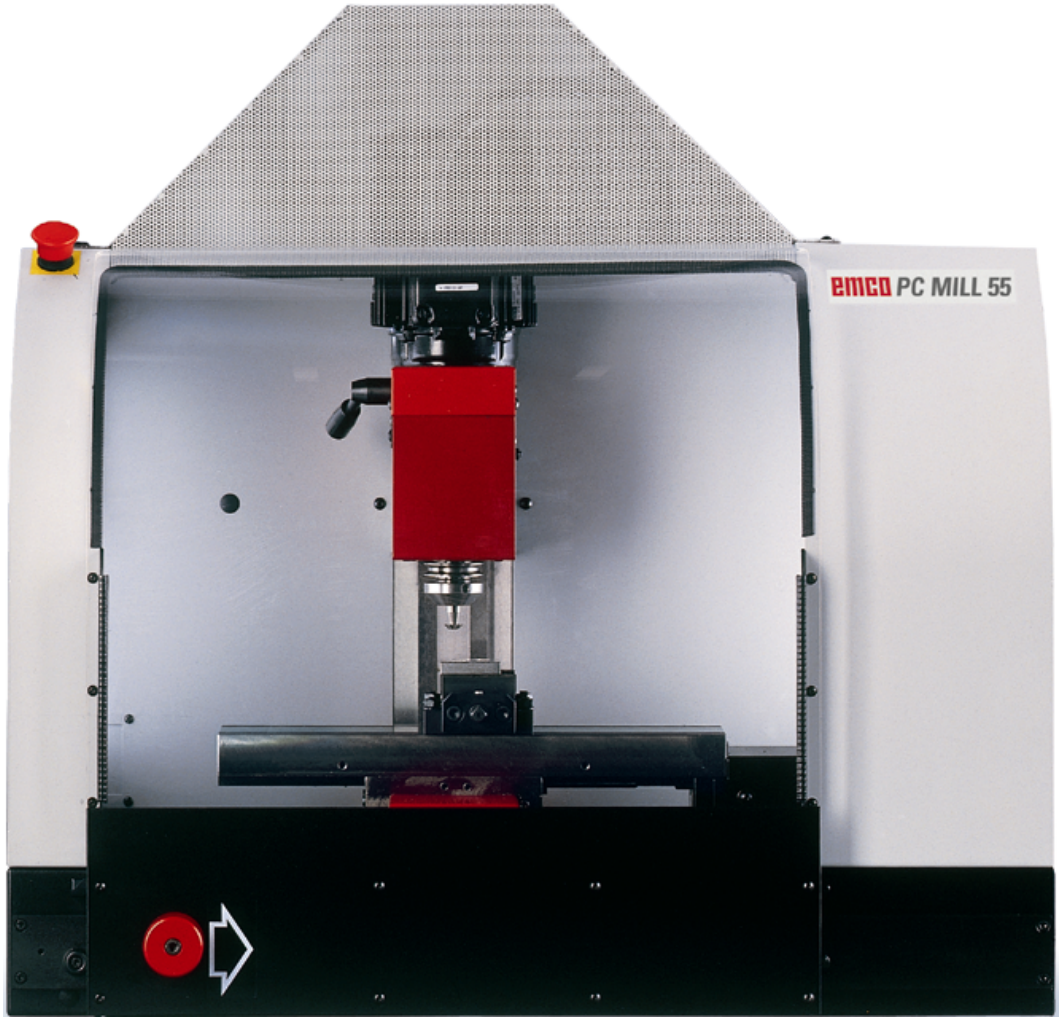
innovative machine tools

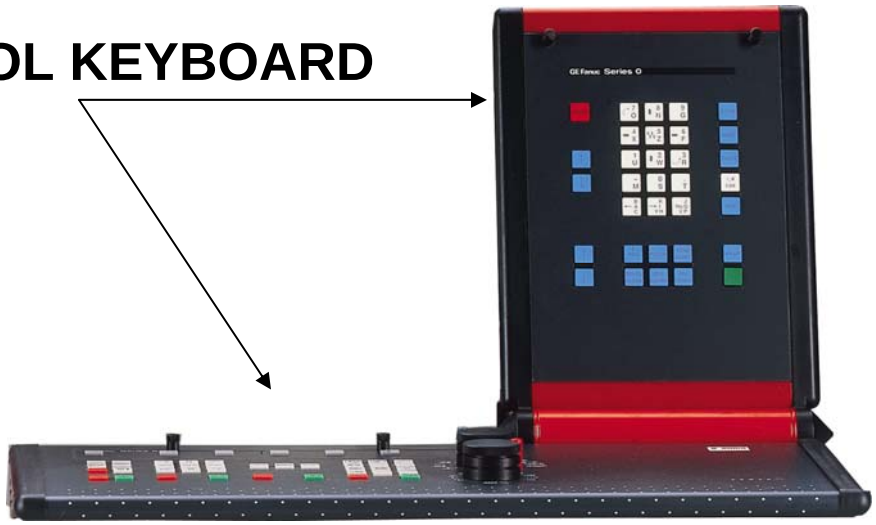


SIEMENS 840D
50/55 MILL
TRAINING GUIDE

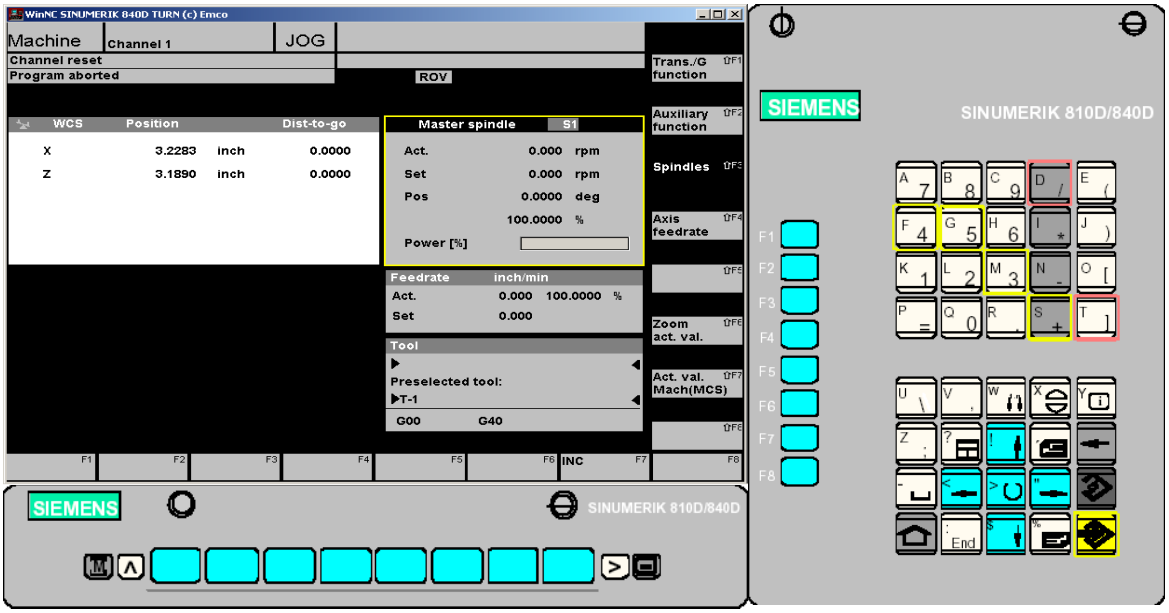
10/15/03 Version 1
Made by EMCO
Authored by Chad Hawk

Training Index

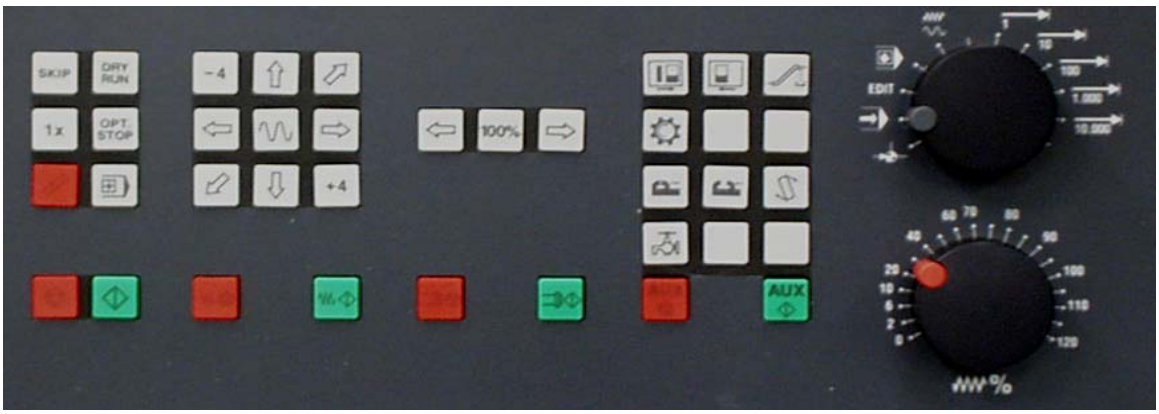
CONTROL KEYBOARD



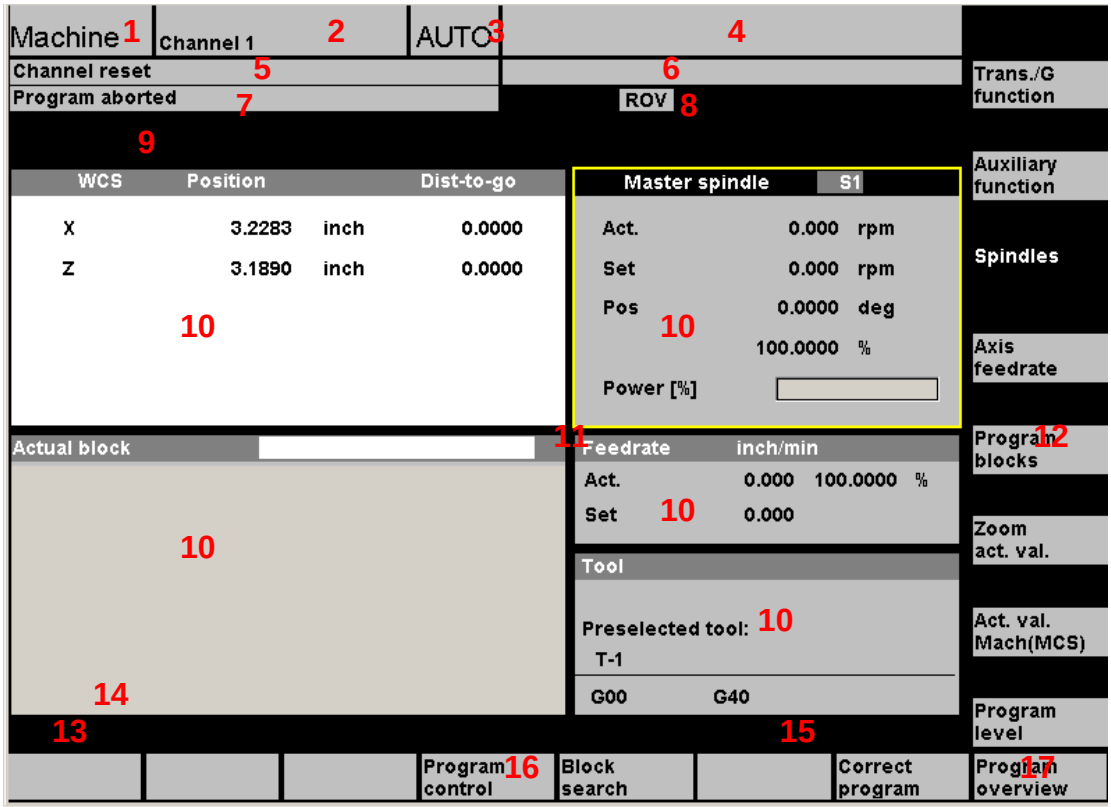
Siemens 840D CONTROL



MACHINE CONTROL



SIEMENS 840D SCREEN



1. Display of the active Operating Area
2. Display of the active channel
3. Operating mode, when a sub mode is active, it also will be displayed (e.g. REF, INC)
4. Program path and name of the selected program
5. Channel status
6. Channel operating messages
7. Program status
8. Channel status display (SKIP, DRY, SBL)
9. Alarm and message line
10. Working window, NC display the working windows (program editor) and NC displays (feed, tool) available in the active Operating Area are displayed here.
11. The selected window is marked with a border and the headline is displayed inverted. The keyboard inputs are effective here.
12. Vertical soft keys These 8 fields show the functions of the keys right beside. (at the PC: Shift F1..F8)
13. When this symbol is displayed, the key ^ is active (jump back to superior menu is possible).
14. Dialogue line with operator notes
15. When this symbol is displayed, the key is active (information available).
16. Horizontal soft keys These 8 fields show the functions of the keys below. (at the PC: F1..F8)
17. When this symbol is displayed, the key > is active (more soft key functions available in this line).

SIEMENS 840D KEYS

= Direct jump to the Operating Area Machine

= Jump back to the superior menu (recall)

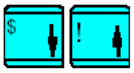
= Expanding the soft key line in the same menu

= Show basic menu (selection Operating Areas)
If pressed again jump back to the previous menu

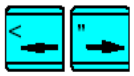
= Confirm alarm

= Show information for the actual operating status - works only when the dialogue line shows an "*i*".

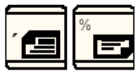
= Select window (when several windows are on the screen)
Keyboard inputs are valid for the selected window only.



= Cursor down / up



= Cursor left / right



= Leaf backward / forward

= Blank

= Clear (Backspace)

= Selection key / Toggle key

- **Selection of predefined input values in input fields and lists, which are marked with this symbol**
- **Activate / deactivate switch box / radio button**

☒◉ **= active**
☐○ **= not active**

= Edit key / Undo

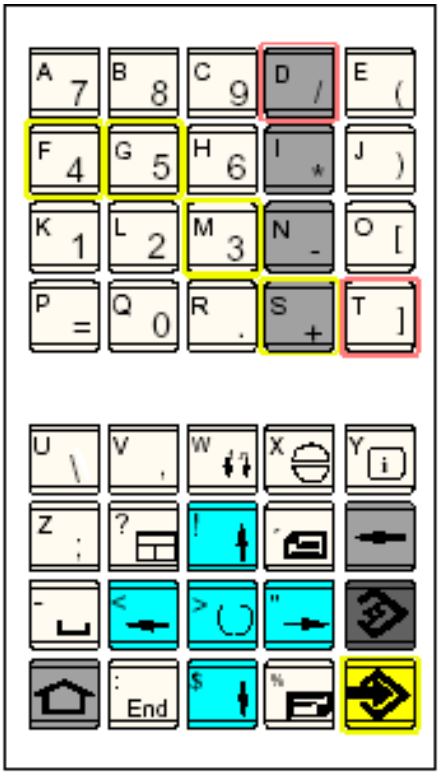
- **Switch to edit mode in tables and input fields**
- **Undo function for table elements and input fileds (leaving a filed with this key does not store the entered value but reestablishes the old value)**

= End Jump to line end (list end)

= Input key

- **Take over an edited value**
- **Open / close directory**
- **Open file**

= Shift key



Address and Numeric Keyboard

The shift key bottom left shifts to the second key function (indicated in the left top edge of the keys).

Example:

Leaf backward
Comma

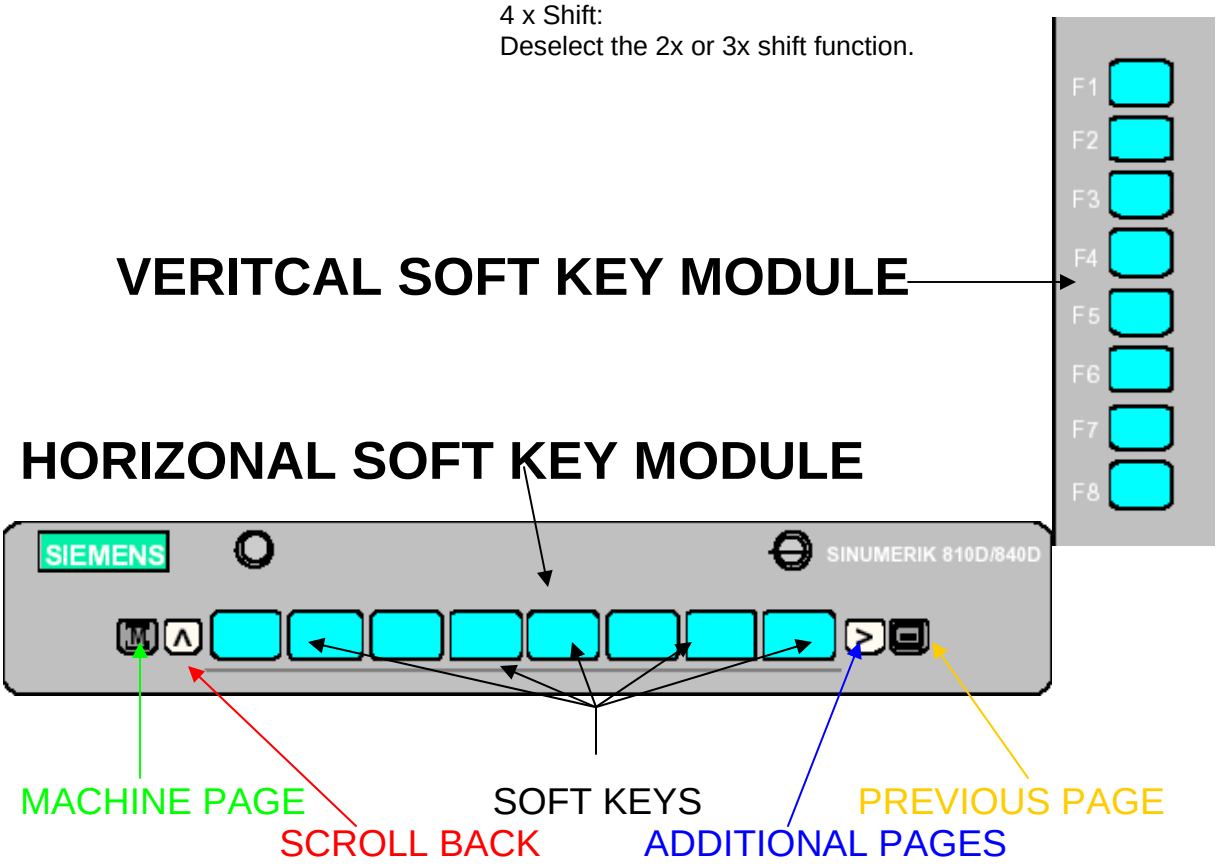
Double-Shift Function

1 x Shift:
For the following key press the second key function will be done, for all following inputs the first key function.

2 x Shift:
For all following key presses the second key function will be done (shift lock).

3 x Shift:
For the following key press the first key function will be done, for all following inputs the second key function.

4 x Shift:
Deselect the 2x or 3x shift function.



MACHINE KEYS

MACHINE FUNCTION KEYS

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

= Press for test run without spindle on (remove raw material from vise)

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

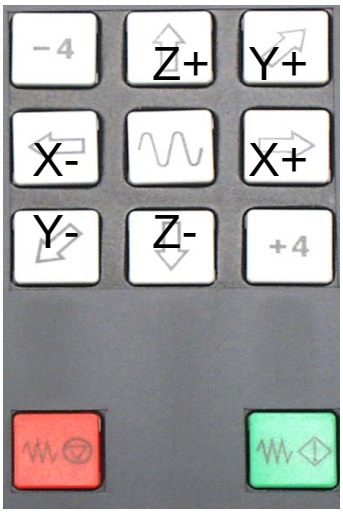
= (Optional stop) for programs with (m1)

= (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

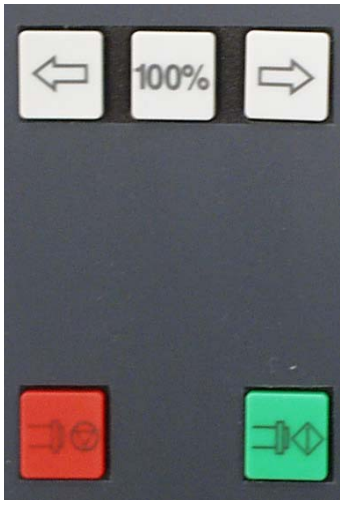


DIRECTION KEYS

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

ACCESSORY FUNCTIONS

Arrow right door open
Arrow left door closed

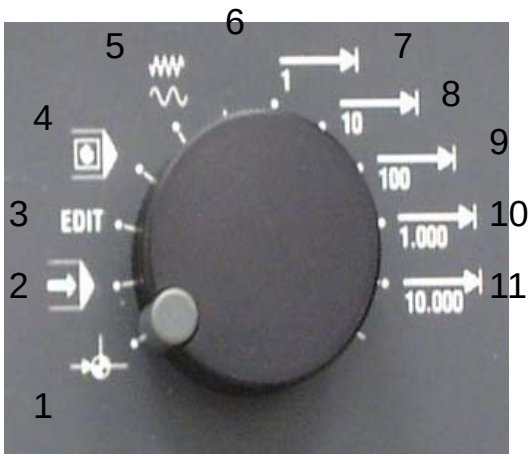
Press for Rotary axis Indexing

Press once vise closed
Press once vise open

Press once coolant on
Press again coolant off

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

MODE DIAL



(1) REF = Zero / Reference or Home mode

(2) 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

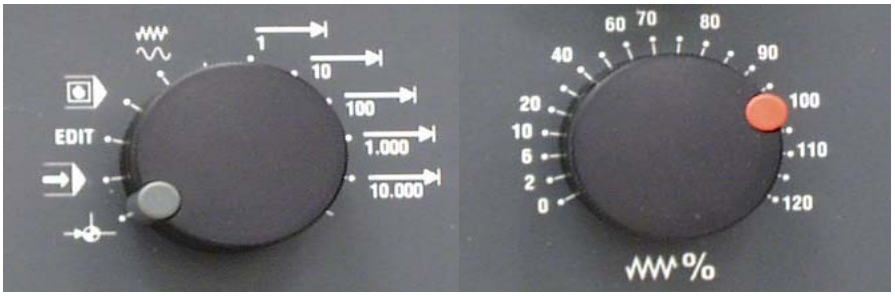
FEED OVERRIDE DIAL

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

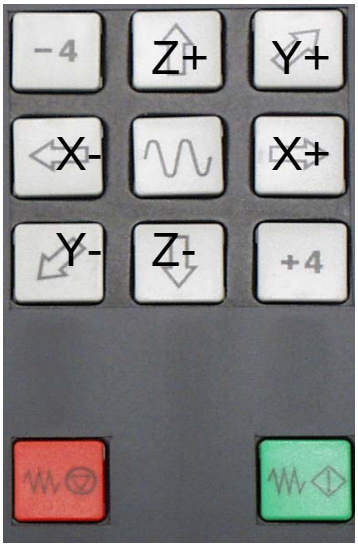
Turning Machine On/Entering Siemens Software

Referencing the Machine

1. Move the MODE dial to ZRN position also know as Reference make sure your feed rate is not on “**0**”



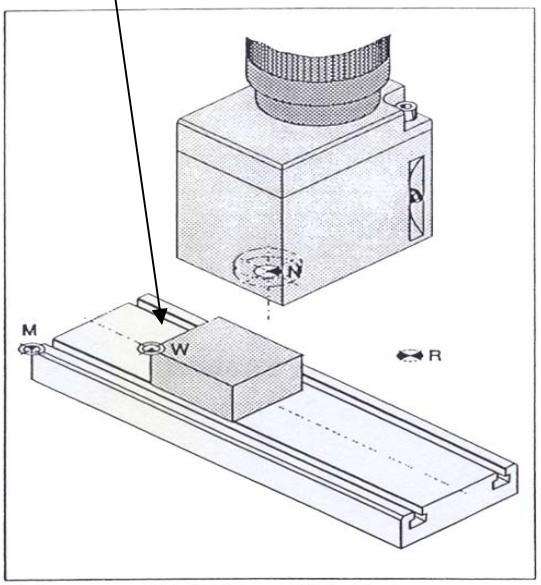
2. Make sure the Door is closed
3. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
4. Press the X- (arrow pointing left) this references the X axis
5. Press the Y- (arrow pointing left) this references the Y axis



Note: Every time you enter Siemens 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 Spindle nose



Reference points in the working area

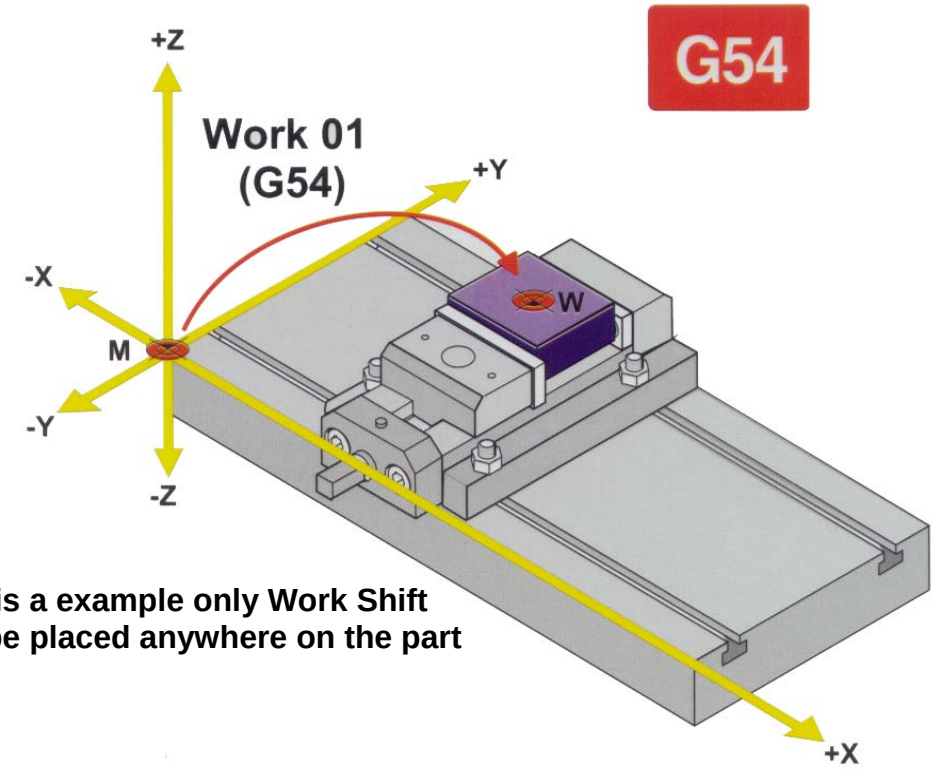
Reference Points of the EMCO Milling Machines

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.



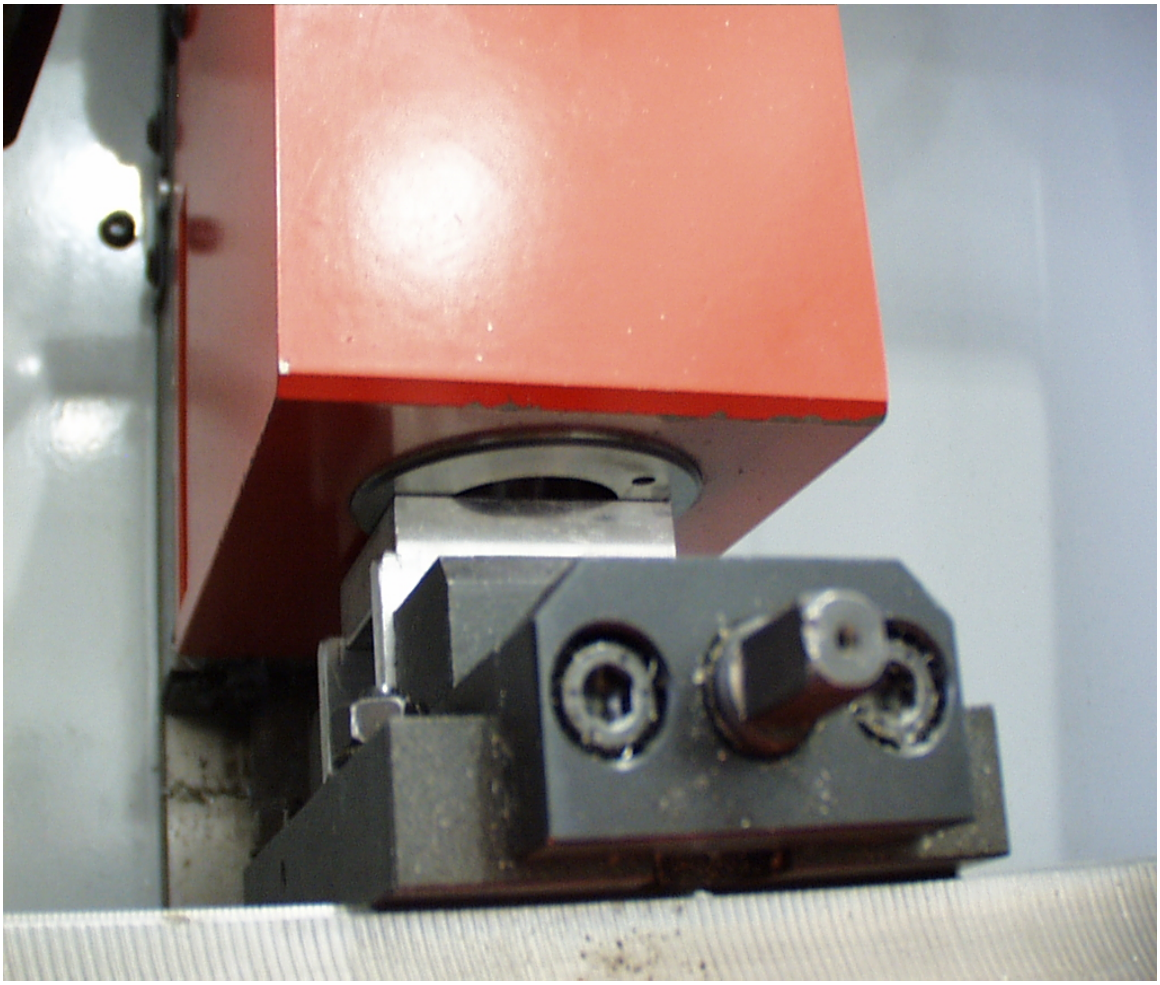
This is a example only Work Shift can be placed anywhere on the part

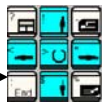
Work Shift:

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

Note: Use Feed Dial or Steps to approach at a slower feed rate.

Use piece of paper between nose and Work Piece

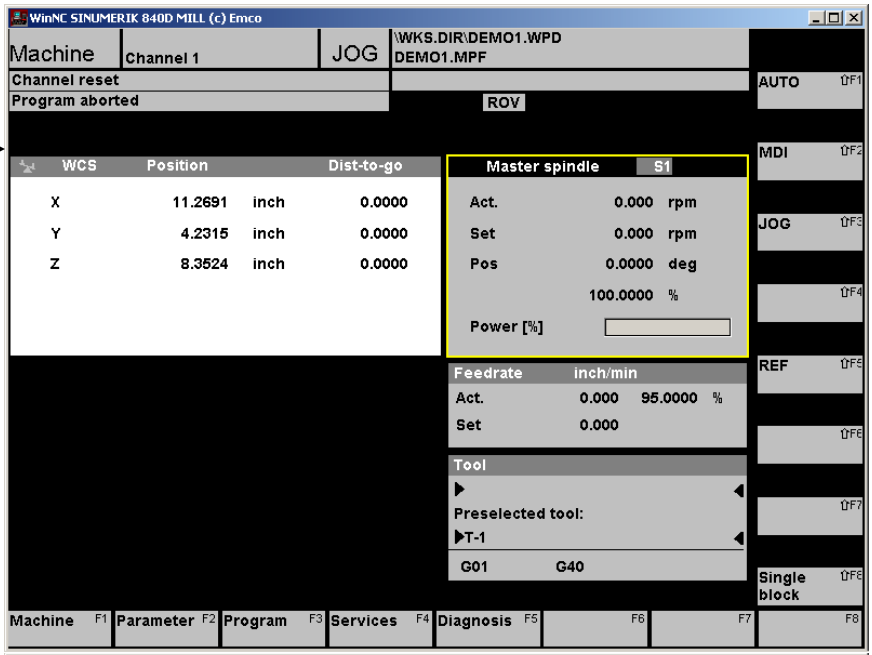


3. Press the  button on the horizontal soft keys
4. Press the Blue horizontal soft key for Parameter F2 (Picture 1)
5. Press the Blue horizontal soft key for Work offset F4 (Picture 1) 
6. Cursor down to Z using cursor keys
7. Press the Blue vertical soft key Accept position ⇧F5 (Picture 2)
   Write the value down

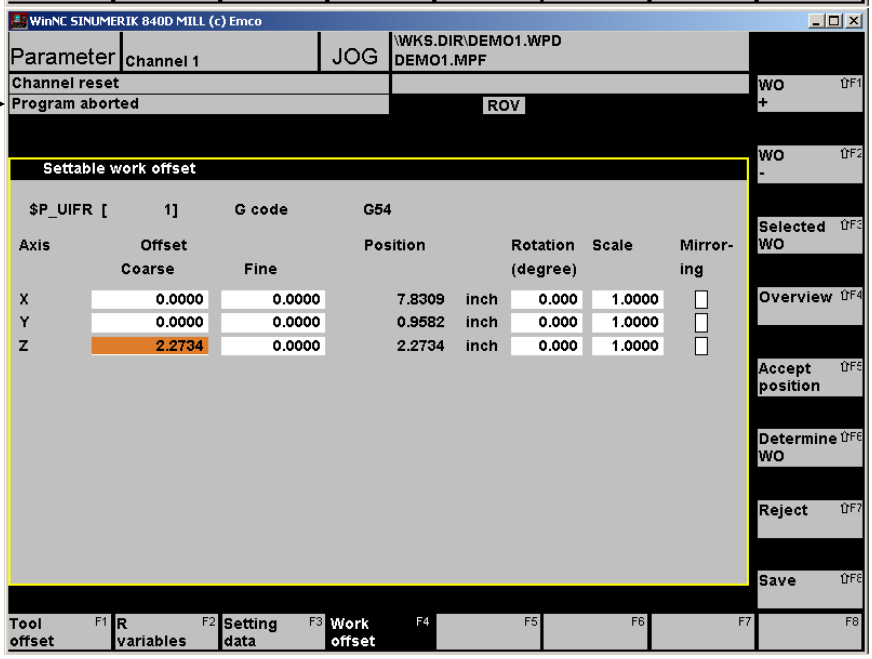
(This value is the distance from the Spindle Nose to the end of the Work Piece)

8. Press the Blue vertical soft key for Save ⇧F8 (Picture 2)

Picture 1



Picture 2



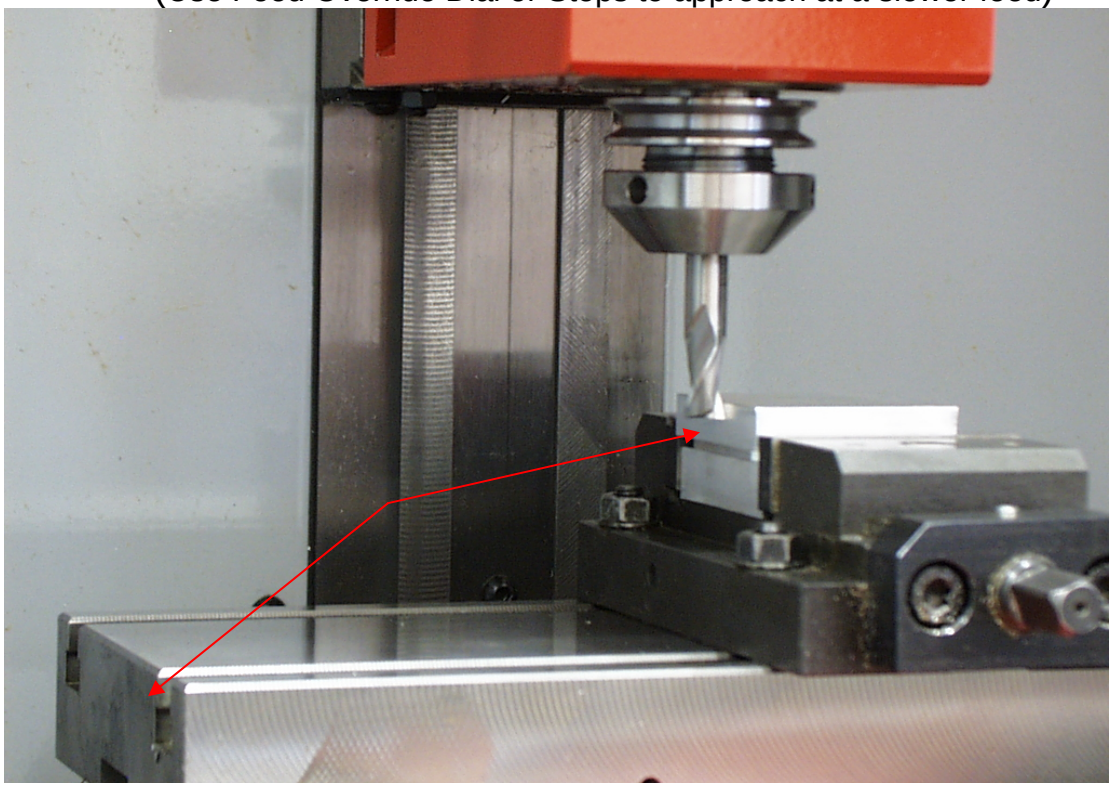
9. Jog Spindle up away from WORK PIECE using Z+
10. Place a edge finder or tool in the Spindle (Example uses 3/8 end mill)
11. Move the MODE Dial to MDI position

12. Press the button on the horizontal soft keys then press Blue horizontal soft key for Program F3
13. Press the button on the horizontal soft keys for active screen on MDI program
14. Type S1000 M3 then press input button

    S1000 = Spindle speed M3 = spindle on clockwise

15. Press CYCLE START (make sure door is closed)
16. Move the MODE Dial to JOG position
17. Jog the Tool to the left side of the Work Piece & touch using the Direction keys.

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



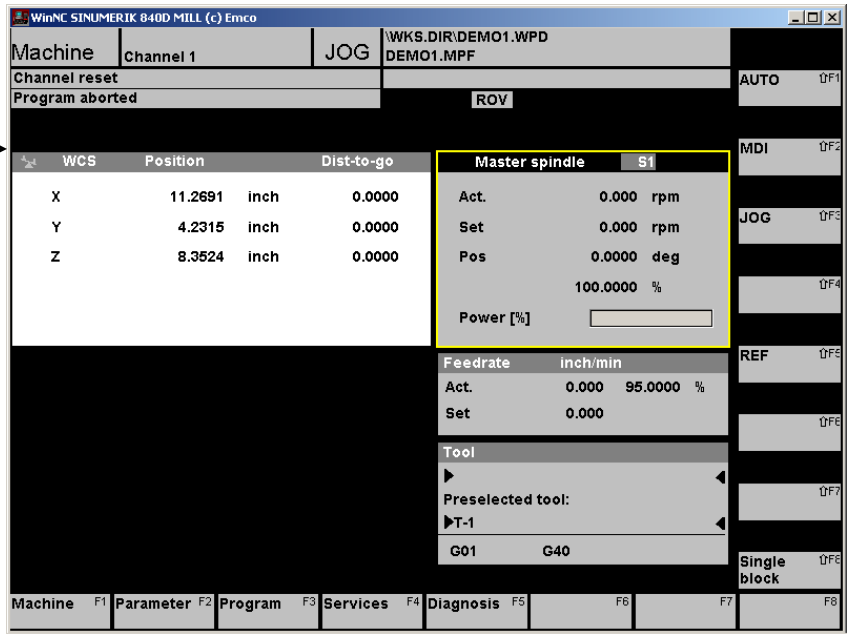
Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

18. Press the  button on the horizontal soft keys
19. Press the Blue horizontal soft key for  (Picture 1)
20. Press the Blue horizontal soft key for  (Picture 1)
21. Cursor down to X using cursor keys 
22. Press the Blue vertical soft key  (Picture 2)
    Write the value down

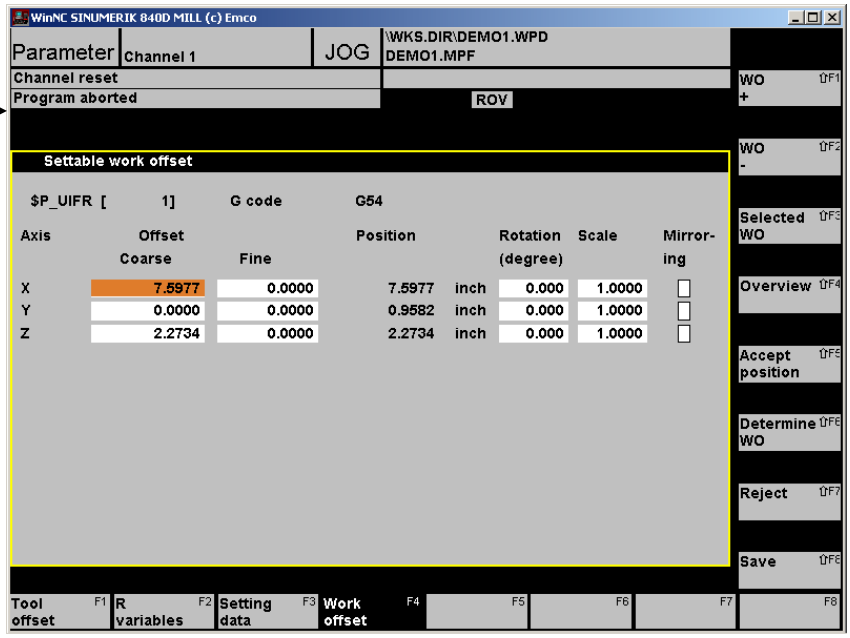
(This value is the distance from the left end of the table to the left side of the Work Piece)

23. Press the Blue vertical soft key for  (Picture 2)

Picture 1 →

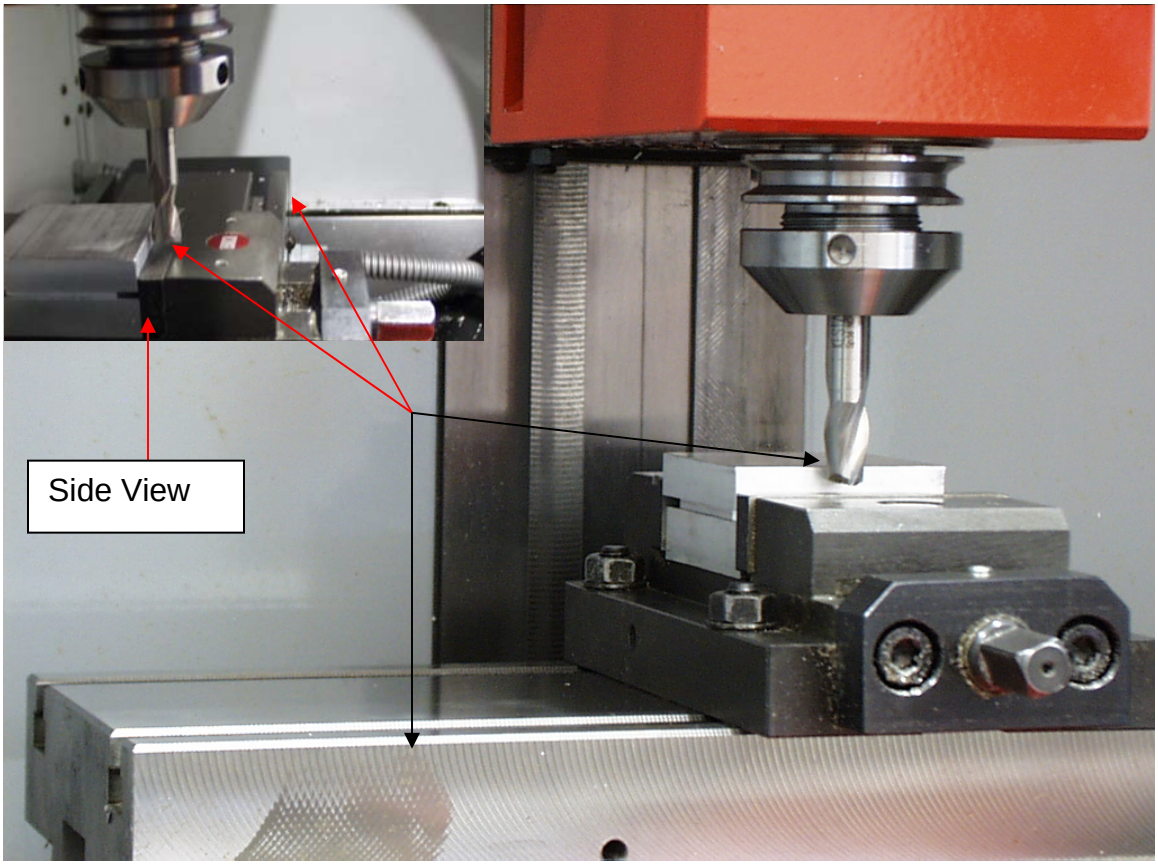


Picture 2 →



24. Spindle up away from WORK PIECE using Z+
25. 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)



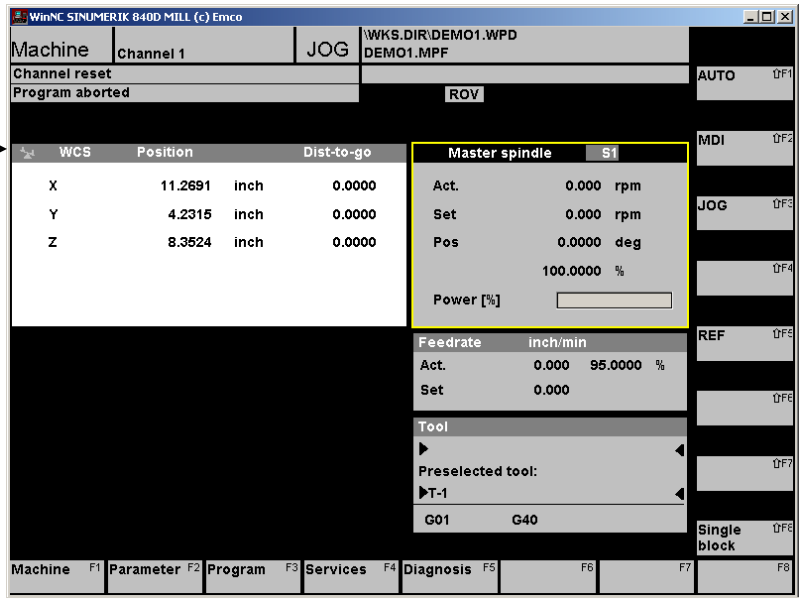
Note: Machine 0 in Y is the center of the spindle to the Front of the Machine bed.

26. Press the  button on the horizontal soft keys
27. Press the Blue horizontal soft key for  (Picture 1)
28. Press the Blue horizontal soft key for  (Picture 1)
29. Cursor down to Y using cursor keys 
30. Press the Blue vertical soft key  (Picture 2)
    Write the value down

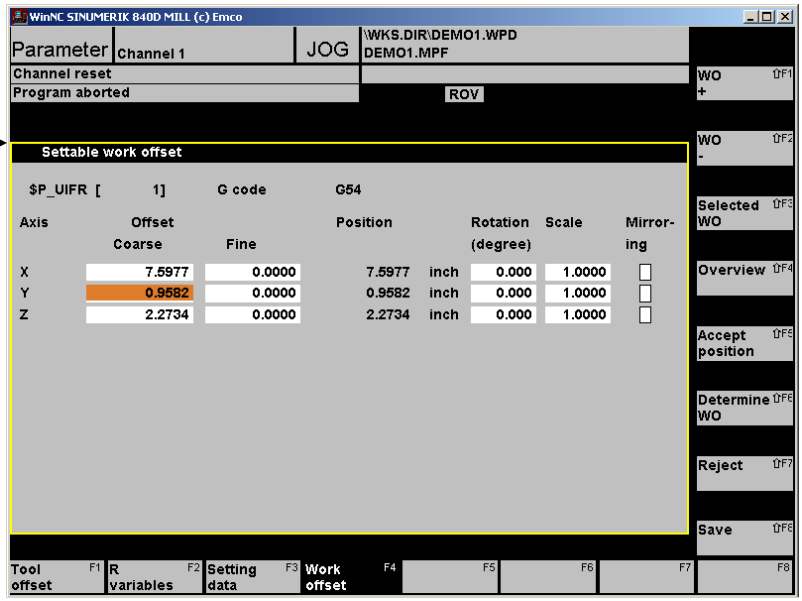
(This value is the distance from the left end of the table to the left side of the Work Piece)

31. Press the Blue vertical soft key for  (Picture 2)

Picture 1 →

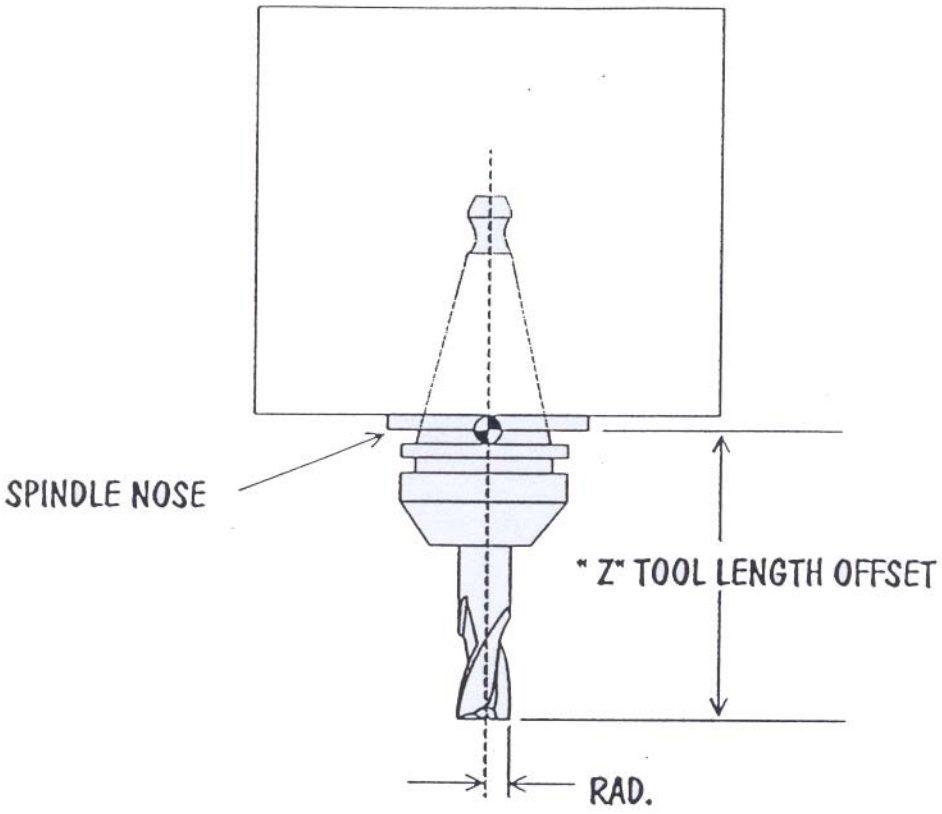


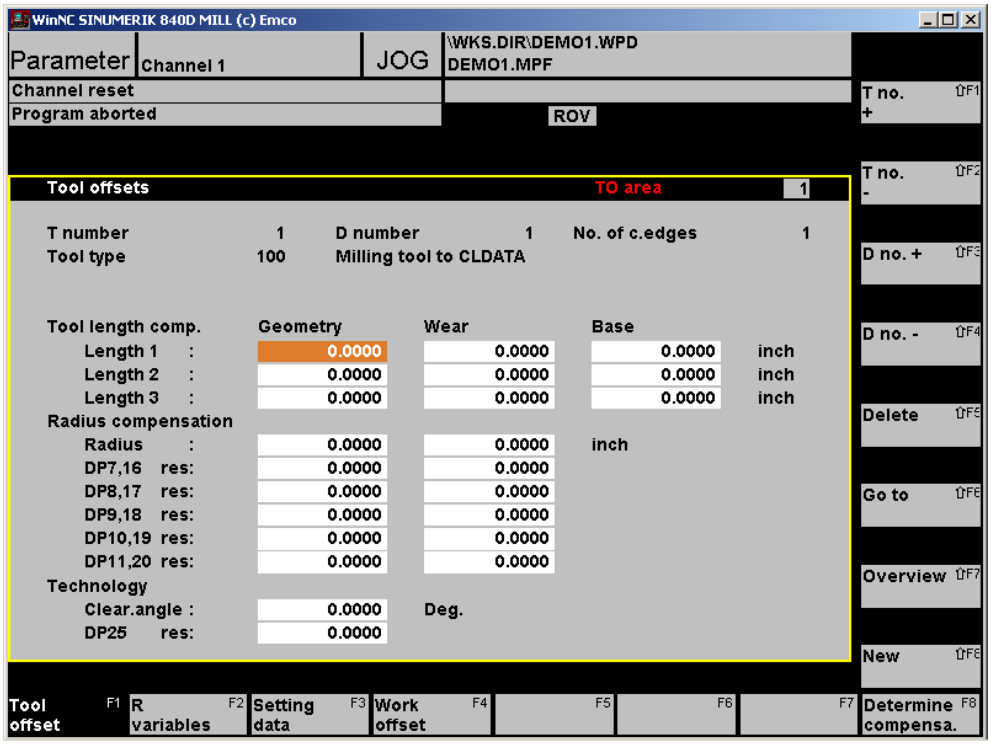
Picture 2 →



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

TOOL OFFSET


SPINDLE NOSE
" Z" TOOL LENGTH OFFSET
RAD.


WinNC SINUMERIK 840D MILL (c) Emco
Parameter
Channel 1
JOG
\WKS.DIR\DEMO1.WPD
DEMO1.MPF
Channel reset
Program aborted
ROV
T no. +
T no. -
D no. +
D no. -
Delete
Go to
Overview
New
Tool offsets
TO area 1
T number 1
D number 1
No. of c.edges 1
Tool type 100
Milling tool to CLDATA
Tool length comp.
Geometry
Wear
Base
Length 1 : 0.0000 0.0000 0.0000 inch
Length 2 : 0.0000 0.0000 0.0000 inch
Length 3 : 0.0000 0.0000 0.0000 inch
Radius compensation
Radius : 0.0000 0.0000 inch
DP7,16 res: 0.0000 0.0000
DP8,17 res: 0.0000 0.0000
DP9,18 res: 0.0000 0.0000
DP10,19 res: 0.0000 0.0000
DP11,20 res: 0.0000 0.0000
Technology
Clear.angle : 0.0000 Deg.
DP25 res: 0.0000
Tool offset
R variables
Setting data
Work offset
Determine compensa.

Tool Offsets

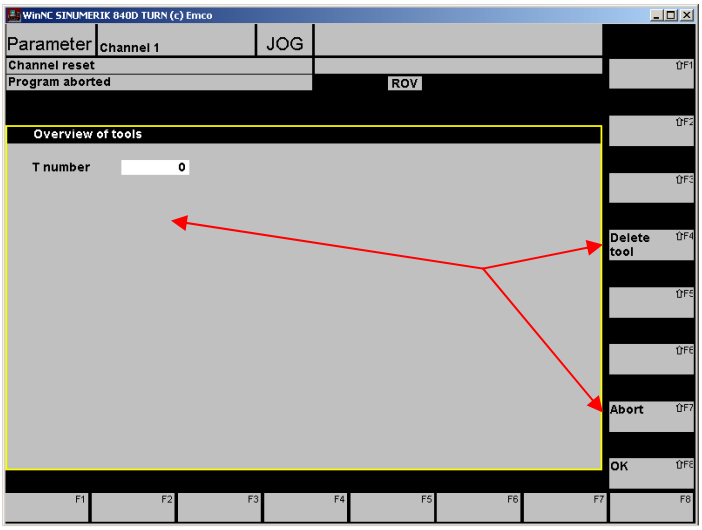
When the Software is loaded there will be tools created already

A. Press the Blue horizontal soft key for **Tool offset** F1

B. Press the Blue vertical soft key for **Overview** ⇧F7

C. Press the Blue vertical soft key for **Delete tool** ⇧F4 until there are no tools remaining

D. Press the Blue vertical soft key for **Abort** ⇧F7

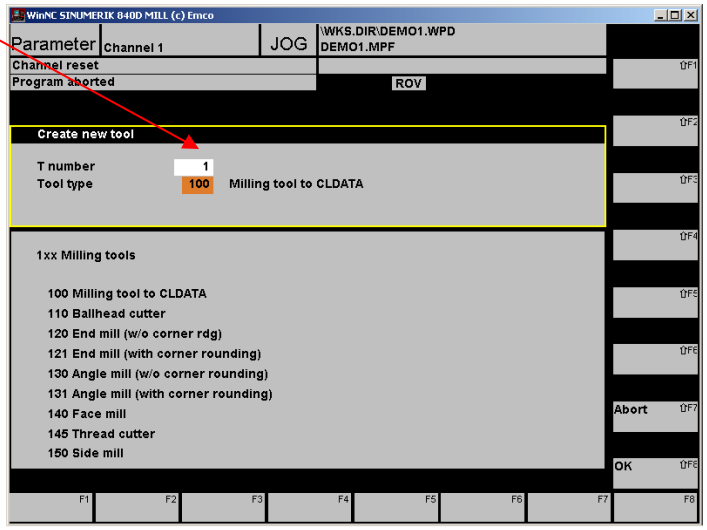


1. Press the Blue vertical soft key for **New** ⇧F6 then press the Blue vertical soft key for **New tool** ⇧F4

2. Cursor to the T number and type in 1

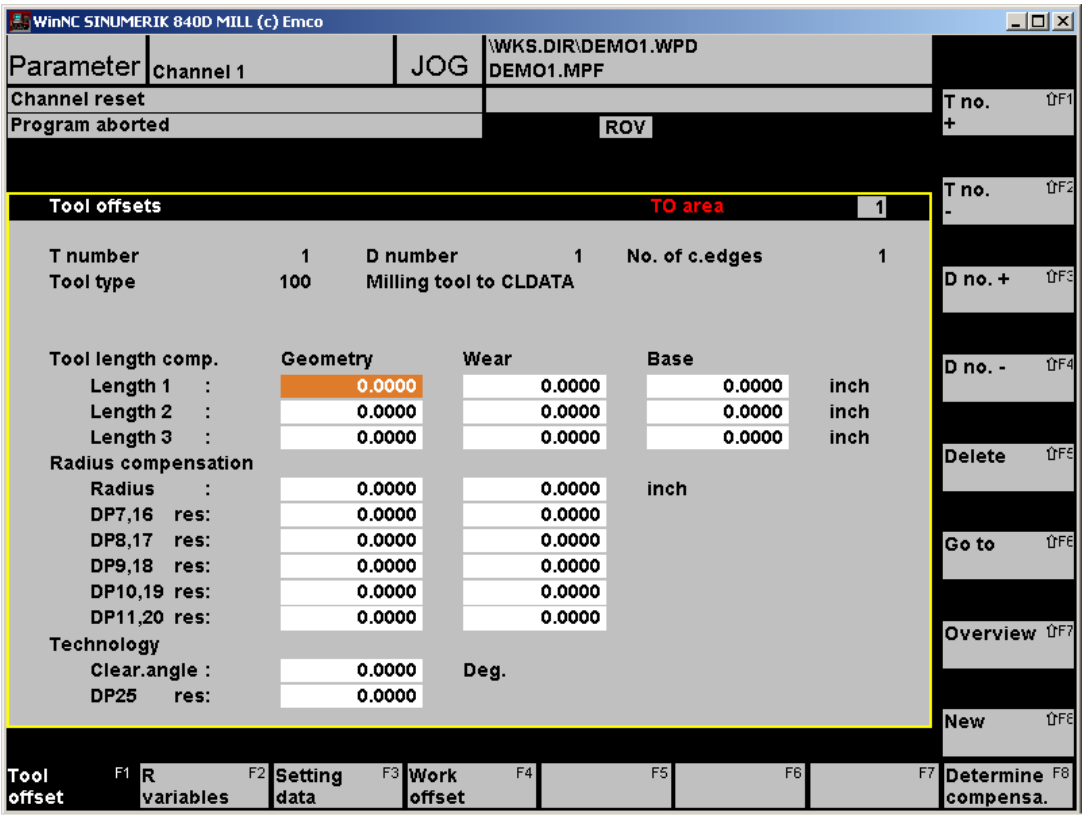
3. Cursor down to Tool type & type in 100 for Milling tool to CLDATA

4. Press the Blue vertical soft key for **OK** ⇧F8



5. Place a tool to be measured in the spindle
6. Jog Tool tip down & touch the Top of the Work Piece

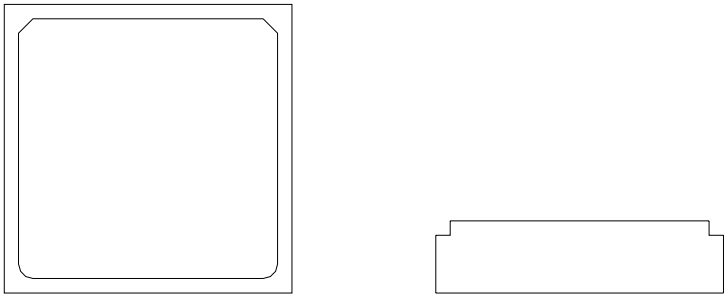
   (Use Feed Dial or Steps to approach at a slower feed)



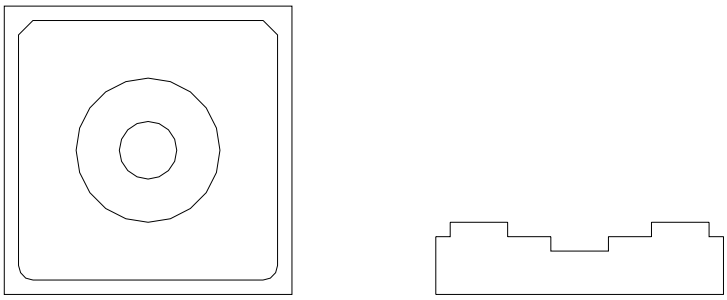
7. To set more Tools Repeat Steps 1thru 5
   - Drills & Taps need no Radius set for them

Program Training

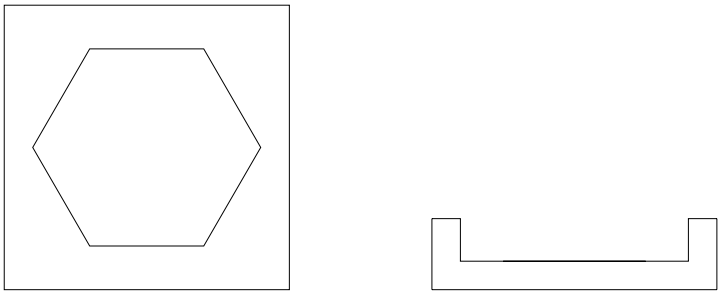
Program O0001



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

- **DELETE ALL PROGRAMS**

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

     Example: **<u>O – 9999</u>**

- **DELETE A WORD**

  1. Press letter then number
  2. Press delete button 

     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; AFORE N5;

     <u>(S1000)</u> 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

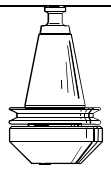
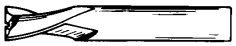
M00 Programmed stop, unconditional
M01 Programmed stop, conditional
M03 Spindle ON clockwise
M04 Spindle ON counter clockwise
M05 Spindle OFF
M25 Open clamping vice
M26 Close clamping vice
M30 Main program end with new start of program
M71 Blow-off ON
M72 Blow-off OFF
M98 Subroutine call-up
M99 Subroutine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

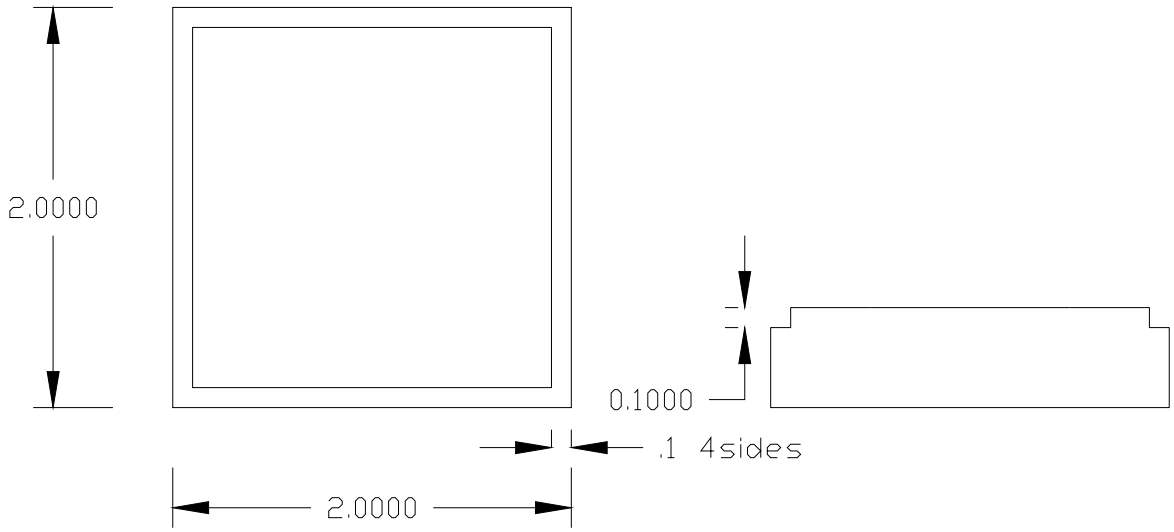
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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

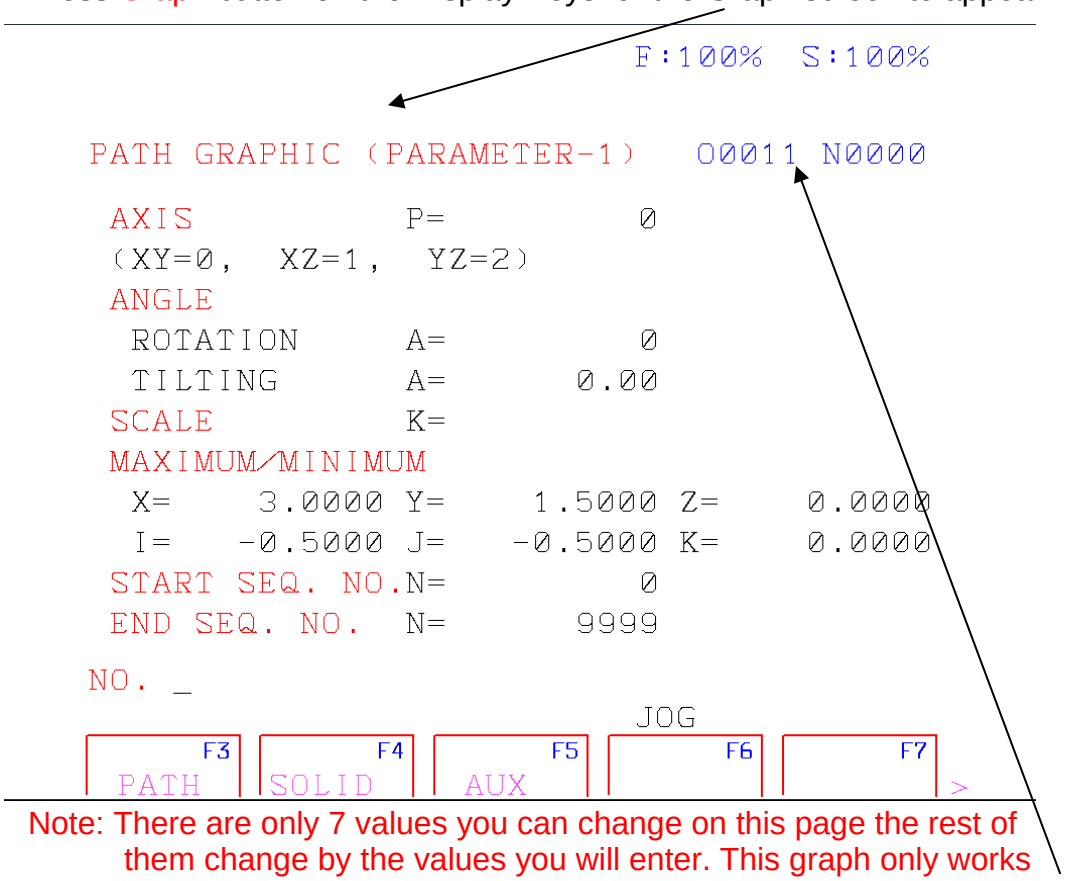
Program O0001



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

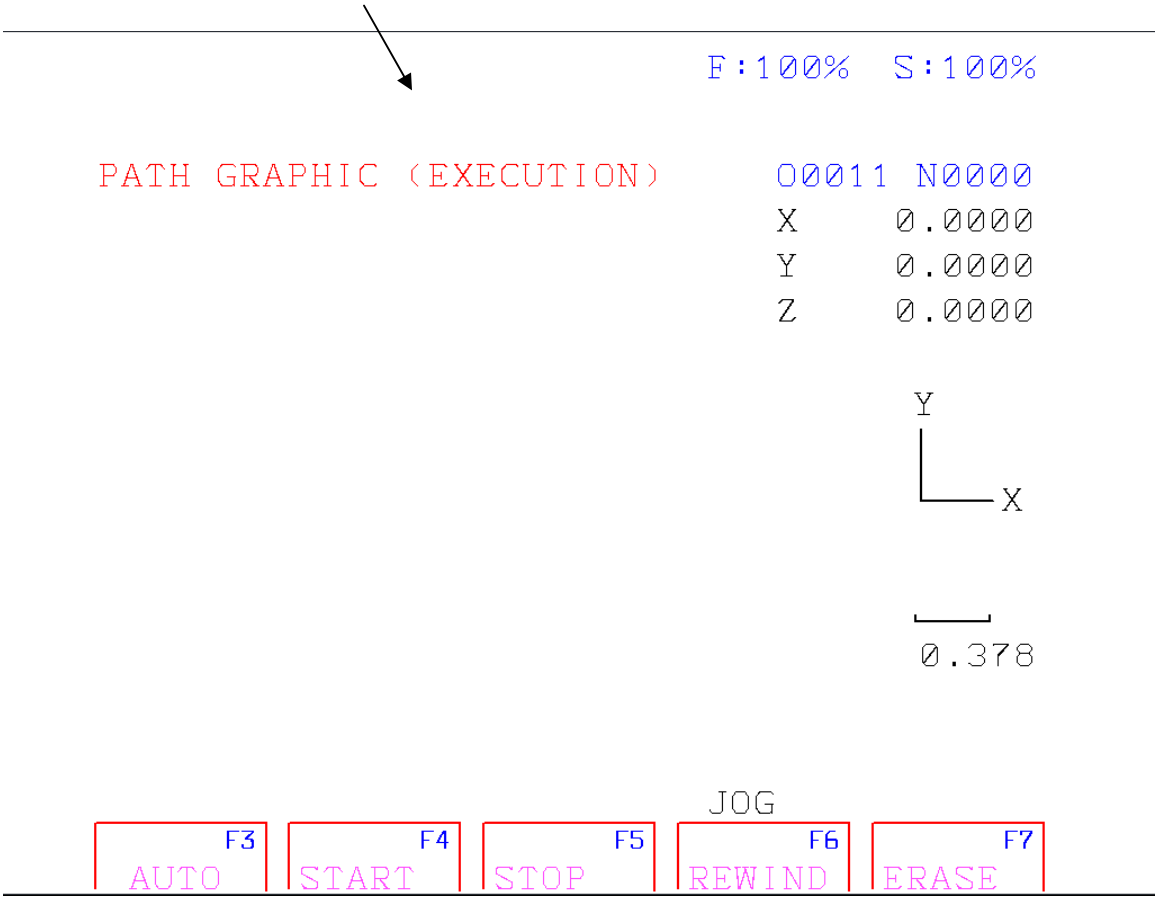
1. Press Graph button on the Display Keys for the Graph screen to appear



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

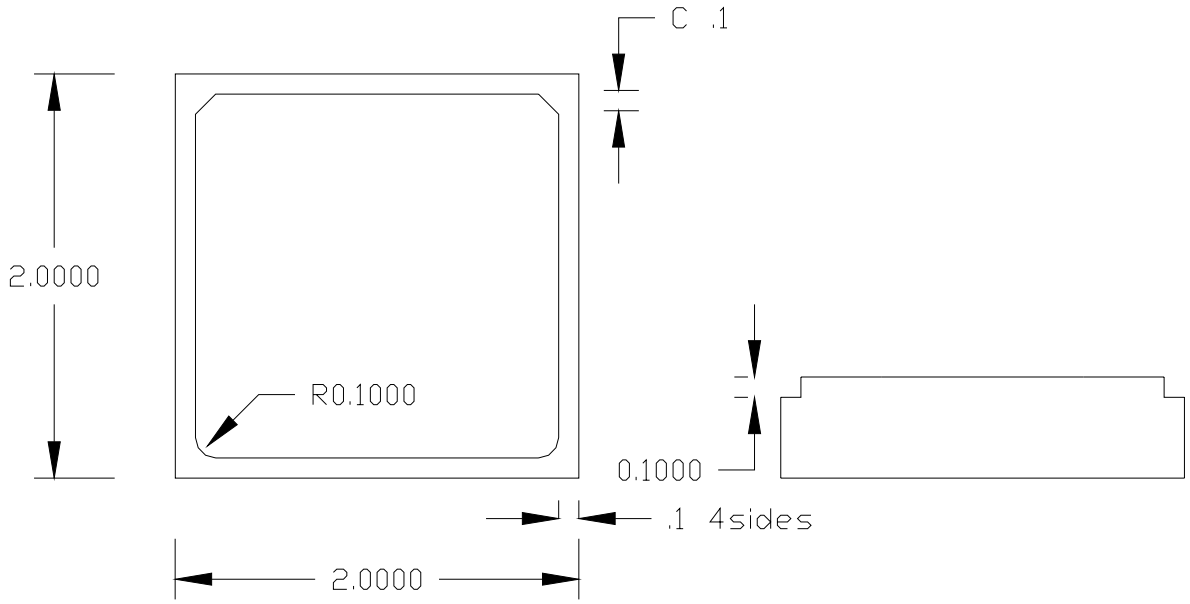
- **Input Program into Fanuc Software from Drive unit**
    1. Press the **Program** display key
    2. Type program number to be read

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

    3. Press (**Input)** key

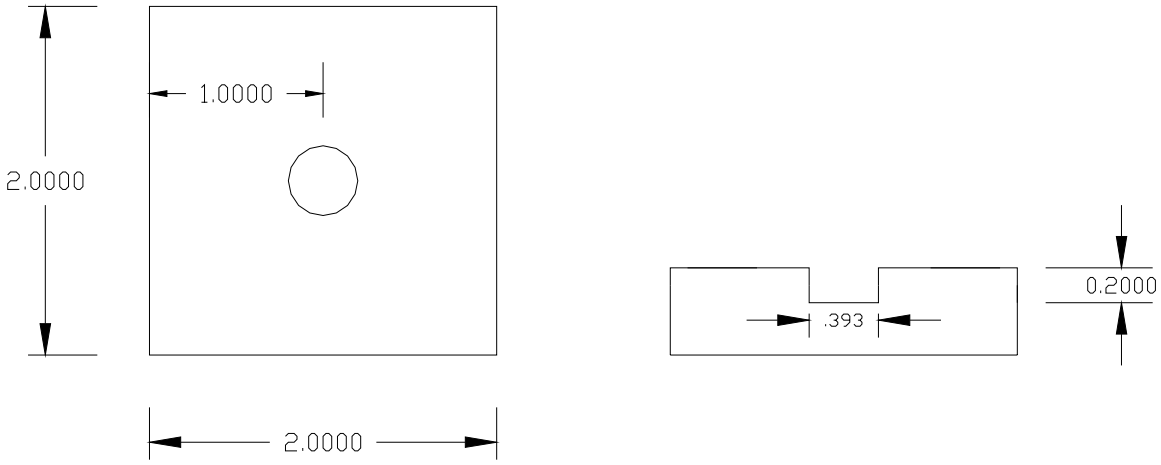
- **Input Offsets into Fanuc Software from Drive unit**
    1. Press the **Menu Offset** display key
    2. Press **(Input)** key

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 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

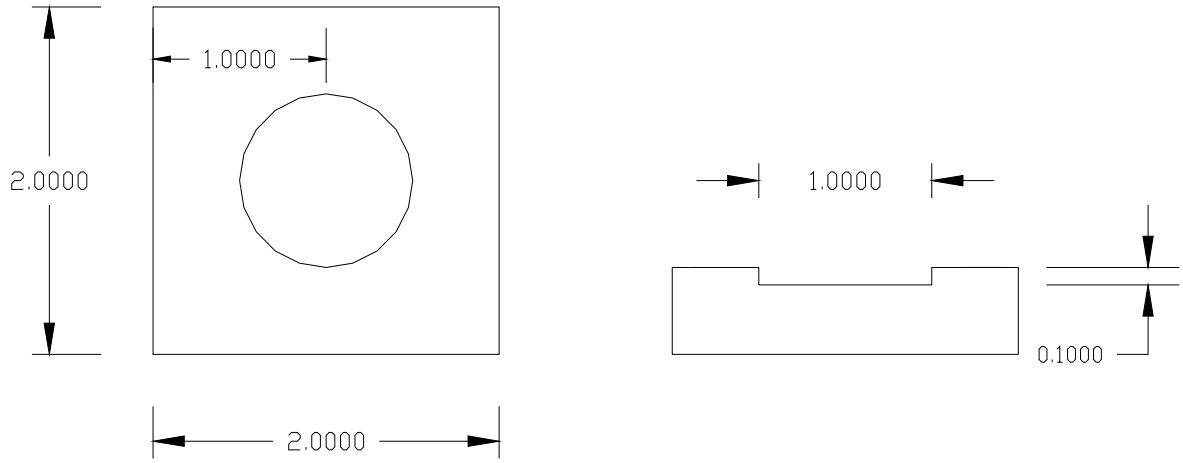
N40 G80

N45 G28 Z3

N50 G28 X2.5 Y2.5

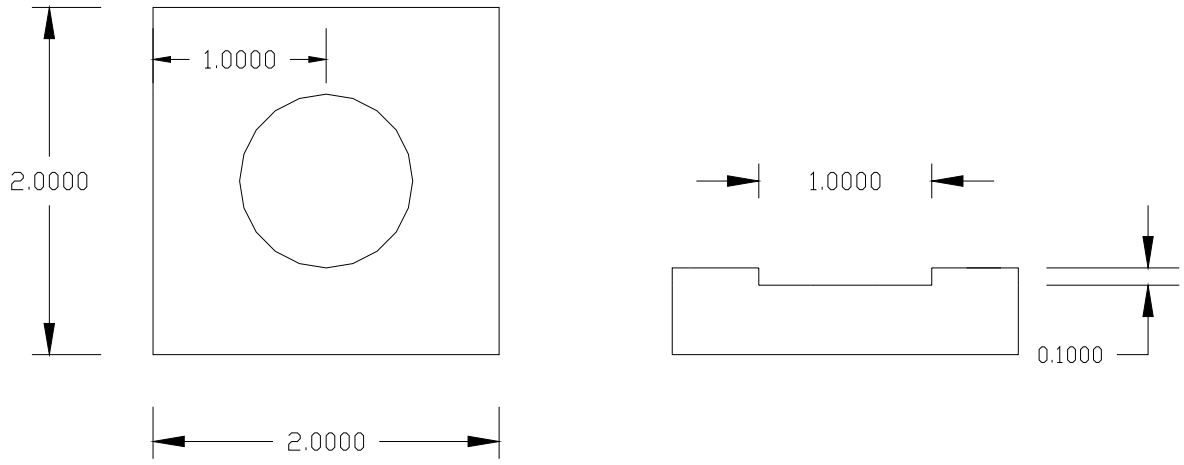
N55 M30

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

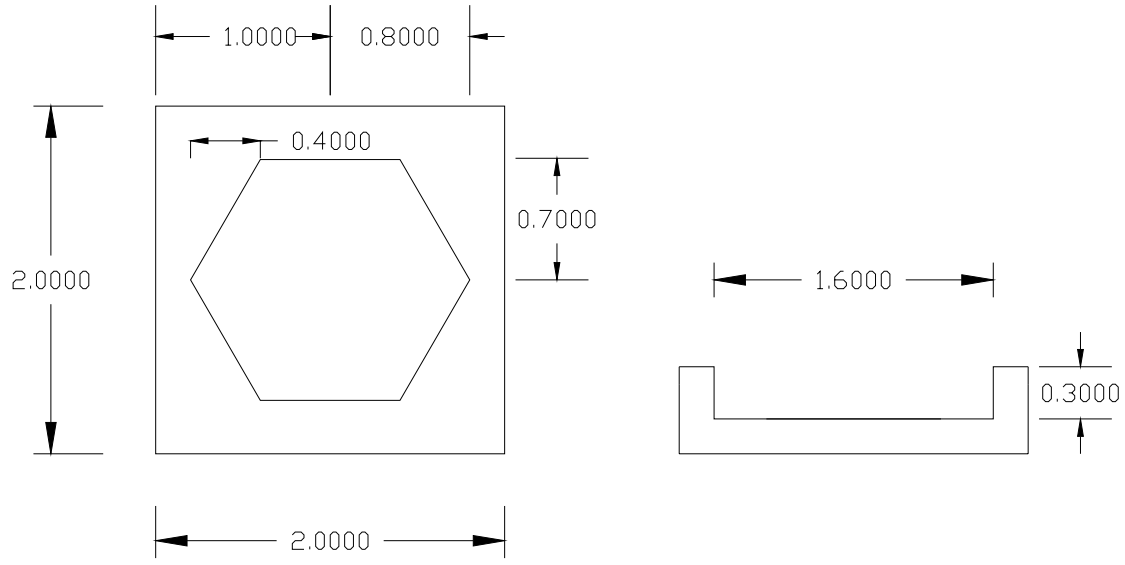
N15 M98 P010002

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