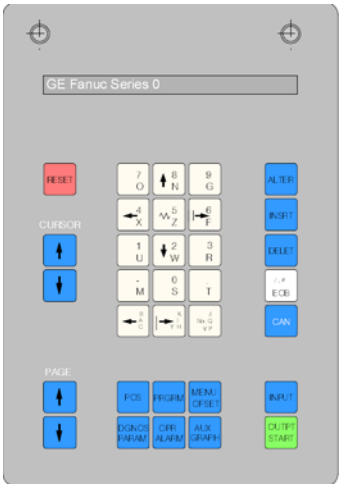
GE FANUC 0
CONCEPT 55 TURN
TEACHER GUIDE

7/13/08 Version 1
Made by EMCO
Authored by Chad Hawk

Training Index

Machine Components

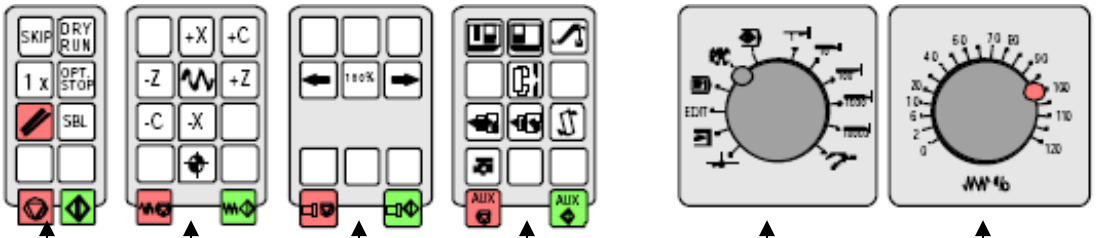
EMCO Control Keyboard



Fanuc 0 Keypad

Fanuc 0 Soft Keys

EMCO Machine Control



Operator Keys

Directional Keys

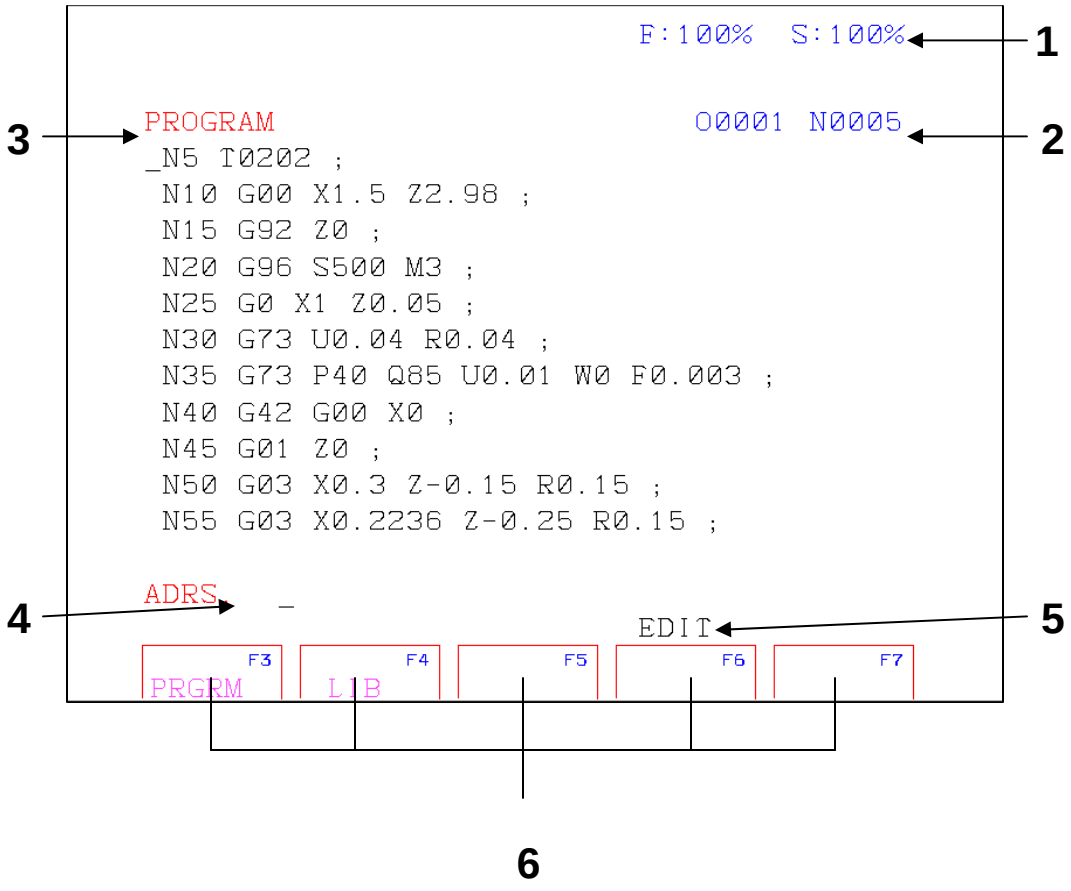
Spindle Keys

Accessory Keys

MODE Dial

Feed Override Dial

The Fanuc O Screen



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

FANUC O KEYS

RESET = cancels most alarms, resets program, interrupts programs

CURSOR MOVEMENT KEYS

CURSOR UP = moves cursor up

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

PAGE UP = moves one page up

PAGE DOWN = moves one page down

CHANGE KEYS

ALTER = alter word (replace word)

INSRT = insert word, create new program

DELET = deletes word / block or a program

EOB = end of block / skip block

CAN = deletes entries in the address

STORE KEYS

INPUT = inputs program / offsets / word / numbers

OUTPT / START = sends program / offsets out

DATA INPUT KEYS

Continually press keys to see all possibilities of that Key.

Press one time a letter appears
Press again a number appears

FUNCTION KEYS (DISPLAY KEYS)

POS

POS = displays actual, relative, machine positions

PRGRM

PRGRM = displays program, library page

MENU OFSET

MENU / OFSET = displays wear, geometry, work shifts pages

DGNOS PARAM

DGNOS / PARAM = displays parameters, diagnostic pages

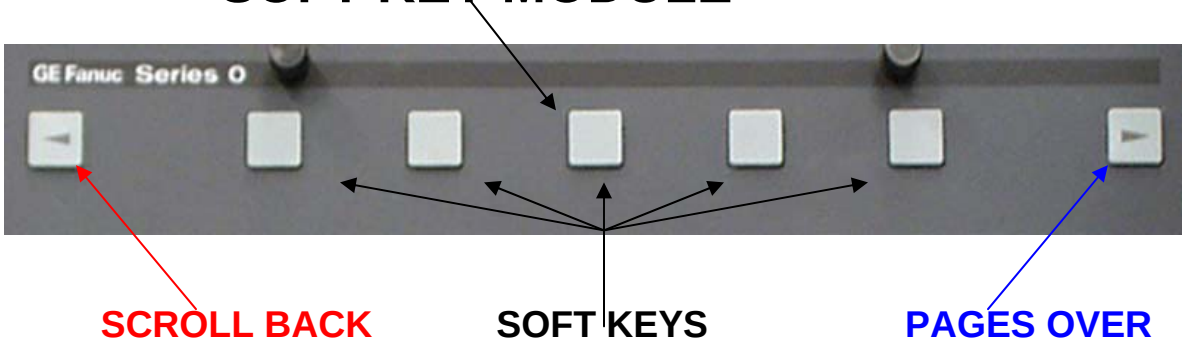
OPR ALARM

OPR / ALARM = displays operator & alarm messages

AUX GRAPH

AUX / GRAPH = displays 2–D graph simulation

SOFT KEY MODULE



EMCO MACHINE KEYS

= Operator Keys

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

DRY RUN **= Press for test run without spindle on and rapids only (remove raw material from vise)**

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

OPT. STOP **= (Optional stop) for programs with (m1)**

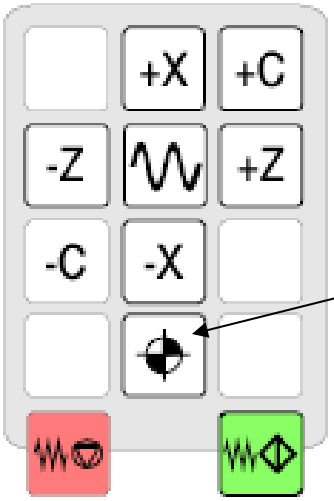
= (Reset) cancels most alarms, resets program, interrupts programs

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

= (Cycle stop) program hold, feed hold

= (Cycle start) program start

Note: Skip, Dry Run, Optional Stop, and Single Block will show at the top of the screen when pressed. When pressed again they will disappear and turn off.



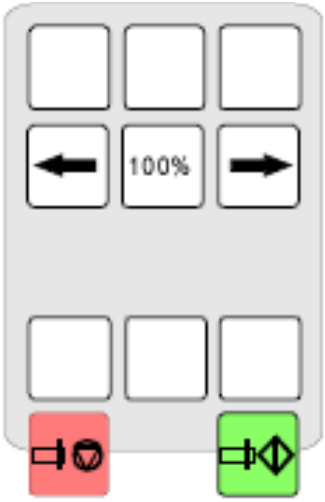
DIRECTION KEYS

These keys control axes directional movements

+4 & -4 = Additional axes

Reference all (Doesn't work for 55 Turn's)

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



SPINDLE OVERRIDE KEYS

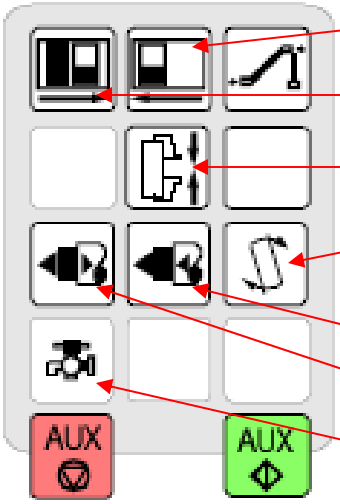
Arrow key pointing right increase the Spindle speed (120% high)

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 once chuck open
Press again chuck closed

Press turret index's one time clockwise
Each time pressed

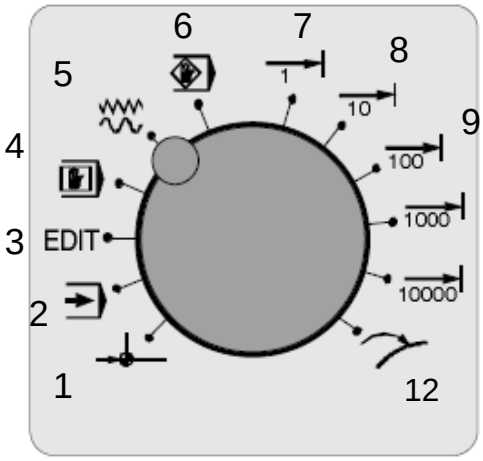
Press tailstock moves backward

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) 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, Z

(6) SIEMEN MODE (Not used on Fanuc)

(7) STEPS = .0001 or tenths

(8) STEPS = .0010 or thousands

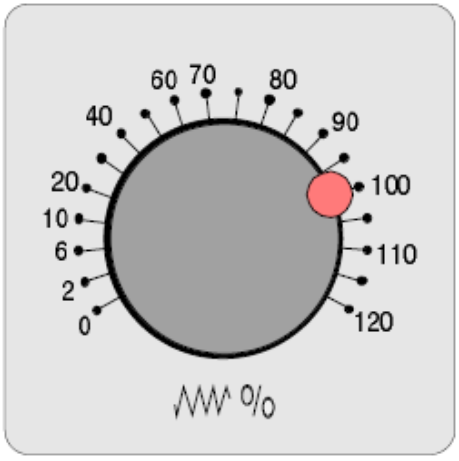
(9) STEPS = .0100 or ten thousands

(10) STEPS = .1000 or hundred thousands

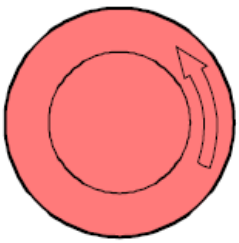
(11) STEPS = .1000 or hundred thousands

(12) SIEMEN MODE (Not used on Fanuc)

FEED OVERRIDE DIAL



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

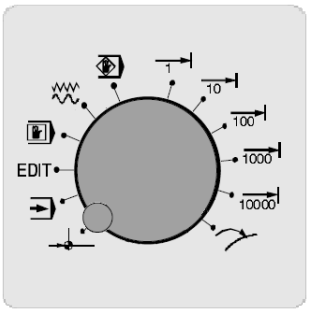


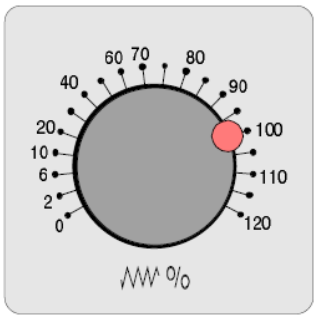
E Stop or Emergency Stop

Turning the Machine On/Entering Fanuc Software

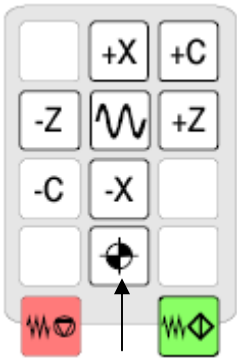
Referencing the Machine

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





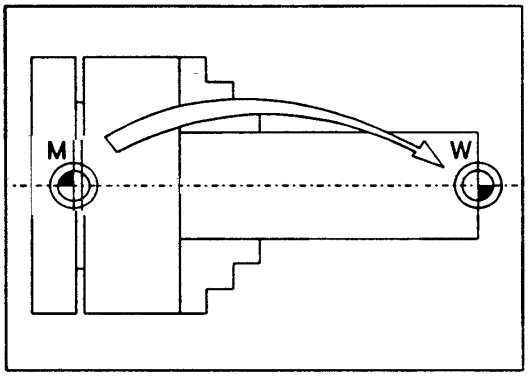
2. Make sure the Door is closed
3. Press the X+ (arrow pointing up) this references the X axis. (Wait until X is fully reference)
4. Press the Z- (arrow pointing left) this references the Z axis



Reference all axis doesn’t work for 55 Turn because of the direction turret travels to be reference

Note: Every time you enter Fanuc 0 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.

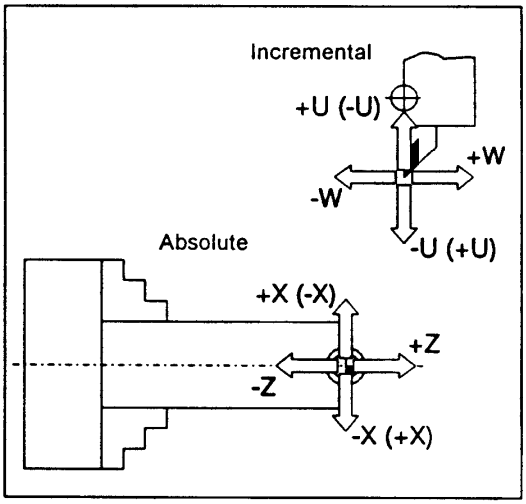
The offset register offers one adjustable zero offset.

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:

Note: There are 2 main ways of doing this Education way or Industry way. Step 1 thru 3 is for the Education way; skip these steps if you are setting up Industry way; go to step 4.

1. Index to a empty ID location
   - Manually index by going to Jog Mode and Pressing Index button 

   OR

   - Programming Index

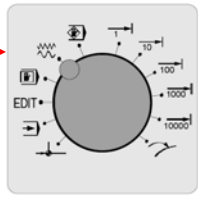
     Rotate Mode Dial to MDI

     Press the PROGRAM display button 
     Until top left of the screen shows 

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

     Press Input button 

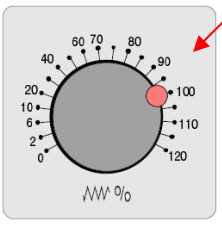
     Then press CYCLE START  **(Door must be closed)**

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 MENU/OFSET button 

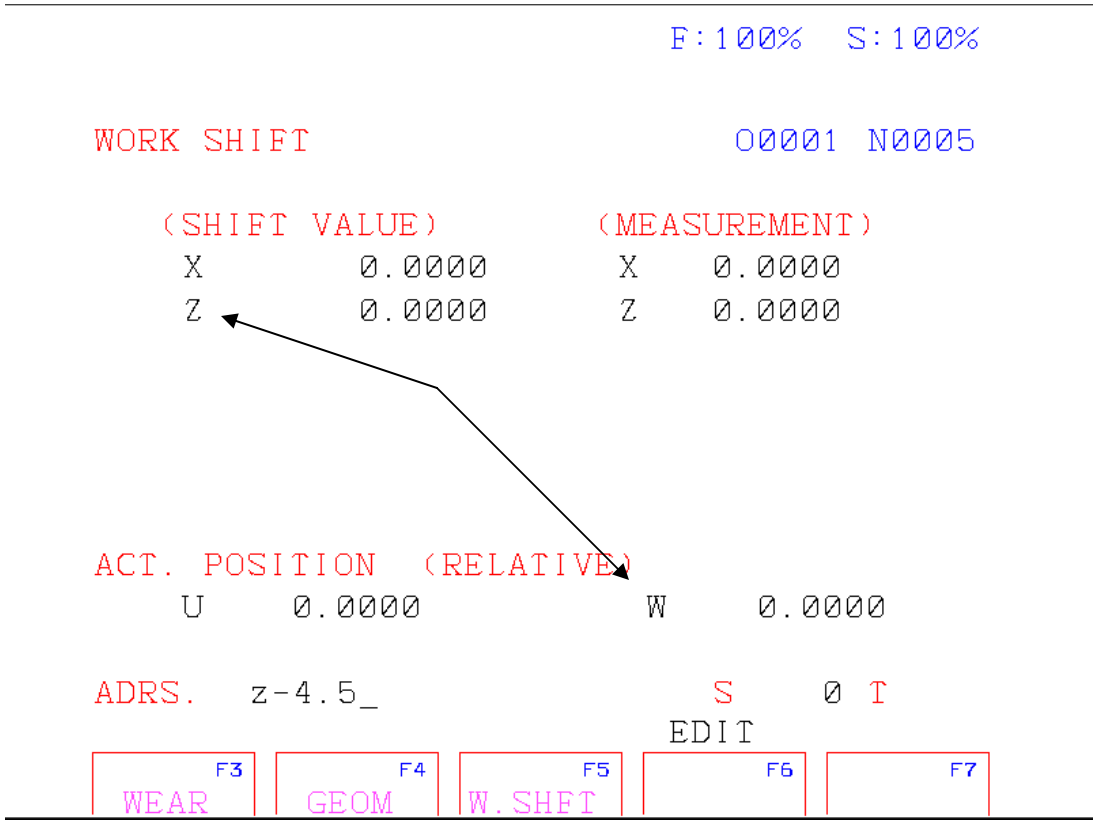
    - Press the W SHIFT Soft key (Gray Button)
5. Make sure the (Shift value) Z is 0 if not type in Z0 and Input

(Industry Way skip steps 6-8 continue on to Tool Offsets)

6. The value that is in the ACTUAL POSITION (RELATIVE) W type this value in (SHIFT VALUE) Z as a negative number
7. Press Input 

8. Jog TURRET away from WORK PIECE using Z+

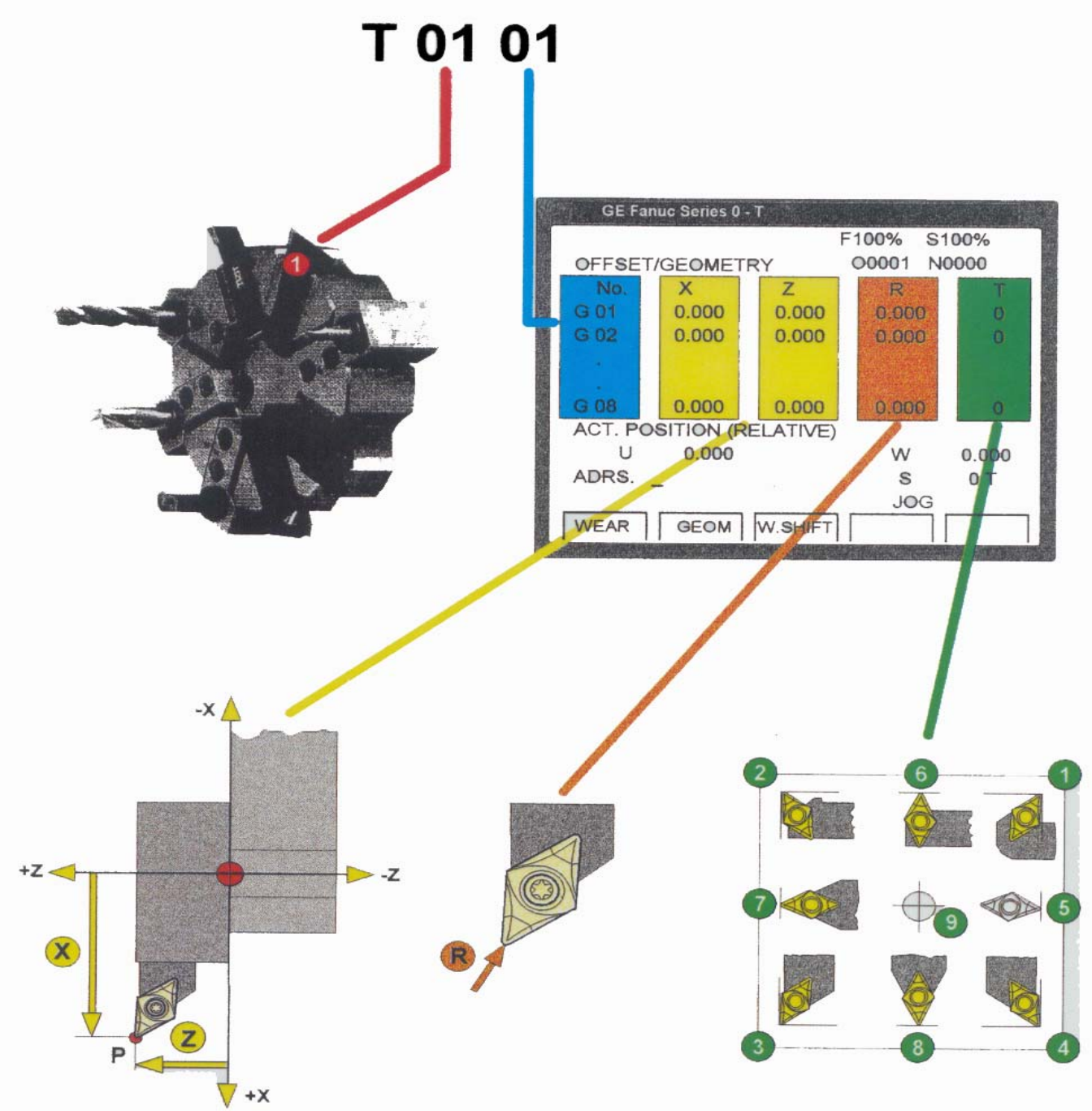
This value is the distance from the Spindle Nose to the end of the Work Piece



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

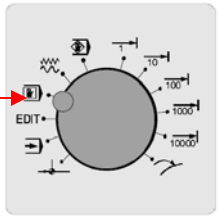
NEVER put a value in SHIFT VALUE X

TOOL OFFSETS



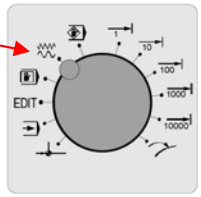
Tool Offsets

1. Index the TURRET to the tool to be measured



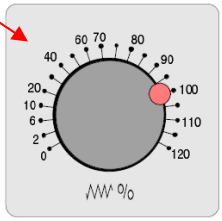
   - To do this Move the MODE Dial to MDI position
   - Press the Program (display button) PRGRM
   - Type tool number then press INPUT button INPUT
     Example: T0200
     1. For Scratching

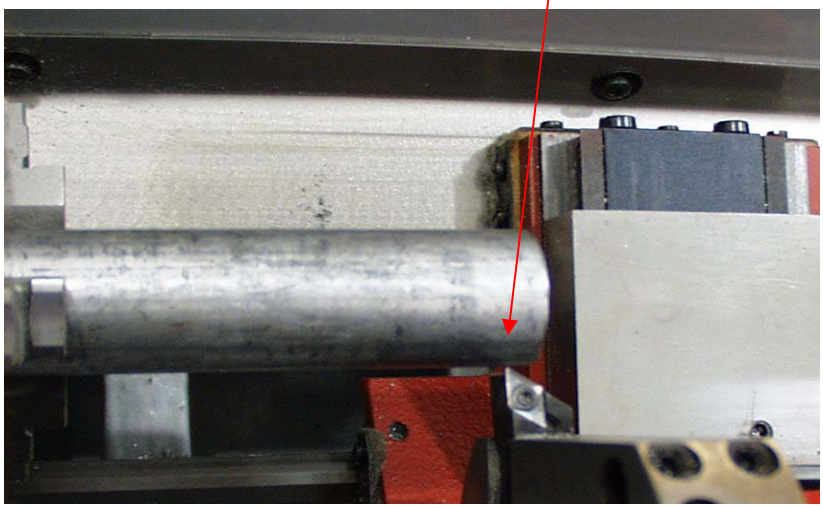
        Type S1000 for RPM press INPUT then Type M03 for spindle on clockwise press INPUT
   - Then press CYCLE START (make sure door is closed)
2. Move the MODE Dial to JOG position



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 MENU/OFSET button 

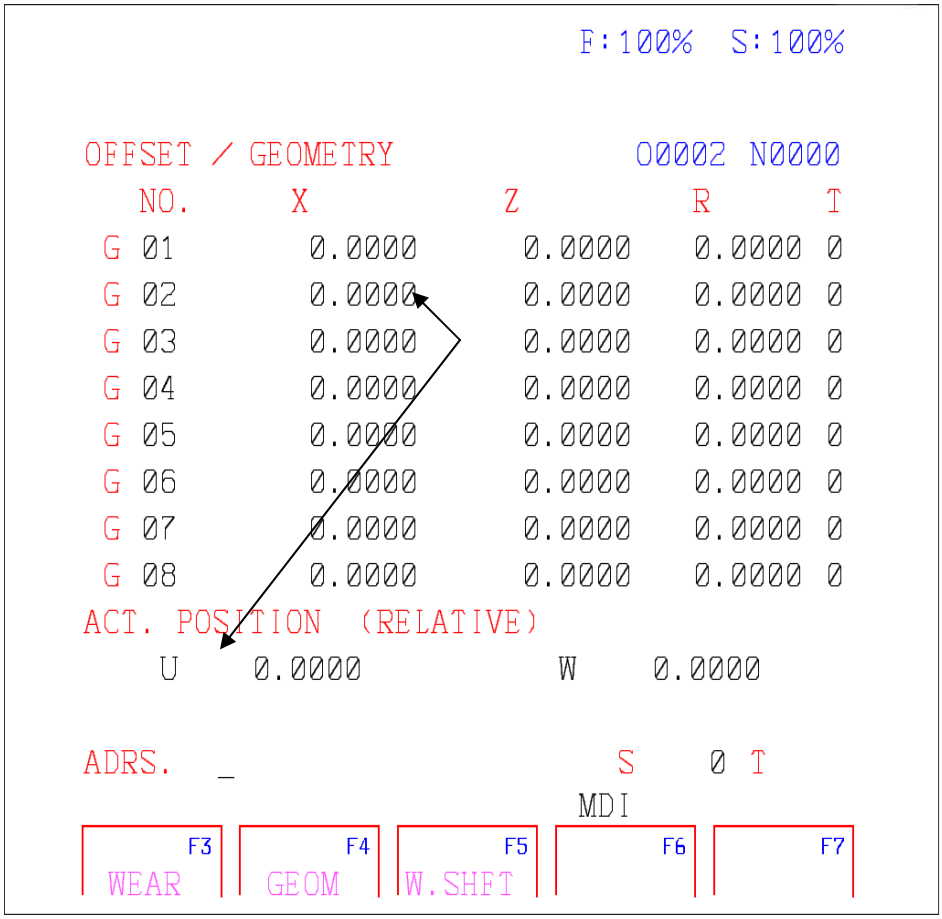
    - Press the GEOM Soft key
5. Take the value in Actual Position (Relative) U and subtract the Diameter of the Work Piece being scratched
6. Type this value in G02 for X (If the tool being use is T0202)

    Example: U is 2.962 Type X 1.962 (If stock is 1”dia.)

7. Then press INPUT 

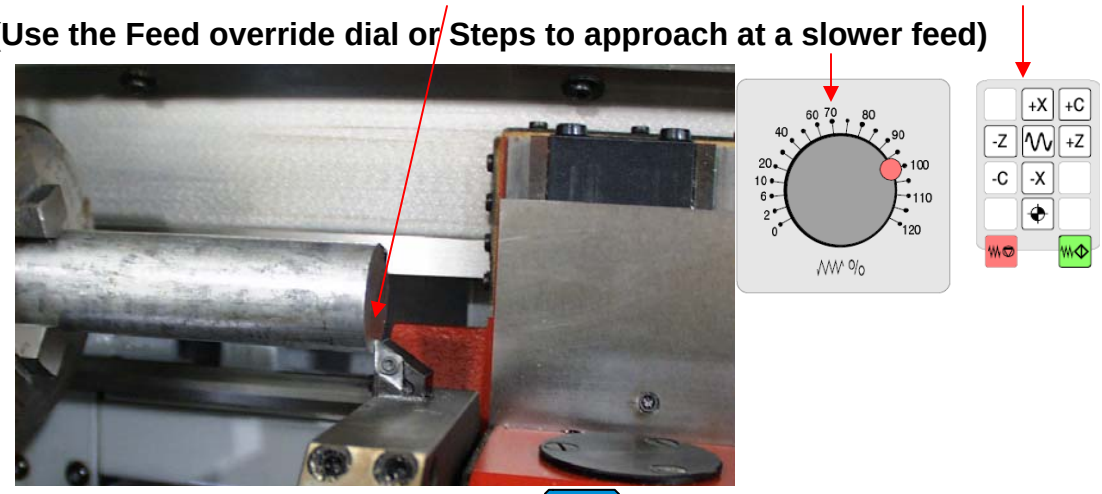
8. Jog TURRET away from WORK PIECE using X+

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



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)



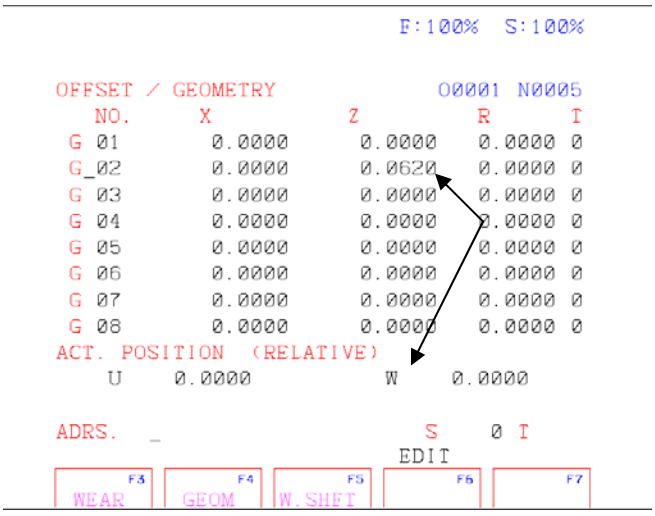
10. Press the MENU/OFSET button 

- Press the GEOM Soft key

11. The Value in the Actual Position (Relative) W type this value in G02 for Z (If the tool being use is T0202)

    Example: W is .625 Type Z .625

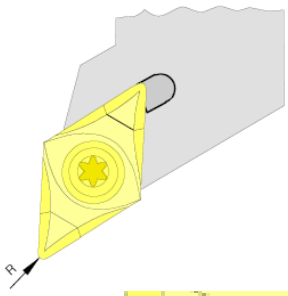
12. Then press INPUT button 



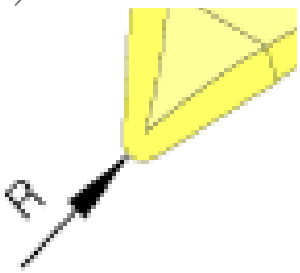
Note: Industry way the value for Z will be a large value
(This is the distance from spindle nose to the program 0 / front of the work piece)

13. R is the Tool Tip Radius

- Cursor over to the R column and type in the value from below that matches the insert type then press input button



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

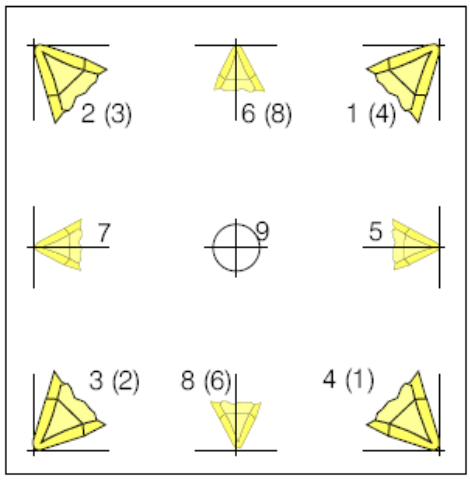
55° insert = .015 Parting Off or Groove = .003

80° insert = .032 35° insert = .010

Threading insert = .001

14. T for cutter comp cutting direction

- Cursor over to the T column and type in the number from below that matches the tool direction then press input button



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

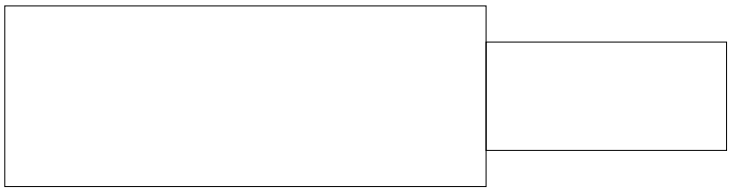
Emco 55 Turning machines the numbers to use are in the brackets.

All machines that have a turret on the bottom of the spindle 0 line will also use the bracket #'s. Machines with turret on top will use regular #'s

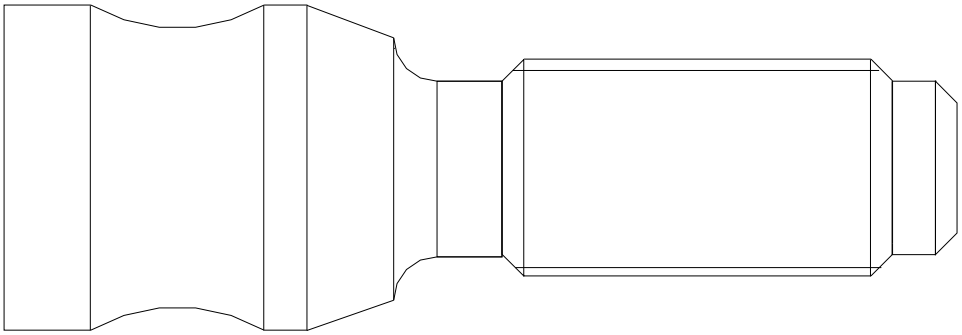
15. Jog TURRET away from WORK PIECE using Z+
16. Repeat steps for all OD tools (STEPS 1-15)

Program Training

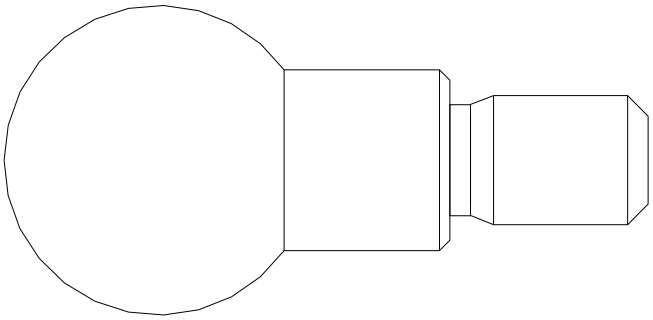
Program O0001



Program O0003



Program O0004



Change the Mode Dial to Edit
Press the Program Button
To do functions on pages 21 - 23

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
To do functions on pages 21 - 23

PRGRM

- ## DELETE A PROGRAM

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

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

- ## DELETE ALL PROGRAMS

1. Press letter o plus the – & 9999

2. Press delete button DELET

Example: **<u>O – 9999</u>**

- ## DELETE A WORD

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

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

- ## 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.

Change the Mode Dial to Edit
Press the Program Button
To do functions on pages 21 - 23

PRGRM

- ## 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)

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 Interpolaton
	+	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 hold drilling, cut-in cycle in Z
	+	G75	G75	G77	Cut in cycle in X
	+	G76	G76	G78	Multiple threading cycle
	+	G50	G92	G92	Coord.syst.set., Spindle speed limit
1	•	G00	G00	G00	Positioning (rapid traverse)
		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	Inkremental 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

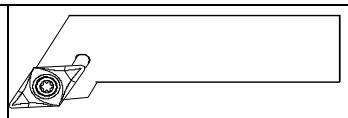
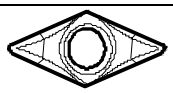
R Radius, retraction, cycle parameter

S Spindle speed

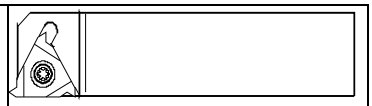
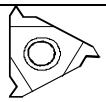
T Tool called out

X, Z Position data in absolute

Tool Position 2 needed for Program 1, 2, 3, 4

260 601	Right hand Turning Tool	No. SDJCR 1210 D07	
271056	Indexable inserts for Aluminum	No. DCGT 070204-27 H10T	

Tool Position 4 needed for Program 2, 3, 4

260 620	OD-thread tool Right	Max. Pitch 1,5 mm (.040”) No. NL 1210-2 RH	
260 621	Indexable inserts for OD-thread tool	Pitch 0,5 - 1,5 mm (.040”) No. 16ER T A60° S36T	

Program screen & Edit mode

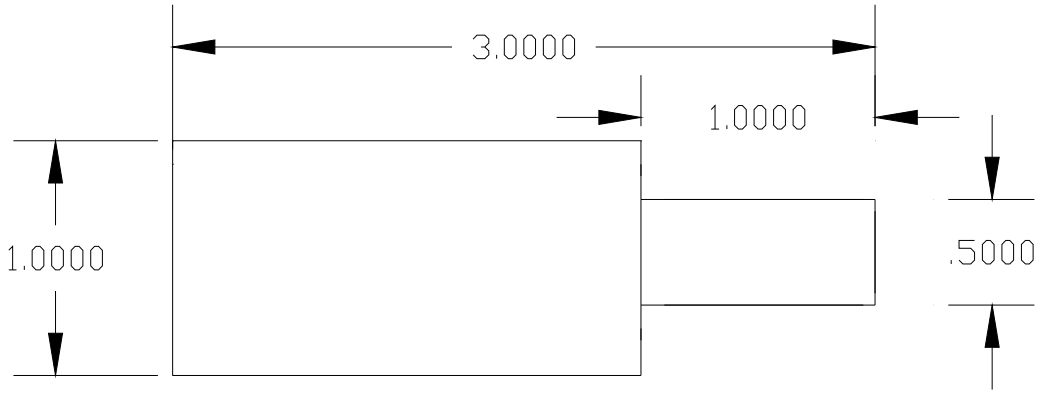
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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 O0001



G73 U = Depth of Cut R = Retract Value

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 (3.25 x 1 alum)

N10 G40 G70 G80 G90.....Default G Codes (Not Needed)

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

N20 G0 Z2.0……..safe move

N25 T0202 S550 M3 (Finish Tool 55°)

N30 G0 X1.0 Z.1……start point of cycle

N35 G73 U.03 R.015……..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……..1st diameter of contour

N60 Z-1.0……length of contour

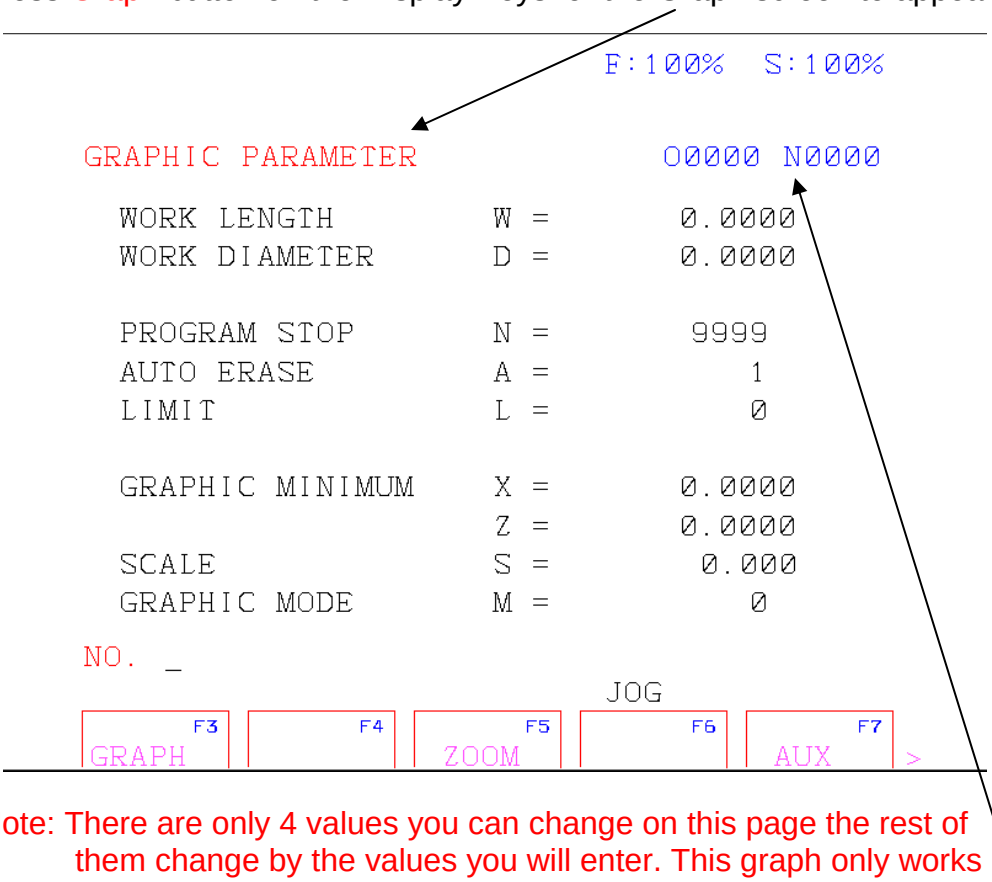
N65 X1.0……..diameter of contour

N70 G0 Z2.0……safe move

N75 M30……end of program

2D Simulation

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



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

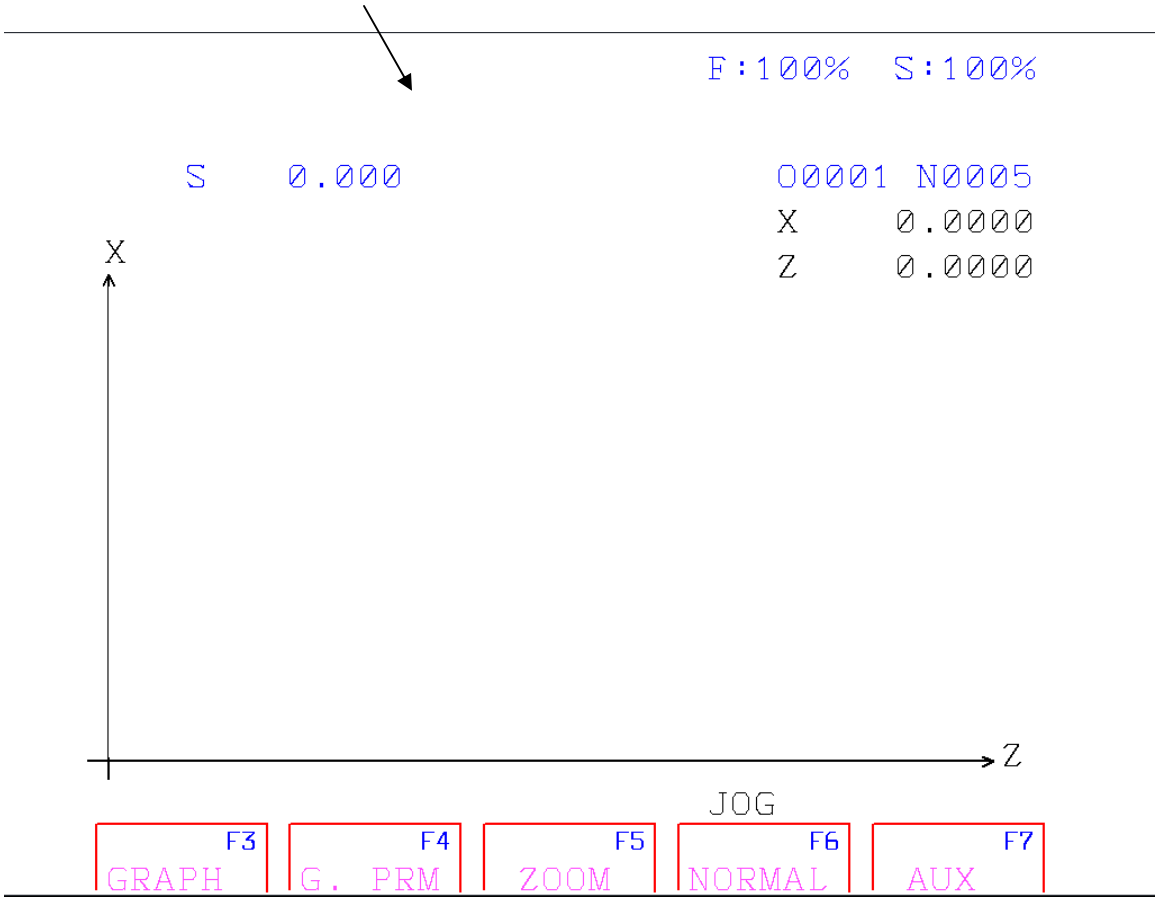
2. Work Length W = Overall length of stock in the Z direction this is a + value

3. Work Diameter D = Overall diameter of stock in the X direction + value

4. Graphic Minimum 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 Minimum 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 **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 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 (**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)

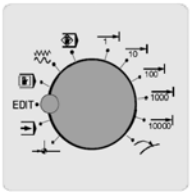
    3. Press (**Input)** key

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

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 

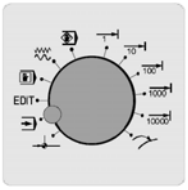
1. Rotate the Mode dial to Edit

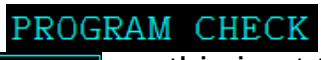


2. Press the Program button 

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

4. Rotate the Mode dial to MEM



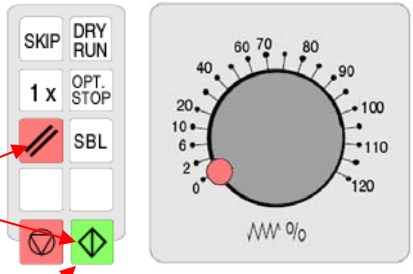
5. If screen is not in 

then press program button or the softkey 

so this is at the top left of the screen

6. Press the Single Block button for the program to run one line at a time. 

Note: Use one hand on the feed override dial slowly increasing it and the other pressing cycle start and close to the reset button



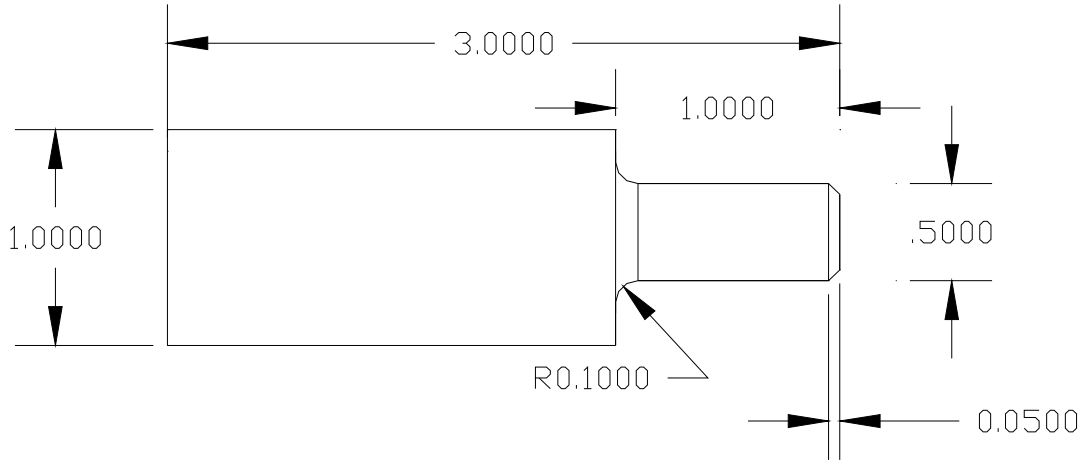
7. Press Cycle Start and continue

(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 using C/R's



G73 U = Depth of Cut R = Retract Value

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)

O0002(Demo 2)

N5 (3.25 x 1" alum)

N10 G40 G70 G80 G90

N15 G95 **G96** G98…….....sfpm

N20 G0 Z2.0……..safe move

N25 T0202 S550 M3 (Finish Tool 55°)

N30 G0 X1.0 Z.1……start point of cycle

N35 G73 U.03 R.015……..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

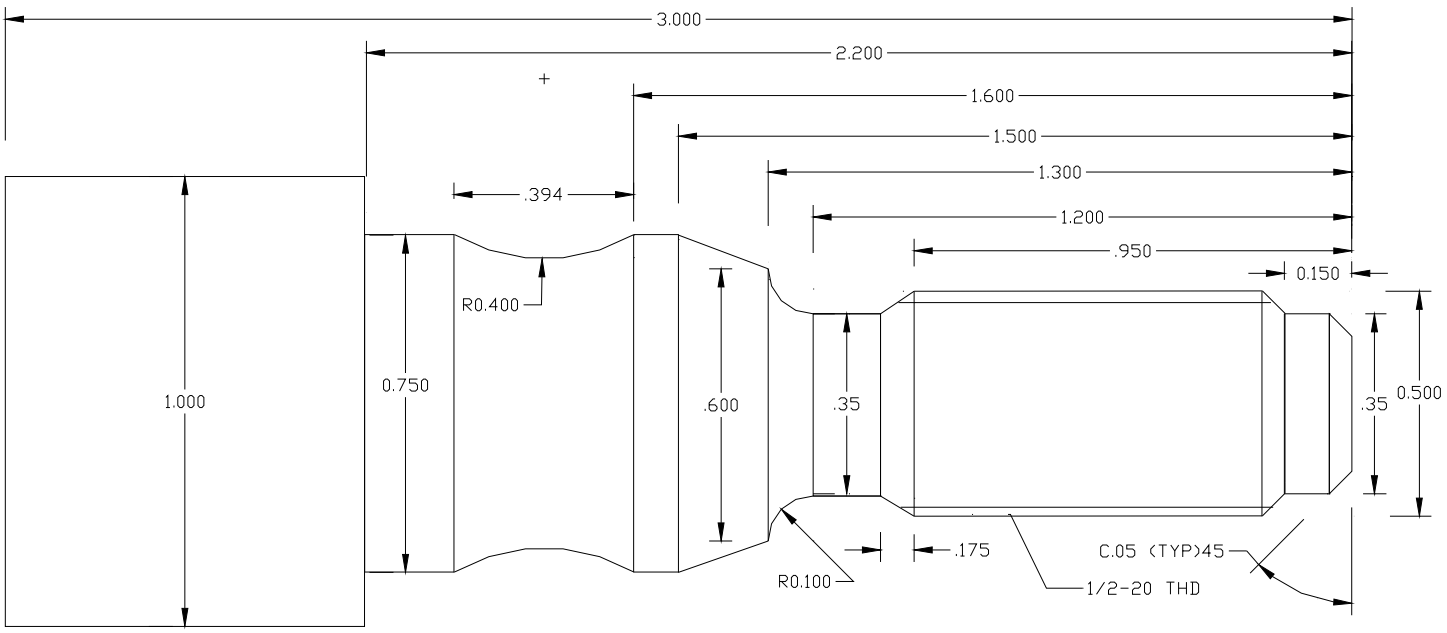
N60 Z-1.0 R.1……length of contour

N65 X1.0……diameter of contour

N70 G0 Z2.0……safe move

N75 M30……end of program

Program O0003



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

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

Micro IN is the value without the decimal point

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

HINT:
Threading

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

IPM = RPM X PITCH

$RPM = \frac{IPM}{PITCH} = \frac{50}{.05} = 1000\ RPM$ 78 is max for a Concept 55 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 O0003

O0003 (Demo 3)
N5 (Stock 3.25 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 Z2 ……Safe Index Pos
N140 G97 S560 M3 ……...Threading 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 Z2 ……...……Safe 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.

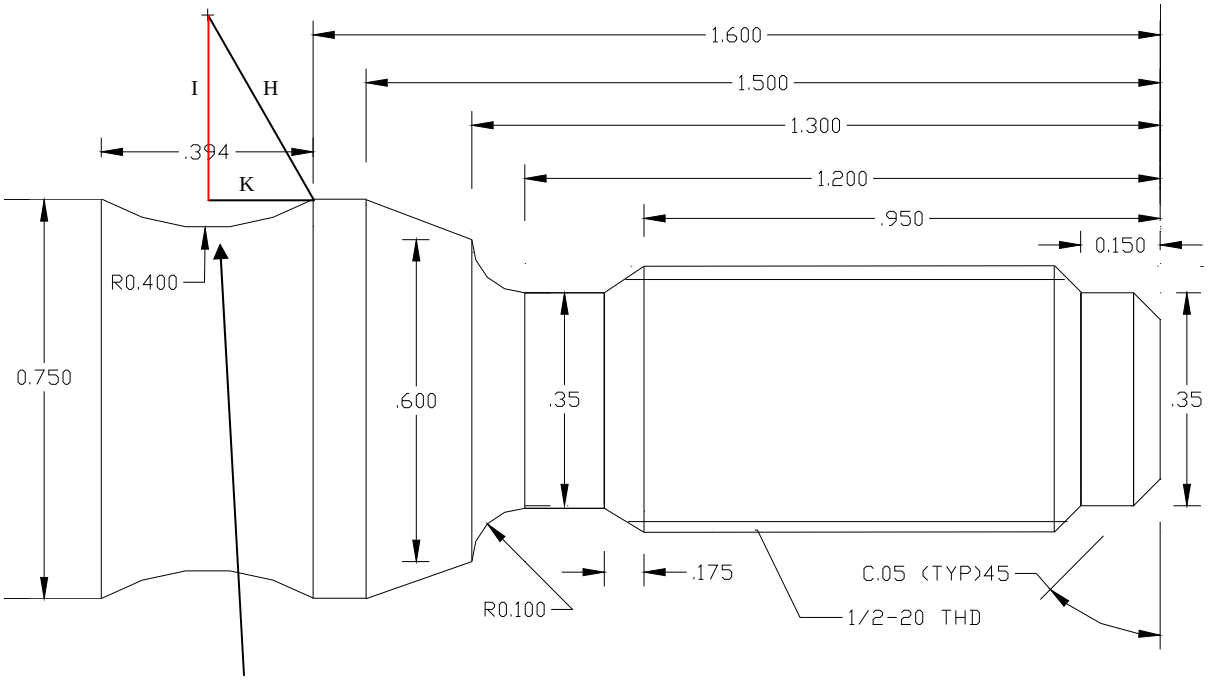
4. All programs can be used as Sub Programs or Main Programs M99 means program is Sub, M30 means program is a Main

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

O0006 (Tie Programs)
N5 (Stock 3.25 x 1 alum)
N10 M98 P010002 (Demo 2 C/R)
N15 M98 P010003 (Demo 3)
N20 M30
Changing Item
Note: Change the end of O0002 and O0003 to M99 for running them as SUB PROGRAMS

I and J Test



Find the I and J for the arc in the picture

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

$\sqrt{}$

Sally Can Tell Oscar Has A Hat On Always

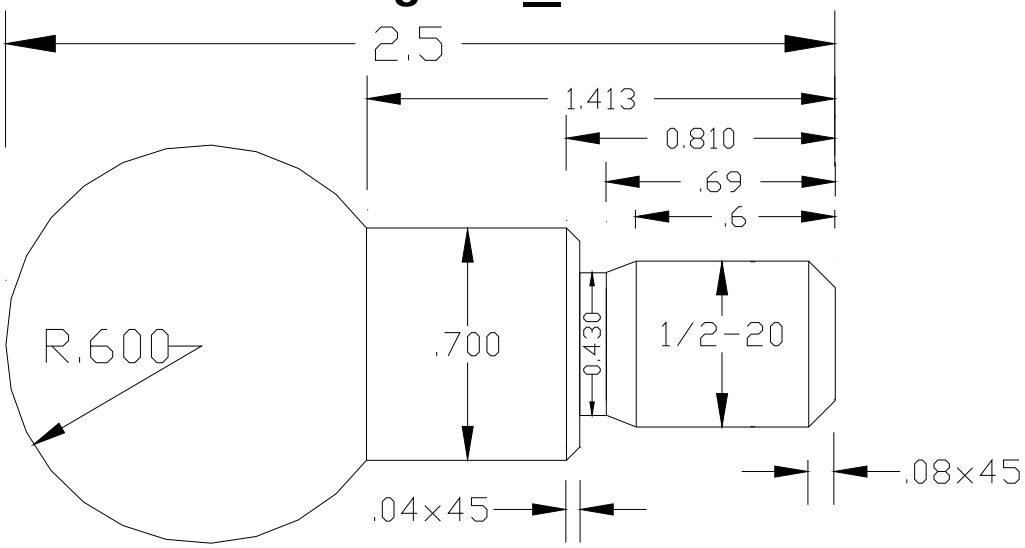
$S \frac{O}{H}$

$C \frac{A}{H}$

$T \frac{O}{A}$

SINE COSINE TANGENT

Program O0004



O0004 (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 P010005 (SUB PROGRAM FOR BACK SIDE)
N160 M30

Program O0005

O0005 (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)**

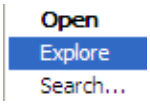
   - **Press** 

to allow you to exit
   - **Press** 

and  **together to exit the Software**

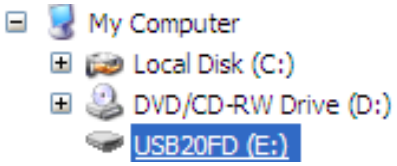
2. **Make sure USB is plug into port**

3. **Open Explorer**

   - **Right Click on Either My Computer, My Documents or any Folder on the Desktop**
   - **Move mouse to** 

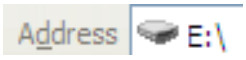
(Explorer)
   - **Left Click**
   - **If you right clicked on My computer skip to step 4 if not then Left Click on My Computer**

4. **Copy Drive directory**



   - **Click on you USB drive**

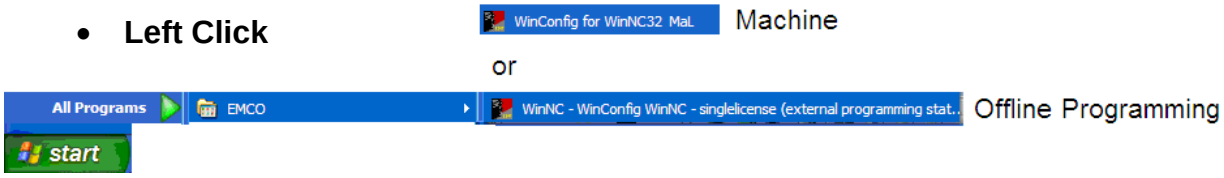
   - **At the top of the active screen or page in the Address copy or remember drive info**



   - **Close the active screen or page using either Alt and F4 or**  **at top of the active screen**

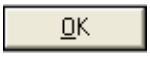
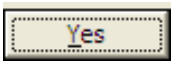
5. **Setting up WinConfig**

   - **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)**
   - **Left click on white box**

   - **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**