PRACTICAL MANUFACTURING AI

CNC MILL SIMULATOR

User manual · Fanuc 0i-MF style control · version 2.0



Mateusz Żuławski · www.mateuszzulawski.com

November 2026

CONTENTS

01 About the simulator
02 Quick start
03 Screen layout
04 Operator panel and modes
05 MDI keypad and key-in buffer
06 Control screens
07 3D view
08 Working with programs
09 Machine setup procedures
10 Alarms, collisions and warnings
11 G and M codes
12 Machine configuration
13 Data, backup, language
14 Troubleshooting
15 Limitations

01 ABOUT THE SIMULATOR

The simulator runs in a web browser and models a 3-axis vertical machining centre with a control based on **Fanuc 0i-MF**. It has the control's screens and MDI keypad, an operator panel, a G-code interpreter, a 3D view with material removal, and detection of collisions and process errors.

WHAT IT IS FOR

Learning the control, checking programs before they reach the machine, practising work offsets and tool measurement, and showing what mistakes do — with no risk to a machine.

WHO IT IS FOR

CNC operators and setters, people moving over from another control, students and instructors.

WHAT YOU NEED

A current browser (Chrome, Edge, Firefox, Safari) and an internet connection to load the 3D graphics. Nothing to install, no account.

WARNING

This is a training simulator of one documented machine (chapter 12). Real Fanuc machines differ in parameters and machine-builder macros. The simulator does not replace your machine's manual or a careful first run of the program on the machine.

The Polish and English versions behave identically and share the same data. The control screens are in English, as on a real Fanuc. Help, simulator messages and settings are in the selected language.

02 QUICK START

1. Open the simulator. On a computer, and on a phone held sideways, the whole console fits on one screen.
2. On the operator panel press **MEM** — the mode lamp lights up.
3. In the bottom right corner of the 3D view choose a simulation speed of **×5** or **×20**.
4. Press **CYCLE START**. Program O1000 runs facing, a contour with cutter radius compensation, a pocket, spot drilling, drilling and tapping.
5. Watch the positions on the POS screen, the current block above the 3D view and the material being removed. **FEED HOLD** stops motion, **CYCLE START** resumes, **RESET** aborts the program.

> **TIP**
> Switch on **SINGLE BLOCK** to make the program stop after every block. Each **CYCLE START** then runs one block.

03 SCREEN LAYOUT



THE WHOLE SIMULATOR CONSOLE — NUMBERS REFER TO THE TABLE

#	ELEMENT	WHAT IT DOES
1	Header	Brand and links, the **⛶ FIT TO SCREEN** / **↕ SCROLL VIEW** switch, **FULL SCREEN** (if the browser allows it), language switch **PL**.
2	CNC screen	Control screens: positions, program, offsets, alarms, help. At the bottom, the `>` key-in buffer with messages, and the status line: mode, STRT/HOLD/STOP, MTN, ALM, clock.

#	ELEMENT	WHAT IT DOES
3	Soft keys	Five keys under the screen — their meaning changes with the screen. ◄ returns to the main set, ► opens the (OPRT) operations.
4	MDI keypad	Addresses and digits into the buffer, screen function keys, editing, RESET, cursor.
5	3D view	Table, stock, tool and toolpaths. Labels show the state, current block, tool, spindle speed and feed. Buttons for views, toolpath, new stock and simulation speed.
6	Operator panel	Modes, CYCLE START, FEED HOLD, emergency stop, program switches, overrides, spindle, manual jog and the handwheel.

COMPUTER, TABLET, PHONE



PHONE HELD SIDEWAYS — THE WHOLE CONSOLE ON ONE SCREEN

In a landscape window (computer, tablet, phone turned sideways) the simulator switches to **fit to screen** by itself: the whole console is scaled to the window, with no scrolling.

On a phone held upright you get the **scroll view**, with a hint to turn the phone. The header button switches the view by hand, and the browser remembers your choice.

On a phone you can also pinch to zoom in and read small labels.

04 OPERATOR PANEL AND MODES



OPERATOR PANEL

MODES

MODE	USED FOR	CYCLE START
EDIT	Editing the main program in the PROG screen editor.	does nothing
MEM	Automatic operation from memory — runs the main program (selected in PROG → DIR).	starts / resumes the program
MDI	Keying in and running a few blocks, e.g. `T3 M06`, `G43 H03 Z100.`, `S1000 M03`.	runs the MDI buffer
JOG	Manual axis motion: choose X/Y/Z, hold − or +. RAPID switches to rapid jog.	does nothing
HND	Manual handwheel: axis, step ×1 (0.001 mm), ×10, ×100, ×1000 (1 mm). Turn the wheel or press −1 / +1.	does nothing
REF	Reference return to machine home: choose an axis and press +. Z first!	does nothing

> **GOOD TO KNOW**
> You cannot change mode while a program is active (including when it is stopped at M00 or in SINGLE BLOCK). Press RESET first.

BUTTONS AND SWITCHES

ELEMENT	FUNCTION
CYCLE START	Starts a program in MEM/MDI; resumes after FEED HOLD, M00, M01, G88 and SINGLE BLOCK.
FEED HOLD	Stops axis motion; the program stays active. During tapping the stop happens only after the tap has retracted.
EMERG. STOP	Immediate stop, spindle and coolant off, EMG alarm. Release the button (press it again) and press RESET.
SINGLE BLOCK	Stops after every program block (not in the middle of a drilling cycle).
BLOCK SKIP	Skips blocks that start with `/` (also `/1` … `/9`).
OPT STOP	When on, the program stops at M01.
FEED / JOG	Feed override 0–150% (also sets JOG speed; 100% = 2000 mm/min). At 0% the axes stand still.
SPINDLE	Spindle override 50–120%. Affects the feed in G95.
RAPID	Rapid override: F0 (5%), 25, 50, 100%.
CW STOP CCW	Manual spindle in JOG, HND and REF modes (last S, or 1000 rpm).
COOL	Coolant — status on screen only, no effect on the process.

05 MDI KEYPAD AND KEY-IN BUFFER



MDI KEYPAD

Address and digit keys type into the **key-in buffer** — the `>` line at the bottom of the screen. What is in the buffer goes on with a key or soft key: **INPUT**, **INSERT**, **MEASUR** and others.

- **SHIFT** — the key's second character (the small one in the corner: P, Q, R, I, J, K, L, D…).
- **EOB** — end of block `;` (a new line in the editor).
- **CAN** — deletes the last character in the buffer.
- **INPUT** — enters the buffer value (offsets, work coordinates); in the editor it works like INSERT.
- **INSERT** / **ALTER** / **DELETE** — insert the buffer at the cursor, replace a word, delete a word (EDIT, MDI).
- **POS** **PROG** **OFS/SET** **SYSTEM** **MESSAGE** **GRAPH** **HELP** — screen selection.
- **RESET** — aborts the program, clears alarms, resets modal state, stops the spindle.
- **PAGE ↑↓** and the arrows — cursor in tables, the editor and lists.

COMPUTER KEYBOARD

KEY	FUNCTION
letters, digits, `. - + / # ;`	add a character to the buffer (like the address keys)
Backspace	CAN
Enter	INPUT (when focus is not on a button)
arrows, Page Up / Down	cursor in the OFFSET/WORK tables and in the editor
Space / Enter on **-** **+**	JOG while the key is held down
arrows on the handwheel	±1 step, Page Up / Down ±10 steps (focus on the wheel)

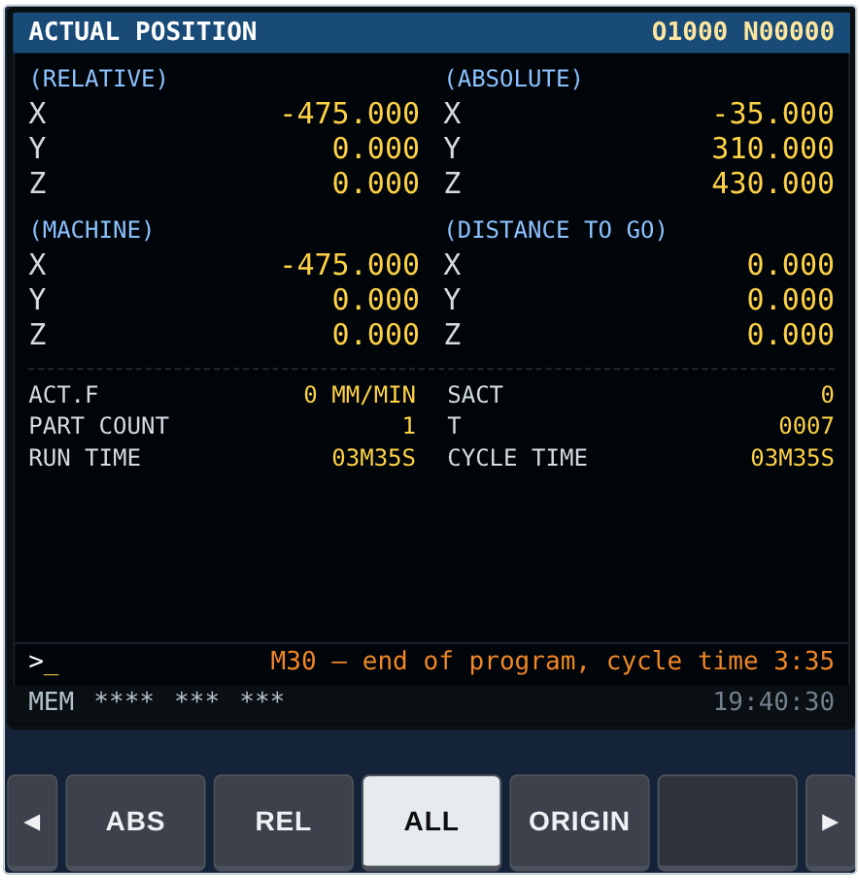
> **TIP**
> In the program editor (EDIT or MDI mode) you can type and paste with your phone's or computer's normal keyboard — the text then goes straight into the program.

06 CONTROL SCREENS

POS — POSITIONS



POS → ABS

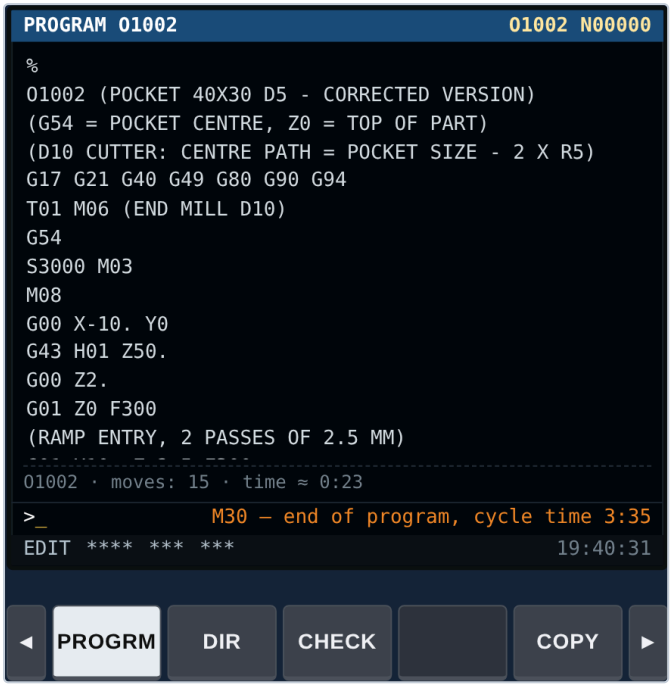


POS → ALL

- **ABS** — position in the active work coordinate system (G54–G59, with length compensation if G43 is active).
- **REL** — relative position that you zero yourself.
- **ALL** — relative, absolute, machine and distance to go, all at once.
- **ORIGIN** — zeroes REL: key in the axes (e.g. `Z`) or nothing to zero them all.

Under the positions: actual feed (ACT.F), spindle speed (SACT), part count, tool, run time and cycle time.

PROG — PROGRAM

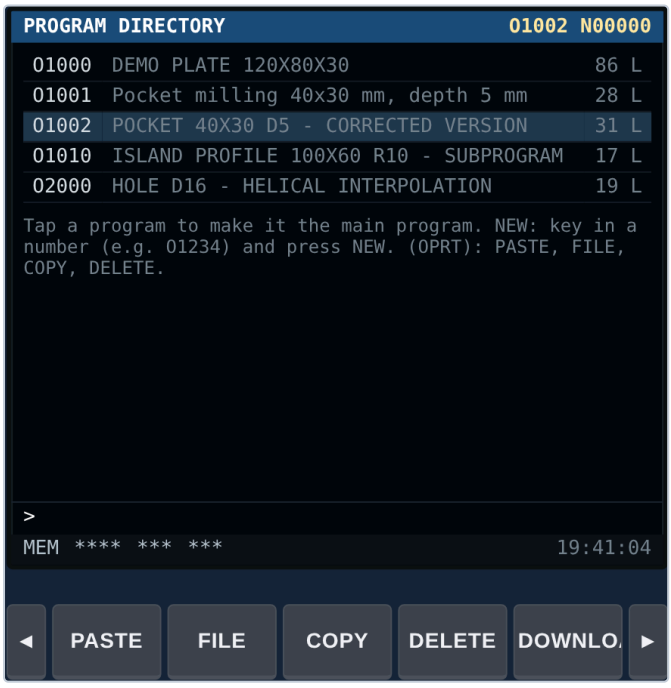


EDIT: EDITOR AND PROGRAM CHECK LINE

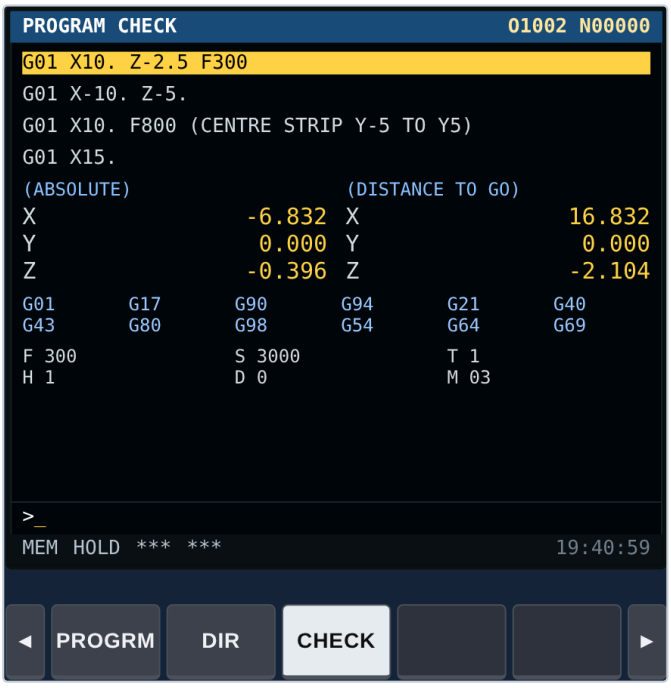


MEM: CURRENT BLOCK HIGHLIGHTED

- **PROGRM** — in EDIT the main program editor, in MDI the MDI buffer, in MEM the block list with the current block highlighted. Under the editor, a check line: number of moves, time ≈, alarm or warnings.
- **DIR** — program list; tap a program to make it the main program. **(OPRT)**: **PASTE**, **FILE**, **COPY**, **DELETE**, **DOWNLOAD**.
- **CHECK** — current and next blocks, absolute position, distance to go, modal codes, and F, S, T, H, D, M.
- **COPY** — copies the program to the clipboard.



DIR WITH THE (OPRT) OPERATIONS



PROGRAM CHECK

OFS/SET — OFFSETS AND SETTINGS



OFFSET: H AND D OFFSETS, VALUE IN THE BUFFER

OFFSET — 20 offsets: length H (GEOM, WEAR) and radius D (GEOM, WEAR). Tap a cell or move the cursor with the arrows, key in a value and press **INPUT** (replace) or **+INPUT** (add).

(OPRT): **NO.SRH** — jump to a number, **MEASUR** — measure length (H GEOM), **INP.C.** — enter the relative Z.



WORK: G54–G59 WITH OPERATIONS

WORK — the EXT external offset and the G54–G59 work coordinate systems (X, Y, Z in machine coordinates). The active system is highlighted.

MEASUR sets the offset from the current position: key in where the tool axis is now in that system, e.g. `X-66.` or `Z0` (chapter 09).



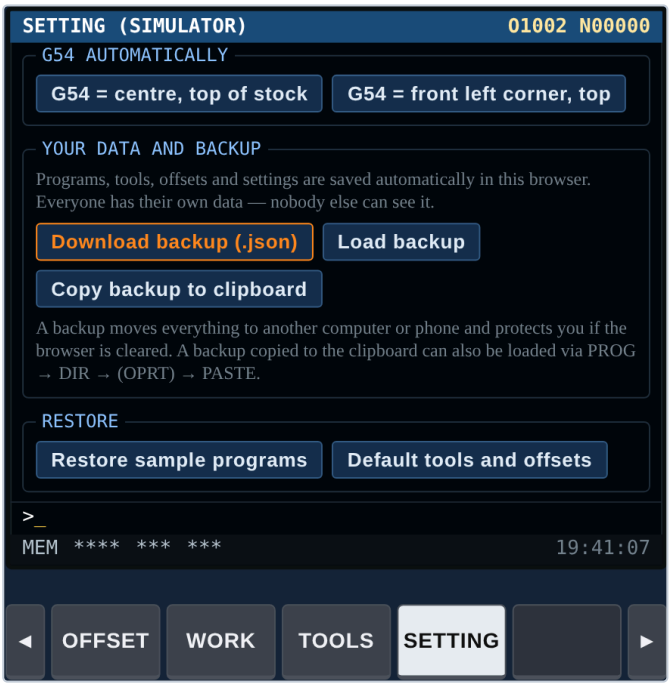
TOOLS: ACTUAL TOOL DIMENSIONS

TOOLS — the T1–T20 magazine: type, diameter, length from the spindle gauge line, flute length, pitch (tap) and description. **+ add tool** creates a new one.

> **WARNING**
> These are the **physical** tool dimensions the simulation uses. The control uses the H and D offsets from the OFFSET screen. Just as on a machine: a wrong H offset means a crash, even when the tool itself is correct.



SETTING (TOP)



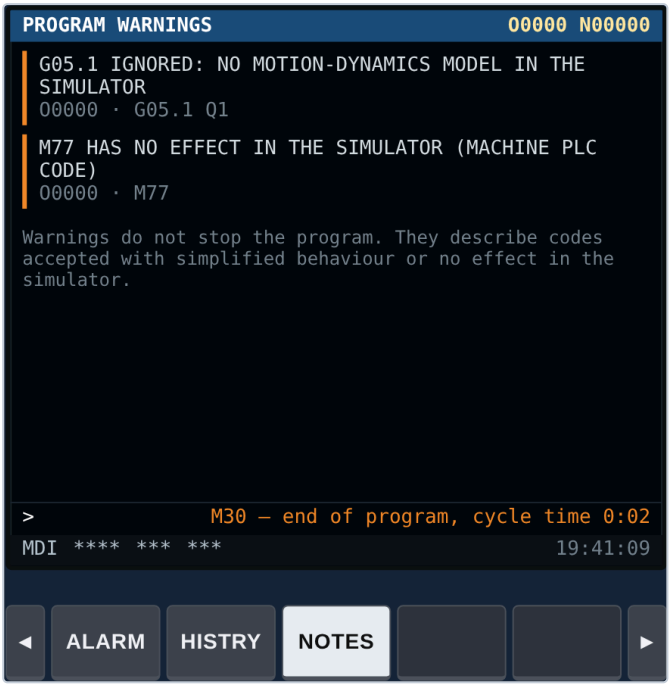
SETTING (BOTTOM): BACKUP AND RESTORE

SETTING: numbers without a decimal point (Fanuc: `X10` = 0.010 mm, or calculator: 10 mm), stock size, material accuracy (~0.5 or ~0.25 mm), new stock on every MEM start, behaviour after M00/M01, automatic G54 (centre or corner), backup, restoring programs and offsets.

MESSAGE — ALARMS, HISTORY, NOTES



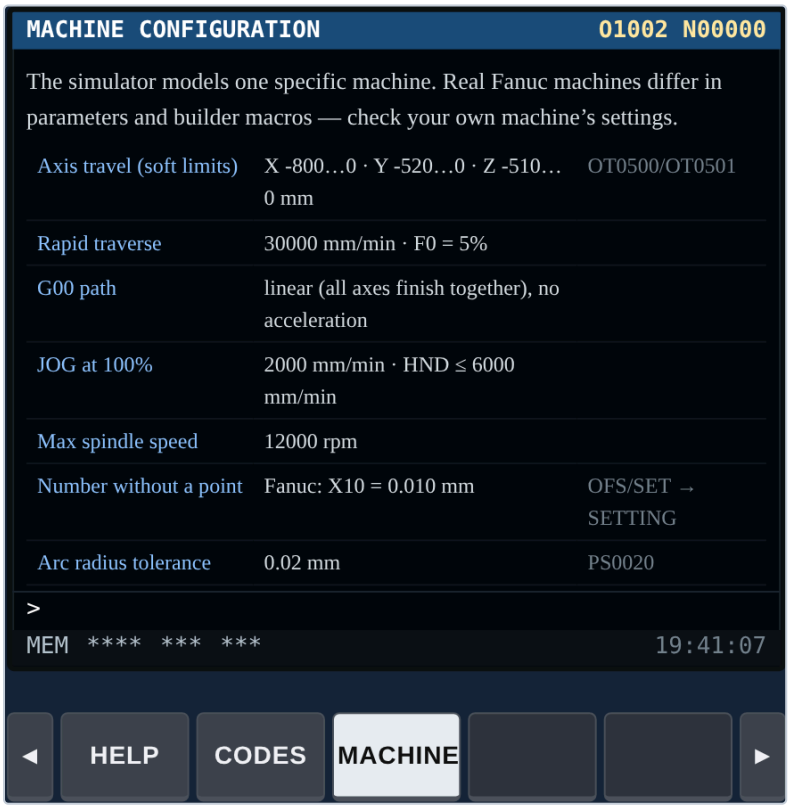
ALARM: ACTIVE ALARM AND THE BLOCK THAT CAUSED IT



NOTES: CODES WITH NO EFFECT OR SIMPLIFIED

ALARM shows active alarms with the program block; when an alarm occurs the screen switches here by itself. **HISTRY** — alarm history with times. **NOTES** — warnings for the current program: codes the simulator accepted with no effect or in a simplified form. Warnings do not stop the program.

GRAPH AND SYSTEM

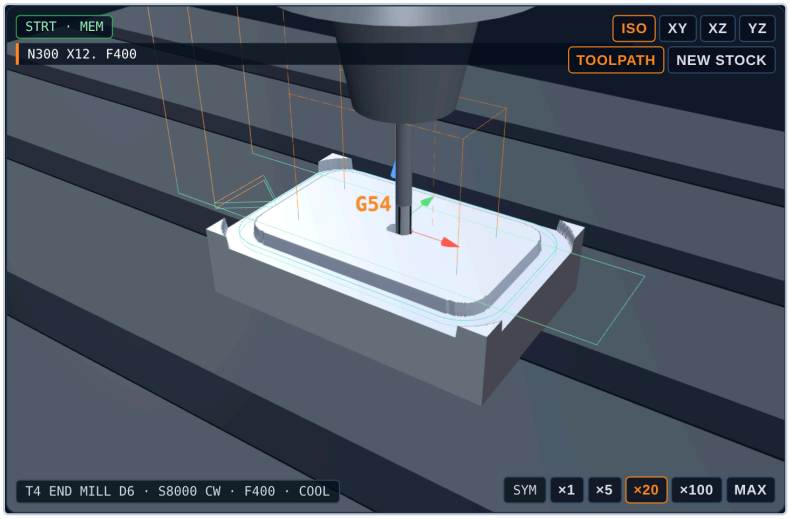


SYSTEM → MACHINE

GRAPH — buttons for the ISO, XY, XZ, YZ views and the toolpath; on a phone it scrolls to the 3D view.

SYSTEM (or **HELP**): **HELP** — quick guide, **CODES** — supported G/M codes and alarms, **MACHINE** — configuration of the simulated machine (chapter 12).

07 3D VIEW



3D VIEW WHILE A PROGRAM RUNS

- **Rotate**: one finger or the left mouse button. **Zoom**: two fingers or the wheel. **Pan**: two fingers or the right mouse button.
- **ISO** **XY** **XZ** **YZ** — preset views.
- **TOOLPATH** — shows or hides the toolpaths.
- **NEW STOCK** — renews the stock (e.g. after a collision).
- **SIM ×1 … MAX** — simulation speed (×1 = real time). The cycle time on the POS screen always counts machine time.

G00 moves from the program preview	cutting moves from the preview
path actually travelled (feed)	path actually travelled (G00)

The arrows labelled G54 (or G55–G59) show the zero of the active work coordinate system. Labels above the view: state and mode, the current block or the preview summary. Labels below the view: tool, spindle speed, direction, feed, coolant.



GAP READOUT WHILE EDGE FINDING

GAP-TO-MATERIAL READOUT

In JOG, HND and REF modes, when the tool is closer than 3 mm to the material, the label **"Gap to material …"** appears. On contact it shows **TOUCH**, and when the tool goes into the material, **"IN MATERIAL …"**. This replaces the paper or edge finder when setting work offsets.

A tangential touch on a wall removes no material. A collision is raised only when the tool goes in deeper than 0.05 mm, because the spindle is usually stopped during setting.

08 WORKING WITH PROGRAMS

RUNNING

1. In **PROG** → **DIR** tap a program — it becomes the main program.
2. Choose **MEM** mode, check the line under the program (time ≈, any alarm or warnings) and the toolpath in the 3D view.
3. **CYCLE START**. By default each MEM start begins with fresh stock (change this in SETTING).



MDI: A FEW BLOCKS TO RUN

MDI

MDI mode opens the MDI buffer on the PROG screen. Key in blocks (on-screen keypad with **INSERT**, or a normal keyboard) and press **CYCLE START**. Modal state after MDI stays — as on a machine — until RESET.

Typical use: `T3 M06`, `G90 G54 G00 X0 Y0`, `G43 H03 Z100.`, `S2000 M03`, `G91 G28 Z0`.

EDITING AND MANAGING

- **New program**: DIR → key in a number (e.g. `O1234`) → **NEW**. EDIT mode opens.
- **Editing**: **EDIT** mode. Type straight into the editor, or use the buffer and **INSERT** / **ALTER** / **DELETE**. Saving is automatic.
- **Program number** comes from the O line: changing `O1001` to `O1005` in the editor renumbers the program (if the number is free).
- **Pasting**: DIR → (OPRT) → **PASTE**, paste the whole program, **SAVE**. An existing number is overwritten only on the second press.
- **From a file**: **FILE** loads .nc, .txt, .tap and similar text files.
- **Downloading**: **DOWNLOAD** saves the program as .nc (as .txt in the Claude app preview).
- **Deleting**: key in the number (or leave the buffer empty for the main program) → **DELETE** twice.

> **WARNING**
> **Decimal point.** As on a standard Fanuc, a number without a point is in least input increments: `X10` = 0.010 mm, `X10.` = 10 mm. You can switch on calculator mode in SETTING.

SAMPLE PROGRAMS

PROGRAM	CONTENTS
O1000	Demo on a 120×80×30 plate: facing (T2), contour with G41 in subprogram O1010 (T3), pocket (T4), spot drilling G81 (T5), drilling G83 (T6), tapping G84 M8×1.25 (T7).
O1001	A "40×30" pocket with a programming error: the cutter centre path is ±20/±15 with no allowance for the radius — it produces 50×40 with an uncut island. A good example to analyse.
O1002	Corrected 40×30×5 pocket with a Ø10 end mill: ramp entry, a centre pass and the perimeter at ±15/±10.
O1010	Subprogram for the 100×60 R10 island contour with G41 D03 (called by O1000).
O2000	Ø16 hole by G03 helical interpolation with a Ø6 end mill.

STOPS AND CHANGES DURING A RUN

- **M00** / **M01**: the program stops, and the spindle and coolant stop. **CYCLE START** restores them and continues. In SETTING you can choose "program needs M03" — then a program without M03 after M00 ends in a collision, as on machines that behave that way.
- **Offsets during a run**: at M00/M01 or in SINGLE BLOCK you can change, for example, the H/D wear or a tool. After **CYCLE START** the rest of the program is recalculated; the new value applies from the next `G43 H` or `D`.
- **RESET** aborts the program, cancels G43 and modal codes, stops the spindle; the next start begins at the top of the program.

09 MACHINE SETUP PROCEDURES

By default G54 is at the centre of the 120×80×30 stock and Z0 is on its top face. H and D offsets have the same numbers as the tools. The procedures below let you practise setting up from scratch.

9.1 TOOL CHANGE

1. **MDI** mode, key in `T3 M06` , **CYCLE START**.
2. Z goes to home, the change takes about 2.5 s, the spindle stops. Tools that are not in the magazine raise an alarm.

9.2 G54 X AND Y FROM AN EDGE

1. Load a tool (e.g. T3 Ø12) and lower it in **JOG** or **HND** below the top face, next to the material.
2. In **HND** mode bring the side of the tool to the left wall, reducing the step to ×10 and ×1, until **TOUCH** appears in the 3D view.
3. **OFS/SET** → **WORK** → cursor on G54. Key in where the tool axis should be in the system you want: with zero at the centre of a 120 mm plate and a Ø12 cutter against the left wall that is `X-66.` (−60 − 6).

4. **(OPRT)** → **MEASUR**. Repeat for Y with the front wall: `Y-46.` (−40 − 6).

9.3 G54 Z

1. Touch the top face with the tool (TOUCH).
2. **WORK** → cursor on G54 Z. **MEASUR works from the ABS coordinate**: after RESET (G49 — length compensation off) key in `Z` plus the tool length, e.g. `Z100.` for T1. If you ran `G43 H01 Z100.` in MDI before, key in `Z0`.
3. **(OPRT)** → **MEASUR**. The message line reminds you whether the tool length was taken into account.

9.4 TOOL LENGTH (H)

1. Touch a surface of known height in G54 with the tool (top of the material = Z0).
2. **OFFSET** → cursor on (H) GEOM for that tool → key in `Z0` → **(OPRT)** → **MEASUR**. Result: H = machine Z − G54 Z − the height you keyed in.
3. Setting-block method: touch the block with the spindle nose or a master tool, **POS** → **REL** → key in `Z` → **ORIGIN**. Then touch with each tool in turn and press **INP.C.** on its (H) GEOM.

9.5 SIZE CORRECTION WITH WEAR

SITUATION	CORRECTION
Depth 0.05 mm too shallow	H WEAR of that tool `-0.05` (+INPUT) — the tool goes lower.
Outside contour (G41/G42) 0.04 mm oversize on the diameter	D WEAR `-0.02` — the tool moves closer to the contour.
Pocket with compensation 0.04 mm undersize	D WEAR `-0.02` — the walls move outwards.

> **GOOD TO KNOW**
> D offsets only work in programs with G41/G42 (e.g. the O1010 contour). O1001 and O1002 do not use cutter radius compensation — there the size depends on the path written in the program.

9.6 REFERENCE RETURN

In **REF** mode choose Z and press **+**, then X and Y. In a program: `G91 G28 Z0`. Note: `G90 G28 Z0` first goes to work Z0 — the simulator shows the result just as the machine would.

10 ALARMS, COLLISIONS AND WARNINGS

COLLISION
Detected contact: G00 in the material, cutting with no spindle rotation (including M03 S0), shank, holder or spindle nose in the material, tool into the table.

PROCESS
A simulator rule — the tool would break: sideways motion of a drill or tap in the material, tapping pitch F/S different from the tap's pitch.

NOTES
A warning in MESSAGE → NOTES: a code accepted with no effect or simplified. The program keeps running.

After an alarm the machine stops and the screen switches to MESSAGE. Remove the cause and press **RESET**. After a collision the material stays damaged — **NEW STOCK** in the 3D view renews it.

ALARM	MEANING	WHAT TO DO
PS0003 / PS0007 / PS0009	Too many digits / wrong decimal point / illegal character	Fix the word in the block shown on the ALARM screen.
PS0010	Unknown G code	Check the G number; the list is in SYSTEM → CODES.
PS0011	Feed zero (G01/G02/G03 without F, G95 without S)	Add F (and S in G95).
PS0020	Arc radius: start and end at different distances from the centre	Fix I/J/K or R, or the end point (tolerance 0.02 mm).
PS0022	Arc without I/J/K or R	Add the centre or the radius.
PS0033 / PS0041	Cutter radius compensation: no solution / interference (e.g. tool radius larger than an inside arc)	Use a smaller tool, fix the contour or D.
PS0034	Start-up or cancel of radius compensation on an arc	G41/G42 and G40 only in a linear block.
PS0045	G73/G83 without Q	Add the peck depth Q.
PS0077 / PS0078	Nesting too deep (more than 4) / program or N number not found	Check M98 P and M99 P and the DIR list.
OT0500 / OT0501	Axis overtravel + / –	RESET, then move the other way (JOG/HND).
EMG	Emergency stop	Release the button, RESET.
SIM	Function not supported by the simulator (macros, G68, G52…), a cycle outside G17, T not in the magazine	Change the program; the limitations are listed in chapter 15.

> **TIP**
> Before starting, look at the check line under the program: it shows an alarm with its block, overtravel and the number of warnings before the machine moves.

11 G AND M CODES

GROUP	CODES
Full	`G00 G01 G02 G03` (I J K or R, full circle, helix), `G04`, `G10 L1/L2/L10–L13`, `G17 G18 G19`, `G20 G21`, `G28 G30`, `G43 G44 G49`, `G53`, `G54–G59`, `G73 G74 G76 G80–G89` (G17), `G90 G91`, `G94 G95`, `G98 G99`
Simplified	`G41 G42` — G17 only, outside corners rounded with an arc; `G76` — shift +X by Q; `G88` — stop at the bottom, retract after CYCLE START
M	`M00 M01 M02 M03 M04 M05 M06 M07 M08 M09 M19 M29 M30`, `M98 P… L…` and `M98 Pxxxxyyyy`, `M99`, `M99 Pn`
With a warning	`G05 G05.1 G08 G09 G61 G61.1 G63`, unknown M codes, `G53` in G91
Cancel codes	`G13.1 G15 G40.1 G49.1 G50 G50.1 G64 G67 G69 G80.5` — accepted with no effect (they switch off functions the simulator does not have)
Alarm	Macros (`#`, IF, WHILE, G65/G66), `G07.1 G12.1 G16 G27 G31 G39 G45–G48 G51 G51.1 G52 G54.1 G68 G87 G92`

The full list with alarms: SYSTEM → **CODES**.

12 MACHINE CONFIGURATION

PARAMETER	VALUE
Axis travel	X −800…0, Y −520…0, Z −510…0 mm (machine home = 0, 0, 0)
Rapid traverse	30,000 mm/min; override F0 (5%), 25, 50, 100%; linear G00
JOG / handwheel	2000 mm/min at 100%; handwheel up to 6000 mm/min
Spindle	max. 12,000 rpm (higher S is clamped with a warning)
Tool change	M06: Z to home, XY stays, about 2.5 s, spindle stops; magazine T1–T20
Offsets	20 × H and D (geometry + wear); EXT and G54–G59
Cycles	G17 only; G73 retract 0.5 mm; G83 clearance 1.0 mm; rigid tapping (feed override inactive)
RESET	cancels G43 and modal state, stops the spindle, rewinds the program
M00 / M01	spindle and coolant stop; CYCLE START restores them (can be changed in SETTING)
Arc tolerance	0.02 mm
Table and stock	table Z −460; stock 120×80×30 (adjustable), front left corner at X −500 Y −350
Holder	40 mm, Ø52; spindle nose Ø92
Material	height map, cell ~0.5 mm (optionally ~0.25 mm)
Dynamics	no acceleration — cycle time is a lower estimate

13 DATA, BACKUP, LANGUAGE

- Programs, tools, offsets and settings are saved **automatically in the browser**, on this device. Everyone has their own data; nobody else can see it.
- The Polish and English versions use **the same data**. Switching language deletes nothing. Tool names and program comments stay in the language they were typed in.
- The data is lost if you clear the browser's data. Safari may also delete it after about a week without a visit to the page. In private mode nothing is saved — the simulator warns you about it.

1. **Backup**: OFS/SET → SETTING → **Download backup (.json)**. The file holds all programs, tools, offsets and settings.

2. **Moving**: on the other device, **Load backup**. Instead of a file you can also use **Copy backup to clipboard** and paste it in PROG → DIR → (OPRT) → **PASTE**.

3. **Restoring**: **Restore sample programs** and **Default tools and offsets** (both need a second press).

The **PL** / **EN** button in the header switches the language of the whole page.

14 TROUBLESHOOTING

SYMPTOM	CAUSE AND FIX
CYCLE START does nothing	A mode other than MEM or MDI, an active alarm, or EMG. Look at the message line above the status line.
I cannot change mode	A program is active (including at M00 or in SINGLE BLOCK). Press RESET.
A message instead of the 3D view	No internet, a blocked CDN or no WebGL. The control, interpreter and collision checks keep working. Reload the page when the connection is back.
A 0.01 mm move instead of 10 mm	A number without a point: `X10` = 0.010 mm. Write `X10.` or switch on calculator mode in SETTING.
“No rotation” collision after M00	SETTING is set to “program needs M03”. Add M03 after M00 or change the setting.
G00 collision on the Z approach	Wrong H offset, wrong G54 Z or missing G43. Compare H in OFFSET with the length in TOOLS and check G54 in WORK.
MEASUR gives a Z off by the tool length	MEASUR works from ABS: with G49 key in Z + tool length, with G43 active key in Z0.
The D offset changes nothing	The program has no G41/G42 — radius compensation works only with them.
A program has “disappeared”	A different browser or device, cleared data or private mode. Load your .json backup.
Text too small on a phone	Turn the phone sideways, use FULL SCREEN or pinch to zoom.

15 LIMITATIONS

- No macros (`#`, IF, WHILE, G65/G66) and no `G68 G51 G52 G92 G54.1 G16 G87` — they raise an alarm and are never skipped.
- Cutter radius compensation rounds outside corners with an arc. The part contour is correct, but the tool path at the corner differs from Fanuc type C.
- No acceleration or look-ahead: cycle time is a lower estimate; G05.1, G08 and G61 give a warning.
- SINGLE BLOCK stops at block boundaries, not at the stages of drilling cycles.
- The material is a height map with a ~0.5 mm cell (optionally ~0.25 mm): walls are smoothed to the cell, and holes smaller than a cell are not visible. There are no undercuts, because these tools cannot make them in 3 axes.
- Vice, fixtures, guards and the tool magazine are not modelled for collisions.
- PROCESS is a set of simulator rules — there is no model of forces, temperature or tool wear.
- Alarm numbers are based on Fanuc 0i-F; on your machine the numbers, texts and parameters may differ.

Bottom line. The simulator shows what a program will do on this specific, documented machine. On a real machine always check the offsets, the work coordinate system and the first run with reduced rapid.

PRACTICAL MANUFACTURING AI

MATEUSZ ŻUŁAWSKI

More on CNC and the practical use of AI in manufacturing:

mateuszzulawski.com

LinkedIn

© 2026 Mateusz Żuławski · Practical Manufacturing AI. Training simulator. FANUC is a trademark of FANUC Corporation; this simulator is not affiliated with it.