



GE FANUC 21 CM 105 TURN TRAINING GUIDE

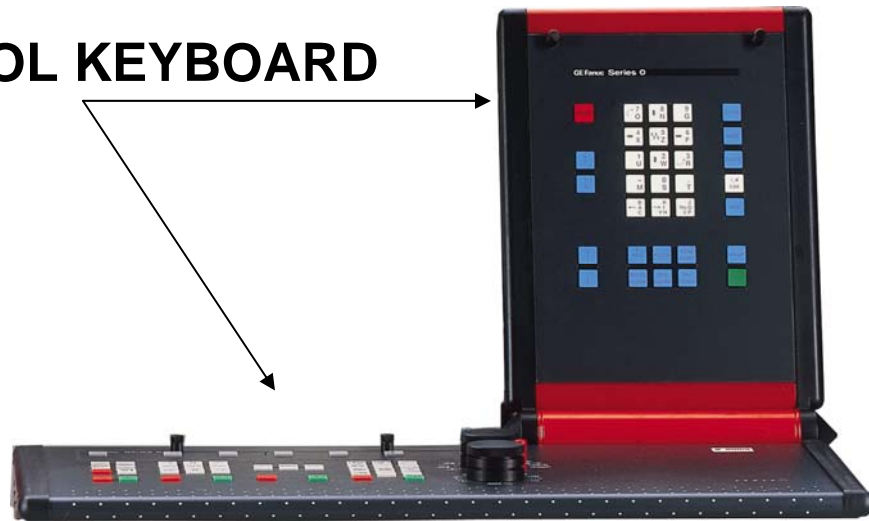
4/29/08 Version 1
Made by EMCO
Authored by Chad Hawk

Training Index

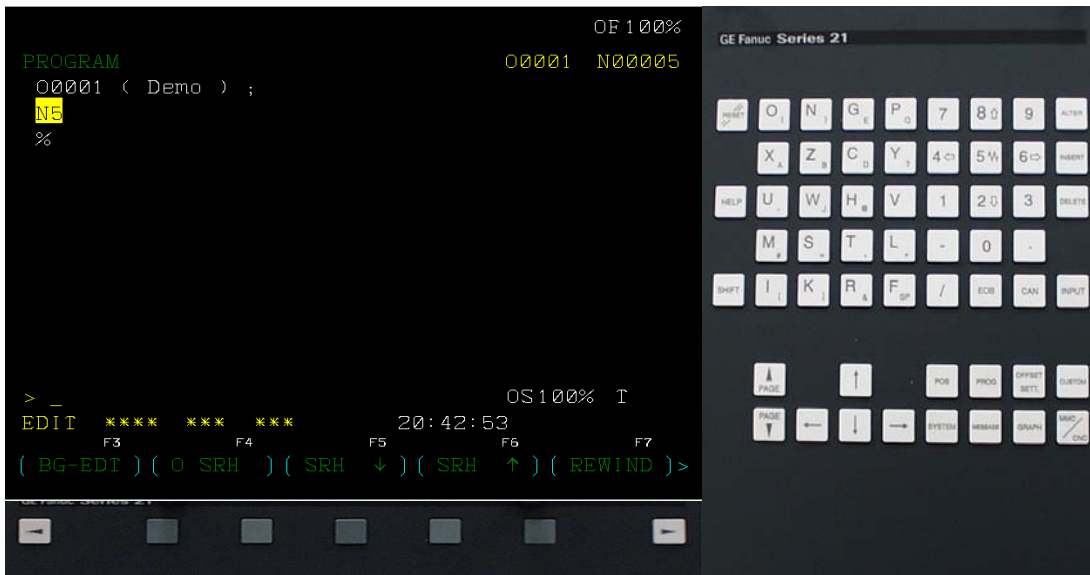
Control Keyboard.....	Pg 1
• Fanuc 21 Control	
• Machine Control	
Fanuc 21 Screen	Pg 2
Fanuc 21 Keys	Pg 3
• Data Input Keys	
• Change Keys	
Cursor Movement Keys	Pg 4
• Function Keys (Display Keys)	
• Soft Key Module	
Machine Keys	Pg 5
• Machine Function Keys	
Direction Keys	Pg 6
• Spindle Override Keys	
• Accessory Functions	
Mode Dial	Pg 7
• Feed Override Dial	
Referencing the Machine	Pg 8
Work Shift Description (Picture)	Pg 9
Work Shift (How to do Work Shift)	Pg 10
Tool Offset Description (Picture)	Pg 12
Tool Offset (How to do Tool Offsets for X)	Pg 13
• Manually programming Turret Index	
• Manually programming Spindle on	
Tool Offset (How to do Tool Offsets for Z)	Pg 15

Program Training	Pg 17
Inserting a New Program	Pg 18
• Calling a Existing Program up	
• Insert a word	
• Insert a End of Block	
Delete a Program	Pg 19
• Delete all Programs	
• Delete a word	
• Delete a Block	
Cancel word	Pg 20
• Alter a word	
• Search for number Block	
• Search for word	
G Groups	Pg 21
G Codes	Pg 22
M Codes	Pg 23
• Used Addresses	
Program 1	Pg 25
2D simulation (Setup)	Pg 26
Input & Output the Programs & offsets thru the Fanuc Software	Pg 28
Running a Program	Pg 29
Program 1 (C & R)	Pg 30
Program 2 (G73 and G72 Description)	Pg 31
G78 Description	Pg 32
Program 2	Pg 33
Sub Programming.....	Pg 34
Test for Subs and I & J test.....	Pg 35
Program 3 (Ball)	Pg 36
Program 4 (Ball)	Pg 37
Appendix	

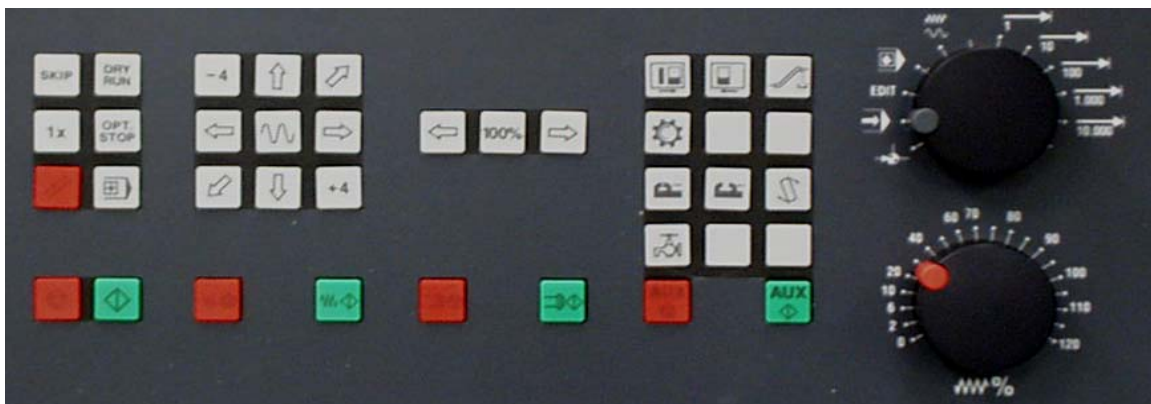
CONTROL KEYBOARD



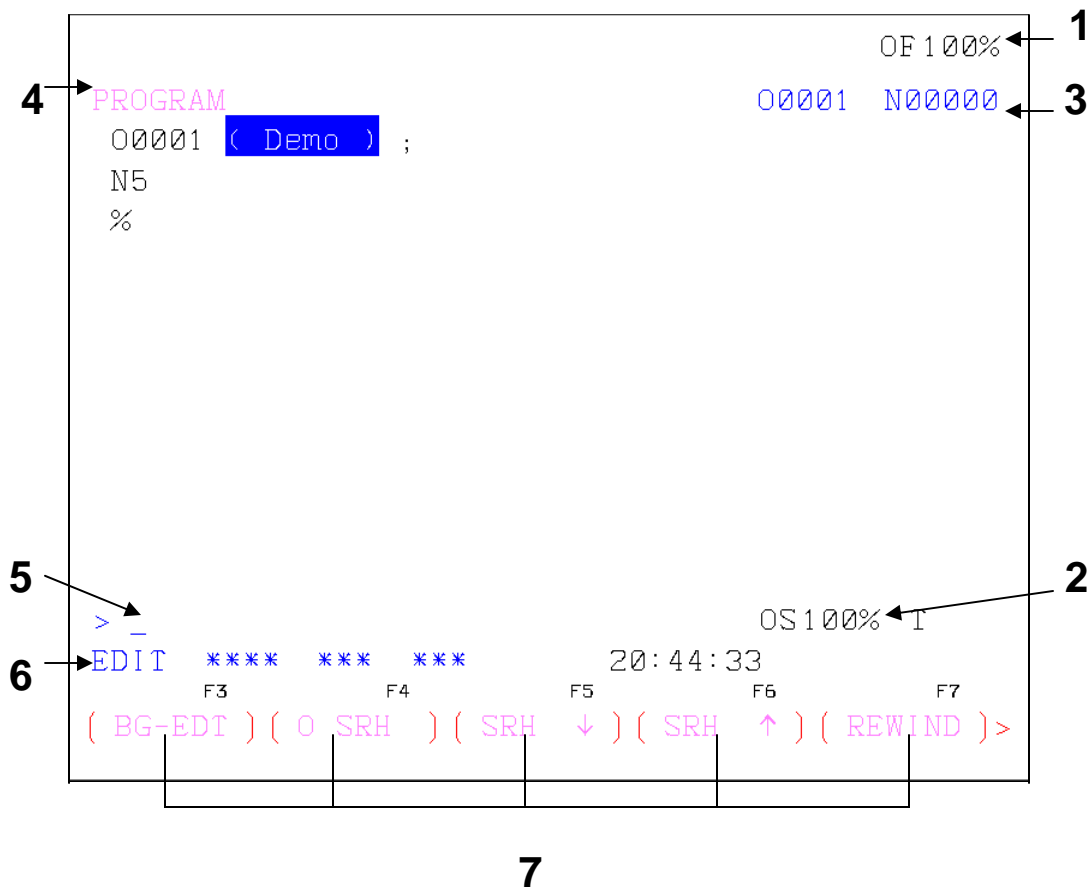
FANUC 21 CONTROL



MACHINE CONTROL



The Fanuc 21 Screen



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

FANUC 21 KEYS



**RESET = cancels most alarms, resets program,
interrupts programs**

CURSOR MOVEMENT KEYS



CURSOR UP = moves cursor up



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



PAGE UP = moves one page up



PAGE DOWN = moves one page down



CURSOR RIGHT = moves cursor right



CURSOR LEFT = moves cursor left

CHANGE KEYS



ALTER = alter word (replace word)



INSRT = insert word, create new program



DELET = deletes word / block or programs



EOB = end of block



CAN = deletes entries in the address one by one



INPUT = input offsets / words or numbers

DATA INPUT KEYS



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

FUNCTION KEYS (DISPLAY KEYS)



POS = displays actual, relative, machine positions



PROG = displays program, library page



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



SYSTEM = displays parameters, diagnostic pages

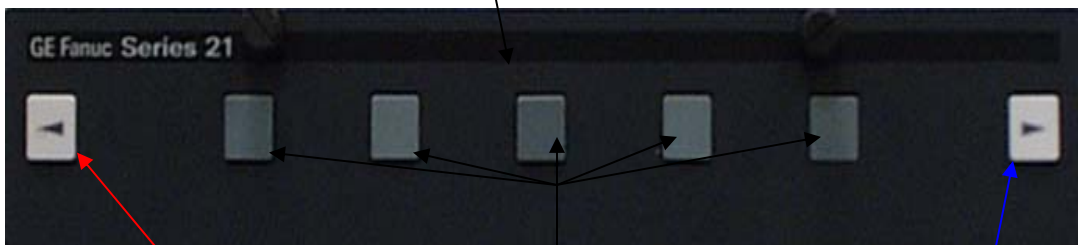


MESSAGE = displays operator & alarm messages



AUX / GRAPH = displays 2-d graph simulation

SOFT KEYS



SCROLL BACK

SOFT KEYS

PAGES OVER

MACHINE KEYS

MACHINE FUNCTION KEYS



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



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



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



= (Optional stop) for programs with (m1)



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



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



= (Cycle stop) program hold, feed hold



= (Cycle start) program start

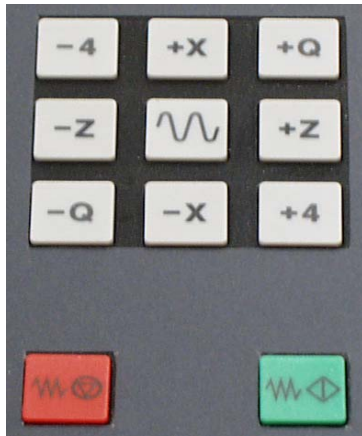


= (Agreement button) used for open/closing door or to jog axis with the door open



= (Mode Key) Automatic & Hand Mode

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

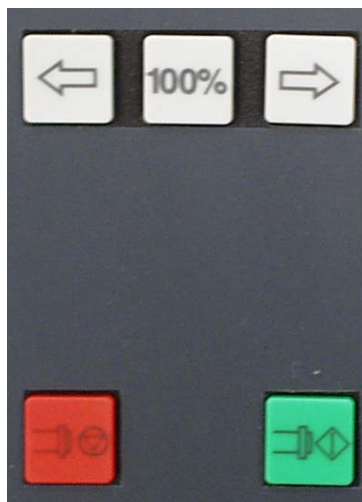


DIRECTION KEYS

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)

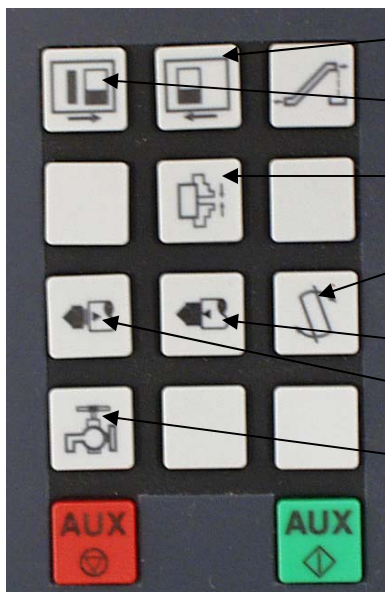
Arrow key pointing left decrease the Spindle speed (50% low)

100% key jumps speed to 100%

Spindle stop (Red) / Spindle start (Green)

Works all modes except EDIT & ZRN (Reference)

ACCESSORY FUCTIONS



Arrow right door open

Arrow left door closed

Press once chuck open

Press again chuck closed

**Press turret index's one time clockwise
Each time pressed**

Press tailstock moves backward

Press tailstock moves forward

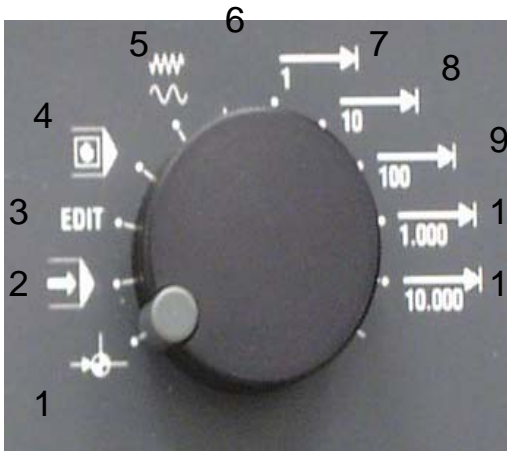
Press once coolant on

Press again coolant off

Press auxiliary drives on (Green)

Press auxiliary drives off (Red)

MODE DIAL

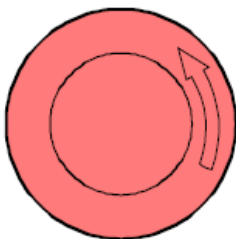


- (1) REF = Zero / Reference or Home mode
- (2) MEM = Automatic mode for running a program
- (3) EDIT = Edit mode for program changes or entering a new program
- (4) MDI = Manual Data Input mode for manually running the machine
- (5) JOG = Manual moving the axis in x or z
- (6) STEPS = Incremental feed movements
- (7) STEPS = .0001 or tenths
- (8) STEPS = .001 or thousands
- (9) STEPS = .010 or ten thousands
- (10) STEPS = .100 or hundred thousands
- (11) STEPS = .100 or hundred thousands

FEED OVERRIDE DIAL





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

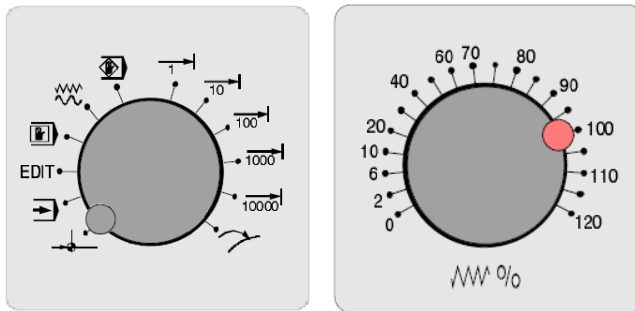


E Stop or Emergency Stop

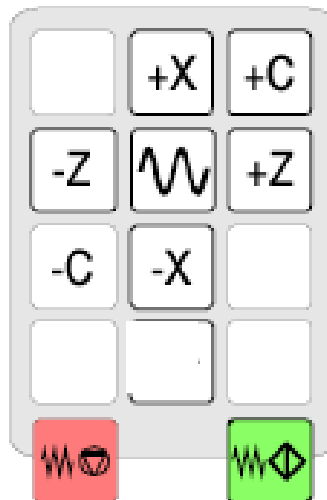
Turning the Machine On/Entering Fanuc Software

Referencing the Machine

1. Press the **AUX** button  (This turns on the Auxiliary Drives)
2. Press the **Agreement** button  **Open** door then **Shut** door
(This Initialize the safety circuits on the Machine door)
3. Move the MODE dial to REF position also know as Reference make sure your feed rate is not on "0"

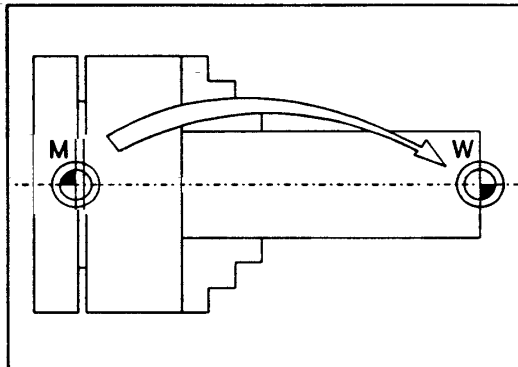


4. Make sure the Door is closed
5. Press the X+ (arrow pointing up) this references the X axis.
(Wait until X is fully reference)
6. Press the Z- (arrow pointing left) this references the Z axis



Note: Every time you enter Fanuc 21 Software or Turn the Machine On you must reference the axis

WORK SHIFT



Zero offset from machine zero point M to workpiece zero point W

With EMCO lathes the machine zero "M" lies on the rotating axis and on the end face of the spindle flange. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

The offset register offers one adjustable zero offset.

When you define a value in the offset register, this value will be considered with program start and the coordinate zero point will be shifted from the machine zero M to the workpiece zero W.

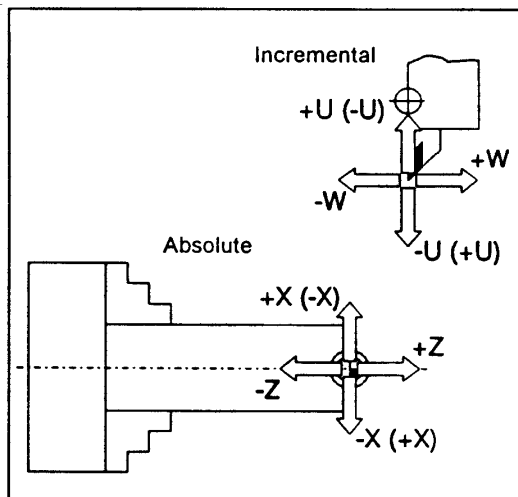
The workpiece zero point can be shifted within a program with "G92 - Coordinate system setting" in any number.

More informations see in the command description.

The Coordinate System

The X coordinate lies in the directions of the cross slide, the Z coordinate in the direction of the longitudinal slide.

Coordinate values in minus directions describe movements of the tool system towards the workpiece. Values in plus direction away from the workpiece,



Absolute coordinates refer to a fixed position, incremental coordinates to the tool position. The bracket values for X, -X, U, -U are valid for the PC TURN 50 because the tool is in front of the turning centre on this machine.

Coordinate System for Absolute Value Programming

The origin of the coordinate system lies at the machine zero "M" or at the workpiece zero "W" following a programmed zero offset.

All target points are described from the origin of the coordinate system by the indication of the respective X and Z distances.

X distances are indicated as the diameter (as dimensioned on the drawing).

Coordinate System for Incremental Value Programming


The origin of the coordinate system lies at the tool mount reference point "N" or at the cutting tip after a tool call-up.

The U coordinate lies in the direction of the cross slide, the W coordinate in the direction of the longitudinal slide. The plus and minus directions are the same as for absolute value programming.

With incremental value programming the actual paths of the tool (from point to point) are described. X distances are indicated as the diameter.

Work Shift:

1. Index to a empty ID location

- Manually index by going to Jog Mode and Pressing Index button 

OR

- Programming Index

Rotate Mode Dial to MDI

Press the PROGRAM display button

Until top left of the screen shows **PROGRAM (MDI)**

Type T0100 (if the ID location wanted is position 1)

Press Input button



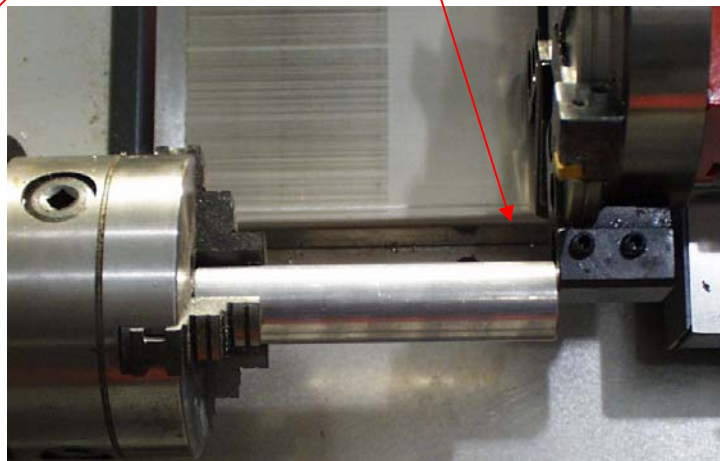
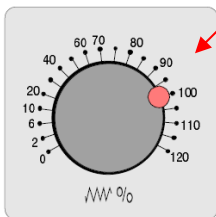
Then press CYCLE START  (Door must be closed)

2. If the Dial is not in Jog rotate Mode Dial to Jog

3. Jog the TURRET to the face of the Work Piece & touch using the Direction keys.

(Use piece of paper between TURRET and Work Piece)

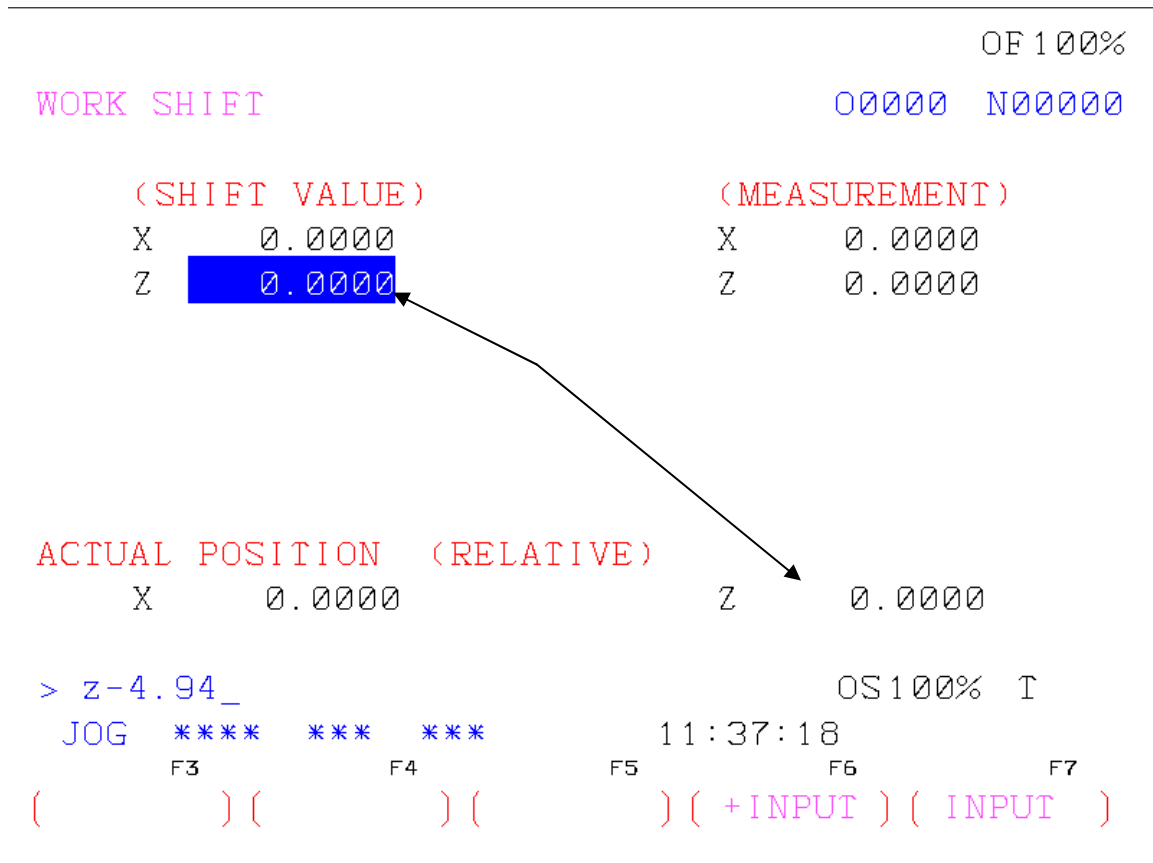
(Use the Feed override dial or Steps to approach at a slower feed)



4. Press the OFFSET/SETT button until Work Shift page appears
5. Make sure (Shift Value) Z is 0 if not highlight Z under (SHIFT VALUE) type 0 and Input
6. The value that is in the ACTUAL POSITION (RELATIVE) Z type this value in (SHIFT VALUE) Z as a negative number
7. Then press INPUT button
8. Jog TURRET away from WORK PIECE using Z+

OFFSET
SETT.

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

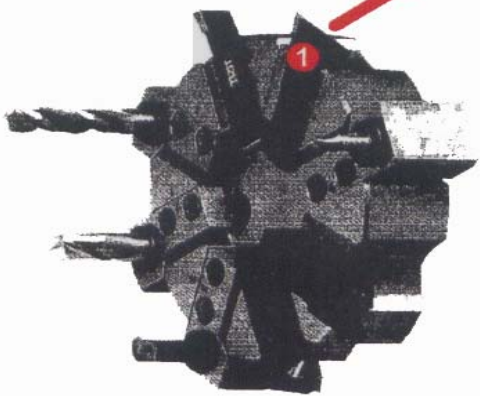


Note: Machine 0 is the turret face touching the spindle nose.

NEVER put a value in SHIFT VALUE X

TOOL OFFSETS

T 01 01



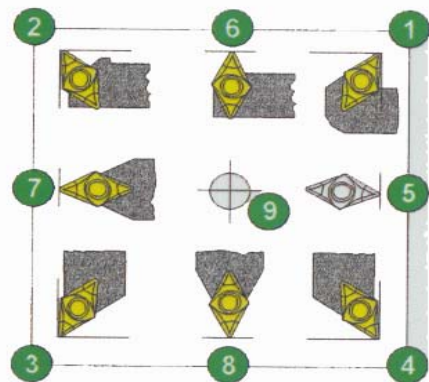
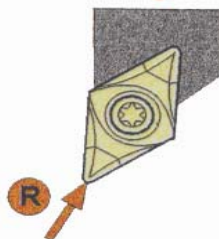
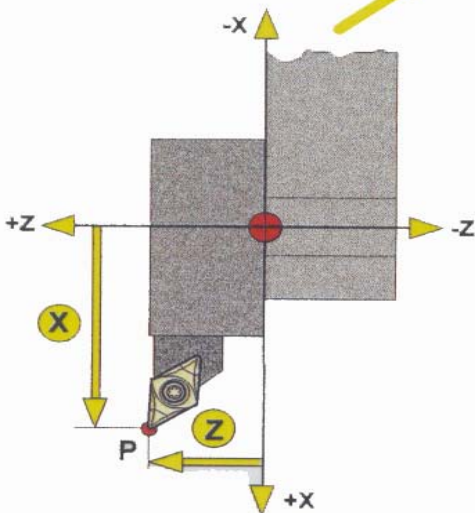
GE Fanuc Series 0 - T

F100% S100%
O0001 N0000

OFFSET/GEOMETRY				
No.	X	Z	R	T
G 01	0.000	0.000	0.000	0
G 02	0.000	0.000	0.000	0
G 08	0.000	0.000	0.000	0

ACT. POSITION (RELATIVE)
U 0.000 W 0.000
ADRS. S 0.000
JOG

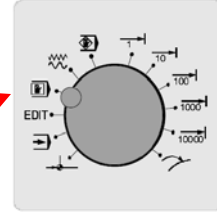
WEAR GEOM W.SHIFT



Tool Offsets

1. Index the TURRET to the tool being measured

- Move the MODE Dial to MDI position
- Press Program button until **PROGRAM (MDI)** is at the top left of the screen



- Type tool number then press INPUT button



Example: T0200

1. For scratching type S1000 M03 press INPUT button

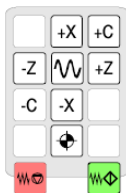
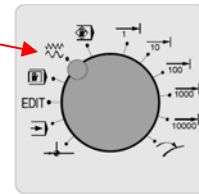


S = Spindle speed M03 = spindle on clockwise

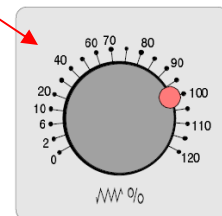
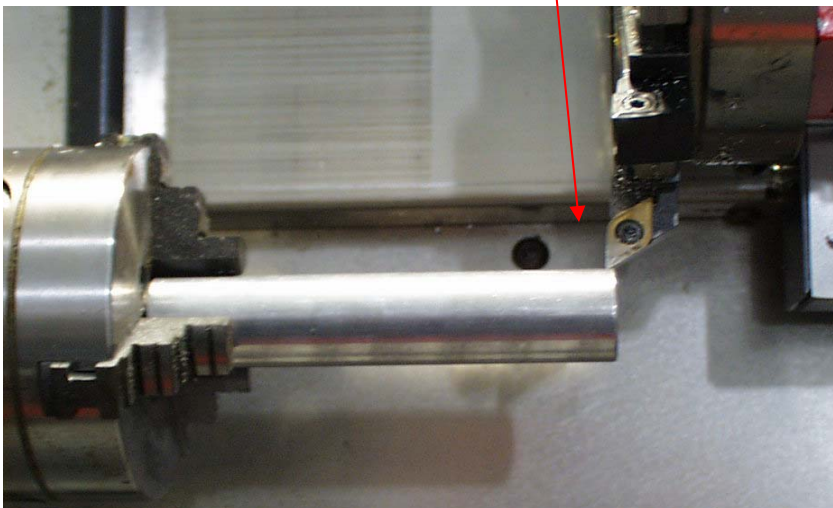
- Press CYCLE START  (make sure door is closed)

2. Move the MODE Dial to JOG position

3. Jog TOOL TIP to the WORK PIECE & touch TOOL TIP to the DIAMETER of the WORK PIECE using the Direction keys.



(Use the Feed override dial or Steps to approach at a slower feed)



4. Press the OFFSET/SETT button until Geometry page appears

OFFSET
SETT.

5. Type X and the Diameter being scratched

Example: X1 (If the diameter being scratch is 1"dia.)

6. Cursor down and over to highlight the G0 X location for the tool that is being measured

(Picture below shows tool 2 is being measured)

7. Then press soft key for

F4
(MEASUR)

8. Jog TURRET away from WORK PIECE using X+

This value is the distance from an I.D. Tool Station to the Tool Tip

OF 100%

OFFSET / GEOMETRY

00000 N00000

NO.	X	Z	R	T
G01	0.0000	0.0000	0.0000	0
G02	0.0000	0.0000	0.0000	0
G03	0.0000	0.0000	0.0000	0
G04	0.0000	0.0000	0.0000	0
G05	0.0000	0.0000	0.0000	0
G06	0.0000	0.0000	0.0000	0
G07	0.0000	0.0000	0.0000	0
G08	0.0000	0.0000	0.0000	0

ACTUAL POSITION (RELATIVE)

X	0.0000	Z	0.0000
---	--------	---	--------

> 1.097_

OS100% T

JOG **** **

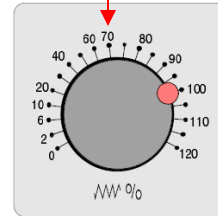
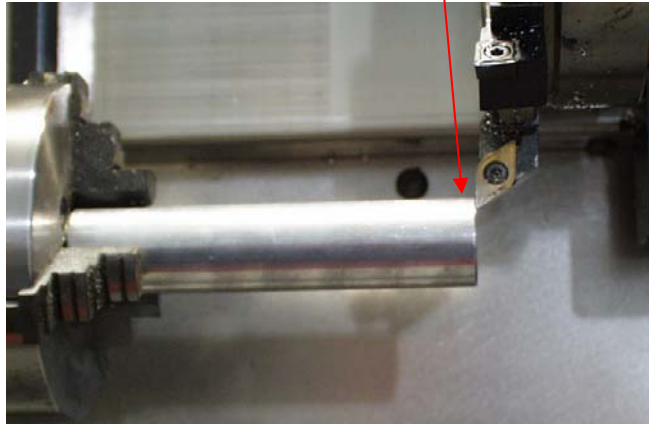
11:38:33

F3 F4 F5 F6 F7

{ NO.SRH } { MEASUR } { INP.C. } { +INPUT } { INPUT } >

9. Jog TOOL TIP to the end of the WORK PIECE & touch TOOL TIP to the FACE of the WORK PIECE using the Direction keys.

(Use the Feed override dial or Steps to approach at a slower feed)



10. Press the OFFSET/SETT button until Geometry page appears

OFFSET
SETT.

11. Type Z and 0 for reading from work shift 0

Example: Z0

12. Cursor down and over to highlight the G0 Z location for the tool that is being measured

(Picture below shows tool 2 is being measured)

13. Then press soft key for

F4
(MEASUR)

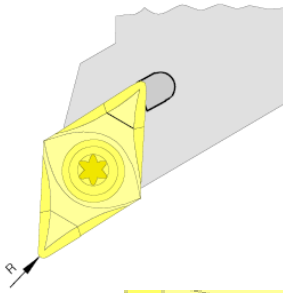
```

OF 100%
OFFSET / GEOMETRY                                00000 N00000
NO.      X      Z      R      T
G01      0.0000  0.0000  0.0000  0
G02      0.0000  0.0000  0.0000  0
G03      0.0000  0.0000  0.0000  0
G04      0.0000  0.0000  0.0000  0
G05      0.0000  0.0000  0.0000  0
G06      0.0000  0.0000  0.0000  0
G07      0.0000  0.0000  0.0000  0
G08      0.0000  0.0000  0.0000  0
ACTUAL POSITION (RELATIVE)
X      0.0000      Z      0.0000
>
OS 100% T
JOG  ***  ***  ***  11:39:22
F3      F4      F5      F6      F7
( NO.SRH ) ( MEASUR ) ( INP.C. ) ( +INPUT ) ( INPUT ) >

```

Note: This is the distance from spindle nose to the program 0 / front of the work piece

14. R is the Tool Tip Radius



Note: Most insert packages or tool holders specify this value. If cutter comp is not used then the R value is not used

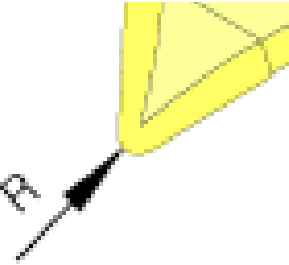
Type in the value for the tip radius

Emco tooling radius

55° insert = .015 Parting Off or Groove = .003

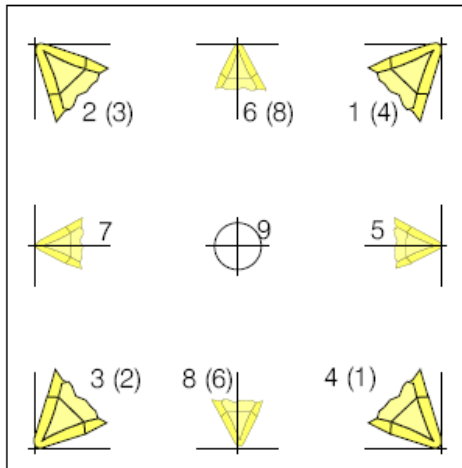
80° insert = .032 35° insert = .010

Threading insert = .001



15.

T for cutter comp cutting direction



Note: **The T is Direction that the Tool Points. Tool doesn't need to look like Tool in the picture**

All machines that have a turret on the bottom will also use the bracket #'s. Machines with turret on top will use regular #'s

16. Jog TURRET away from WORK PIECE using Z+

17. Repeat steps for all OD tools (STEPS 1-16 starting on pg 14)

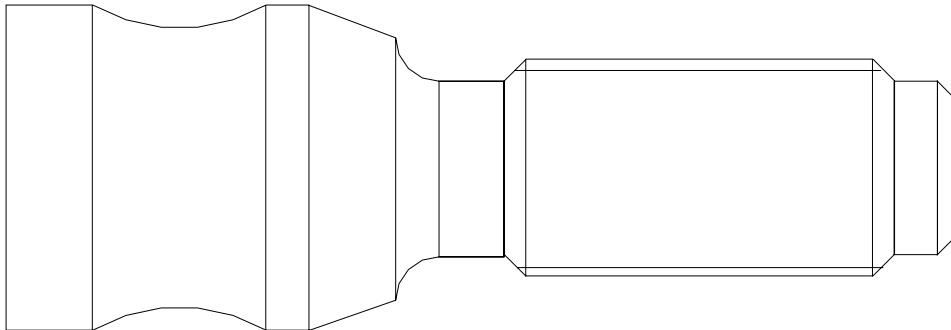
Note: The T is Direction that the Tool Points. Tool does not need to look like Tool in the Picture

Program Training

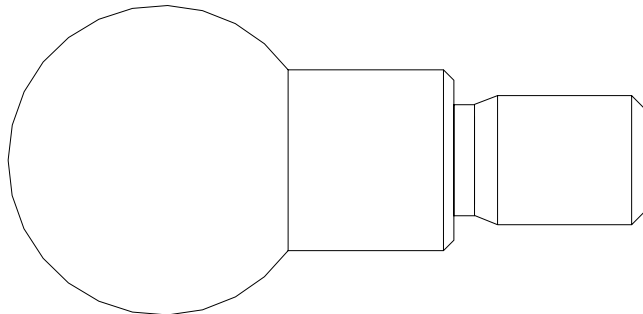
Program O0001



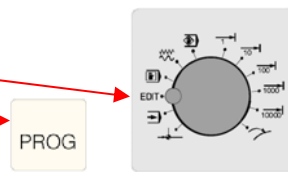
Program O0002



Program O0003



Change the Mode Dial to Edit
Press the Program Button



- **INSERT A NEW PROGRAM**

1. Press letter o then a program number between 1- 8999
2. Press insert button



Example: Q0001 OR Q1

- **CALL A EXISTING PROGRAM UP**

1. Press letter o then program number in the directory
2. Press cursor down button



- **INSERT A WORD**

1. Press letter then number
2. Press insert button



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

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

- **INSERT END OF BLOCK**

1. Press the (EOB) button



2. Press insert button



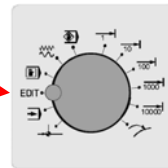
HINT: at the end of each number line needs an End Of Block looks like a Semicolon (;)

Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT

USE INPUT FOR ALL OTHER SCREENS AND MODES.

Change the Mode Dial to Edit
Press the Program Button



- **DELETE A PROGRAM**

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



Example: O0001 OR O1

- **DELETE ALL PROGRAMS**

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



Example: O – 9999

- **DELETE A WORD**

1. Highlight the Word
2. Press delete button



- **DELETE A BLOCK OR LINE NUMBER**

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



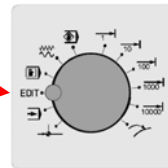
- **CANCEL MISTYPED WORD (Backspace)**

1. Press cancel button



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

Change the Mode Dial to Edit
Press the Program Button



- **ALTER A WORD**

1. Highlight the word needed altered type the change
2. Press alter button



- **SEARCH FOR NUMBER BLOCK**

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



- **SEARCH FOR WORD**

1. Type in word & number
2. Press cursor down button



- **SEARCH FOR LETTER**

1. Press letter
2. Press cursor down button



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

Groups of G codes

There are 3 groups of G-Codes; Emco Group uses the C group of G-Codes.
In relation to the other two Groups the only differences is the # for the G-Code

Gr.	Command			Function
	A	B	C	
0	+ G04	G04	G04	Dwell
	+ G07.1	G07.1	G07.1	Cylindrical Interpolation
	+ G10	G10	G10	Data setting
	+ G11	G11	G11	Data setting Off
	+ G28	G28	G28	Return to reference point
	+ G70	G70	G72	Finishing cycle
	+ G71	G71	G73	Stock removal in turning
	+ G72	G72	G74	Stock removal in facing
	+ G73	G73	G75	Pattern repeating
	+ G74	G74	G76	Deep hole drilling, cut-in cycle in Z
	+ G75	G75	G77	Cut in cycle in X
	+ G76	G76	G78	Multiple threading cycle
	+ G90	G92	G92	Coord.syst.set., Spindle speed limit
	• G00	G00	G00	Positioning (rapid traverse)
1	G01	G01	G01	Linear interpolation clockwise
	G02	G02	G02	Circular interpolation clockwise
	G03	G03	G03	Circular interp. counterclockwise
	G90	G77	G20	Longitudinal turning cycle
	G92	G78	G21	Thread cutting cycle
	G94	G79	G24	Face turning cycle
	G32	G33	G33	Thread cutting
2	G96	G96	G96	Constant cutting speed
	• G97	G97	G97	Direct spindle speed programming
3	• -	G90	G90	Absolute programming
	-	G91	G91	Incremental programming
5	G98	G94	G94	Feed per minute
	• G99	G95	G95	Feed per revolution
6	G20	G20	G70	Inch data input
	G21	G21	G71	Metric data input
7	• G40	G40	G40	Cancel cutter radius compensation
	G41	G41	G41	Cutter radius compensation left
	G42	G42	G42	Cutter compensation right
10	• G80	G80	G80	Cancel cycles
	G83	G83	G83	Drilling cycle
	G84	G84	G84	Tapping cycle
	G85	G85	G85	Reaming cycle
11	• -	G98	G98	Return to initial plane
	-	G99	G99	Return to withdrawal plane
16	G17	G17	G17	Plane selection XY
	G18	G18	G18	Plane selection ZX
	G19	G19	G19	Plane selection YZ
21	G12.1	G12.1	G12.1	Polar Coordinate Interpolation ON
	G13.1	G13.1	G13.1	Polar Coordinate Interpolation OFF

Example

G70 in the C group is programming in inches

G20 in the A & B group is programming in inches

Both are exactly the same but the G #

Survey of commands G-CODES (Group C): Mostly used

Model	G00	Rapid traverse
Model	G01	Linear interpolation in working feed
Model	G02	Circular interpolation, clockwise
Model	G03	Circular interpolation, counter-clockwise
Non-Model	G04	Dwell, active block by block
Non-Model	G28	Approach reference point
Model	G40	Deselect cutter radius compensation
Model	G41	Cutter radius compensation left
Model	G42	Cutter radius compensation right
Model	G70	Dimensions in inch
Model	G71	Dimension in millimeter
Non-Model	G72	Finishing cycle
Non-Model	G73	Longitudinal turning cycle
Non-Model	G78	Multiple Thread cutting cycle
Model	G80	Deselect drilling cycles
Model	G83	Drilling cycle
Model	G90	Absolute value programming
Model	G91	Incremental value programming
Model	G92	Set coordinates zero point / speed limitation
Model	G94	Feed in inch/min
Model	G95	Feed in inch/rev
Model	G96	Constant cutting speed (Surface Footage)
Model	G97	Constant speed
Model	G98	Return to start plane

Bold print = is the Default codes that are on at all times until changed

Note: Most CONTROLS only take up to 4 G codes per line

Survey of commands M- CODES : Mostly used

M00	Programmed stop unconditional
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
M05	Spindle OFF
M20	Tailstock sleeve backward
M21	Tailstock sleeve forward
M25	Release clamping device
M26	Close clamping device
M30	Main program end with new start of program
M71	Blow-off ON (cleaning clamping device)
M72	Blow-off OFF
M98	Subroutine called up
M99	Subroutine end

Only one M-command for one Block

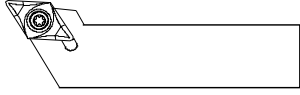

Used Addresses

A	Angle
C	Chamfer
F	Feed rate, thread pitch
G	Path, movement function
I, K	Circle parameter
U, W	Incremental, cycle parameter
M	Miscellaneous, machine function
N	Block number 1 to 9999, macro call out
O	Program number 1 to 9499
P	Dwell, subroutine, cycle parameter
Q	Cutting depth, cycle parameter
R	Radius, retraction, cycle parameter
S	Spindle speed
T	Tool called out
X, Z	Position data in absolute

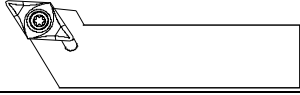

Need one of each tool and insert (Inch or Metric)

Position	Part #	Discription	Picture
-----------------	---------------	--------------------	----------------

Metric Tool



T0202	271 050	Finishing tool left	
	271 056	Indexable inserts for aluminum	

Inch Tool

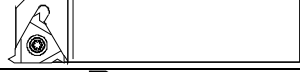

T0202	270185A	Finishing tool left	
	270186A	Indexable inserts for aluminum	

And use Threading tool

Metric Tool

T0404	271 110	OD-threading tool right	
	271 115	Indexable inserts for OD-threading, right	

Inch Tool

T0404	270188A	OD-threading tool right	
	270189A	Indexable inserts for OD-threading, right	

Program screen & Edit mode

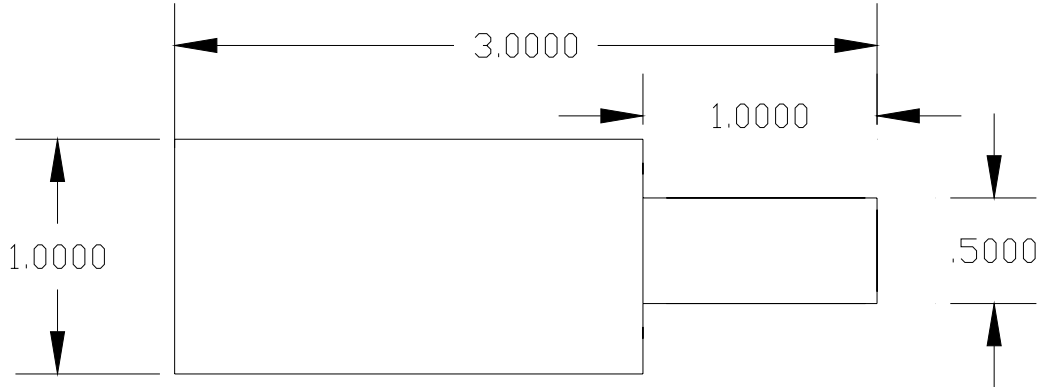
To edit / change a program / insert new programs & input or output excising programs & offsets

Program screen & MDI mode

To manually program the spindle speed / move the axis (X,Z) to a specified location and or Index to a certain tool

Note: Material is 2011-T3 Alum, All feeds & speeds are programmed for this type of Aluminum

Program Q0001



1st **G73** U = Depth of Cut R = Retract Value

2nd **G73** P = First Block number of the Contour (Block number after the 2nd G73)

Q = Last Block number of the Contour F = Feed rate for cycle

(Facing in a cycle)

O0001 (Demo 1).....	Label Line Only
N5 (3 x 1 alum).....	Part Label Optional
N10 G40 G70 G80 G90	Default G Codes
N15 G95 G96 G98	sfpm
N20 G0 G28 U0 W0.....	safe move
N25 T0202 S700 M4 (Finish Tool 55°).....	Tool, Speed, Direction
N30 G0 X1.0 Z.1.....	start point of cycle
N35 G73 U.06 R.03.....	cycle parameters
N40 G73 P45 Q65 F.004.....	cycle begin / end lines
N 45 G0 X0.....	first line of cycle
N50 G1 Z0.0.....	Z to Face of part
N55 X.5.....	1 st diameter of contour
N60 Z-1.0.....	length of contour
N 65 X1.0.....	diameter of contour
N70 G0 G28 U0 W0.....	safe move
N75 M30.....	end of program

2D Simulation

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

OF 100%

GRAPHIC PARAMETER 00001 N00000

WORK LENGTH	W =	0.0000
WORK DIAMETER	D =	0.0000
PROGRAM STOP	N =	0
AUTO ERASE	A =	1
LIMIT	L =	0
GRAPHIC CENTER	X =	0.0000
	Z =	0.0000
SCALE	S =	0.000
GRAPHIC MODE	M =	0

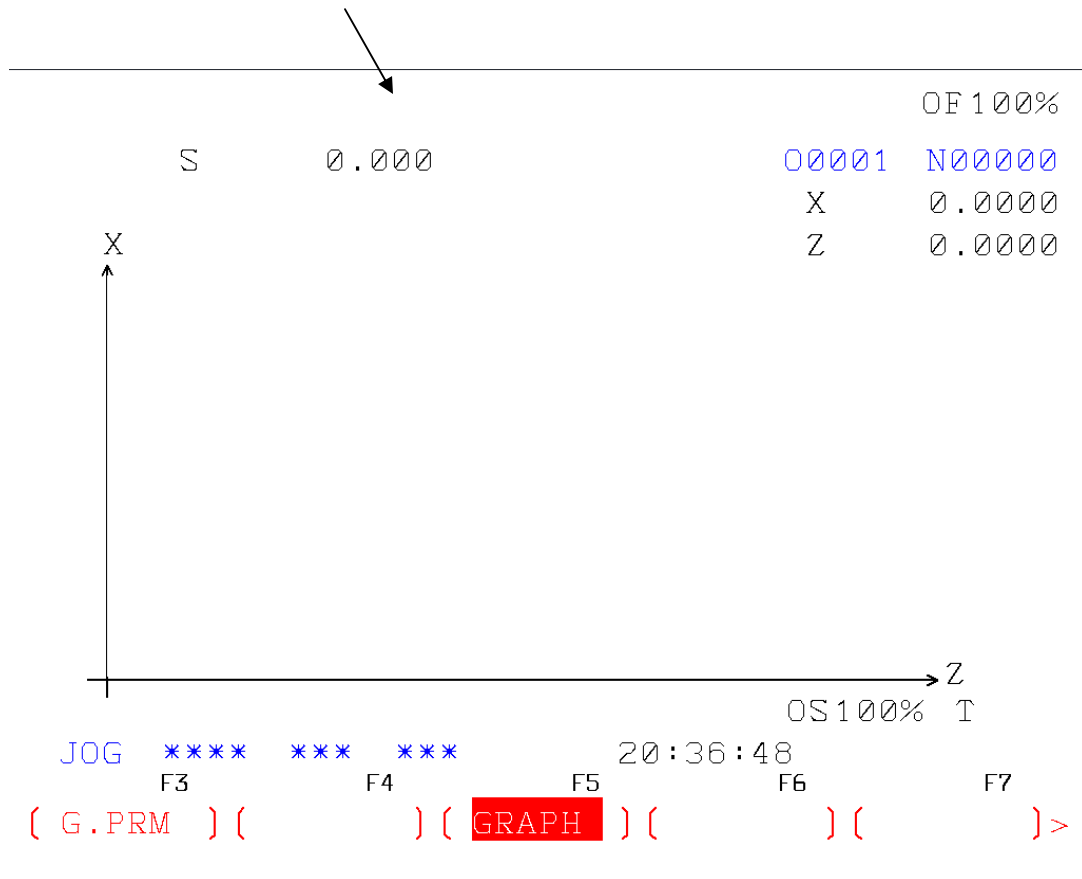
> _ OS100% T

JOG **** * F3 20:21:13 F4 F5 F6 F7

[G.PRM] () (GRAPH) () () >

Note: There are only 4 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. Work Length W = Overall length of stock in the Z direction this is a + value
3. Work Diameter D = Overall diameter of stock in the X direction + value
4. Graphic Center X = any area you wish to see past X0. Usually only if a Drill or a Tap is being used place a – value to see the tool movements for X pasted 0
Example -.100 is a common value entered
5. Graphic Center Z = this value is always a negative number and this is the area you wish to view. The longest Z- number in the program is normally used here
6. Press the Soft key **Graph** for Simulation screen



7. Now press **Cycle start** and you will see the tool movements of the program

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

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

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

Note: If you want to use USB use C and then follow instruction in 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 O and program number
(O0002) or (O2)
3. Press the right Arrow key on the Soft keys
4. Press Punch then press Exec

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

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


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

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

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

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

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

PROG

3. Call up Program to be run / cut
(Example O1 for program 1)

4. Rotate the Mode dial to MEM

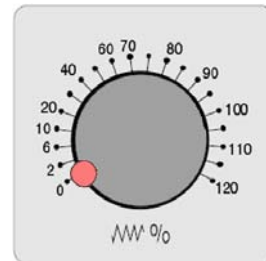
5. If screen is not in **PROGRAM CHECK** then press program button until this is at the top left of the screen

PROG

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



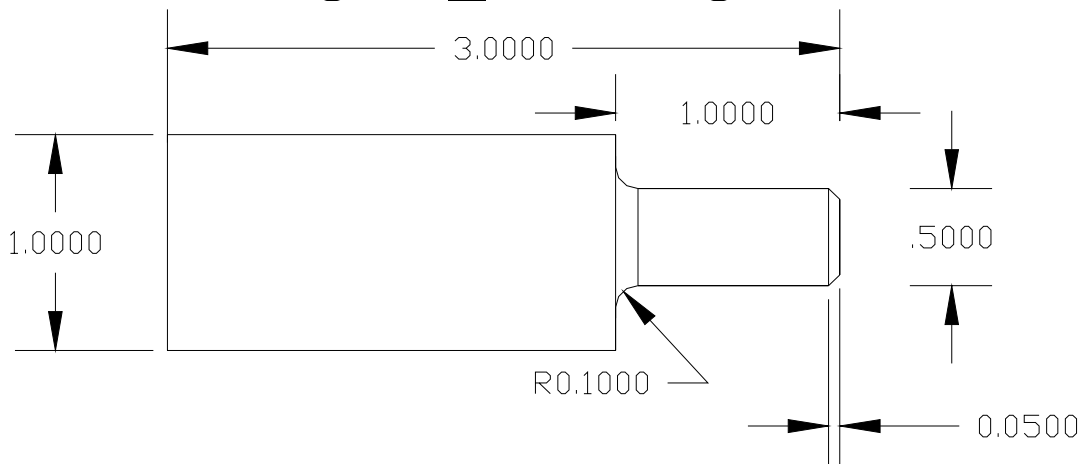
7. Press Cycle Start and continue

(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 Q0001 using C/R's



1st G73 U = Depth of Cut R = Retract Value

2nd G73 P = First Block number of the Contour (Block number after the 2nd G73)

Q = Last Block number of the Contour F = Feed rate for cycle

(Facing in a cycle)

O0001(Demo 1)

N5 (3 x 1 alum)

N10 G40 G70 G80 G90

N15 G95 **G96** G98.....sfpm

N20 G0 Z2.0.....safe move

N25 T0202 S700 M4 (Finish Tool 55°)

N30 G0 X1.0 Z.1.....start point of cycle

N35 G73 U.06 R.03.....cycle parameters

N40 G73 **P45 Q65** F.004.....cycle begin and end lines

N45 G0 X0.....first line of cycle

N50 G1 Z0.0.....movement to face of part

N55 X.5 **C.05**.....1st diameter of contour

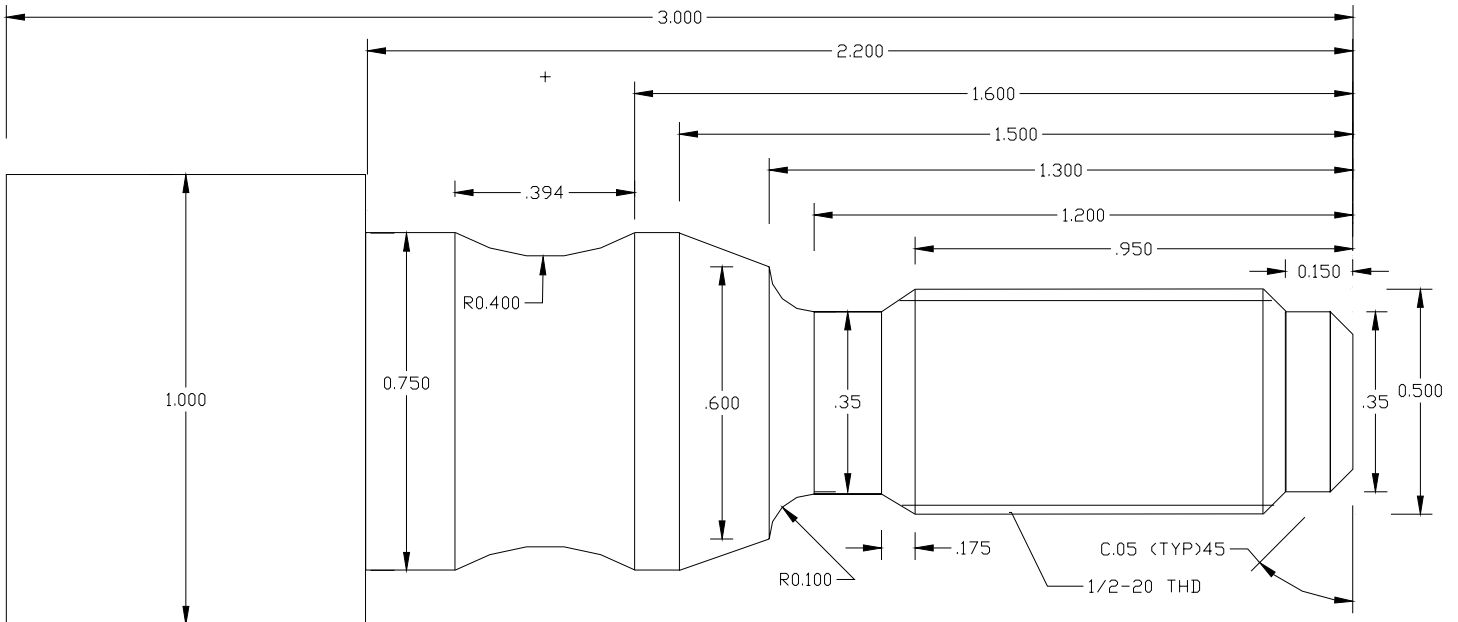
N60 Z-1.0 **R.1**.....length of contour

N65 X1.0.....diameter of contour

N70 G0 Z2.0.....safe move

N75 M30.....end of program

Program Q0002



1st G73 **U** = Depth of Cut **R** = Retract Value

2nd G73 **P** = First Block number of the Contour (Block number after the 2nd G73)
Q = Last Block number of the Contour **U** = Allowance for Finish cut in X
W = Allowance for Finish cut in Z **F** = Feed rate for the cycle

HINT:

The X **BEFORE** G73 example (X 1.25) should be (=) to or (>) than X at the **END** of the Cycle. X at the end of the cycle determines stock size

G72 **P** = First Block number of the Contour (Block number after G73)
Q = Last Block number of the Contour

HINT:

BEFORE the G72 call a spindle **SPEED** higher and **FEED** rate lower

If possible change tool to a 55 degrees for FINISHING & 80 degree for ROUGHING

G78 CYCLE MULTIPLE

Example for 1/2 20 thread

1ST G78

P = Is 6 Digits divided in 2 Digit groups

P = 1st two digits is number of FINISH PASSES 01

2ND two digits is PULL OUT ANGLE 00

3rd two digits is angle of the THREADS 60 degrees

Q = Minimum cutting DEPTH 0020 (Micro IN)

R = Finishing OFFSET .001

2nd G78

X = Minor DIA. X .434

Z = Length of THREAD from (0) call out Z -1.05

P = Depth of THREAD Radial 0330 (Micro IN)

Q = First cutting DEPTH 0120 (Micro IN)

F = Thread PITCH .050

Example G78 P010060 Q0020 R.001

G78 X.434 Z-1.125 P0330 Q0120 F.05

Micro IN is the value without the decimal point

Example: .1000 is shown as 1000 (show all 4 place values)

HINT:

$$\text{TPI} = \frac{1}{20} = (F) .05$$

$$\text{IPM} = \text{RPM} \times \text{PITCH}$$

$$\text{RPM} = \frac{\text{IPM}}{\text{PITCH}} = \frac{50}{.05} = 1000 \text{ RPM} \qquad 196 \text{ is max for a Concept 105 Machine}$$

Make sure the X value before the G78 is larger than the MAJOR Diameter and the Z is at least 2 times the PITCH before cutting threads

Example: N100 G0 X.55 Z.1 ; THIS IS THE START POINT FOR G78
N105 G78 ;

Program Q0002

O0002 (Demo 2)
N5 (Stock 3.5 x 1 alum)
N10 G0 Z2
N15 G96 T0202 S550 M3 (Finish Tool 55°)
N20 G0 X1.1 Z.1.....Safe start for Facing
N25 Z0.....Face of part
N30 G1 X-.02 F.002.....Facing past Zero
N35 G0 X1.0 Z.1.....Start point of cycle
N40 G73 U.04 R.02.....Cycle parameters
N45 G73 P50 Q115 U.01 W.005 F.004.....Cycle finish offsets
N50 G0 G42 X.2.....Turning CRC on
N55 G1 Z0.....Face of part
N60 X.35 C.05
N65 Z-.15
N70 X.5 C.05
N75 Z-.950
N80 X.35 Z-1.125
N85 Z-1.3 R.1
N90 X.6
N95 X.75 Z-1.5
N100 Z-1.6
N105 G2 X.75 Z-1.994 R.4
N110 G1 Z-2.2
N115 G1 X1.0
N120 G0 G40 X1.1.....Cancel CRC
N125 S700 F.002
N130 G72 P50 Q120
N135 G0 Z2Safe Index Pos
N140 G97 S560 M3Threading Speed in RPM
N145 T0404 (Threading Tool Right Hand)
N150 X.55 Z.1.....Start Pos. Thread Cycle
N155 G78 P010060 Q0020 R.001.....Threading cycle
N160 G78 X.434 Z-1.125 P0330 Q0120 F.05
N165 G0 Z2Safe Return
N170 M30.....End of Program

1. To make a program tie 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.

TEST FOR SUB PROGRAMS

O0005 (Tie Programs)

N5 (Stock 3.25 x 1 alum)

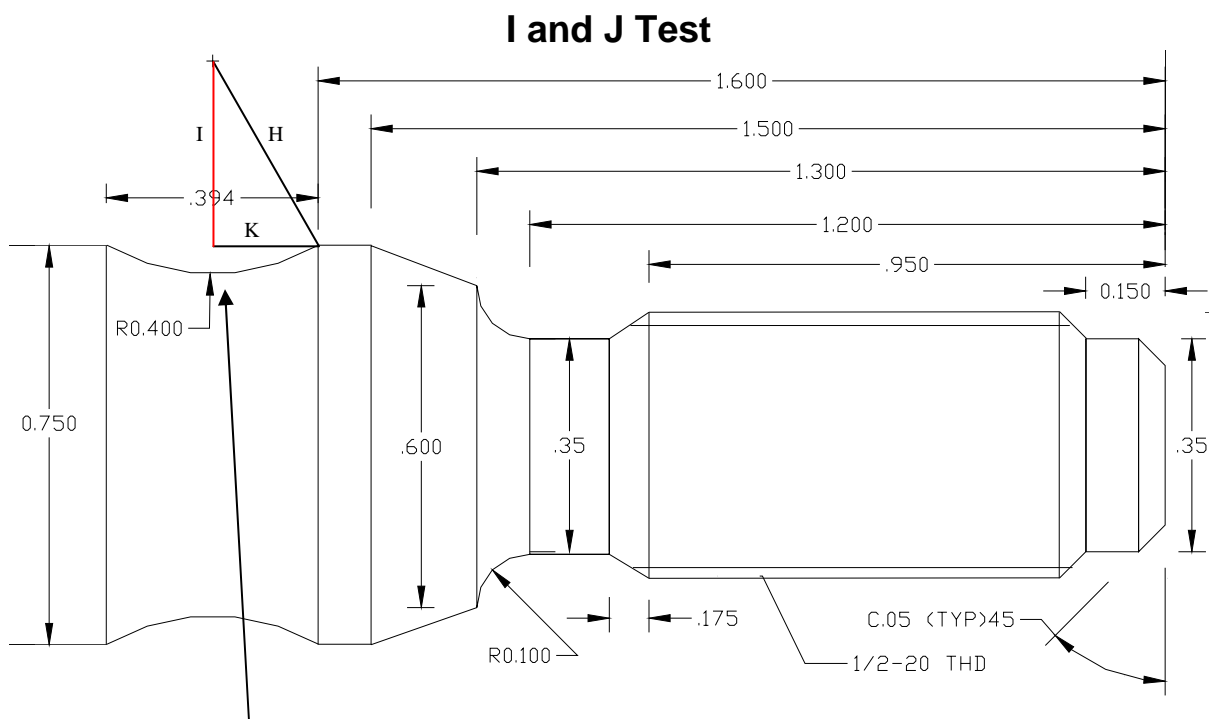
N10

N15

N20

Changing Item

Note: Change the end of O0001 and O0002 to M99 for running them as SUB PROGRAMS



Find the I and J for the arc in the picture

$$A^2 (\text{K leg}) + B^2 (\text{I leg}) = C^2 (\text{H radius})$$



S $\frac{O}{H}$

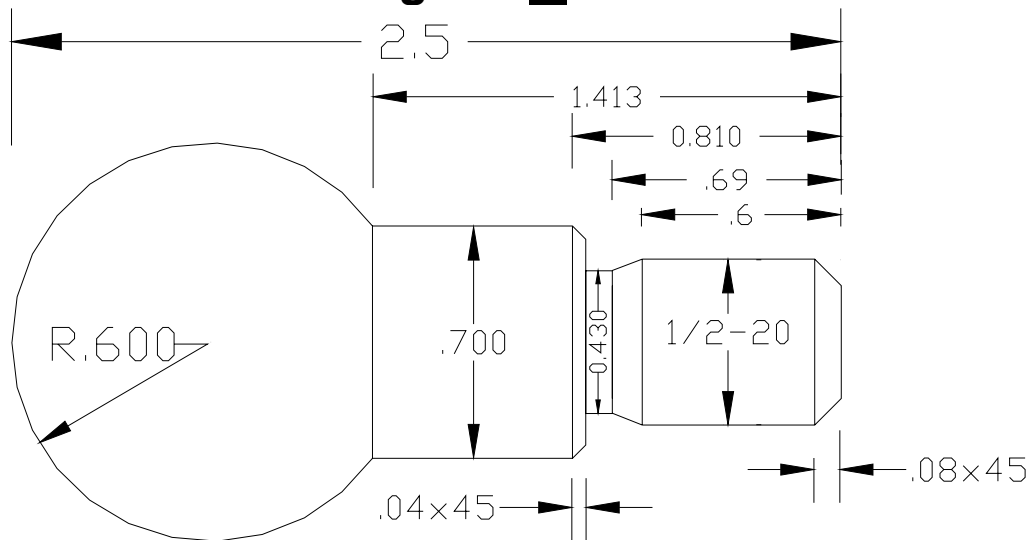
C $\frac{A}{H}$

T $\frac{O}{A}$

Sally Can Tell Oscar Has A Hat On Always

SINE COSINE TANGENT

Program Q0003



O0003 (Ball Hitch)

N5 (Stock 2.5625 x 1.25)

N10 G0 Z2

N15 G96 T0202 S550 M3 (Right Hand Finish Tool 55°)

N20 G0 Z.1

N25 Z0

N30 G1 X-.02 F.003

N35 G0 X1.25 Z.1

N40 G73 U.03 R.015

N45 G73 P50 Q95 U.01 W.005 F.004

N50 G0 G42 X.24

N55 G1 Z0

N60 X.5 C.08

N65 Z-.6

N70 X.43 Z-.69

N75 Z-.770

N80 X.7 C.04

N85 Z-1.413

N90 G3 X1.2 Z-1.92 R.6

N95 G1 X1.25

N100 G0 G40 X1.3

N105 S700 F.002

N110 G72 P50 Q100
N115 G0 Z2.0
N120 G97 S560 M3
N125 T0404 (Threading tool Right hand)
N130 X.55 Z.1
N135 G78 P010060 Q0020 R.001
N140 G78 X.434 Z-.69 P0330 Q0100 F.05
N145 G0 Z2.0
N150 M30 (Flip Part around) **Note: change M30 to M00 after touch off**
Then start back at line N150 to run the back side
N155 M98 P010004 (SUB PROGRAM FOR BACK SIDE)
N160 M30

Program Q0004

O0004 (Back Side Ball Hitch)

N5 G96

N10 G10 P0 Z- ←

Need to touch with turret to the face of stock to get the number for the (Z-) after you cut the first side. Now press Position and the number that is in Machine for (Z) place this number on line N10 for Z as (-).

N15 T0202 S550 M3 (Right Hand Finish Tool 55°)

N20 X1.25 Z.200

N25 G73 U.03 R.015

N30 G73 P35 Q55 U.01 W.005 F.003

N35 G0 G42 X0

N40 G1 Z0

N45 G3 X1.2 Z-.6 R.6

N50 G1 Z-.69

N55 X1.25

N60 G0 G40 X1.3

N65 S700 F.002

N70 G72 P35 Q60

N75 Z2

N80 G10 P0 Z- **(the original work shift)**

N85 M99




Might need to subtract from the Z- on line N10 at least .0625

This is the difference between the Stock size on the print and the Stock size recommended. This way the ball will blend together in the middle of the part. The other thing that can be done is to face .03125 on each side of the part as it is being machine

Appendix

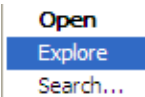
Changing Drive to USB Port

1. Close out the SW (software)

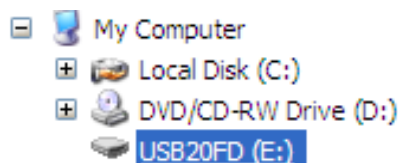
- Press  to allow you to exit
- Press  and  together to exit the Software

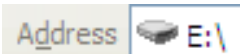

2. Make sure USB is plug into port

3. Open Explorer

- Right Click on Either My Computer, My Documents or any Folder on the Desktop
- Move mouse to  (Explorer)
- Left Click
- If you right clicked on My computer skip to step 4 if not then Left Click on My Computer

4. Copy Drive directory

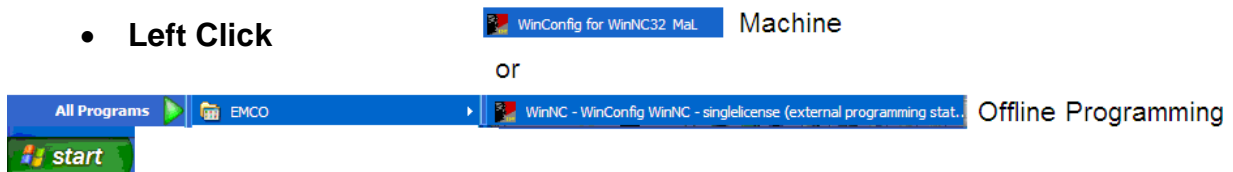


- Click on you USB drive
- At the top of the active screen or page in the Address copy or remember drive info 
- Close the active screen or page using either Alt and F4 or  at top of the active screen





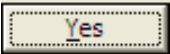
5. Setting up WinConfig

- 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



6. In Winconfig

- Left Click on  (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)
- Left Click on  (Close)
- Left Click on  (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