GE FANUC O
105/125 TURN TRAINING GUIDE
Training Index

Fanuc O Control ................................................................. Pg 1
  • Machine Control
Fanuc O Screen ................................................................. Pg 2
Fanuc O Keys ................................................................. Pg 3
  • Cursor Movement Keys
  • Change Keys
  • Store Keys
Data Input Keys ................................................................. Pg 4
  • Function Keys (Display Keys)
  • Soft Key Module
Machine Keys ................................................................. Pg 5
  • Machine Function Keys
Direction Keys ................................................................. Pg 6
  • Spindle Override Keys
  • Accessory Functions
Mode Dial ................................................................. Pg 7
  • Feed Override Dial
Pc Keyboard Keys ................................................................. Pg 8
Referencing the Machine ................................................................. Pg 9
Work Shift Description (Picture) ................................................................. Pg 10
Work Shift (How to do Work Shift) ................................................................. Pg 11
Tool Offset Description (Picture) ................................................................. Pg 13
Tool Offset (How to do Tool Offsets for X) ................................................................. Pg 14
  • Manually programming Turret Index
  • Manually programming Spindle on
Tool Offset (How to do Tool Offsets for Z) ................................................................. Pg 16
Program Training ................................................................. Pg 18
Inserting a Program .............................................................. Pg 19
  • Calling a Program up
  • Insert a word
  • Insert a End of Block
Delete a Program ................................................................. Pg 20
  • Delete all Programs
  • Delete a word
  • Delete a Block
Cancel word ................................................................. Pg 21
  • Alter a word
  • Search for number Block
  • Search for word
G Codes ................................................................. Pg 22
M Codes ................................................................. Pg 23
  • Used Addresses
Program 1 ................................................................. Pg 25
Input and Output of Programs to software from 3 1/2 Floppy ........ 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 (SUB for 3) ................................................... Pg 33
FANUC O CONTROL

PROGRAM
_N5 T0002 ;
N10 G00 X1.5 22.98 ;
N15 G02 Z0 ;
N20 G96 S500 M3 ;
N25 G0 X1 Z0.05 ;
N30 G73 U0.04 R0.04 ;
N35 G73 P40 Q65 U0.01 W0 F0.203 ;
N40 G42 G00 X0 ;
N45 G01 Z0 ;
N50 G03 X0.3 Z-0.15 R0.15 ;
N55 G03 X0.2258 Z-0.25 R0.15 ;

AD:RS. _ EDIT

MACHINE CONTROL
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

```
PROGRAM 00001 N0005
  _N5 T0202 ;
  N10 G00 X1.5 Z2.98 ;
  N15 G92 Z0 ;
  N20 G96 S500 M3 ;
  N25 G0 X1 Z0.05 ;
  N30 G73 U0.04 R0.04 ;
  N35 G73 P40 Q85 U0.01 W0 F0.003 ;
  N40 G42 G00 X0 ;
  N45 G01 Z0 ;
  N50 G03 X0.3 Z-0.15 R0.15 ;
  N55 G03 X0.2236 Z-0.25 R0.15 ;
```
FANUC O 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

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 = displays actual, relative, machine positions
PRGRM = displays program, library page
MENU / OFSET = displays wear, geometry, work shifts pages
DGNOS / PARAM = displays parameters, diagnostic pages
OPR / ALARM = displays operator & alarm messages
AUX / GRAPH = displays 2–D graph simulation

SOFT KEY MODULE
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**
  - (Reset) cancels most alarms, resets program, interrupts programs

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

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

- **Cycle start**
  - (Cycle start) program start

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

- **Mode Key**
  - (Mode Key) Automatic & Hand Mode
**DIRECTION KEYS**

These keys control axes directional movements

+4 & -4 = Additional axes

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 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 Axes and the Z Axes. 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 ⏳
   **Open** door then **Shut** door (This Initialize the safety circuits on the Machine door)

3. Move the MODE dial to ZRN 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 axes.

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

**Note:** Every time you enter Fanuc O Software or Turn the Machine On you must reference the axes
WORK SHIFT

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.
Work Shift:

1. Move the MODE dial to JOG position

2. Index to a ID tool holder position
   Press \$ will index one tool position at a time

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
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

T 01 01
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)
   - Type tool number then press INPUT button
     Example: T0202

   1. Option for Scratching
      Type S1000 for RPM press then Type M04 for
      spindle on counter clockwise press

      ▪ 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

13. Jog TURRET away from WORK PIECE using Z+

14. The R will be Tool Tip Radius

15. The T is the Tool Direction or Tool Type

16. Repeat steps for all OD tools (STEPS 1-15)

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
• INSERT A PROGRAM
  1. Press letter o then program number
  2. Press insert button

  Example: _0001 OR _01

• CALL A PROGRAM UP
  1. Press letter o then program number
  2. Press cursor down button

• INSERT A WORD
  1. Press letter then number
  2. Press insert button

  Example: press once letter _ appears press again number 7 appears

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

  Example: _N5 G01 X 0.25; G01 is the word being inserted

• INSERT END OF BLOCK
  1. Press the (EOB) button
  2. Press insert button

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

  Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT ONLY. USE INPUT ALL OTHER APPLICATIONS.
• DELETE A PROGRAM
  1. Press letter o then program number
  2. Press delete button
     Example: O0001 OR O1

• DELETE ALL PROGRAMS
  1. Press letter o plus the – & 9999
  2. Press delete button
     Example: O – 9999

• DELETE A CODE
  1. Press letter then number
  2. Press delete button
     Example: press once S appears press again 0 appears

  HINT: Deleting a word; place the cursor on the left side
     before the word being deleted
  Example: BEFORE N5_S1000; AFTER N5;
          (S1000) is the word being deleted?

• DELETE A BLOCK OR LINE NUMBER
  1. Type the number line
  2. Press delete button
     Example: _N10 G0 X1.0 F.003; make sure cursor is on
             the line being deleted (_N10)
• CANCEL MISTYPED CODE
  1. Press cancel button

  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.

• ALTER A CODE
  1. Type the code needed altered
  2. Press alter button

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

• SEARCH FOR NUMBER BLOCK
  1. Press letter n and the number of the block
  2. Press cursor down button

  Example: (N50)

  HINT: The arrow button pointing down

• SEARCH FOR CODE
  1. Type in code & number
  2. Press cursor down button

  Example: (M30)

• 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-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
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
M8     Coolant ON
M9     Coolant 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

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Part Number</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Copying tool left</td>
<td>No. SDJCL 1212 D07</td>
<td>271 050</td>
</tr>
<tr>
<td></td>
<td>Indexable inserts for alu</td>
<td>No. DCGT 070204-27 H10T</td>
<td>271 056</td>
</tr>
</tbody>
</table>

Tool Position 4 needed for Program 2, 3, 4

<table>
<thead>
<tr>
<th>Tool</th>
<th>Description</th>
<th>Part Number</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>OD-thread tool left</td>
<td>No. NL 12-3 LH</td>
<td>271 100</td>
</tr>
<tr>
<td></td>
<td>Indexable inserts for OD-thread tool, left</td>
<td>pitch 0.5 - 1.5 mm, No. 16EL T A60° S36T</td>
<td>271 105</td>
</tr>
</tbody>
</table>

Program screen & Edit mode
- To edit / change a program / insert new programs & input or output excising programs & offsets

Program screen & MDI mode
- To manually program the spindle speed / move the axis (X,Y,Z) to a specified location and or Index to a certain tool

Note: Material is 2011-T3 Alum, All feeds & speeds are programmed for this type of Aluminum
Program Q0001

G73 U = Depth of Cut   R = Retract Value
G73 P = First Block number of the Contour (Block number after the 2\textsuperscript{nd} G73)
   Q = Last Block number of the Contour   F = Feed rate for cycle

N5  (Stock 3 x. 75 alum)(Work Shift Number)
N10 G40 G70 G80 G90..........................active codes
N15 G95 \textbf{G96} G98..........................active codes
N20 G0 Z2.0........................................safe move
N25 T0202 S1000 M4.........................Right hand tool speed call out
N30 G0 X.75 Z.1...............................start point of cycle
N35 G73 U.04 R.02..............................cycle parameters
N40 G73 \textbf{P45} \textbf{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..........................................1\textsuperscript{st} diameter of contour
N60 Z-1.0..........................................length of contour
N65 X.75.........................................diameter of contour
N70 G0 Z2.0......................................safe move
N75 M30..........................................end of program
• **Changing I/O to floppy drive (Must change 1st)**
  1. Move the Mode Dial to **EDIT**
  2. Press **Parameter** on the display keys
  3. Page down until you see Parameter (Setting 1)
  4. Change the (I/O) to (A)

• **Send Program from software to floppy disk**
  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

• **Send Offsets from software to floppy disk**
  1. Press the **Menu Offset** display key
  2. Press (**Output Start**) key

• **Input Program into Software**
  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 Software**
  1. Press the **Menu Offset** display key
  2. Press (**Input**) key
N5  (Stock 3 x .75 alum)(Work Shift Number)
N10 G40 G70 G80 G90..............................active codes
N15 G95 **G96** G98...............................active codes
N20 G0 Z2.0........................................safe move
N25 T0202 S1000 M4.........................Right Hand tool and speed call out
N30 G0 X.75 Z.1.................................start point of cycle
N35 G73 U.04 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**.................................1\textsuperscript{st} diameter of contour
N60 Z-1.0 **R.1**.................................length of contour
N65 X.75............................................diameter of contour
N70 G0 Z2.0........................................safe move
N75 M30..........................................end of program
**Program Q0002**

G73  \( U \) = Depth of Cut  \( R \) = Retract Value

G73  \( P \) = First Block number of the Contour (Block number after the 2\(^{nd} \) 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 tools to a 55 degree for **FINISHING** & 80 degree for **ROUGHING**
G78 CYCLE (MULTIPLE THREAD)
Example for 1/2 20 thread

1\textsuperscript{st} G78

\begin{itemize}
  \item P = Is 6 Digits divided in 2 Digit groups
  \item P = 1\textsuperscript{st} two digits is number of FINISH PASSES \hspace{1cm} 01
  \item 2\textsuperscript{nd} two digits is PULL OUT ANGLE \hspace{1cm} 00
  \item 3\textsuperscript{rd} two digits is angle of the THREADS \hspace{1cm} 60 degrees
  \item Q = Minimum cutting DEPTH \hspace{1cm} 0020 (um)
  \item R = Finishing OFFSET \hspace{1cm} .001
\end{itemize}

2\textsuperscript{nd} G78

\begin{itemize}
  \item X = Minor DIA. \hspace{1cm} X .434
  \item Z = Length of THREAD from (0) call out \hspace{1cm} Z -1.05
  \item P = Depth of THREAD Radial \hspace{1cm} 0330 (um)
  \item Q = First cutting DEPTH \hspace{1cm} 0120 (um)
  \item F = Thread PITCH \hspace{1cm} .050
\end{itemize}

HINT:
Threading

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

IPM = RPM X PITCH

\[
\begin{array}{c}
\text{IPM} \\
\text{RPM} = \text{PITCH} = .05 = 1000 \text{ RPM}
\end{array}
\]

196 is max for a 105 or a 125 Turning 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 ; \hspace{1cm} THIS IS THE START POINT FOR G78
N105 G78
Program Q0002

N5 G96 (Stock 3 x .75).......................Constant cutting speed
N10 G0 Z2
N15 T0202 S1000 M4.........................Right hand Tool & Speed
N20 G0 X.8 Z.1..............................Safe start for Facing
N25 Z0........................................Face of part
N30 G1 X-.02 F.003........................Facing past Zero
N35 G0 X.75 Z.1.............................Start point of cycle
N40 G73 U.04 R.02.........................Cycle parameters
N45 G73 P50 Q110 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 G0 G40 X.785...........................Cancel CRC
N120 S1500 F.002
N125 G72 P50 Q115
N130 G0 X1.3 Z2............................Safe Index Pos
N135 G97 S1000 M3..........................Threading Speed in RPM
N140 T0404.................................Threading Tool
N145 X.55 Z.1...............................Start Pos. Thread Cycle
N150 G78 P010060 Q0020 R.001............Threading cycle
N155 G78 X.434 Z-1.125 P0330 Q0120 F.05
N160 X1.3 Z2.0..............................Safe Return
N165 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.
N5 G96 (Stock 2.52 x 1.25) (W.Shift)
N10 G0 Z2
N15 T0202 S1000 M4
N20 G0 X1.3 Z.1
N25 Z0
N30 G1 X-.02 F.003
N35 G0 X1.25 Z.1
N40 G73 U.04 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 S1500 F.002
N110 G72 P50 Q100
N115 G0 X1.3 Z2
N120 G97 S1000 M3
N125 T0404 (Threading tool)
N130 X.55 Z.1
N135 G78 P010060 Q0020 R.001
N140 G78 X.434 Z-.69 P0330 Q0100 F.05
N145 G28 U0 W0
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
N6 G92 U0 W
Need to touch off with turret to get the number for the (W) after you cut the first side. This number will only work repeatedly if stock is cut exactly the same size.
N10 T0202 S1000 M4 (Right Hand)
N15 X1.25 Z.1
N20 G73 U.04 R.02
N25 G73 P30 Q50 U.01 W.005 F.004
N30 G0 G42 X0
N35 G1 Z0
N40 G3 X1.2 Z-.6 R.6
N45 G1 Z-.69
N50 X1.25
N55 G0 G40 X1.3
N60 S1500 F.002
N65 G72 P30 Q55
N70 Z2
N75 G92 U0 W0
N80 M99