![emco innovative machine tools]

GE FANUC O
155 MILL
TRAINING GUIDE

Training Index

Appendix

FANUC O CONTROL

MACHINE CONTROL

The Fanuc O Screen

```
                                    F:100%  S:100% ←————— 1

3 ——→ PROGRAM                       O0003 N0005 ←————— 2
        _N5
         %










4 ——→ ADRS.    _                                         5
                                    EDIT ←————————————
        F3          F4          F5          F6          F7
      PRGRM      LIB

                            6
```
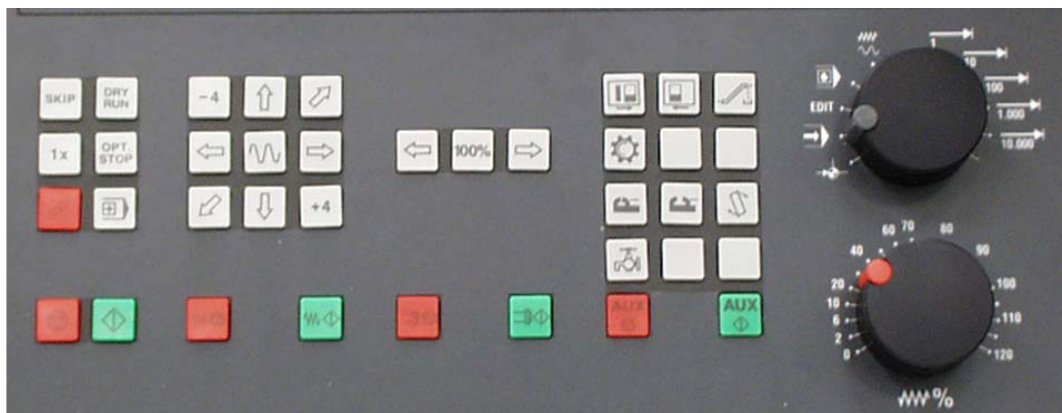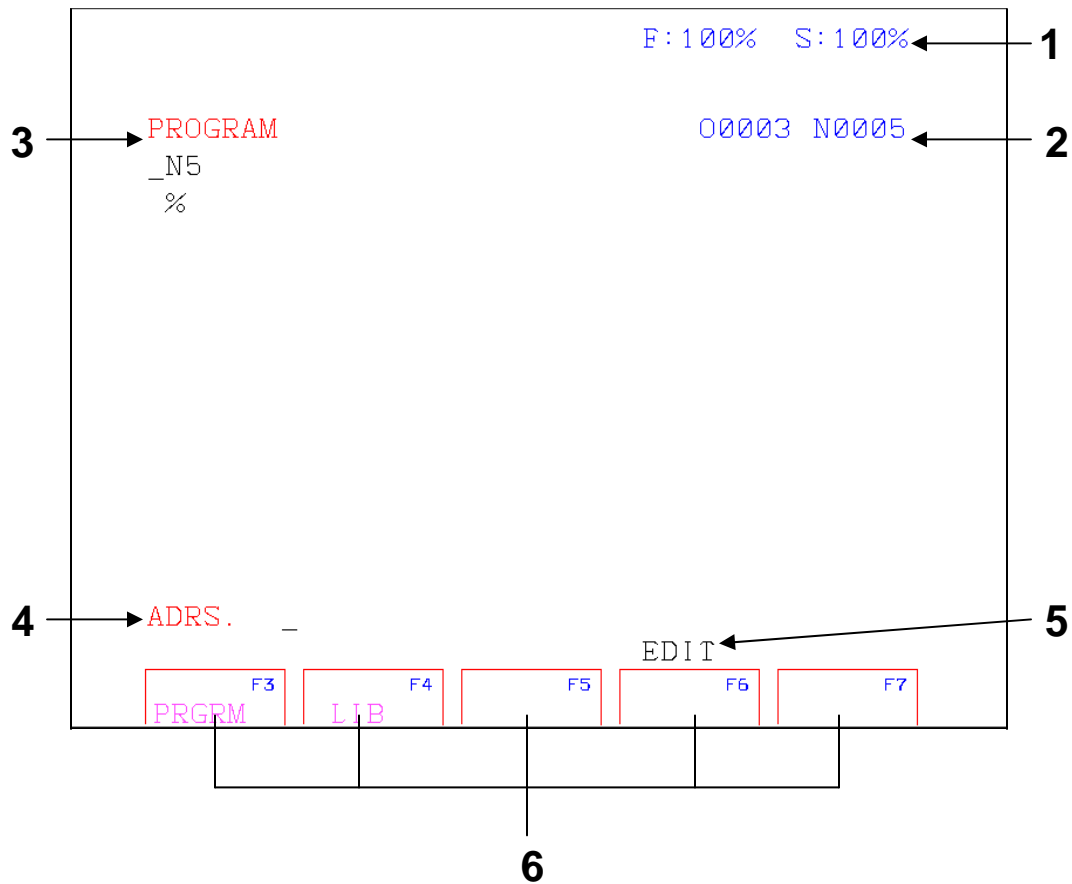
1.   **Displays of Feed and Spindle Speed override**

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 = displays actual, relative, machine positions

PRGRM = displays program, library page

MENU / OFSET = displays Offsets, Work shifts

DGNOS / PARAM = displays parameters, diagnostic pages

OPR / ALARM = displays operator & alarm messages

AUX / GRAPH = displays 2–D graph

SOFT KEY MODULE
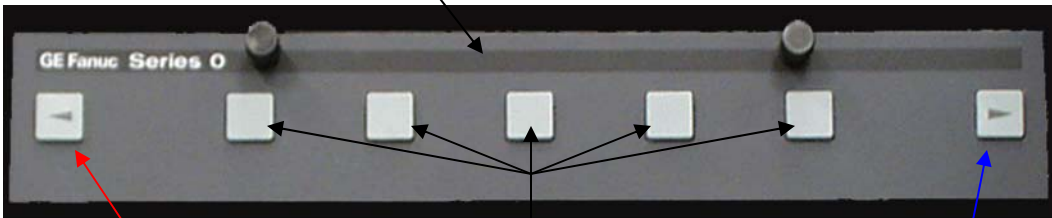
SCROLL BACK   SOFT KEYS   PAGES OVER

MACHINE KEYS

MACHINE FUNCTION KEYS

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

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

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

OPT. STOP = **(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**
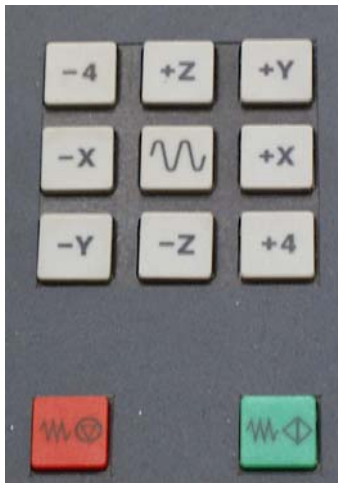
DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

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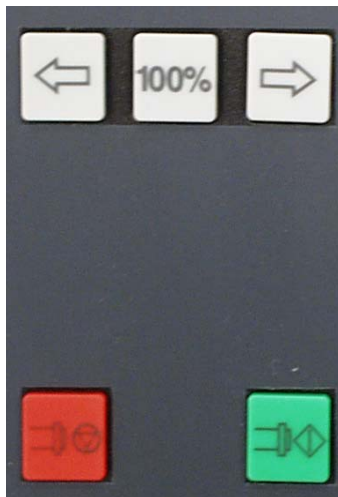
SPINDLE OVERRIDE KEYS

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
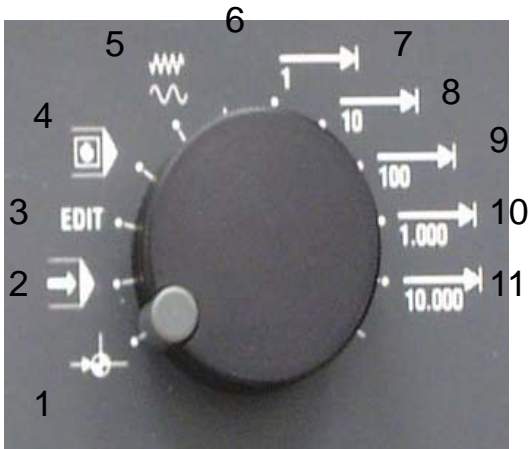
(1) ZRN = Zero / Reference or Home mode

(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 you 3D view

7. F12 then F11 then F3 then F11 then F3 gives Graph

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 when they will move the axes if used as numbers. Use numbers on the keyboard.

8

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 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".
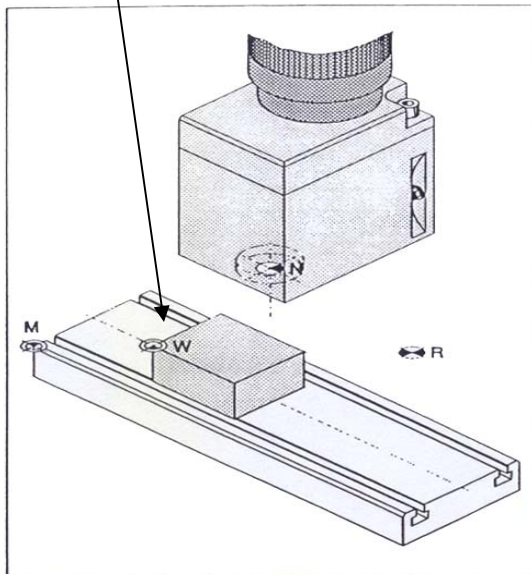Required after every power failure.

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.

Reference points in the working area

G54

Work 01
(G54)
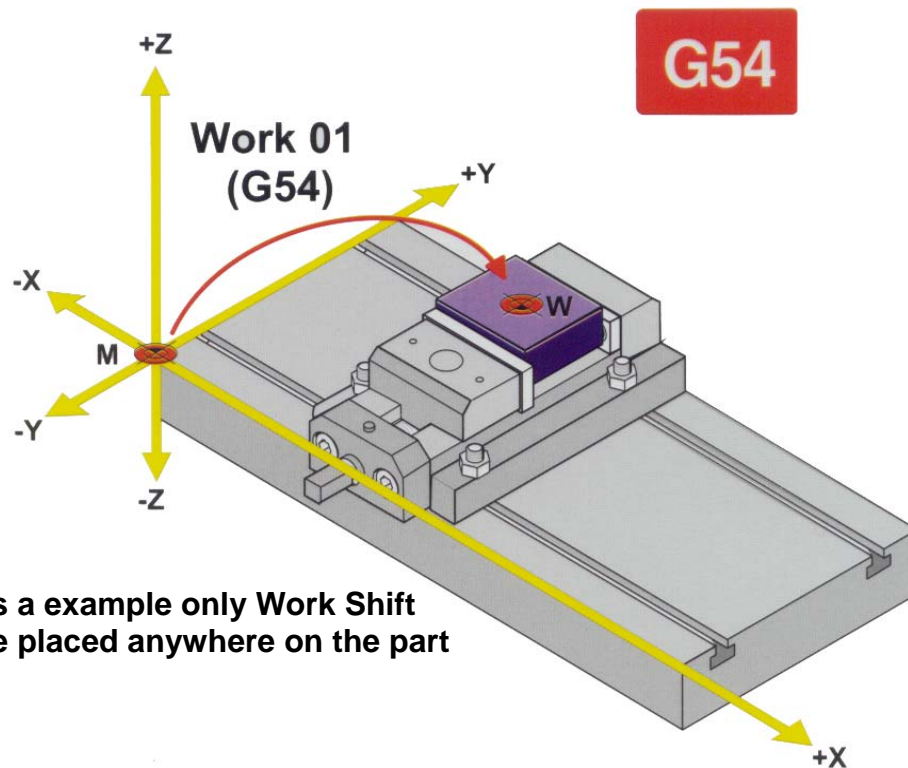
This is a example only Work Shift can be placed anywhere on the part

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 **MENU OFSET**

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 **INPUT**

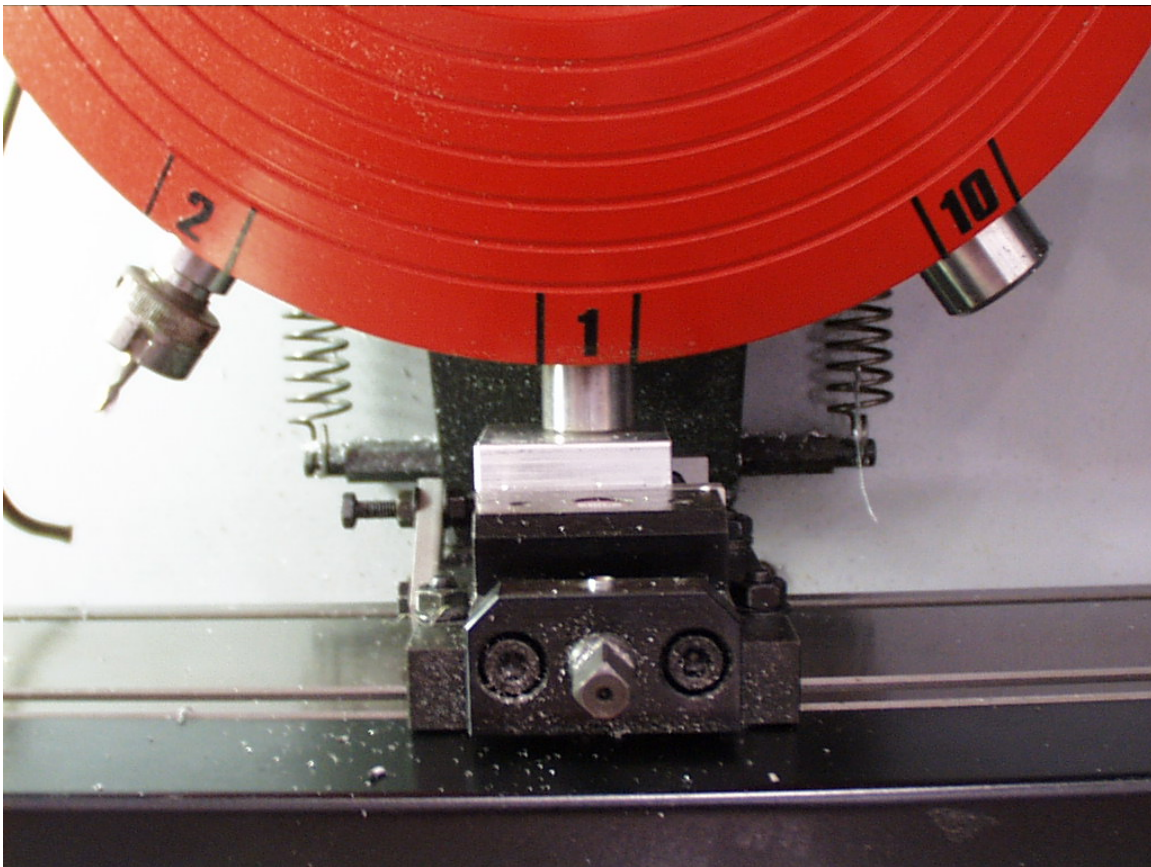This value is the distance from the top of the Machine bed to the top of the Work Piece.

```
                                    F:100%  S:100%
Example 1

   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          F4        F5          F6          F7
   OFFSET                   WORK
```

```
                                    F:100%  S:100%
Example 2
   WORK COORDINATES                   O0001 N0005

      NO.      DATA        NO.         DATA
      00    X   0.0000     02      X   0.0000
            Y   0.0000             Y   0.0000
            Z   0.0000             Z   0.0000


     _01    X   0.0000     03      X   0.0000
            Y   0.0000             Y   0.0000
            Z   0.0000             Z   0.0000


   ADRS.    _
                                    EDIT
      F3          F4        F5          F6          F7
   OFFSET                   WORK
```

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 `MENU OFSET`

- 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)

```
                                F:100%  S:100%
 ┌─────────────┐
 │ Example 1   │
 └─────────────┘
  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 │ │ F4 │ │ F5 │ │ F6 │ │ F7 │
  OFFSET               WORK
```

```
                                F:100%  S:100%
 ┌─────────────┐
 │ Example 2   │
 └─────────────┘
  WORK COORDINATES               O0001 N0005

     NO.     DATA        NO.      DATA
     00   X  0.0000      02   X   0.0000
          Y  0.0000           Y   0.0000
          Z  0.0000           Z   0.0000


    _01   X  0.0000      03   X   0.0000
          Y  0.0000           Y   0.0000
          Z  0.0000           Z   0.0000


  ADRS.    _
                            EDIT
  ┌────┐ ┌────┐ ┌────┐ ┌────┐ ┌────┐
  │ F3 │ │ F4 │ │ F5 │ │ F6 │ │ F7 │
  OFFSET               WORK
```

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)
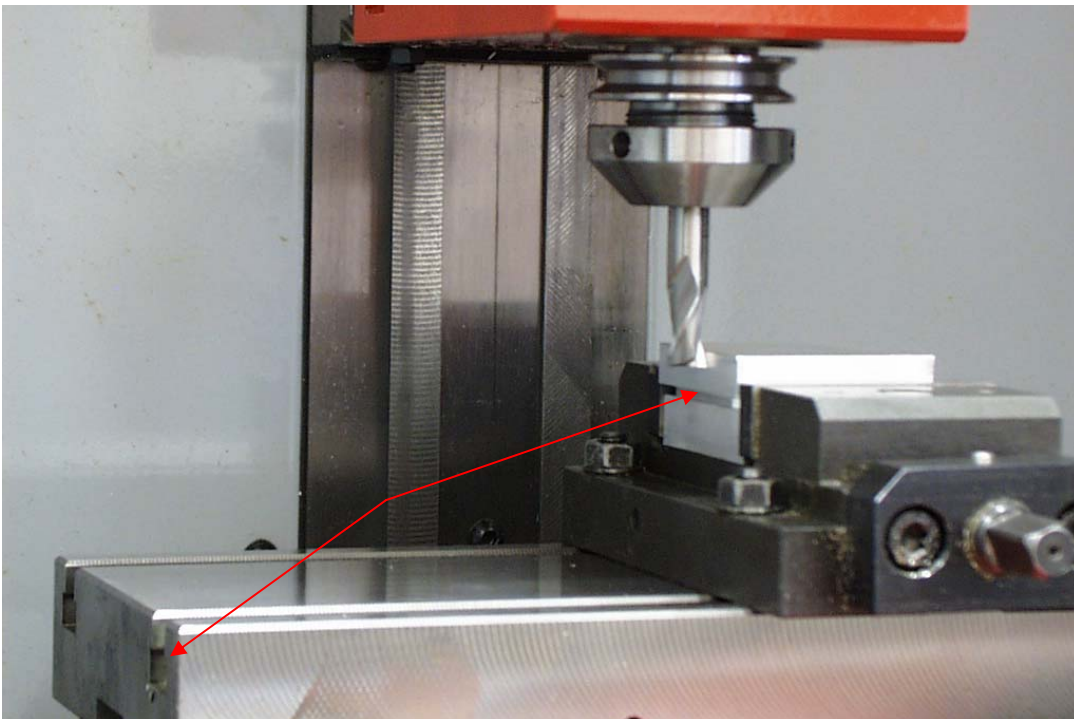

Side View
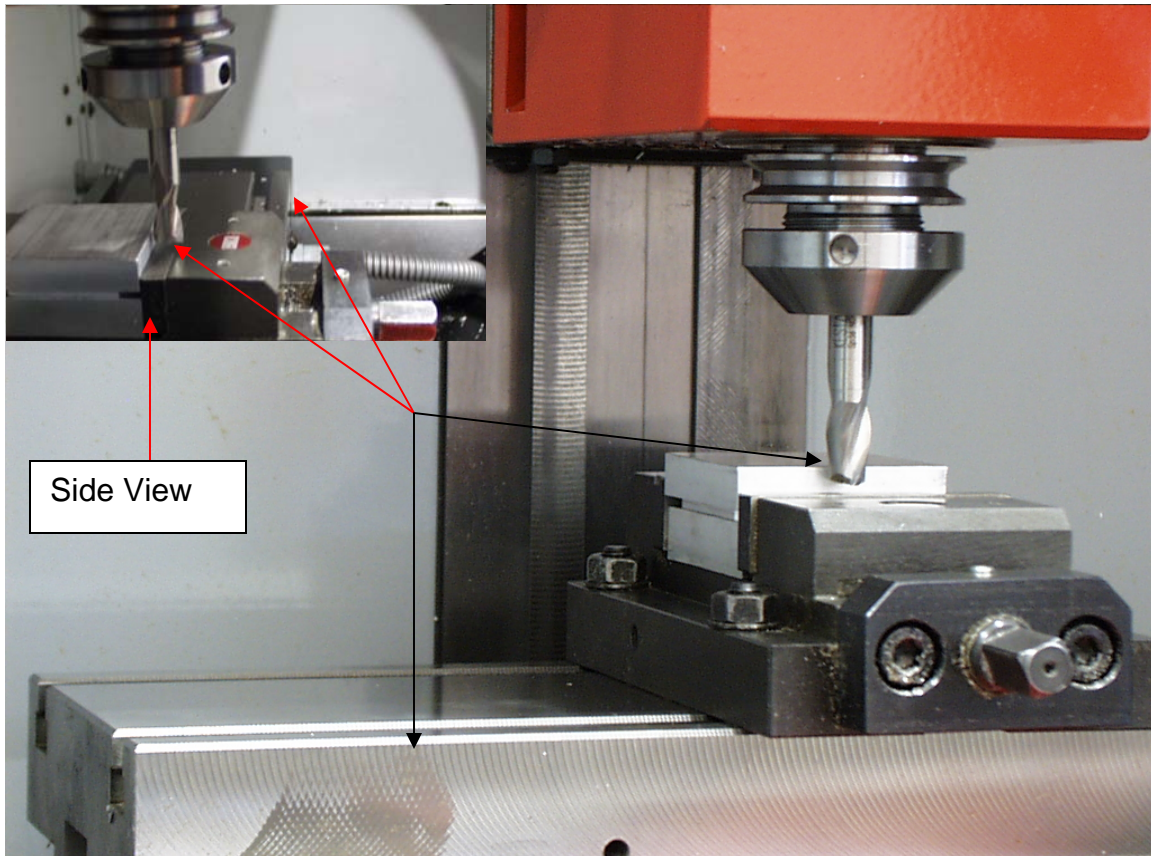
Note: Machine 0 in Y is the center of the spindle to the Front of the Machine bed.

20. Press the MENU/OFSET button [MENU OFSET]

- 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)

```
Example 1                          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         F4        F5      F6        F7
OFFSET                       WORK
```

```
Example 2                          F:100%  S:100%

WORK COORDINATES                       O0001 N0005

    NO.       DATA          NO.        DATA
    00    X   0.0000        02    X   0.0000
          Y   0.0000              Y   0.0000
          Z   0.0000              Z   0.0000


   _01    X   0.0000        03    X   0.0000
          Y   0.0000              Y   0.0000
          Z   0.0000              Z   0.0000


 ADRS.    _
                             EDIT
    F3         F4        F5      F6        F7
OFFSET                       WORK
```
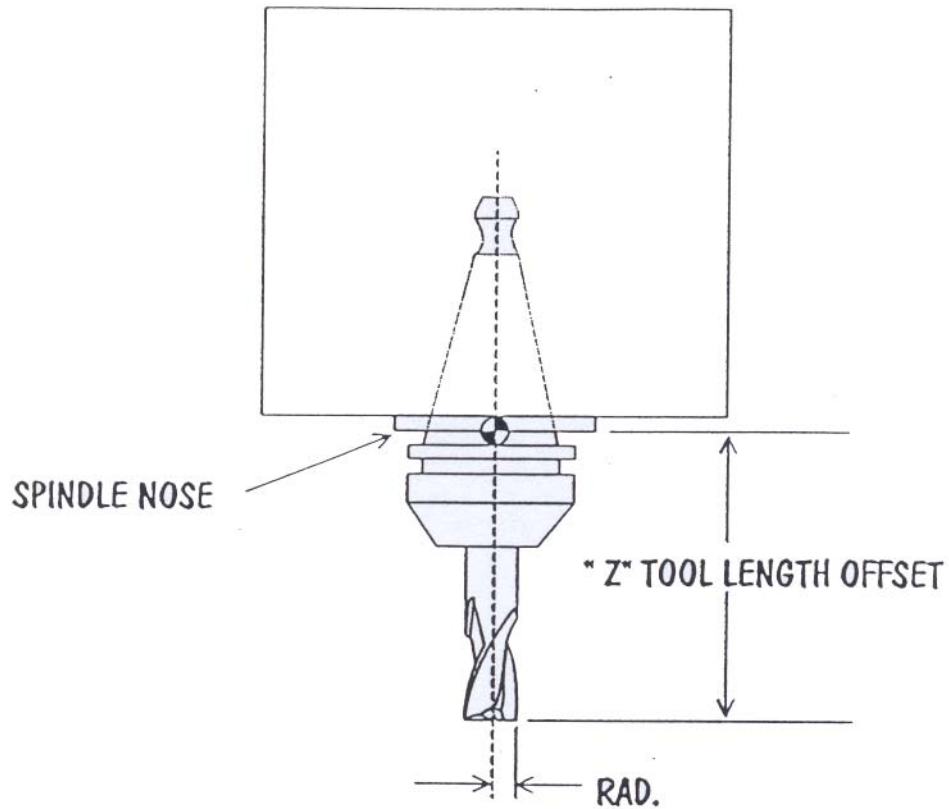
24. Jog the Tool up above the Work Piece using Z+

TOOL OFFSET

SPINDLE NOSE

" Z" TOOL LENGTH OFFSET

RAD.

```
                                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          F4         F5        F6        F7
  OFFSET                              WORK
```

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 **MENU OFSET**
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

```
                                        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
```
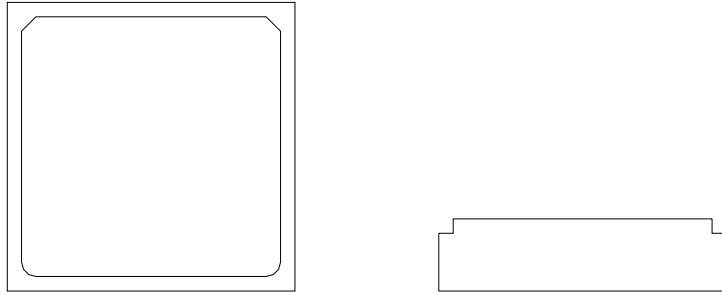
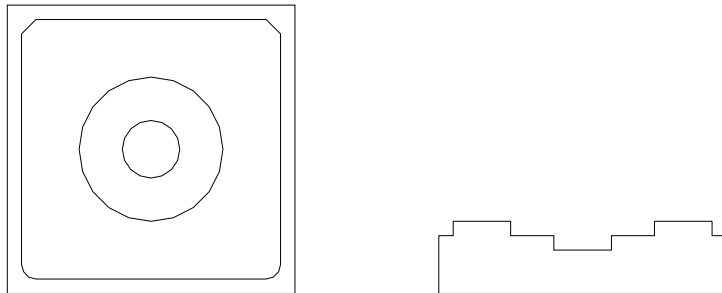| F3	F4	F5	F6	F7
OFFSET			WORK	

NOTE: When you use a T the H = Height

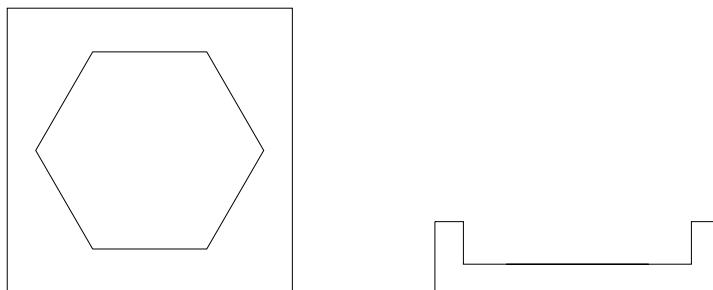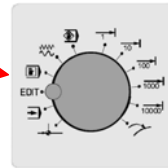When you use a G41 or G42 the H = Radius

Program Training

Program O0001

Program O0006

Program O0007

Change the Mode Dial to Edit
Press the Program Button
To do functions on this page

PRGRM

- ## 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**

        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   **/ , # EOB**
    2. Press insert button   **INSRT**
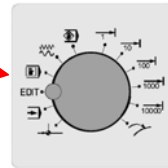
        HINT: at the end of each number line needs an End Of Block

        looks like a Semicolon **(;)**

        Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT ONLY.
USE INPUT ALL OTHER APPLICATIONS.

Change the Mode Dial to Edit
Press the Program Button — PRGRM
To do functions on this page

- ## DELETE A PROGRAM

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

     Example: <u>O0001</u> OR <u>O1</u>

- ## DELETE ALL PROGRAMS

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

     Example: <u>O – 9999</u>

- ## DELETE A CODE

  1. Press letter then number
  2. Press delete button `DELET`

     Example: press once <u>S</u> 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; AFTER N5;

     (<u>S1000</u>) is the word being deleted?
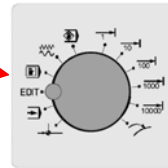
- ## DELETE A BLOCK OR LINE NUMBER

  1. Type the number line
  2. Press delete button `DELET`

     Example: _N10 G0 X1.0 F.003; make sure cursor is on

     the line being deleted (_N10)

Change the Mode Dial to Edit
Press the Program Button
To do functions on this page

`PRGRM`

- **CANCEL MISTYPED CODE**

    1. Press cancel button `CAN`

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

- **ALTER A CODE**

    1. Type the code needed altered

    2. Press alter button `ALTER`

    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 CODE**

    1. Type in code & 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)**

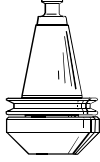## Survey commands M-CODES: Mostly used

M00   Programmed stop, unconditional
M01   Programmed stop, conditional
M03   Spindle ON clockwise
M04   Spindle ON counter clockwise
M05   **Spindle OFF**
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 910	**Collet holder**	For ESX-16 collets	
152 800	**(9.0-10.0mm)⌀ 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 <u>O</u>0001
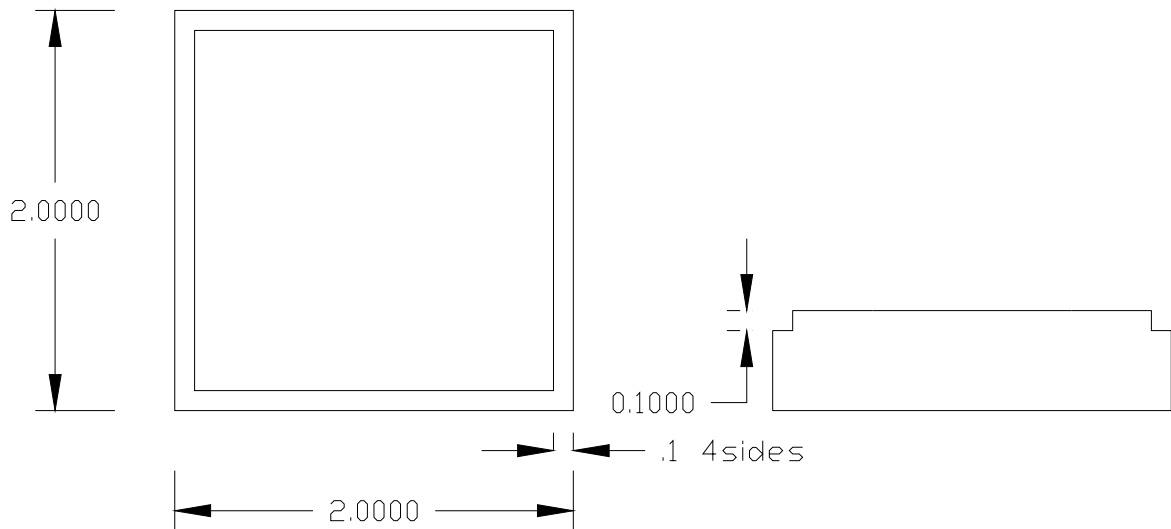
N5 G00 G17 G40 G80 (Demo 1) (2 X 2 X .5 Alum.)
N10 G90 G94 G98
N15 **G54**
N20 G43 T1 H1 M0   (3/8 or 10mm Endmill)
N25 S1800 M3
N30 G0 Z1
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9
N55 X1.9
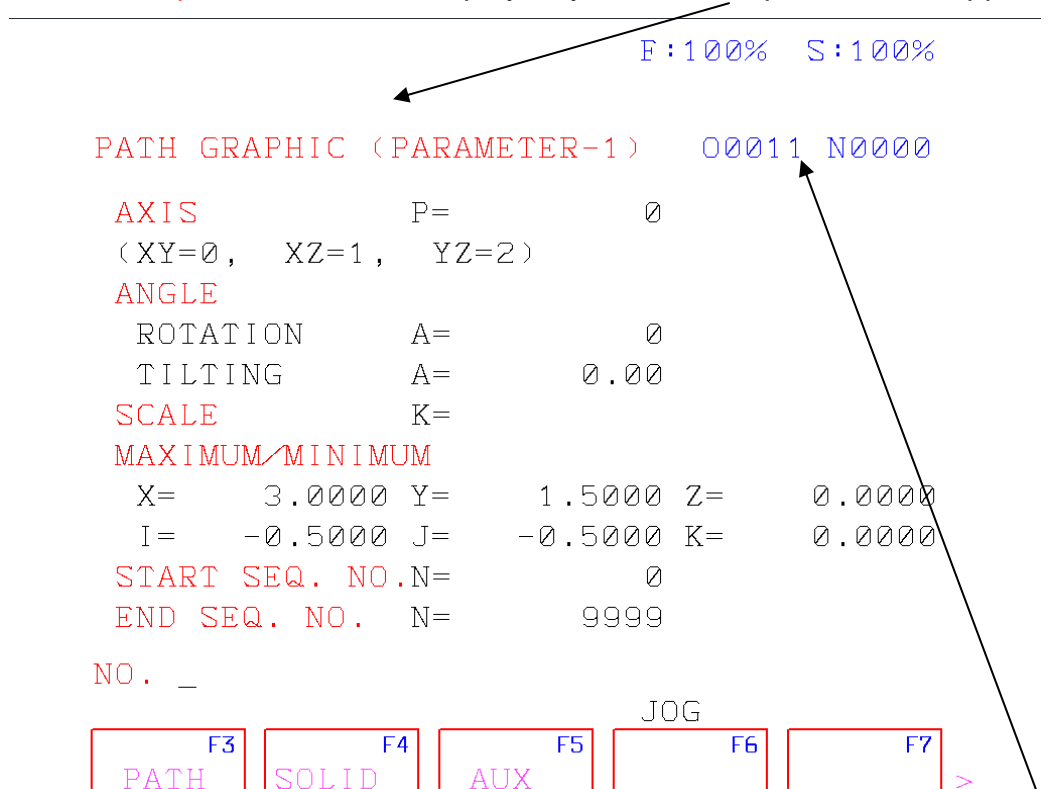N60 Y.1
N65 X.1
N70 Y1
N75 G0 G40 X-1
N80 G28 Z3
N85 G28 X2.5 Y2.5
N90 M30

2D Simulation

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

```
                                F:100%   S:100%


     PATH GRAPHIC (PARAMETER-1)   O0011 N0000

      AXIS           P=            0
      (XY=0,  XZ=1,  YZ=2)
      ANGLE
       ROTATION      A=            0
       TILTING       A=         0.00
      SCALE          K=
      MAXIMUM/MINIMUM
       X=     3.0000 Y=     1.5000 Z=     0.0000
       I=    -0.5000 J=    -0.5000 K=     0.0000
      START SEQ. NO.N=            0
      END SEQ. NO.  N=         9999

     NO. _
                                JOG
```

F3	F4	F5	F6	F7
PATH	SOLID	AUX		>

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

F:100% S:100%

PATH GRAPHIC (EXECUTION)   O0011 N0000
                                 X   0.0000
                                 Y   0.0000
                                 Z   0.0000

                                          Y
                                          └── X

                                          └─┘
                                          0.378

                              JOG
| F3 | F4 | F5 | F6 | F7 |
| AUTO | START | STOP | REWIND | ERASE |
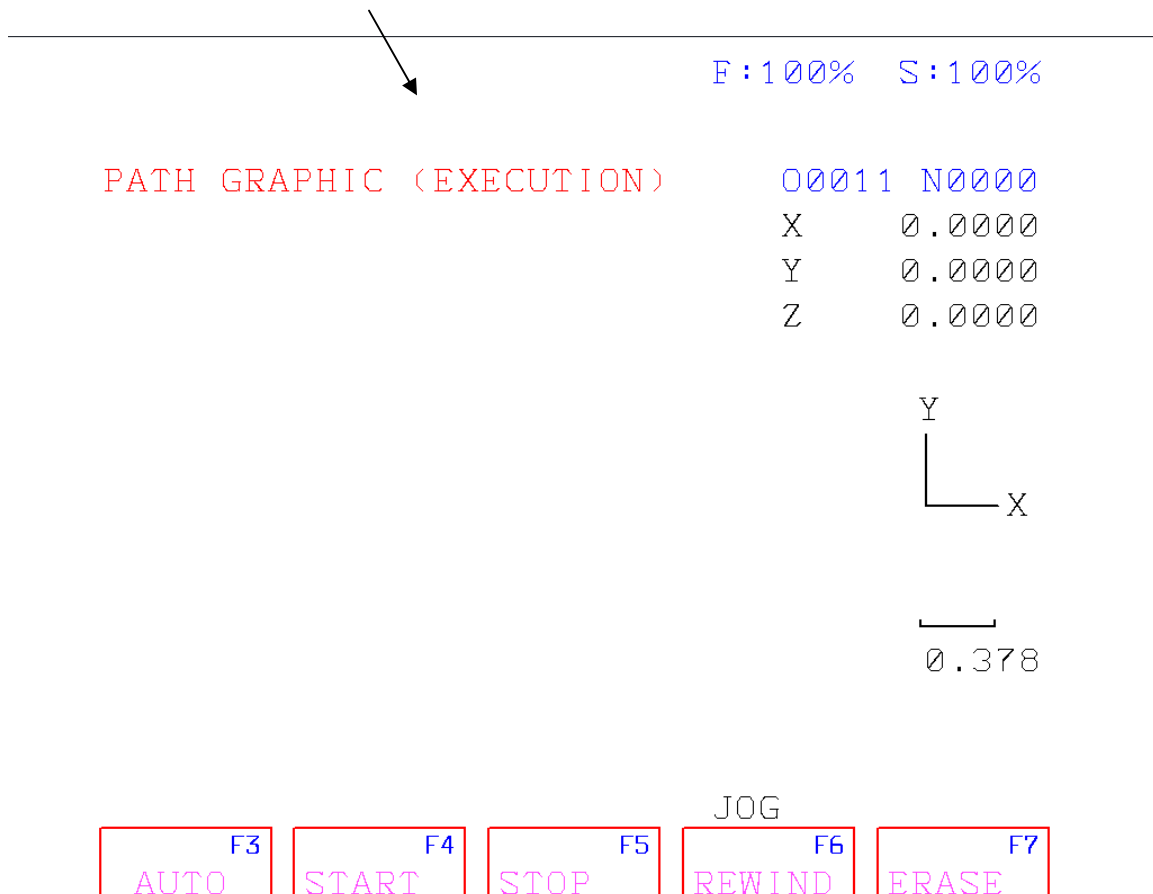
Note: If you press the AUTO on this screen this will auto scale for you. You will
need to press the arrow left on the soft keys to go back and enter your values
that you originally had.

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)

  Note: If you want to use USB use C and then follow 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 <u>O</u> and program number
       (<u>O</u>0002) or (<u>O</u>2)

    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 <u>O</u> and program number
       (<u>O</u>0002) or (<u>O</u>2)

    3. Press (**Input)** key

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

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 **PRGRM**
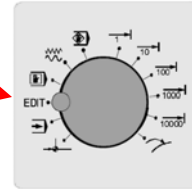
3. Call up Program to be run / cut (Example O1 for program 1)
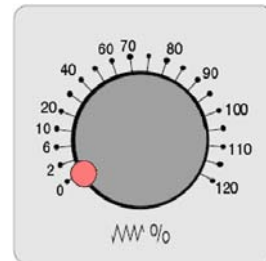
4. Rotate the Mode dial to Auto
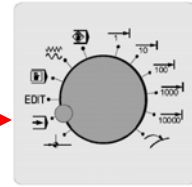
5. If screen is not in **PROGRAM CHECK** then press the check soft key

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, Z and looks right; you can take single block off and run the program)

8. Press Cycle Start one more time

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

Program O0002 (C & R)
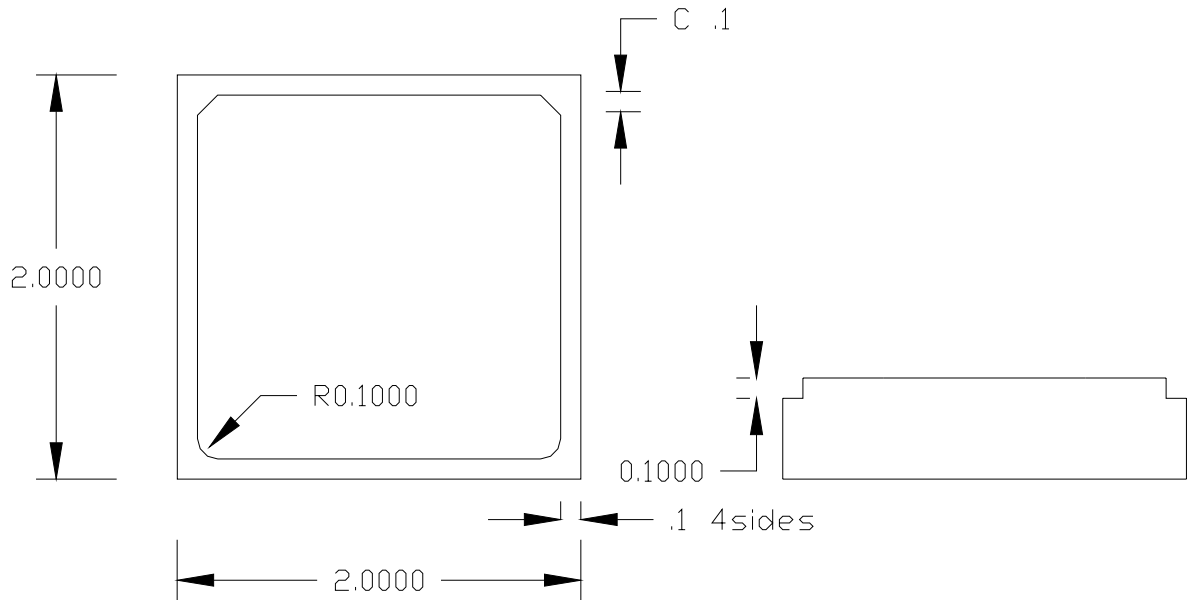
N5 G00 G17 G40 G80 (Demo 2) (2 X 2 X .5 Alum.)

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 <u>O</u>0003 (Deep Hole Drilling)
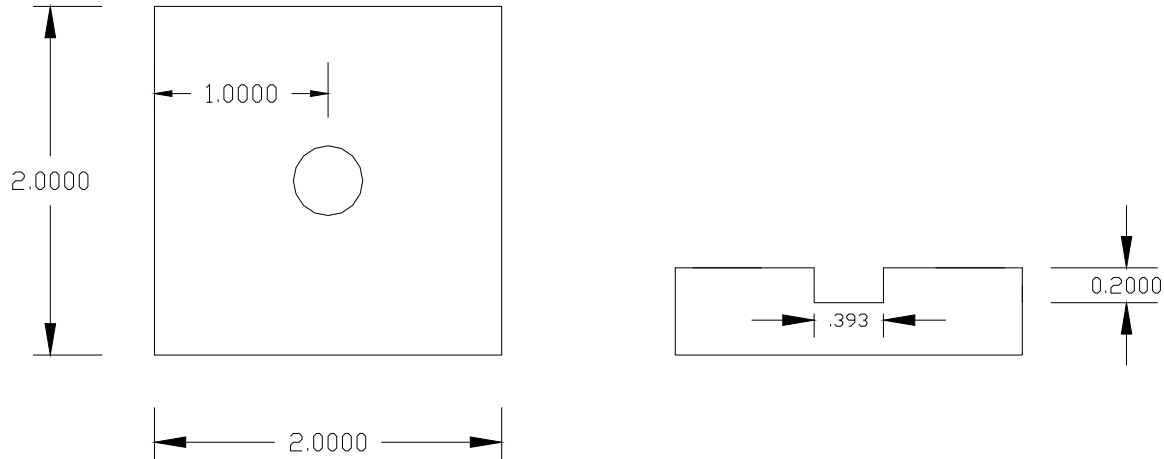
G83 X = Location of hole Y = location of hole
   Z = Overall Depth of hole P = Dwell at bottom of hole
   R = Retract after Cycle Q = incremental peck depth per pass
   K = Incremental repeats only used with G91 **F = Feed rate**

N5 G54 (Demo 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.05

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

N40 G80

N45 G28 Z3

N50 G28 X2.5 Y2.5

N55 M30

Program O0004 (I & J)
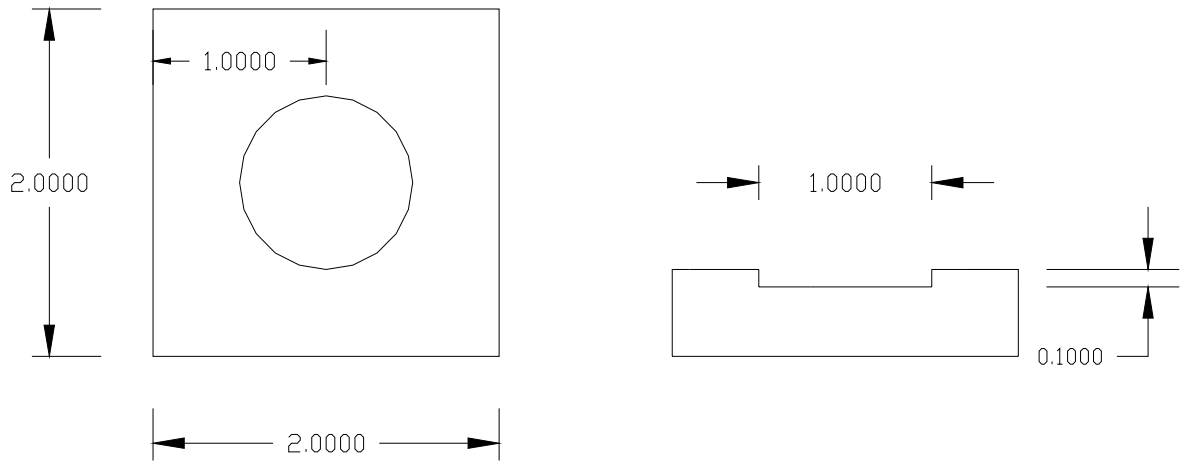
N5 G54 (Demo 4) (2 X 2 X .5 Alum.)

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 <u>O</u>0005 (R)
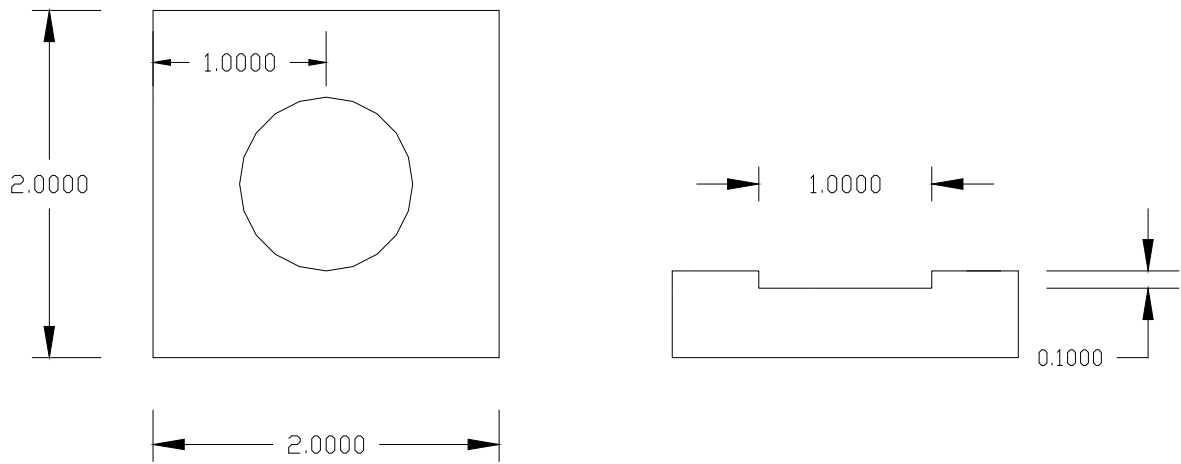
N5 G54 (Demo 5) (2 X 2 X .5 Alum.)

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 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

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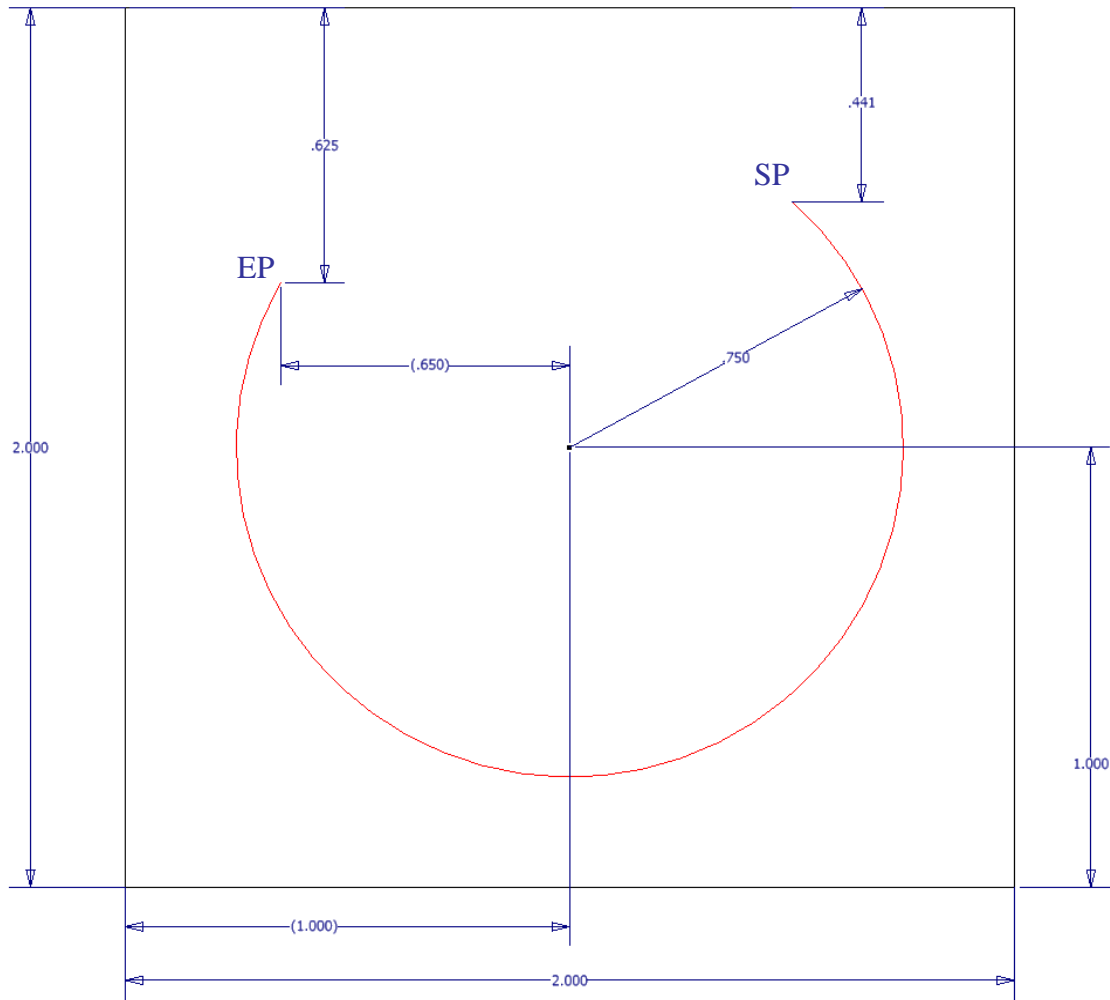
Using T1 (3/8 End Mill) and cutting on the inside of the red arc starting at the SP

```
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
```
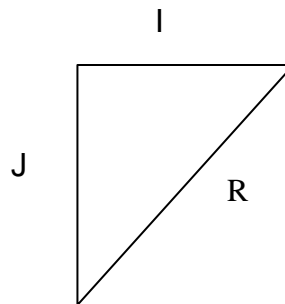
A^2 (K leg) $+ B^2$ (I leg) $= C^2$ (H radius)

<u>Degrees</u>
<u>S</u>ally <u>C</u>an <u>T</u>ell <u>O</u>scar <u>H</u>as <u>A</u> <u>H</u>at <u>O</u>n <u>A</u>lways
<u>S</u>INE <u>C</u>OSINE <u>T</u>ANGENT

S $\frac{O}{H}$

C $\frac{A}{H}$

T $\frac{O}{A}$

35

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)
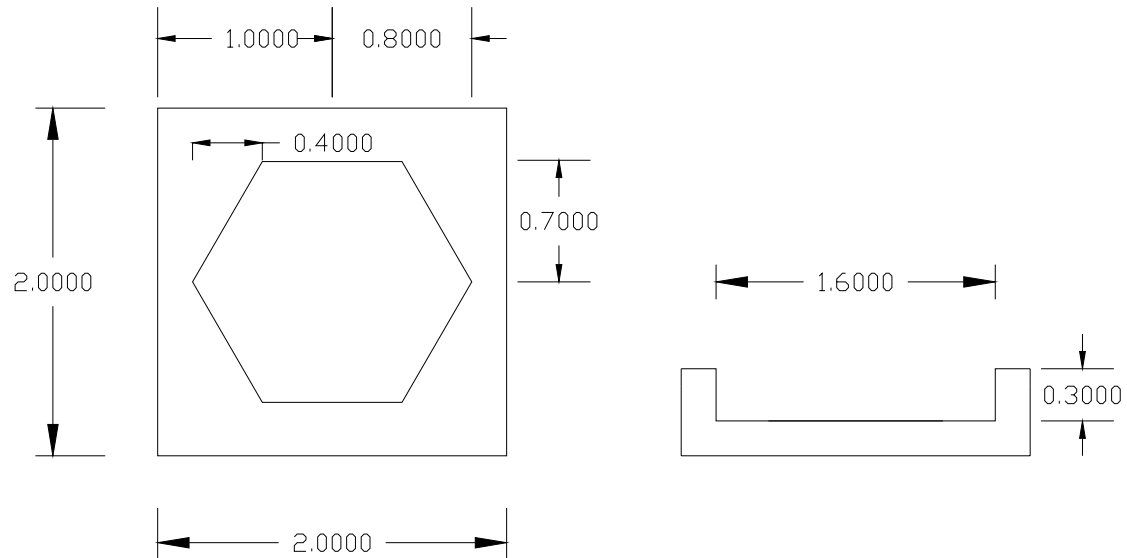
N5 G54 (Demo 7) (2 X 2 X .5 Alum.)

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 O0008 (Sub for program 7)

N5 G91 (Sub Prog. for Prog. 7)

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
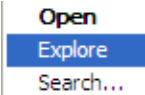
Changing Drive to USB Port

1. **Close out the SW (software)**

   - **Press** **to allow you to exit**

   - **Press** SKIP **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** Open Explore Search... **(Explorer)**
   - **Left Click**

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

     My Computer
     Local Disk (C:)
     DVD/CD-RW Drive (D:)
     USB20FD (E:)
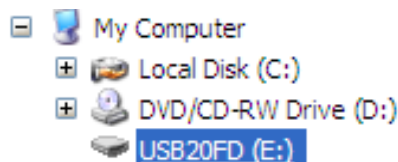
4. **Copy Drive directory**

   - **Click on you USB drive**

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

     Address E:\
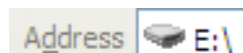
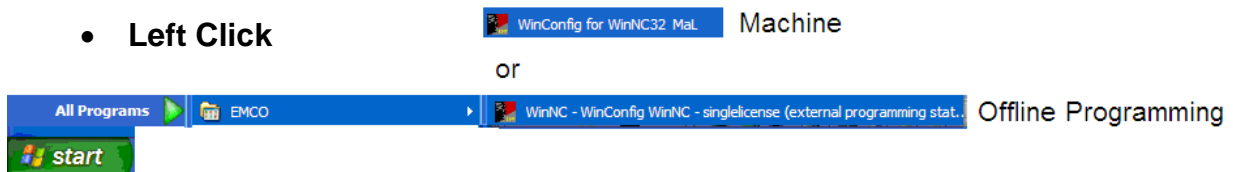   - **Close the active screen or page using either Alt and F4 or** **at top of the active screen**
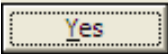
5. Setting up WinConfig

- **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**   WinConfig for WinNC32 MaL   Machine

  or

All Programs ▶ EMCO ▶ WinNC - WinConfig WinNC - singlelicense (external programming stat..   Offline Programming

start

6. In Winconfig

- **Left Click on** [INI] **(INI) button**

- **Double Left Click on** Directories **(Directories)**

- **Left click on white box**

  Import / Export directory

- **Either Press Ctrl and V (this will paste in the info) or type in USB directory**

- **Left Click on** OK **(OK)**

- **Left Click on** ✕ **(Close)**

- **Left Click on** Yes **(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**