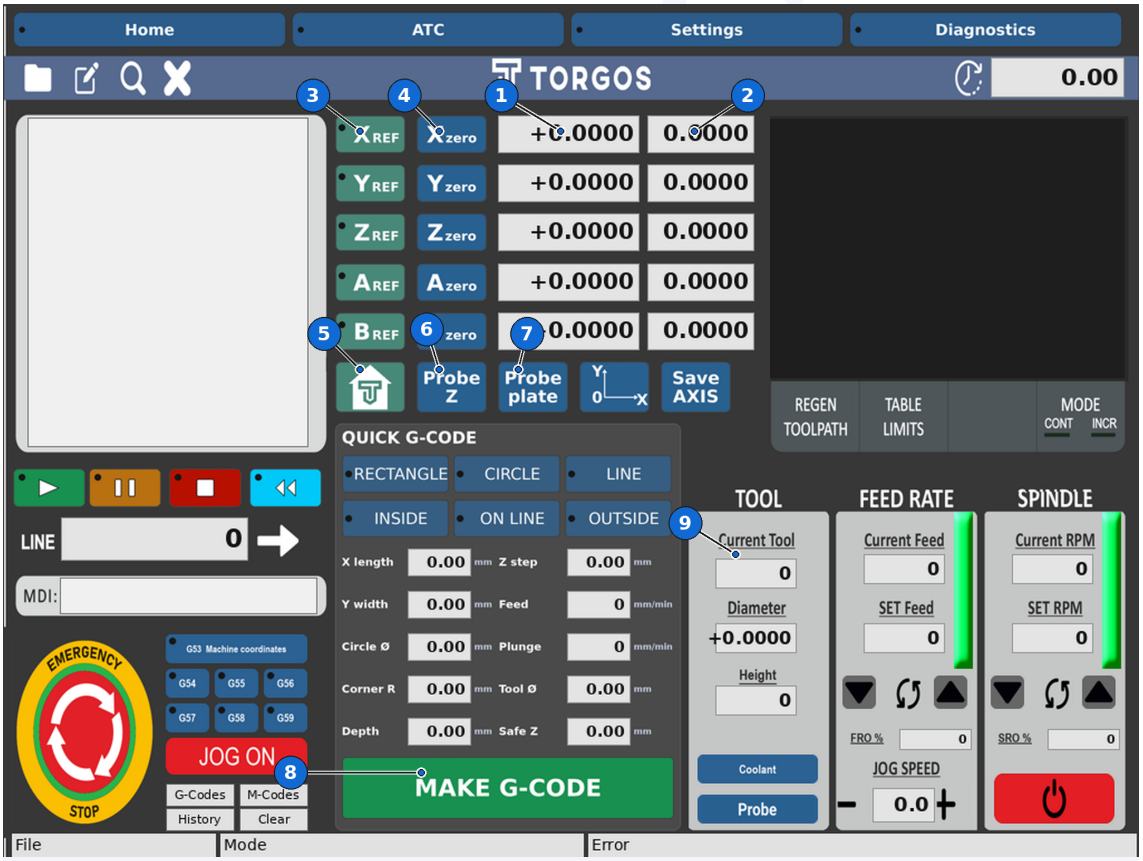
TORGOS screenset for Mach3

User and installation manual

Version 2026.11.10 · torgoscnc.gr



The main screen — what is what

1. Work coordinates (G54). Measured from the zero you set on the part
2. The work zero in machine coordinates, the offset of the active G54. Click to edit: 0 makes G54 coincide with G53
3. REF: drives the axis to its reference point. The dot shows whether it has been there
4. ZERO: sets the work position of the axis to zero where the tool stands now
5. The House: drives all axes to the reference point and asks which tool is in the spindle
6. Probe Z: measures the tool on the fixed tool setter and sets the Z zero
7. Probe plate: finds the corner of the part with the corner plate and sets X0, Y0 and Z0
8. Quick G-code: creates the program for a rectangle, circle or line without CAM
9. The tool in the spindle and its diameter

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

1. What the TORGOS screenset is

The TORGOS screenset is a new operator screen for Mach3, designed by a CNC engineer for everyday work at the machine tool. The default Mach3 screen has small buttons and hides the everyday functions inside menus. The TORGOS screenset is designed to be read clearly, and everything the operator needs every day is on the first page: tool measuring, zeroing and the reference points. On the same page there is a quick G-code panel that does simple jobs without CAM. For machines with an automatic tool changer (ATC) there is a separate tab, with the tool magazine and the measurement of all tools with one press. It works on a 1024×768 screen or larger, in Greek or English.

Licence

The screenset and the manual are the work of TORGOS CNC. The licence you bought covers one business: you may use them on as many machines as you have. You may not modify, translate or pass them on to third parties, by any means or tool.

What it assumes

The screenset is a screen for a Mach3 that already runs correctly. It does not set up a machine tool from scratch and does not change its settings:

- the steps per unit,
- the ports and pins (Ports & Pins),
- the travel limits,
- the backlash (Backlash),
- the tool-change procedure.

Those stay as you have them. If your machine does not move correctly today, the machine must be set up first and the screen installed afterwards.

Measuring and probe

The measurements with a tool setter, touch plate, corner plate and 3D probe work on any machine with a probe input in Mach3.

Automatic tool change

The tool magazine, MEASURE ALL and the automatic tool change are designed for TORGOS machines and for machines we have rebuilt. Every automatic changer is different, with its own cylinders, clamp, bridge and outputs, so on a machine from another maker these buttons may not work without adaptation, and adaptation may mean a retrofit. If you are not sure, ask us before you buy.

Requirements

1) Mach3 on Windows, with a profile that already works.
2) Windows 8.1 or newer, 64-bit, for the installer. On old 32-bit XP or 7 the installation is done by hand, as described in section 10.
3) A 1024×768 screen or larger.
4) For measuring, a tool setter, a touch plate or a 3D probe, wired to the Mach3 probe input.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

2. Installation

The installation is done with a single file, **TORGOS-screenset-Setup.exe**, on the machine-tool computer that has Mach3 installed.

Note. Mach3 must be closed during the installation.

The program asks four things. After every answer we press Enter. No question has a default: a bare Enter does not move on, it waits for an answer.

1) **Language**: 1 for Greek, 2 for English. The whole package, screen, images and messages, is installed in the language you chose.
2) **How the machine changes tools**: 1 by hand, 2 automatically. If the machine tool has a manual or semi-automatic change, we choose 1. If it has an automatic tool changer, we choose 2. The choice decides how the Probe Z, MEASURE ONE and MEASURE ALL buttons work internally.
3) **Which profile**: you pick the profile you work with today. The program does not touch it. It creates a copy named "your profile TORGOS SCREEN", with all its settings and macros, and installs the screen inside that copy. It is essentially a clone of your own profile, and if yours worked correctly, the clone will work too.
4) **Whether to install the M142 and M143 macros**: we answer Y. They are Probe Z and the corner plate, and the program asks separately because these two need the values in TorgosProbe.txt to work. We answer N only if we already have our own M142 and M143 in the profile and want to keep them.

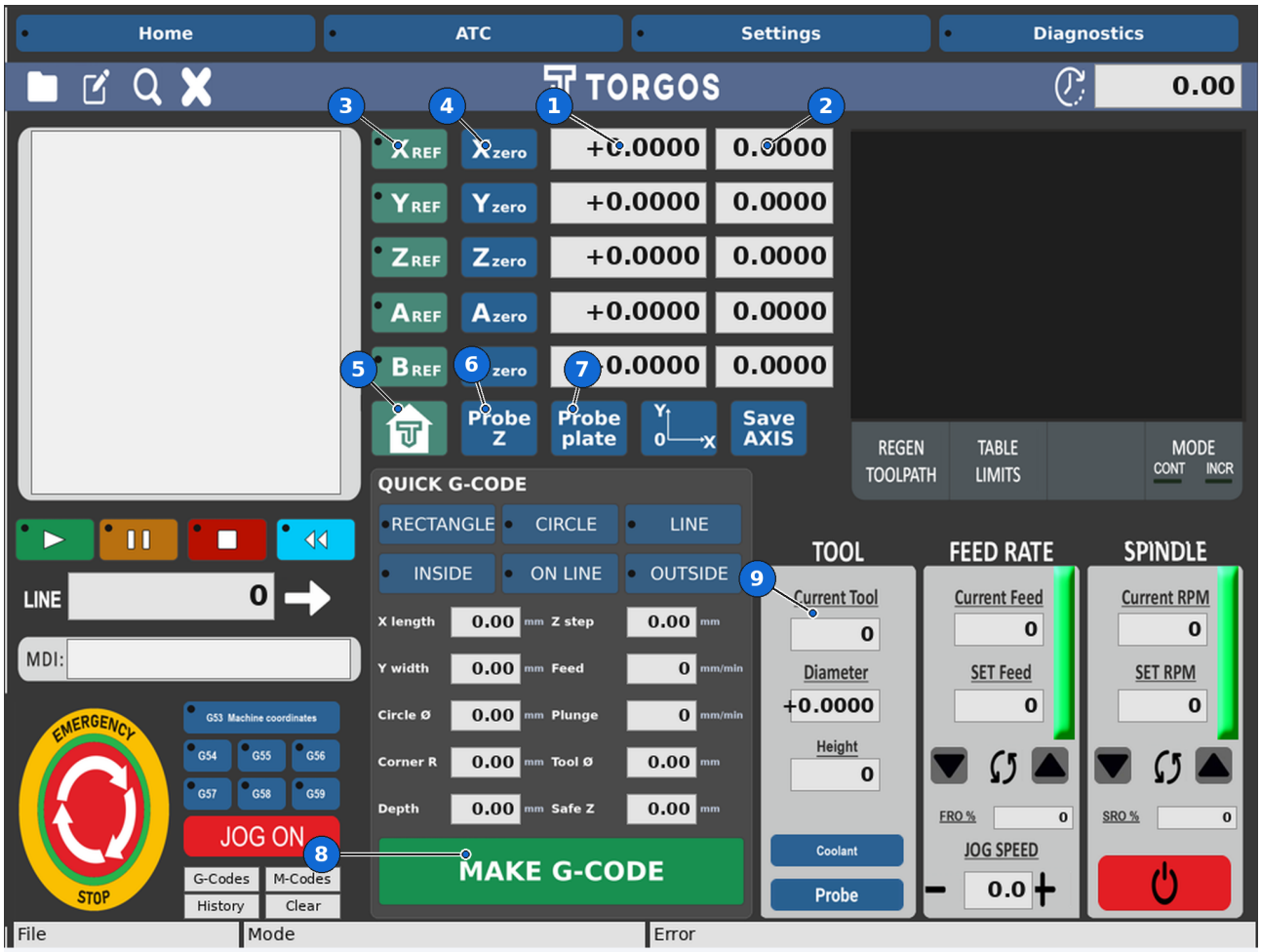
The program copies the screen and images into C:\Mach3, puts the button macros into the new profile, writes into the profile that it loads the TORGOS screen and runs the macro pump, reads the machine's M6Start and fills by itself whatever numbers it finds there into TorgosProbe.txt. It keeps a log at **C:\Mach3\TORGOS-egkatastasi.txt**, with date, version, profile and the files installed, and for every file it replaces it keeps a dated copy (.bak-…).

After the installation you start Mach3 and pick the "… TORGOS SCREEN" profile. The screen loads by itself. If you see the old screen, go to View › Load Screens and pick TORGOS-v2.set.

Note. The first time Mach3 opens a new screen, the G-code window bar may appear displaced. This is known Mach3 behaviour. Close and reopen it and it will not happen again.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

3. The main screen



The main screen — what is what

1. Work coordinates (G54). Measured from the zero you set on the part
2. The work zero in machine coordinates, the offset of the active G54. Click to edit: 0 makes G54 coincide with G53
3. REF: drives the axis to its reference point. The dot shows whether it has been there
4. ZERO: sets the work position of the axis to zero where the tool stands now
5. The House: drives all axes to the reference point and asks which tool is in the spindle
6. Probe Z: measures the tool on the fixed tool setter and sets the Z zero
7. Probe plate: finds the corner of the part with the corner plate and sets X0, Y0 and Z0
8. Quick G-code: creates the program for a rectangle, circle or line without CAM
9. The tool in the spindle and its diameter

In the centre are the DROs, the axis position readouts. Every axis has two readouts. The left one shows the tool position in the work system, relative to the zero you have set on the part, and moves with the tool. The right one shows where that zero lies in machine coordinates, that is the offset of the active system, as on the Offsets tab of Mach3. It does not change when you move the machine, it changes when you zero. Above JOG ON are the G54 to G59 coordinate systems, with a dot on the active one, and G53 for machine coordinates. G53 is not a system you select: its button only changes what the left column shows, work position or machine position, and the dot of the active G54 to G59 stays lit, because that is where the program keeps running.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.



REF and ZERO, next to each axis

REF drives the axis to its reference point, that is the home switch, so the machine learns where it is. The dot next to it lights when the axis has been referenced. ZERO sets the work position of the axis to zero where the tool stands now. Zero with G53 switched off, so that the left column shows the zero you have just set: with G53 on, ZERO still works, but the screen keeps showing machine position.

The right-hand readout can be clicked and accepts a number, like every Mach3 DRO. If you want the work coordinates to coincide with the machine coordinates, that is G54 to read the same as G53, type 0 into the right-hand readout of each axis. Each of G54 to G59 has its own zero, so if you want them all equal to G53, repeat it with each one active.



Reference point of all axes, the House

It drives all axes to their reference point in the preset order. Before anything moves, it asks which tool is in the spindle. We answer 0 when the spindle is empty and all tools are in the magazine, otherwise the tool number. On machines with an automatic changer it accepts only pockets declared with the HOLDER button on the ATC tab. On a manual installation it accepts any number from 1 to 12.

The question is there to protect against a hasty move by the operator. Mach3 remembers the tool number from the last time it was closed. If it closed with tool 3 assigned and you changed the tool by hand in the meantime, when it reopens it will still believe it holds tool 3. Because every tool change starts with the rule "if the same tool is requested, do nothing", the program would cut with the wrong tool without any warning. The question at the reference point prevents that.

TORGOS CNC

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.



Probe Z

It measures the tool on the fixed tool setter and sets the work Z zero. What exactly it does depends on whether the machine changes tools by hand or automatically, and is described in section 7.



Probe plate

It finds the corner of the part with the corner plate and sets X0, Y0 and Z0. It is the same button as CORNER X0 Y0 on the ATC tab, section 7.



ZERO X Y with Z lifted

It first raises Z to the safe height and then goes to the work X0 Y0. We use it to return to zero without dragging the tool across the part.

Quick G-code



Right below the DROs is the Quick G-code. It creates the program for three basic shapes, rectangle, circle and line, and lets you decide whether the tool runs INSIDE, ON or OUTSIDE the line of the shape. Its fields are:

X length / Y width: the dimensions of the rectangle (mm)

Circle Ø: the diameter of the circle (Ø)

Corner R: the rounding of the rectangle corners, with which you can go all the way to an oval (mm)

Depth: the thickness of material to be cut (mm)

Z step: the depth of each pass, from which the number of passes is calculated (mm)

Feed: the cutting speed (mm/min)

Plunge: the speed at which the tool descends into the material (mm/min)

Tool Ø: the diameter of the tool to be used (Ø)

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

Safe Z: the height to which the tool lifts for rapid moves (mm)

Pressing MAKE G-CODE creates the program and loads it ready to run. It starts from the work X0 Y0 Z0, so you must have zeroed on the part first. For a facing pass, a pocket or a simple cut-out you do not need to open a CAD program.

TORGOS CNC

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

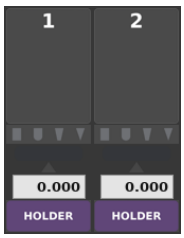
4. The ATC tab



The ATC tab — tools and measuring

1. The tool magazine with 12 pockets. Each pocket shows which tool it holds and whether a holder exists
2. HOLDER: declares which pockets really exist on your machine. Done once
3. The tool data of the pocket. The diameter is saved to the tool table
4. MEASURE ONE: measures the tool in the spindle and writes its length to the table
5. MEASURE ALL: measures every tool in the magazine, one by one
6. ZERO Z with the touch plate, and CORNER X0 Y0 with the corner plate
7. HOLE CENTRE and PART CENTRE with the 3D probe
8. SENSOR: tests the signal by hand. Moves nothing
9. The coordinates, with a switch for machine or work
10. Tool change: which tool the spindle holds now and which pocket it will change to
11. The probing device you are using: tool setter, touch plate or 3D probe

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.



The tool magazine and the HOLDER button

At the top of the tab is the tool magazine with 12 pockets. Every pocket shows which tool it holds and, below it, has a button named HOLDER, with which you declare which pockets really exist on your machine. This is done once and the setting stays after Mach3 closes. On a manual installation the pockets do not affect operation.



TOOL IN THIS POCKET and SAVE TO TABLE

When you select a pocket, the TOOL IN THIS POCKET box shows the data of its tool: cutting and shank diameter, flute length, overall length, number of flutes, angle, tip radius and type (FLAT, BALL NOSE, TAPER, V). With the SAVE TO TABLE button the diameter is written into the Mach3 tool table and the rest is stored permanently in the profile.



TOOL CHANGE: ASSIGN and CHANGE

The TOOL CHANGE box shows which tool the spindle holds now, the "In spindle" one, and which pocket it will change to. ASSIGN tells Mach3 which tool you are holding, without any motion. You need it after a manual change, so that Mach3 knows what is in the spindle. On machines with an automatic changer it accepts only pockets with a HOLDER. CHANGE sends a tool-change command (M6) to the pocket you picked, and then your own change procedure runs, as described in section 8.



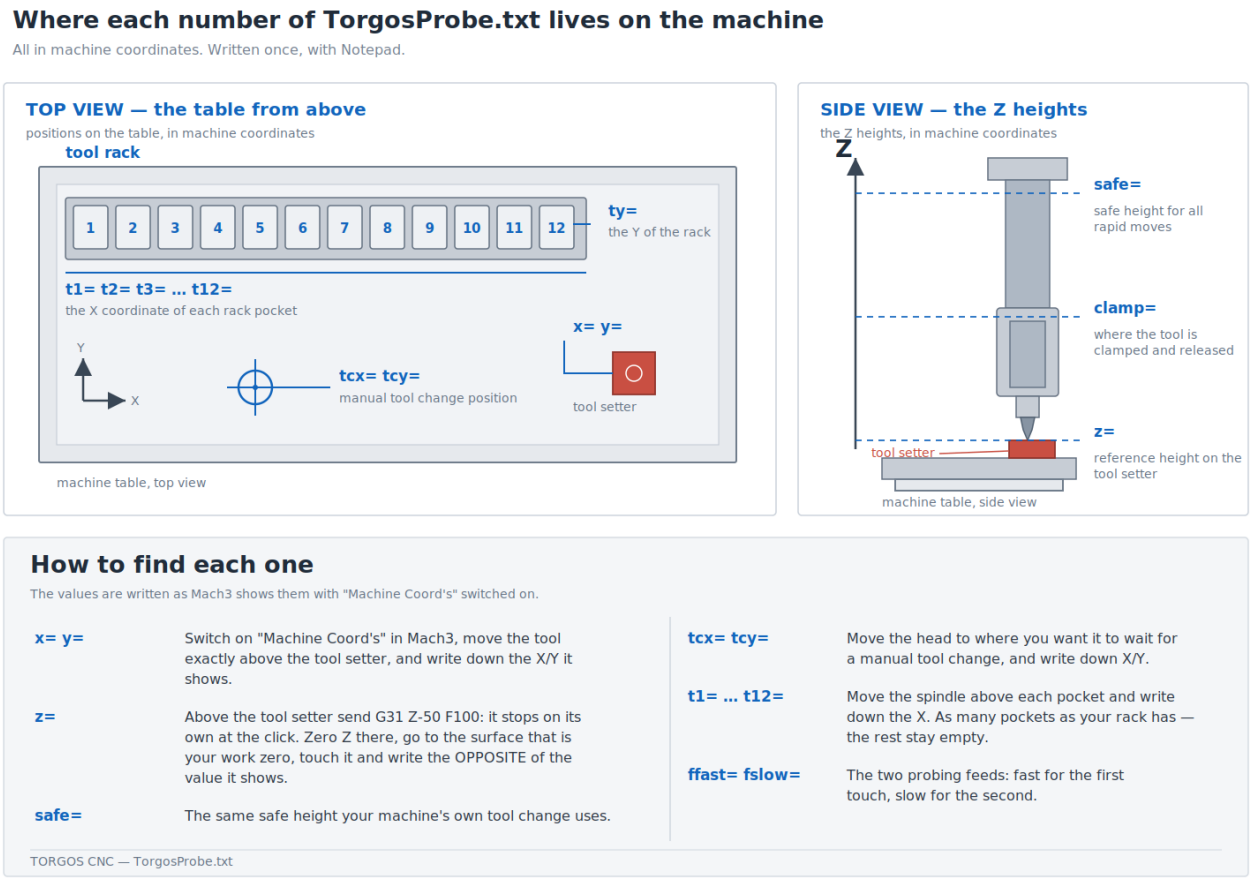
SENSOR, the signal test

It is a test that moves nothing. You press the button, touch the sensor by hand, and the lamp shows whether Mach3 received the signal. It is good practice before every working day, and it is mandatory before the first measurement, for the reason explained in section 6. To its right is the probing-device switch: tool setter, touch plate or 3D probe.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

5. The settings of your own machine, TorgosProbe.txt

The position of the sensor and of the magazine differ on every machine tool, which is why no such number is written inside the macros. They are all in one text file, **C:\Mach3\TorgosProbe.txt**, which opens with Notepad. While a number is missing, the matching button does not move the machine but stops and reports which one is missing. Decimals are written with a dot and not a comma: 1000.5 is right, 1000,5 cannot be read.



The installer reads your profile's M6Start and fills in by itself whatever values it finds there. If you run it again, it keeps the numbers you have already written and only adds what is missing. For tool measuring the basics are needed:

x= y=: the position of the tool setter in machine coordinates. You find it by switching on Machine Coord's in Mach3 and bringing the tool exactly above the sensor.

safe=: the safe Z height for rapid moves, in machine coordinates. It is the same one your machine's tool change uses.

z=: the reference height on the sensor. How to find it is described in section 5.1.

ffast= fslow=: the two probing speeds, the fast one of the first touch and the slow one of the second (mm/min).

If you change tools by hand and use our M6Start (section 8), you additionally need:

tcx= tcy=: the position the spindle goes to for the change, in machine coordinates. If they are missing, the change stops with a message saying so, without anything moving.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

If you have a magazine with automatic change, you additionally need:

atc= probe=: 1 or 0: whether the machine has a magazine, and whether it measures the tool on every change. The installer writes atc= from your answer to the second question.

t1= … t12=: the X position of each magazine pocket. You fill in as many as your machine has and the rest stay empty.

ty=: the Y position of the magazine. Stays empty if the magazine rides on the gantry.

bridge= clamp=: the X position the bridge moves out to before a change, and the Z height at which the tool is clamped.

px= py= retx=: a second plate, only if it is different from the tool setter. Empty means the same.

5.1 How to find z=

1) With the tool you usually zero with and Z raised, you send from the MDI **G53 G0 X**x **Y**y, with the values you wrote in x= and y=.
2) You send **G31 Z-50 F100**. The tool comes down on its own and stops the moment the sensor gives its signal. It cannot be done by jogging, because jogging does not stop at the touch.
3) You zero Z there.
4) You go to the surface you use as work zero, table or workpiece, touch it as you always do and read the Z.
5) You write the value with the opposite sign in z=: if it shows +17.74, you write -17.74. z= holds for one reference surface: if you zero on the table, Probe Z will find the table again, not the workpiece. On a machine with an automatic changer, z= is the height of the sensor surface in machine coordinates.
6) If the sensor is ever moved, you repeat the above.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

6. Before the first measurement

> **First of all, the SENSOR test.** If Mach3 does not see the signal, probing (G31) never stops and the machine drives the full travel through the sensor. Do not measure anything before this test passes. The first time, keep your hand on the E-Stop.

1) You reference the machine with the House. Without it, machine coordinates mean nothing, and every measuring button refuses to start.
2) You check that x=, y=, z= and safe= in TorgosProbe.txt are right. You send from the MDI **G53 G0 Z**safe and then **G53 G0 X**x **Y**y. The tool must stand exactly above the sensor.
3) You make the first measurement with your hand on the E-Stop.

Every measurement is made with two touches, a fast one to find the sensor and a slow one for accuracy. In all measurements the result is written only after it is confirmed that the tool really touched the sensor within the travel. If it did not, nothing is written and the message says how far it went. Without this check, a cut cable or a plate out of position would mean the machine sets zero in thin air.

7. The measurements, button by button



Probe Z, on the main screen

On a machine with manual change it asks which tool you are putting in and sends the tool-change command (T... M6). From there your machine's own change procedure runs, as described in section 8: with our M6Start it raises Z, goes to the tcx, tcy position, waits for you to change the tool and press OK, and with probe=1 it measures on the sensor and sets the work Z zero from z=. If tcx and tcy are missing, it stops with a message saying so, without moving. The number you give must differ from the tool already held, otherwise the machine's change does nothing.

On a machine with an automatic changer it goes to the sensor with the tool it holds, measures, and sets the work Z zero from z=.



MEASURE ONE

It measures the tool currently in the spindle and writes its length into the Mach3 tool table, under the number declared as In spindle. At the end it applies G43 with that length. The spindle must declare a tool, not 0, otherwise the button asks you to ASSIGN one first. The length is calculated as contact Z minus z=, so z= must be correct.

On a machine with manual change it writes into the table the machine height of the contact, which is a fixed reference because the sensor does not move. It does not touch the work zero and does not apply G43 by itself.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.



MEASURE ALL

It works only on a machine with an automatic changer. It goes through the tools of the pockets that have an X in TorgosProbe.txt, from t1 to t12: it picks each one, measures it, writes its length and returns it to its pocket. If one does not touch the sensor, it returns it to its pocket and stops, reporting in which pocket the problem occurred. On a machine with manual change the button replies that there is no magazine. It is tested on TORGOS machines, and on another machine it may need adaptation, as explained in section 1.



ZERO Z

With the plate on the material and the tool just above it, it goes straight down, touches twice, and sets Z0 at the material surface taking the plate thickness, which is 8 mm, into account. Then it lifts. It needs no number from TorgosProbe.txt. If your plate has a different thickness, tell us, because it is one constant in the macro.

ZERO Z writes the work zero. If you run a program with two tools and lengths from the tool table (G43), you must zero with a tool that is measured, otherwise the second tool will be off by the difference of the lengths.

TORGOS CNC

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.



CORNER X0 Y0, the corner plate

With the 70×70×8 mm corner plate on the corner of the part and the tool above the plate, near the corner, it finds the plate Z, then the X side and the Y side, and sets X0, Y0 and Z0 at the corner of the material. All moves are relative, so no setup is needed. It uses the tool diameter from the tool table, so you must have saved it first.



HOLE CENTRE

With the 3D probe inside the hole, roughly at its centre, it touches both walls on the X axis and goes to the middle, does the same on Y, and zeros X and Y at the centre.



PART CENTRE

With the probe above the part, roughly at its centre, it asks for the size of the part in X and in Y, finds the top surface, goes down outside each side and touches inwards, and zeros X and Y at the centre. The dimensions you give decide how far out the probe goes, which is why they must be the real ones.

TORGOS CNC

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

8. The tool change

The tool change (M6) belongs to your own machine. Its own M6Start.m1s is copied into the new profile and we do not touch it. The CHANGE button and every T… M6 command inside a program run that.

The package's macros\ folder also holds our own optional M6Start.m1s, which reads all its numbers from TorgosProbe.txt. With atc=0 it raises Z, goes to tcx, tcy, waits for you to change and press OK, and if probe=1 it measures and sets the Z zero. With atc=1 it performs the automatic change at the magazine, with bridge, clamp and holders, and if probe=1 it measures and writes the length into the tool table with G43. You put it into the profile by hand only if you want it, and the automatic-change version applies to TORGOS machines, as explained in section 1.

9. If something stops working

Your original profile always stays untouched. If you open Mach3 with it, the machine works as it did before the installation. The first step in every problem is to run the installer again, which is harmless, keeps your numbers and fixes almost everything.

Symptom	Cause and fix
The G54 to G59 and tab dots are black.	The macro pump is not running. Config › General Config › Run Macro Pump, or run the installer again.
A button does absolutely nothing.	Its macro is missing from the profile, or you opened a different profile. Pick the "… TORGOS SCREEN" one and run the installer again.
"Numbers missing" or "TorgosProbe.txt not found".	Open C:\Mach3\TorgosProbe.txt and fill in the key the message names. Dot for decimals.
The measurement goes down and does not stop.	E-Stop. Mach3 does not see the probe signal. Run the SENSOR test and check the probe input in Ports & Pins: enabled, right pin, Active Low.
The measurement says the sensor already shows pressed.	The signal is inverted or the cable shorted. Flip Active Low on the input or check the cable.
The length is off by a constant amount.	Wrong z=. Repeat section 5.1.
The House does not accept the tool number.	On a machine with an automatic changer, the pocket has no HOLDER. On the ATC tab switch on the HOLDER of the pocket and press the House again.
I want my old screen back.	Open Mach3 with your original profile. Nothing has changed there.

10. Manual installation

Needed only if the installer cannot run, that is on 32-bit Windows. They are the same steps, by hand, from the zip:

1) Copy TORGOS-v2.set (or TORGOS-v2-en.set) into C:\Mach3\.
2) Copy the Bitmaps\TorgosUI folder (and TorgosUI-en for English) into C:\Mach3\Bitmaps\.

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

3) Copy the three files of Bitmaps\splash\ directly into C:\Mach3\Bitmaps\, not into a subfolder.
4) Copy the macros\ (or macros-en\) folder into your profile's macro folder, for example C:\Mach3\macros\\. Do not replace the machine's M6Start.m1s, nor an existing macropump.m1s.
5) Copy macros\TorgosProbe.txt into C:\Mach3\ and fill it in.
6) In Mach3: Config › General Config › Run Macro Pump. Then View › Load Screens › TORGOS-v2.set, and restart.

11. Support

Support is by email for 30 days from purchase and covers the installation, filling in TorgosProbe.txt and the operation of the screen buttons. It does not cover machine setup, wiring and sensors, Mach3 settings outside the screenset, changes you made inside the macros, and adapting the automatic-change functions to a machine that is not a TORGOS. For those there is the remote setup: we connect to the machine computer, fill in the values of your own machine tool and test the measurements together until they work correctly.

When you write, attach **C:\Mach3\TORGOS-egkatastasi.txt** and **C:\Mach3\TorgosProbe.txt**. They answer most questions before we ask them.

TORGOS CNC · Heraklion, Crete · torgoscnc.gr

Note. The screenset and this manual are the intellectual property of TORGOS CNC. The licence covers one business. Modifying, translating or redistributing them, by any means or tool, is not permitted without written permission.

Appendix: the screenset M-codes

These are the macros the installer puts into the new profile. If your machine already has a macro with the same number, the installer keeps yours as .bak. Tell us and we move ours to other numbers.

Macro	Button
M124	SENSOR, test that moves nothing
M127	HOLE CENTRE
M128	PART CENTRE
M129	MEASURE ALL
M131	Quick G-code test
M132	ZERO X Y with Z lifted
M142	Probe Z
M143	CORNER X0 Y0, Probe plate
M145	MEASURE ONE
M146	ZERO Z

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.

Macro	Button
M150	MAKE G-CODE
macropump.m1s	Dots, holders and indicators of the ATC tab

TORGOS CNC

© 2026 TORGOS CNC, torgoscnc.gr. Modification, translation or redistribution, by any means or tool, only with written permission.