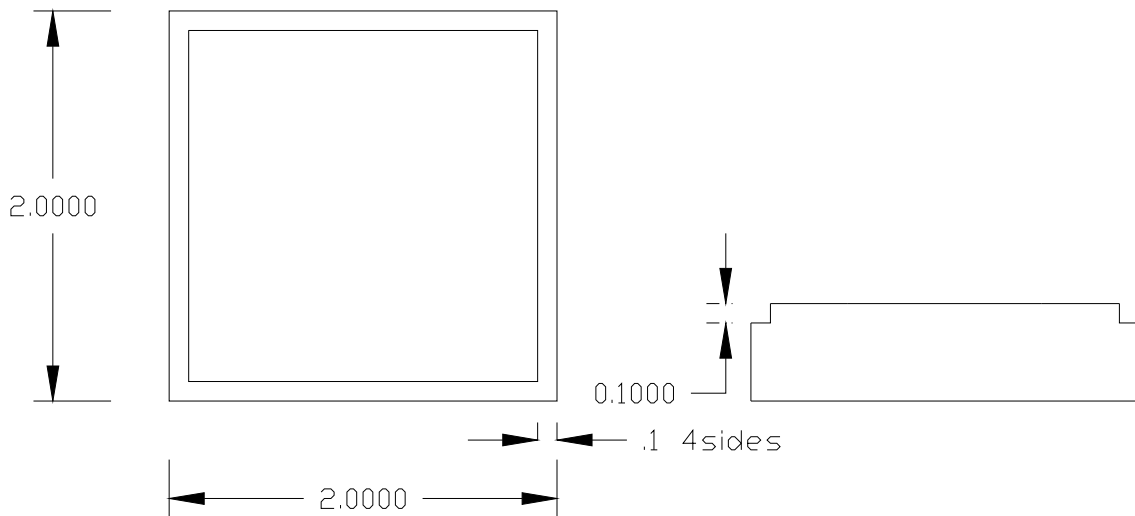
- To edit / change a program / insert new programs & input or output excising programs & offsets

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 M0 (3/8 or 10mm Endmill) Tool call out line

N25 S1800 M3

N30 G0 Z1

N35 X-1 Y1

N40 Z-.1

N45 G1 G41 H11 X.1 F7

N50 Y1.9

N55 X1.9

N60 Y.1

N65 X.1

N70 Y1

N75 G0 G40 X-1

N80 G91

N85 G28 Z0

N90 G90

N95 M30

**Default G code
not needed**

Work shift Call out

Spindle on CW

Safe move above part

Positioning X, Y for cutting

Positioning Z at depth

CRC on and moving to X

Position Top left corner

Position Top right corner

Position Bottom right corner

Position Bottom left corner

Back to start point Y

CRC off and start point X

incremental mode

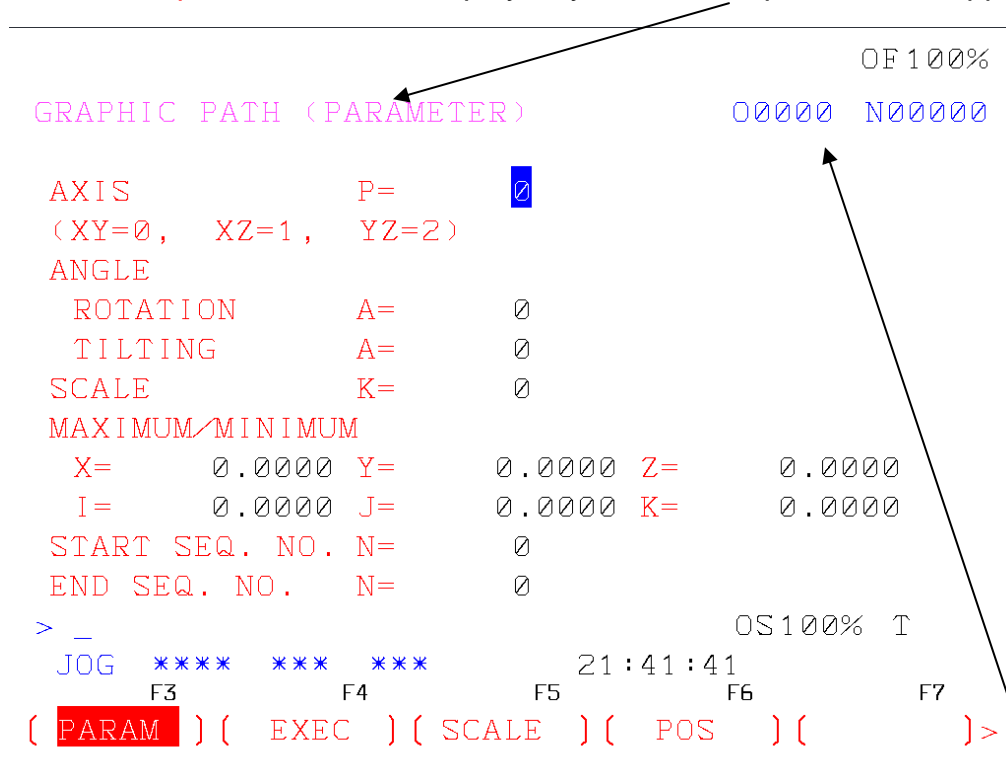
from position to Z home

absolute mode

End of Program

2D Simulation

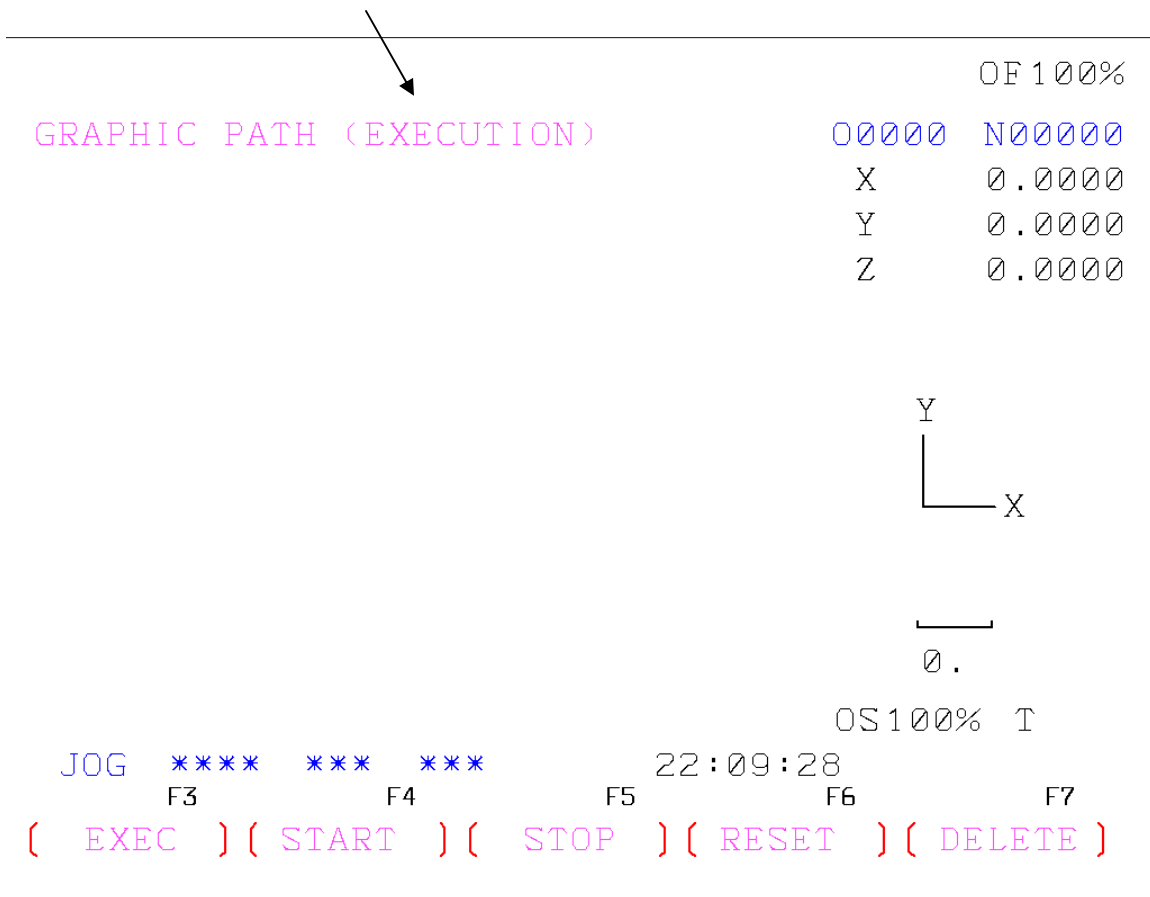
1. Press **Graph** button on the Display Keys for the Graph screen to appear



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



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)**

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

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

1. Press 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

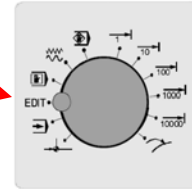
Running a Program

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

1. Rotate the Mode dial to Edit

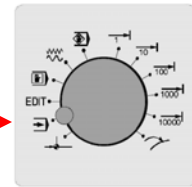
2. Press the Program button

PROG



3. Call up Program to be run / cut
(Example O1 for program 1)

4. Rotate the Mode dial to MEM



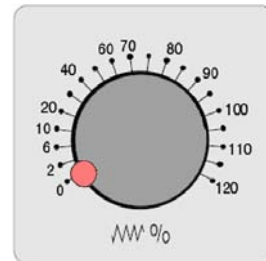
5. If screen is not in **PROGRAM CHECK** then press program button until this is at the top left of the screen

PROG

6. Press the Single Block button for the program to run one line at a time.

SBL

Note: Use one hand on the feed override dial slowly increasing it and the other pressing cycle start and close to the reset button



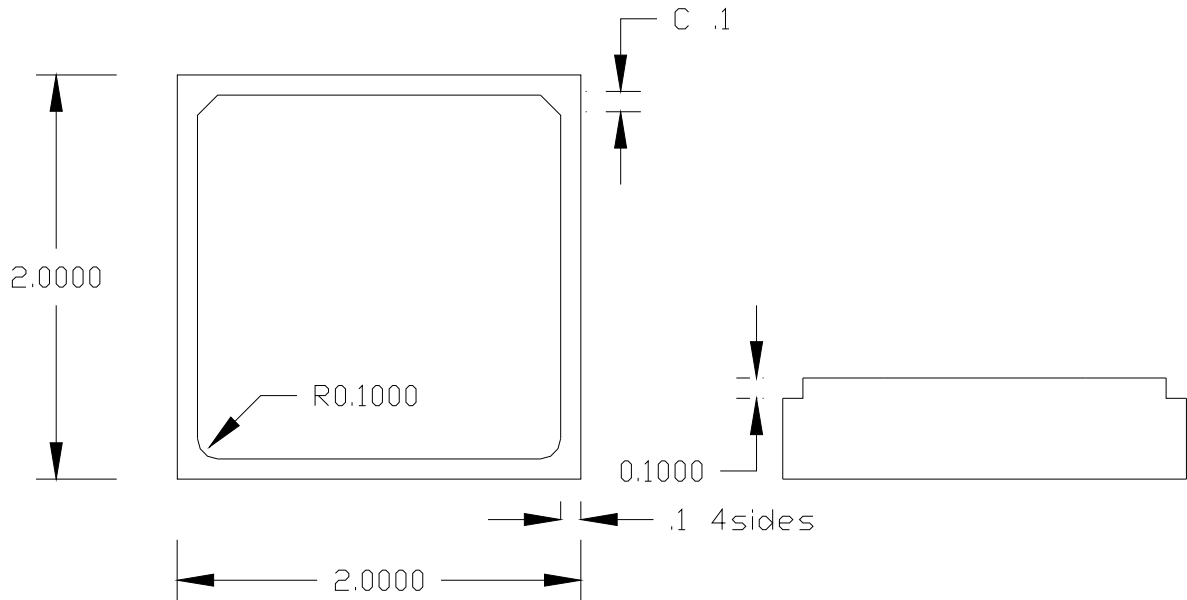
7. Press Cycle Start and continue

(Once the program have moved in the safe called out locations for X, Y, Z and looks right; you can take single block off and run the program)

8. Press Cycle Start one more time

(If there are more than one tool before the next tool use single block to check the offsets locations for the Z only then continue at step 8 again)

Program Q0002 (C & R)



O0002 (Demo 2 C/R) (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 10 mm end mill)

N25 S1800 M3

N30 G0 Z1

N35 X-1 Y1

N40 Z-.1

N45 G1 G41 H11 X.1 F7

N50 Y1.9 **C.1**

N55 X1.9 **C.1**

N60 Y.1 **R.1**

N65 X.1 **R.1**

N70 Y1

N75 G0 G40 X-1

N80 G91 G28 Z0

N85 G28 X0 Y0

N90 G90

N95 M30

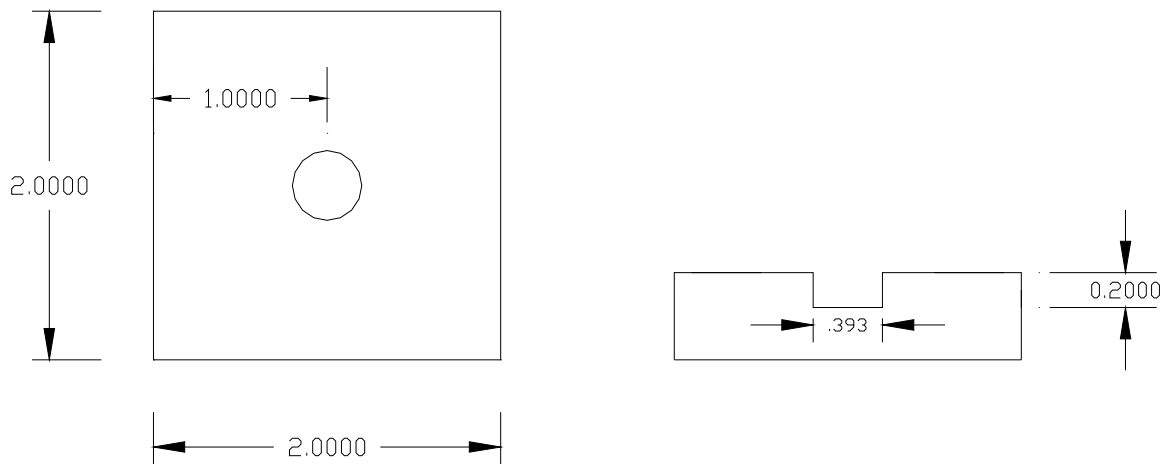
Automatic Chamfer 45 degrees

Automatic Chamfer 45 degrees

Automatic Radius 90 degrees

Automatic Radius 90 degrees

Program Q0003 (Deep Hole Drilling)



G83 X = Location of hole Y = location of hole

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

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

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

O0003 (Demo 3 Drill) (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 G91 G28 Z0

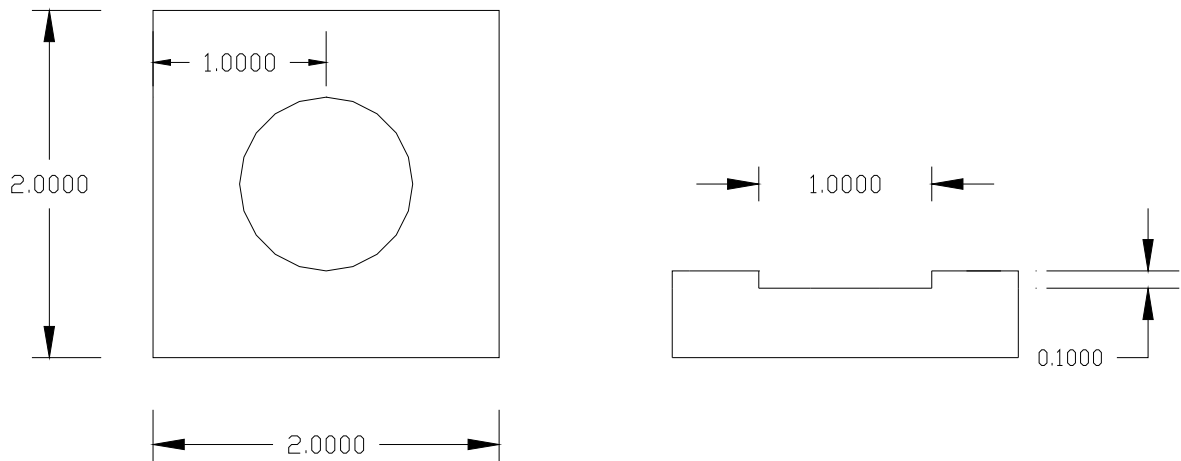
N50 G28 X0 Y0

N55 G90

N60 M30

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

Program Q0004 (R)



O0004 (Demo 3 R) (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

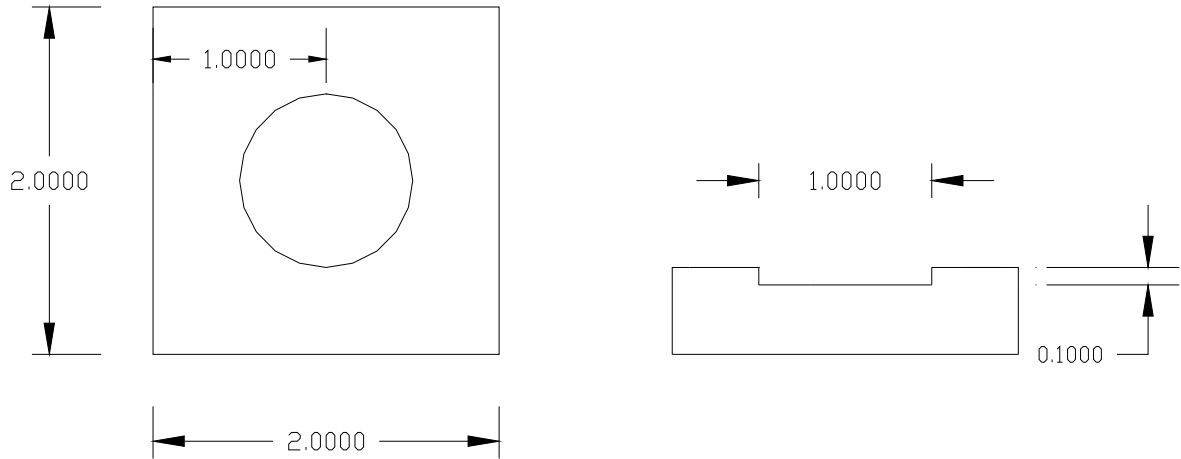
N65 G91 G28 Z0

N70 G28 X0 Y0

N75 G90

N80 M30

Program Q0005 (I & J)



O0005 (Demo 4 I/J) (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

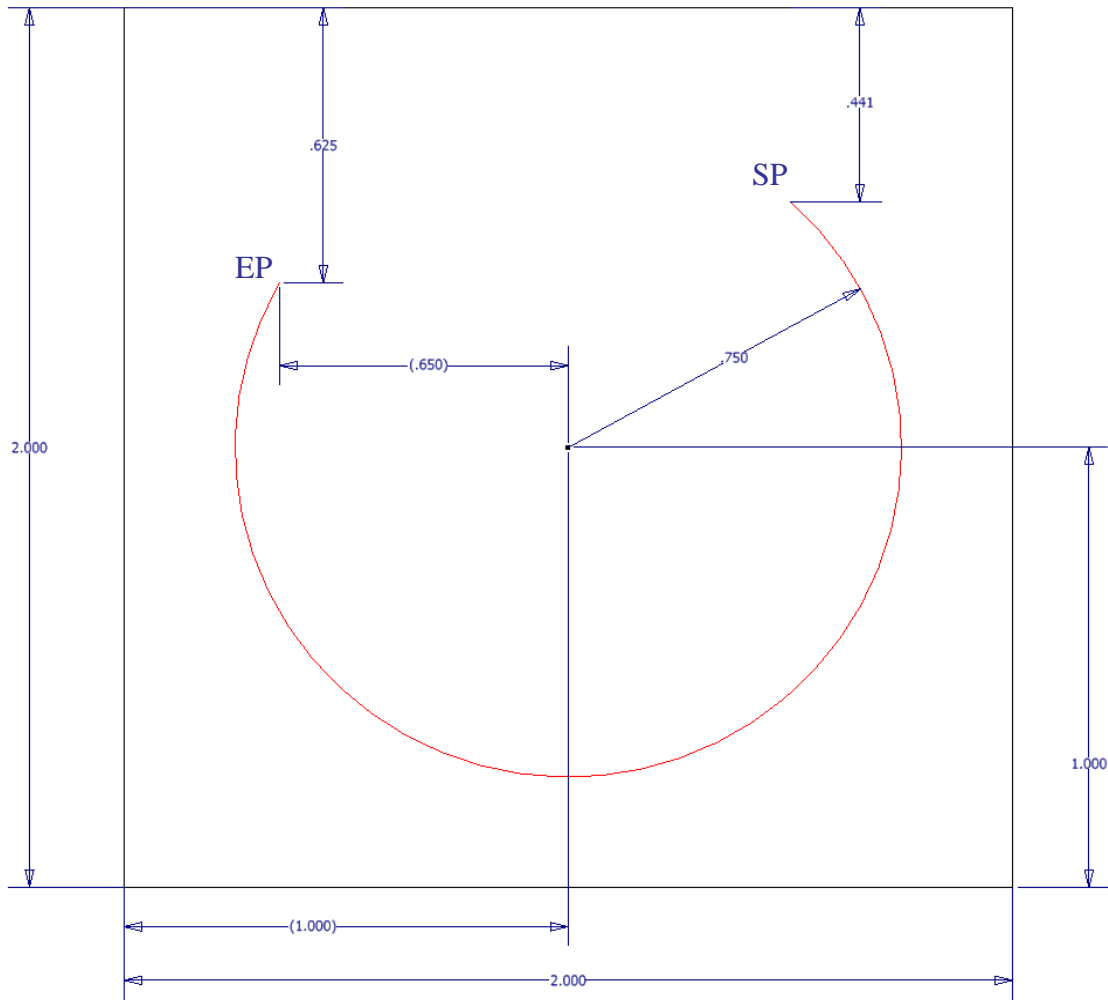
N60 G91 G28 Z0

N65 G28 X0 Y0

N70 G90

N75 M30

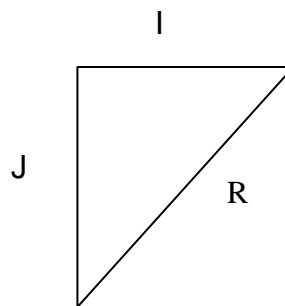
I and J Test



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

```

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



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

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

Degrees

Sally Can Tell Oscar Has A Hat On Always

SINE COSINE TANGENT

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

TEST FOR SUB PROGRAMS

O0006 (Tie Programs)

N5

N10 (Demo 2 C/R)

N15 (Demo 3 Drilling)

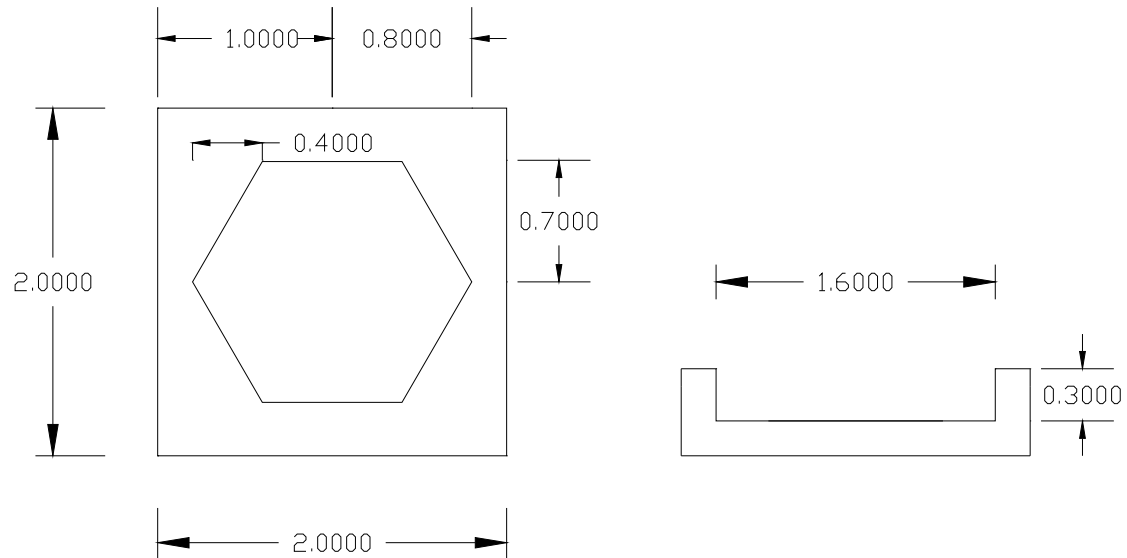
N20 (Demo 5 I & J)

N25 M30

Changing Item

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

Program O0007 (Pocket Milling) (Making a Cycle)



O0007 (Demo 7 Pocket) (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 P030008

N45 G0 G91 G28 Z0

N50 G28 X0 Y0

N55 M30

Program O0008 (Sub for program 7)

O0008 (Sub for Prog 7)

N5 G91

N10 G1 Z-.1 F2

N15 G90

N20 S1800

N25 G41 H11 X.4 Y1.35 F7

N30 X.2 Y1

N35 X.6 Y.3

N40 X1.4

N45 X1.8 Y1

N50 X1.4 Y1.7

N55 X.6

N60 X.2 Y1

N65 X.4

N70 X.8 Y.5

N75 X1.2

N80 X1.6 Y1

N85 X1.2 Y1.5

N90 X.8

N95 X.4 Y1

N100 G0 G40 X1

N105 M99

Shorter Program Test:

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

N5 G91

N10 G1 Z-.1 F2

N15 G90

N20 S1800

N25 G41 H11 X.4 Y1.35 F7

N30 X.2 Y1

N35 X.6 Y.3

N40 X1.4

N45 X1.8 Y1

N50 X1.4 Y1.7

N55 X.6

N60

N65

N70

N75

N80

N60

N65

N70

N75




N80

N85

Appendix

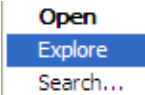
Changing Drive to USB Port

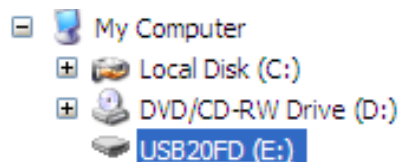
1. Close out the SW (software)

- Press  to allow you to exit
- Press  and  together to exit the Software

2. Make sure USB is plug into port

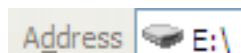
3. Open Explorer

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



4. Copy Drive directory

- Click on you USB drive
- At the top of the active screen or page in the Address copy or remember drive info

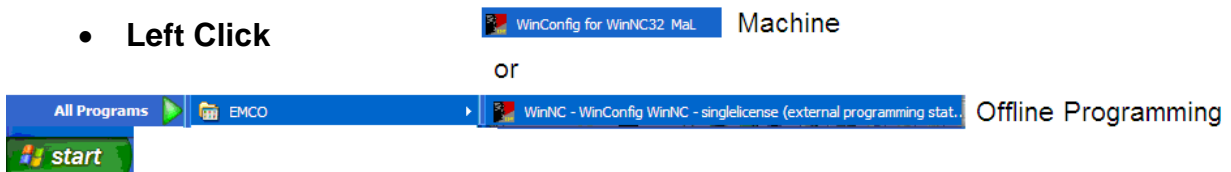


- Close the active screen or page using either Alt and F4 or  at top of the active screen





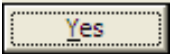
5. Setting up WinConfig

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


- Left Click



6. In Winconfig

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

7. Restart SW (software)

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