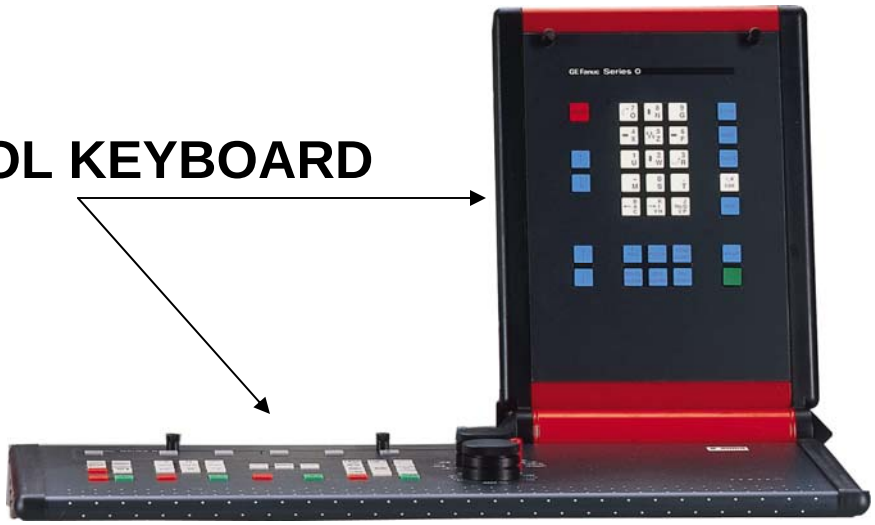
emco group

GE FANUC 21
CM 105 MILL
TRAINING GUIDE

4/29/08 Version 1
Made by EMCO
Authored by Chad Hawk

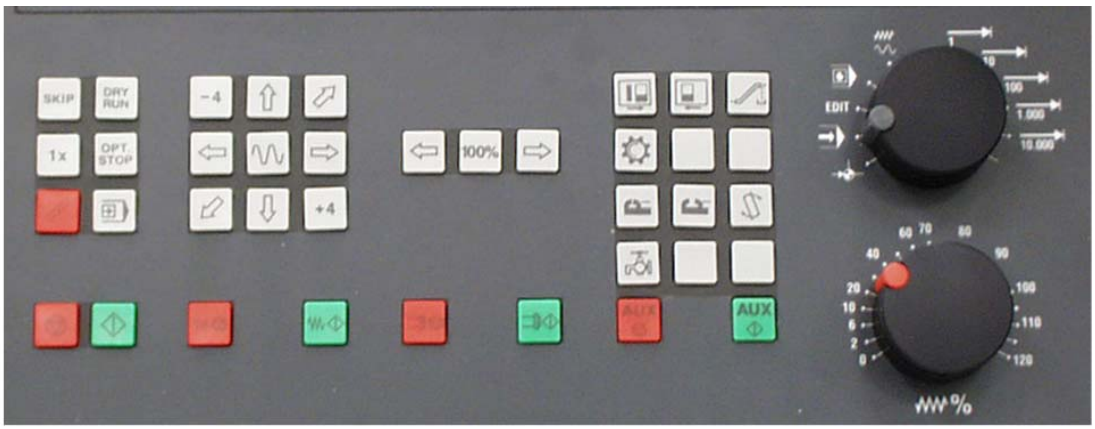
Training Index

CONTROL KEYBOARD

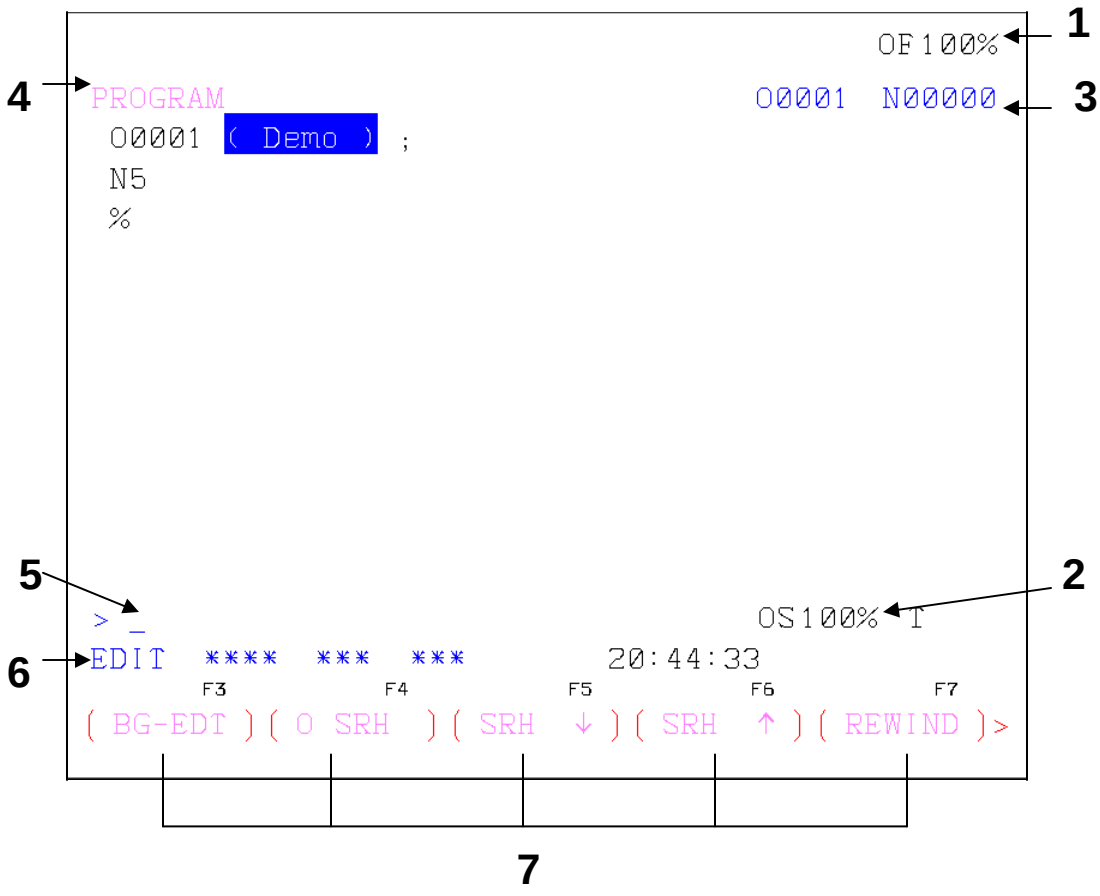


FANUC 21 CONTROL

MACHINE CONTROL



The Fanuc 21 Screen



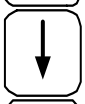
1. **Displays of Feed**
2. **Spindle Speed override**
3. **Display of Program and Number block**
4. **Display of active Screen**
5. **Entry line**
6. **Display of active Mode**
7. **Display of Soft key Functions**

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

POS = displays actual, relative, machine positions

PROG

PROG = displays program, library page

OFFSET SETT.

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

SYSTEM

SYSTEM = displays parameters, diagnostic pages

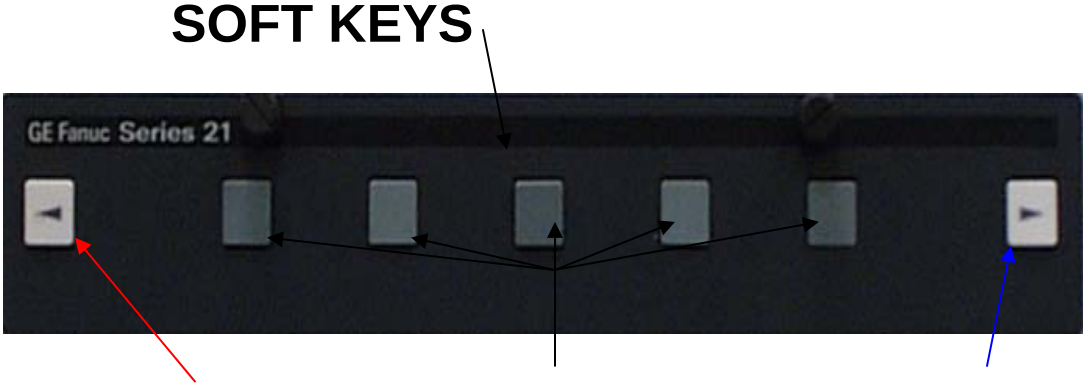
MESSAGE

MESSAGE = displays operator & alarm messages

GRAPH

GRAPH = displays 2–d graph simulation

SOFT KEYS



SCROLL BACK **SOFT KEYS** **PAGES OVER**

MACHINE KEYS

MACHINE FUNCTION KEYS

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

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

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

= (Optional stop) for programs with (m1)

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

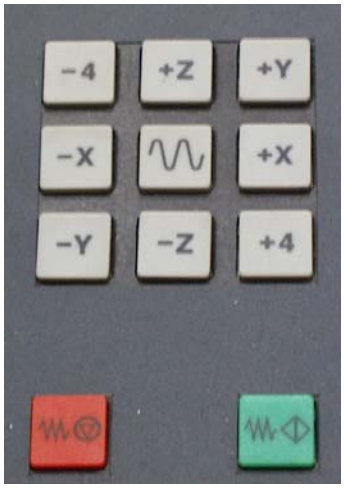
= (Single block) reads one block line at a time

= (Cycle stop) program hold, feed hold

= (Cycle start) program start

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

=(Mode Key) Automatic & Hand Mode
Hand Mode is for moving machine around with door open and works in conjunction with the (Agreement button)

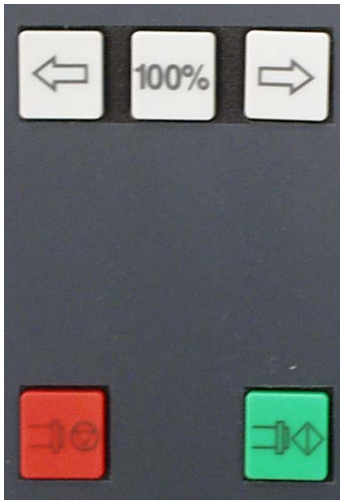


DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

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



SPINDLE OVERRIDE KEYS

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

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

100% key jumps speed to 100%

Spindle stop (Red) / Spindle start (Green)

Works all modes except EDIT & ZRN (Reference)

ACCESSORY FUNCTIONS

Arrow right door open
Arrow left door closed

Press for Rotary axis Indexing

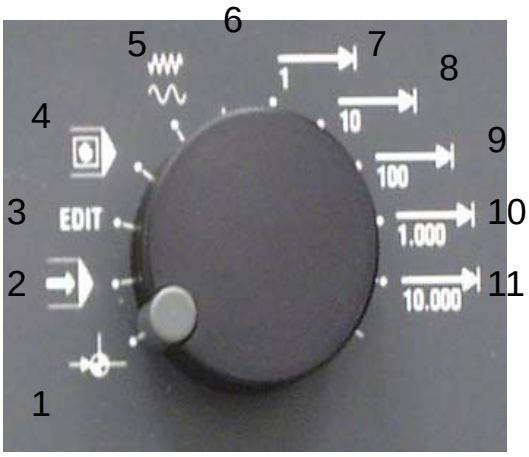
Press once vise closed
Press once vise open

Turret Index

Press once coolant on
Press again coolant off

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

MODE DIAL



(1) REF = Zero / Reference or Home mode

(2) MEM = Automatic mode for running a program

(3) EDIT = Edit mode for program changes or entering a new program

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

(5) JOG = Manual moving the axis in x y z

(6) STEPS = Incremental feed movements

(7) STEPS = .0001 or tenths

(8) STEPS = .001 or thousands

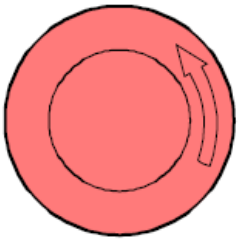
(9) STEPS = .010 or ten thousands

(10) STEPS = .100 or hundred thousands

(11) STEPS = .100 or hundred thousands

FEED OVERRIDE DIAL

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



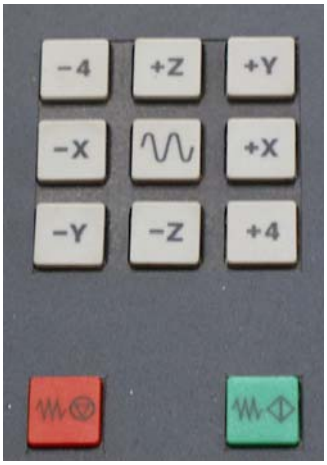
E Stop or Emergency Stop

Turning the Machine On/Entering Fanuc Software

Referencing the Machine

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

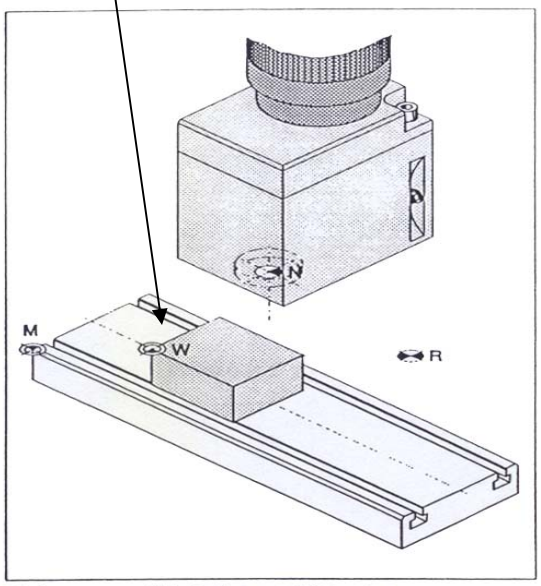
4. Make sure door is closed
5. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
6. Press the X- (arrow pointing left) this references the X axis
7. Press the Y- (arrow pointing left) this references the Y axis



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

WORK SHIFT

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



Reference points in the working area

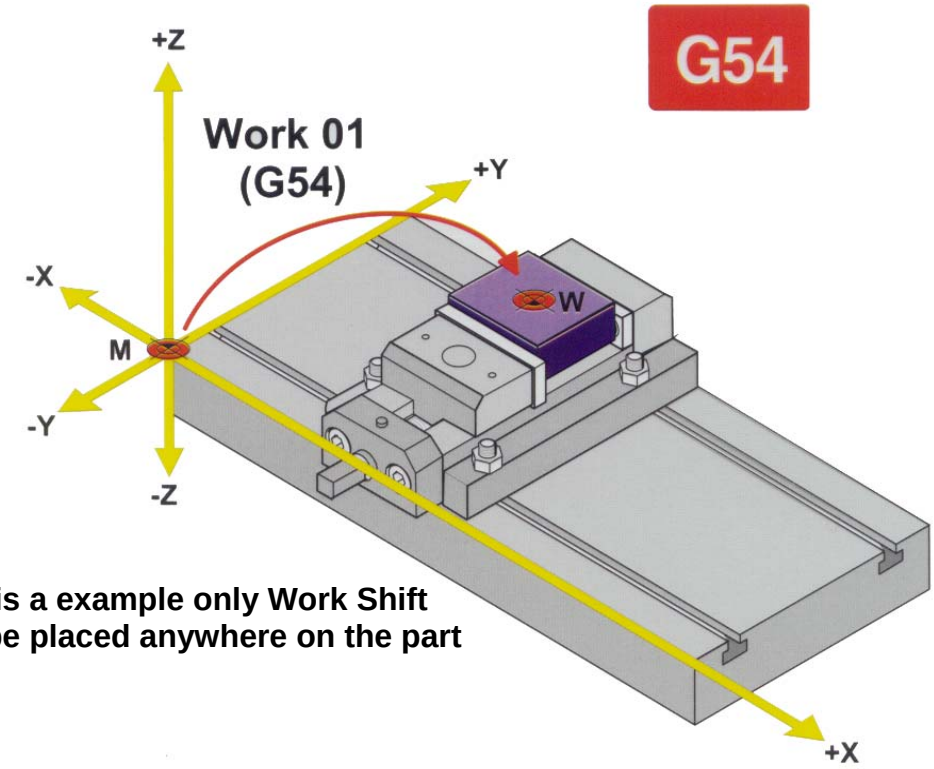
Reference Points of the EMCO Milling Machines

M = Machine zero point
An unchangeable reference point established by the machine manufacturer.
Proceeding from this point the entire machine is measured.
At the same time "M" is the origin of the coordinate system.

R = Reference point
A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the "R".
Required after every power failure.

N = Tool mount reference point
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point
Starting point for the dimensions in the part program. Can be freely established by the programmer and moved as desired within the part program.

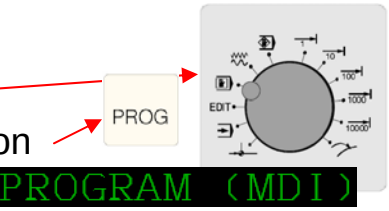


This is a example only Work Shift can be placed anywhere on the part

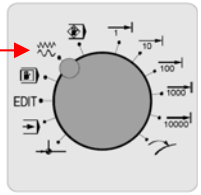
Work Shift:

1. Index to the plug
   - Rotate Mode Dial to MDI
   - Press the PROGRAM display button
     Until top left of the screen shows PROGRAM (MDI)
   - Type T10, M06 (if the tool is in position 2)

     T10=Tool Position M06=Index
   - Then press CYCLE START **(Door must be closed)**



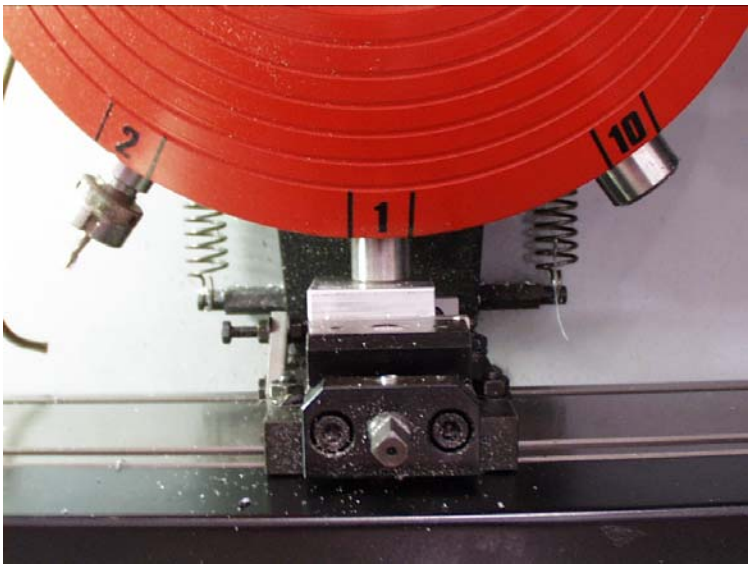
2. Rotate Mode Dial to Jog



3. Jog the plug 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 plug and Work Piece



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

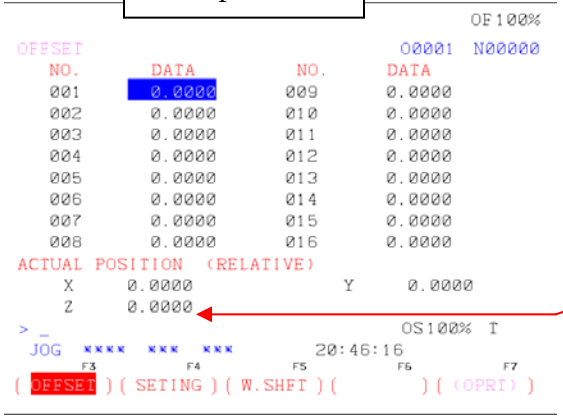
4. Press the OFFSET/SETT button
    - Press the W.SHIFT Soft key (Gray Button)
5. Make sure that X, Y, Z are all 0 if they have values then the Work Shift will be taken from those values not from the machine 0
    - If there is a value then cursor to each one (Example 2) and type 0 and press the input button → INPUT
    - Press the left arrow on the soft keys

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 →

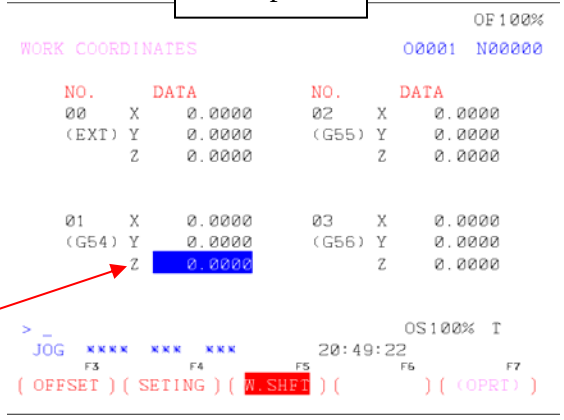
8. Press the W.SHFT Soft key (Gray Button)
9. Cursor down to 01 (G54) location so Z is highlighted (Example 2)
10. Press Input button INPUT

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

Example 1

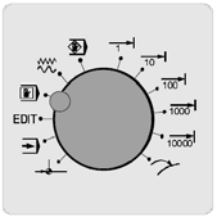


Example 2



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

12. Index to a Tool or Edge finder

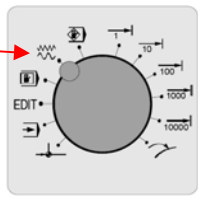


- Move MODE Dial to MDI
- Press the PROGRAM display button PROG until top left of the screen shows PROGRAM （MDI）
- Type T2 M06 (if the tool is in position 2)

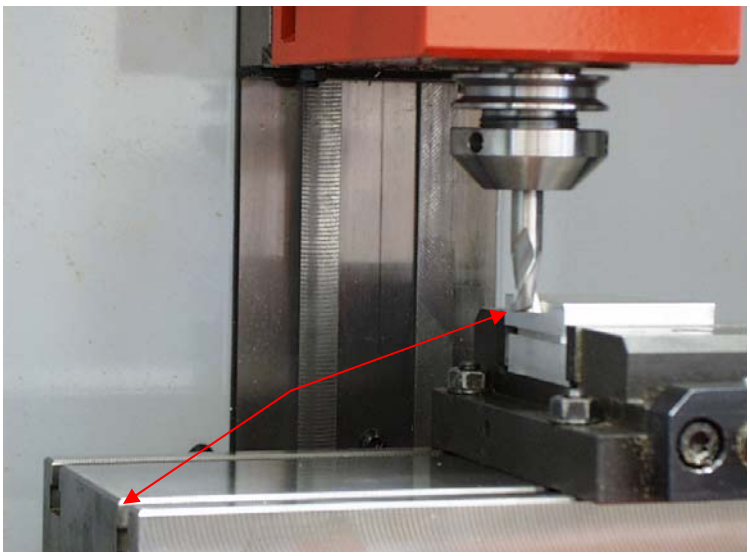
  T2= Tool Position M06=Index

- Press Input button INPUT
- Optional: scratching with the spindle on type S1000 M3 then Press Input button INPUT 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.

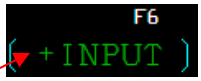


Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

13. Press the OFFSET/SETT button 

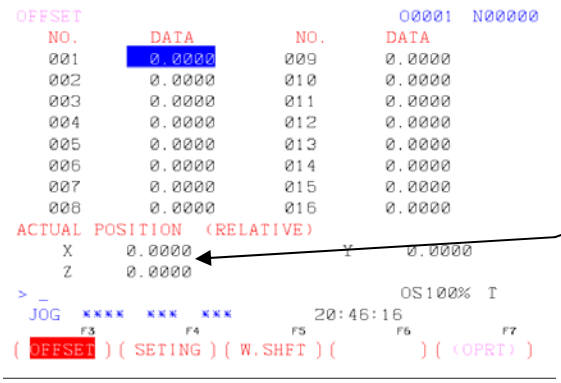
- If the top left of the screen doesn't show OFFSET then press the OFFSET Soft key (Gray button)
- Type the value from the Actual Position Relative X
- Press the left arrow on the soft keys 

14. Press the W.SHFT Soft key (Gray Button) (Example 2)
15. Cursor down to 01 (G54) location highlight X
16. Press Input button 

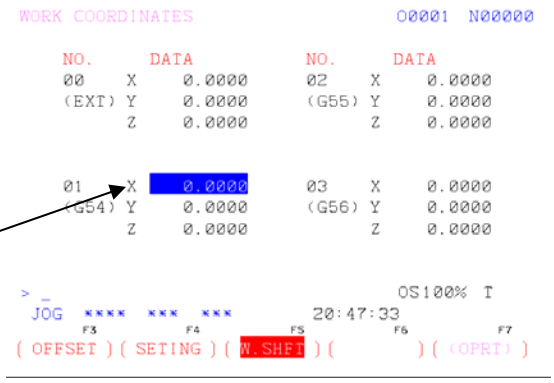
17. Type in the radius for the Tool and press the +input soft key 

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

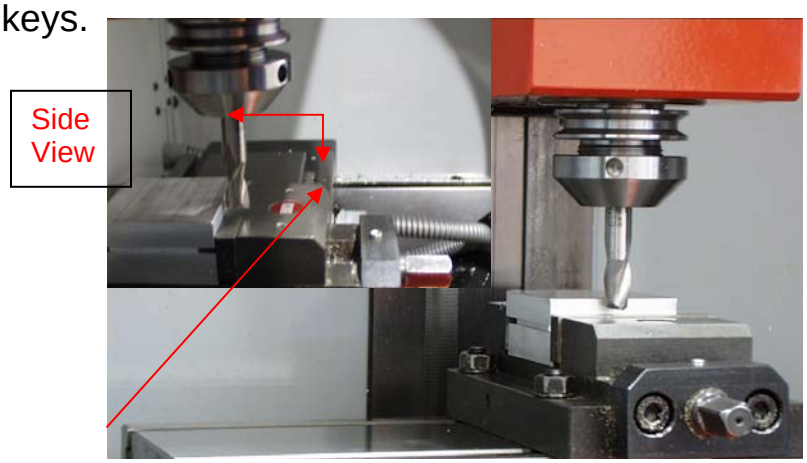
Example 1



Example 2



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.



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

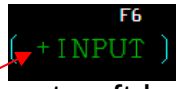
20. If the screen is at OFFSET or WORK COORDINATES then go to the bullets if not then press the OFFSET/SETT button 

- 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)

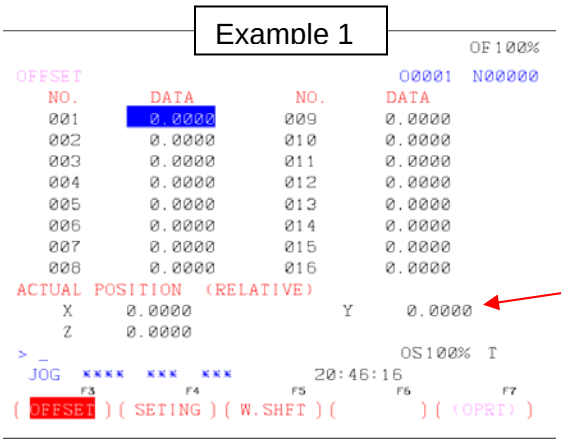
22. Cursor down to 01 (G54) location highlight Y

23. Press Input button 

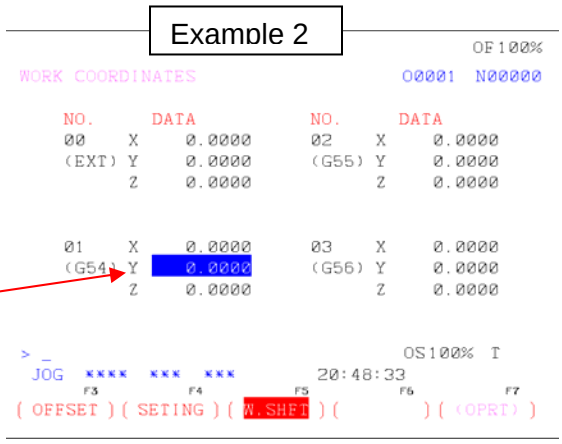
24. Type in the radius for the Tool and press the +input soft key 

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

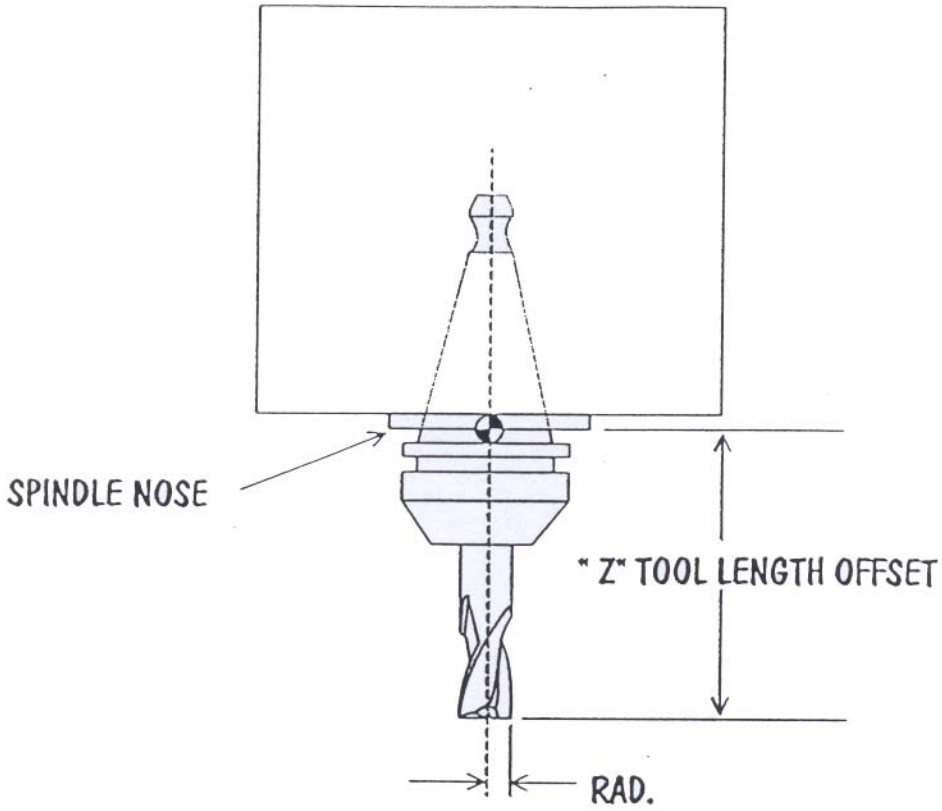


Example 2



25. Jog the Tool up above the Work Piece using 

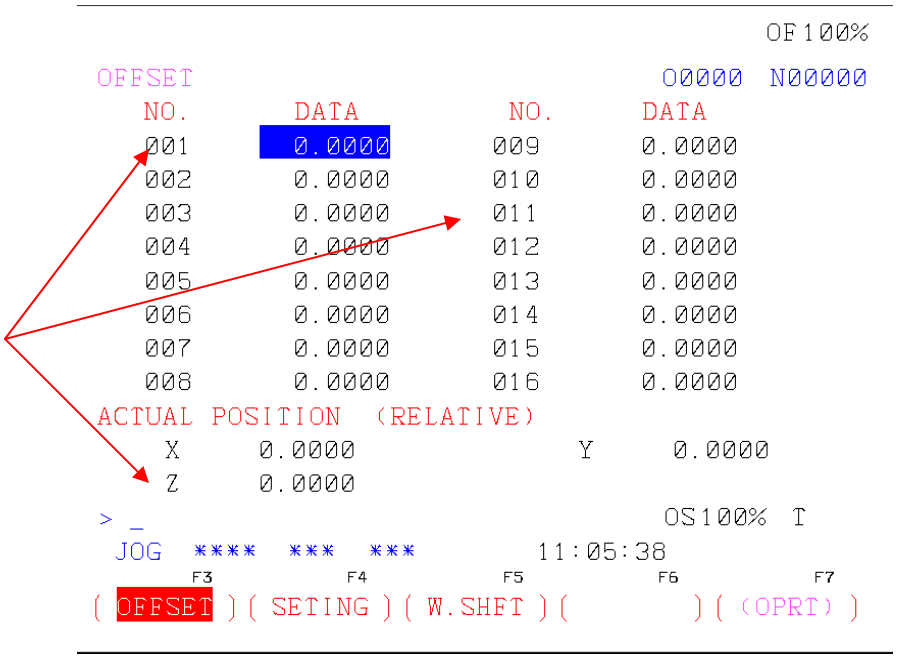
TOOL OFFSET




```
                                                        OF100%
OFFSET                                          O0000  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

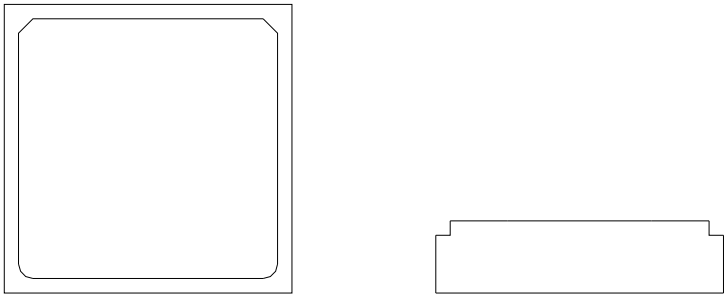


NOTE: When you use a T the H = Height

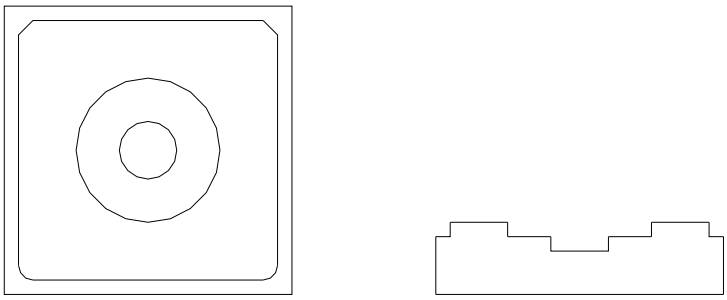
When you use a G41 or G42 the H = Radius

Program Training

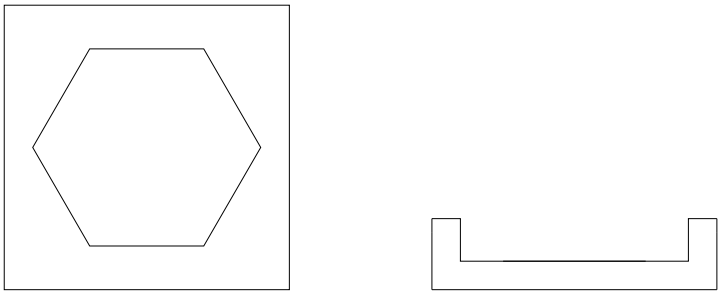
Program O0001



Program O0003



Program O0005



Change the Mode Dial to Edit
Press the Program Button PROG
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 INSERT

  Example: O0001 OR O1

- ## 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 INSERT

  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 EOB
  2. Press insert button INSERT

  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: **<u>O</u>0001 OR <u>O</u>1**

- ## DELETE ALL PROGRAMS

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

   Example: **<u>O – 9999</u>**

- ## 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

PROG

- **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 ↓

- **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

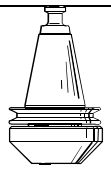
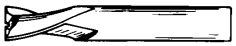
M00	Programmed stop, unconditional
M01	Programmed stop, conditional
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
M05	**Spindle OFF**
M06	Tool Index
M25	Open clamping vice
M26	Close clamping vice
M30	Main program end with new start of program
M71	Blow-off ON
M72	**Blow-off OFF**
M98	Subroutine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

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

Used Addresses

A	Angle
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	**Collet holder**	For ESX-25 collets	
225 100	**(9.0-10.0mm)∅ 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

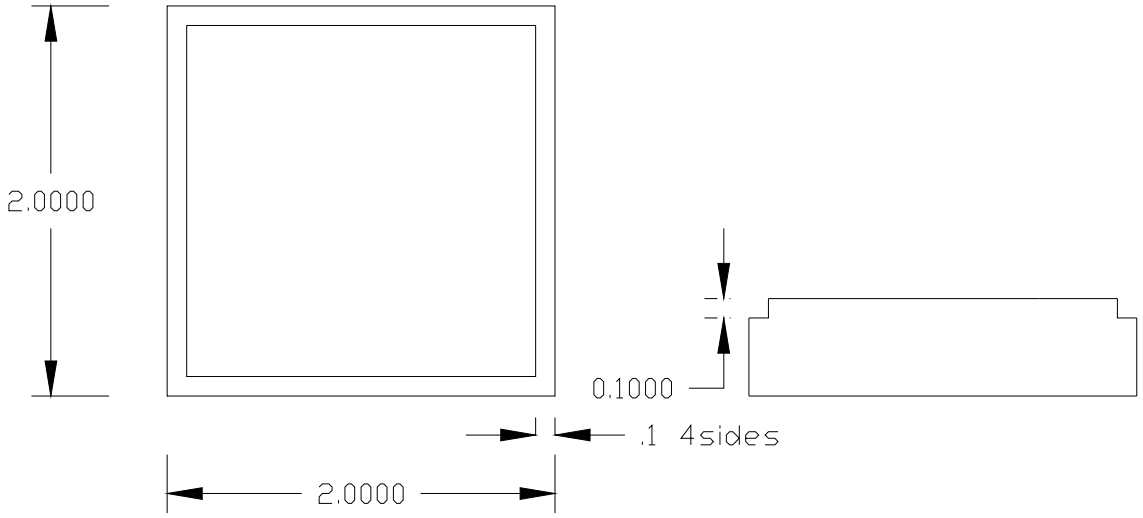
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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 O0001

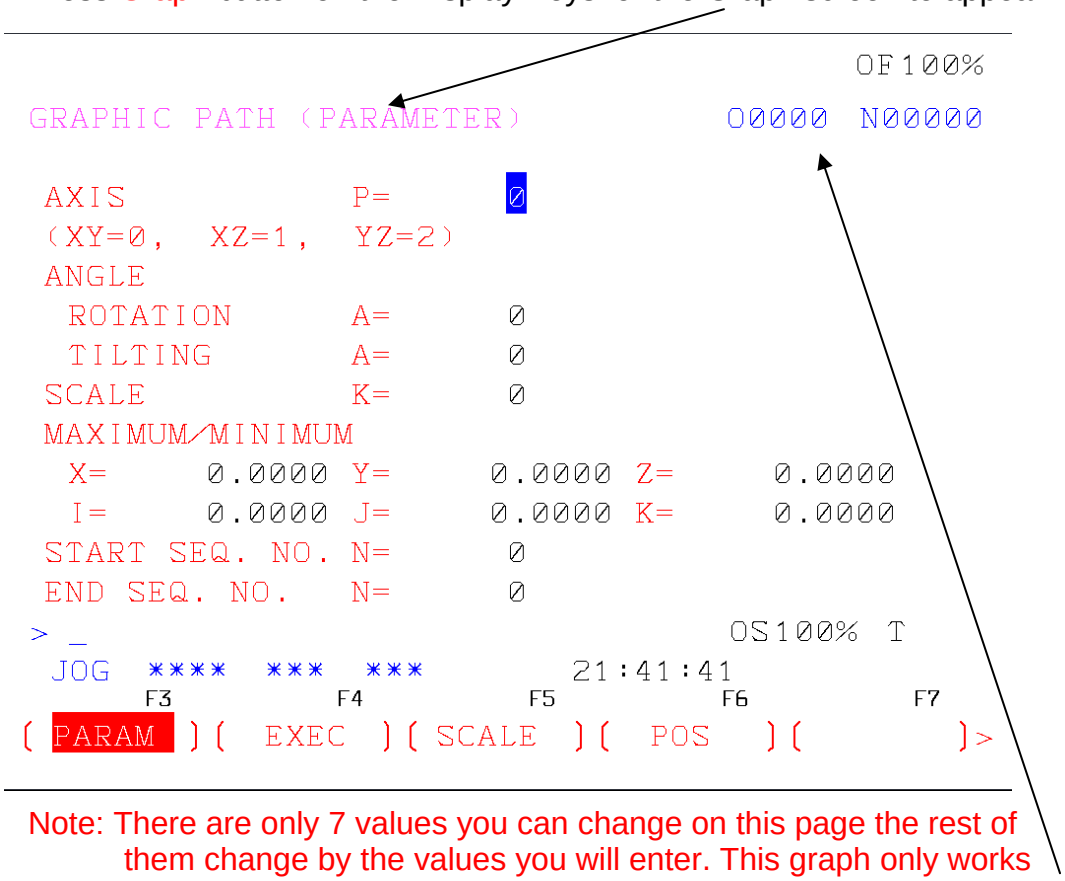


O0001 (Demo 1) (2 X 2 X .5 Alum.)

N5 G00 G17 G40 G80	Default G code
N10 G90 G94 G98	not needed
N15 **G54**	Work shift Call out
N20 G43 T1 H1 M6 (3/8 or 10mm Endmill)	Tool call out line
N25 S1800 M3	Spindle on CW
N30 G0 Z1	Safe move above part
N35 X-1 Y1	Positioning X, Y for cutting
N40 Z-.1	Positioning Z at depth
N45 G1 G41 H11 X.1 F7	CRC on and moving to X
N50 Y1.9	Position Top left corner
N55 X1.9	Position Top right corner
N60 Y.1	Position Bottom right corner
N65 X.1	Position Bottom left corner
N70 Y1	Back to start point Y
N75 G0 G40 X-1	CRC off and start point X
N80 G91	incremental mode
N85 G28 Z0	from position to Z home
N90 G90	absolute mode
N95 M30	End of Program

2D Simulation

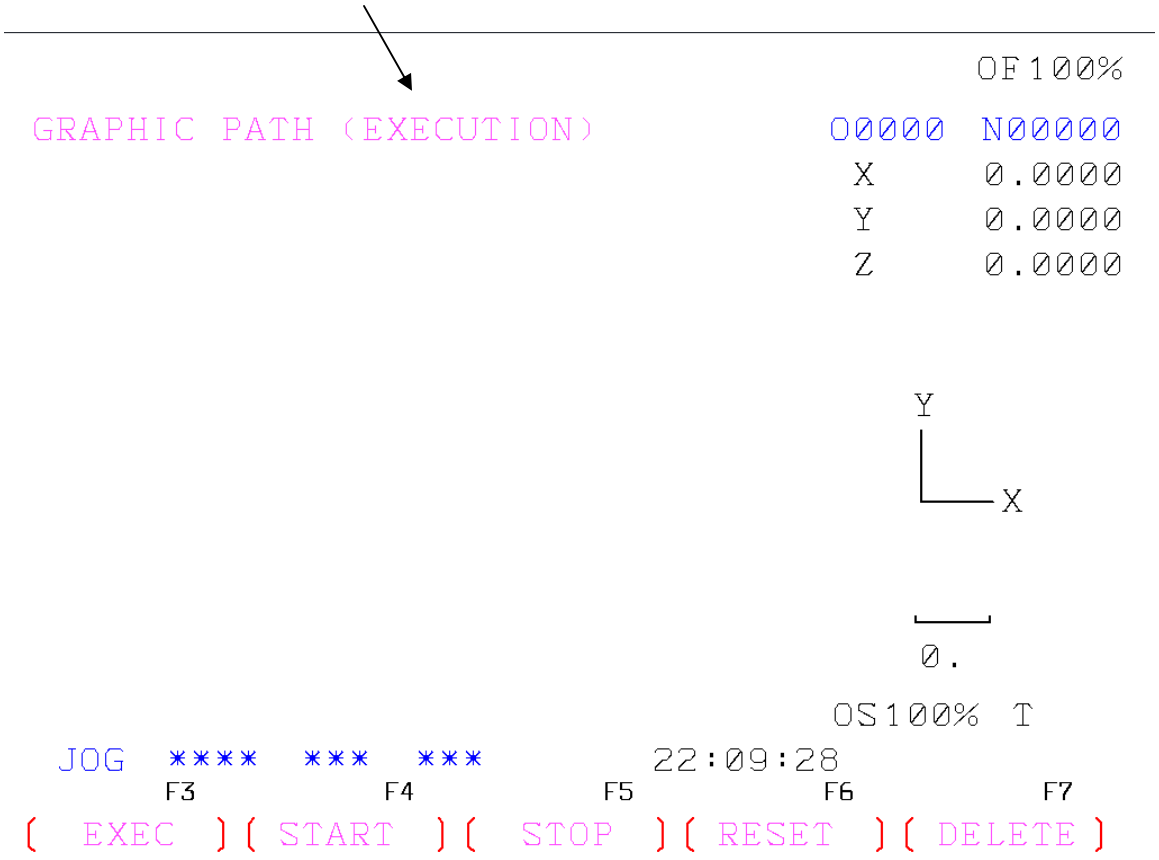
1. Press Graph button on the Display Keys for the Graph screen to appear



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

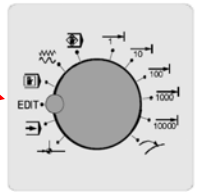
- **Input Offsets into Fanuc Software from Drive unit**

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 

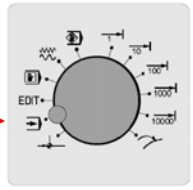
1. Rotate the Mode dial to Edit



2. Press the Program button PROG

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

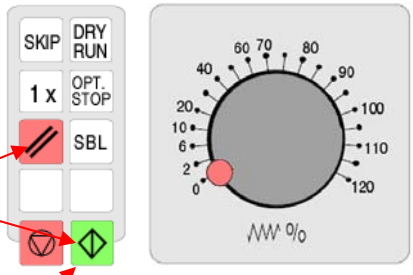
4. Rotate the Mode dial to MEM



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



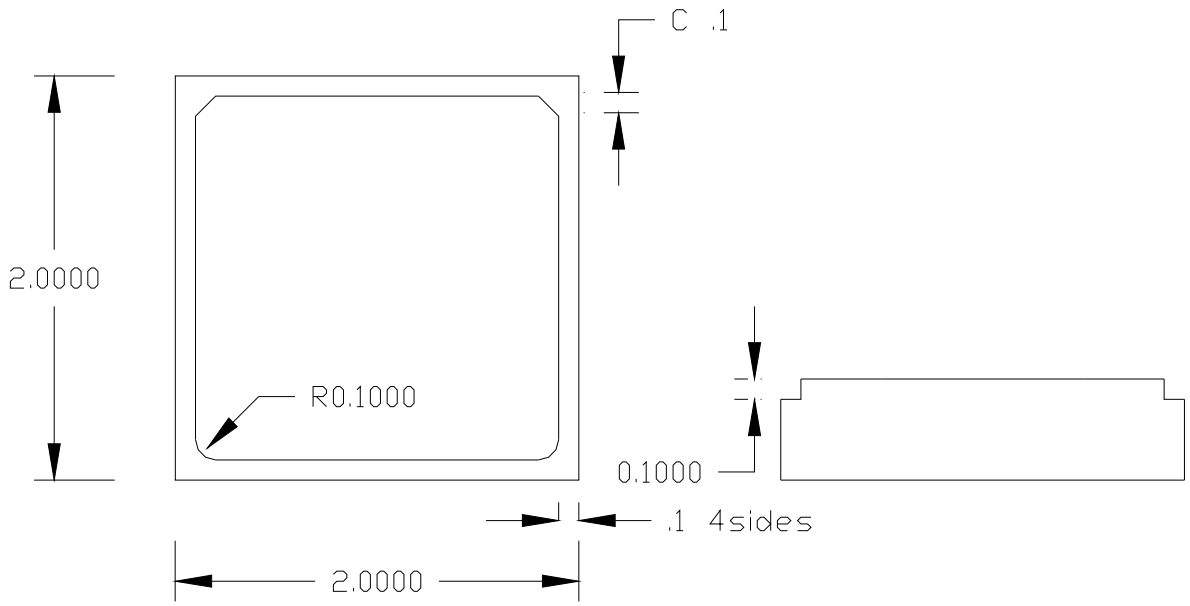
7. Press Cycle Start and continue

(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

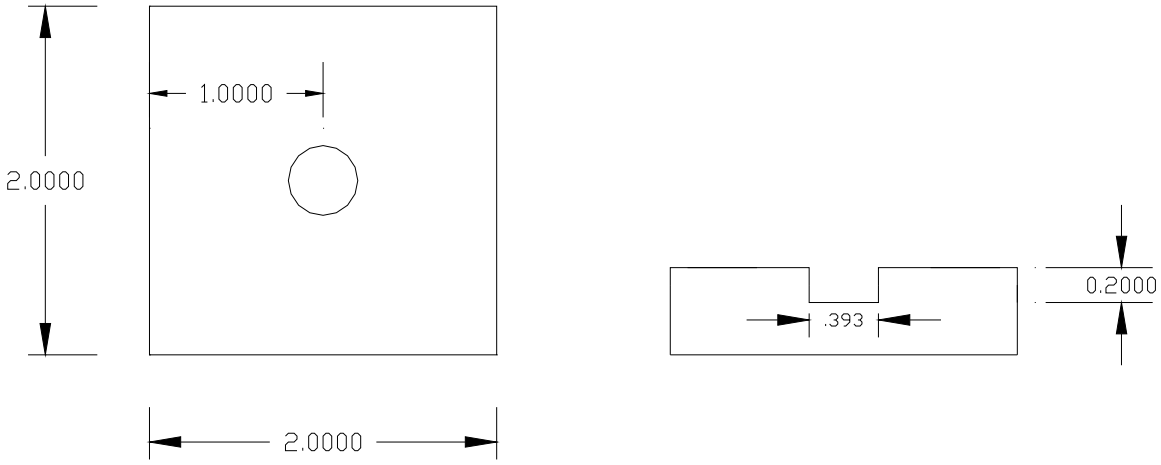
(If there are more than one tool before the next tool use single block to check the offsets locations for the Z only then continue at step 8 again)

Program O0002 (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 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** Automatic Chamfer 45 degrees
N55 X1.9 **C.1** Automatic Chamfer 45 degrees
N60 Y.1 **R.1** Automatic Radius 90 degrees
N65 X.1 **R.1** Automatic Radius 90 degrees
N70 Y1
N75 G0 G40 X-1
N80 G91 G28 Z0
N85 G28 X0 Y0
N90 G90
N95 M30

Program O0003 (Deep Hole Drilling)

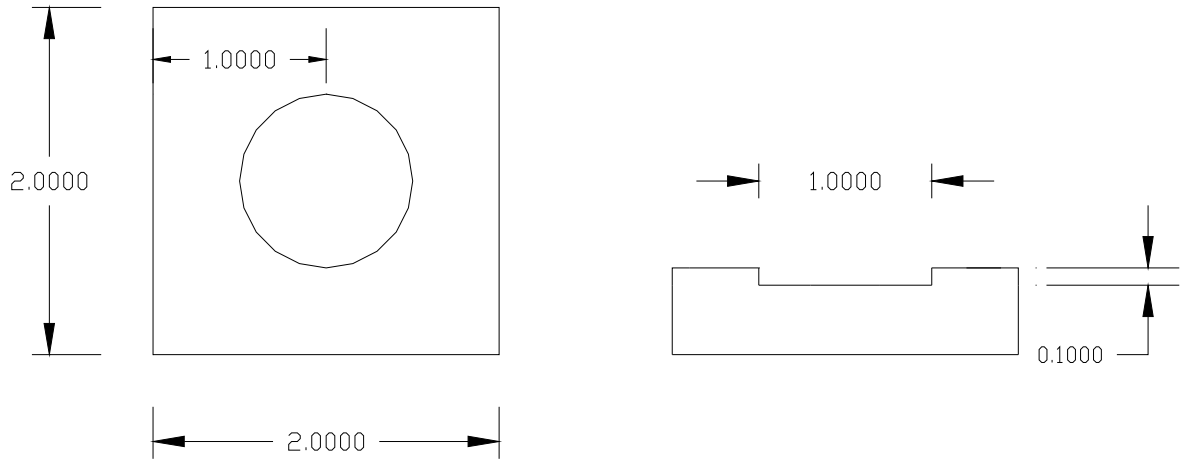


G83 X = Location of hole Y = location of hole
Z = Overall Depth of hole P = Dwell at bottom of hole
R = Retract after Cycle **Q = incremental peck depth per pass**
K = Incremental repeats only used with G91 **F = Feed rate**

O0003 (Demo 3 Drill) (2 X 2 X .5 Alum)
N5 G54
N10 G43 T1 H1 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 G91 G28 Z0
N50 G28 X0 Y0
N55 G90
N60 M30

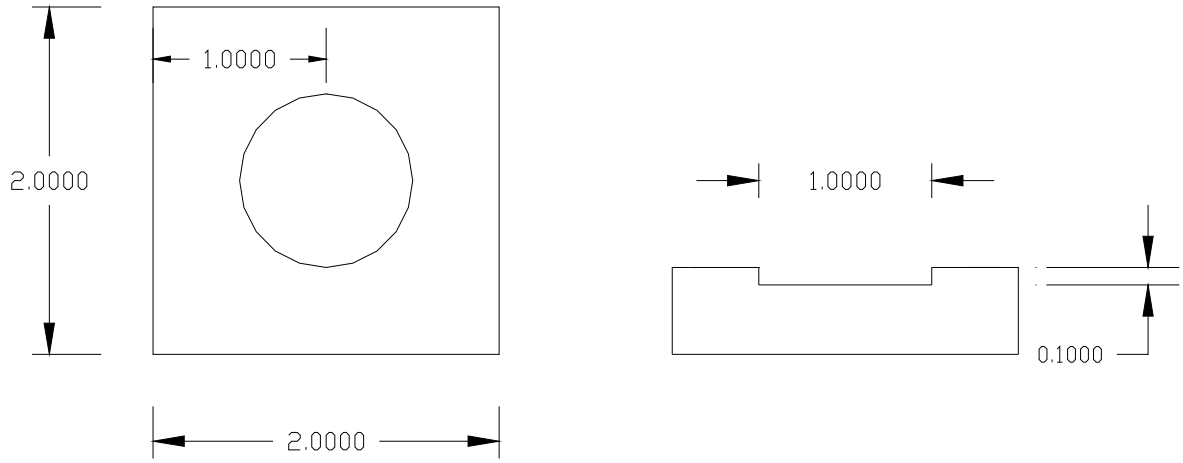
Note: G0, G1, G2, G3 all cancel the drilling cycle

Program O0004 (R)



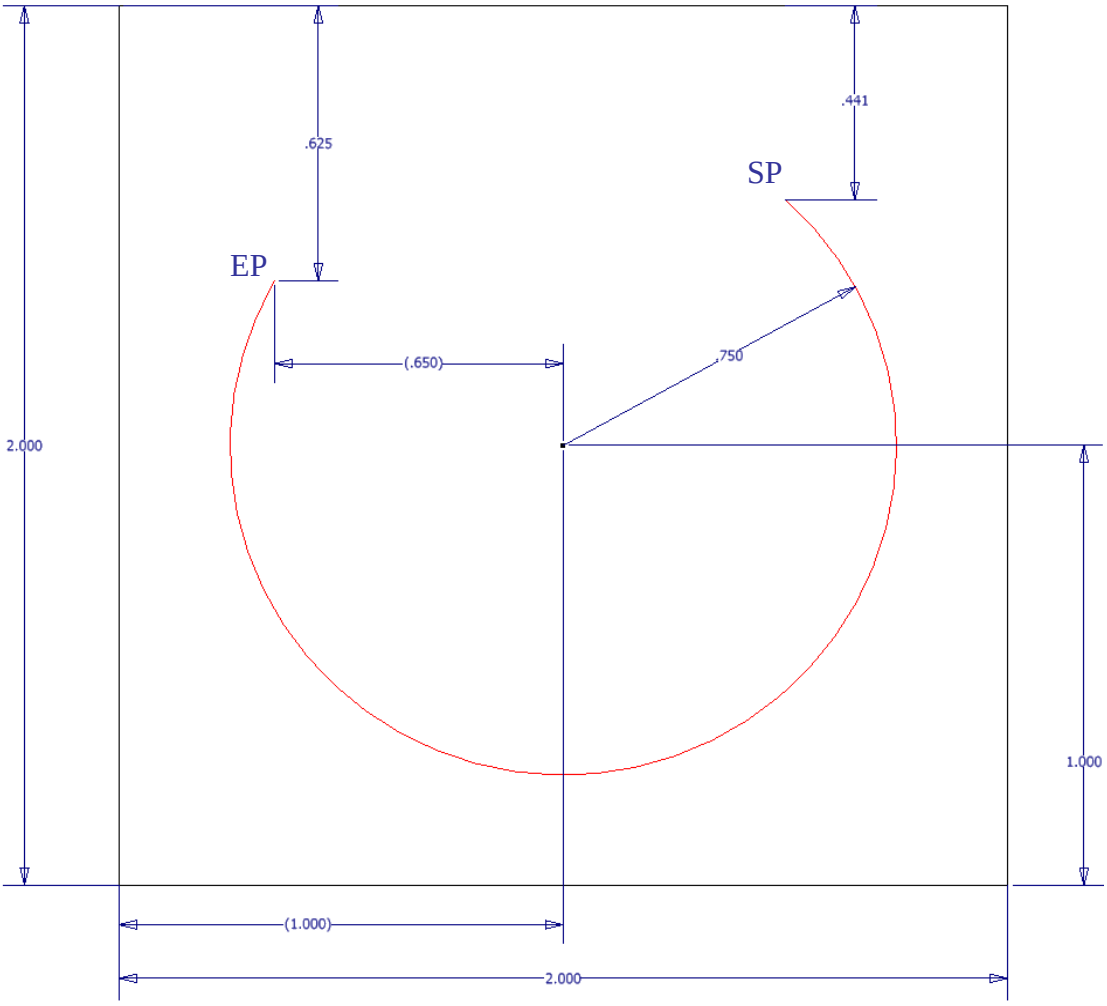
O0004 (Demo 4 R) (2 X 2 X .5 Alum)
N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F5
N50 G2 X1.5 Y1 R.5 (180 Degrees)
N55 G2 X.5 Y1 R.5 (180 Degrees)
N60 G0 G40 X1
N65 G91 G28 Z0
N70 G28 X0 Y0
N75 G90
N80 M30

Program O0005 (I & J)



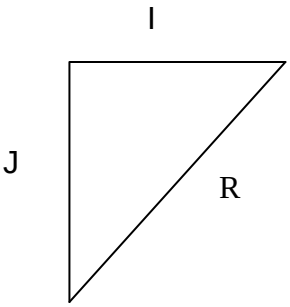
O0005 (Demo 5 I/J) (2 X 2 X .5 Alum)
N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F5
N50 G2 X.5 Y1 I.5 J0 (360 degrees)
N55 G0 G40 X1
N60 G91 G28 Z0
N65 G28 X0 Y0
N70 G90
N75 M30

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 (K leg) + B^2 (I leg) = C^2 (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

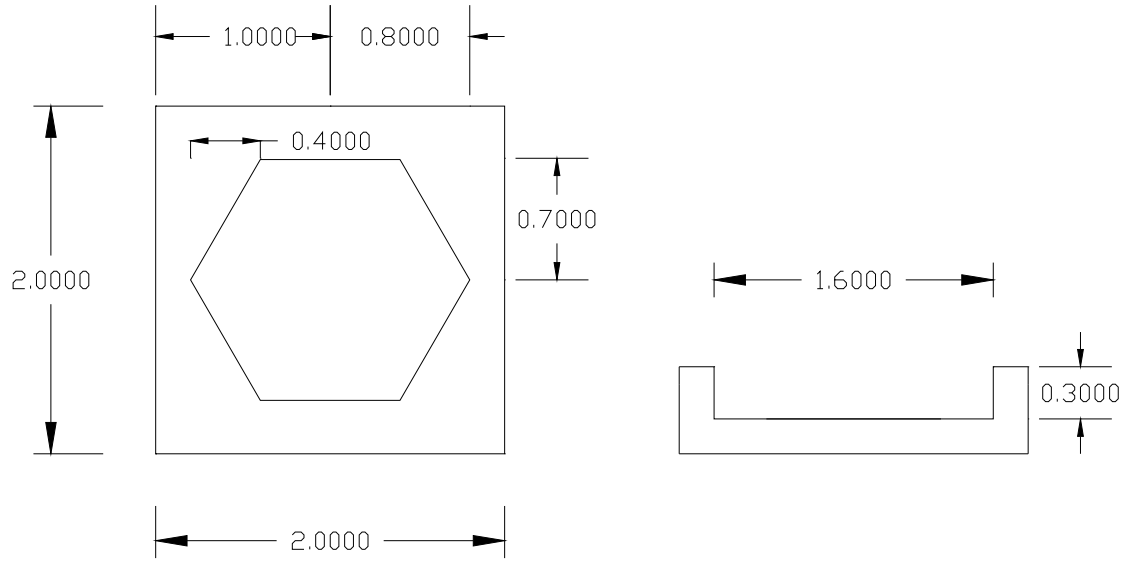
1. To make all programs tie together O0002, O0003, O0004 to run together. Use M98 this calls out Sub programs or Sub routines. Example: M98 P010001
2. After M98 P is identified with 6 digits.
   - The First 2 digits is the number of times program is to be repeated
   - The next 4 digits is the program 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 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 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)**

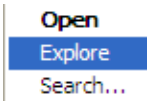
   - **Press** 

to allow you to exit
   - **Press** 

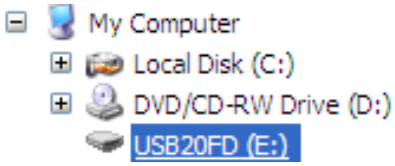
and  **together to exit the Software**

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** 

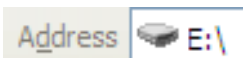
(Explorer)
   - **Left Click**
   - **If you right clicked on My computer skip to step 4 if not then Left Click on My Computer**



4. **Copy Drive directory**

   - **Click on you USB drive**

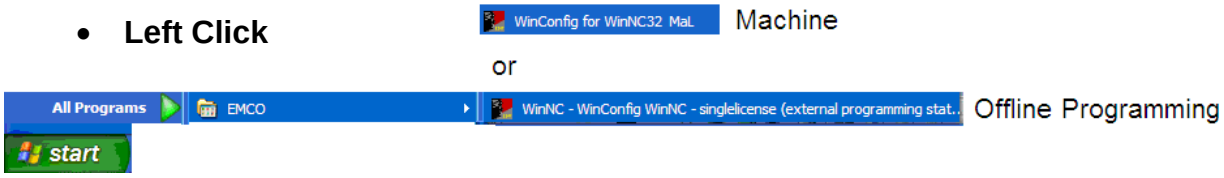
   - **At the top of the active screen or page in the Address copy or remember drive info**



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

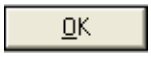
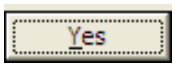
5. **Setting up WinConfig**

   - **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)**
   - **Left click on white box**

   - **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**