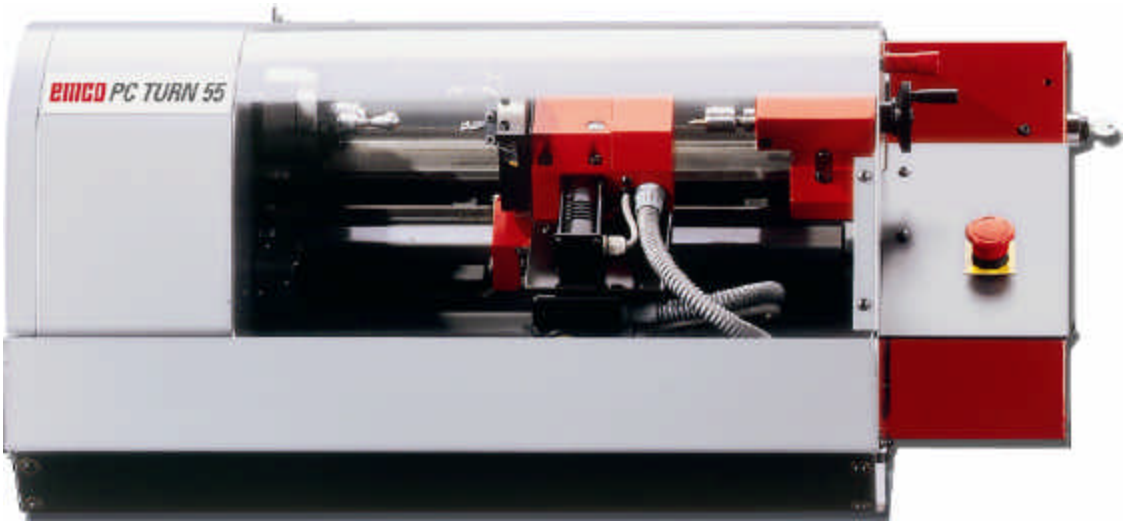
innovative machine tools



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

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

Training Index

Fanuc 0 Screen ..... Pg 1

Fanuc 0 Keys ..... Pg 2

- **Cursor Movement Keys**
- **Change Keys**
- **Store Keys**

Function Keys (Display Keys)..... Pg 3

- **Machine Function Keys**

Direction Keys ..... Pg 4

- **Spindle Override Keys**
- **Accessory Functions**

Mode Dial ..... Pg 5

- **Feed Override Dial**

Pc Keyboard Keys ..... Pg 6

Referencing the Machine ..... Pg 7

Work Shift Description (Picture) ..... Pg 8

Work Shift (How to do Work Shift) ..... Pg 9

Tool Offset Description (Picture) ..... Pg 11

Tool Offset (How to do Tool Offsets for X) ..... Pg 12

- **Manually programming Turret Index**
- **Manually programming Spindle on**

Tool Offset (How to do Tool Offsets for Z) ..... Pg 14

Program Training ……..….. Pg 16

Inserting a New Program ……..….. Pg 17

- **Calling a Existing Program up**
- **Insert a word**
- **Insert a End of Block**

Delete a Program ……... Pg 18

- **Delete all Programs**
- **Delete a word**
- **Delete a Block**

Cancel word …….. Pg 19

- **Alter a word**
- **Search for number Block**
- **Search for word**

G Codes ……... Pg 20

M Codes …… Pg 21

- **Used Addresses**

Program 1 …….. ……... Pg 23

2 D Simulation (Setup) …… Pg 24

Input and Output of Programs & offsets thru Fanuc software….. Pg 26

Program 1 (C & R) …… Pg 27

Program 2 (G73 and G72 Description) . …… Pg 28

G78 Description …… Pg 29

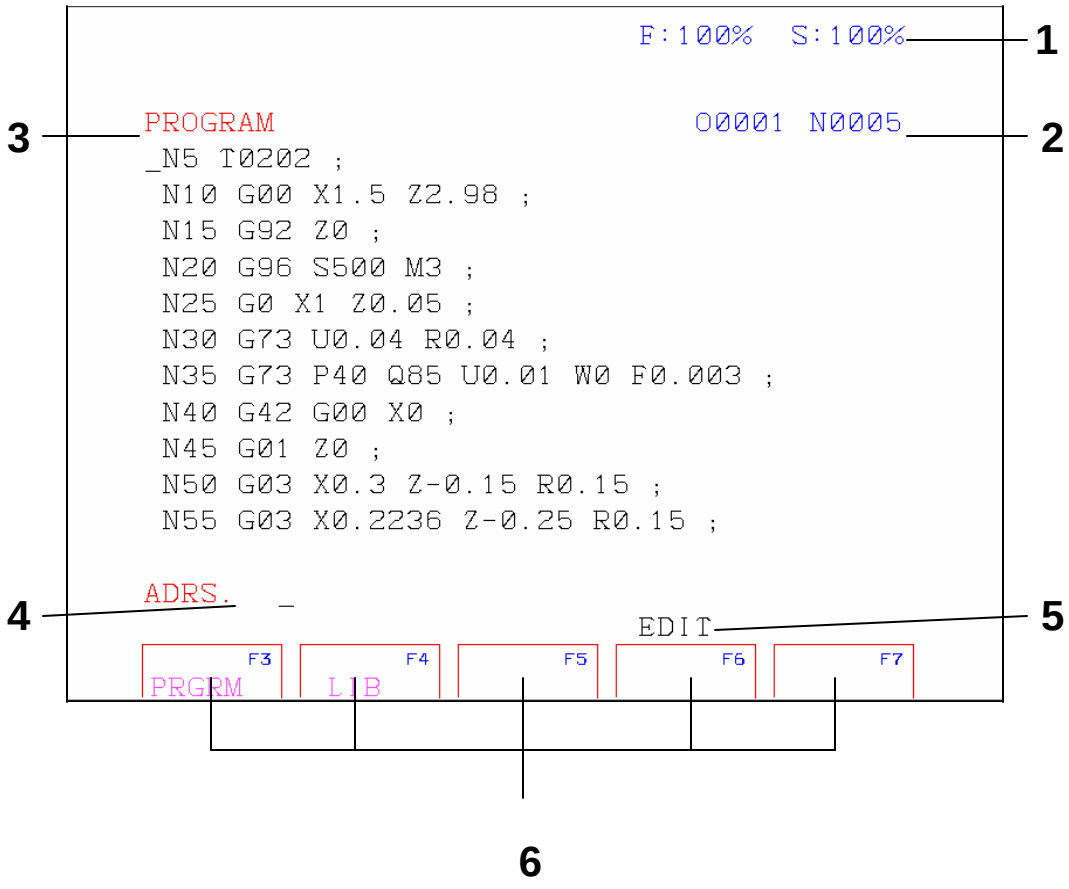
Program 2 ……... Pg 30

Sub Programming..…… Pg 31

Program 3 (Ball) ..…… Pg 32

Program 4 (Ball) ..…… Pg 33

The Fanuc O Screen



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

FANUC O KEYS ON PC KEYBOARD

CURSOR MOVEMENT KEYS

Arrow key pointing up is CURSOR UP = moves cursor up

Arrow key pointing right moves cursor right

Arrow key pointing left moves cursor left

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

PAGE UP = moves one page up

PAGE DOWN = moves one page down

CHANGE KEYS

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

Enter is the INSRT = insert word, create new program

DELETE = deletes word / block or a program

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

Back space is the CAN = deletes entries in the address

STORE KEYS

F10 is INPUT = inputs programs & offsets

F9 is OUTPUT = sends program & offsets out

FUNCTION KEYS (DISPLAY KEYS)

F12 TOGGLES THE MENU FOR THE DISPLAY KEYS

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

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

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

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

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

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

MACHINE FUNCTION KEYS

 # keys is the same as Numeric keypad or 10 key

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

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

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

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

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

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

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

DIRECTION KEYS

These keys control axis directional movements

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

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

SPINDLE OVERRIDE KEYS

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

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

Ctrl & 6 = Spindle stop

Ctrl & 7 = Spindle start

All spindle keys work in all modes except EDIT & ZRN

ACCESSORY FUNCTIONS

Press Ctrl & + for Door open

Press again Door closed

Press Ctrl & 3 for Rotary axis Indexing

Press Ctrl & 0 tailstock backward

Press Ctrl & 9 tailstock forward

Press Ctrl & 2 puff blowing ON

Press again puff blowing OFF

Press Ctrl & 8 auxiliary drives on

Press Ctrl & - auxiliary drives off

Press Ctrl & 1 index Tool turret

Press Ctrl & ~ open chuck

Press again close chuck

MODE CONTROL

F1 TOGGLES THE MENU FOR THE MODE CONTROL

F1 THEN F7 = ZRN for Reference or Home mode

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

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

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

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

F1 THEN F11 = STEPS Incremental feed movements

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

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

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

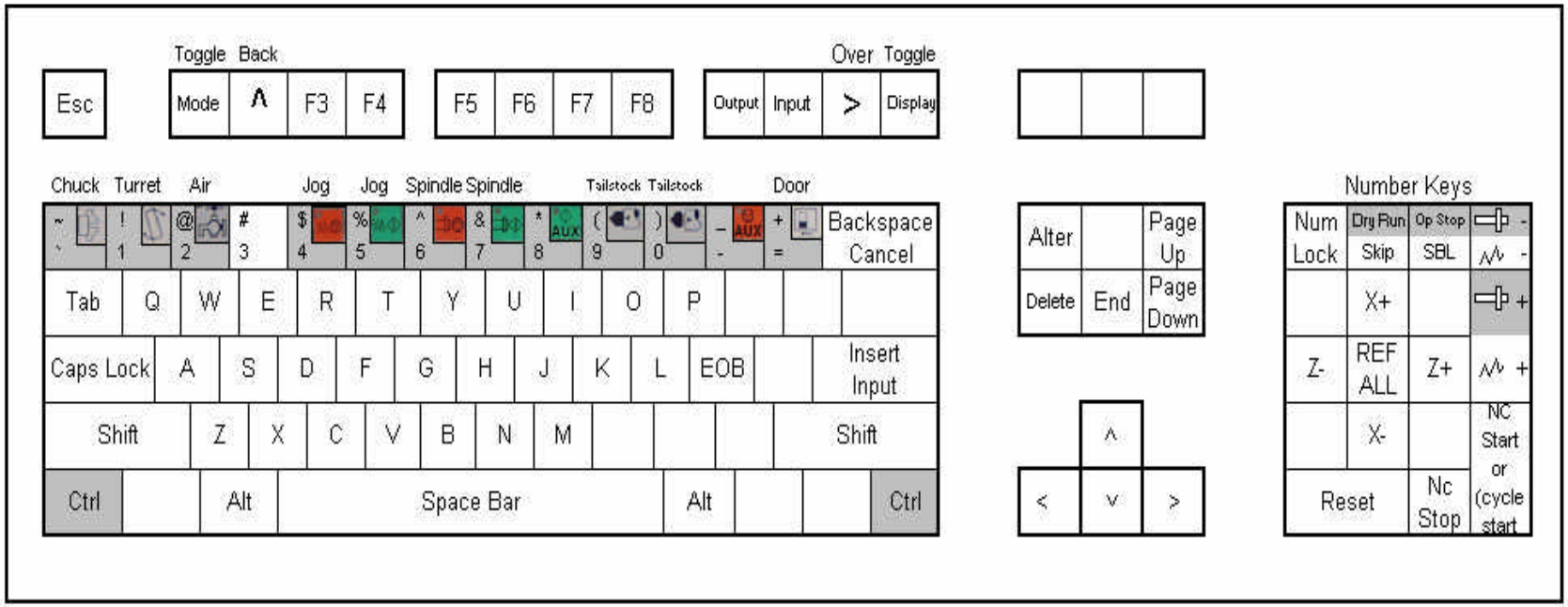
F1 THEN F11 THEN F6 = STEPS 1000 OR .100 hundred thousands

FEED OVERRIDE CONTROL

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

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

These Control feed for jogging in the X-axis / Y-axis / Z axis



1. Any key with Gray highlight Press Ctrl + the key for that function
2. Some keys have two functions to them for 1st function just press the key
3. 2nd function will be Grey press Ctrl + the key for the function
4. Some automative keys when you press them 1 time this will close/turn off press them again will open/turn on
5. F1 is a toggle key for the modes: Zero, Auto, Edit, MDI, Jog and F1 then F11 give Increment Step
6. F12 is a toggle key for the Display screens: Position, Program, Offsets, Parameter, Alarm and F12 then F11 then F3 gives Graph
7. F12 then F11 then F3 then F11 then F3 gives you 3D view
8. Press enter 2 times this is the same as pressing EOB insert
9. Alt + F4 will exit the software back to the desktop
10. The Top right corner will allow the screen to be minimized, restored and close just like a standard windows screen

The machine functions are active only with NUM LOCK on

Keys are active they will move the axes if used as numbers. Use numbers on the keyboard.

Turning the Machine On/Entering Fanuc Software

Referencing the Machine

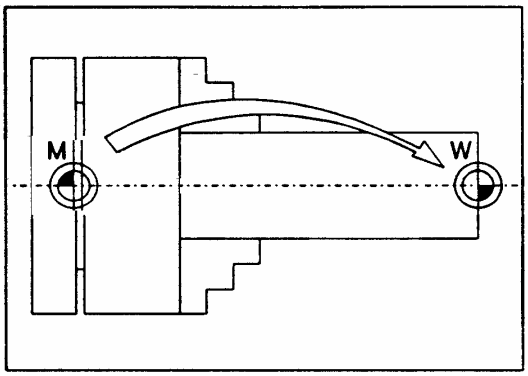
1. Make sure your feed rate is not on "**0**"
2. Make sure door is closed
3. Make sure at the bottom of the screen shows ZRN
4. Press 8 on the # keys this references the X axis.
5. Press 4 on the # keys this references the Z axis

Or just do 5.

6. Press 5 on the # keys this references both axis

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

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.

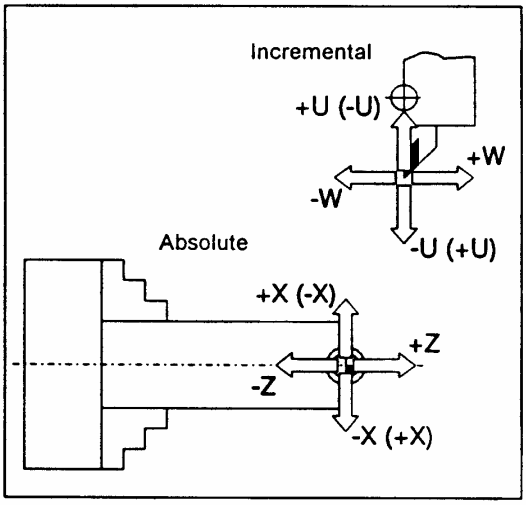
The offset register offers one adjustable zero offset.

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).

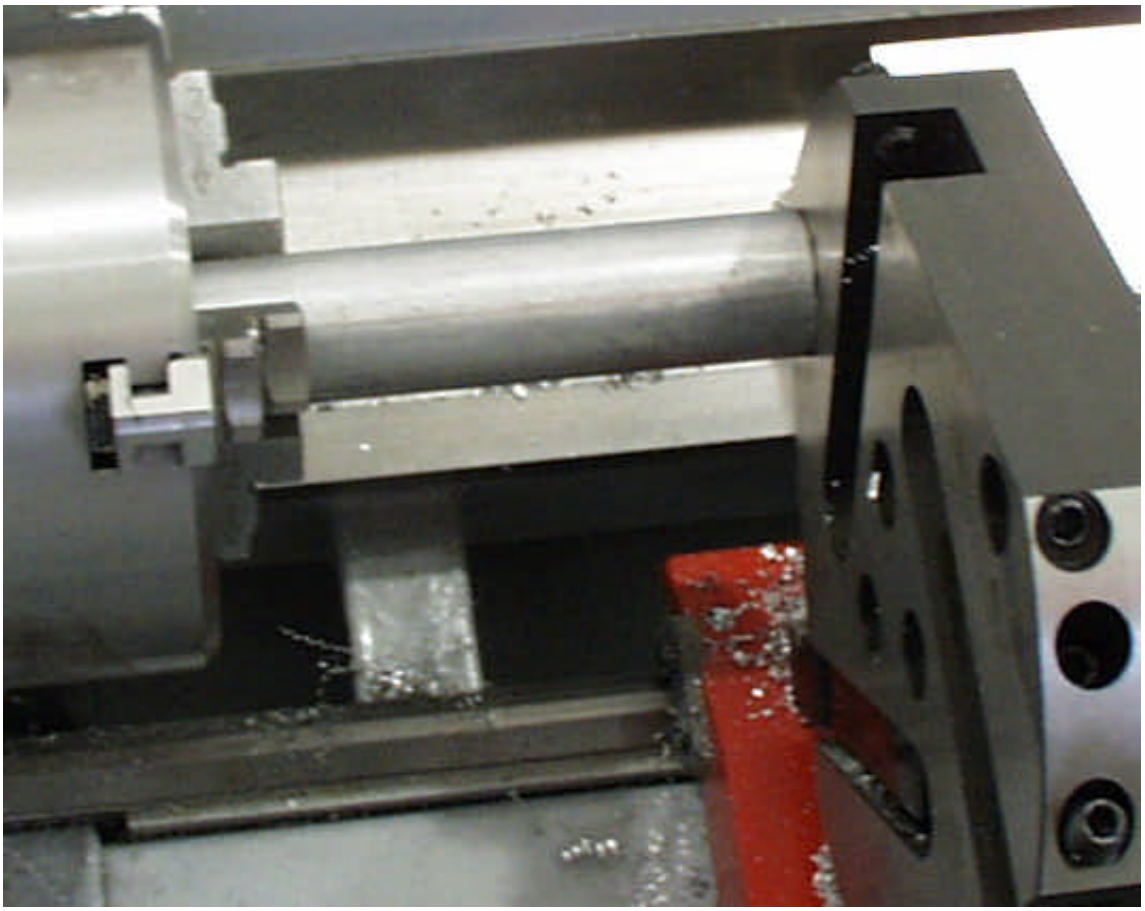
Coordinate System for Incremental Value Programming
The origin of the coordinate system lies at the tool mount reference point "N" or at the cutting tip after a tool call-up.
The U coordinate lies in the direction of the cross slide, the W coordinate in the direction of the longitudinal slide. The plus 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. Press F1 then F6 for JOG position
2. Index to a Empty ID tool position (1, 3, 5)
    - Press Control & 1 will index one tool position at a time
3. Jog the TURRET to the face of the Work Piece & touch using 2,4,6,8 on the # keys.

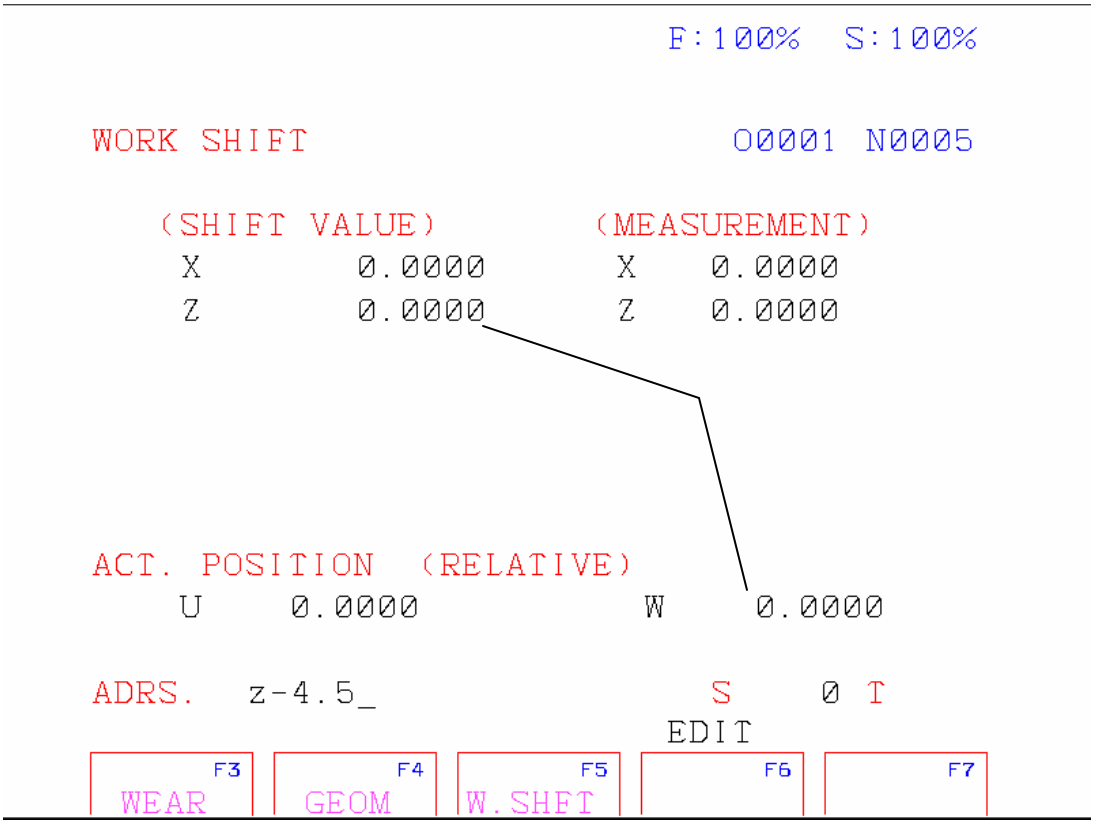
    (Use piece of paper between TURRET and Work Piece)

(Use the Feed – , + or Steps to approach at a slower feed)



4. Press F12 then F5 for OFFSET
5. Press F5 for W SHIFT
6. Make sure the (Shift value) Z is 0 if not type in Z0 and Enter
7. The value that is in the ACTUAL POSITION (RELATIVE) W type this value in (SHIFT VALUE) Z as a negative number
8. Press Enter
9. Jog TURRET away from WORK PIECE using 6 on # keys

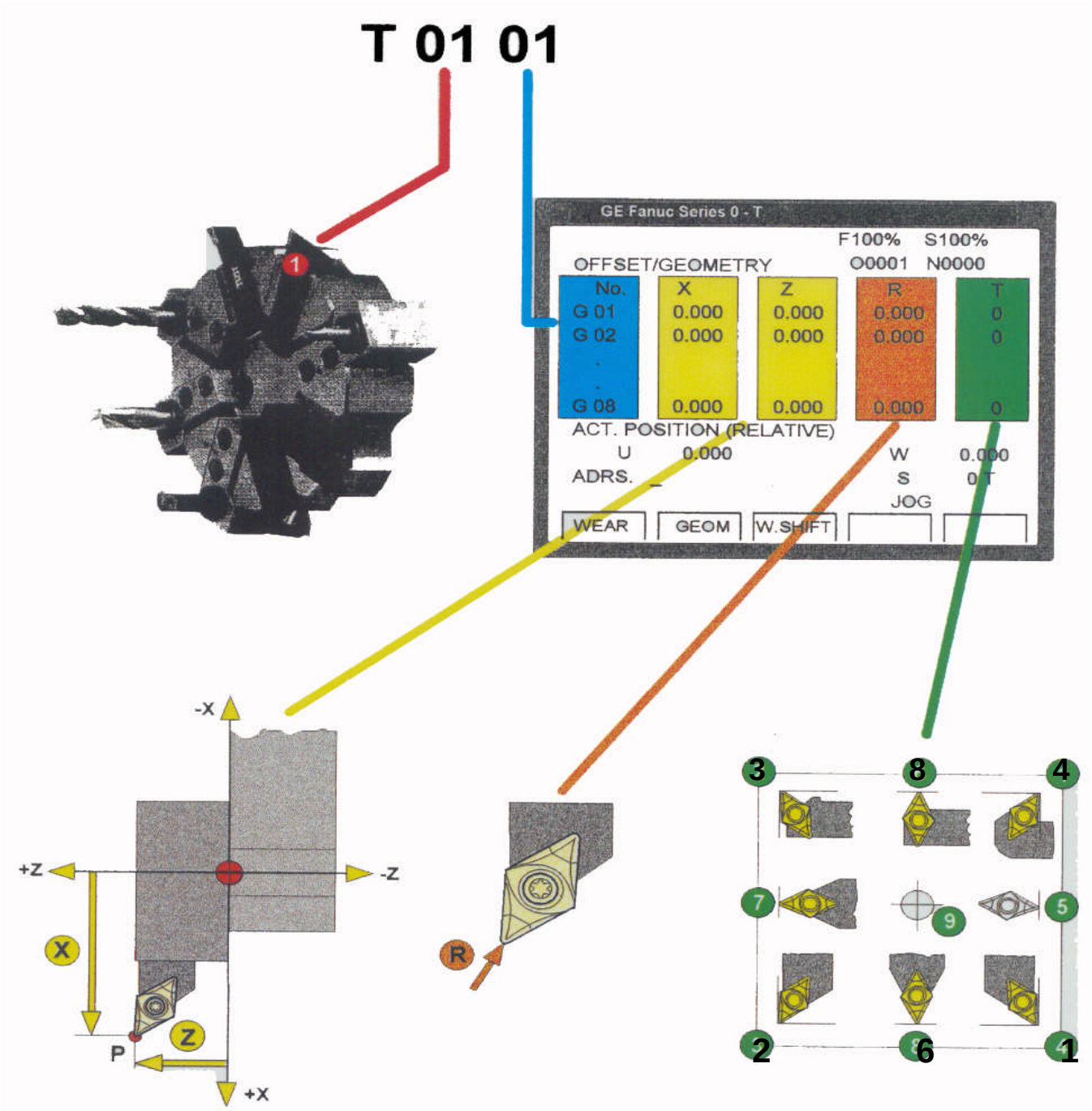
   This value is the distance from the Spindle Nose to the end of the Work Piece



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 a tool to be measured

   To do this

   - Press F1 then F5 for (**MDI)** Press F12 then F4 for (**Program**)
   - Type tool number then press Enter
     Example: T0202

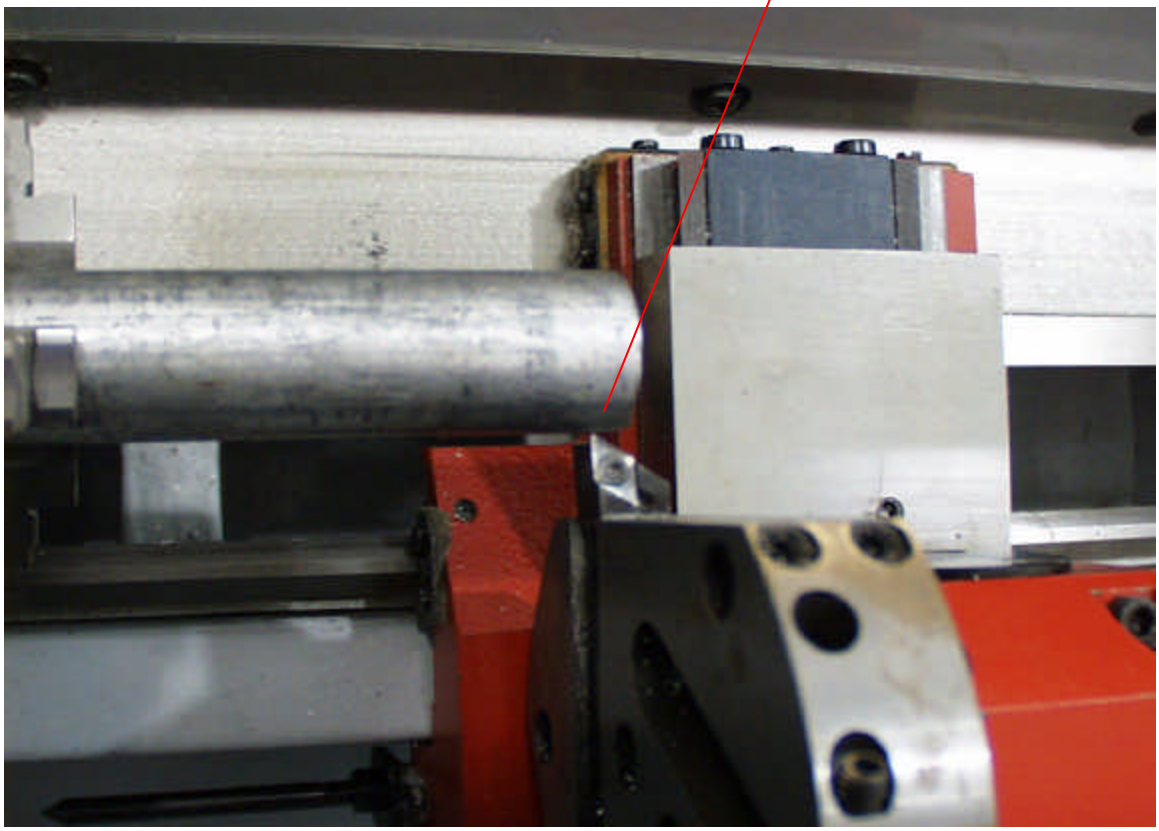
     1. Option for Scratching

        Type S1000 for RPM press Enter then Type M03 for spindle on clockwise press Enter

   - Then press Enter on the # keys (make sure door is closed)

2. Press F1 then F6 for JOG
3. Jog TOOL TIP to the WORK PIECE & touch TOOL TIP to the DIAMETER of the WORK PIECE using 2,4,6,8 on the # keys.

   (Use the Feed – , + or Steps to approach at a slower feed)

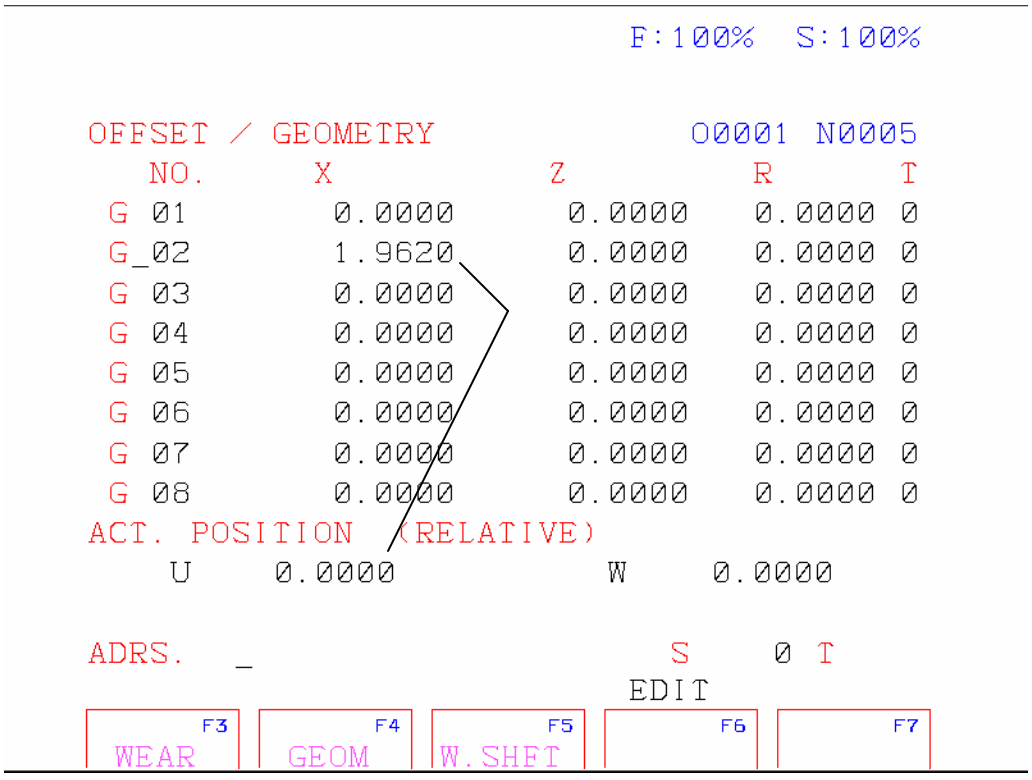


4. Press F12 then F5 for OFFSET then press F4 for Geometry
5. Take the value in Actual Position (Relative) U and subtract the Diameter of the Work Piece being scratched
6. Type value in G02 for X (If the tool being use is T0202)

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

7. Then press Enter
8. Jog TURRET away from WORK PIECE using 8 on # keys

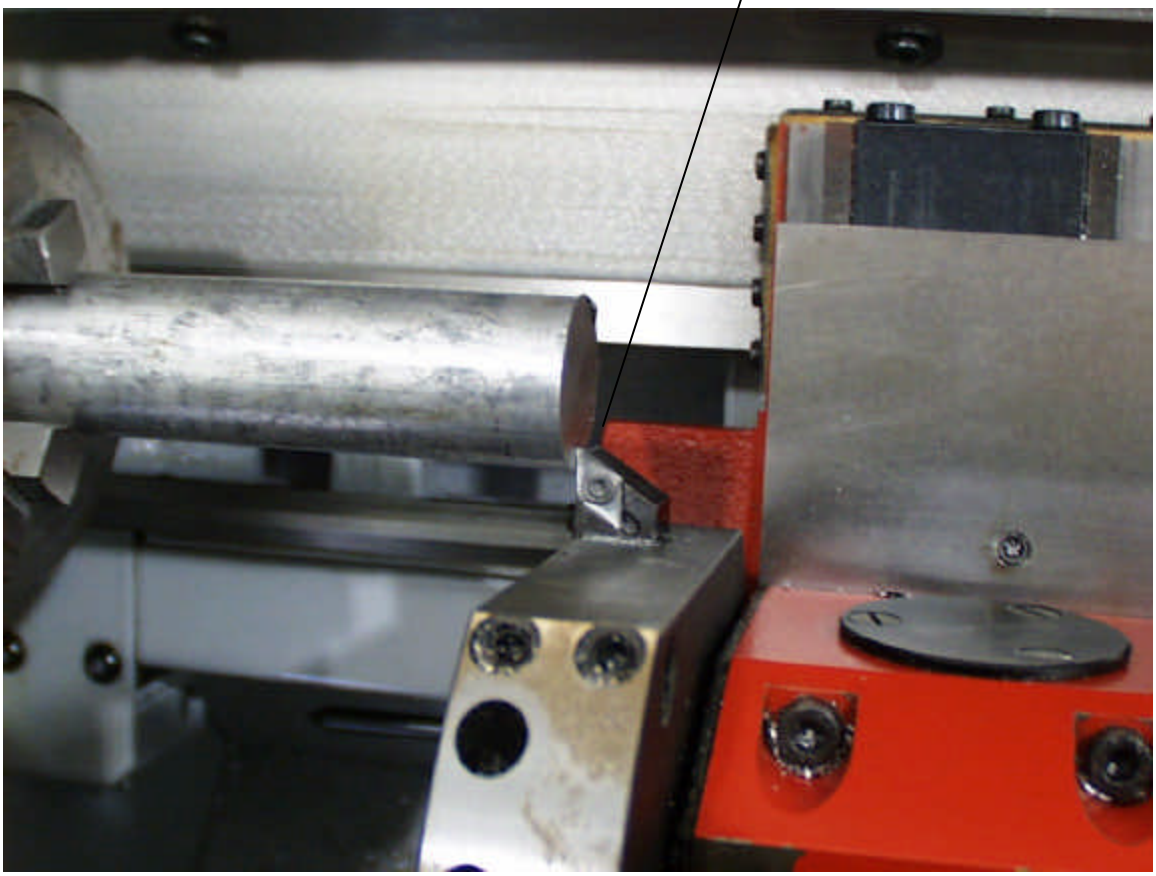
This value is the distance from an I.D. Tool Station to the Tool Tip



9. Jog TOOL TIP to the end of the WORK PIECE & touch TOOL TIP to the FACE of the WORK PIECE using 2,4,6,8 on the # keys.

(Use the Feed – , + or Steps to approach at a slower feed)

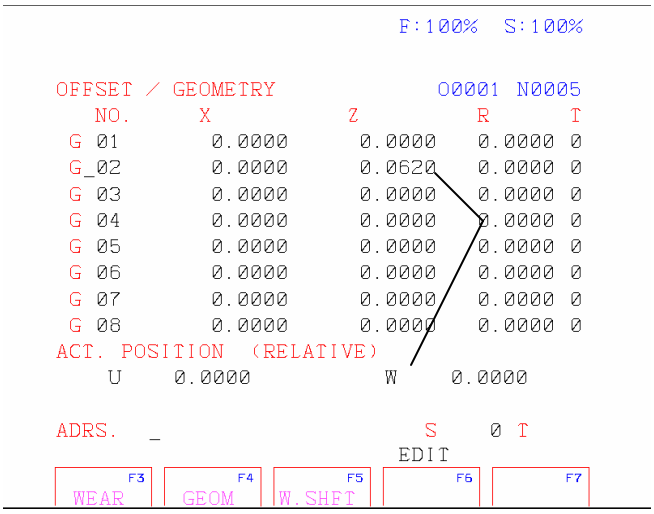
10. Press F12 then F5 for Offset then press F4 for geometry



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 .062 Type Z .062

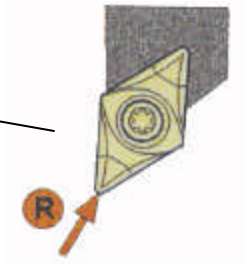
12. Then press Enter

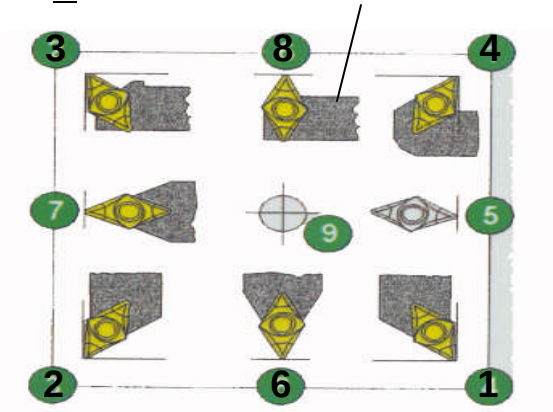


13. Jog TURRET away from WORK PIECE using 6 on the # keys

14. The R will be Tool Tip Radius

15. The T is the Tool Direction or Tool Type



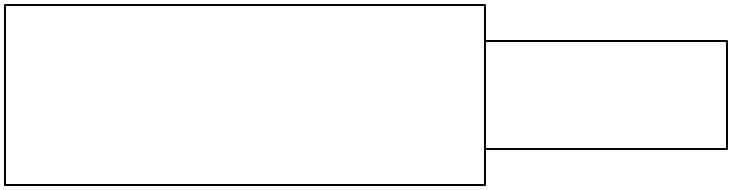


16. To set more OD tools repeat steps 1 thru 15

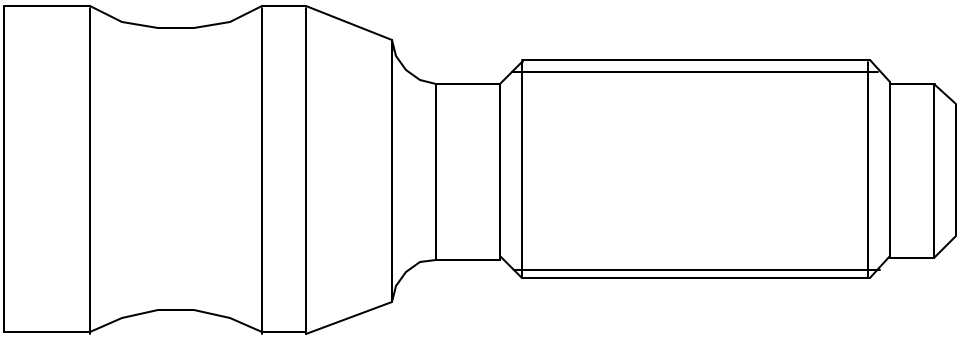
Note: The T is Direction that the Tool Points. Tool does not need to look like Tool in the Picture

Program Training

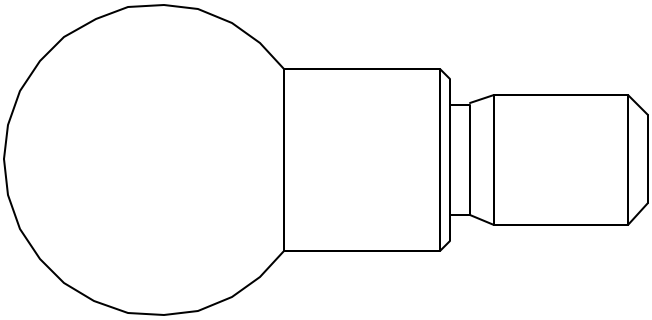
Program O0001



Program O0002



Program O0003



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

- **INSERT A NEW PROGRAM**
    1. Press letter O then program number
    2. Press Enter

    Example: O0001 OR O1
- **CALL A EXISTING PROGRAM UP**
    1. Press letter O then program number
    2. Press arrow pointing down
- **INSERT A WORD**
    1. Press letter then number
    2. Press Enter

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

    Example: _**N5 G01 X 0.25; G01** is the word being inserted
- **INSERT END OF BLOCK**
    1. Press (;)
    2. Press Enter
    3. Or press Enter 2 times

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

    Example: N5 G01 X1.00 F.003;

- **DELETE A PROGRAM**

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

    Example: **O0001 OR O1**

- **DELETE ALL PROGRAMS**

    1. Press letter o plus the – & 9999

    2. Press Delete

    Example: **O – 9999**

- **DELETE A WORD**

    1. Press letter then number
    2. Press Delete

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

    Example: BEFORE N5_S1000; AFTER N5;

    (S1000) is the word being deleted?

- **DELETE A BLOCK OR LINE NUMBER**

    1. Type the number line
    2. Press Delete

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

- **CANCEL MISTYPED WORD**

  1. Press Backspace

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

- **ALTER A WORD**

  1. Type the word needed altered
  2. Press Insert

  Example: Make sure the cursor is to the left of the words being altered (_N5 CHANGE TO _N10)

- **SEARCH FOR NUMBER BLOCK**

  1. Press letter n and the number of the block
  2. Press arrow pointing down

  Example:(N50)

- **SEARCH FOR WORD**

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

  2. Press arrow pointing down

- **SEARCH FOR LETTER**

  1. Press letter
  2. Press arrow pointing down

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

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

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

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

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 authorized

Used Addresses

C Chamfer

F Feed rate, thread pitch

G Path function

I, K Circle parameter

M Miscellaneous function

N Block number 1 to 9999

O Program number 1 to 9499

P Dwell, subroutine, cycle parameter

Q Cutting depth

R Radius, retraction, cycle parameter

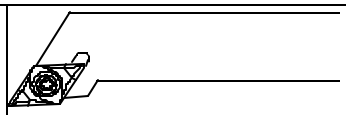
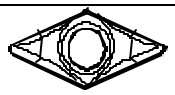
S Spindle speed

T Tool called out

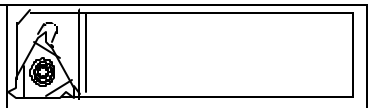
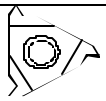
X, Z Position data in absolute

; Block end

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

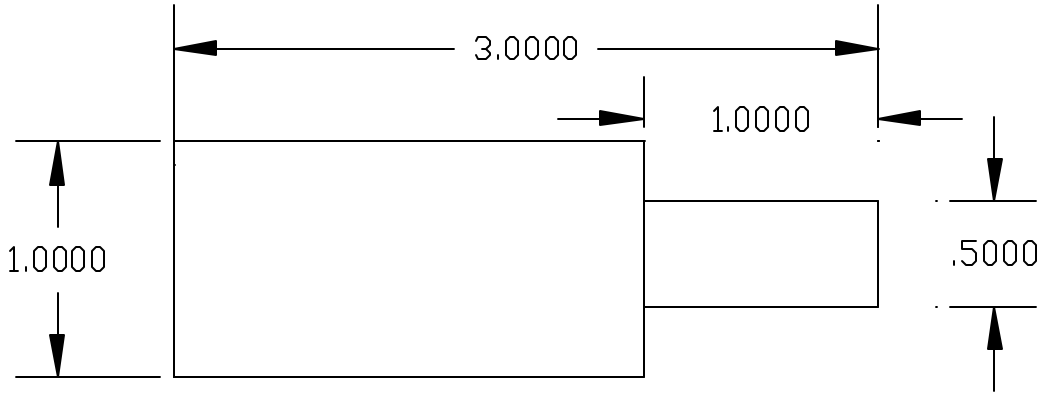
- **To edit / change a program / insert new programs & input or output excising programs & offsets**

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)

N5 (Demo 1) (3.25 x 1 alum)

N10 G40 G70 G80 G90

N15 G95 **G96** G98

N20 G0 Z2.0……..safe move

N25 T0202 S550 M3 (Right Hand 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

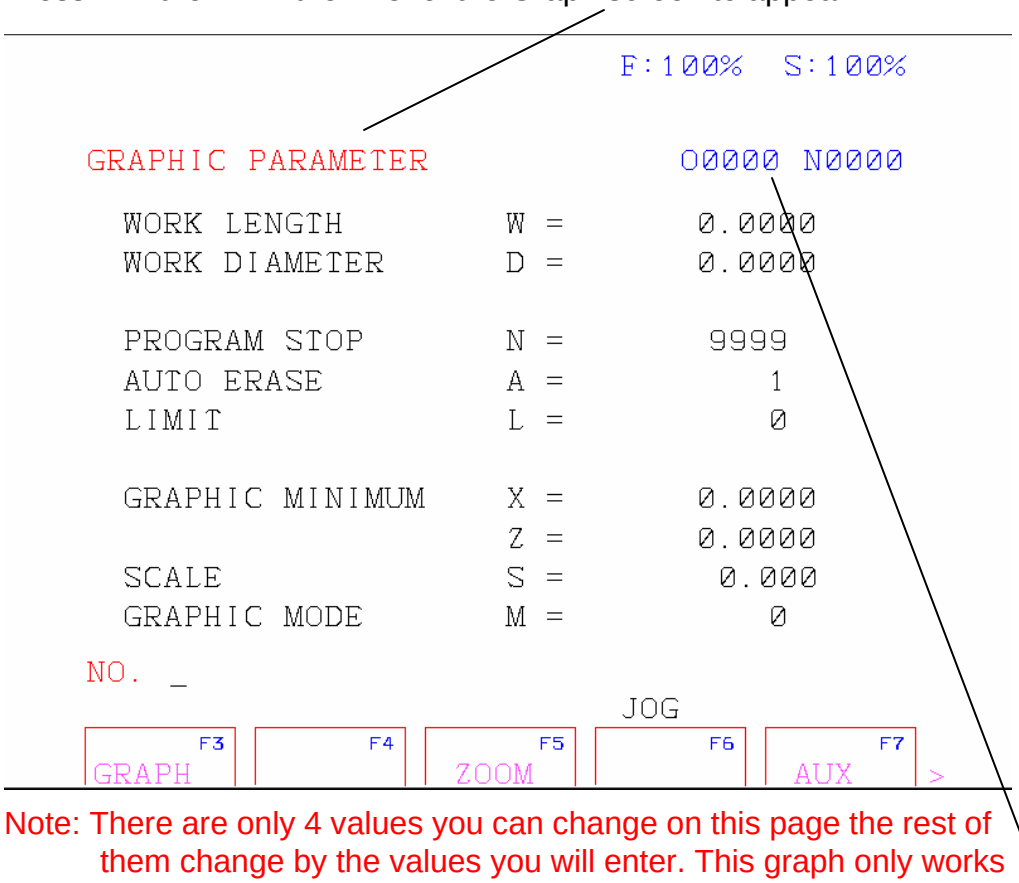
N65 X1.0……..diameter of contour

N70 G0 Z2.0……safe move

N75 M30……end of program

2D Simulation PC Keyboard

1. Press F12 then F11 then F3 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

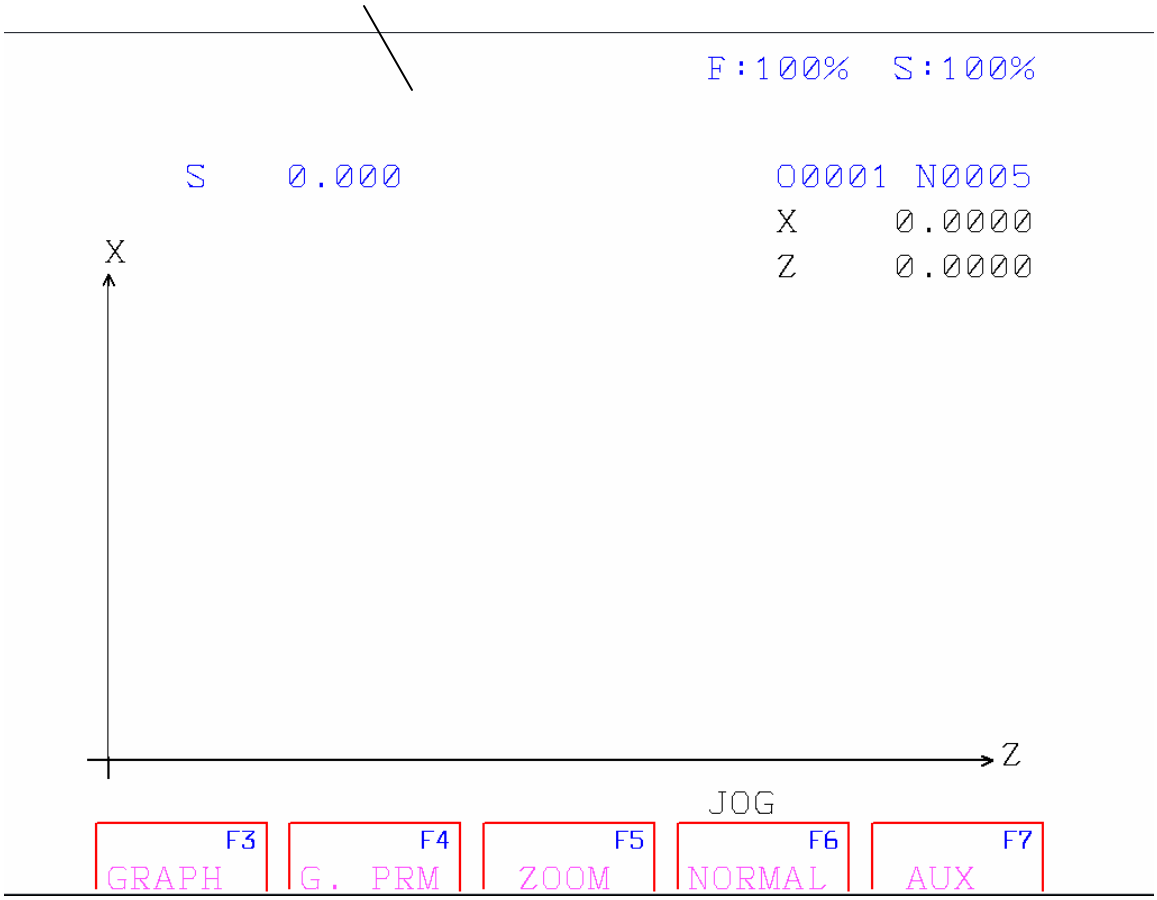
2. Work Length W = Overall length of stock in the Z direction this is a + value

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

4. Graphic 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 F4 for Simulation screen



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

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

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

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

    3. Press F9 for Output

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

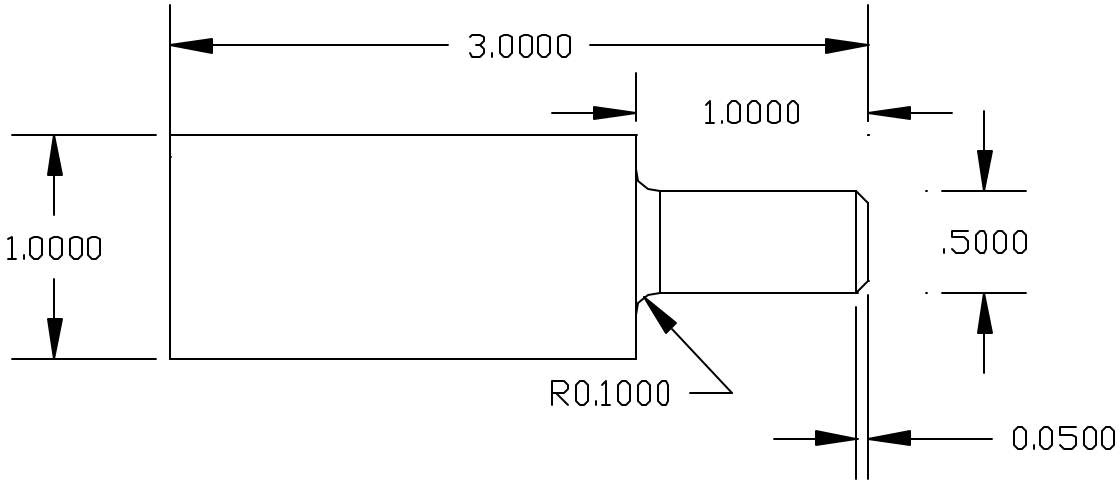
- **Input Program into Fanuc Software from Drive unit**
    1. Press F12 then F4 for **Program**
    2. Type program number to be read

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

    3. Press F10 for input

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

Program O0001 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)

N5 (Demo 1) (3.25 x 1 alum)

N10 G40 G70 G80 G90

N15 G95 **G96** G98

N20 G0 Z2.0……..safe move

N25 T0202 S550 M3 (Right Hand 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

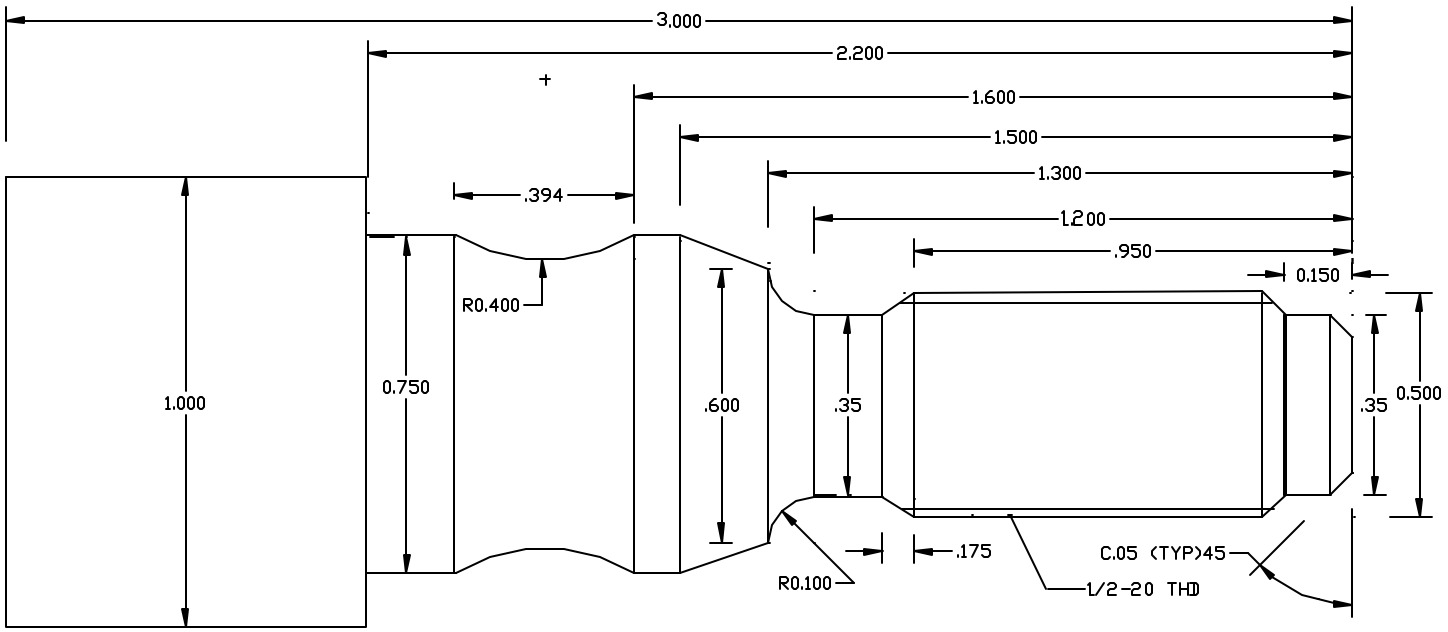
N60 Z-1.0 R.1……length of contour

N65 X1.0……..…….diameter of contour

N70 G0 Z2.0……safe move

N75 M30……end of program

Program O0002



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

$RPM = \frac{IPM}{PITCH} = \frac{28}{.05} = 560\ RPM$

30 is max IPM for 50 Machines
78 is max for a new 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 O0002

N5 (Demo 2) (Stock 3.25 x 1)
N10 G0 Z2
N15 G96 T0202 S550 M3 (Right Hand 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

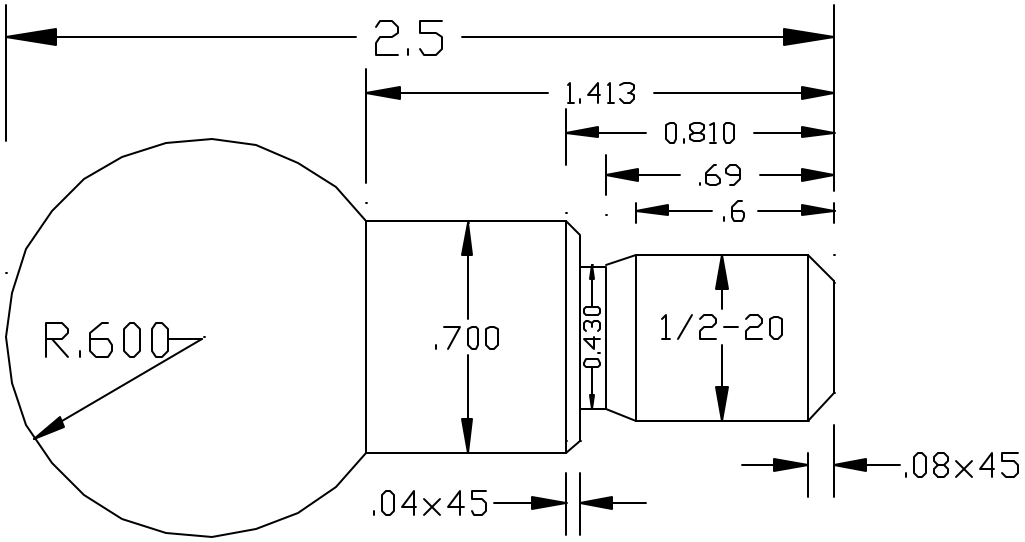
1. To make a program tie together use M98 this calls out Sub programs or Sub routines.
   Example: M98 P010001

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.

Program O0003



N5 (Ball Hitch) (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 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 O0004

N5 G96 (Back side of Ball Hitch)
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 machined