emco

innovative machine tools

GE FANUC 21
155/325II TCM
TRAINING GUIDE

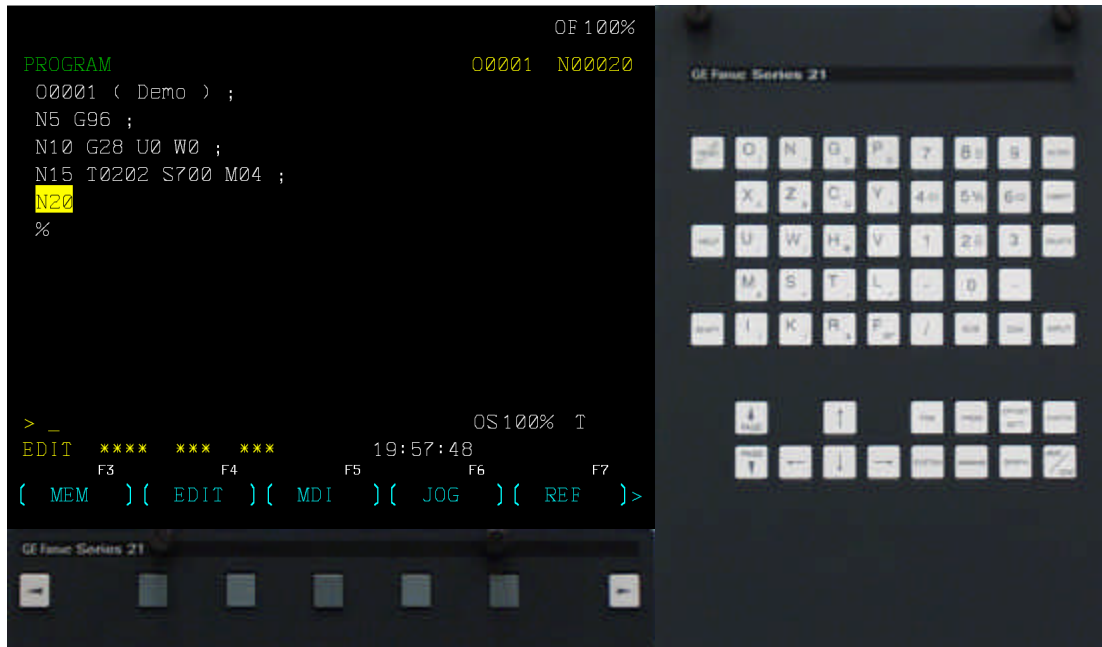# Training Index

FANUC 21 CONTROL

MACHINE CONTROL

The Fanuc 21 Screen

```
                                               OF 100%          1

 PROGRAM                          O0001   N00000               3

   O0001 ( Demo ) ;
   N5
   %








 > _
EDIT    ****    ***    ***           20:44:33
     F3          F4          F5          F6          F7
 ( BG-EDT )( O SRH  )( SRH   ↓ )( SRH   ↑ )( REWIND )>
```

4 → PROGRAM

5 → > _

6 → EDIT

2 → OS100% T

7
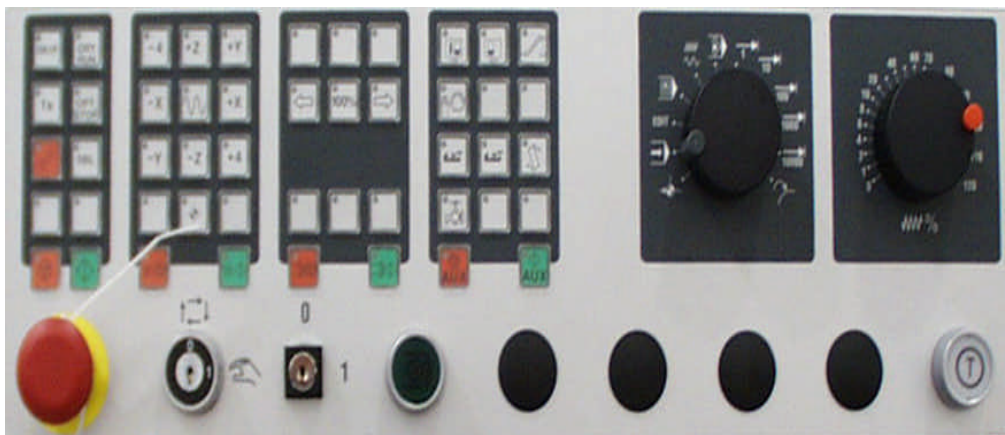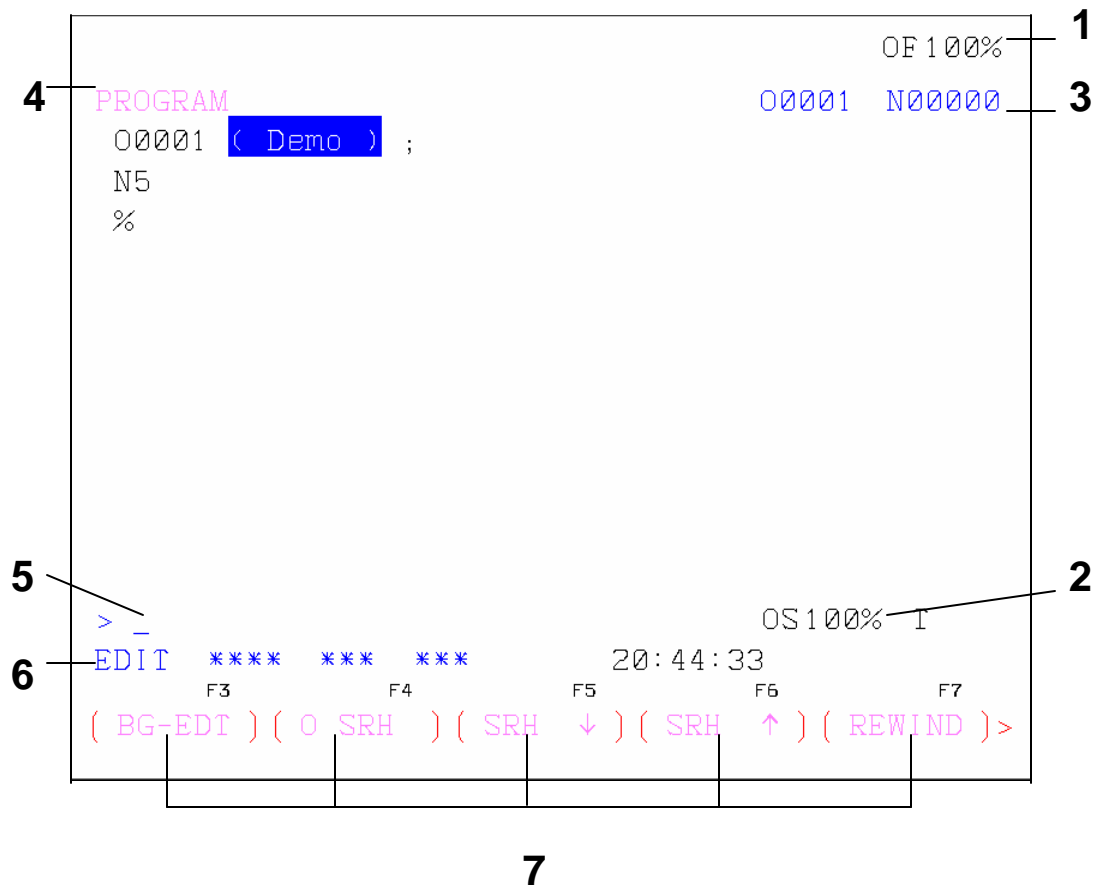
1. **Displays of Feed**

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 `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 = alter word (replace word)**

`INSRT` **INSRT = insert word, create new program**

`DELET` **DELET = deletes word / block or programs**

`/ , # EOB` **EOB = end of block**

`CAN` **CAN = deletes entries in the address one by one**

`INPUT` **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 | **POS = displays actual, relative, machine positions** |

| PROG | **PROG = displays program, library page** |

| OFFSET SETT. | **OFFSET/ SETTINGS = displays wear, geometry, work shifts pages** |

| SYSTEM | **SYSTEM = displays parameters, diagnostic pages** |

| MESSAGE | **MESSAGE = displays operator & alarm messages** |

| GRAPH | **AUX / GRAPH = displays 2–d graph simulation** |

SOFT KEYS

GE Fanuc Series 21
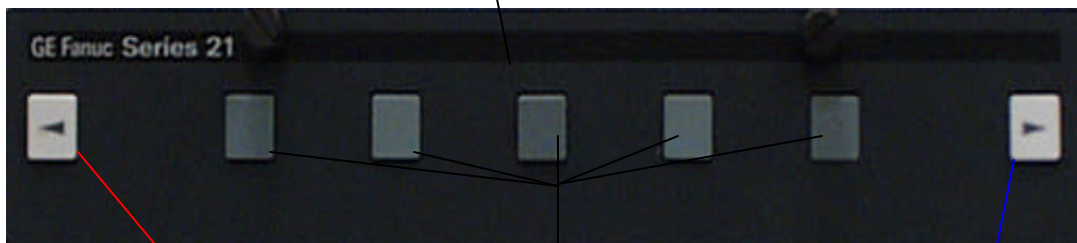
SCROLL BACK   **SOFT KEYS**   **PAGES OVER**

MACHINE KEYS

MACHINE FUNCTION KEYS

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

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

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

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

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

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

= **(Cycle stop) program hold, feed hold**

= **(Cycle start) program start**

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

= **(Mode Key) Automatic & Hand Mode**
Hand Mode is for moving machine around with door open and works in conjunction with the (Agreement button)

=**(Cycle start) program start**
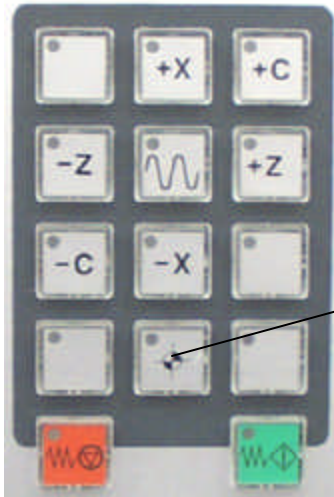
DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

Reference all 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)
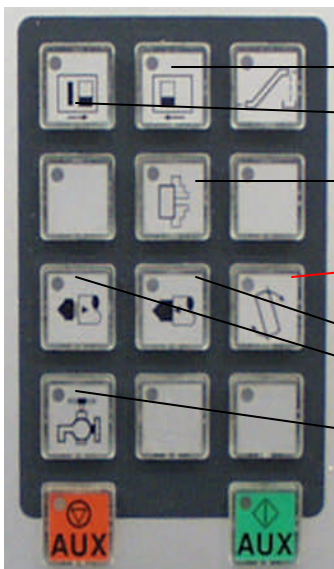
ACCESSORY FUCTIONS

Arrow right door closed
Arrow left door open

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
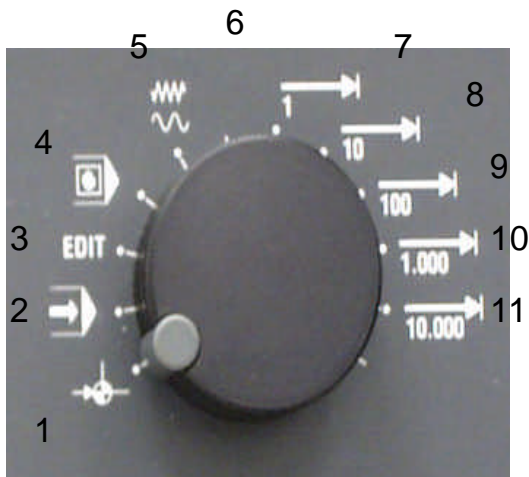
(1) **REF** = Zero / Reference or Home mode

(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**

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 ⓣ and hold then **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**"
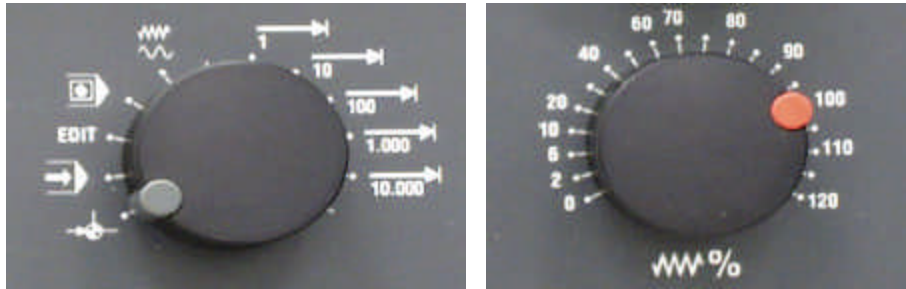
4. Press the X+ (arrow pointing up) this references the X axis.

5. Press the Z+ (arrow pointing left) this references the Z axis

6. For all axis press the Reference key
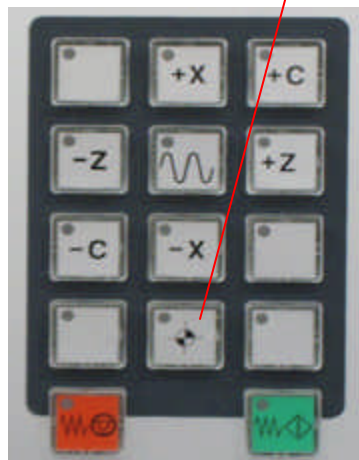
Note: Every time you enter Fanuc 21 Software or Turn the Machine On you must reference the axis

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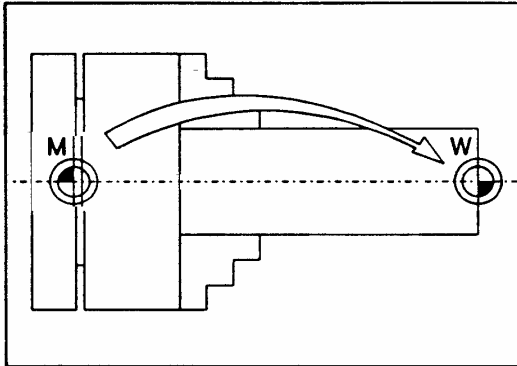
Zero offset from machine zero point M to workpiece zero point W

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,

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.

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.

Work Shift:

1. Move the MODE dial to JOG position

2. Jog the TURRET to the face of the Work

   Piece & touch using the Direction keys.

3. Use piece of paper between TURRET and Work Piece

   A. Turn the key to hand position

   B. Press the (T) and hold while pressing the direction keys
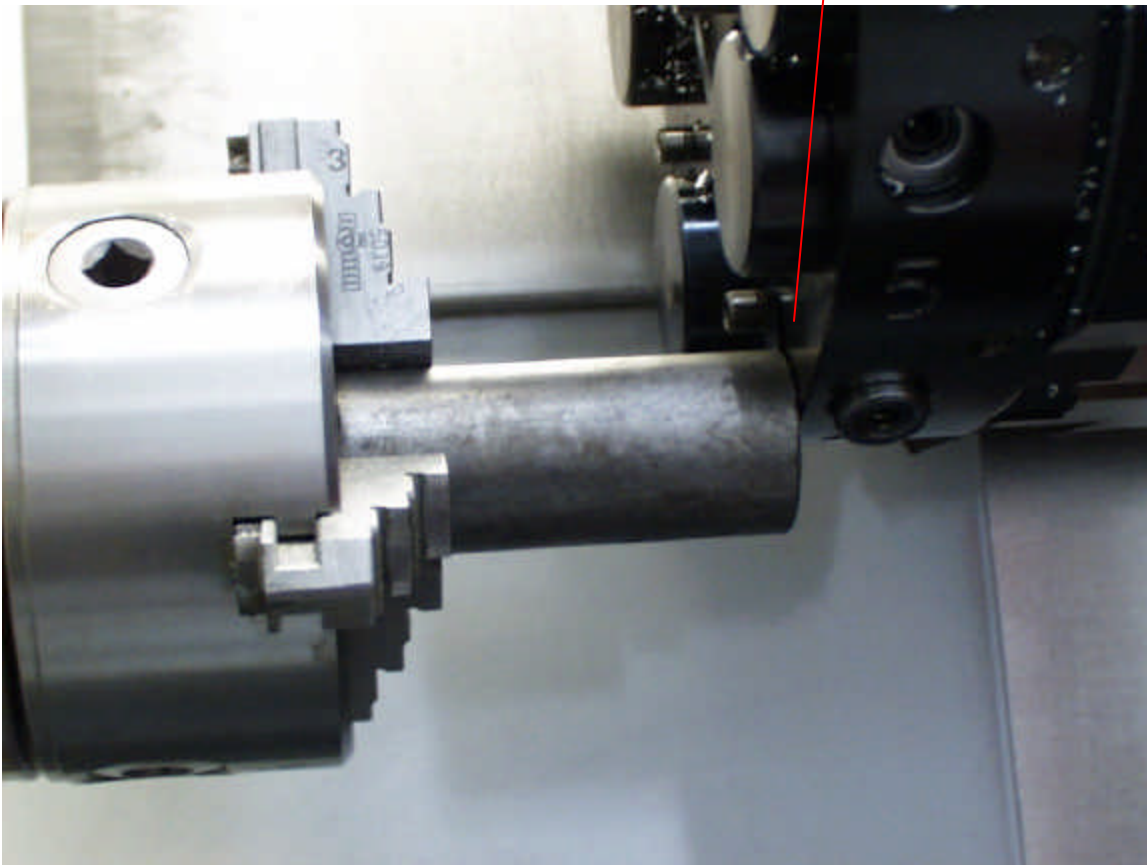
4. Press the OFFSET/SETT button until Work Shift page appears OFFSET SETT.

5. Make sure the (Shift value) Z is 0 if not highlight Z (Shift Value) type in 0 & press Input button 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 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

```
                                                OF 100%

 WORK SHIFT                              O0001  N00020

     (SHIFT VALUE)              (MEASUREMENT)
     X       0.0000             X       0.0000
     Z       0.0000             Z       0.0000




 ACTUAL  POSITION   (RELATIVE)
      X       0.0000              Z       0.0000

 >  _                                    OS 100%  T
 EDIT   ****   ***   ***        22:10:41
         F3           F4          F5         F6           F7
 [        ][ W.SHFT ][         ][          ][ (OPRT) ]>
```
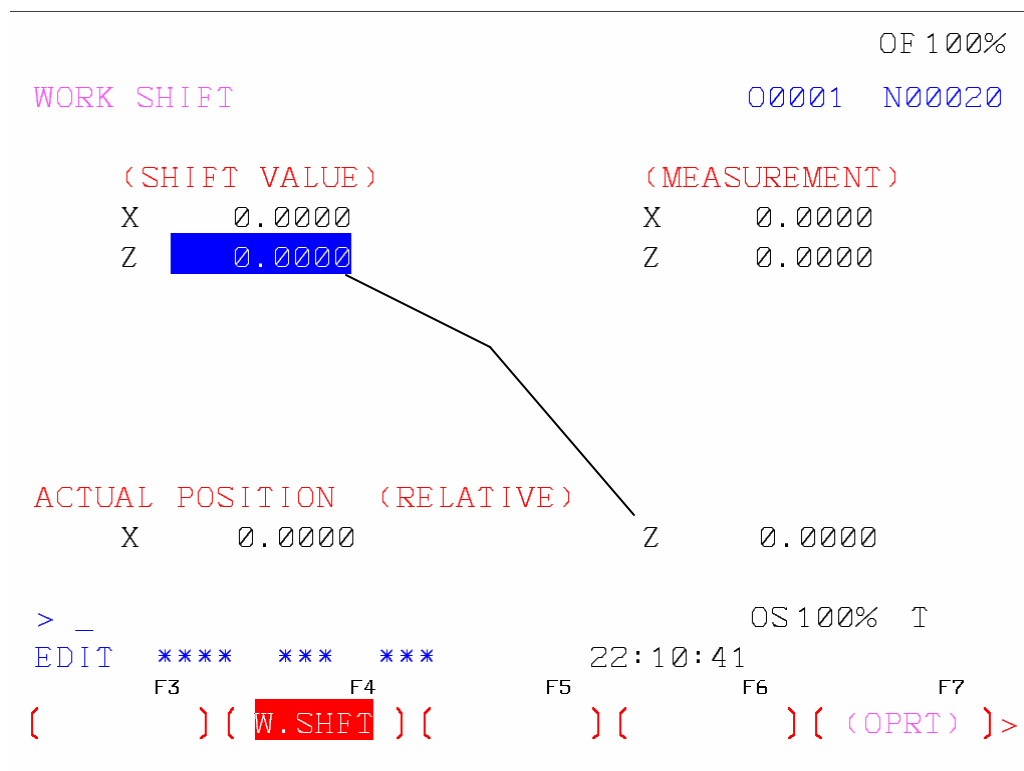
Note: Machine 0 is the turret face touching the spindle nose.

NEVER put a value in SHIFT VALUE X

TOOL OFFSETS
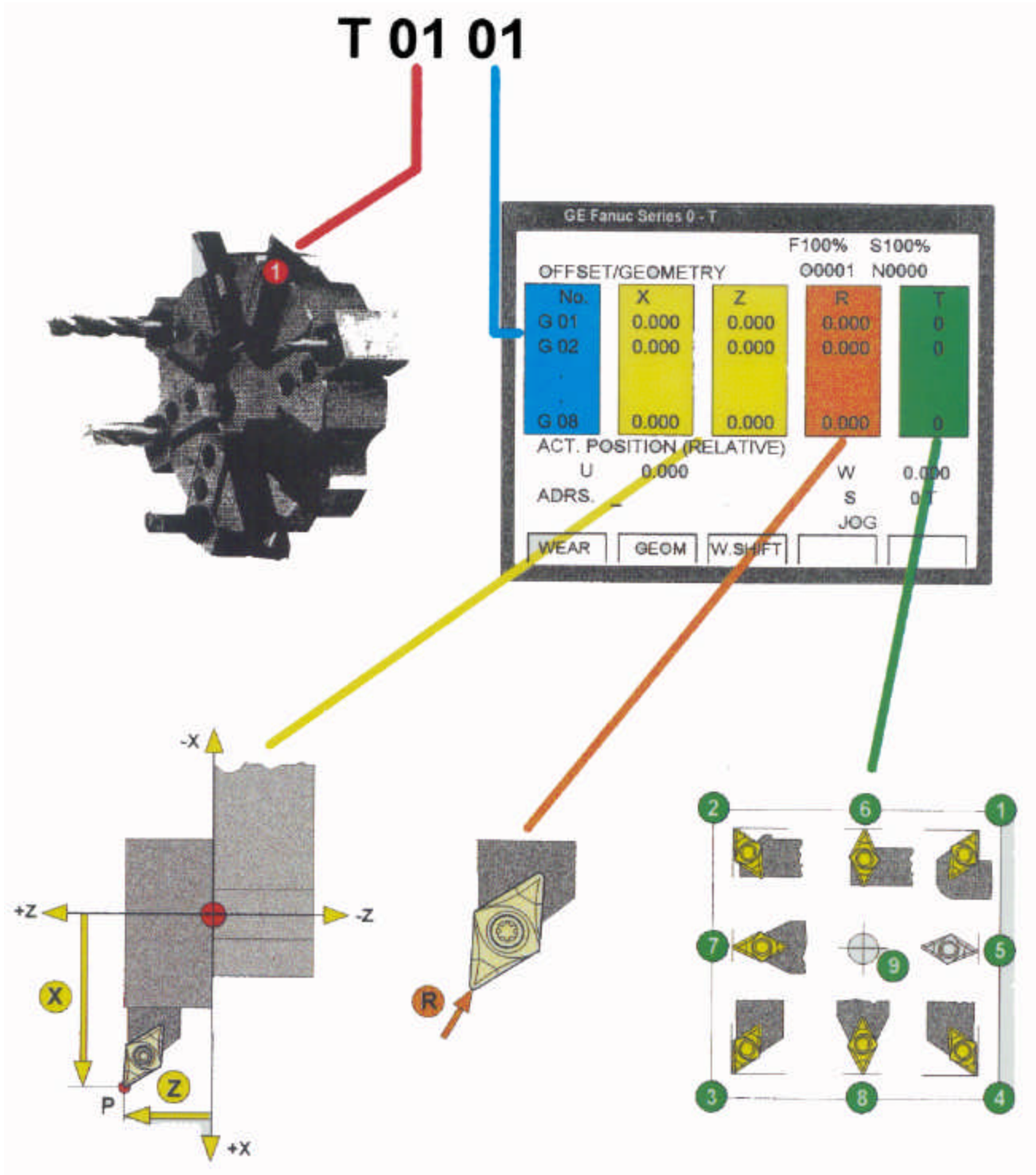
Tool Offsets

1. Index the TURRET to a tool to be measured

   - Move the MODE Dial to MDI position
   - Press Program button [PROG]
   - Type tool number then press INPUT button [INPUT]

     Example: T0200

     1. For Scratching type S1000 then M04 press INPUT button [INPUT]

        This is for Spindle speed & spindle on clockwise

2. Then press CYCLE START [icon] (make sure door is closed)
3. Move the MODE Dial to JOG position
4. Jog TOOL TIP (X axis) to the WORK PIECE & touch TOOL TIP to the DIAMETER of the WORK PIECE using the Direction keys.
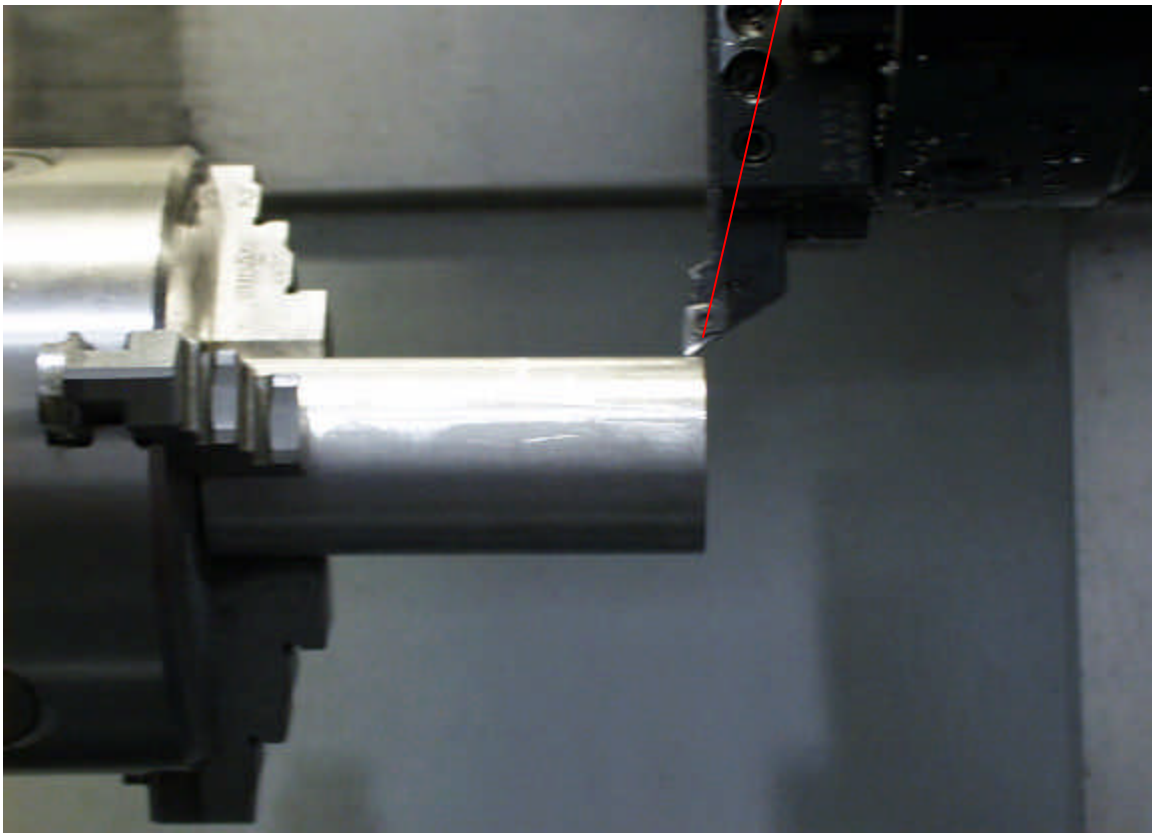
5. Press the OFFSET/SETT button until Offset / Geometry page appears `OFFSET SETT.`

6. Actual Position (Relative) X subtract the Diameter of the Work Piece being scratched

7. Type value in G02 for X (If the tool being use is T0202)

8. Then press INPUT `INPUT`

9. Jog TURRET away from WORK PIECE using X+

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

```
                                                 OF 100%
OFFSET ╱ GEOMETRY                     O0001  N00020
  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
EDIT   ****   ***   ***        22:15:51
       F3          F4          F5          F6          F7
[  WEAR  ] [  GEOM  ] [       ] [        ] [ (OPRT) ]
```

This value is the distance from the center of the Tool Station to the Tool Tip

10. Jog TOOL TIP (Z axis) to the end of the WORK PIECE & touch TOOL TIP to the FACE of the WORK PIECE using the Direction keys.

11. Press the OFFSET/SETT button until Offset / Geometry page appears OFFSET SETT.

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

13. Then press INPUT button

Example: Z is .062   Type .062

```
                                              OF 100%
   OFFSET / GEOMETRY                  O0001  N00020
     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
   EDIT  ****   ***   ***          22:18:30
         F3          F4          F5         F6         F7
   [  WEAR  ] [  GEOM  ] [       ] [       ] [ (OPRT) ]
```
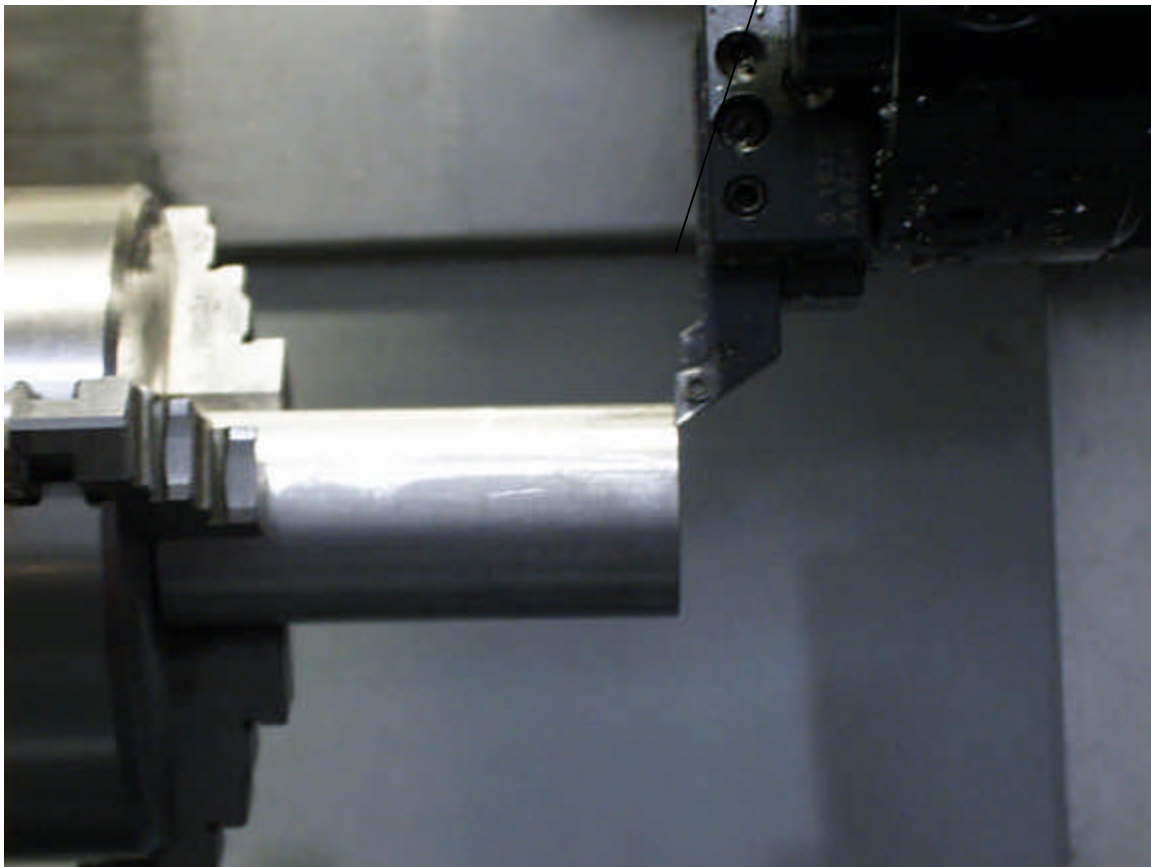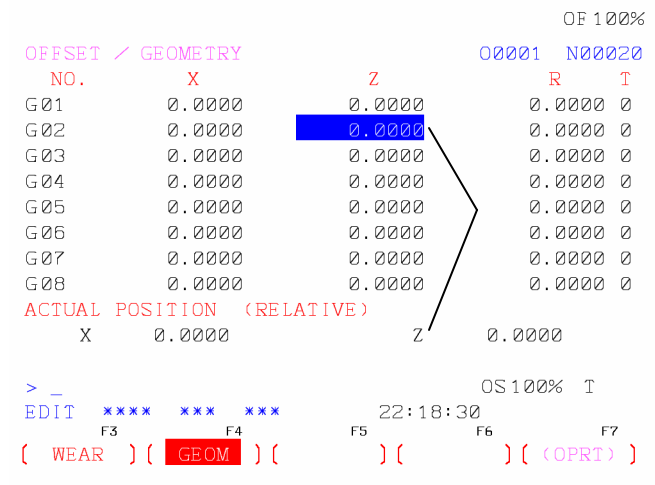
14. Jog TURRET away from WORK PIECE using Z+

15. The <u>R</u> will be Tool Tip Radius

16. The <u>T</u> is the Tool Direction or Tool Type
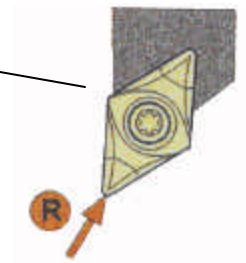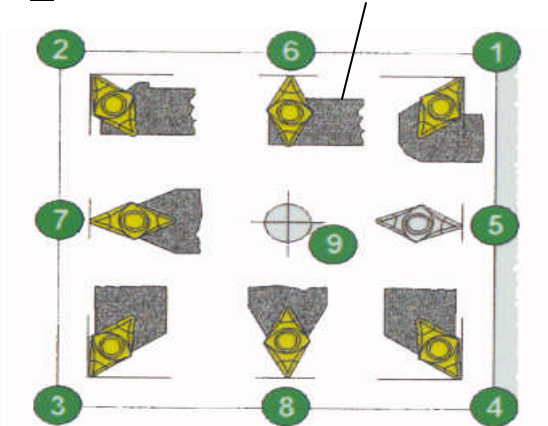
Note: Direction that the Tool is pointing.
        Tool doesn't need to look like Tool in the Picture

17. Repeat steps for all OD tools (STEPS 1-16 under Tool Offset)

Live Tools Offset

1. Index the TURRET to a Axial Tool

   - Move the MODE Dial to MDI position
   - To do this press Program button PROG
   - Type tool number then press INPUT button INPUT

     Example: T1100

2. Then press CYCLE START ◆ (make sure door is closed)

3. Move the MODE Dial to JOG position

4. Jog TOOL TIP (Z axis) to the end of the WORK PIECE & touch TOOL TIP to the FACE of the WORK PIECE using the Direction keys.
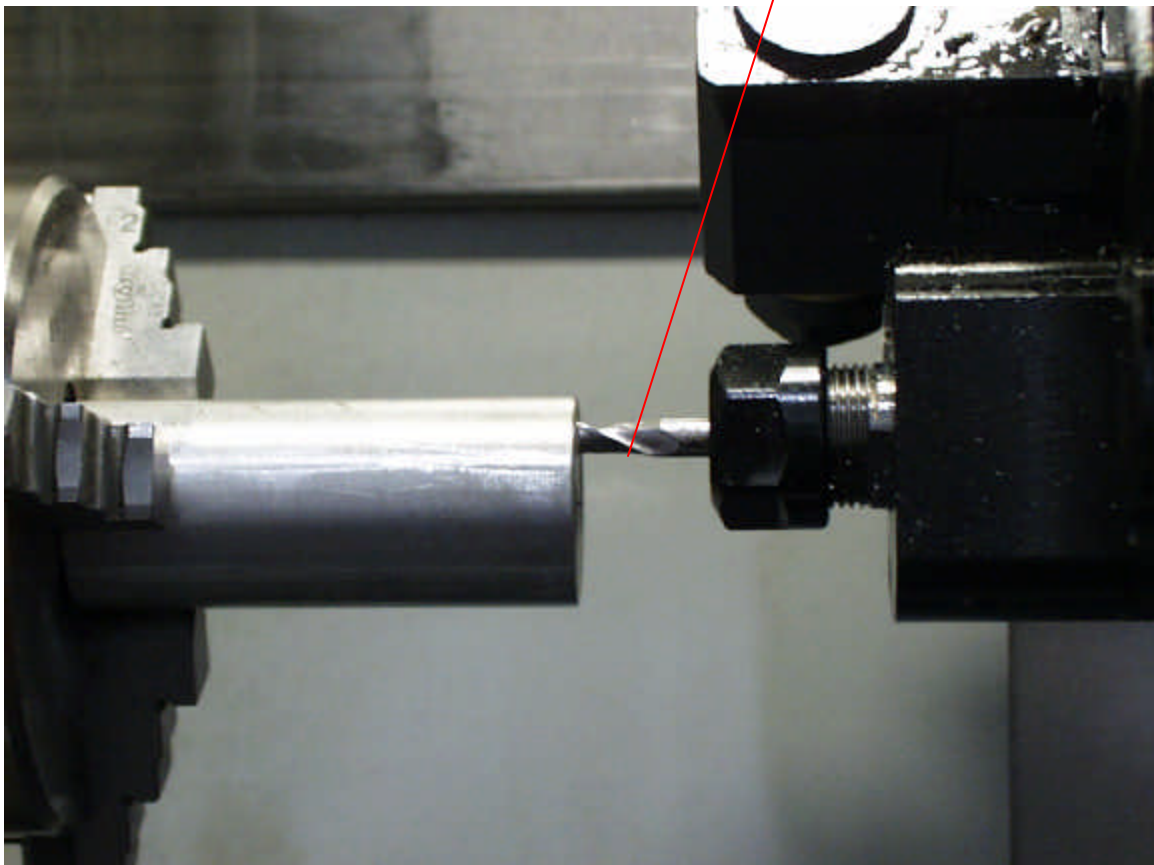
5. Press the OFFSET/SETT button until Offset / Geometry page

   appears ⬚OFFSET SETT.

6. The Value in the Actual Position (Relative) Z type this value in

   G11 for Z (If the tool being use is T1111)

7. Then press INPUT button ⬚INPUT

   Example: Z is 3.125   Type 3.125

```
                                                  OF 100%
     OFFSET / GEOMETRY                      O0001  N00020
        NO.          X                Z           R       T
     G09        0.0000          0.0000        0.0000  0
     G10        0.0000          0.0000        0.0000  0
     G11        0.0000          0.0000        0.0000  0
     G12        0.0000          0.0000        0.0000  0
     G13        0.0000          0.0000        0.0000  0
     G14        0.0000          0.0000        0.0000  0
     G15        0.0000          0.0000        0.0000  0
     G16        0.0000          0.0000        0.0000  0
     ACTUAL POSITION   (RELATIVE)
        X     0.0000               Z    0.0000

     > _                                  OS100%  T
     EDIT   ****   ***   ***        22:21:03
             F3            F4      F5         F6            F7
     [  WEAR  ][  GEOM  ][        ][        ][ (OPRT) ]
```

8. The X will be 20mm neg. (-.7874 inch) type this in then press
   INPUT button ⬚INPUT this is read as a Diameter value.
   (The live tool positions 1,3,5,7,9,11 are offset 10mm negative
   from the even positions 2,4,6,8,10,12.)

9. The R is the radius of the End Mill being used. If drill is used no
   radius is needed.

10. The T will always be 0 for live tools

18

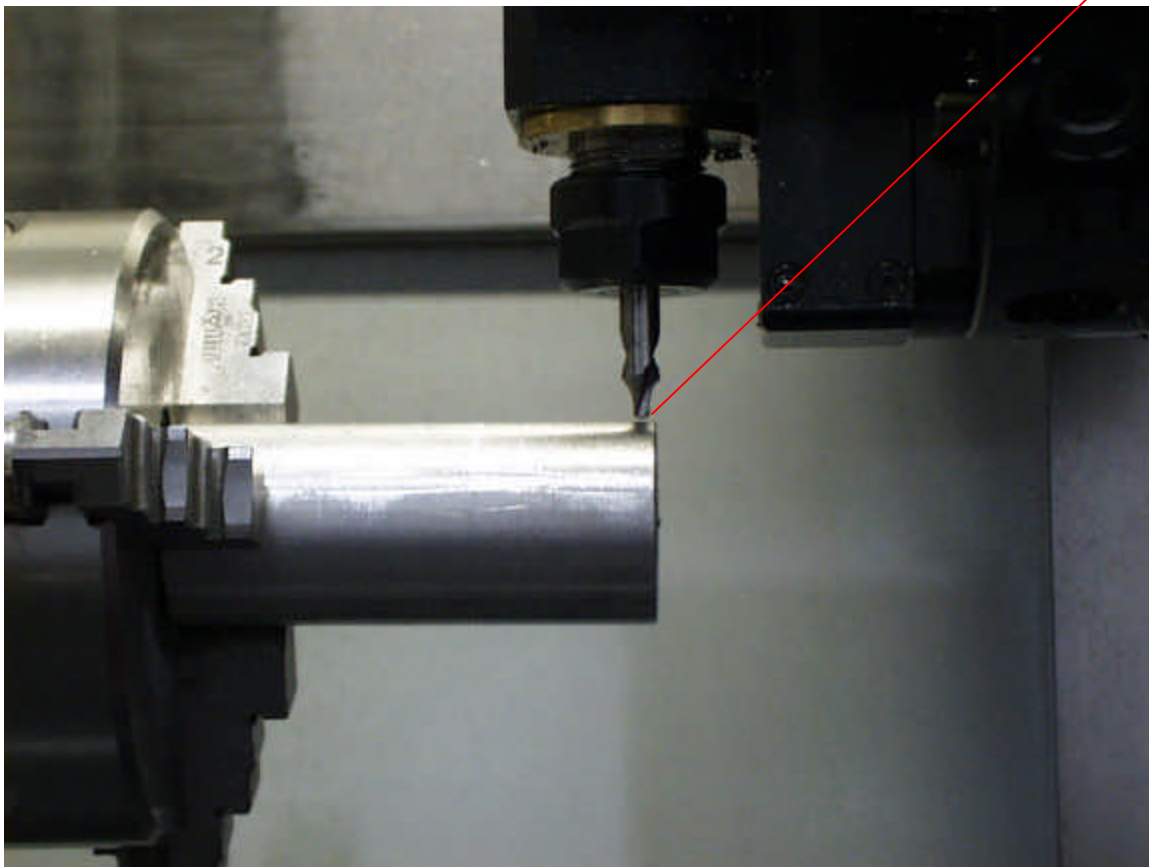1. Index the TURRET to a Radial Tool

   - Move the MODE Dial to MDI position

   - To do this press Program button PROG

   - Type tool number then press INPUT button INPUT

     Example: T0900

2. Then press CYCLE START (make sure door is closed)

3. Move the MODE Dial to JOG position

4. Jog TOOL TIP (X axis) to the WORK PIECE & touch TOOL TIP

   to the diameter of the WORK PIECE using the Direction keys.

5. Press the OFFSET/SETT button until Offset / Geometry page appears $\boxed{\substack{\text{OFFSET} \\ \text{SETT.}}}$

6. Actual Position (Relative) X subtract the Diameter of the Work Piece being scratched

7. Type value in G09 for X (If the tool being use is T0909)

8. Then press INPUT $\boxed{\text{INPUT}}$

9. Jog TURRET away from WORK PIECE using X+

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

```
                                                    OF 100%
OFFSET ╱ GEOMETRY                        O0001  N00020
   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
EDIT   ****   ***   ***          22:15:51
        F3          F4        F5          F6          F7
[  WEAR  ] [  GEOM  ] [        ] [        ] [ (OPRT) ]
```
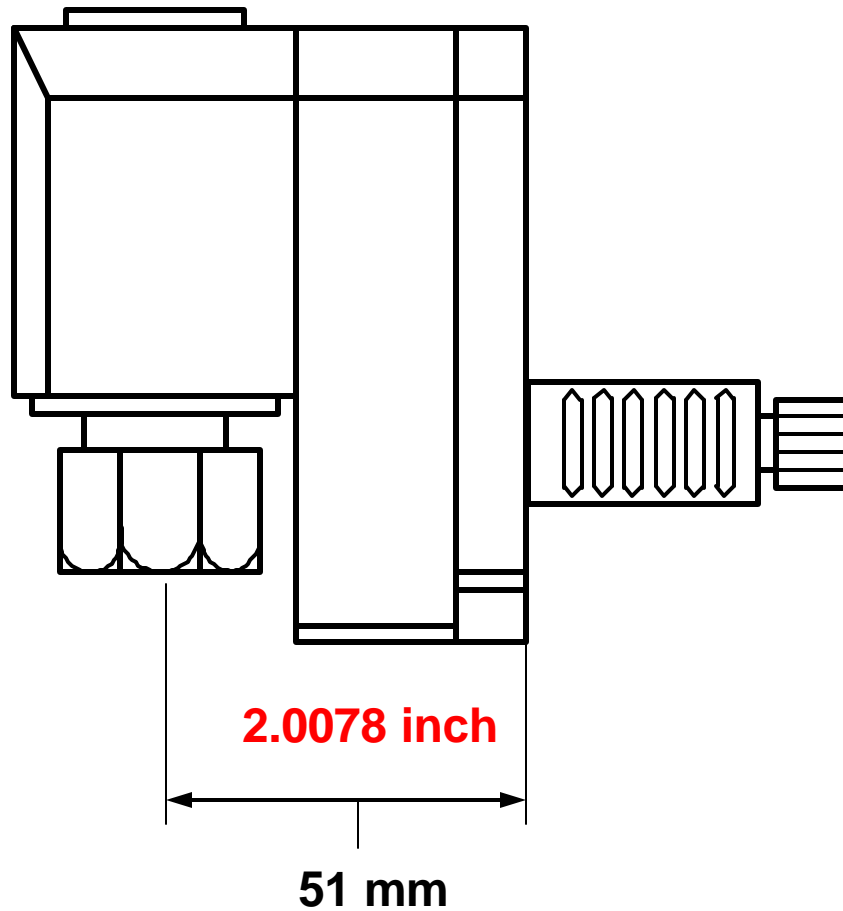
This value is the distance from the center of the Tool Station to the Tool Tip

10. For Z type in 2.0078 inch which is (51 mm)

11. The R is the radius of the End Mill that is used. If a Drill or Tap is used no radius is needed.
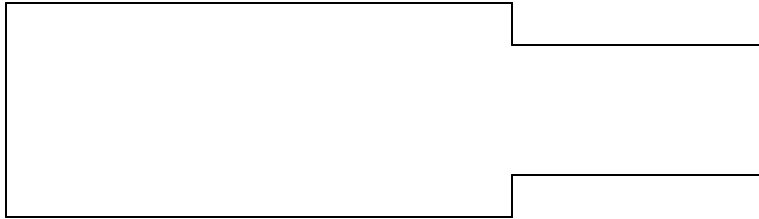
12. The T will always be 0 for live tools.

2.0078 inch

51 mm

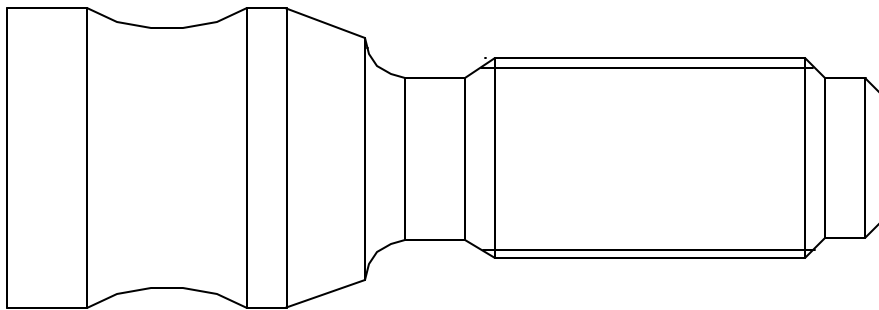**Note: Length is preset from tool manufacture (WTO), provided by EMCO Maier**
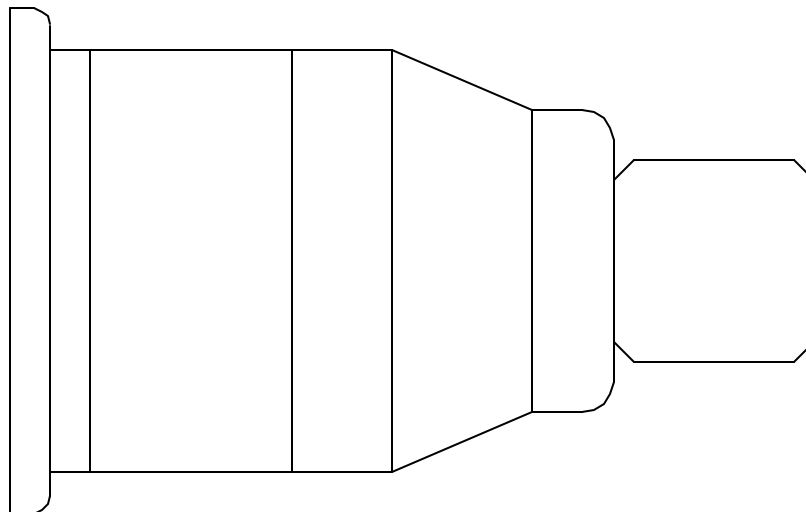
Program Training

Program O0001

Program O0002

## Program O0003

Change the Mode Dial to Edit & Press the [PROG] **to do functions below & on the next 2 Pages**

- ## INSERT A NEW PROGRAM

    1. Press letter o then program number
    2. Press insert button [INSRT]

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

- ## 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] or input [INPUT]

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

    ### Example: **N5** **G01 X 0.25**; **<u>G01</u>** is the word being inserted

- ## INSERT END OF BLOCK

    1. Press the (EOB) button [/ , # EOB]
    2. Press insert button [INSRT] or input [INPUT]

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

    ### Example: **N5 G01 X1.00 F.003;**

NOTE: IN EDIT & IN PROGRAM USE INSERT OR INPUT.

USE INPUT FOR ALL OTHER APPLICATIONS.

- ## 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. Highlight the Word
  2. Press delete button `DELET`

- ## DELETE A BLOCK OR LINE NUMBER

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

- ## CANCEL MISTYPED WORD (Backspace)

  1. Press cancel button `CAN`

     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

- **ALTER A WORD**

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

- **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)

Survey of commands G-GODES (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
G7.0   Cylindrical Interpolation OFF
G7.1   Cylindrical Interpolation
G10   Data Setting
G11   Data Setting Off
G12.1 Polar Coordinate Interpolation On
G13.1 Polar Coordinate Interpolation Off
G17   Plane Selection XY
G18   Plane Selection ZX
G19   Plane Selection YZ
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
G99   Return to withdrawal plane
Bold print = is the Default codes that are on at all times until changed

Survey of commands M- CODES: Mostly used
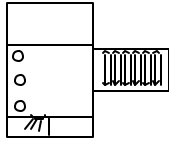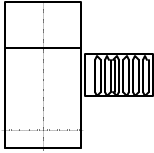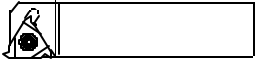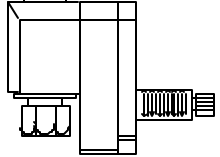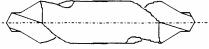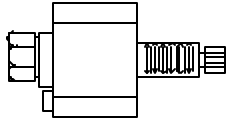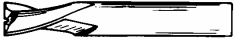
M00   Programmed stop unconditional
M02   Main program end, no new start of program
M03   Spindle ON clockwise
M04   Spindle ON counter clockwise
M05   **Spindle OFF**
M08   Coolant On
M09   Coolant Off
M13   Driven Tool On Clockwise
M14   Driven Tool On Counterclockwise
M15   Driven Tool Off
M20   Tailstock sleeve backward
M21   Tailstock sleeve forward
M23   Collection Tray Backward
M24   Collection Tray Forward
M25   Release clamping device
M26   Close clamping device
M30   Main program end with new start of program
M52   C-axis On
M53   C-axis 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

Need one of each tool and holder / collet's

Position	Part #	Discription	Picture
T0202	A8Z 240	Radial tool holder VDI 16, M4	
	271 050	Finishing tool left	
	271 056	Indexable inserts for aluminum	
T0404	A8Z 230	Radial tool holder VDI 16, M3	
	271 110	OD-threading tool right	
	271 115	Indexable inserts for OD-threading, right	
T0909	A8Z 480	Radial milling holder VDI 16	
	152 760	∅.236" ESX 16 COLLET (6.0mm)	
	573 770	Center drill, HSS (6mm)	
T1111	A8Z 470	Axial milling holder VDI 16	
	152 760	∅.236" ESX 16 COLLET (6.0mm)	
	764 304	ø6 mm / shank- ø6 mm	

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 <u>O0001</u>
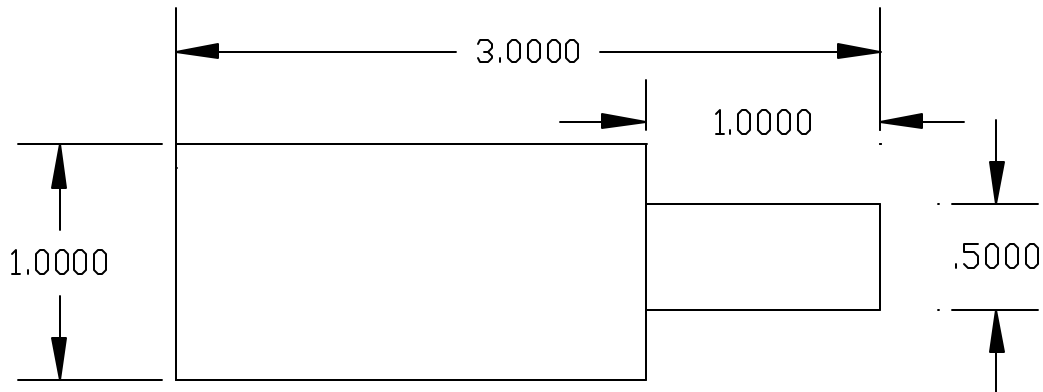
G73   U = Depth of Cut   R = Retract Value

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……………………active codes

N15 G95 **G96** G98…………………………active codes

N20 G0 G28 U0 W0………………………..safe move

N25 T0202 S700 M4…….……………….(Right Hand Finish Tool 55°)

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

N35 G73 U.06 R.02………………………..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 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                      O0001  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   ****   ***   ***          20:21:13
          F3              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

30

6. Press the Soft key Graph for Simulation screen

```
                                                      OF 100%

         S          0.000                    O0001   N00000
                                               X      0.0000
   X                                           Z      0.0000












                                                      →Z
                                              OS 100%  T
   JOG    ****    ***    ***        20:36:48
          F3             F4         F5        F6          F7
 [ G.PRM  ] [           ] [ GRAPH  ] [        ] [          ] >
```
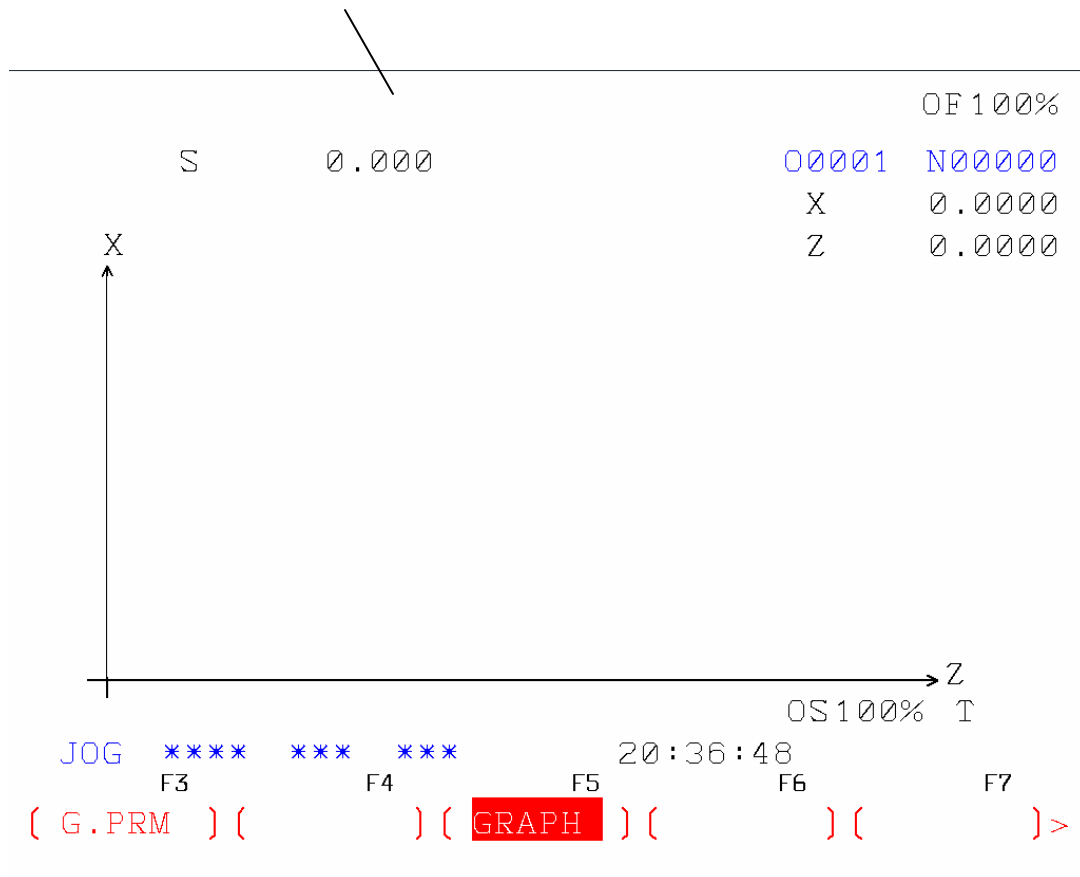
7. Now press Cycle start and you will see the tool movements of the program

- **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)

- **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

Program <u>O</u>0001 using C/R's
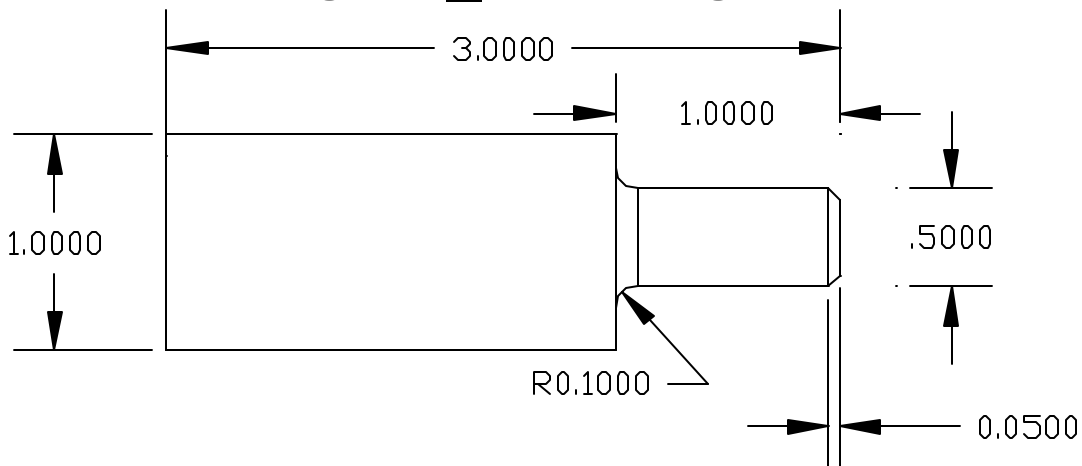
G73   U = Depth of Cut   R = Retract Value

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)

O0001 (Demo 1)

N5 (3.25 x 1 alum)

N10 G40 G70 G80 G90

N15 G95 **G96** G98

N20 G0 G28 U0 W0.……………….…..safe move

N25 T0202 S700 M4 (Left Hand Finish Tool 55°)

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

N35 G73 U.06 R.02………………………..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 G28 U0 W0………………….……safe move

N75 M30………………………….………end of program

Program <u>O</u>0002
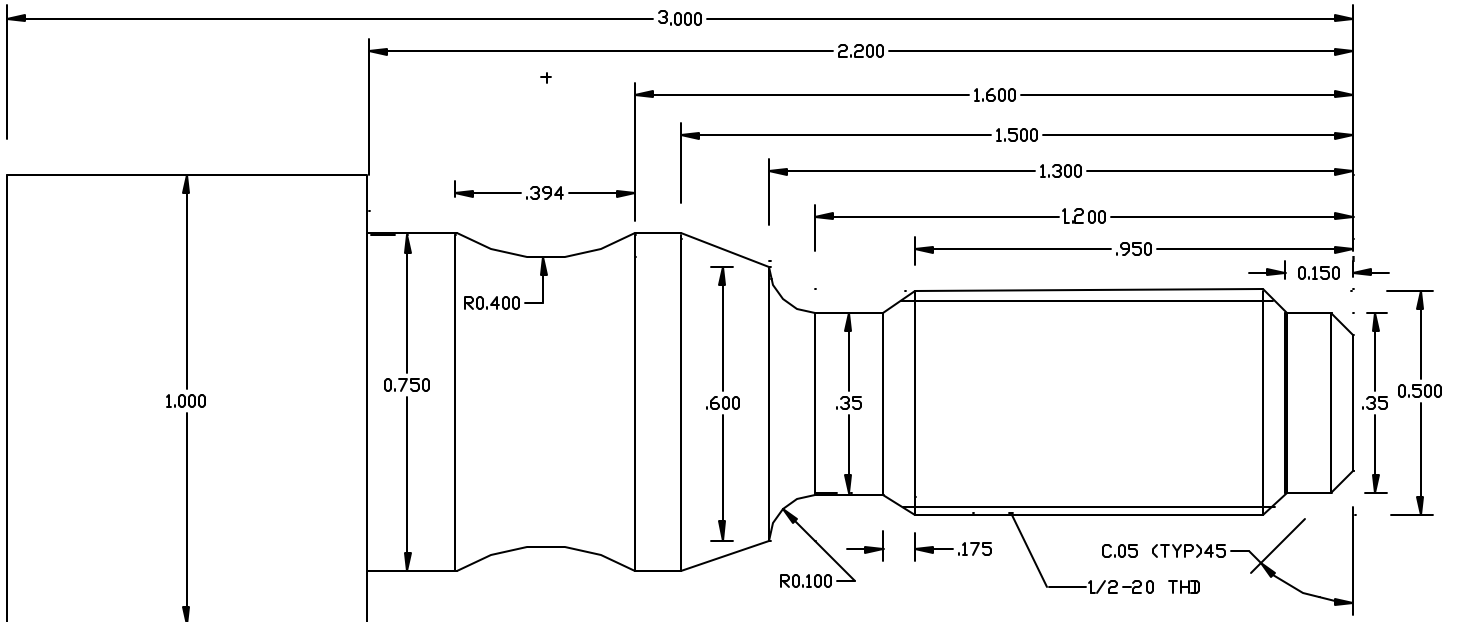
G73   **U** = Depth of Cut   **R** = Retract Value

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

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

IPM = RPM X PITCH

$$RPM = \frac{IPM}{PITCH} = \frac{50}{.05} = 1000 \ RPM$$
                            300 is max for a 155 machine
                            315 is max for a 325II 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

O0002 (Demo 2)
N5 (Stock 3.25 x 1)
N10 G0 G28 U0 W0
N15 G96 T0202 S700 M4 (Left 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.06 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 S1000 F.002
N130 G72 P50 Q120
N135 G0 G28 U0 W0………………………Safe Index Pos
N140 G97 S1000 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 G28 U0 W0...………………………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 <u>O</u>
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
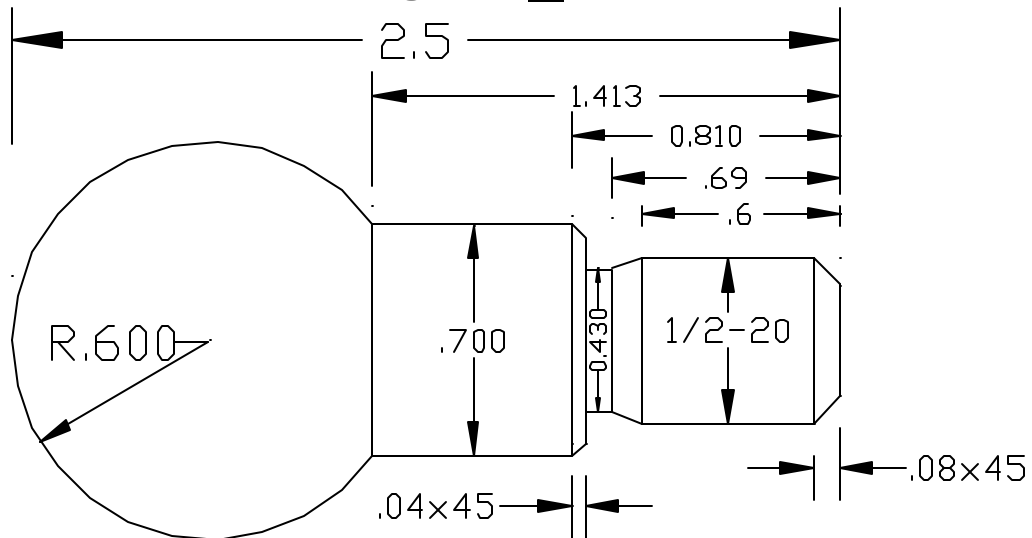
O0003 (Ball Hitch)
N5 (Stock 2.5625 x 1.25)
N10 G0 G28 U0 W0
N15 G96 T0202 S700 M4 **(Left Hand Finish Tool 55°)**
N20 G0 Z.1
N25 Z0
N30 G1 X-.02 F.003
N35 G0 X1.25 Z.1
N40 G73 U.06 R.02
N45 G73 P50 Q95 U.01 W.005 F.004
N50 G0 G42 X.24
N55 G1 Z0
N60 X.5 C.08
N65 Z-.6
N70 X.43 Z-.69
N75 Z-.770
N80 X.7 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 S1000 F.002
N110 G72 P50 Q100

N115 G0 G28 U0 W0
N120 G97 S1000 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 G28 U0 W0
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 <u>O0004</u>

O0004 (Back side of 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 S700 M4 (Left Hand Finish Tool 55°)
N20 X1.25 Z.200
N25 G73 U.06 R.02
N30 G73 P35 Q55 U.01 W.005 F.004
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 S1000 F.002
N70 G72 P35 Q60
N75 G28 U0 W0
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

Program O0005
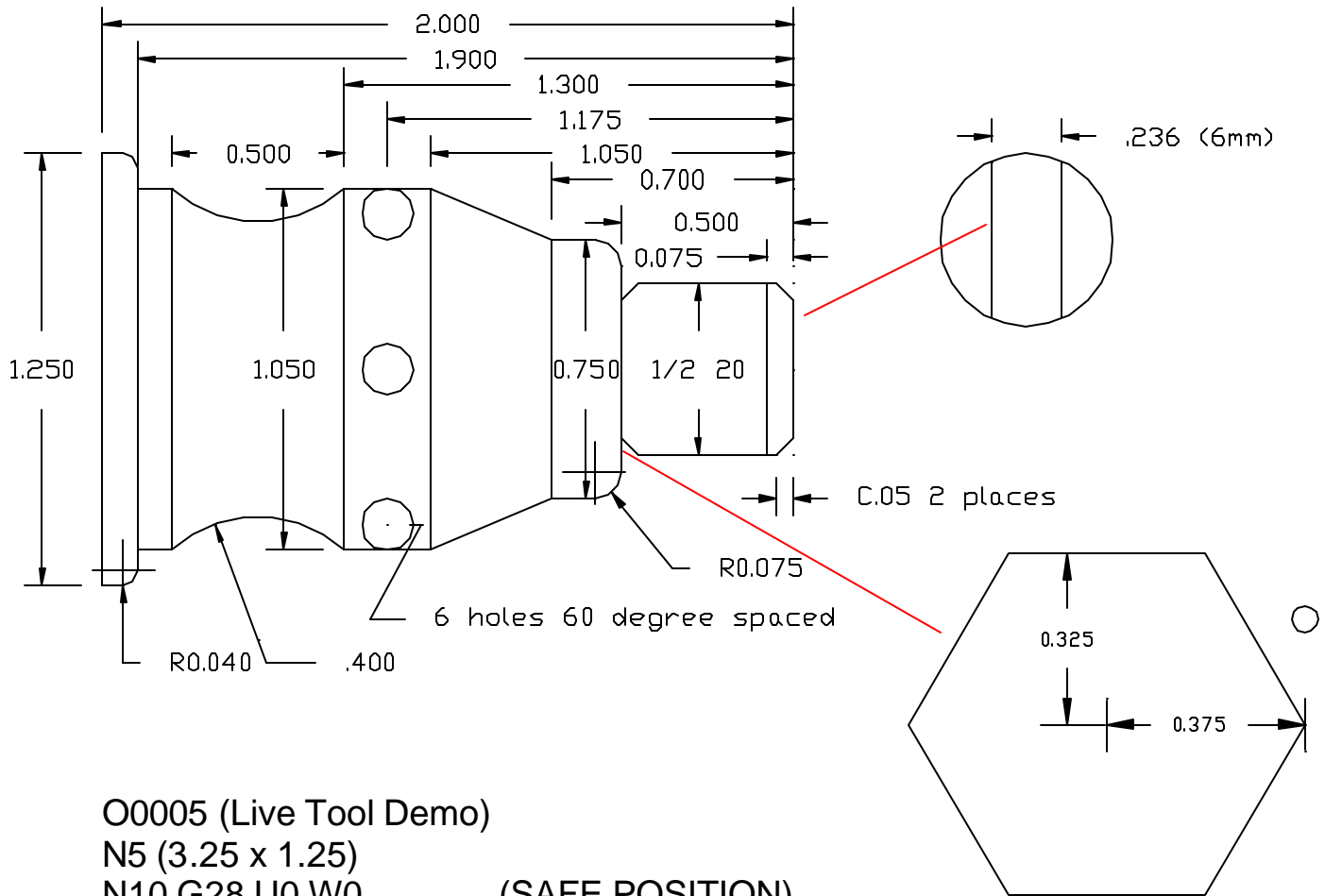
O0005 (Live Tool Demo)
N5 (3.25 x 1.25)
N10 G28 U0 W0   (SAFE POSITION)
N15 T0202   (FINISHING TOOL LEFT HAND)
N20 G96 S700 M4
N25 X1.3 Z0   (FACING)
N30 G1 X-.02 F.003
N35 G0 X1.25 Z.1
N40 G73 U.1 R.05   (ROUGHING CYCLE)
N45 G73 P50 Q110 U.01 W.005 F.004
N50 G0 G42 X.3
N55 G1 Z0
N60 G1 X.5 C.05
N65 Z-.4
N70 X.45 Z-.5
N75 X.75 R.075
N80 Z-.7

N85 X1.05 Z-1.1
N90 Z-1.3
N95 G2 X1.05 Z-1.8 R.400
N100 G1 Z-1.9
N105 X1.25 R.04
N110 Z-2.0
N115 G0 G40 X1.26
N120 S900 F.002
N125 G72 P50 Q115   (FINISHING CYCLE)
N130 G28 U0 W0   (SAFE INDEX MOVE)
N135 G97 T0404 S1000 M3   (THREADING TOOL RIGHT HAND)
N140 G0 X.55 Z.2
N145 G78 P010060 Q0020 R.001   (THREADING CYCLE)
N150 G78 X.414 Z-.45 P0330 Q0100 F.05
N155 G28 U0 W0

USING THE C-AXIS AS A Y-AXIS

N160 M5
N165 T1111   (6 mm END MILL in AXIAL HOLDER)
N170 M52   (TURN ON C AXIS)
N175 M13   (LIVE SPINDLE ON CLOCKWISE)
N180 G97 S1500   (DIRECT RPM)
N185 G28 C0   (REFERENCE C AXIS)
N190 G12.1   (PCI ON)
N195 G0 X1.3 Z-.7 C0   (SAFE MOVE)
N200 G1 G41 X.750 C.2 F.008   (CRC ON)
N205 G1 C0 F.004   (POSITION 1)
N210 X.375 C-.325   (POSITION 2)
N215 X-.375   (POSITION 3)
N220 X-.750 C0   (POSITION 4)
N225 X-.375 C.325   (POSITION 5)
N230 X.375   (POSITION 6)
N235 X.750 C0   (POSITION 7)
N240 C-.200   (SAFE MOVE)
N245 G1 G40 X.9 C-.4   (CRC OFF)
N250 G13.1   (PCI OFF)
N255 G0 X.9 Z-.25 M15   (LIVE SPINDLE OFF)
N260 M53   (C-AXIS OFF)

USING THE C-AXIS AS DEGREES

```
N265 M52                        (C-AXIS ON)
N270 M13                        (LIVE SPINDLE CLOCKWISE)
N275 G97 S1500                  (DIRECT RPM)
N280 G28 C0                     (REFERENCE C-AXIS)
N285 G0 X.8                     (MAKING SLOT ON THE FRONT OF PART)
N290 C90                        (TURN C IN DEGREES)
N295 G1 X0 F.003
N300 G0 C270                    (TURN C IN DEGREES)
N305 G1 X.8
N310 G28 U0 W0
N315 T0909                      (CENTER DRILL IN RADIAL HOLDER)
N320 G97 S1800                  (LIVE SPINDLE CLOCKWISE)
N325 G0 X1.1                    (MAKING DRILLED HOLES AROUND PART)
N330 Z-1.2
N335 C60                        (DEGREES)
N340 G1 X.65 F.003
N345 G0 X1.1
N350 C120                       (DEGREES)
N355 G1 X.65
N360 G0 X1.1
N365 C180                       (DEGREES)
N370 G1 X.65
N375 G0 X1.1
N380 C240                       (DEGREES)
N385 G1 X.65
N390 G0 X1.1
N395 C300                       (DEGREES)
N400 G1 X.65
N405 G0 X1.1
N410 C360                       (DEGREES)
N415 G1 X.65
N420 G0 X1.1 M15
N425 M53                        (C-AXIS OFF)
N430 G28 U0 W0
N435 G97
N440 T0404 S1000 M3             (THREADING TOOL RIGHT HAND)
N445 G0 X.55 Z.2                (DEBURRING THREADS)
N450 G78 P010060 Q0330 R.001
N455 G78 X.414 Z-.45 P0330 Q0100 F.05
N460 G28 U0 W0                  (SAFE MOVE)
N465 M30
```