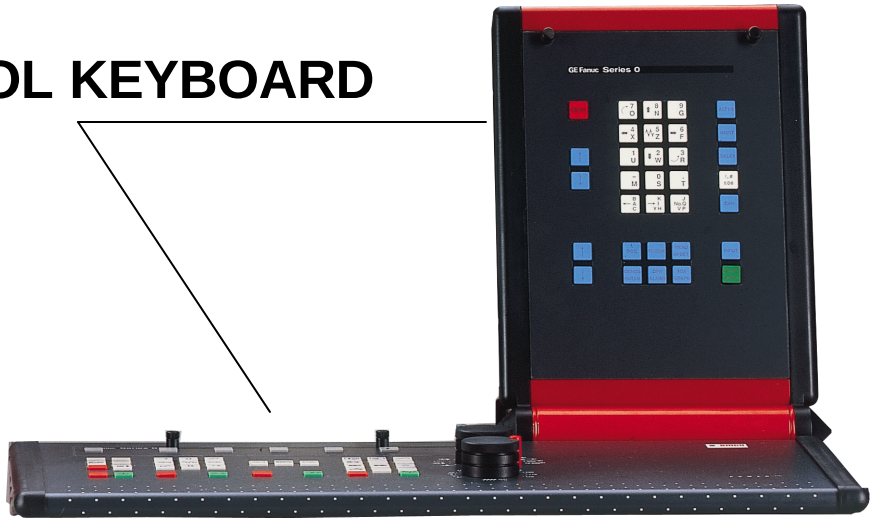
emco
innovative machine tools

GE FANUC O
105 MILL
TRAINING GUIDE

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

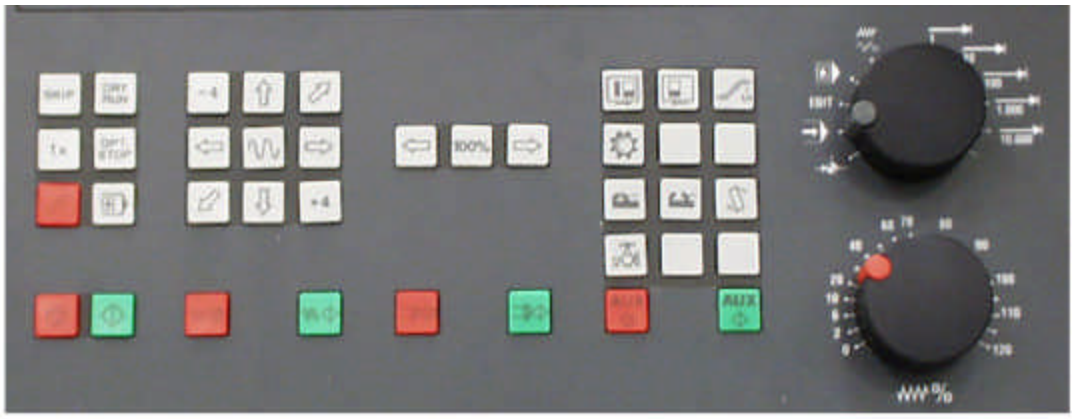
Training Index

CONTROL KEYBOARD

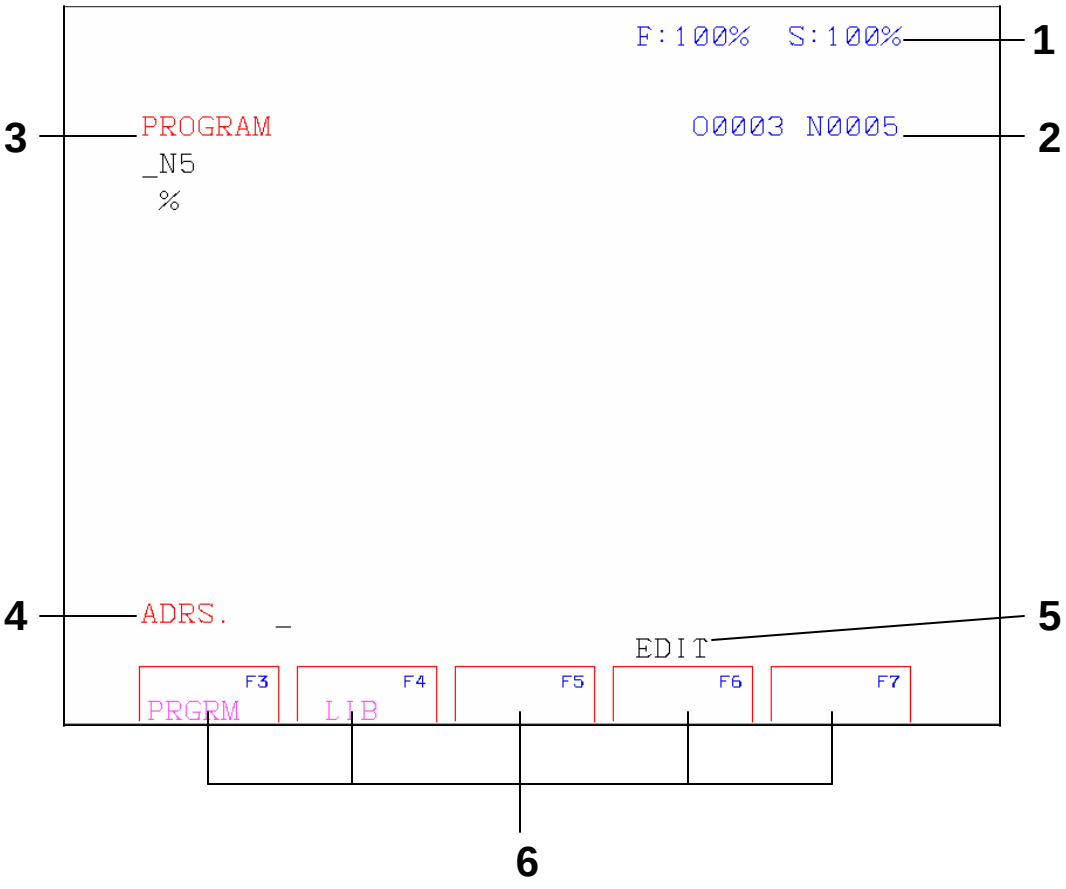


FANUC O CONTROL

MACHINE CONTROL



The Fanuc O Screen



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

FANUC O 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

CHANGE KEYS

ALTER = alter word (replace word)

INSRT = insert word, create new program

DELET = deletes word / block or a program

EOB = end of block, skip block

CAN = deletes entries in the address

STORE KEYS

INPUT = inputs program / offsets / word / numbers

OUTPT / START = sends program / offsets out

DATA INPUT KEYS

Continually press keys to see all possibilities of that Key.

Press one time a letter appears
Press again a number appears

FUNCTION KEYS (DISPLAY KEYS)

POS

POS = displays actual, relative, machine positions

PRGRM

PRGRM = displays program, library page

MENU OFSET

MENU / OFSET = displays Offsets, Work shifts

DGNOS PARAM

DGNOS / PARAM = displays parameters, diagnostic pages

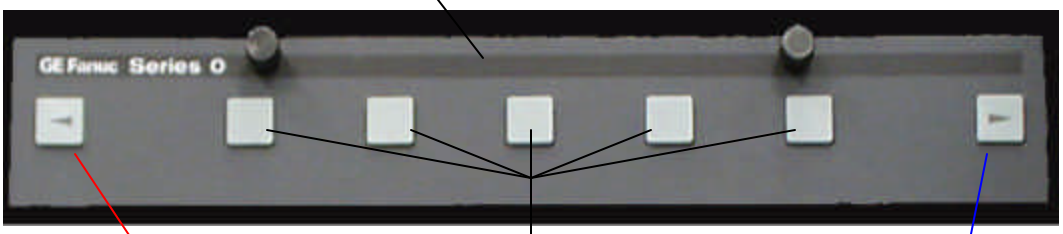
OPR ALARM

OPR / ALARM = displays operator & alarm messages

AUX GRAPH

AUX / GRAPH = displays 2–D graph & 3D simulation

SOFT KEY MODULE



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

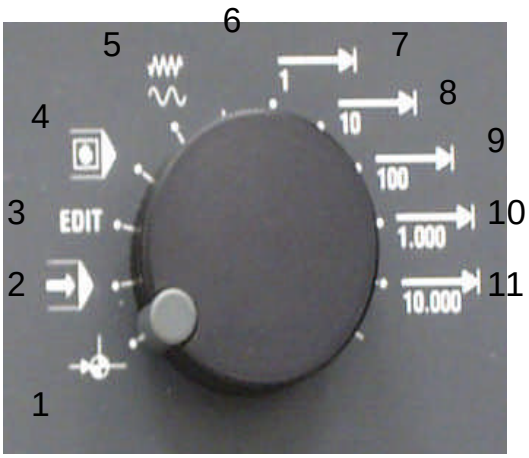
Press once vise closed
Press once vise open

Turret Index

Press once coolant on
Press again coolant off

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

MODE DIAL



(1) ZRN = Zero / Reference or Home mode

(2) AUTO = Automatic mode for running a program

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

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

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

(6) STEPS = Incremental feed movements

(7) STEPS = .0001 or tenths

(8) STEPS = .001 or thousands

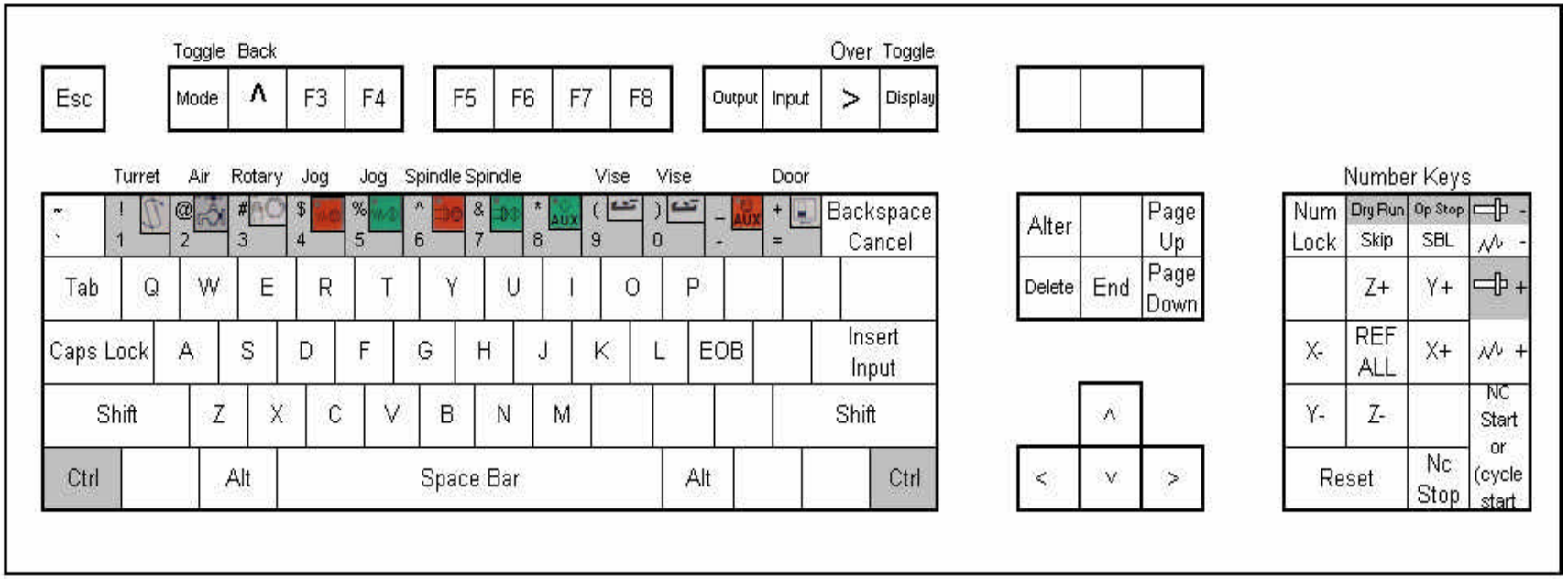
(9) STEPS = .010 or ten thousands

(10) STEPS = .100 or hundred thousands

(11) STEPS = .100 or hundred thousands

FEED OVERRIDE DIAL

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



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

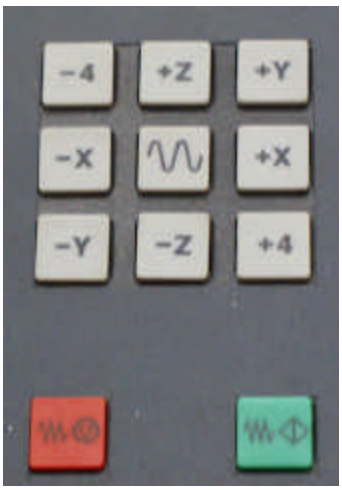
Keys are active they will move the axes if used as numbers. Use numbers on the keyboard.

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 ZRN position also know as Reference make sure your feed rate is not on "**0**"

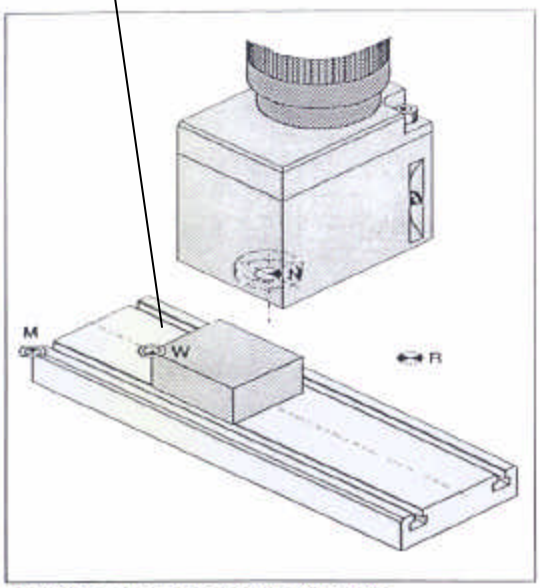
4. Make sure door is closed
5. Press the Z+ this references the Z axis.
6. Press the X- this references the X axis
7. Press the Y- this references the Y axis



Note: Every time you enter Fanuc O 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

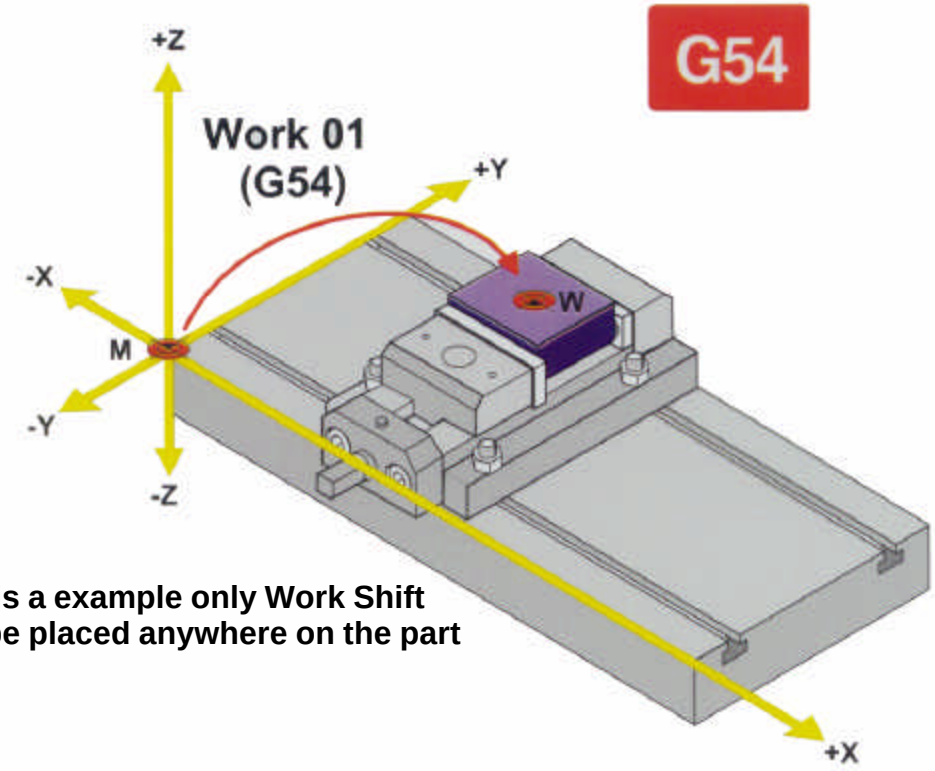
Reference Points of the EMCO Milling Machines

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.

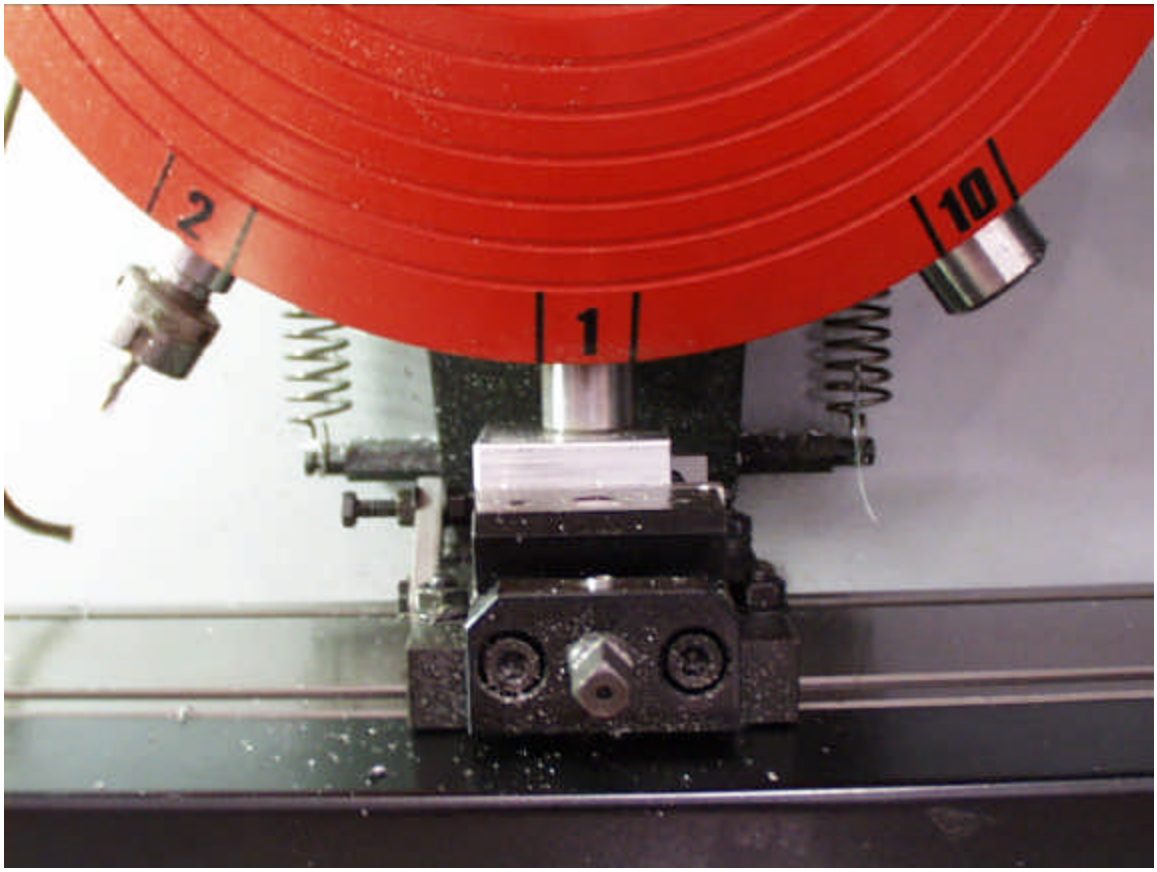


This is a example only Work Shift can be placed anywhere on the part

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 piece of paper between nose and Work Piece)

NOTE: Use the Stump that has been provided with the Machine

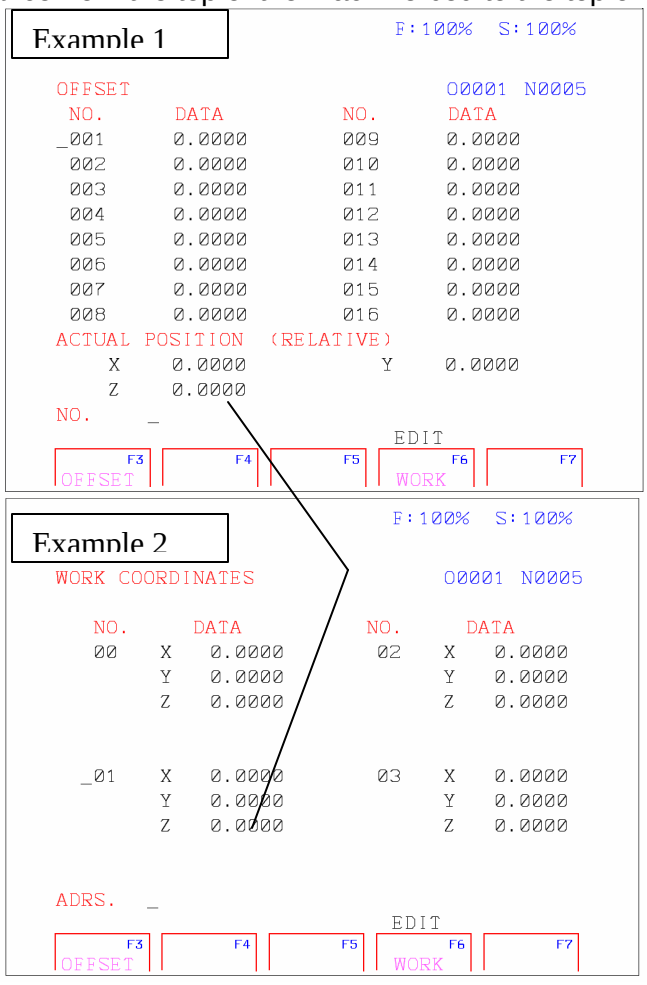


3. Press the MENU/OFSET button 

4. Press the WORK Soft key (Gray Button) Example 2
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
6. Press the OFFSET Soft key (Gray Button)
    - Example 1 in the picture below
    - Record the value in the Actual Position Relative Z
7. Press the WORK Soft key (Gray Button) Example 2
8. Move Cursor to 01 location
9. Recorded value type in Work Coordinates 01(Z) which is G54

    Example: Type Z 2.463 press Input button 

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

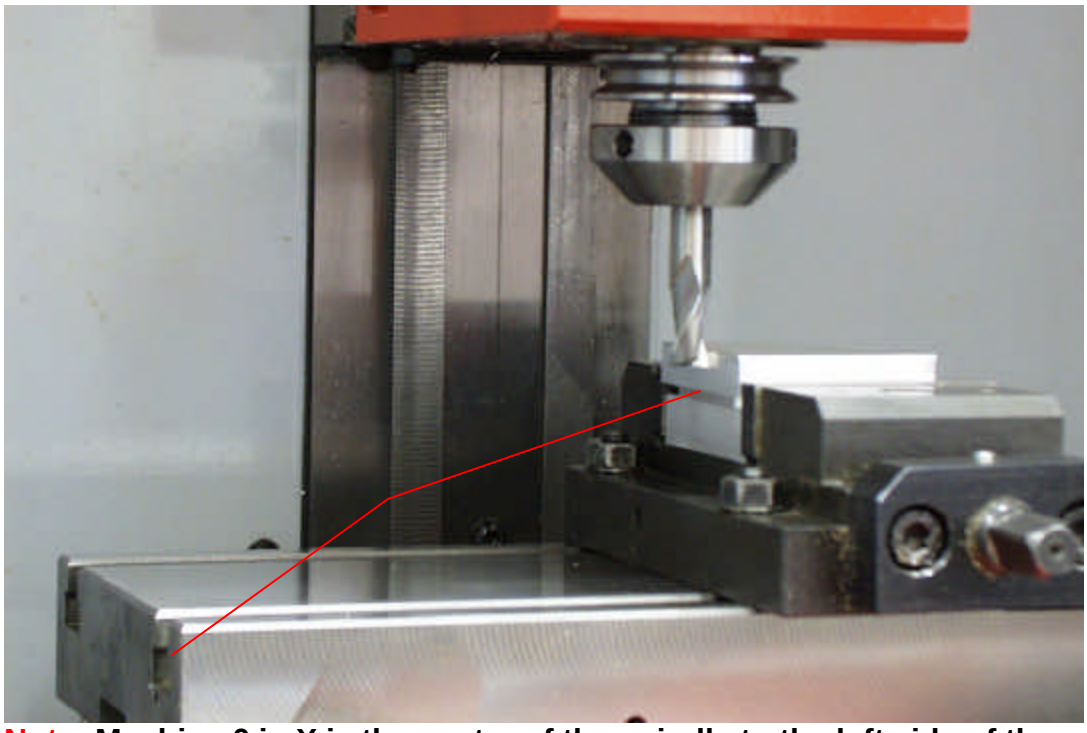


Note: Machine 0 is the spindle nose touching the top of the Machine bed.

10. Jog Spindle up away from WORK PIECE using Z+
11. Either follow step 12 or follow step 13 when finished go on to step 14
12. Index to a edge finder or tool (Ex. 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)
13. For Scratching move MODE Dial to MDI

    - Press the PROGRAM display button PRGRM until top of the screen shows MDI (Program)
    - Type S1000 INPUT M03 INPUT then  cycle start
    - Then type T1 INPUT M6 INPUT then  (Door must be closed)

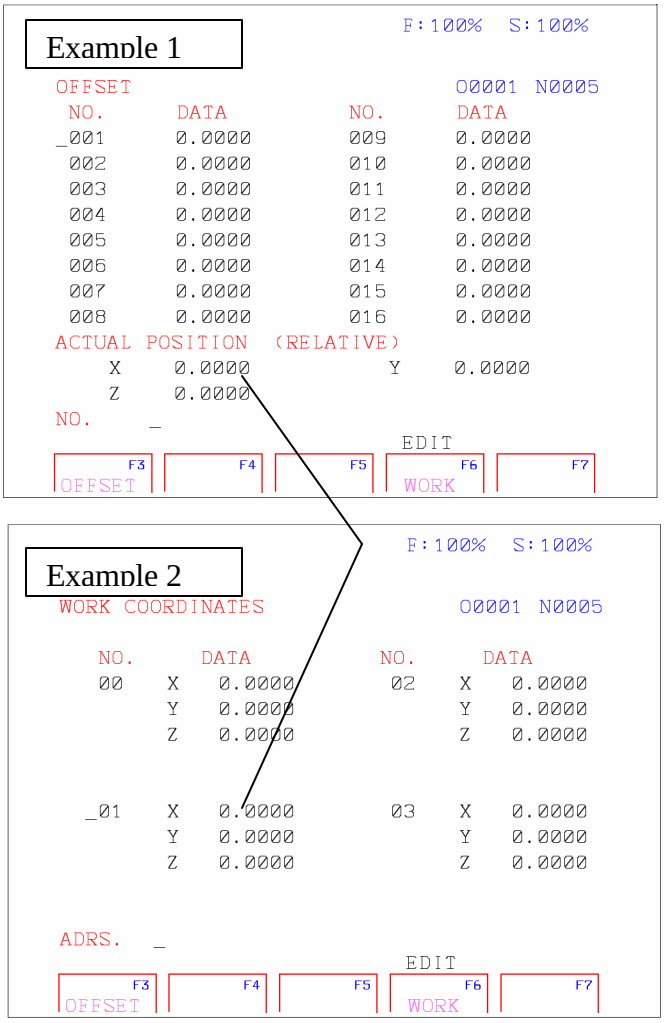
      S=RPM M03=Spindle on Clockwise T1=Tool Position M6=Index
    - Move MODE Dial to Jog then Jog the Tool to the left side of the Work Piece & touch using the Direction keys.



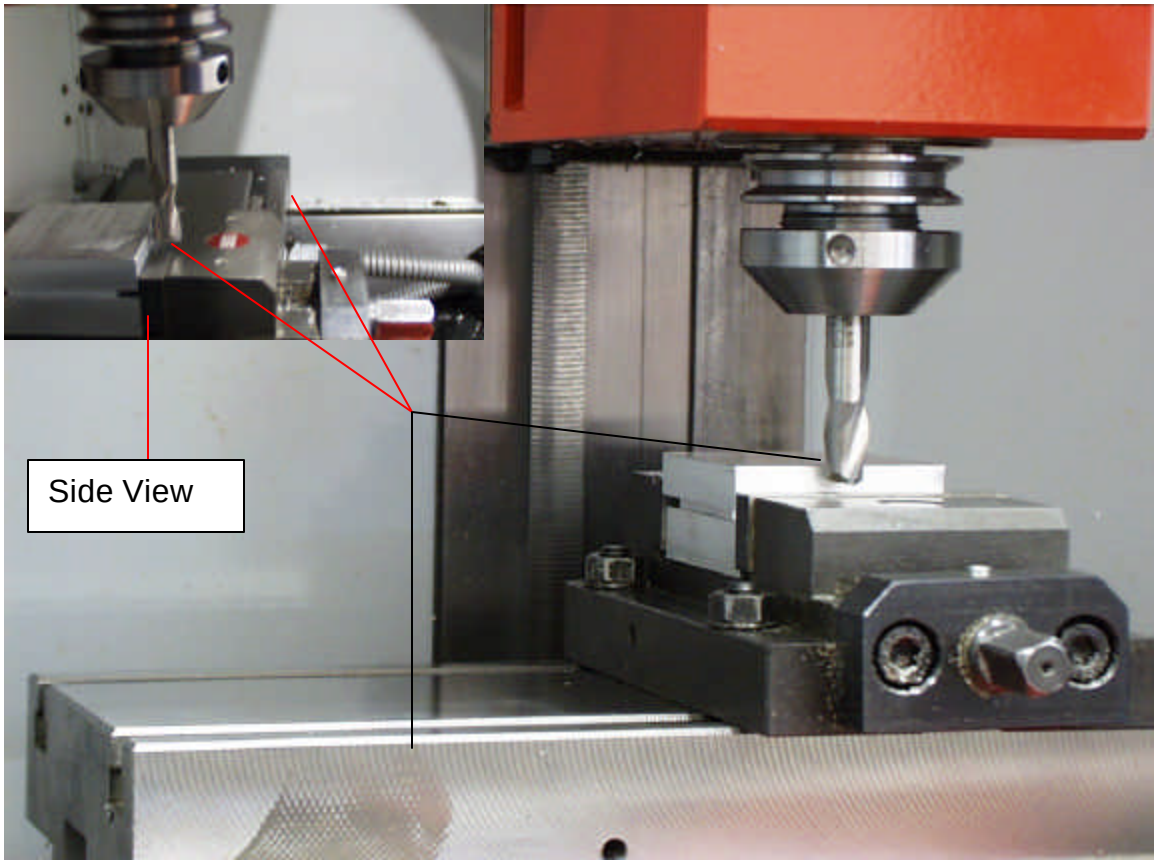
Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

14. Press the MENU/OFSET button 

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



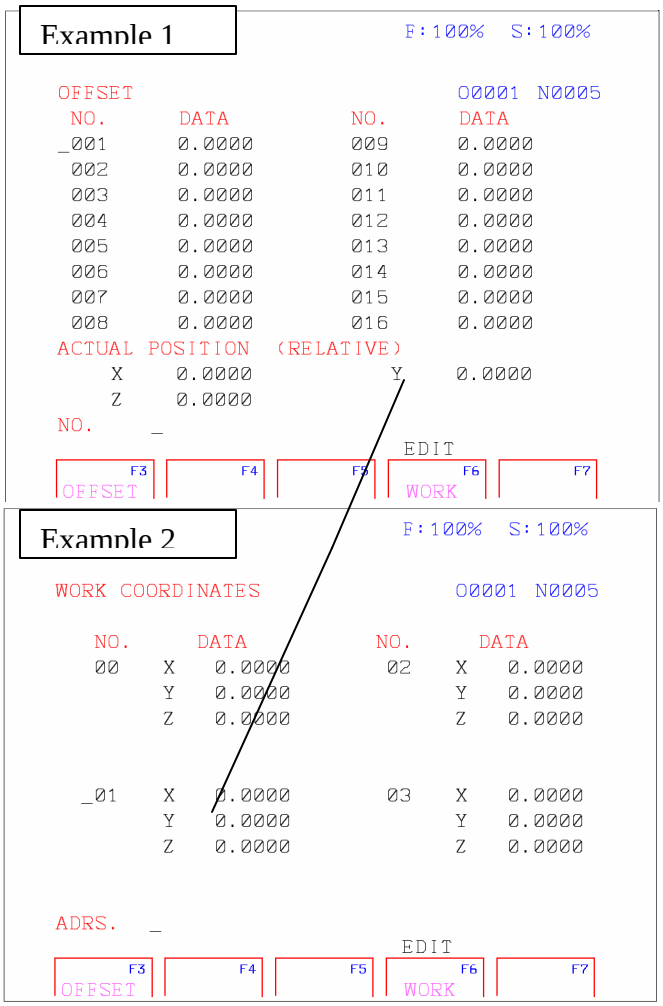
18. Jog Spindle up away from WORK PIECE using Z+
19. Jog the Tool to the Front of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps to approach at a slower feed)



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

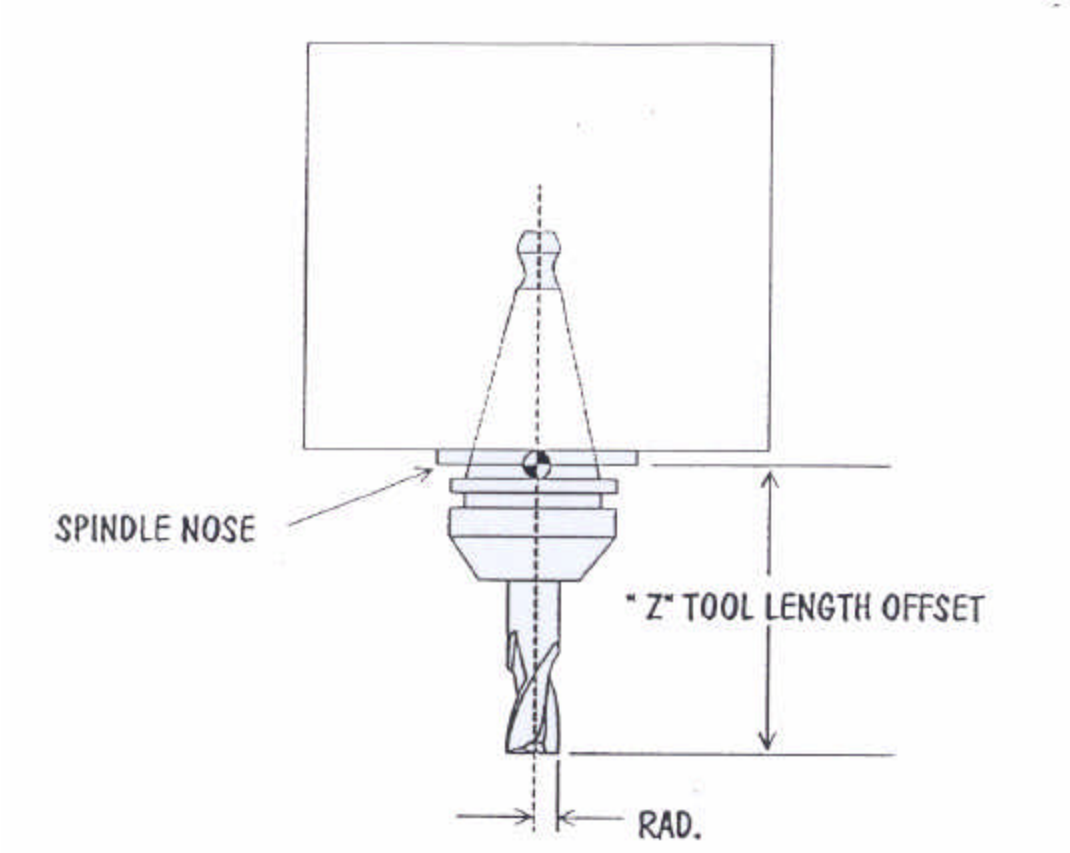
20. Press the MENU/OFSET button 

    - Example 1 in the picture below
    - Record the value in the Actual Position Relative Y
21. Press the WORK Soft key (Gray Button) Example 2
22. Move Cursor to 01 location
23. The Recorded value plus the radius of the tool being used to scratch (3/8) type in Work Coordinates 01 (Y)



24. Jog the Tool up above the Work Piece using Z+

TOOL OFFSET



F:100% S:100%

OFFSET O0001 N0005

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

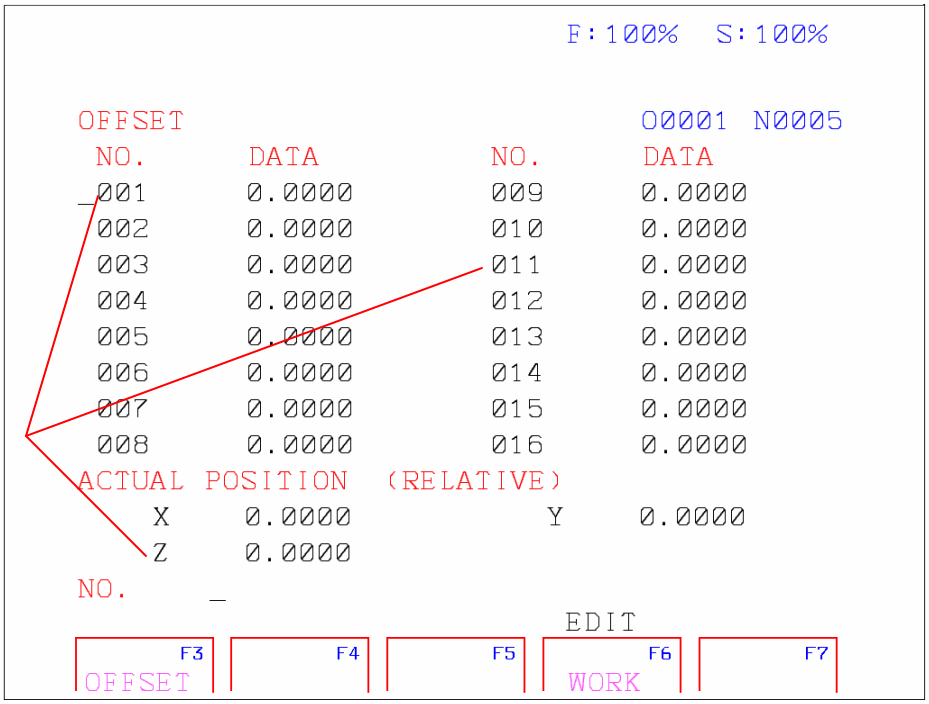
NO. _

EDIT

F3 OFFSET | F4 | F5 | F6 WORK | F7

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 MENU/OFSET 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 4
   - Drills & Taps need no Radius set for them

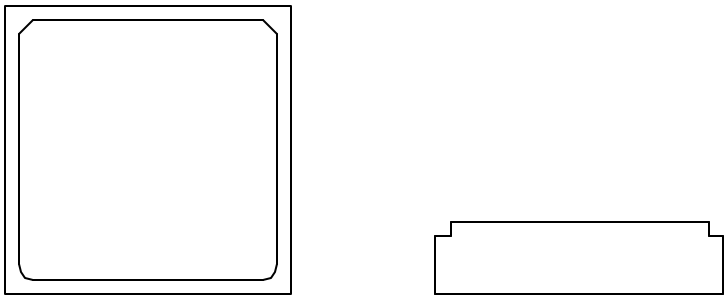


NOTE: When you use a T the H = Height

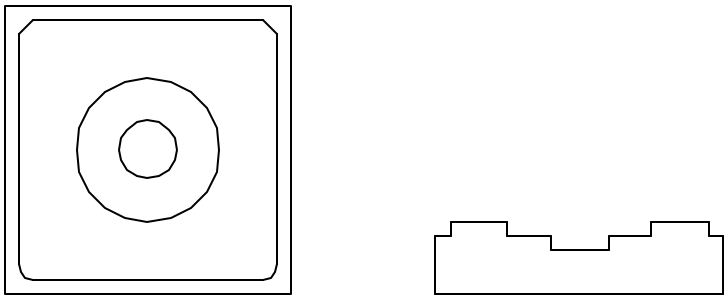
When you use a G41 or G42 the H = Radius

Program Training

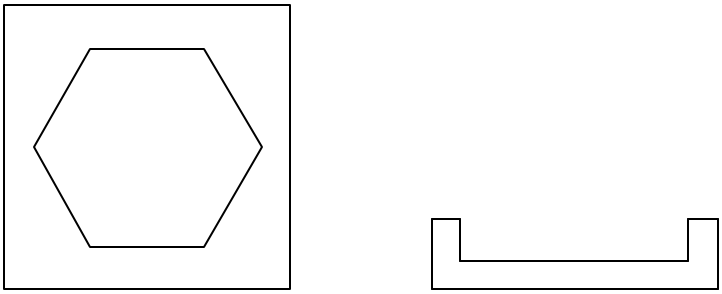
Program O0001



Program O0003



Program O0005



Change the Mode Dial to Edit & Press the 

to do functions below & on the next 2 Page

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

     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 

     Example: press once letter O appears press again number 7 appears

     HINT: When inserting a word place the cursor one word on the left before the place being inserted

     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 ONLY.
USE INPUT ALL OTHER APPLICATIONS.

- **DELETE A PROGRAM**

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

     Example: **O0001 OR O1**

- **DELETE ALL PROGRAMS**

  1. Press letter o plus the – & 9999

  2. Press delete button 

     Example: **O – 9999**

- **DELETE A WORD**

  1. Press letter then number
  2. Press delete button 

     Example: press once S appears press again 0 appears

     **HINT: ** Deleting a word; place the cursor on the left side before the word being deleted

     Example: BEFORE N5_S1000; AFORE N5;

     (S1000) is the word being deleted?

- **DELETE A BLOCK OR LINE NUMBER**

  1. Type the number line
  2. Press delete button 

     Example: _N10 G0 X1.0 F.003; make sure cursor is on the line being deleted (_N10)

- **CANCEL MISTYPED WORD**

  1. Press cancel button 

  HINT: In the ADRS. (Address) at the lower left of the screen is the word and numbers that you typed in. Before pressing insert check if what was typed in is correct. If not press cancel and retype word and numbers.

- **ALTER A WORD**

  1. Type the Word needed altered
  2. Press alter button 

  Example: Make sure the cursor is to the left of the words being altered (_N5 CHANGE TO _N10)

- **SEARCH FOR NUMBER BLOCK**

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

  Example:(N50)

  HINT: The arrow button pointing down

- **SEARCH FOR WORD**

  1. Type in Word & number **Example: (M30)**

  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 Deselect miller radius compensation
G41 Miller radius compensation left
G42 Miller radius compensation right
G43 Tool length compensation positive
G44 Tool length compensation negative
G49 Deselect tool length compensation
G53 Machine coordinate system
G54 Zero point shift 1
G55 Zero point shift 2
G56 Zero point shift 3
G57 Zero point shift 4
G58 Zero point shift 5
G59 Zero point shift 6
G73 Chip break cycle
G80 Delete drilling cycle (G83 to G85)
G81 Drilling cycle
G83 Excavation drilling cycle
G90 Absolute value programming
G91 Incremental value programming
G94 Feed in inch/min
G95 Speed with feed in inch/revolution
G97 Spindle speed per minute
G98 Retract to plane of start (drilling cycles)

Survey commands M CODES: Mostly used

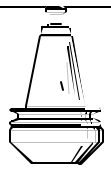
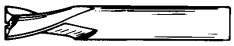
M00 Programmed stop, unconditional
M01 Programmed stop, conditional
M03 Spindle ON clockwise
M04 Spindle ON counter clockwise
M05 Spindle OFF
M06 Tool change
M08 Coolant ON
M09 Coolant OFF
M19 Orientated spindle stop
M25 Release clamping device
M26 Close clamping device
M30 Main program end with new 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

F1Z 910	**Collet holder**	For ESX-16 collets	
152 800	**(9.0-10.0mm)Æ 3/8”**	**ESX 16 COLLETS**	
764 308	Acc. to DIN 327, shape B cutting-ø10 mm / shank-ø10mm	**Slot end mill, HSS**	

Program screen & Edit mode

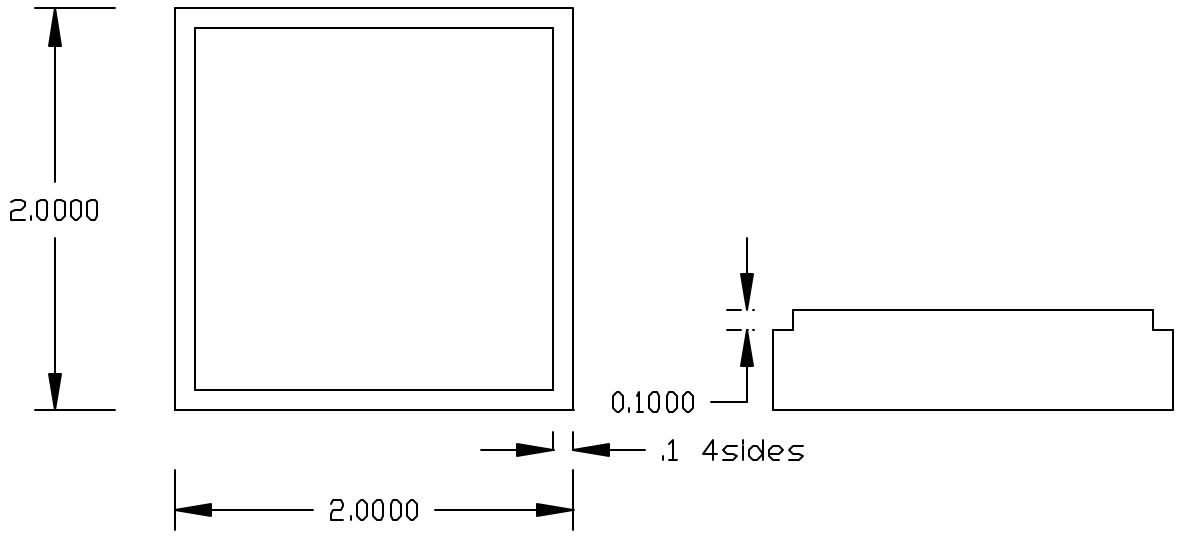
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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

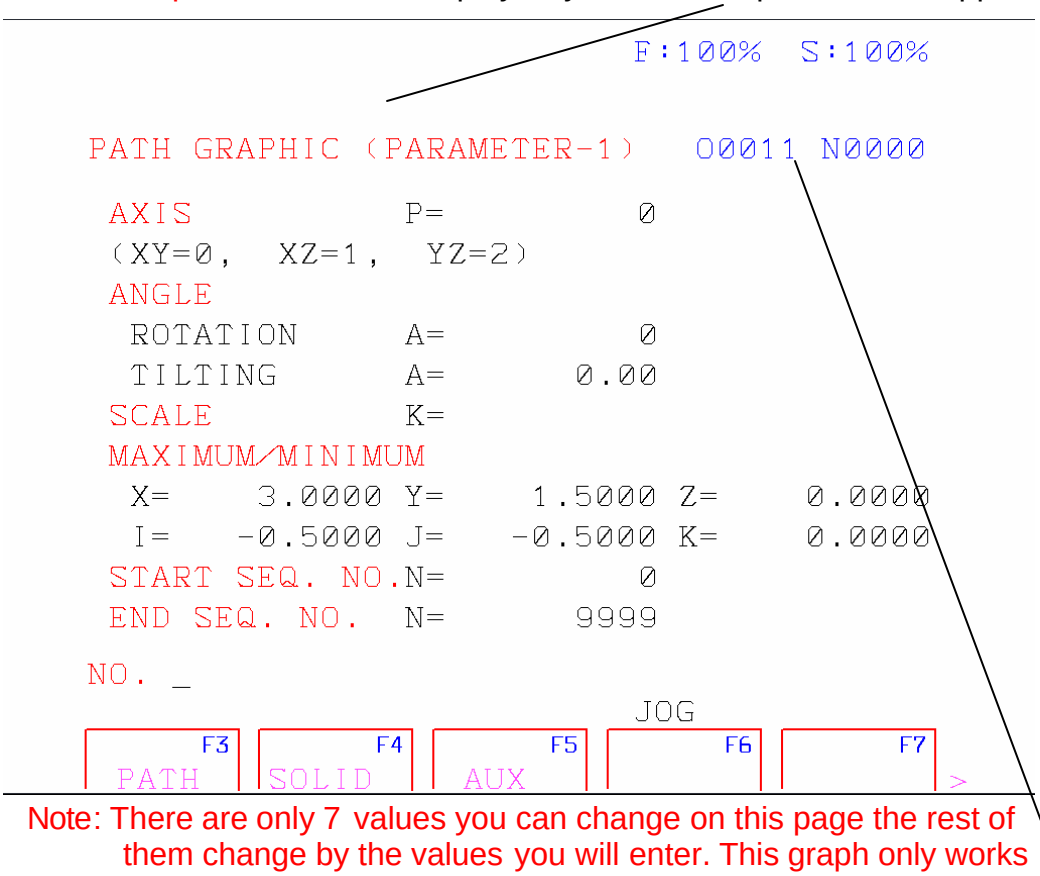
Program O0001



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

2D Simulation

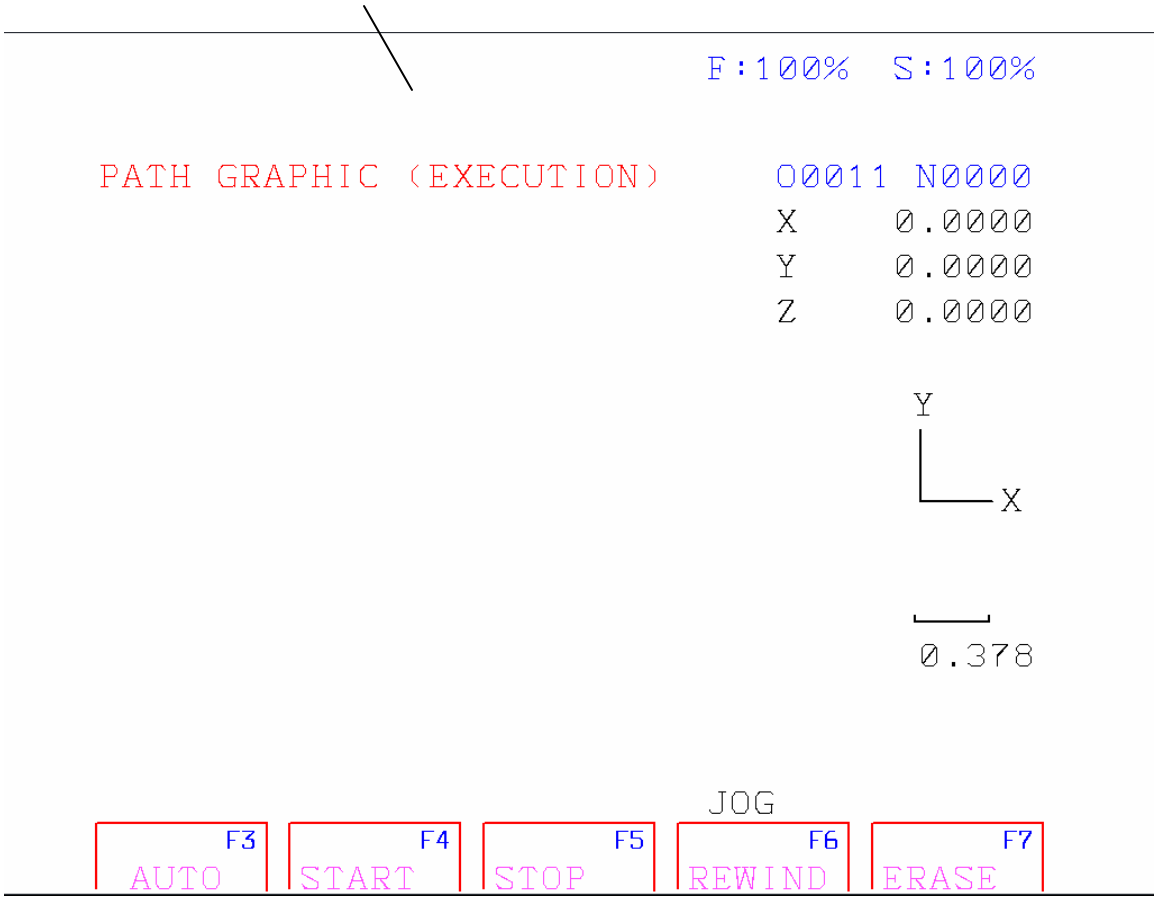
1. Press Graph button on the Display Keys for the Graph screen to appear



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 PATH
10. Press the Soft key EXEC for Execution screen



Note: If you press the AUTO 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.

11. 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 **Parameter** on the display keys
    3. Page down until you see Parameter (Setting 1)
    4. Cursor down to 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 (**Output Start)** key

- **Output Offsets from Fanuc software to Drive unit**
    1. Press the **Menu Offset** display key
    2. Press (**Output Start)** key

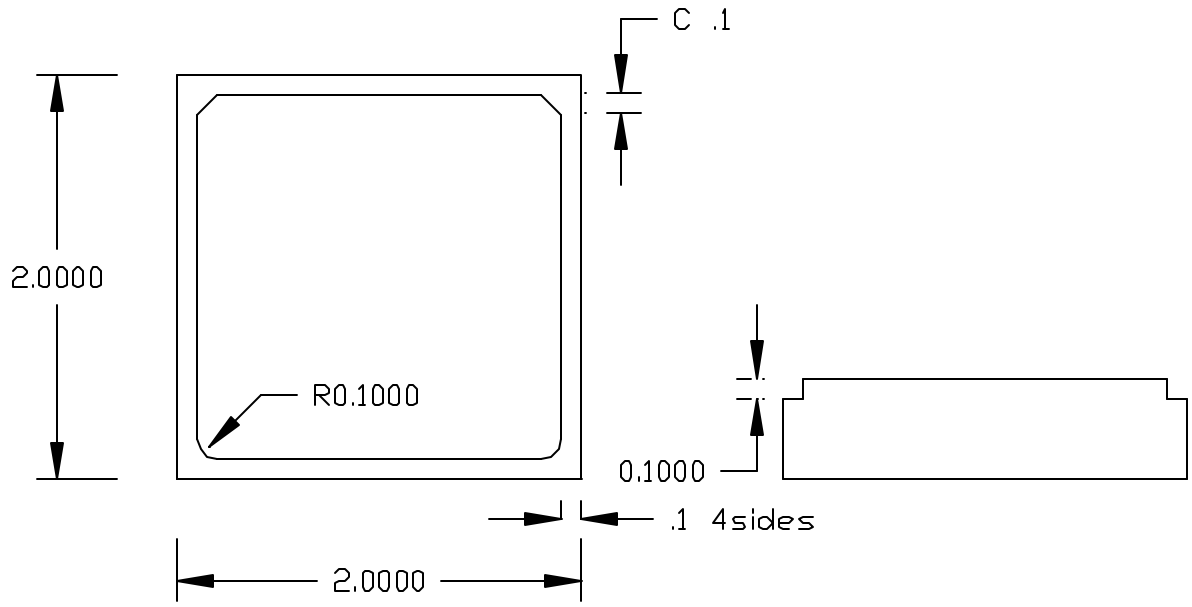
- **Input Program into Fanuc Software from Drive unit**
    1. Press the **Program** display key
    2. Type program number to be read

       Example: letter O and program number (O0002) or (O2)

    3. Press (**Input)** key

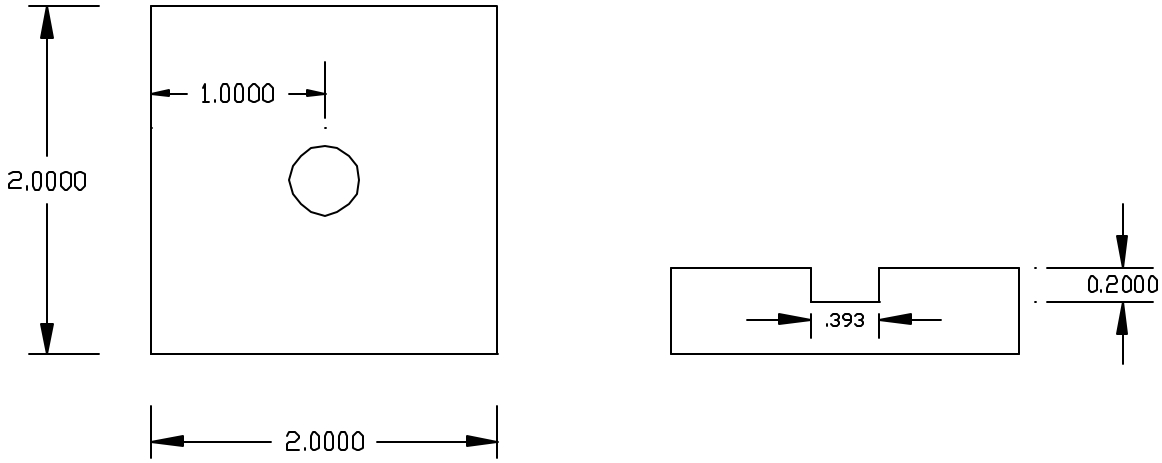
- **Input Offsets into Fanuc Software from Drive unit**
    1. Press the **Menu Offset** display key
    2. Press **(Input)** key

Program O0001 (C & R)



N5 G00 G17 G40 G80 (Demo 1) (2 X 2 X .5 Alum.)
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**

N5 G54 (Demo 2) (2 X 2 X .5 Alum.)

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

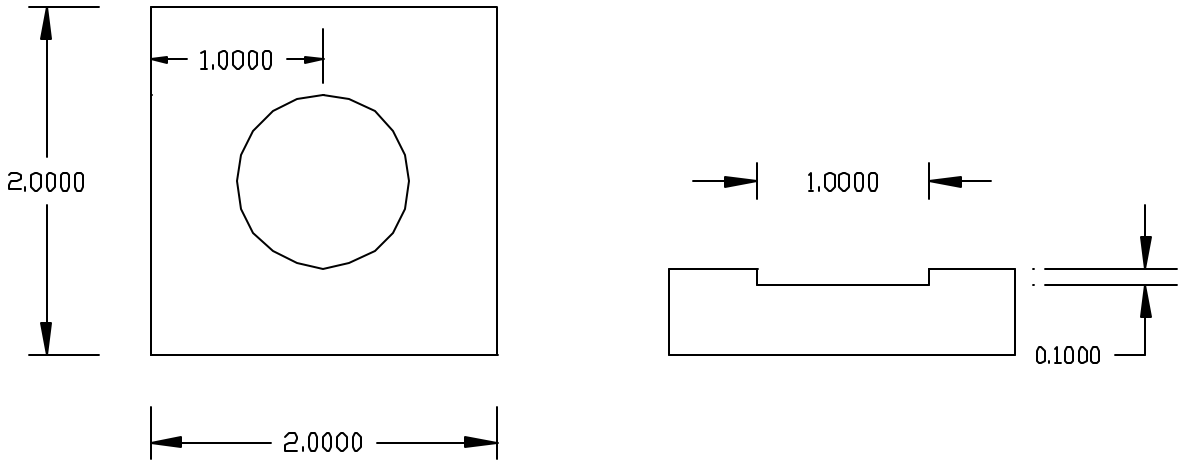
N40 G80

N45 G28 Z1.5

N50 G28 X2.5 Y2.5

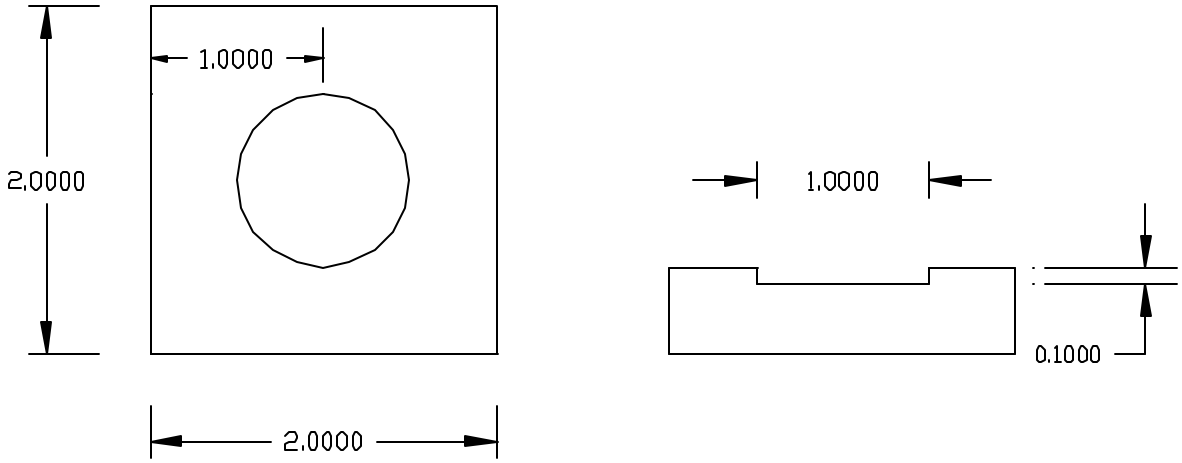
N55 M30

Program O0003 (I & J)



N5 G54 (Demo 3) (2 X 2 X .5 Alum.)
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)



N5 G54 (Demo 3) (2 X 2 X .5 Alum.)

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)

N5 G54 (Tie Prog. 1,2,3 together)

N10 M98 P010001

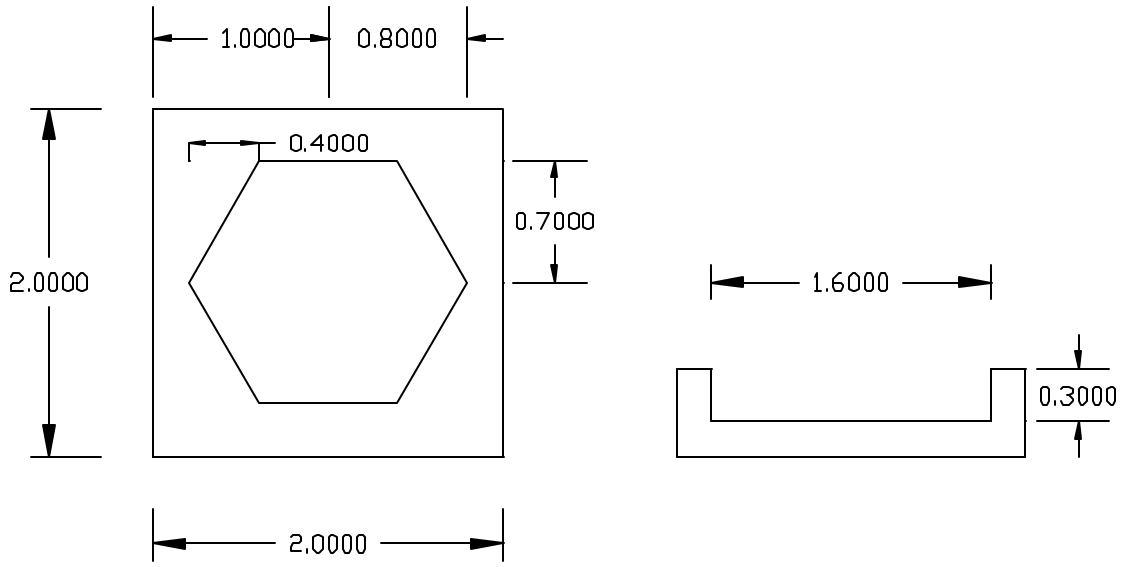
N15 M98 P010002

N20 M98 P010003

N25 M30

Program O0005 (Pocket Milling)

(Making a Cycle)



N5 G54 (Demo 5) (2 X 2 X .5 Alum.)

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)

N5 G91 (Sub Prog. for Prog. 5)

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