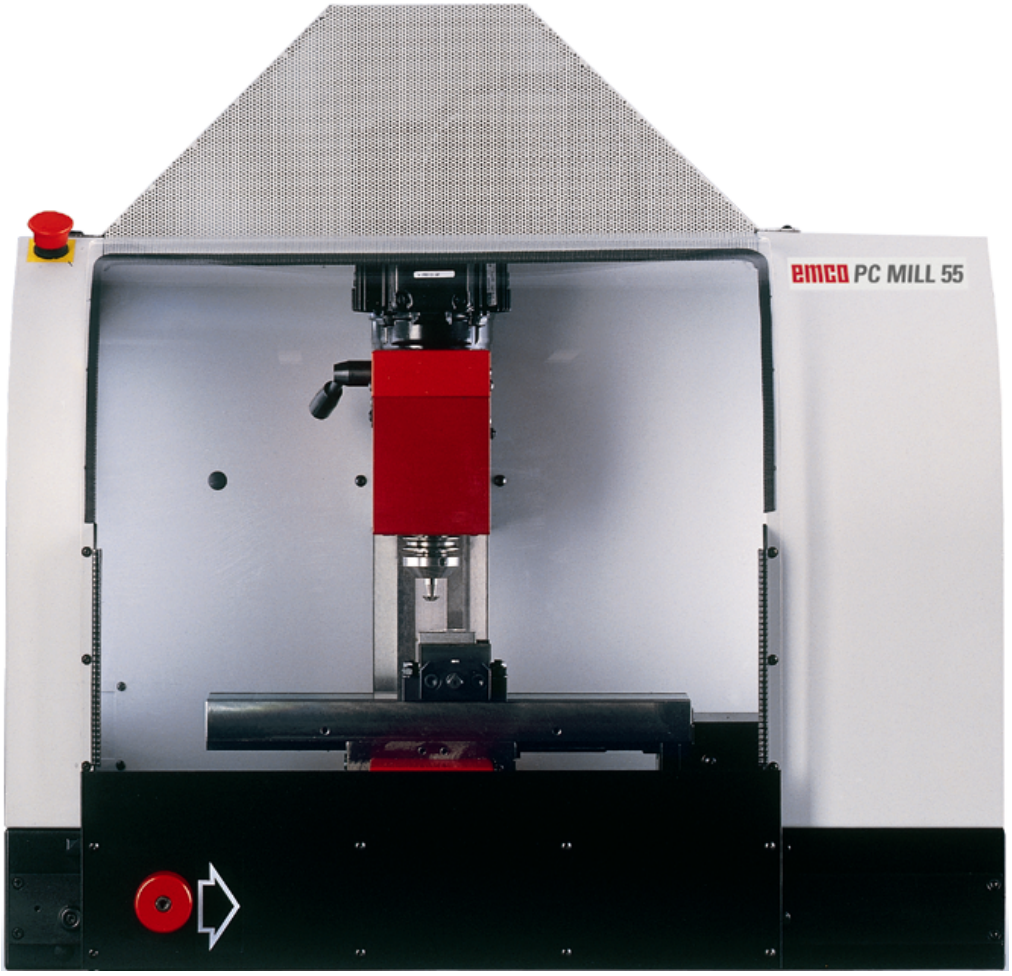
innovative machine tools



GE FANUC O
50/55 MILL
TRAINING GUIDE
ON PC KEYBOARD

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

TRAINING INDEX

Fanuc 0 Screens …… Pg 1

Fanuc 0 keys …….. Pg 2

- **Cursor Movement keys**
- **Change keys**

Function keys (Display keys) ……... Pg 3

- **Machine Function keys**

Direction keys …….. Pg 4

- **Spindle Overrides keys**
- **Accessory Functions**

Mode Dial …… Pg 5

- **Feed Override Dial**

Pc Keyboard picture …… Pg 6

Reference the machine …….. Pg 7

Work Shift (Description) …… Pg 8

Work Shift for Z …… Pg 9

Work Shift for X …… Pg 11

- **Manually index the Tool Turret**
- **Manually turning on Spindle**

Work Shift for Y …… Pg 13

Tool Offsets (Description) …… Pg 15

Setting Tool Offset …… Pg 16

Program Training …….. Pg 17
Insert a New Program …….. Pg 18
- **Call a Existing Program up**
- **Insert a Code**
- **Insert a End of Block**

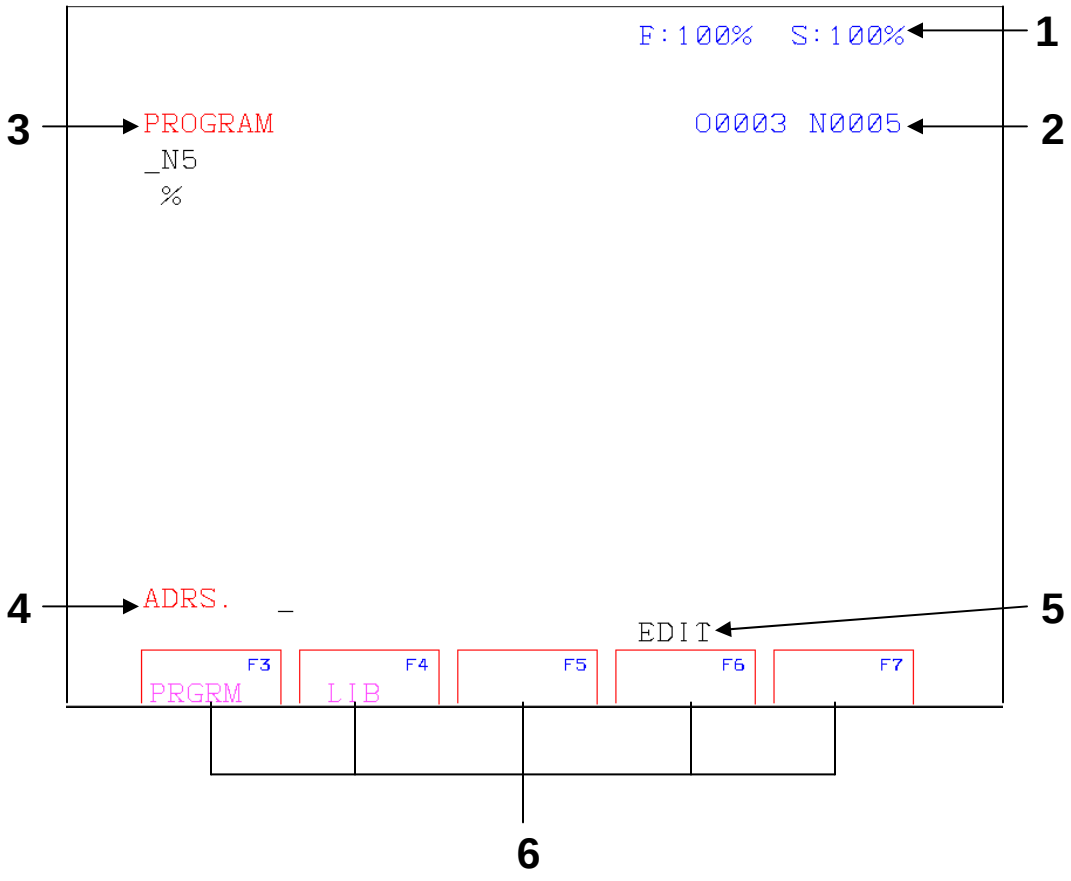
Delete a Program …….. Pg 19
- **Delete all Programs**
- **Delete a Code**
- **Delete a Block or Line number**

Cancel Mistyped Codes ……... Pg 20
- **Alter a Code**
- **Search for Code**
- **Search for Letter**

G Codes …….. Pg 21
M Codes …….. Pg 22
- **Used Addresses**

Program O0001 …… Pg 24
2D simulation (setup) …….. Pg 25
Input and Output the Programs & offsets thru the Fanuc software Pg 27
Program O0001 (C/R) …….. Pg 28
Program O0002 (Deep hole Drilling) …… Pg 29
Program O0002 (I/J) …… Pg 30
Program O0003 (R's) …… Pg 31
Program O0004 (Sub Programming) …….. Pg 32
Program O0005 (Pocket Milling) …….. Pg 33
Program O0006 (Sub For program 5) …… Pg 34

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 ON PC KEYBOARD

Number 0 on # keys is RESET = cancels most alarms, resets program, interrupts programs

CURSOR MOVEMENT KEYS

Arrow key pointing up is CURSOR UP = moves cursor up

Arrow key pointing right = moves cursor right

Arrow key pointing left = moves cursor left

Arrow key pointing down is CURSOR DOWN = moves cursor down, search function, program call up

PAGE UP = moves one page up

PAGE DOWN = moves one page down

CHANGE KEYS

Insert key is the ALTER = alter word (replace word)

Enter is the INSRT = insert word, create new program

DELETE = deletes word / block or a program

Enter pressed twice is EOB = End of a block or line

Back space is the CAN = deletes entries in the address

STORE KEYS

F10 is INPUT = inputs programs & offsets

F9 is OUTPUT = sends program & offsets out

FUNCTION KEYS (DISPLAY KEYS)

F12 TOGGLES THE MENU FOR THE DISPLAY KEYS

F12- F3 is for POS = displays actual, relative & all positions

F12- F4 is for PRGRM = displays program, library page

F12- F5 is for OFFSET = displays offset & work pages

F12- F6 is for PARAM = displays parameters & diagnostic pages

F12- F7 is for ALARM = displays operator & alarm message

F12- F11- F3 is for GRAPH = displays 2–d graph simulation

MACHINE FUNCTION KEYS

 # keys is the same as Numeric keypad or 10 key

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

Press Ctrl & / on # keys = (DRY RUN) Test run without spindle on (Remove raw material from chuck)

Press Ctrl & * on # keys = (Optional stop) for programs with (m1)

Press 0 on the # keys = (Reset) cancels most alarms, resets program, interrupts programs

Press * on # keys = (Single block) reads one block at a time

Press . on # keys = (Cycle stop) program hold, feed hold

Press Enter on # keys = (Cycle start) program start

DIRECTION KEYS

These keys control axis directional movements

1 on the # keys moves Y axis -
2 on the # keys moves Z axis -
4 on the # keys moves X axis -
6 on the # keys moves X axis +
8 on the # keys moves Z axis +
9 on the # keys moves Y axis +

Ctrl & 4 = Feed stop
Ctrl & 5 = Feed start
Both works in all modes but EDIT & ZRN

SPINDLE OVERRIDE KEYS

Ctrl & + on the # keys increase the spindle speed (50% to 120% highest)

Ctrl & - on the # keys decrease the spindle speed (120% to 50% lowest)

Ctrl & 6 = Spindle stop

Ctrl & 7 = Spindle start

Works in all modes except EDIT & ZRN

ACCESSORY FUNCTIONS

Press Ctrl & + for Door open

press again Door closed

Press Ctrl & 3 for Rotary axis Indexing

Press Ctrl & 0 vise open

Press Ctrl & 9 vise closed

Press Ctrl & 2 coolant on

press again coolant off (only 100/125/155)

Press Ctrl & 2 puff blowing on

press again puff blowing off (only 50/55)

Press Ctrl & 8 auxiliary drives on

Press Ctrl & - auxiliary drives off

MODE CONTROL

F1 TOGGLES THE MENU FOR THE MODE CONTROL

F1 THEN F7 = ZRN for Reference or Home mode

F1 THEN F3 = AUTO for Automatic mode for running a program

F1 THEN F4 = EDIT mode for program changes or entering a new program

F1 THEN F5 = MDI for Manual Data Input mode for manually programming and running the machine

F1 THEN F6 = JOG for Manual moving the axis in X, Y, or Z

F1 THEN F11 = STEPS Incremental feed movements

F1 THEN F11 THEN F3 = STEPS 1 OR .0001 or tenths

F1 THEN F11 THEN F4 = STEPS 10 OR .001 thousands

F1 THEN F11 THEN F5 = STEPS 100 OR .010 ten thousands

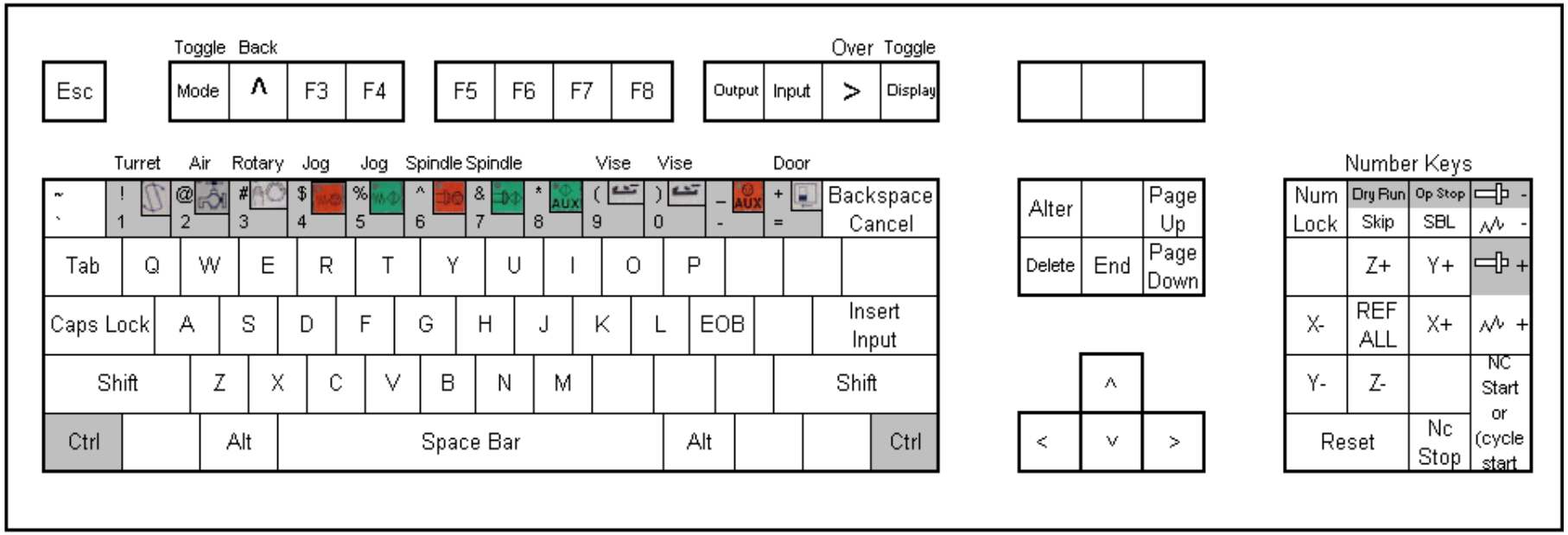
F1 THEN F11 THEN F6 = STEPS 1000 OR .100 hundred thousands

FEED OVERRIDE CONTROL

+ on the # keys increase the feed rate speed (0% to 120%)

- on the # keys decrease the feed rate speed (120% to 0%)

These Control feed for jogging in the X, Y, Z axis



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

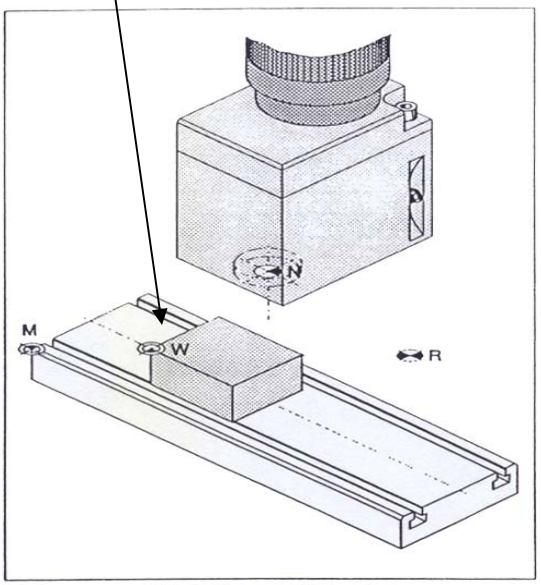
Referencing the Machine

1. **Make sure your feed rate is not on “0”**
2. **Make sure door is closed**
3. **Follow steps 4, 5, 6 or just do 7**
4. **Press 8 on the # keys this references the Z axis.**
5. **Press 4 on the # keys this references the X axis**
6. **Press 1 on the # keys this references the Y axis**
7. **Press 5 on the # keys this references all axis**

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

WORK SHIFT

Pages 8 – 16 is setting the Work shift & offsets to the lower left corner & the top of the part with the Spindle nose



Reference points in the working area

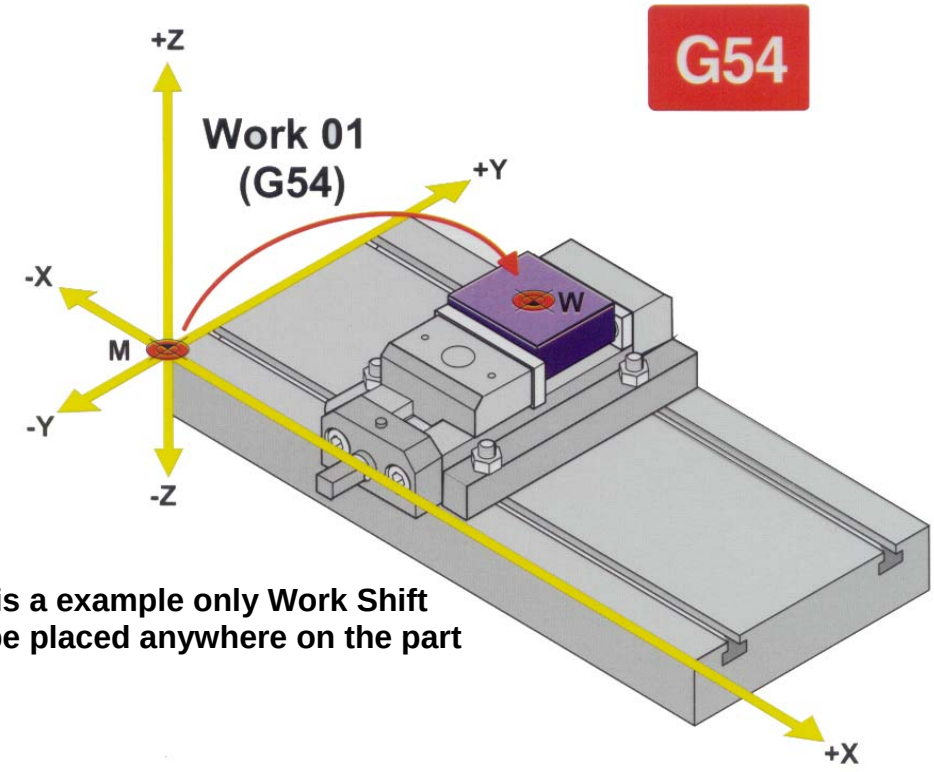
Reference Points of the EMCO Milling Machines

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

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

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

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



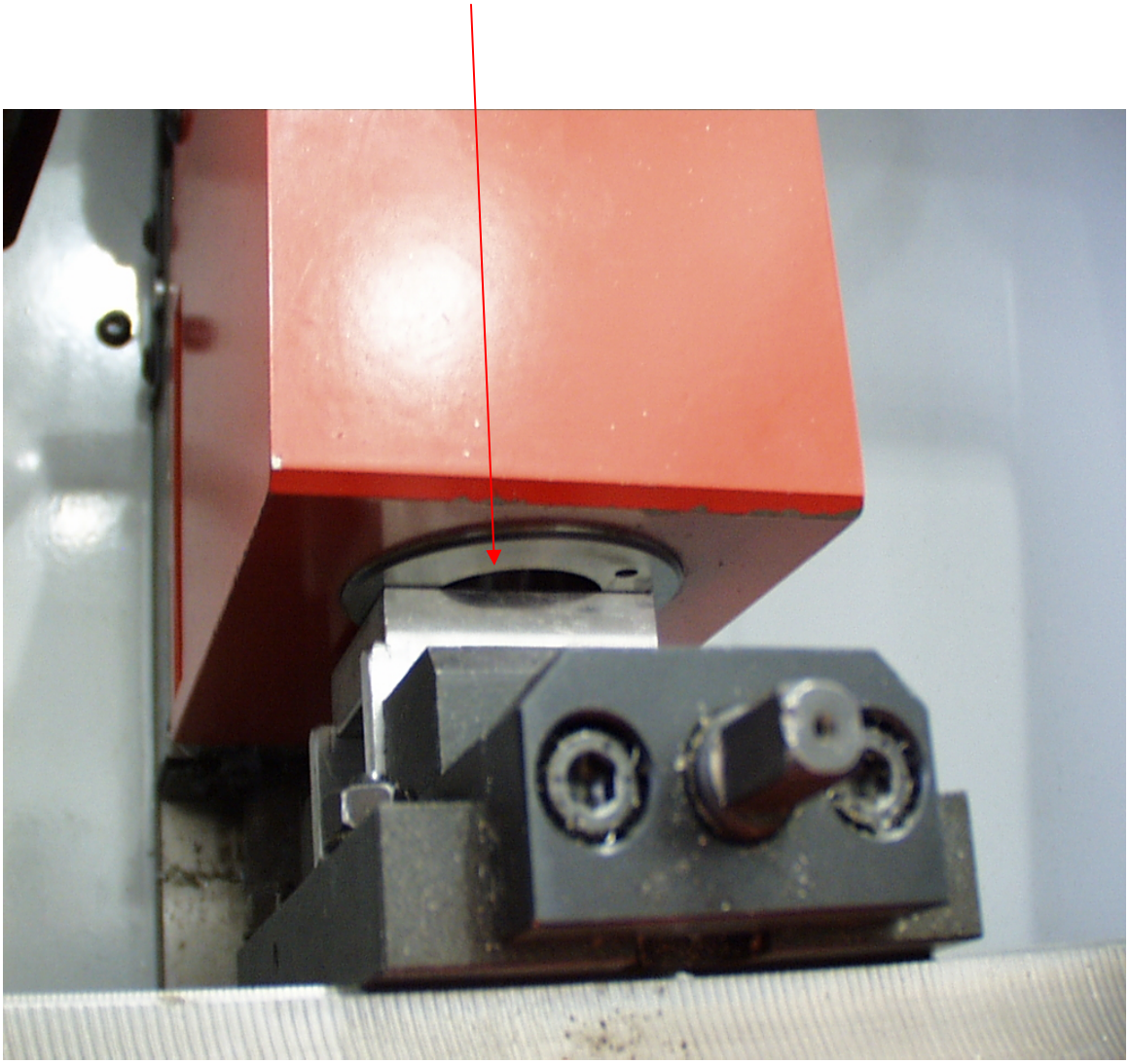
This is a example only Work Shift can be placed anywhere on the part

Work Shift:

1. Press F1 then F6 for JOG mode
2. Jog the Spindle nose to the top of the Work Piece & touch using the # keys 1,2,4,6,8,9.

   (Use Feed -, + or Steps to approach at a slower feed rate)

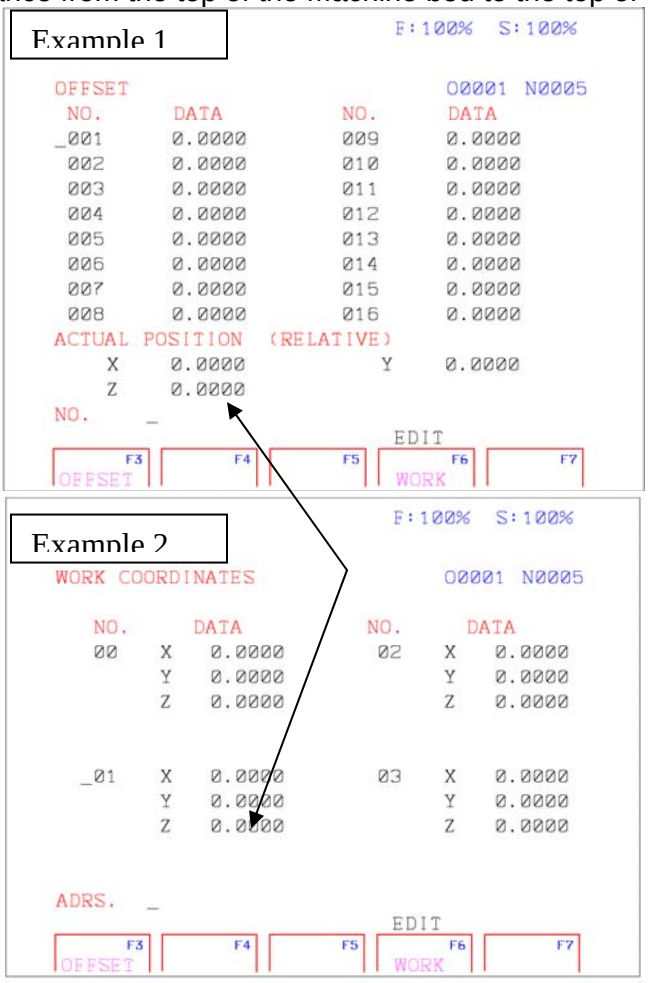
   (Use piece of paper between nose and Work Piece)



3. Press F12 then F5 for OFFSET screen
4. Press F6 for WORK 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 F3 for OFFSET
    - Example 1 in the picture below
    - Record the value in the Actual Position Relative Z
7. Press F6 for WORK Example 2
8. Move Cursor to 01 location
9. Recorded value type in Work Coordinates 01(Z) which is G54

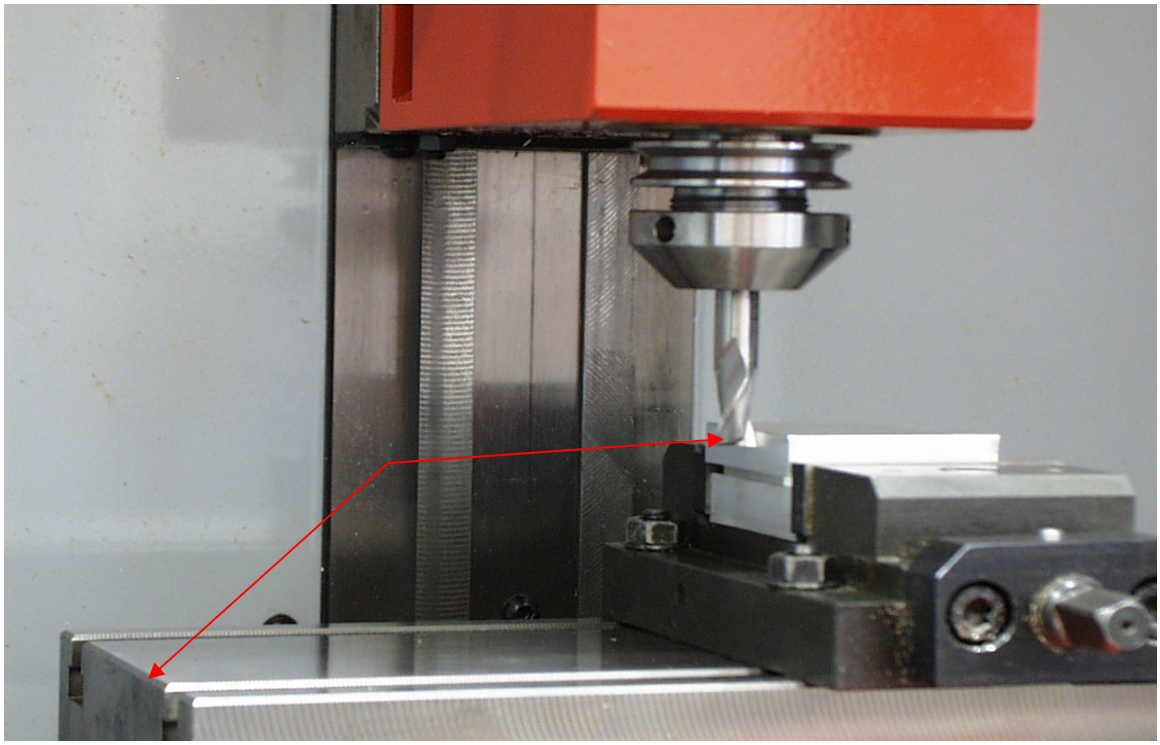
    Example: Type Z 2.463 press Enter

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



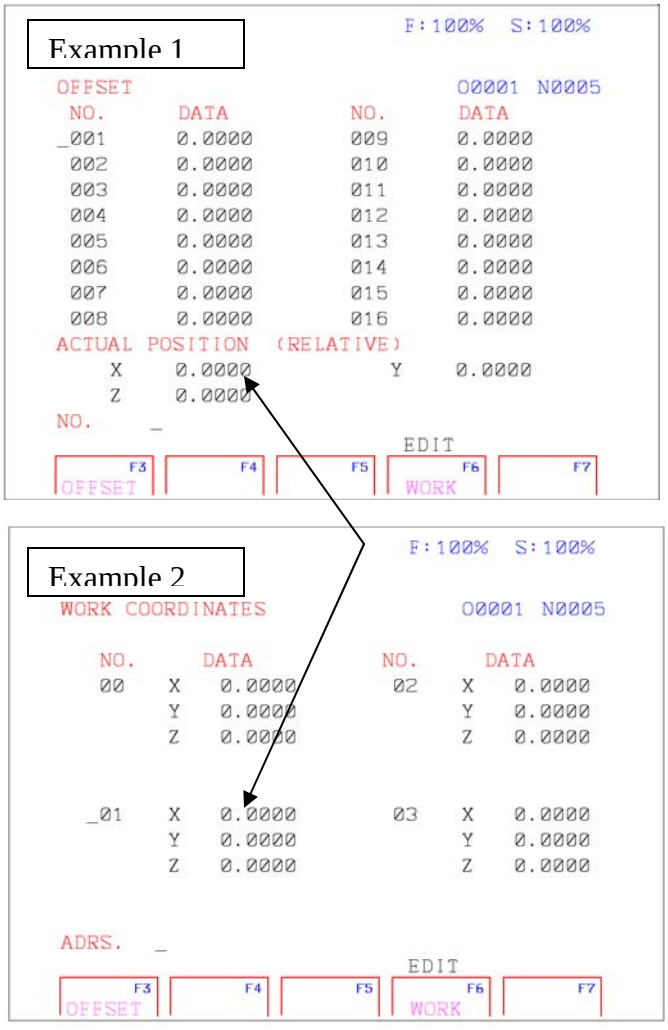
Note: Machine 0 is the spindle nose touching the top of the Machine bed.

10. Jog Spindle up away from WORK PIECE using 8 on # keys
11. Place a edge finder or a tool in the Spindle (Ex. use 3/8 endmill)
12. Either Follow step 13 or follow step 14 then go to step 15
13. Jog the Tool to the left side of the Work Piece & touch using 1,2,4,6,8,9 on the # keys. (Use Feed Dial or Steps to approach slower)
14. For Scratching: Press F1 then F5 for MDI Mode
    - Press F12 then F4 for PROGRAM display press until top of the screen shows MDI (Program)
    - Type S1000 for spindle speed then type M03 Spindle on Clockwise press Enter
    - Then press Enter on the # keys (Make sure door is closed)
    - Press F1 then F6 for Jog mode
    - Jog the Tool to the left side of the Work Piece & touch using 1,2,4,6,8,9 on # keys.

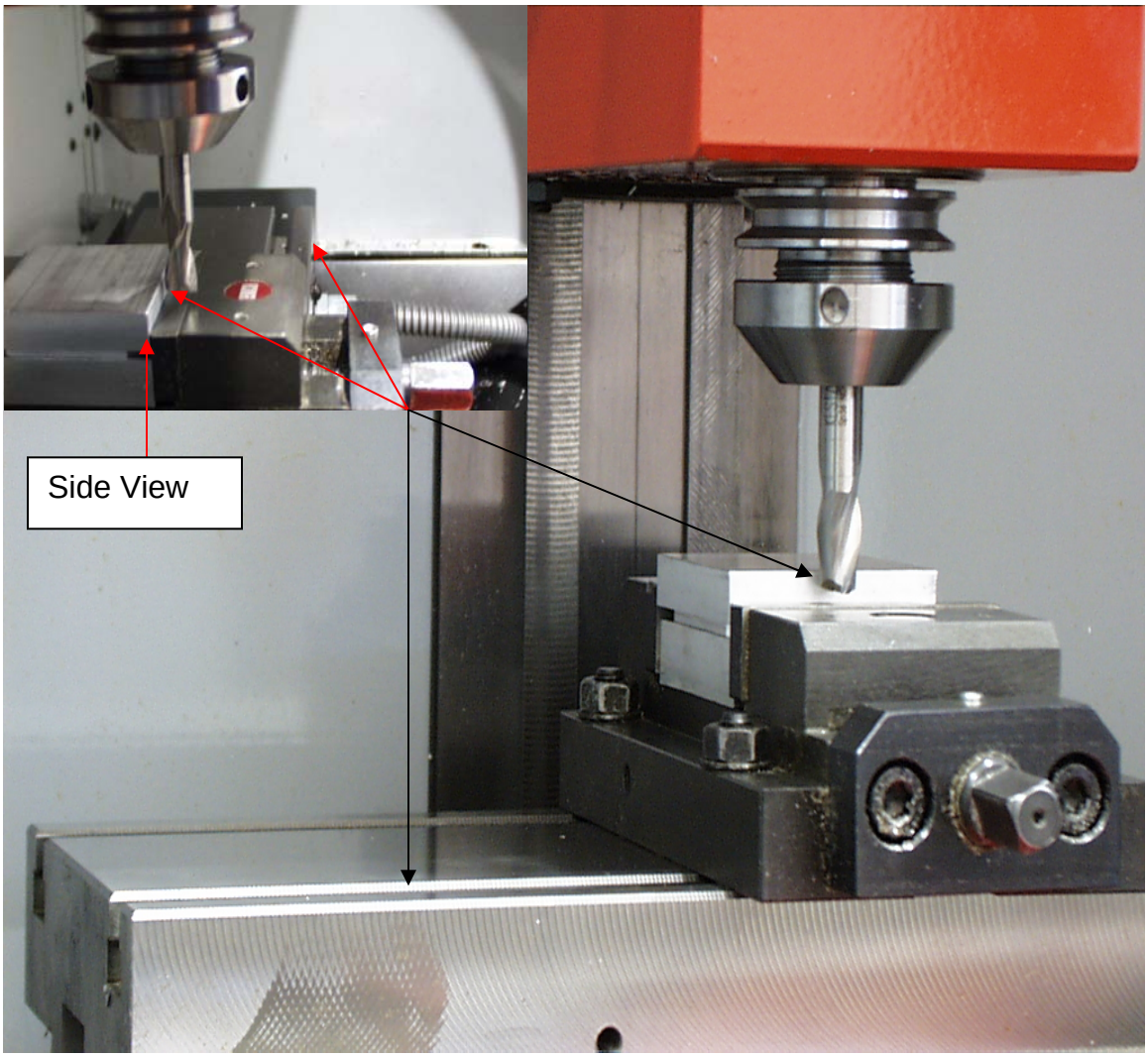


Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

15. Press F12 then F5 OFFSET SCREEN
    - Example 1 in the picture below
    - Record the value in the Actual Position Relative X
16. Press F6 for WORK Example 2
17. Move Cursor to 01 location
18. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (X)

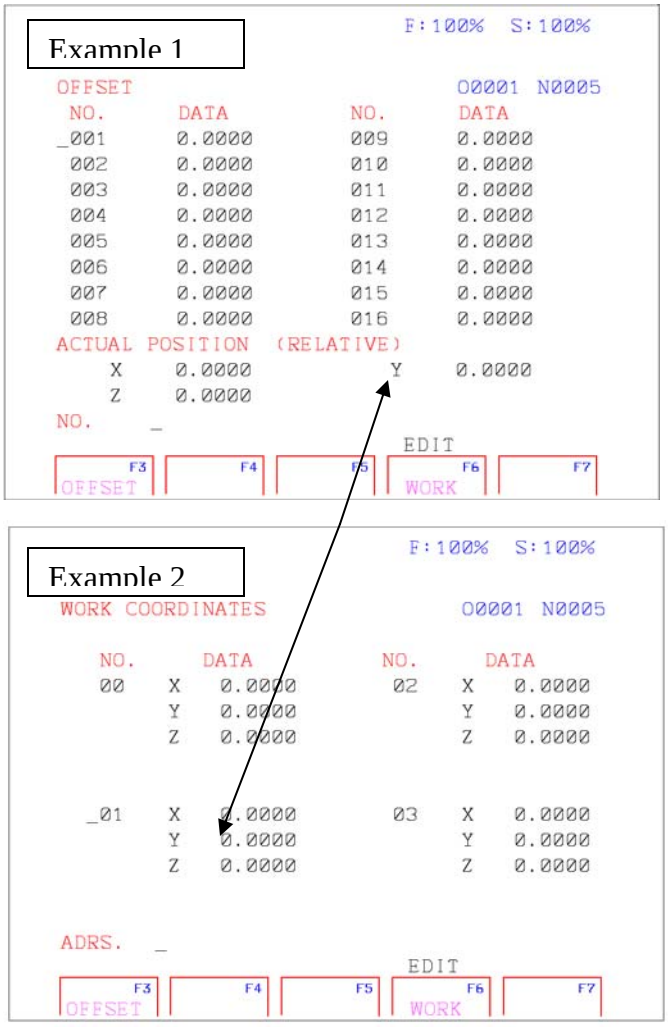


19. Jog Spindle up away from WORK PIECE using 8 on # keys
20. Jog the Tool to the Front of the Work Piece & touch using 1,2,4,6,8,9 on # keys.
    (Use Feed Dial or Steps to approach slower)



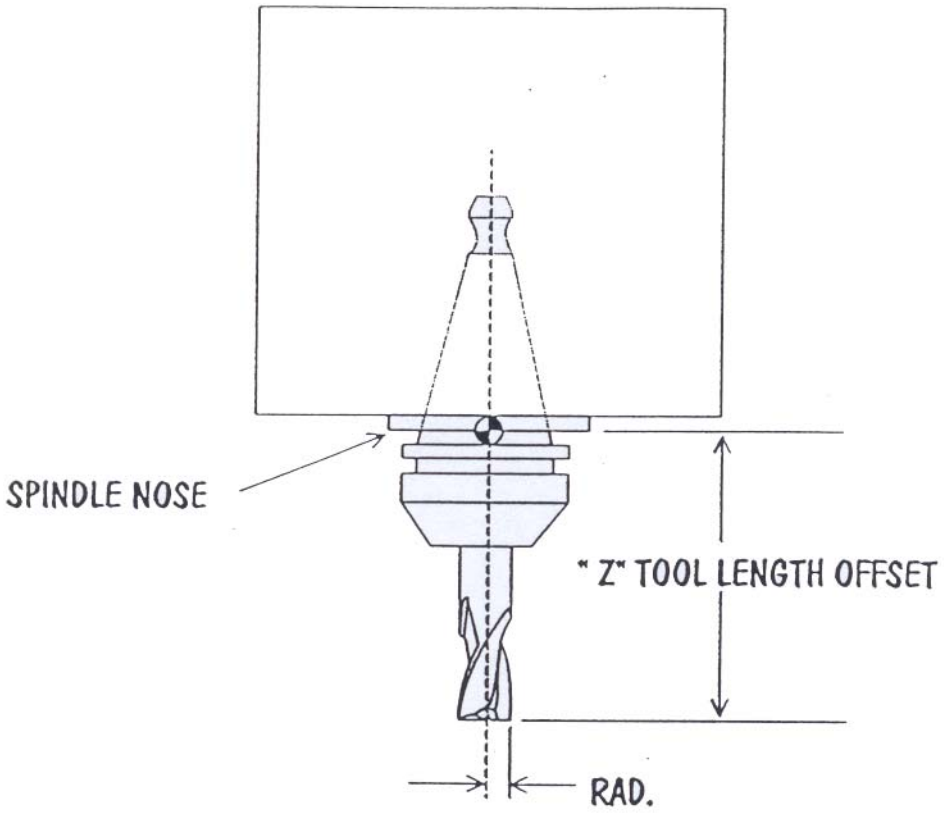
Note: Machine 0 in Y is the center of the spindle to the Front of the Machine bed.

21. Press F12 then F5 OFFSET screen
    - Example 1 in the picture below
    - Record the value in the Actual Position Relative Y
22. Press F6 for WORK Example 2
23. Move Cursor to 01 location
24. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (Y)



25. Jog the Tool up above the Work Piece using 8 on # keys

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

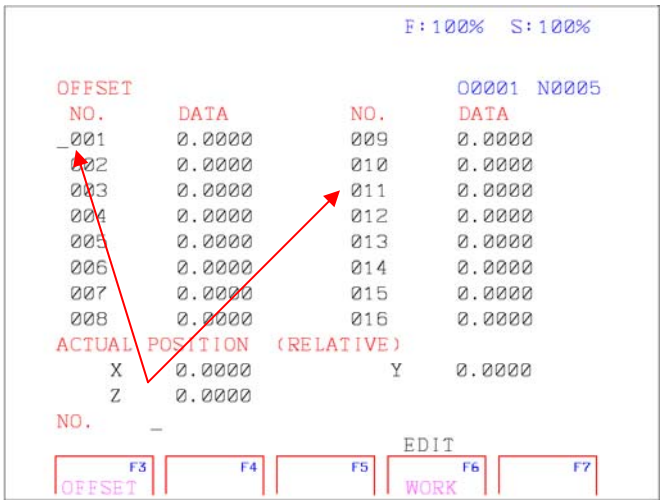
Z 0.0000

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 slower)
2. Press F12 then F5 for OFFSET
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 do not need a Radius set for them

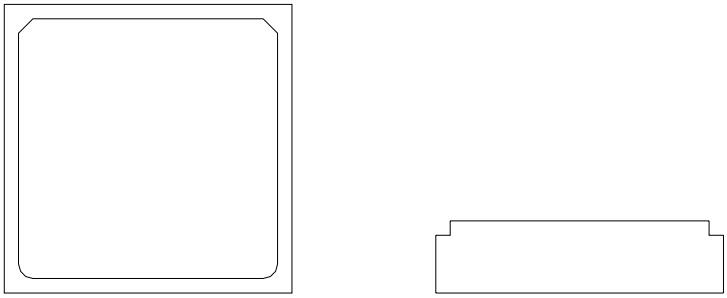


NOTE: When you use a T the H = Height

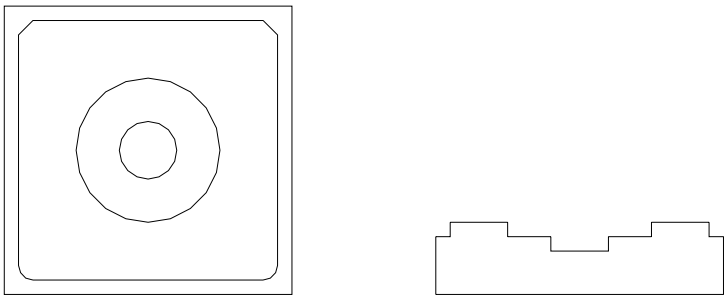
When you use a G41 or G42 the H = Radius

Program Training

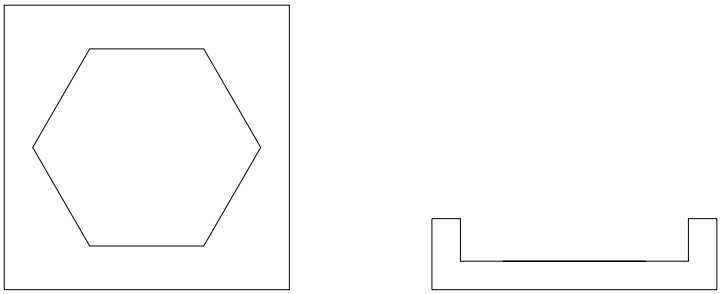
Program O0001



Program O0003



Program O0005



Press F1 then F4 for Edit & Press F12 then F4 for Program to do functions below & on the next 2 Pages

- ## INSERT A NEW PROGRAM

    1. Press letter O then program number
    2. Press Enter

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

- ## CALL A EXISTING PROGRAM UP

    1. Press letter O then program number
    2. Press arrow pointing down

- ## INSERT A WORD

    1. Press letter then number
    2. Press Enter

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

    Example: _**N5 G01 X 0.25; <u>G01</u>** is the word being inserted

- ## INSERT END OF BLOCK

    1. Press (;)
    2. Press Enter
    3. Or press enter 2 times

    HINT: at the end of each number block needs an <u>End Of Block</u> looks like a Semicolon **(;)**

    Example: N5 G01 X1.00 F.003;

- **DELETE A PROGRAM**

    1. Press letter o then program number
    2. Press Delete

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

- **DELETE ALL PROGRAMS**

    1. Press letter o plus the – & 9999

    2. Press Delete

    Example: **<u>O – 9999</u>**

- **DELETE A WORD**

    1. Press letter then number
    2. Press Delete

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

    Example: BEFORE N5_S1000; AFTER N5;

    <u>(S1000)</u> is the word being deleted?

- **DELETE A BLOCK OR LINE NUMBER**

    1. Type the number block
    2. Press Delete

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

- **CANCEL MISTYPED WORD**

  1. Press Backspace

  HINT: In the ADRS. (Address) at the lower left of the screen is the word and numbers that's been typed in. Before pressing enter check if what was typed in is correct. If not press backspace and retype word or number.

- **ALTER A WORD**

  1. Type the word needed altered
  2. Press Insert

  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 arrow pointing down

  Example:(N50)

- **SEARCH FOR WORD**

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

  2. Press arrow pointing down

- **SEARCH FOR LETTER**

  1. Press letter
  2. Press arrow pointing down

  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

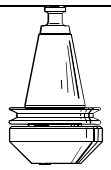
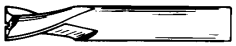
M00 Programmed stop, unconditional
M01 Programmed stop, conditional
M03 Spindle ON clockwise
M04 Spindle ON counter clockwise
M05 Spindle OFF
M25 Open clamping vice
M26 Close clamping vice
M30 Main program end with new start of program
M71 Blow-off ON
M72 Blow-off OFF
M98 Subroutine call-up
M99 Subroutine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

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

Used Addresses

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

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

F1Z 010	**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

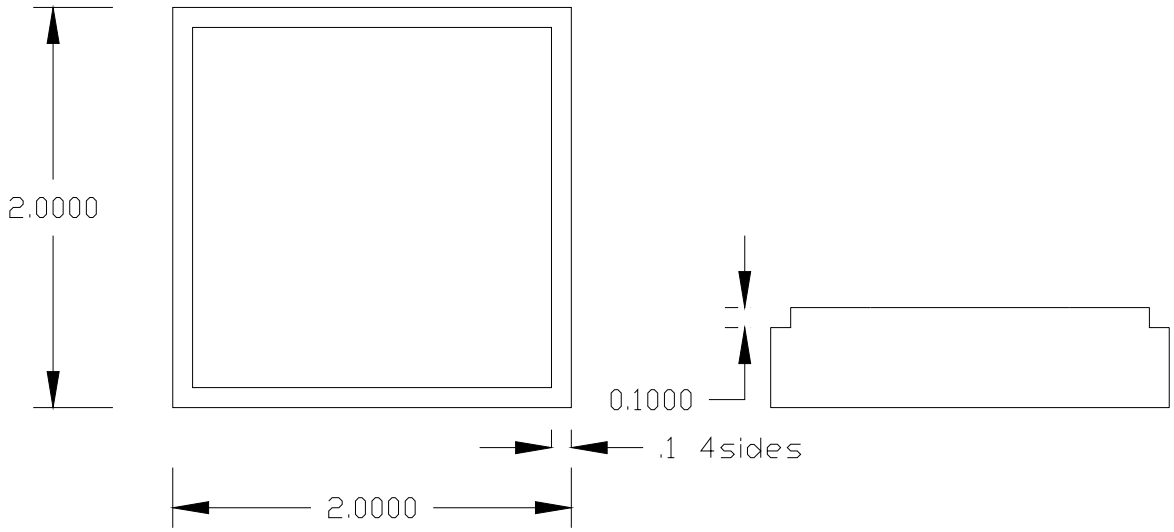
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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

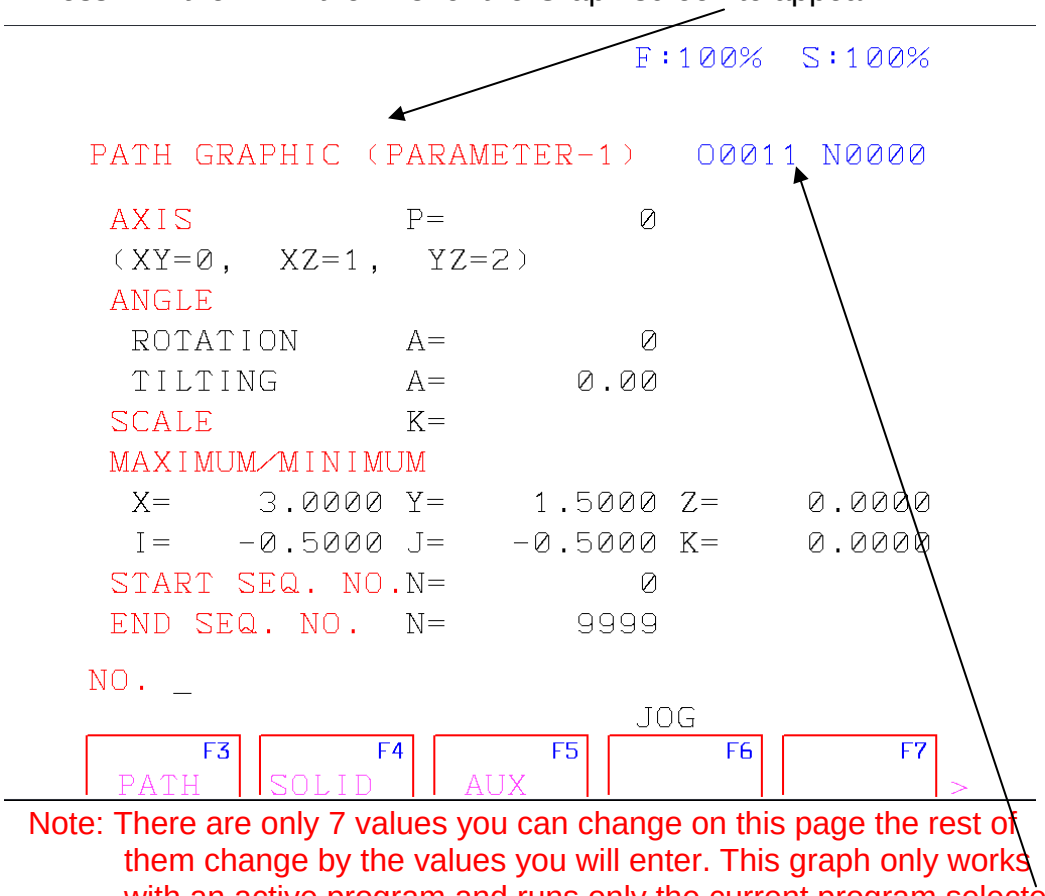
Program O0001



N5 G00 G17 G40 G80 (Demo 1) (2 X 2 X .5 Alum.)
N10 G90 G94 G98
N15 **G54**
N20 G43 T1 H1 M0 (3/8 or 10mm Endmill)
N25 S1800 M3
N30 G0 Z1
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9
N55 X1.9
N60 Y.1
N65 X.1
N70 Y1
N75 G0 G40 X-1
N80 G28 Z3
N85 G28 X2.5 Y2.5
N90 M30

2D Simulation PC Keyboard

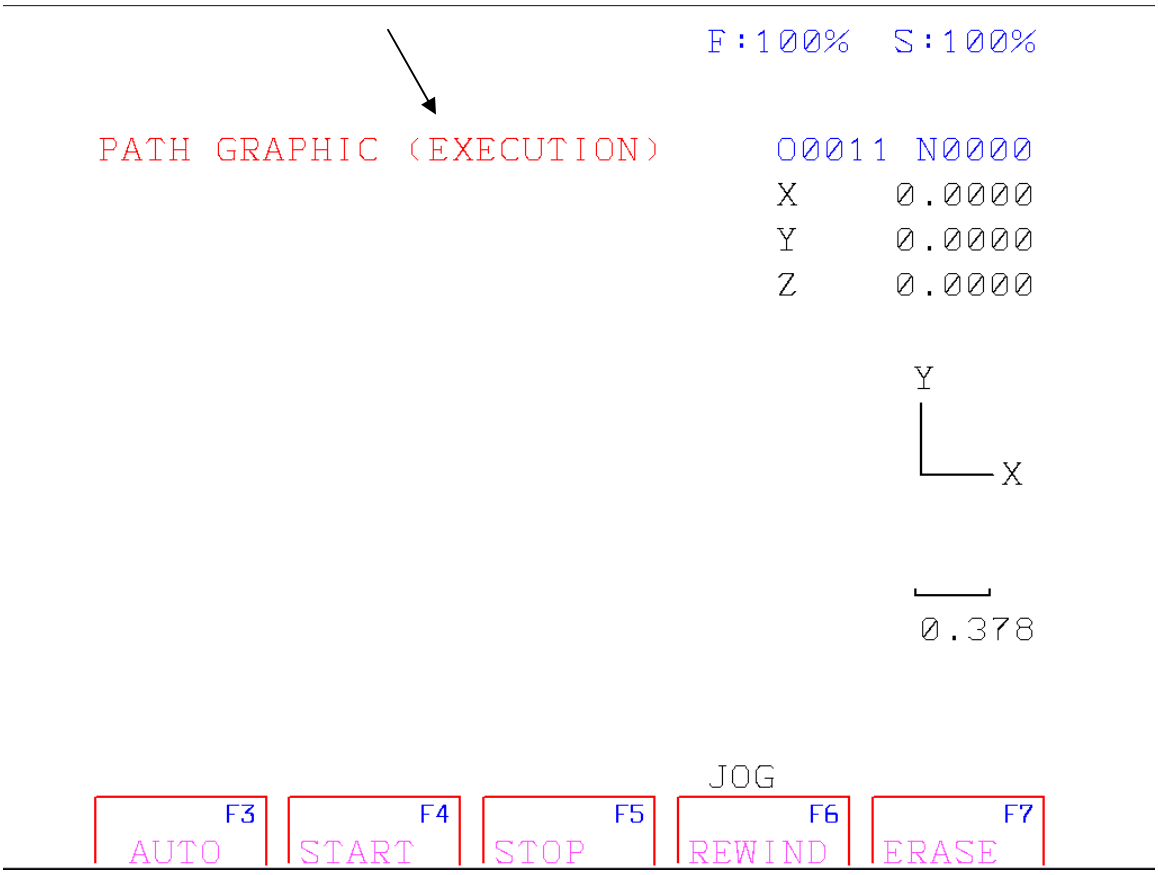
1. Press F12 then F11 then F3 for the Graph screen to appear



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 F3 for PATH
10. Press F5 for Execution screen



Note: If you press F3 on this screen this will auto scale for you. You will need to press F2 (scroll back) to go back and enter your values that you originally had.

11. Now press Enter on the # Keys for Cycle start or F4 for 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. Press F1 then F4 for **EDIT**
    2. Press F12 then F6 for **Parameter**
    3. Page down until you see Parameter (Setting 1)
    4. Cursor down to (I/O)
    5. Type A (for the floppy drive) then press Enter key

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

- **Output Program from Fanuc software to Drive unit**
    1. Press F12 then F4 for **Program**
    2. Type program number to be send out

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

    3. Press F9 for Output

- **Output Offsets from Fanuc software to Drive unit**
    1. Press F12 then F5 **Offset**
    2. Press F9 for Output

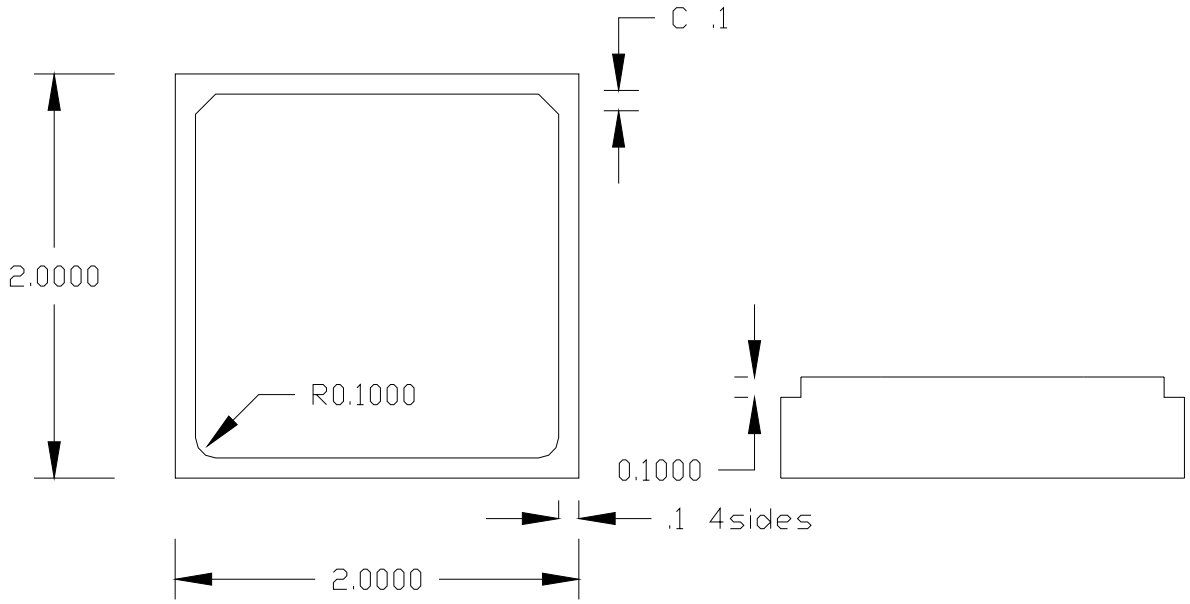
- **Input Program into Fanuc Software from Drive unit**
    1. Press F12 then F4 for **Program**
    2. Type program number to be read

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

    3. Press F10 for input

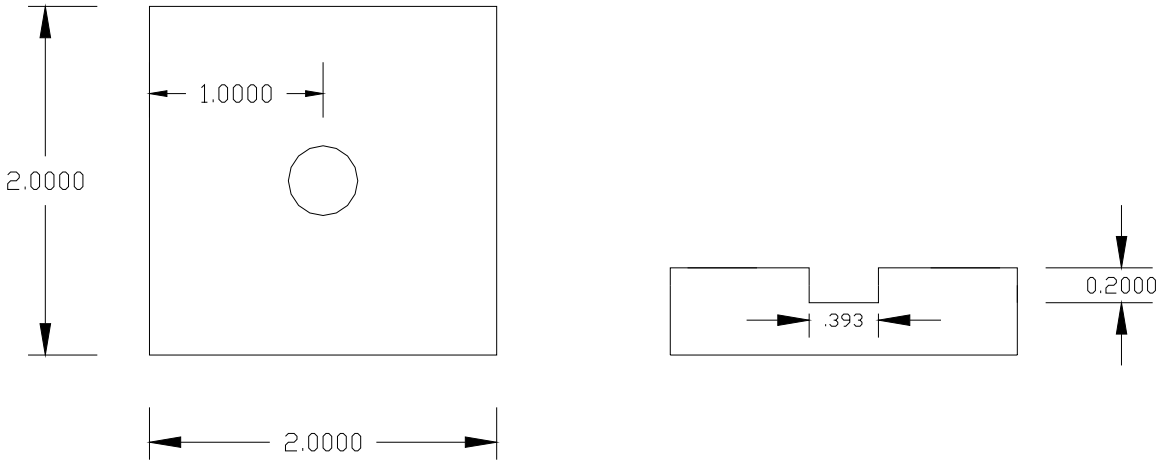
- **Input Offsets into Fanuc Software from Drive unit**
    1. Press F12 then F5 **Offset**
    2. Press F10 for input

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 M0 (3/8 or 10 mm end mill)
N25 S1800 M3
N30 G0 Z1
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9 **C.1**
N55 X1.9 **C.1**
N60 Y.1 **R.1**
N65 X.1 **R.1**
N70 Y1
N75 G0 G40 X-1
N80 G28 Z3
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 M0 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.05

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

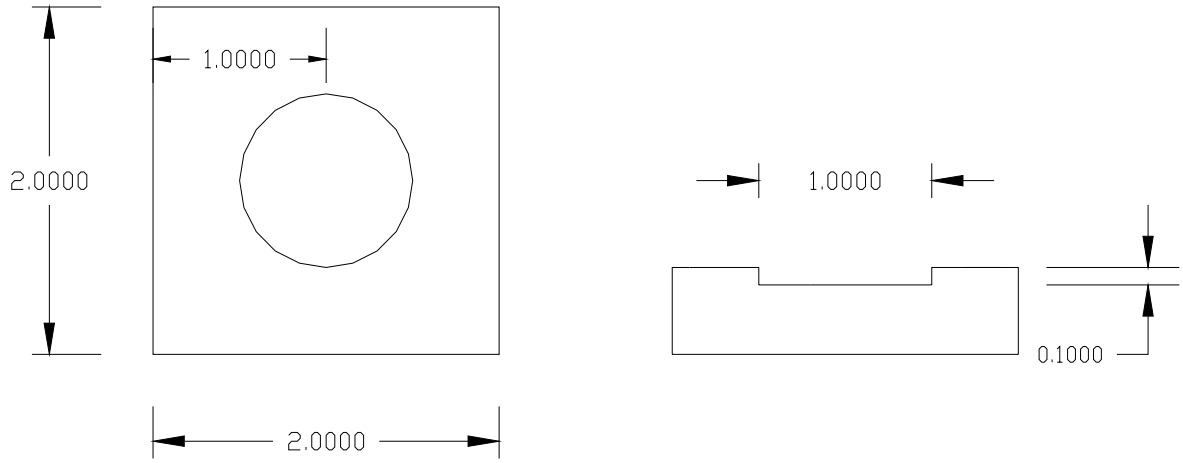
N40 G80

N45 G28 Z3

N50 G28 X2.5 Y2.5

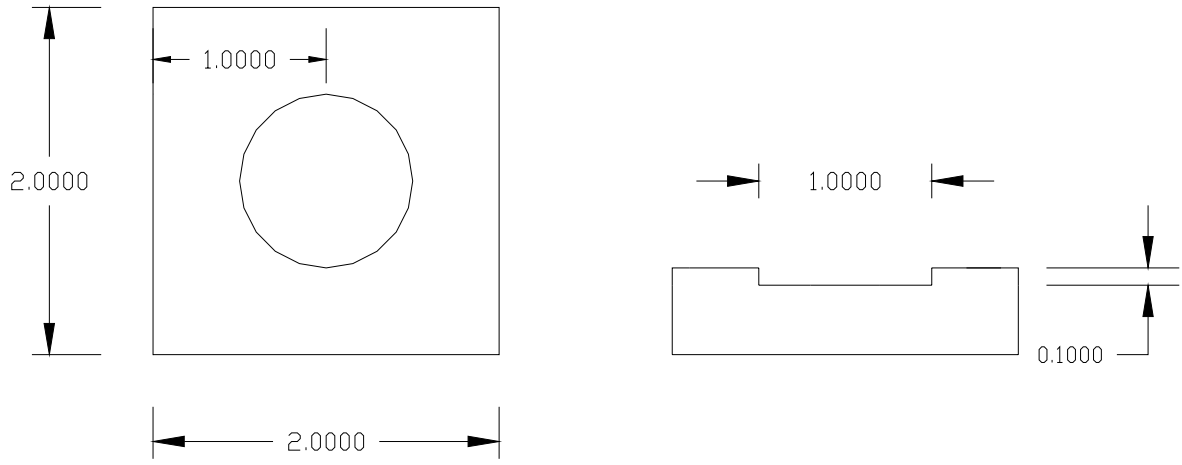
N55 M30

Program O0003 (I & J)



N5 G54 (Demo 3) (2 X 2 X .5 Alum.)
N10 G43 T1 H1 M0 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F5
N50 G2 X.5 Y1 I.5 J0 (360 degrees)
N55 G0 G40 X1
N60 G28 Z3
N65 G28 X2.5 Y2.5
N70 M30

Program O0003 (R)



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

N10 G43 T1 H1 M0 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z-.1 F3

N40 S1800

N45 G1 G42 H11 X.5 F5

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

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

N60 G0 G40 X1

N65 G28 Z3

N65 G28 X2.5 Y2.5

N70 M30

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

Program O0004 (Main Program)

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

N10 M98 P010001

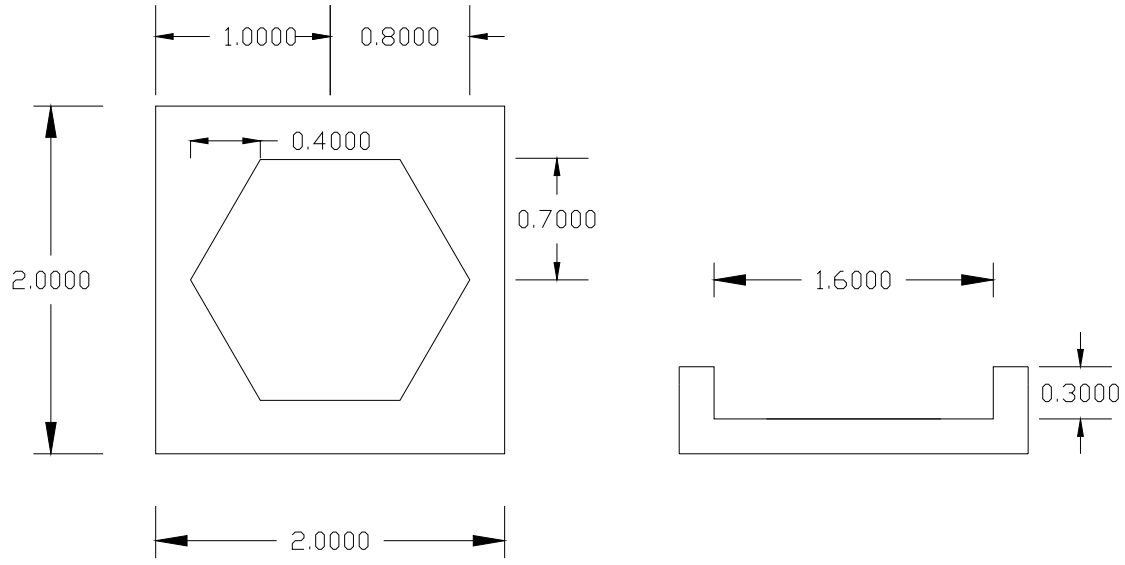
N15 M98 P010002

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 M0 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z0 F3

N40 M98 P030006

N45 G0 G28 Z3

N50 G28 X2.5 Y2.5

N55 M30

Program O0006 (Sub for program 5)

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