GE FANUC O
105/125 MILL
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 Z Work Shift) ......................... Pg 11

Work Shift (How to do X Work Shift) ......................... Pg 13
  • Manually starting the Spindle

Work Shift (How to do Y Work Shift) ......................... Pg 15

Tool Offset Description (Picture) .......................... Pg 17

Tool Offset (How to do Tool Offsets) ......................... Pg 18
Program Training ................................................................. Pg 19
Inserting a Program .................................................................. Pg 20
  • Calling a Program up
  • Insert a word
  • Insert a End of Block
Delete a Program ....................................................................... Pg 21
  • Delete all Programs
  • Delete a word
  • Delete a Block
Cancel word ............................................................................... Pg 22
  • Alter a word
  • Search for number Block
  • Search for word
G Codes ....................................................................................... Pg 23
M Codes ....................................................................................... Pg 24
  • Used Addresses
Program 1 (Contour Out Side) ...................................................... Pg 26
Input and Output of Programs to software from 3 1/2 Floppy ...... Pg 27
Program 1 (C & R) ....................................................................... Pg 28
Program 2 (I & J) ....................................................................... Pg 29
Program 2 (R) ............................................................................. Pg 30
Program 3 (Drilling) .................................................................... Pg 31
Program 4 (Main Program) using M98 (sub programs) .......... Pg 32
Program 5 (Main for Pocket Milling) ........................................ Pg 33
Program 6 (Sub program for program 5) ................................. Pg 34
The Fanuc O Screen

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

RESET

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 Offsets, Work shifts
DGNOS / PARAM = displays parameters, diagnostic pages
OPR / ALARM = displays operator & alarm messages
AUX / GRAPH = displays 2-D graph simulation

SOFT KEY MODULE

SCROLL BACK  SOFT KEYS  PAGES OVER
MACHINE KEYS

MACHINE FUNCTION KEYS

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

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

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

= (Optional stop) for programs with (m1)

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

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

= (Cycle stop) program hold, feed hold

= (Cycle start) program start

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

=(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 for Rotary axes Indexing
Press once vise closed
Press once vise open

Turret Index
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. Make sure door is closed

5. Press the Z+ this references the Z axes.

6. Press the X- this references the X axes

7. Press the Y- this references the Y axes

**Note:** Every time you enter Fanuc O Software or Turn the Machine On you must reference the axes
Reference Points of the EMCO Milling Machines

M = Machine zero point
An unchangeable reference point established by the machine manufacturer.
Proceeding from this point the entire machine is measured.
At the same time "M" is the origin of the coordinate system.

R = Reference point
A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the "R".
Required after every power failure.

N = Tool mount reference point
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point
Starting point for the dimensions in the part program.
Can be freely established by the programmer and moved as desired within the part program.
Work Shift:

1. Move the MODE dial to JOG position

2. Jog the tool STUMP to the top of the Work Piece & touch using the Direction keys. (Use piece of paper between nose and Work Piece)

NOTE: Use the Stump that has been provided with the Machine
3. Press the MENU/OFSET button

4. Press the WORK Soft key (Gray Button) Example 2

5. Make sure that X, Y, Z are all 0 if they have values then the Work Shift will be taken from those values not from the machine 0

6. Press the OFFSET Soft key (Gray Button)
   - Example 1 in the picture below
   - Record the value in the Actual Position Relative Z

7. Press the WORK Soft key (Gray Button) Example 2
8. Move Cursor to 01 location

9. Recorded value type in Work Coordinates 01(Z) which is G54

   Example: Type Z 2.463 press Input button

   This value is the distance from the top of the Machine bed to the top of the Work Piece.

Note: Machine 0 is the spindle nose touching the top of the Machine bed.
10. Jog Spindle up away from WORK PIECE using Z+

11. Either follow step 12 or follow step 13 when finished go on to step 14

12. Index to a edge finder or tool (Ex. 3/8 end mill) Press
   - Jog the Tool to the left side of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps to approach at a slower feed)

13. For Scratching move MODE Dial to MDI
   - Press the PROGRAM display button until top of the screen shows MDI (Program)
   - Type S1000 M03 T1 M6
     - S=RPM  M03=Spindle on Clockwise  T1=Tool Position  M6=Index
   - Then press CYCLE START (Door must be closed)
   - Move MODE Dial to Jog then Jog the Tool to the left side of the Work Piece & touch using the Direction keys.

Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.
14. Press the MENU/OFSET button
   - Example 1 in the picture below
   - Record the value in the Actual Position Relative X
15. Press the WORK Soft key (Gray Button) Example 2
16. Move Cursor to 01 location
17. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (X)
18. Jog Spindle up away from WORK PIECE using Z+
19. Jog the Tool to the Front of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps to approach at a slower feed)

**Note:** Machine 0 in Y is the center of the spindle to the Front of the Machine bed.
20. Press the MENU/OFSET button
   - Example 1 in the picture below
   - Record the value in the Actual Position Relative Y
21. Press the WORK Soft key (Gray Button) Example 2
22. Move Cursor to 01 location
23. The Recorded value plus the radius of the tool being used to scratch (3/8) type in Work Coordinates 01 (Y)

24. Jog the Tool up above the Work Piece using Z+
TOOL OFFSET

OFFSET

<table>
<thead>
<tr>
<th>NO.</th>
<th>DATA</th>
<th>NO.</th>
<th>DATA</th>
</tr>
</thead>
<tbody>
<tr>
<td>001</td>
<td>0.0000</td>
<td>009</td>
<td>0.0000</td>
</tr>
<tr>
<td>002</td>
<td>0.0000</td>
<td>010</td>
<td>0.0000</td>
</tr>
<tr>
<td>003</td>
<td>0.0000</td>
<td>011</td>
<td>0.0000</td>
</tr>
<tr>
<td>004</td>
<td>0.0000</td>
<td>012</td>
<td>0.0000</td>
</tr>
<tr>
<td>005</td>
<td>0.0000</td>
<td>013</td>
<td>0.0000</td>
</tr>
<tr>
<td>006</td>
<td>0.0000</td>
<td>014</td>
<td>0.0000</td>
</tr>
<tr>
<td>007</td>
<td>0.0000</td>
<td>015</td>
<td>0.0000</td>
</tr>
<tr>
<td>008</td>
<td>0.0000</td>
<td>016</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

ACTUAL POSITION (RELATIVE)

| X | 0.0000 |
| Z | 0.0000 |

NO. — EDIT

F3 F4 F5 F6 F7
1. Jog Tool tip down & touch the Top of the Work Piece  
   (Use Feed Dial or Steps to approach at a slower feed)
2. Press the MENU/OFSET button
3. The value in Actual Position (Relative) Z; type this value in Offset NO. 001(H1) If tool is going to be T1
4. Place the Radius in the corresponding Offset 011 (H11)
   • This is for the cutter compensation when using G41 or G42
5. To set more Tools Repeat Steps 1 thru 4
   • Drills & Taps need no Radius set for them

NOTE: When you use a T the H = Height
When you use a G41 or G42 the H = Radius
Program Training

Program O0001

Program O0003

Program O0005
Change the Mode Dial to Edit & Press the \[PRGRM]\ to do functions below & on the next 2 Page

- **INSERT A PROGRAM**
  1. Press letter o then program number
  2. Press insert button \[INSRT]\  
     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 \[INSRT]\  
     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 \[/#, EOB]\  
  2. Press insert button \[INSRT]\  
     **HINT**: at the end of each number line needs an End Of Block looks like a Semicolon (;)  
     Example: N5 G01 X1.00 F.003;

**NOTE:** **IN EDIT & IN PROGRAM USE INSERT ONLY.**  
**USE INPUT ALL OTHER APPLICATIONS.**
**DELETE A PROGRAM**

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

Example: O0001 OR O1

**DELETE ALL PROGRAMS**

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

Example: O – 9999

**DELETE A WORD**

1. 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 WORD
  1. Press cancel button

  **HINT:** In the ADRS. (Address) at the lower left of the screen is the word and numbers that you typed in. Before pressing insert check if what was typed in is correct. If not press cancel and retype word and numbers.

• ALTER A WORD
  1. Type the Word 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 WORD
  1. Type in Word & number (M30)
  2. Press cursor down button

• SEARCH FOR LETTER
  1. Press letter
  2. Press cursor down button

  **HINT:** This goes to the first (G). Follow steps 1 & 2 cursor goes to the next (G)
Survey commands G CODES: Mostly used

G00  Rapid motion
G01  Linear interpolation in working feed
G02  Circular interpolation, clockwise
G03  Circular interpolation, counter-clockwise
G04  Dwell time, active block by block
G09  Exact hold
G17  Selection of plane X-Y
G18  Selection of plane Z-X
G19  Selection of plane Y-Z
G20  Dimension in inch
G21  Dimension in millimeter
G28  Approach reference point
G40  Deselect miller radius compensation
G41  Miller radius compensation left
G42  Miller radius compensation right
G43  Tool length compensation positive
G44  Tool length compensation negative
G49  Deselect tool length compensation
G53  Machine coordinate system
G54  Zero point shift 1
G55  Zero point shift 2
G56  Zero point shift 3
G57  Zero point shift 4
G58  Zero point shift 5
G59  Zero point shift 6
G73  Chip break cycle
G80  Delete drilling cycle (G83 to G85)
G81  Drilling cycle
G83  Excavation drilling cycle
G90  Absolute value programming
G91  Incremental value programming
G94  Feed in inch/min
G95  Speed with feed in inch/revolution
G97  Spindle speed per minute
G98  Retract to plane of start (drilling cycles)
Survey commands M CODES: Mostly used

M00         Programmed stop, unconditional
M01         Programmed stop, conditional
M03         Spindle ON clockwise
M04         Spindle ON counter clockwise
M05         Spindle OFF
M06         Tool change
M08         Coolant ON
M09         Coolant OFF
M19         Orientated spindle stop
M25         Release clamping device
M26         Close clamping device
M30         Main program end with new start of program
M71         Blow-off ON
M72         Blow-off OFF
M98         Subroutine call-up
M99         Subroutine end

A maximum of three M commands allowed for each program block!

Used Addresses

C          Chamfer
F          Feed rate, thread pitch
G          Path function
H          Tool height, tool radius
I, J, K    Circle parameter, scale factor, K number of repetition
M          Miscellaneous function
N          Block number 1 to 9999
O          Program number 1 to 9499
P          Dwell, subroutine
Q          Cutting depth or shift value
R          Radius, retraction height
S          Spindle speed
T          Tool called out
X, Y, Z    Position data
;          Block end
Tools needed for Programs 1, 2, 3, 4, 5, 6

<table>
<thead>
<tr>
<th>Part Number</th>
<th>Description</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1Z 910</td>
<td><strong>Collet holder</strong></td>
<td>For ESX-16 collets</td>
</tr>
<tr>
<td>152 800</td>
<td>(9.0-10.0mm)Ø 3/8”</td>
<td>ESX 16 COLLETS</td>
</tr>
<tr>
<td>764 308</td>
<td>Acc. to DIN 327, shape B cutting-Ø10mm / shank-Ø10mm</td>
<td>Slot end mill, HSS</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 2024-T4 Alum, All feeds & speeds are programmed for this type of Aluminum
Program Q0001

N5 G00 G17 G40 G80
N10 G90 G94 G98
N15 G54
N20 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N25 S1800 M3
N30 G0 Z1.5
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9
N55 X1.9
N60 Y.1
N65 X.1
N70 Y1
N75 G0 G40 X-1
N80 Z1.5
N85 X2.5 Y2.5
N90 M30
• **Changing I/O to floppy drive**
  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**
Program Q0001 (C & R)

N5 G00 G17 G40 G80
N10 G90 G94 G98
N15 G54
N20 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N25 S1800 M3
N30 G0 Z1.5
N35 X-1 Y1
N40 Z-.1
N45 G1 G41 H11 X.1 F7
N50 Y1.9 C.1
N55 X1.9 C.1
N60 Y.1 R.1
N65 X.1 R.1
N70 Y1
N75 G0 G40 X-1
N80 Z1.5
N85 X2.5 Y2.5
N90 M30
Program O0002 (I & J)

```
N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1.5
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F7
N50 G2 X.5 Y1 I.5 J0 (360 degrees)
N55 G0 G40 X1
N60 Z1.5
N65 X2.5 Y2.5
N70 M30
```
Program Q0002 (R)

N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1.5
N25 X1 Y1
N30 Z.1
N35 G1 Z-.1 F3
N40 S1800
N45 G1 G42 H11 X.5 F7
N50 G2 X1.5 Y1 R.5 (180 Degrees)
N55 G2 X.5 Y1 R.5 (180 Degrees)
N60 G0 G40 X1
N65 Z1.5
N65 X2.5 Y2.5
N70 M30
Program Q0003 (Deep Hole Drilling)

G83 X = Location of hole  Y = location of hole
Z = Overall Depth of hole  P = Dwell at bottom of hole
R = Retract after Cycle  Q = incremental peck depth per pass
K = Incremental repeats only used with G91  F = Feed rate

N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z1.5
N25 X1 Y1
N30 Z.05
N35 G83 Z-.2 R.1 Q.05 F3
N40 G80
N45 Z1.5
N50 X2.5 Y2.5
N55 M30
1. To make all programs tie together or all programs O0001 thru O0003 to run 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.

6. To link all 3 programs together follow Program O0004
   - Program O0001, O0002(R), O0003 must all have M99 at the end to link together

---

**Program O0004 (Main Program)**

```
N5 G54
N10 M98 P010001
N15 M98 P010002 (R)
N20 M98 P010003
N25 M30
```
Program Q0005 (Pocket Milling)
(Making a Cycle)

N5 G54
N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)
N15 S1500 M3
N20 G0 Z2
N25 X1 Y1
N30 Z.1
N35 G1 Z0 F3
N40 M98 P030006
N45 G0 Z1.5
N50 X2.5 Y2.5
N55 M30
Program O0006 (Sub for program 5)

N5 G91
N10 G1 Z-.1 F2
N15 G90
N20 S1800
N25 G41 H11 X.4 Y1.35 F7
N30 X.2 Y1
N35 X.6 Y.3
N40 X1.4
N45 X1.8 Y1
N50 X1.4 Y1.7
N55 X.6
N60 X.2 Y1
N65 X.4
N70 X.8 Y.5
N75 X1.2
N80 X1.6 Y1
N85 X1.2 Y1.5
N90 X.8
N95 X.4 Y1
N100 G0 G40 X1
N105 M99