innovative machine tools

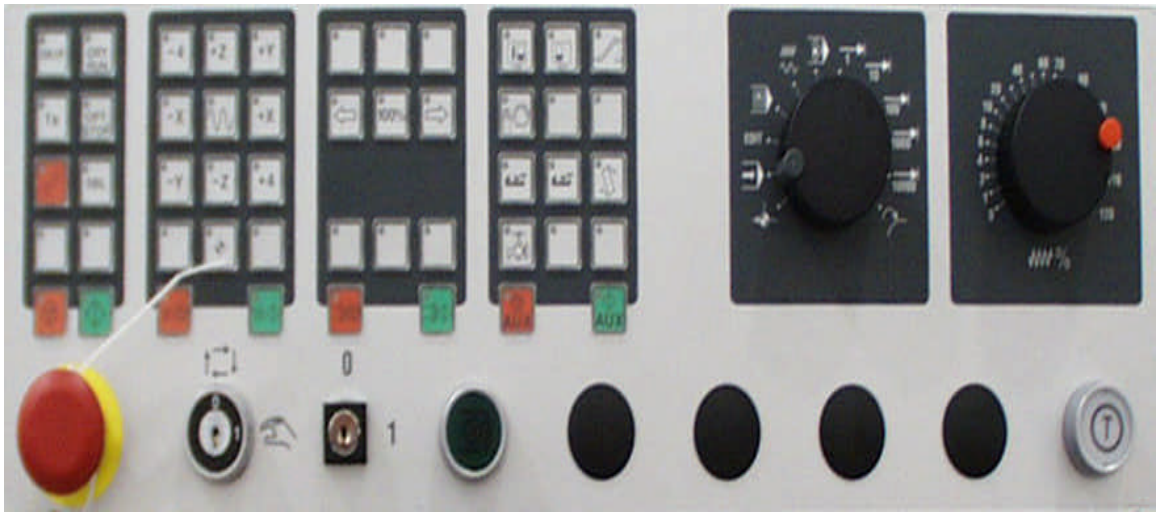
GE FANUC 21
155 MILL
TRAINING GUIDE

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

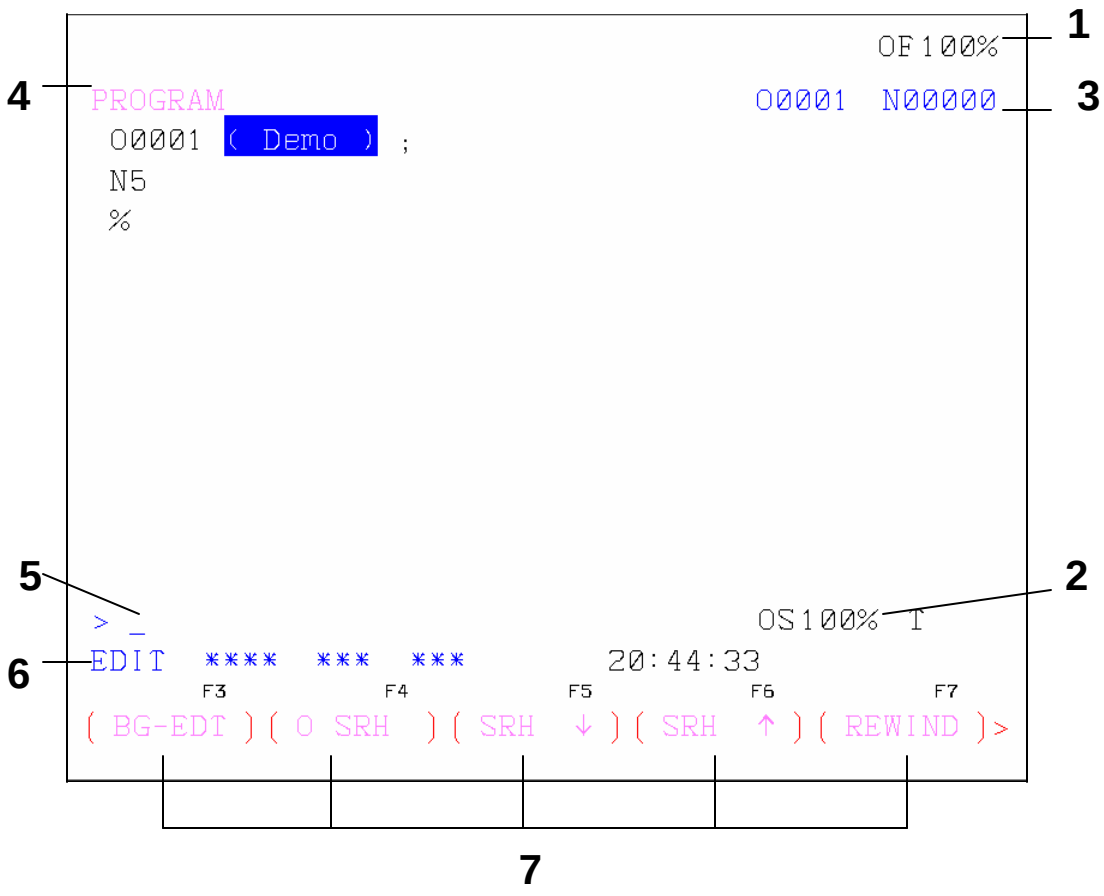
Training Index

FANUC 21 CONTROL

MACHINE CONTROL



The Fanuc 21 Screen



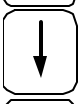
1. **Displays of Feed**
2. **Spindle Speed override**
3. **Display of Program and Number block**
4. **Display of active Screen**
5. **Entry line**
6. **Display of active Mode**
7. **Display of Soft key Functions**

FANUC 21 KEYS

RESET = cancels most alarms, resets program, interrupts programs

CURSOR MOVEMENT KEYS

CURSOR UP = moves cursor up



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

PAGE UP = moves one page up

PAGE DOWN = moves one page down

CURSOR RIGHT = moves cursor right

CURSOR LEFT = moves cursor left

CHANGE KEYS

ALTER = alter word (replace word)

INSRT = insert word, create new program

DELET = deletes word / block or programs

EOB = end of block

CAN = deletes entries in the address one by one

INPUT = input offsets / words or numbers

DATA INPUT KEYS

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

FUNCTION KEYS (DISPLAY KEYS)

POS = displays actual, relative, machine positions

PROG = displays program, library page

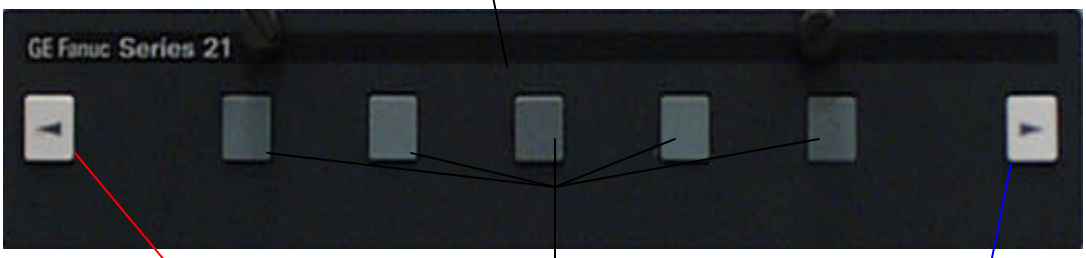
OFFSET/ SETTINGS = displays wear, geometry, work shifts pages

SYSTEM = displays parameters, diagnostic pages

MESSAGE = displays operator & alarm messages

GRAPH = displays 2–d graph simulation

SOFT KEYS



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

MACHINE KEYS

MACHINE FUNCTION KEYS

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

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

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

= (Optional stop) for programs with (m1)

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

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

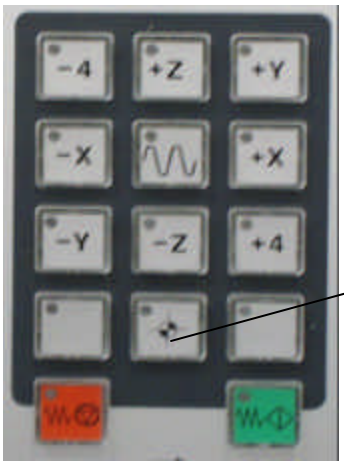
= (Cycle stop) program hold, feed hold

= (Cycle start) program start

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

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

=(Cycle start) program start



DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

Reference all axis

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

SPINDLE OVERRIDE KEYS

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

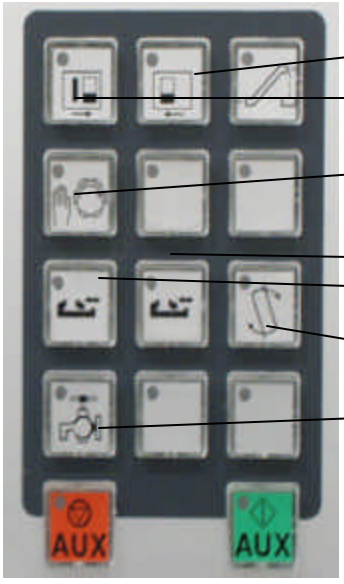
Arrow key pointing left decrease the Spindle speed (50% low)

100% key jumps speed to 100%

Spindle stop (Red) / Spindle start (Green)

Works all modes except EDIT & ZRN (Reference)

ACCESSORY FUNCTIONS



Arrow right door closed
Arrow left door open

Press once rotary rotate

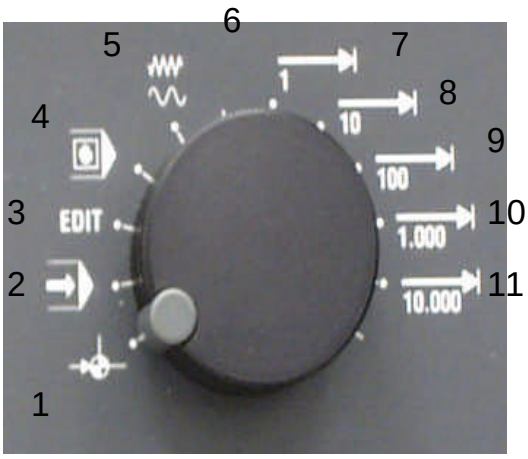
Press vise open
Press vise closed

Press for Tool turret Index

Press once coolant on
Press again coolant off

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

MODE DIAL



(1) REF = Zero / Reference or Home mode

(2) MEM = Automatic mode for running a program

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

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

(5) JOG = Manual moving the axis in X, Y or Z

(6) STEPS = Incremental feed movements

(7) STEPS = .0001 or tenths

(8) STEPS = .001 or thousands

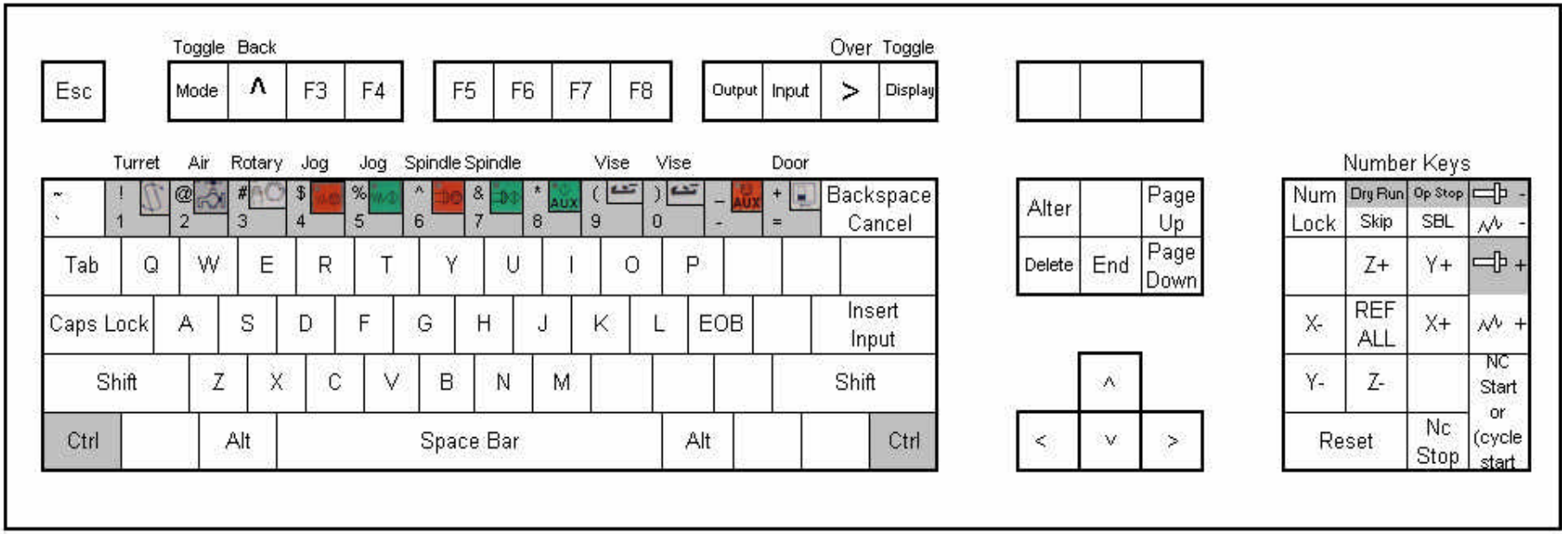
(9) STEPS = .010 or ten thousands

(10) STEPS = .100 or hundred thousands

(11) STEPS = .100 or hundred thousands

FEED OVERRIDE DIAL

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



1. Any key with Gray highlight Press Ctrl + the key for that function
2. Some keys have two functions to them for 1st function just press the key
3. 2nd function will be Grey press Ctrl + the key for the function
4. Some automative keys when you press them 1 time this will close/turn off press them again will open/turn on
5. F1 is a toggle key for the modes: Zero, Auto, Edit, MDI, Jog and F1 then F11 give Increment Step
6. F12 is a toggle key for the Display screens: Position, Program, Offsets, Parameter, Alarm and F12 then F11 then F3 gives Graph
7. F12 then F11 then F3 then F11 then F3 gives you 3D view
8. Press enter 2 times this is the same as pressing EOB insert
9. Alt + F4 will exit the software back to the desktop
10. 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. 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**"

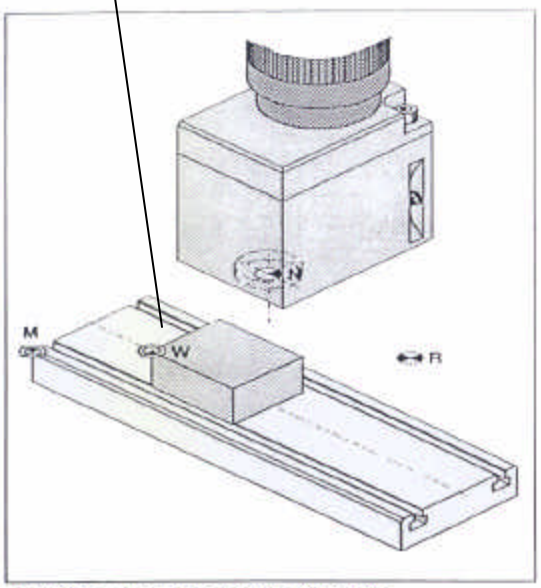
4. Make sure the door is closed
5. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
6. Press the X- (arrow pointing left) this references the X axis
7. Press the Y- (arrow pointing left) this references the Y axis

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 Tool stump



Reference points In the working area

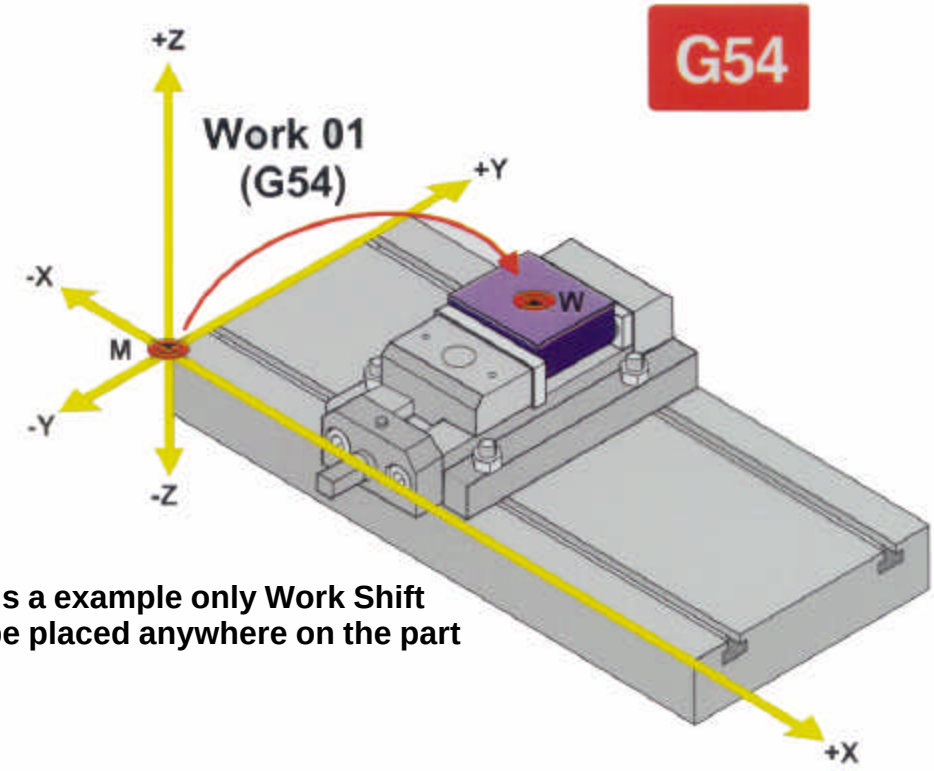
Reference Points of the EMCO Milling Machines

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

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

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

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



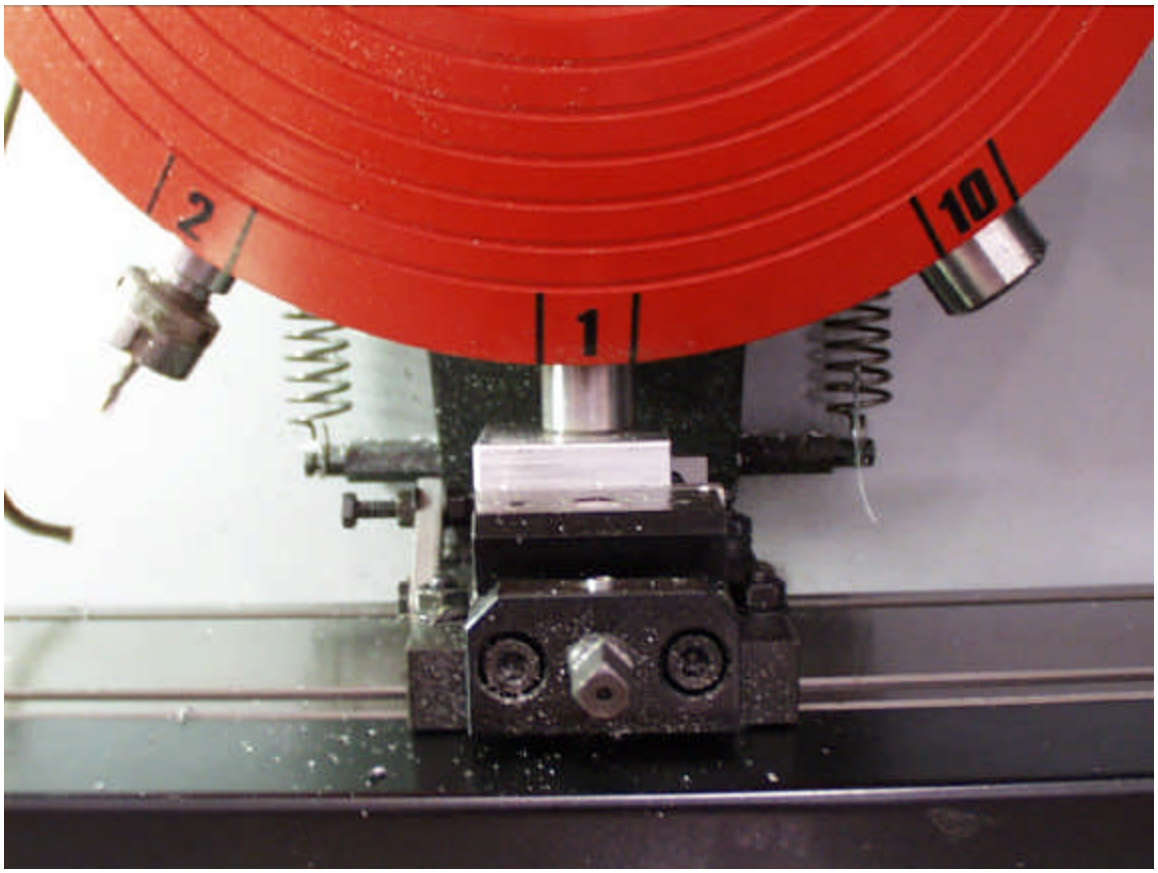
This is a example only Work Shift can be placed anywhere on the part

Work Shift:

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

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

(Use piece of paper between nose and Work Piece)

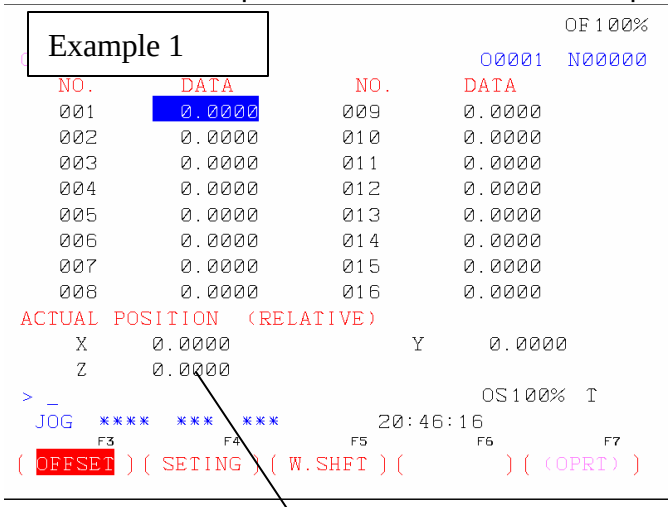


3. Press the OFFSET/SETT button 

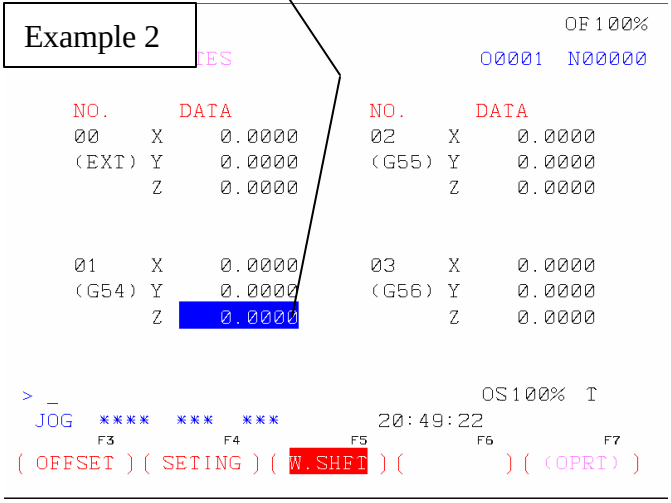
   - Press the W.SHIFT
4. Make sure that X, Y, Z are all 0 if they have values then the Work Shift will be taken from those values not from the machine 0
5. Press the OFFSET Soft key (Gray Button)
   - Record the value in the Actual Position Relative Z (Example 1)
6. Press the W.SHFT Soft key (Gray Button) (Example 2)
7. Move Cursor to 01 (G54) location so Z is highlighted
8. Recorded value type in Work Coordinates 01 (G54) Z
   Example: Type 2.463 press Input button 

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

Example 1



Example 2

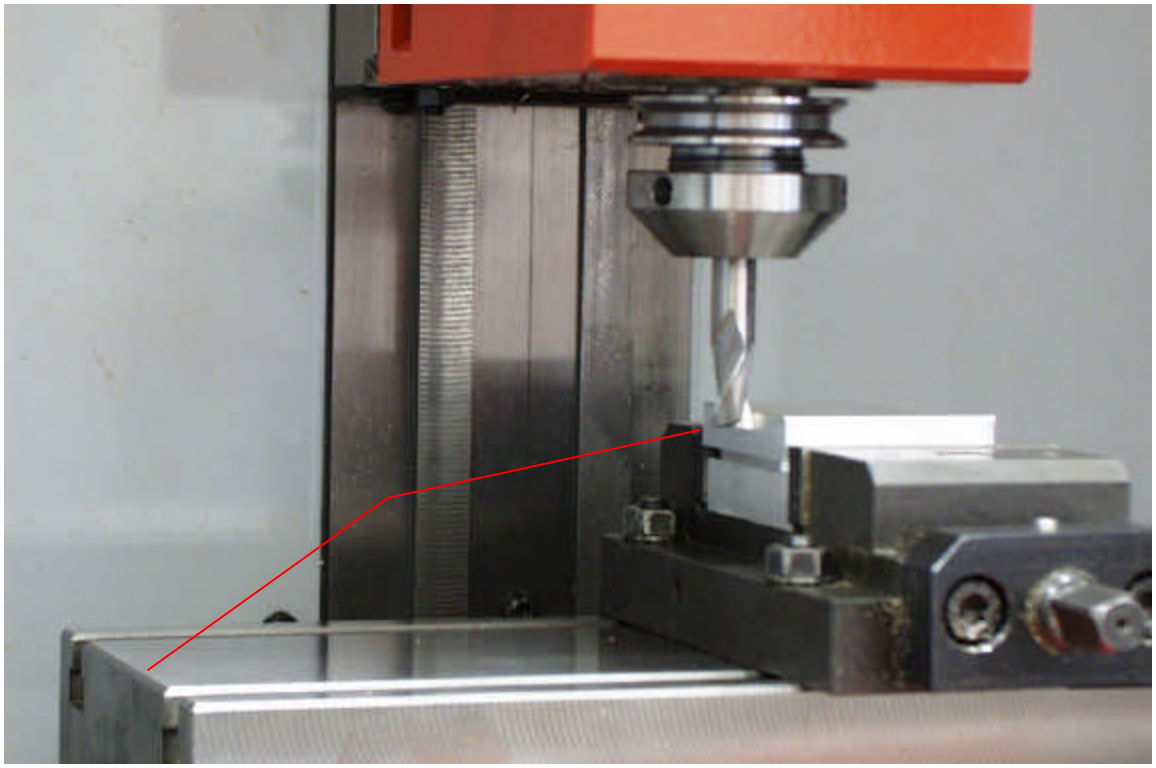


9. Jog Spindle up away from WORK PIECE using Z+
10. Either follow step 11 or follow step 12 when finished go on to step 13
11. Index to a edge finder or tool (Example uses 3/8 end mill) Press 
    - Jog the Tool to the left side of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps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

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

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

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

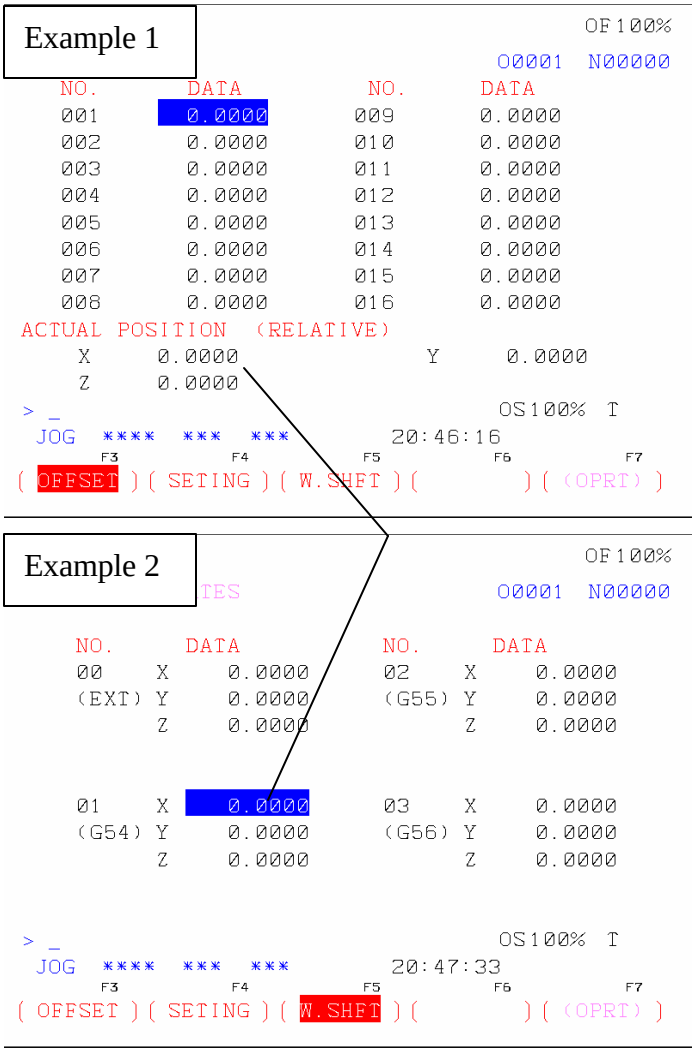


Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

13. Press the OFFSET/SETT button 

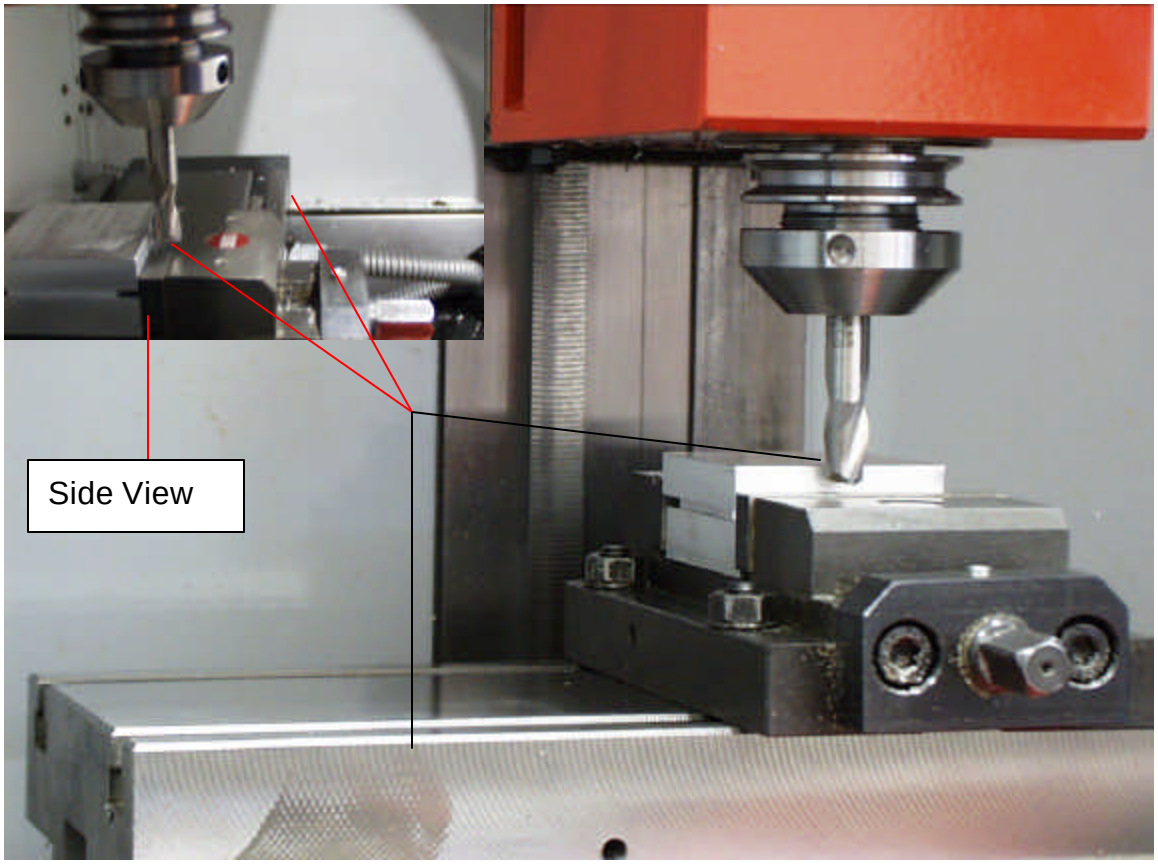
    - Example 1
    - Record the value in the Actual Position Relative X
14. Press the W.SHFT Soft key (Gray Button) (Example 2)
15. Move Cursor to 01 (G54) location highlight X
16. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (G54) X

Example: Type 8.463 press Input button 



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

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



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

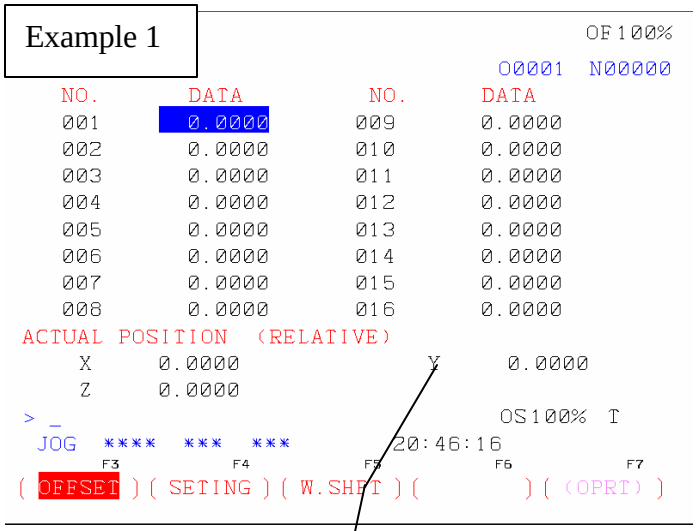
19. Press the OFFSET/SETT button 

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

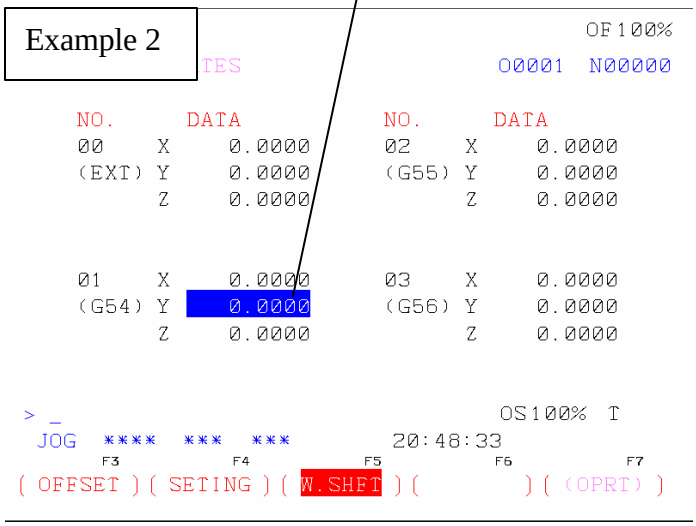
Example: Type 8.463 press Input button 

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

Example 1

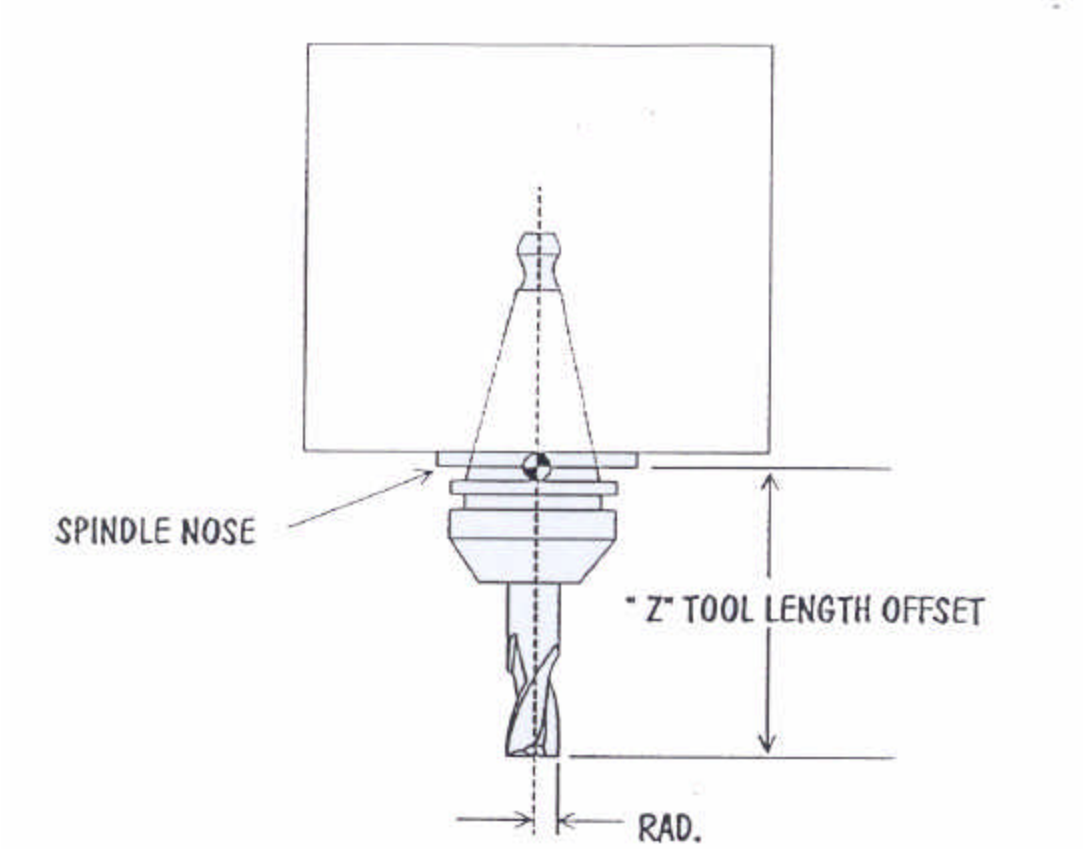


Example 2



23. Jog the Tool up above the Work Piece

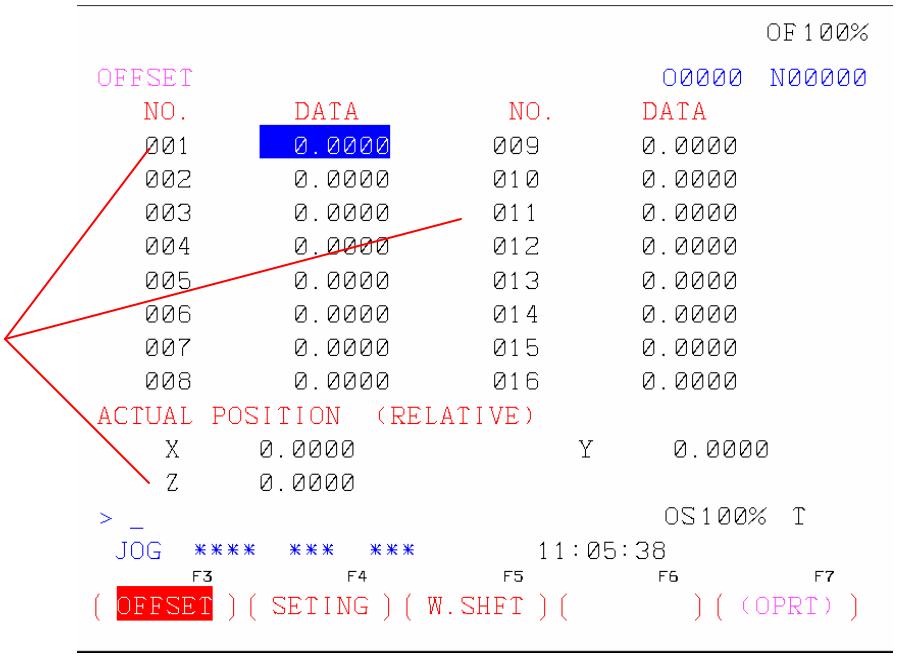
TOOL OFFSET




```
                                                        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

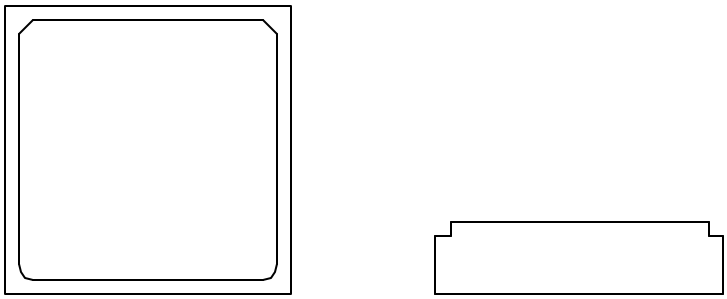


NOTE: When you use a T the H = Height

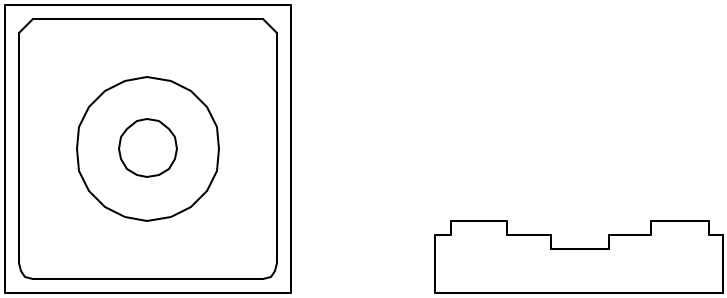
When you use a G41 or G42 the H = Radius

Program Training

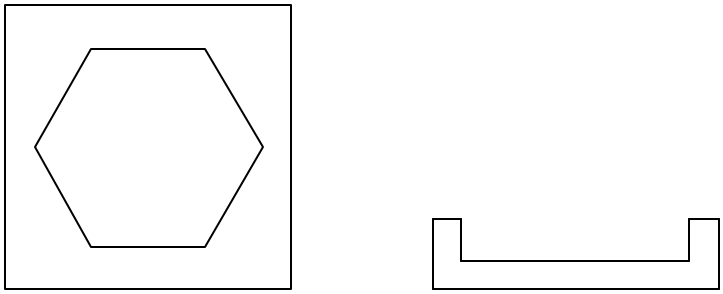
Program O0001



Program O0003



Program O0005



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

- **INSERT A NEW PROGRAM**
  1. Press letter o then program number
  2. Press insert button INSRT

  Example: O0001 OR O1

- **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 INSRT or input INPUT

  HINT: When inserting a word to the left of the highlighted word the new word will be placed

  Example: **N5 G01 X 0.25; G01** is the word being inserted

- **INSERT END OF BLOCK**
  1. Press the (EOB) button / , # EOB
  2. Press insert button INSRT or input INPUT

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

  Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT OR INPUT.

USE INPUT FOR ALL OTHER APPLICATIONS.

- **DELETE A PROGRAM**

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

     Example: **O0001 OR O1**

- **DELETE ALL PROGRAMS**

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

     Example: **O – 9999**

- **DELETE A WORD**

  1. Highlight the Word
  2. Press delete button DELET

- **DELETE A BLOCK OR LINE NUMBER**

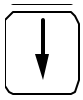
  1. Type the number line and highlight the number line
  2. Press delete button DELET

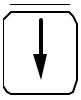
- **CANCEL MISTYPED WORD (Backspace)**

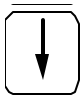
  1. Press cancel button CAN

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

- **ALTER A WORD**
  1. Highlight the word needed altered type the change
  2. Press alter button 

- **SEARCH FOR NUMBER BLOCK**
  1. Press letter n and the number of the block
  2. Press cursor down button 

- **SEARCH FOR WORD**
  1. Type in word & number
  2. Press cursor down button 

- **SEARCH FOR LETTER**
  1. Press letter
  2. Press cursor down button 

HINT: This goes to the first (G). Follow steps 1 & 2 cursor goes to the next (G)

Survey commands G-CODES : Mostly used only

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

Survey commands M functions:

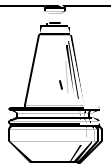
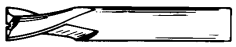
M00 Programmed stop, unconditional
M01 Programmed stop, conditional
M02 Main programmed end
M03 Spindle ON clockwise
M04 Spindle ON counter clockwise
M05 Spindle OFF
M06 Tool change
M08 Coolant ON
M09 Coolant OFF
M19 Orientated spindle stop
M25 Release clamping device
M26 Close clamping device
M30 Main program end with new start of program
M71 Blow-off ON
M72 Blow-off OFF
M98 Subroutine call-up
M99 Subroutine end

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

Used Addresses

C Chamfer
F Feed rate, thread pitch
G Path function
H Tool height, tool radius
I, J, K Circle parameter, scale factor, K number of repetition
M Miscellaneous function
N Block number 1 to 9999
O Program number 1 to 9499
P Dwell, subroutine
Q Cutting depth or shift value
R Radius, retraction height
S Spindle speed
T Tool called out
X, Y, Z Position data
; Block end

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

Q1Z 910	**Collet holder**	For ESX-25 collets	
152 800	**(9.0-10.0mm)Æ 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

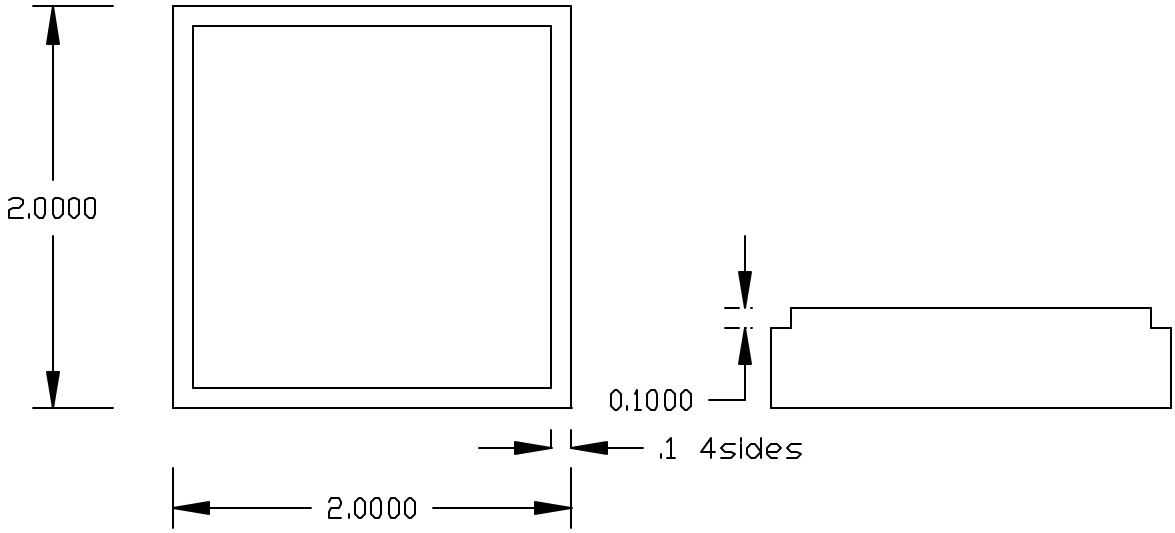
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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

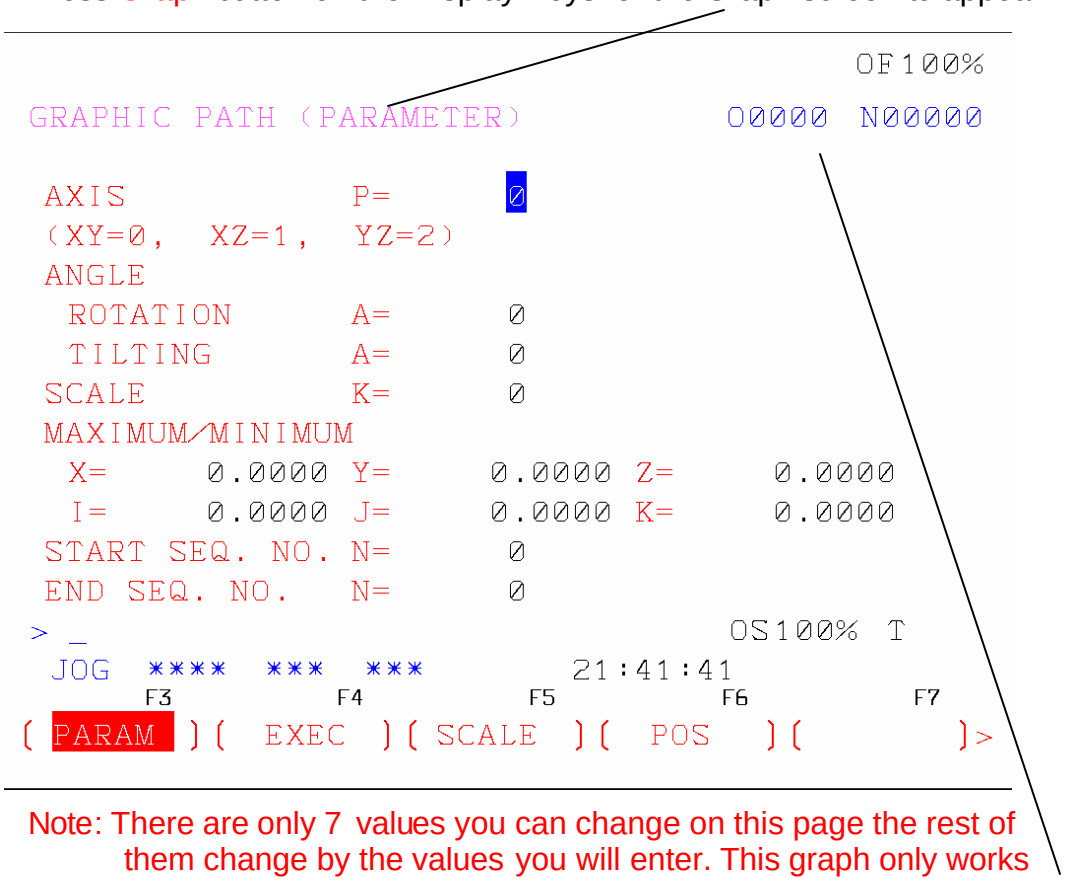
Program O0001



O0001 (Demo 1) (2 X 2 X .5 Alum.)
N5 G00 G17 G40 G80
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
N55 X1.9
N60 Y.1
N65 X.1
N70 Y1
N75 G0 G40 X-1
N80 G28 Z1.5
N85 G28 X2.5 Y2.5
N90 M30

2D Simulation

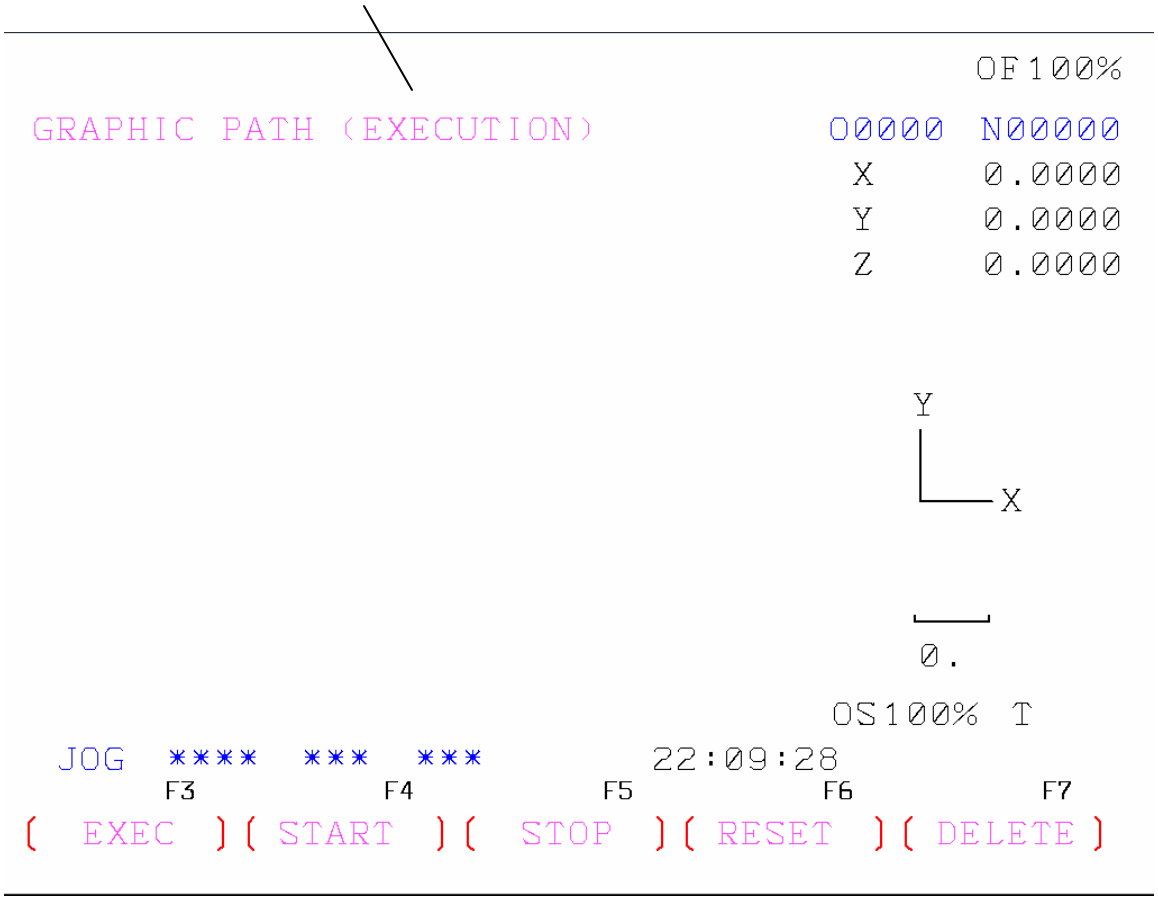
1. Press Graph button on the Display Keys for the Graph screen to appear



Note: There are only 7 values you can change on this page the rest of them change by the values you will enter. This graph only works with an active program and runs only the current program selected

2. Axis P = 0 means G17 1 means G18 2 means G19
3. Maximum/Minimum X = Overall Length of the stock in X direction this is a positive value
4. Maximum/Minimum Y = Overall Width of the stock in Y direction this is a positive value
5. Maximum/Minimum Z = Overall Height of the stock in Z direction this is a positive value
6. Maximum/Minimum I = This value is normally a negative number and this is the viewable area passed X0 going negative
7. Maximum/Minimum J = This value is normally a negative number and this is the viewable area passed Y0 going negative
8. Maximum/Minimum K = This value is normally a negative number and this is the viewable area passed Z0 going negative

9. Press the Soft key EXEC for Execution screen



Note: If you press the EXEC on this screen this will auto scale for you. You will need to press the arrow left on the soft keys to go back and enter your values that you originally had.

10. Now press Cycle start or Soft Key Start and you will see the tool movements of the program

- **Changing I/O to floppy drive** (Only need to do this once stays default)

  1. Move the Mode Dial to **EDIT**
  2. Press **System** on the display keys
  3. Page down until you see Parameter (Manual)
  4. Cursor down to the I/O
  5. Type A (for the Floppy Drive) press Input key

Other Drives useable: B (Drive), C (Drive), P (Printer), 1 , 2 (Com Ports)

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

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

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

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

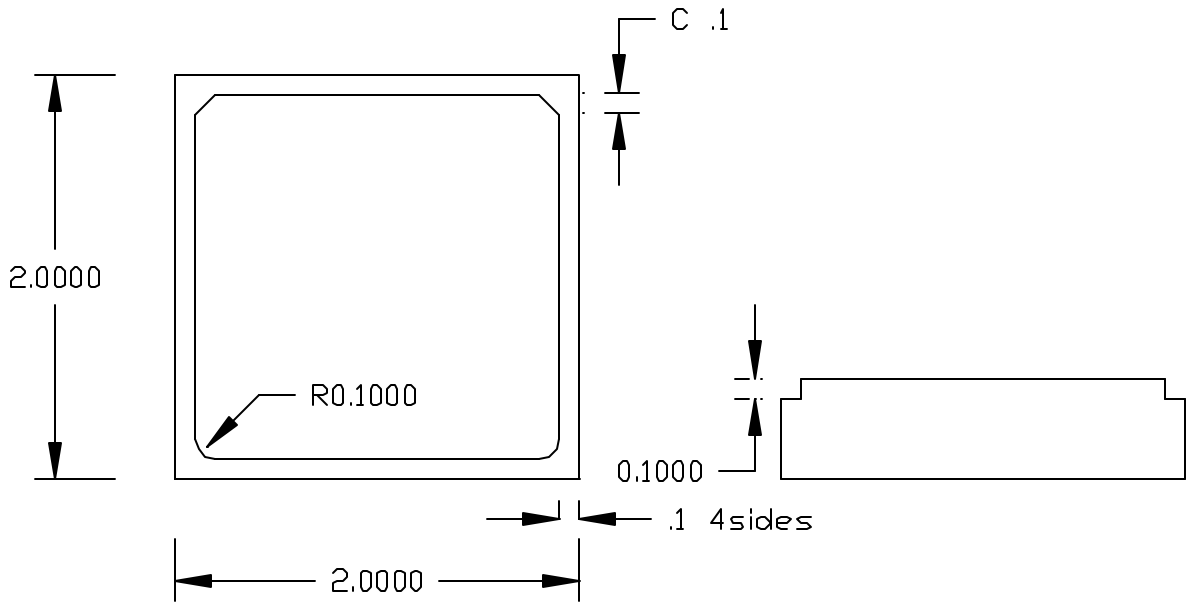
- **Input Program into Fanuc Software from Drive unit**

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

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

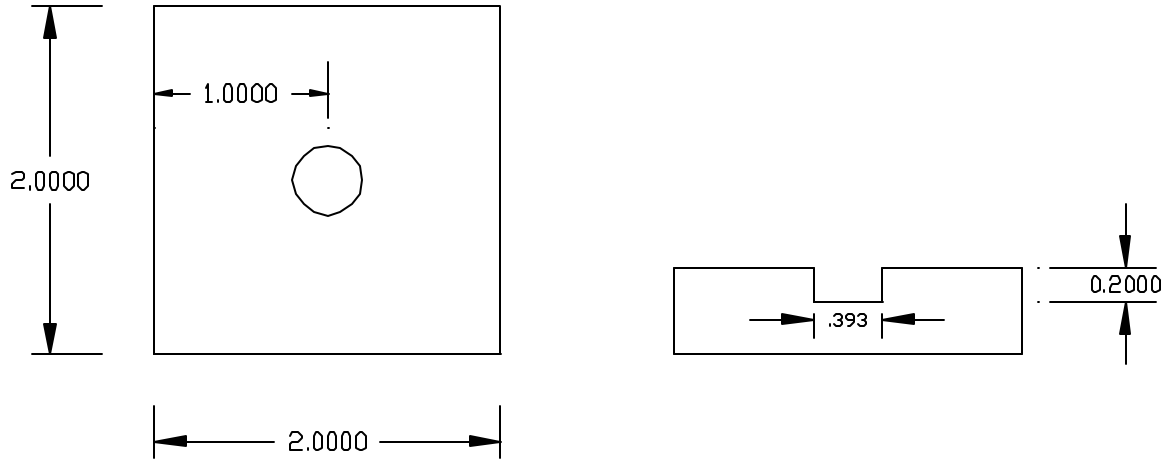
  1. Press the **Offset/Sett** display key
  2. Press (OPRT)
  3. Press the right Arrow key on the Soft keys
  4. Press Read then press Exec

Program O0001 (C & R)



O0001 (Demo 1) (2 X 2 X .5 Alum.)
N5 G00 G17 G40 G80
N10 G90 G94 G98
N15 **G54**
N20 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N25 S1800 M3
N30 G0 Z1
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9 **C.1**
N55 X1.9 **C.1**
N60 Y.1 **R.1**
N65 X.1 **R.1**
N70 Y1
N75 G0 G40 X-1
N80 G28 Z1.5
N85 G28 X2.5 Y2.5
N90 M30

Program O0002 (Deep Hole Drilling)



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

O0002 (Demo 2) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.05

N35 G83 Z-.2 R.1 Q.05 F3

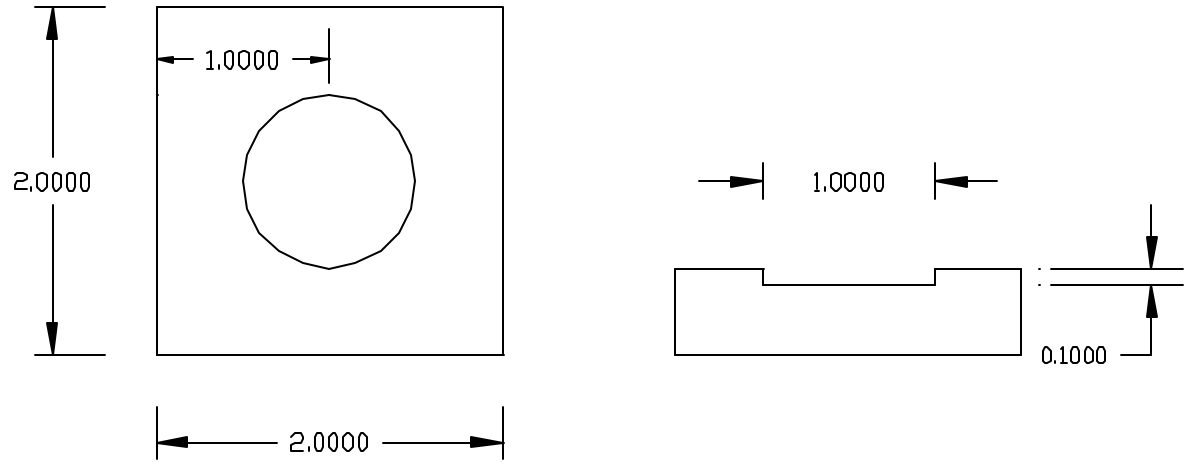
N40 G80

N45 G28 Z1.5

N50 G28 X2.5 Y2.5

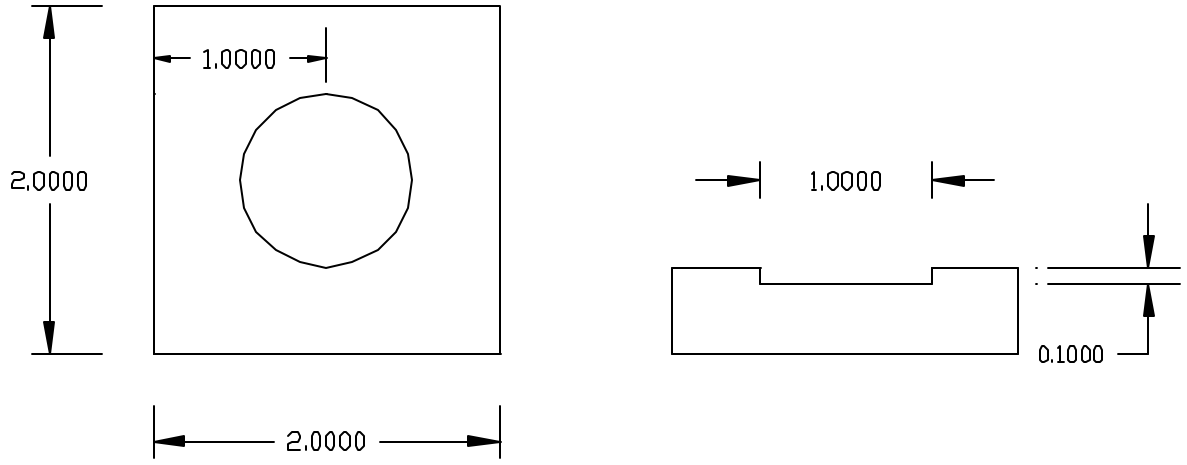
N55 M30

Program O0003 (I & J)



O0003 (Demo 3) (2 X 2 X .5 Alum)
N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F5
N50 G2 X.5 Y1 I.5 J0 (360 degrees)
N55 G0 G40 X1
N60 G28 Z1.5
N65 G28 X2.5 Y2.5
N70 M30

Program O0003 (R)



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

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z-.1 F3

N40 S1800

N45 G1 G42 H11 X.5 F5

N50 G2 X1.5 Y1 R.5 (180 Degrees)

N55 G2 X.5 Y1 R.5 (180 Degrees)

N60 G0 G40 X1

N65 G28 Z1.5

N65 G28 X2.5 Y2.5

N70 M30

1. To make all programs tie together or all programs O0001 thru O0003 to run together. Use M98 this calls out Sub programs or Sub routines.
   Example: M98 P010001
2. After M98 P is identified with 6 digits.
   - The First 2 digits is the number of times program is to be repeated
   - The next 4 digits is the program number without the letter O
3. Programs that are being used as a Sub Programs must end with M99 instead of M30.
4. All programs can be used as Sub Programs or Main Programs M99 means program is Sub, M30 means program is a Main
5. A main Program can also use M99 at the end.
   - Program is being used to repeat without cutting multiple parts.
   - This is mainly used for Demo's for just seeing Tool movements.
6. To link all 3 programs together follow Program O0004
   - Program O0001, O0002(R), O0003 must all have M99 at the end to link together

Program O0004 (Main Program)

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

N5 G54

N10 M98 P010001

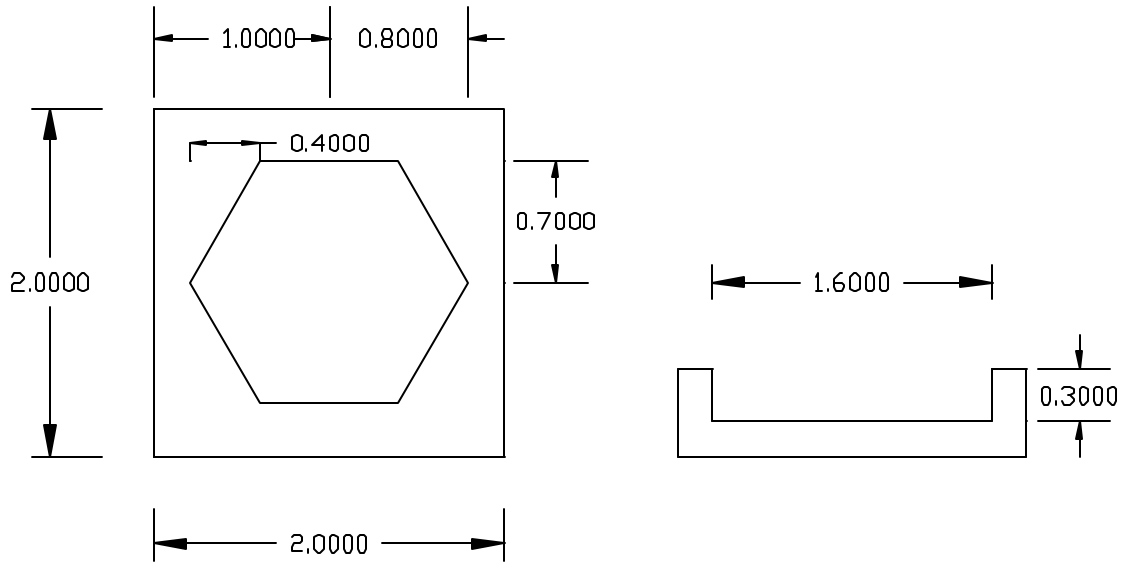
N15 M98 P010002

N20 M98 P010003

N25 M30

Program O0005 (Pocket Milling)

(Making a Cycle)



O0005 (Demo 5) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z0 F3

N40 M98 P030006

N45 G0 G28 Z1.5

N50 G28 X2.5 Y2.5

N55 M30

Program O0006 (Sub for program 5)

O0006 (Sub Prog. for Prog. 5)

N5 G91

N10 G1 Z-.1 F2

N15 G90

N20 S1800

N25 G41 H11 X.4 Y1.35 F7

N30 X.2 Y1

N35 X.6 Y.3

N40 X1.4

N45 X1.8 Y1

N50 X1.4 Y1.7

N55 X.6

N60 X.2 Y1

N65 X.4

N70 X.8 Y.5

N75 X1.2

N80 X1.6 Y1

N85 X1.2 Y1.5

N90 X.8

N95 X.4 Y1

N100 G0 G40 X1

N105 M99