



TNC-L32

Two-Axis CNC Lathe Controller

Standalone control of the cross slide and carriage, with a chuck encoder for synchronised threading and rigid tapping.

OPERATION MANUAL

Document No.	TC-TNC-L32-OM-001
Revision	R1.0
Product Rev.	1.0
Firmware	2.2
Issued	September 2026

THIS MANUAL CONTAINS INFORMATION FOR INSTALLING AND OPERATING THE FOLLOWING PRODUCT: TNC-L32, TWO-AXIS CNC LATHE CONTROLLER.

"Tiny Controls" and the Tiny Controls logo are copyrights of Tiny Controls Pvt. Ltd. Other trademarks, trade names and service marks owned or registered by any other company and used in this manual are the property of their respective companies.

Tiny Controls Private Limited

Office B-17/A, Nishat Park, Kakrola, New Delhi – 110078

Works Khewat 863-Killa 103/8/2(1-18), Village Barhana, Tehsil Beri, District Jhajjar, Haryana – 124107

Sales & Support +91 99111 93212 | Office +91 99111 93210 | anil@tinycontrols.com | www.tinycontrols.com

Contents

Revision History	iv
Chapter 1 — Safety Information	1
1.1 Warning Levels	1
1.2 Electrical Safety	1
1.3 Motion Safety	1
1.4 General Safety Rules	2
1.5 Emergency vs. Decelerated Stops	2
Chapter 2 — Product Overview	4
2.1 Key Features	4
2.2 How the Machine Is Configured	4
Chapter 3 — Installation and Wiring	5
3.1 Terminals	5
3.2 Axis Drives	6
3.3 Spindle	6
3.4 Chuck Encoder	7
3.5 Switches and Probe	7
3.6 Pendant	8
Chapter 4 — Quick Start	9
Chapter 5 — Technical Specifications	10
5.1 What This Product Does Not Have	11
Chapter 6 — Controls and Display	12
6.1 Screen Layout	13
6.2 Reading the Position Display	14
6.3 The Modal State Panel	14
6.4 The Hint Bar	15
6.5 Keypad Layout	15
6.6 What the Keys Do, by Screen	15
6.7 Shifted Combinations	17
6.8 Typing a Value	17
Chapter 7 — Getting the Machine Moving	18
7.1 First Motion	18
7.2 The Five Inversion Settings	18
7.3 If a Motor Loses Steps	18
7.4 Jogging	19
7.5 Establishing a Reference	19
7.6 Setting a Work Offset	19
7.7 Choosing the Work Coordinate System	20
Chapter 8 — Setting Up a Job, Start to Finish	21
8.1 Home the Machine	21
8.2 Choose the Work Offset	21
8.3 Establish X Against the Work	21
8.4 Establish Z Against the Face	22
8.5 Load the Program	22

8.6 Check the Plot Against the Part	23
8.7 Give It Lead-In Space.....	23
8.8 Park the Tool Where the First Move Can Reach It Safely	23
8.9 Dry Run First	23
8.10 Cut.....	23
Chapter 9 — Tools.....	24
9.1 The Tool Table	24
9.2 Editing the Table	25
9.3 Qualifying a Tool	25
9.4 Tool Changes and the Z Retract	26
9.5 Loads But Will Not Run	27
Chapter 10 — Files.....	28
Chapter 11 — G-code Reference	30
11.1 Motion.....	30
11.2 Modal Groups.....	30
11.3 Canned Cycles	30
11.4 Peck Drilling — G83 and G74	31
11.5 Rigid Tapping — G84.....	32
11.6 M Codes	33
11.7 Nose Radius Compensation	34
11.8 What Is Refused at Load	35
Chapter 12 — When the Control Disagrees With Your Program	37
Chapter 13 — Viewing, Editing, Manual Entry and Macros	38
13.1 Viewing and Editing a Program	38
13.2 Manual Entry	39
13.3 Macros.....	39
Chapter 14 — Watching a Job Run.....	41
14.1 Reading the Toolpath Plot.....	41
14.2 The G-code View.....	42
14.3 The Work Coordinate System and the Plot.....	42
Chapter 15 — Running a Program.....	44
15.1 Overrides	44
Chapter 16 — Threading	45
16.1 Index Registration	45
16.2 Lead-In	45
16.3 Run-Out.....	46
16.4 Verifying Threading Without Cutting	46
Chapter 17 — Commissioning the Threading Sync.....	48
17.1 Prove the Encoder Is Counting	48
17.2 Prove the Speed Reading	48
17.3 Prove the Index	48
17.4 One Turn, One Pitch, by Hand	48
17.5 Two Passes Must Register	49
17.6 Dry Run at Speed	49
17.7 Cut Air, Then Cut Metal.....	49
Chapter 18 — Commissioning a Machine	51
Chapter 19 — The Pendant.....	52

Chapter 20 — Updating the Firmware.....	53
20.1 What Happens at Every Power-On	53
20.2 Making a Stick	53
20.3 What It Refuses, and Why	53
20.4 If an Update Is Interrupted	53
20.5 The Tool Table After an Update	54
Chapter 21 — Settings Reference	55
21.1 How the Menu Is Organised.....	55
21.2 Changing a Value	58
21.3 Controller	59
21.4 Machine — General and Axes	62
21.5 Machine — Spindle	65
21.6 Machine — Homing.....	66
21.7 Machine — Limits, E-Stop and Toolpost.....	68
21.8 Machine — PLC	69
21.9 Machine — Threading	71
21.10 Machine — Probe.....	72
21.11 System	73
21.12 Where Settings Live	75
21.13 What Survives a Power Cycle	75
21.14 Two Settings That Are Easily Confused.....	75
Chapter 22 — Messages and Alarms	76
22.1 Alarms	76
22.2 Homing and Reference	76
22.3 Loading and Running	77
22.4 Settings and Storage.....	77
Chapter 23 — Warranty and Support.....	78
23.1 Warranty.....	78
23.2 Exclusions	78
23.3 Returns.....	78
23.4 Support.....	78
Appendix A — Connection Diagram.....	79
Appendix B — Integrator Technical Specifications.....	81
B.1 Step Rate Ceiling	81
B.2 Threading Follower Limits.....	82
B.3 Tapping Depth Overshoot.....	82
B.4 Other Enforced Limits	82
B.5 On-Card Files.....	83
Appendix C — Complete Settings Menu Tree.....	84
Appendix D — Quick Reference Card.....	86
D.1 Cockpit Keys.....	86
D.2 Shifted Combinations.....	86
D.3 Before the First Cut.....	86
D.4 Three Things That Bite	87

Revision History

Revision	Date	Timestamp	Sections affected	Summary of change
R1.0	September 2026	2026-09-18	All	First issue.

Chapter 1 — Safety Information

Read this chapter before wiring, configuring or running a machine on the TNC-L32. A lathe turns an unguarded workpiece at speed within reach of whoever is standing at it.

1.1 Warning Levels

Level	Meaning
DANGER	An imminently hazardous situation which, if not avoided, will result in death or serious injury.
WARNING	A potentially hazardous situation which, if not avoided, could result in death or serious injury.
CAUTION	A situation which, if not avoided, may result in minor injury or damage to equipment.
NOTE	Information needed for correct operation. No injury risk.

1.2 Electrical Safety

WARNING

Disconnect the supply before making or changing any wiring, and before connecting or disconnecting the pendant. The pendant cable carries the display and keypad link and must not be hot-plugged.

WARNING

Wire an emergency-stop switch that removes power at the drives, and test it before every session. The controller stops motion and drops the spindle enable when its emergency-stop input opens, but a control input is not a substitute for a chain that removes drive power.

CAUTION

Set PENDANT ESTOP to PRESENT only when the pendant emergency stop is actually wired. Claiming an emergency stop the machine does not have changes what you expect the machine to do.

1.3 Motion Safety

DANGER

Never change a tool, measure the work or reach near the chuck while the spindle is turning. Stop the spindle and wait for it to come to rest.

DANGER

Check that the workpiece is held properly and every guard is in place before starting the spindle or a program. On a lathe the work is the part that is moving.

WARNING

Soft limits ship switched off. Until they are set and turned on in MACHINE → LIMITS & E-STOP, nothing in the control stops a program driving the cross slide into the chuck.

Soft limits are machine coordinates and do nothing on an axis that has not been referenced. An unreferenced machine has no travel protection of any kind.

WARNING

The control does not reposition before the first line of a program. The first motion block runs from wherever the slides are standing to its own endpoint, in its own mode — a rapid if that line is a G0. Jog clear in X and Z before pressing Start.

WARNING

A threading pass waits for the spindle index before it begins. A pass that appears to have hung is almost always a spindle that is not turning: it will start the moment the chuck does.

WARNING

Pause is deferred inside a synchronised cycle — a threading pass, a G76 or a tapping cycle. The panel reads HOLD AFTER THIS PASS and the machine keeps cutting for up to one more pass. Section 11.5 gives the reason.

Stop is not deferred. It halts the axes immediately while the spindle coasts, which on an engaged tap will strain it and may snap it. Stop remains the right key in an emergency.

CAUTION

Homing ships switched off. Prove every home and limit switch by hand on SYSTEM → Diagnostics → INPUTS before enabling a cycle that deliberately drives an axis into one.

1.4 General Safety Rules

- Only trained personnel may install, configure or operate this controller.
- Reference the machine before running any program. A program will not start until both axes are referenced, and this is why.
- Pause requests a feed hold from every screen without exception.
- Never bypass a guard, a limit switch or the emergency-stop chain to finish a job.
- Never leave a running program unattended.
- Dry-run a new program with the tool clear of the work, and read the toolpath plot, before cutting.
- Keep this manual with the machine.

1.5 Emergency vs. Decelerated Stops

- Stop aborts motion and stops the spindle from any screen whenever the machine is moving, running a program or running a macro. Only once everything is idle does it revert to meaning Esc.
- Every abort is a decelerated stop — Stop, Esc in MDI, a macro aborting, a stalled spindle, a homing abort after the fast pass. The axes come to rest along the path they were on, at the axis

acceleration, so they stay on the programmed line and no step is lost: the DRO is still right afterwards.

- It takes a fraction of a second; until the axes are at rest a new move, a jog or a datum is refused with `STOPPING - TRY AGAIN`.
- Three things still stop dead, deliberately: an E-stop, a hard limit, and the fast pass of a homing cycle, whose switch may be closer than a stopping distance. A probe or home switch tripping also stops at once — that is the probe working, not an abort.

Chapter 2 — Product Overview

The TNC-L32 is a standalone two-axis CNC lathe controller. It drives X, the cross slide, and Z, the carriage, as step-and-direction axes, and commands the spindle either as an analog 0–10 V reference to a variable-frequency drive or as a third step channel. A chuck encoder gives it measured spindle speed and the angular reference that synchronised threading is built on.

Everything needed to set up and run a job is on the controller itself — configuration, tool table, file browser, program editor, manual data input, toolpath plot and diagnostics. No PC is required at the machine.

2.1 Key Features

- Two linear axes, X cross slide and Z carriage, each with independent steps per millimetre, rates, acceleration, backlash and soft limits.
- Spindle driven as an analog 0–10 V reference, or as a step channel the controller commands directly. One setting selects which.
- Chuck encoder feedback giving measured spindle speed alongside the commanded value, and the index registration that threading depends on.
- Synchronised threading — straight passes and the G76 multi-pass cycle — with every pass registered to the spindle index, so each pass cuts the same helix.
- Rigid tapping, synchronised in either direction, on a machine whose spindle is driven as a step channel.
- Canned cycles that expand into ordinary moves before the job runs, so every generated pass appears on the toolpath plot rather than as a single straight line.
- Nose radius compensation for tapers and radii.
- 800 × 480 colour display carrying a live toolpath plot, a G-code listing that follows the machine, and a modal-state panel read from the parser.
- 24-key keypad, each key carrying a printed function, a character and a shifted character.
- Sixteen tool slots with tip orientation, nose radius, qualified geometry and live wear correction.
- Macro language for tool changes, park routines and touch-off, with branching, arithmetic, probing and input waits.
- SD card for programs and settings, USB host for transfer, and firmware update from a USB stick at power-on.
- Pendant with a handwheel that jogs on the cockpit and scrolls every list screen.

2.2 How the Machine Is Configured

Three things are configured separately. Knowing which is which saves a great deal of hunting, and it decides what can be copied from one machine to another.

Holds	What it is for
Controller settings	How this unit behaves — display, units, key timing, jog rates. Nothing here depends on the lathe it is bolted to.
Machine settings	What the lathe is — axes, spindle, homing, limits, toolpost, probe. Kept in a second file, so a second identical machine is set up by copying this one alone.
Tool table	Where each tip sits relative to the master tool, and how worn it is. Sixteen slots, kept independently of the machine settings.

Chapter 3 — Installation and Wiring

This chapter covers what the controller expects to be connected to it, and which setting describes each connection. Terminal-by-terminal assignments and connector detail are in the hardware reference supplied with the board and are deliberately not repeated here, because two copies of a pinout is how an installer comes to wire from the wrong one.

3.1 Terminals

Every terminal is labelled on the face of the unit. Appendix A, Connection Diagram, carries the full-size drawing; the table below names the same terminals in text, so they can be searched and quoted.

Terminal	Function
24V-IN, GND	Supply input.
24V-Out, 5v-Out, GND	Auxiliary supply outputs for sensors and switches.
X-Pulse +/–, X-Dir +/–	Differential step and direction to the X drive.
Z-Pulse +/–, Z-Dir +/–	Differential step and direction to the Z drive.
S-Pulse +/–, S-Dir +/–	Differential step and direction to the spindle drive, used when the spindle is set to step drive.
0~10V-Out, 0V-In, 10V-In	Analog spindle speed reference and its isolated supply, used when the spindle is set to analog drive.
ENC-A +/–, ENC-B +/–	Chuck encoder, two differential channels.
SpEn-NO, SpEn-Com	Spindle enable relay contact.
M7-NO, M7-Com	Coolant relay contact.
Out-0(H), Out-0(oe), Out-0(oc), Out-1 to Out-4	General-purpose outputs, addressed by the digital-output M codes (section 11.6, M Codes).
X-Home, Z-Home	Home switch input per axis.
All-Limits	Limit switch input, shared by both axes.
Probe	Probe or tool-setter input.
E-Stop	Emergency-stop input.
Start, Stop	External cycle start and stop inputs.
Misc-In	General-purpose input.

WARNING

The limit switches share one input. The controller is told that a limit has tripped, not which axis tripped it, so both limit switches must be wired in series into All-Limits and the axis is identified by looking at the machine.

Three connectors sit on the left-hand end: the pendant, the serial port, and the USB host used for file transfer and firmware update.

3.2 Axis Drives

X and Z are driven as step-and-direction pairs. Two settings describe the timing the drive needs, and both matter more than they look.

Setting	What it describes
PULSE WIDTH	How long the step pulse is held. Opto-isolated drive inputs commonly need 5 μ s and several ask for 7.5 or 10; differential line receivers are content with 1 to 2 μ s.
DIR SETUP	How long the direction signal settles before the step edge the drive latches on. An optocoupler needs microseconds here, not nanoseconds.

NOTE

The two times add, and their sum sets the highest step rate the controller can produce, because one pulse must fit inside half a step interval. Raising either lowers the ceiling. SYSTEM → Diagnostics shows the pair and the resulting rate; Appendix B, Integrator Technical Specifications, gives the relationship.

If an axis moves the wrong way, invert it with X DIR INVERT or Z DIR INVERT rather than compensating in G-code. That setting flips the direction signal at the pin, below everything else, so jogging, programs, homing, probing and threading all see the same change at the same instant.

3.3 Spindle

Set DRIVE to match how the spindle is wired.

Mode	Wiring and setup
ANALOG	An isolated 0–10 V reference to a variable-frequency drive. Set ANALOG SPAN to the speed the drive reaches at full scale — this is the field to reach for when the commanded and measured speeds differ by a constant ratio. EN RELAY sets the enable relay contact sense.
STEP	The controller emits the pulse train itself on the spindle step output, with direction on its own pin. STEPS/REV says what one pulse means and SPIN ACCEL limits how fast the commanded speed may change. Rigid tapping requires this mode.

The enable relay is the same contact in both modes, and the encoder stays in charge of threading in both: Z follows the measured spindle, so if the spindle lags in a heavy cut the pitch stays correct.

WARNING

The chuck encoder is required for threading in both modes. Without it the controller has no angular reference to synchronise the carriage to, and a thread cannot be cut.

Step drive adds a stall check, and it is the reason to want it. With a variable-frequency drive the commanded and measured speeds disagree constantly and legitimately, so divergence tells you nothing. With a commanded pulse train the controller knows exactly what it emitted, so a measurement that does not follow means the drive is not following.

DANGER

A stalled spindle reads identically to a stopped one, and a feed-per-revolution cut then advances Z against a chuck that is not turning. On step drive the controller catches this: a divergence of more than 25%, or 60 rpm, held for half a second while a program is running aborts the job and names both the commanded and the measured speed. Where the encoder reads nothing at all it reports no spindle feedback instead, which is a different fault with a different fix. Below 30 rpm nothing is checked, because the ratio is meaningless there.

NOTE

PULSE WIDTH does not apply to the spindle. The spindle train is a square wave at whatever frequency the speed calls for. That setting is sized for the axes, which are opto-isolated step-and-direction drives needing microseconds; a step spindle is a servo with a differential input that wants clean edges. The controller imposes no speed ceiling of its own here — the top end is bounded by the drive's maximum input frequency, from its datasheet, and SYSTEM → Diagnostics reports the train frequency in kilohertz for exactly that comparison.

3.4 Chuck Encoder

The encoder is decoded on both edges of one channel, with the second channel read as a direction level, so a 5,000-count encoder delivers 10,000 counts per turn rather than its nameplate figure. Two settings describe it, and they are a ratio on purpose.

Setting	Means
ENC CNT/IDX	Counts between one index and the next.
ENC IDX REVS	How many chuck revolutions those counts span, 1 to 16.

A belted encoder is why this is a ratio rather than a single number. A 5,000-count encoder driven 56:84 gives 3,333.33 counts per chuck turn, which no whole number holds, and rounding to 3,333 drifts a thread visibly over 25 mm. Entered as 10,000 counts spanning 3 revolutions the ratio is exact and the arithmetic never rounds. A directly coupled encoder is one revolution.

NOTE

There is no separate index channel on the connector. The index is derived from the count itself, which is why the count has to be right before the index can be: a wrong count puts the index at an arbitrary angle. Chapter 17, Commissioning the Threading Sync, walks through proving both.

3.5 Switches and Probe

Home switches, the probe and the limit inputs are read the same way electrically. What varies is whether the contact behind the input rests open or closed, and there is a setting for each.

Setting	Set it to INVERT when
HOMING → SWITCH INV	The home switches are normally closed.
PROBE → PROBE INVERT	The probe contact is normally closed.
SPINDLE → ENC	The chuck encoder counts the wrong way.

Setting	Set it to INVERT when
INVERT	
SPINDLE → EN RELAY	The spindle enable relay contact is normally closed. Set it to NC.
AXES → X / Z DIR INVERT	The motor turns the wrong way.

CAUTION

Get the probe sense wrong and the input reads as permanently touched, so a probing cycle refuses to start. Confirm every switch shows ACTIVE on SYSTEM → Diagnostics → INPUTS when you trip it by hand before relying on it.

3.6 Pendant

The pendant connects by a single cable and carries its own display, six keys and a handwheel. Its contrast and bias ratio are set from CONTROLLER → PENDANT on the main panel and are pushed to it again whenever it re-identifies, so a pendant that browns out comes back on your settings rather than its own.

Chapter 4 — Quick Start

This chapter takes a wired and powered machine to a first cut. Every step names the chapter that covers it in full; nothing here is a second source of truth for a value.

1. **Power on.** The top bar reads NOT REFERENCED, which is normal and is not an error (Chapter 6, Controls and Display).
2. **Set the steps per millimetre** for both axes in MACHINE → AXES. Nothing else matters until these are right, because every distance the machine moves is this number (section 21.4, Machine — General and Axes).
3. **Jog each axis** and measure what it actually moved. A wrong distance means steps per millimetre; a wrong direction means the axis direction inversion (Chapter 7, Getting the Machine Moving).
4. **Set the spindle maximum speed** and, on analog drive, the analog span. Command a mid speed and check that the S card agrees (section 21.5, Machine — Spindle).
5. **Set the toolpost**, front or rear, and the tool count to the tools you actually have. Get the toolpost right first: it mirrors every X direction on the screen (section 21.7, Machine — Limits, E-Stop and Toolpost).
6. **Qualify T1** as the master tool at zero, then every other tool as a difference from it (Chapter 9, Tools).
7. **Touch off the part.** Skim a diameter, tap X-Zero and type the measured size; face the end and hold Z-Zero (Chapter 8, Setting Up a Job, Start to Finish).
8. **Load a program** from the card and read the toolpath plot, which draws the part coloured per tool (Chapter 14, Watching a Job Run).
9. **Dry-run it once** with the tool clear, then press Start (Chapter 15, Running a Program).

NOTE

What new operators get wrong.

The large number on each axis card is the work coordinate and the small one is the machine coordinate. Confusing them is the commonest way to crash a lathe.

Soft limits and homing both ship switched off, so a new machine has no travel protection until they are set (Chapter 1, Safety Information).

The first block of a program runs from wherever the slides are standing. Jog clear before pressing Start.

Chapter 5 — Technical Specifications

AXES 2	TOOL SLOTS 16	DISPLAY 800 × 480	PROGRAM 262,144 moves
------------------	-------------------------	-----------------------------	---------------------------------

Ranges below are the limits the controller enforces on each setting. The usable range on any given machine is narrower, and is set by its mechanics and drives.

Parameter	Value
Axes	Two — X cross slide, Z carriage
Axis command	Step and direction per axis
Spindle command	Isolated 0–10 V analog reference, or a step channel
Spindle feedback	Quadrature chuck encoder, decoded on both edges of one channel
Step pulse width	1 to 20 μ s, default 5 μ s
Direction setup time	1 to 20 μ s, default 8 μ s
Steps per millimetre	1 to 100,000 per axis
Maximum feed rate	1 to 60,000 mm/min per axis
Acceleration	1 to 10,000 mm/s ² per axis
Backlash compensation	0 to 5 mm per axis
Spindle speed limits	Minimum-speed setting 0 to 5,000 rpm; maximum-speed setting 1 to 20,000 rpm
Encoder counts per index	4 to 65,536, with a divisor of 1 to 16 chuck revolutions for a geared encoder
Tool table	16 slots, each with tip orientation, nose radius, geometry and wear
Tool wear range	± 2 mm per axis
Program capacity	262,144 moves
Display	800 × 480 colour
Keypad	24 keys
Pendant	128 × 64 display, six keys and a handwheel
Program storage	SD card. Programs run from the card only
Transfer	USB host, for file transfer and firmware update
Settings storage	SD card, as editable text

NOTE

Derived limits — the figures that follow from a combination of settings rather than being set directly — are in Appendix B, Integrator Technical Specifications, each stated with the setting that governs it. The step-rate ceiling is the one an integrator most often needs.

5.1 What This Product Does Not Have

These absences are design decisions, and knowing them saves time looking for a feature that was never fitted.

- No rotating tool axis and no driven tooling, so there is no side drilling and no milling on the part.
- No pattern-repeat cycle. A profile is roughed and finished by the cycles in Chapter 11, G-code Reference.
- No roughing of a profile that reverses in either axis. That shape is a pocket, and a turning cycle cannot cut one; it is refused by name rather than roughed with a formula that does not apply to it.
- No probing from inside a program. The probe cycle is available from a macro (Chapter 13, Viewing, Editing, Manual Entry and Macros).
- No turret indexing protocol. Turret mode selects the offset and stops for the index to be made.
- No interference check on nose radius compensation. A feature smaller than the nose radius is not detected, which is why a compensated profile is dry-run and read from the plot first (section 11.7, Nose Radius Compensation).
- No running of programs from the USB stick. A stick is a transfer medium; a program pulled mid-cut fails the read that feeds the parser with the spindle turning.
- No settings copy held inside the unit. Settings live on the card as text, and a machine with no card in the slot starts on factory defaults (section 21.12, Where Settings Live).

Chapter 6 — Controls and Display

The controller is operated from the panel, which carries the display and the keypad, and from the pendant, which carries a smaller display, six keys and a handwheel.

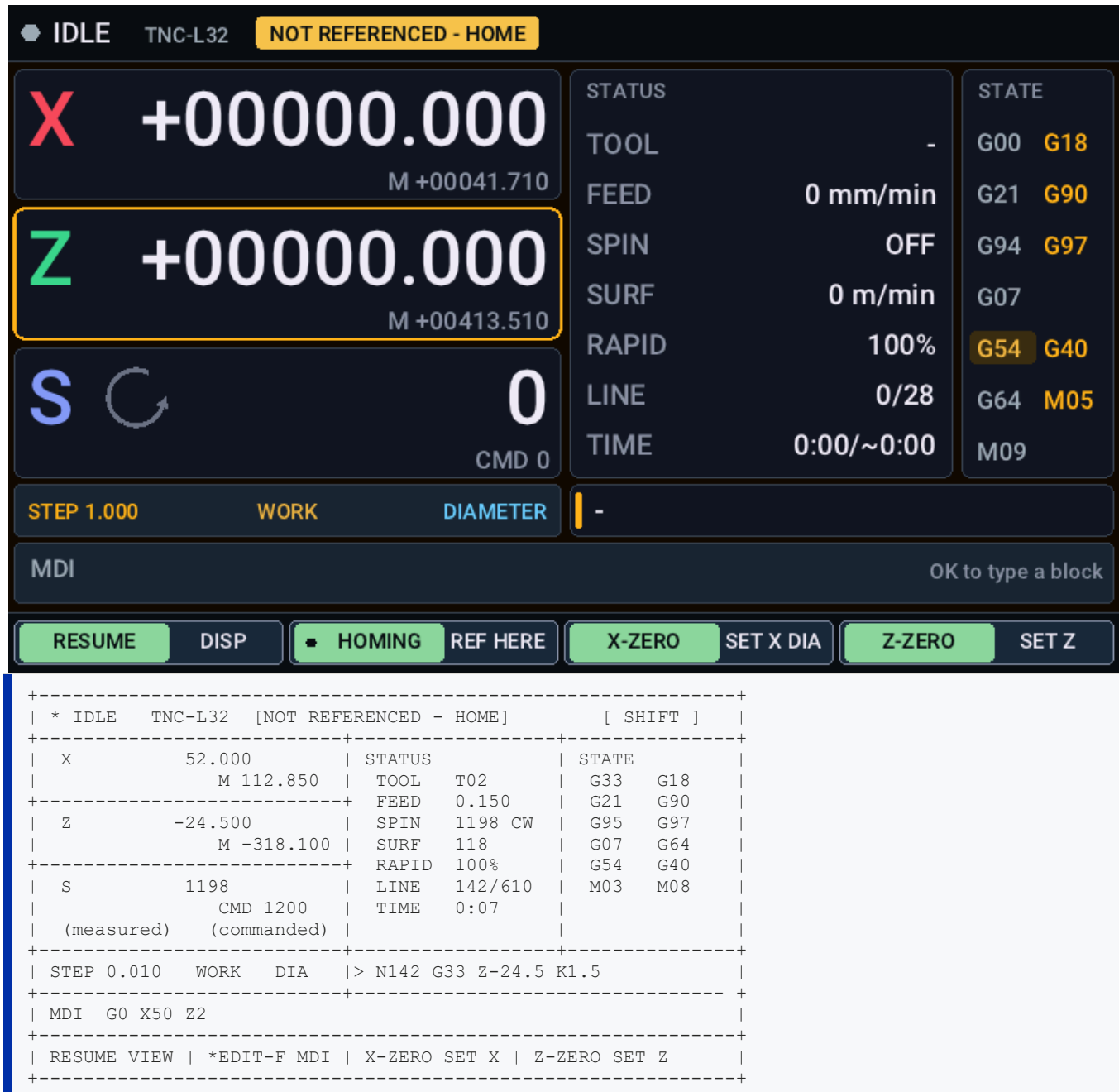
Figure 6.1 — Panel and Pendant



6.1 Screen Layout

The main screen — the cockpit — carries the axis cards on the left, machine status and the modal state on the right, and the executing block and manual-entry strip along the bottom.

Figure 6.2 — The Cockpit



The strip under the axis cards carries what the position numbers mean: the distance one jog key press will move, which coordinate frame the large numbers are in, and whether X is shown as a diameter or a radius. The bottom bar is the hint bar, described in section 6.4, The Hint Bar.

6.2 Reading the Position Display

Both coordinates are always on the card. The large number is one frame and the small number below it is the other, prefixed W or M. Tapping G53/G54 swaps which one is large, so you gain a reading rather than trading one away, and the strip reads WORK or MACHINE so the large number is never ambiguous.

X is held internally as a radius throughout. The display is the only place it becomes a diameter, and only when diameter display is switched on.

The S card reads the same way. Its large number is the measured spindle speed from the chuck encoder and the line below it is what was commanded. A chuck turned by hand still reads, which is the quickest check that the encoder is alive.

NOTE

A NOT REFERENCED chip in the top bar means the position was restored from the last power cycle but the axes have not been homed since. Those numbers are remembered, not measured, and a program will refuse to start.

6.3 The Modal State Panel

The state panel is the complete modal set, read from the parser's own record for the line being executed, so it reports what the control believes rather than what someone typed into a table.

Token	Means
G00 G01 G02 G03 G33	Motion mode. G33 is a thread pass.
G18	Plane. Fixed — this machine has one.
G20 G21	Inches or millimetres.
G90 G91	Absolute or incremental.
G94 G95	Feed per minute or feed per revolution.
G96 G97	Constant surface speed or fixed rpm.
G07 G08	X words read as diameter or as radius.
G61 G64	Exact stop or continuous path.
G54 – G59	Work coordinate system, shown in amber.
G40 G41 G42	Nose radius compensation off, left of travel, or right of travel.
M03 M04 M05	Spindle forward, reverse, stopped.
M08 M09	Coolant.

CAUTION

Read the motion token before you read the position. A G33 thread pass and a G01 feed move look identical on the axis cards and behave nothing alike — one is timed by the chuck and cannot be feed-overridden.

The compensation token is worth a glance before a taper. G40 there means the tip point is tracing the program, which is correct for diameters and faces and leaves a taper proud (section 11.7, Nose Radius Compensation).

6.4 The Hint Bar

The bar along the bottom names the actions that have no printed key of their own. A key whose cap already says what it does — Feed-Ovr, G53/G54, Spin-T — is not on it, because a chip repeating the legend under your finger wastes a quarter of the bar.

A dot before a key name means hold it. X-Zero and Z-Zero carry no dot, and the omission is deliberate: on those two it is the tap that does the unguessable thing, so the tap is what the chip names. The hold needs no chip, because the cap is printed X-Zero and holding it zeroes.

The first slot changes with the machine's state. Idle, it says Resume cycles the view; while held, it says Start is what resumes the cut. Both are keys whose printed legend does not match what they do at that moment, which is what earns a slot.

6.5 Keypad Layout

Twenty-four keys in a 6 × 4 grid. Each key carries up to three tiers: a printed function, an unshifted character, and a shifted character in blue. Shift is one-shot when tapped, applying to the next key only, and latches when held. Tapping it again clears either.

Column 1	Column 2	Column 3	Column 4
Spin-T / 1 / D	Homing / 2 / E	Resume / 3 / P	Start / 4 / Q
Limit-Ovr / 5 / I	Z-Zero / 6 / J	Pause / 7 / K	Go-Zero / 8 / H
Feed-Ovr / 9 / R	X-Zero / 0 / S	Dry-Run / . / T	Edit-File / - / L
Tools / F / M	File / G / save	G53-G54 / Shift	Settings / Spc / insL
Axis / X / delL	Stop / Esc	Cont+ / ↑	OK
Scale / Z / del	Step- / ←	Cont- / ↓	Step+ / →

The table reads as the keypad does, top row first. Stop / Esc is red-capped and OK is green. Scale, Axis, X-Zero and Z-Zero print their function labels in red, because these are the keys that establish a reference or destroy one.

NOTE

Two caps on this keypad do nothing when pressed. Limit-Ovr has nothing to override: continuous limit monitoring is not fitted, so there is no trip to drive off. Go-Zero is deliberate — on a lathe X0 is the spindle centreline, so a blind rapid to work zero would drive the tool through the part, and there is no order to move the two axes in that is safe without knowing where the tool should go first.

The four keys carrying arrows are named for what they do to an axis rather than for the arrow. Step- and Step+ move one jog step per press; Cont- and Cont+ jog continuously while held. On every screen other than the cockpit the same four move a cursor or a selection, in the direction their arrow points.

6.6 What the Keys Do, by Screen

Key	Cockpit	Elsewhere
Tools	Open the tool table	—

Key	Cockpit	Elsewhere
File	Open the file browser	Tool table: hold to save the table. Settings: hold to save the configuration
Settings	Open the settings menu	—
Stop / Esc	Abort motion, stop the spindle, clear an alarm	Back to the cockpit, or up one settings level
Cont+ / Cont-	Continuous jog while held	Move the selection, or choose a tool
Step- / Step+	One jog step per press	Move the field cursor, switch file pane, or nudge a value
OK	—	Open the value entry, cycle a list field, enter a directory, or load a program
Axis	Select the jog axis; hold to switch diameter and radius display	—
Scale	Cycle the jog step size	—
Start	Start, resume a hold, or resume a macro	—
Pause	Feed hold — works on every screen	Feed hold
Resume	Cycle the view: cockpit, toolpath, G-code	Return to the cockpit
X-Zero / Z-Zero	Tap to type what the position reads as; hold to zero it	Tool table: hold to capture that axis's geometry from the position display
G53/G54	Tap to swap the work and machine display; hold to pick G54–G59	Settings: acts as the shift modifier
Homing	Tap to home; hold to reference in place	—
Spin-T	Tap to stop the spindle; hold to start it	—
Feed-Ovr	Tap to type an override; hold to return to 100%	—
Edit-File	Hold for manual entry; tap to edit the loaded program	Files: open the file in the viewer or editor

WARNING

Up aborts a running job from every screen, whichever one is showing. With Stop and Pause, these are the three keys that never depend on what is displayed.

6.7 Shifted Combinations

A long press — press and hold — is the shifted action on a function key. Shift itself is reserved for the character tier, where a long press cannot work, because a line editor needs repeat-on-hold and the two meanings would collide on one key.

Combination	Does
Shift + Step- / Step+	Tool table: nudge the field. Settings: change the value by ten steps
Shift + OK	Files: copy the selected file to the other pane
Shift + Dry-Run	Files: delete the selected file
Shift + Scale	Any text entry: delete a character

NOTE

On the cockpit, G53/G54 is not a modifier — it is the work-offset key its legend says it is. Shift is the modifier everywhere else, which is why the cockpit has no shifted combinations.

6.8 Typing a Value

Every editable number on every screen uses the same entry: configuration fields, tool geometry and wear, the axis datum, the feed override.

It states the range, warns while you type if you are outside it, and refuses rather than clamps when you press OK. A clamp would store a number you did not type and then contradict you. Esc cancels.

NOTE

A field whose range shows no decimal places is a count rather than a measurement — tip type, tool count, work offset, feed override. Typing a fraction into one is refused rather than rounded, because a silent round to zero reads exactly like a decimal point that does not work. A whole number written with a decimal point is accepted.

Settings offering two or three choices are not typed at all. OK cycles them, which is faster and needs no keyboard.

Chapter 7 — Getting the Machine Moving

7.1 First Motion

1. Select the axis. Axis toggles between X and Z, and the selected card is outlined.
2. Select the step. Scale cycles the four jog steps — 1, 10, 100 and 1000 microns. These are fixed rather than configurable, because the pendant's labels are printed on the hardware and cannot vary. The strip under the cards shows which is active.
3. Press Step+ once. The position should move by exactly that step.

If the axis moves the wrong way, invert it in MACHINE → AXES. Do not compensate in G-code.

7.2 The Five Inversion Settings

Each describes a piece of wiring, and each takes effect the moment it is changed, with no restart and no need to re-home.

Setting	Describes
AXES → X / Z DIR INVERT	The motor turns the other way.
SPINDLE → ENC INVERT	The chuck encoder counts the other way.
SPINDLE → EN RELAY	The enable relay contact is normally closed.
PROBE → PROBE INVERT	The probe contact is normally closed.
HOMING → SWITCH INV	The home switches are normally closed.

The last two describe the contact, not the input circuit. Every input on this board reads low when driven and the controller accounts for that; these settings say whether the switch behind it rests open or closed.

7.3 If a Motor Loses Steps

Four settings, in the order worth trying. The symptom tells you which one it is.

Setting	When it is the answer
PULSE WIDTH	The drive misses pulses regardless of speed. Opto-isolated inputs are the usual case — several ask for 7.5 or 10 μ s against a default of 5. Raising it lowers the top step rate, because a pulse must fit inside half a step interval.
DIR SETUP	The axis loses position specifically on reversals — a facing pass, a threading retract — rather than at speed. That is the direction signal changing too close to the step edge the drive latches on.
SMOOTH TIME	Loss happens on direction changes and corners rather than at speed. This is jerk limiting; zero is a plain trapezoidal ramp. Try 20 to 50 ms.
JUNCTION DEV	The machine lurches through corners. Smaller is more accurate and slower.

If none of them helps, the axis is being asked for more acceleration than it has traction for. Lower the axis acceleration before reaching for anything else.

7.4 Jogging

- Step– and Step+ move one jog step per press.
- Cont– and Cont+ jog continuously while held, decelerating on release. The axis overruns the release point by its true stopping distance, which is correct and not a fault.
- The pendant handwheel gives one jog step per detent, at the pendant's own multiplier.

Jogging is refused while a program is running or held, and while the panel is showing any screen other than the cockpit.

7.5 Establishing a Reference

A program will not start until both axes are referenced. There are two ways to get there and they are not equivalent.

To home the machine, tap Homing. The configured cycle seeks at the fast rate until the switch trips, pulls off, re-seeks slowly, and sets that axis's machine zero at the switch. Axes are done one at a time, in the order set by MACHINE → HOMING → ORDER.

NOTE

Both axes seek in the positive direction by default, which is the lathe convention rather than a preference. The reference point sits at the positive extreme of each axis — X away from the spindle centreline, Z toward the tailstock — so the whole working envelope reads negative from the datum. Your switches belong at those ends.

- A second tap while the cycle is running aborts it.
- The cycle is refused unless MACHINE → HOMING → HOMING is on, and it ships off. Nothing should drive at a switch nobody has proven.
- Where it reports NO SWITCH IN nnn mm, the seek ran its whole allowance without seeing anything. Check in this order: does the switch show ACTIVE on SYSTEM → Diagnostics → INPUTS when tripped by hand; is the home direction for that axis pointing at the switch; is SEEK MAX long enough to reach it from where you started.

To reference in place, hold Homing. This declares the current position to be the reference, with no switches involved, and is what a machine with no switches uses. It establishes a datum, but not a repeatable one: it is wherever the slides happened to be.

WARNING

After a power cycle the position display comes back with the numbers it had, but the axes are not referenced and the cockpit says so. Nothing can prove the slides were not moved by hand with the power off. Home before running.

7.6 Setting a Work Offset

Two gestures on the same two keys, and the difference matters.

Gesture	Does
Tap X-Zero or Z-Zero	Type what this position should read as.
Hold X-Zero or Z-Zero	Make this position zero.

Both write the active work coordinate system, and the machine position never changes. Zeroing is the "read as 0" case of the same operation, which is why one key carries both. The typed form is how a diameter is set: tap X-Zero, type 24.5, and the display reads 24.5 at the current position.

CAUTION

The tap types and the hold zeroes, and that assignment is deliberate. Typing is what you actually do, because touching off means skim, measure, type the measurement, so the frequent gesture is the easy one. The mistakes are also not symmetric: a hold meant as a tap would silently move the datum mid-job with the tool sitting on a diameter it had just measured, scrapping the next cut, whereas the reverse slip merely opens a value box that costs an Esc. The recoverable direction gets the careless gesture.

NOTE

Where the display is showing diameters, the number you type is a diameter. The strip under the cards reads DIA or RAD.

7.7 Choosing the Work Coordinate System

- Tap G53/G54 to swap which frame the large number is in, work or machine. Both are always on the card, so the swap costs no reading.
- Hold G53/G54 to pick the offset, G54 through G59, by typing the number.

CAUTION

Choose the work coordinate system before touching off, never after. Touching off first writes the offsets into whichever system happened to be active, and the numbers you carefully established then belong to a coordinate system you are no longer using.

Chapter 8 — Setting Up a Job, Start to Finish

This is the whole sequence, in order, for a part held in a chuck with a tool already ground and mounted. It assumes the machine is configured (Chapter 18, Commissioning a Machine) and the tool table holds the tool's geometry (Chapter 9, Tools).

8.1 Home the Machine

Tap Homing and wait for both axes. The display now reads machine coordinates from the switches, and the reference gate is satisfied.

8.2 Choose the Work Offset

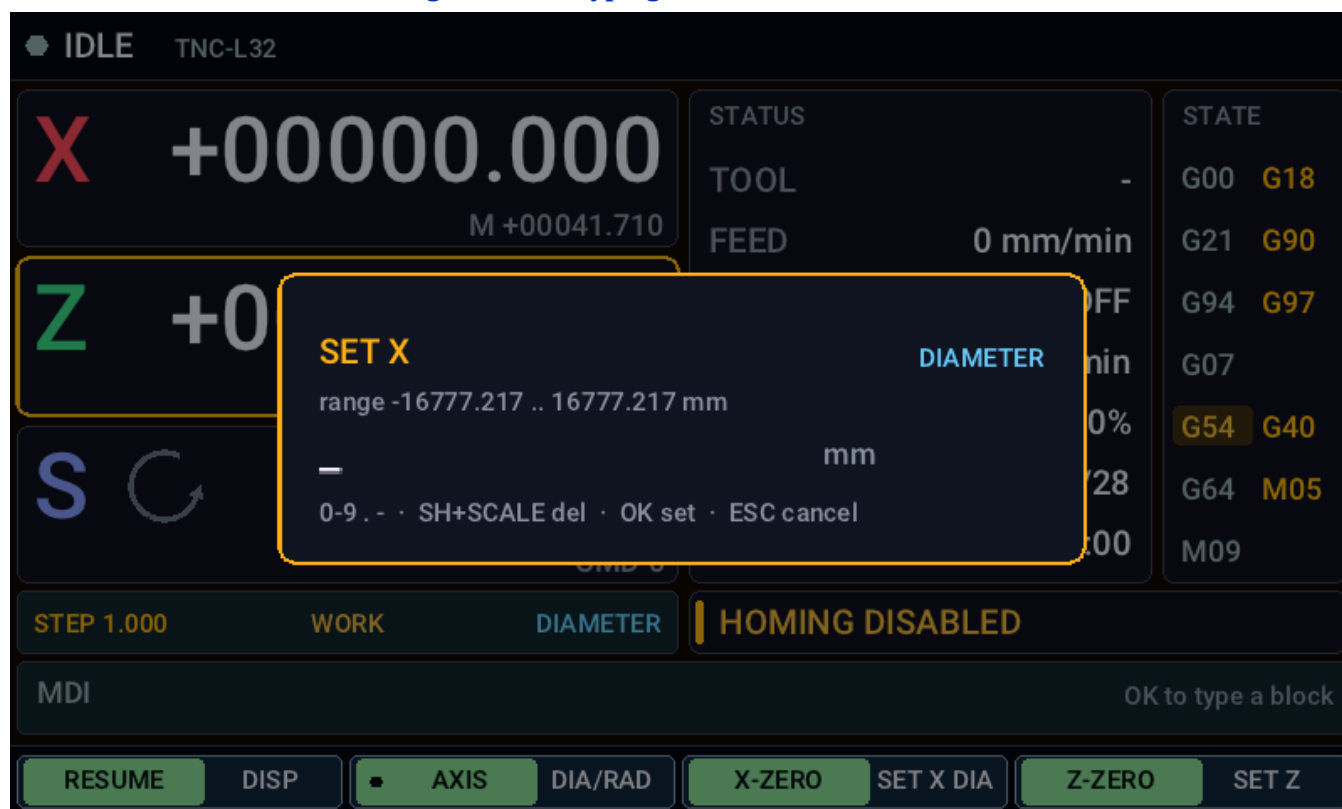
Hold G53/G54 and type the one you want, 54 for a first job. Everything below writes into that offset, so pick it before touching off.

8.3 Establish X Against the Work

X is set from a diameter you can measure, not from a guess.

1. Start the spindle — hold Spin-T, or type M3 S600 into the manual-entry strip.
2. Jog the tool to the work and take a light skim along a short length of the outside diameter. Do not move X after the cut.
3. Retract in Z only, leaving X exactly where it cut. Stop the spindle by tapping Spin-T.
4. Measure the diameter you have just turned with a micrometer.
5. Tap X-Zero, type the measured diameter, and press OK.

Figure 8.1 — Typing the Measured Diameter

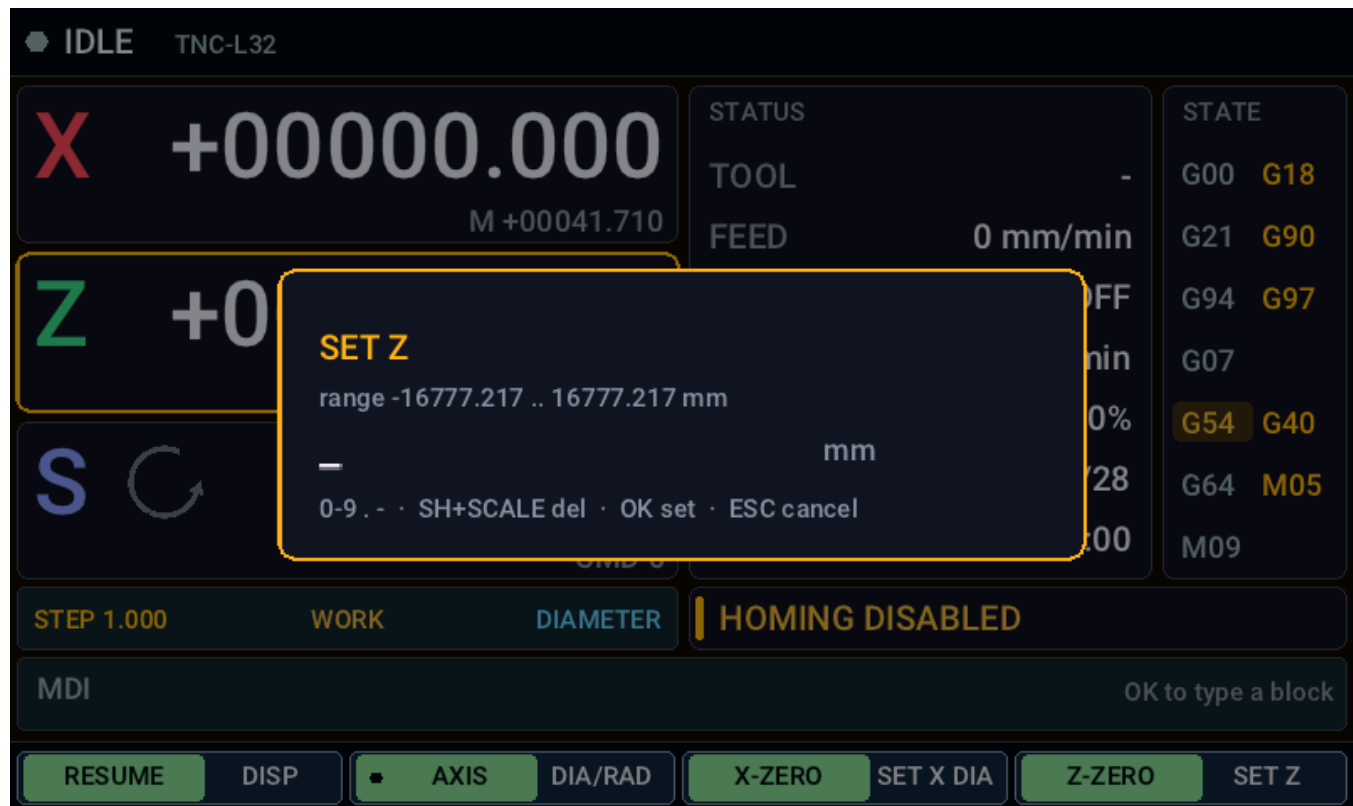


X now reads true diameter. The tool has told the control where it is by cutting, which is the only way that number can be trusted.

8.4 Establish Z Against the Face

Face the end of the part, or touch the tip to the finished face with the spindle stopped. Then, without moving Z, hold Z-Zero to make that face Z0 — or tap Z-Zero and type an offset where the program's Z0 is somewhere else, such as a shoulder or the far end.

Figure 8.2 — Typing a Z Offset



NOTE

The usual lathe convention is Z0 at the finished face, with the work extending in negative Z toward the chuck. The toolpath screen draws the chuck on the left for this reason: if your plot looks mirrored, your Z0 is at the wrong end.

8.5 Load the Program

Press File, pick the file on the SD pane, and press OK. The screen goes to the toolpath so you see what you loaded before you run it.

WARNING

Programs run from the SD card only. The USB pane browses and copies but will not load: a stick pulled mid-program fails the read that feeds the parser with the spindle turning. Copy to the card, then run.

8.6 Check the Plot Against the Part

On the toolpath screen, confirm the chuck is at the end you expect, the profile runs the way you intend, and the extents at the bottom match the stock you have. Feed moves are solid, rapids are dashed, and a thread shows its teeth.

8.7 Give It Lead-In Space

Every pass should start clear of the work and approach through air, rather than beginning its cut at the first point of contact.

- Start the tool at least 2 to 3 mm clear in Z of the face, and clear of the largest diameter in X.
- A threading pass needs more, and the controller adds most of it for you. Section 16.2 gives the distance and how it is derived.
- Leave run-out space at the far end too, especially when threading into a shoulder: the tool has to be retracted while it is still moving (section 16.3, Run-Out).

Where a program was written without lead-in, add it: open the file with Edit-File, edit the first positioning line, and hold File to save. Saving re-parses, so a mistake is caught there rather than at cycle start.

8.8 Park the Tool Where the First Move Can Reach It Safely

DANGER

The control does not reposition before line 1. The first motion block runs from wherever the slides are standing to its own endpoint, in its own mode — a rapid if line 1 is a G0, a feed if it is a G1. The approach is your responsibility. Jog clear in X and Z before you press Start.

8.9 Dry Run First

Where anything is uncertain, retract clear or lift the tool out entirely and run it once. Watch the line counter and the executing block. Press Resume to put the toolpath on screen and watch the tool trace it.

8.10 Cut

Press Start. Pause holds on the path, Start again continues, and Stop aborts. Feed override is on Feed-Ovr: tap and type a percentage, hold to snap back to 100%.

Chapter 9 — Tools

Tools opens the table. Sixteen slots are stored, independently of the machine settings, but the page draws only as many rows as TOOL COUNT — a machine with four tools should not be scrolling through sixteen.

There are two versions of the page, and which one you get comes from CHANGE MODE. A turret's tools are stations, so the count is a fact about the machine and the picture that helps is the turret face. A gang plate's tools are positions on a bar, and the picture that helps is the bar. Manual mode shares the gang page.

9.1 The Tool Table

Figure 9.1 — The Tool Table

TOOL TABLE Up-Down tool · Left-Right field · OK edit · ESC close 1/12

T	TIP TYPE	X GEOM	Z GEOM	R	X WEAR	Z WEAR
T01	3	0.000	0.000	0.800	0.000	0.000
T02	3	0.000	-1.200	0.400	0.000	0.000
T03	2	0.000	2.400	0.400	0.000	0.000
T04	3	0.000	58.400	0.400	0.000	0.000
T05	0	0.000	-0.800	0.000	0.000	0.000
T06	0	0.000	1.600	0.100	0.000	0.000
T07	4	0.000	4.200	0.400	0.000	0.000
T08	8	0.000	-2.100	0.200	0.000	0.000

T01 tip-type 3 · R0.80
insert types · FRONT
X0
MIN SAFE Z 75.000 · TC POS Z NOT SET

TIP TYPE · OK types it · SH+/- nudges 0.010 · TIP TYPE for FRONT post **UNSAVED · HOLD FILE saves**

OK TYPE **X-ZERO/Z-ZEP FROM DRO** DRY-RUN CLR WEAR FILE SAVE

Column	Meaning
T	Slot number, as a T word uses it. A turret shows a bare station number.
Colour chip	The colour this tool's cuts are drawn in on the toolpath plot.
Glyph	The tip, drawn as an oriented diamond pointing the way the nose faces.
TIP TYPE	ISO tip orientation, 0 to 8, as printed on the insert's data sheet. A whole number — it is a code, not a size.
X GEOM / Z GEOM	Qualified tip position, set once.
R	Insert nose radius in mm. This is the decimal one — 0.4, 0.8.
X WEAR / Z WEAR	Live correction, shown amber when it is not zero.

Every depiction of a tip — the row glyph, the rosette in the detail panel, the tool marker on the toolpath — comes from one source, so they cannot disagree with each other. They also mirror with TOOLPOST: the ISO chart is drawn for a rear lathe, so a front post flips every tip vertically on screen while the stored number stays exactly what the data sheet says.

Geometry and wear are added, never merged. Geometry is where the tip is; wear is the small correction dialled in after measuring the part. Clearing wear must not lose the qualification, which is why they are two numbers. Wear is clamped to ± 2 mm, because it is a correction and not a coordinate. The green bar marks the tool the running program has selected.

9.2 Editing the Table

Press	Does
Cont+ / Cont-	Choose the tool.
Step- / Step+	Move the field cursor — the underlined column.
Shift + Step- / Step+	Nudge the field by the tool nudge step, without opening anything.
OK	Open the value entry for that field.
Hold X-Zero / Z-Zero	Capture that axis's geometry from the live position.
Hold Dry-Run	Clear this tool's wear.
Hold File	Save the table.

CAUTION

The cursor starts on TIP TYPE, the leftmost column, and that is a whole-number field. Where you meant the nose radius, move right three columns to R. The value box titles itself with the field name, so check it reads NOSE R before typing a decimal.

Unsaved edits are flagged. They are live immediately, but reach permanent storage only when you hold File.

9.3 Qualifying a Tool

WARNING

Tool offsets are differences from the master tool, not machine coordinates. The work datum lives in the work coordinate system; the tool table says only how far each tip sits from the master's. The machine adds both, so entering machine coordinates as geometry counts the datum twice and every tool cuts one work offset away from where it should.

One tool — T1 by convention — is the master and stays at zero in both axes permanently. Every other tool records the difference.

For the master, once per setup:

1. Home the machine. Offsets are meaningless without a reference.
2. Leave T1's geometry and wear at zero in both axes.
3. Select T0101, skim a diameter, retract in Z only, and measure it.
4. Tap X-Zero and type the measured diameter; face the part and hold Z-Zero on the face. That sets the work coordinate system, which is where the datum belongs.

For every other tool, once each:

1. Select it and bring its tip to the same two surfaces the master was set on — the skimmed diameter and the faced end. Touch off by eye, with a shim, or with the probe.
2. Read the work coordinates off the display. With the master's datum in force, a tool sitting exactly where the master did reads X equal to the measured diameter and Z zero. Whatever it reads instead is the offset, and you enter that difference in X GEOM and Z GEOM.
3. Hold File to save.

A quick check that the whole thing is right: with the machine parked, select each tool in turn and watch the position display. It should read the tip position for whichever tool is selected, jumping by the offset as you change tools. If it does not move, the offset is not reaching the machine.

Thereafter, when a part measures oversize by 0.04 mm, put -0.04 in X WEAR rather than in the geometry.

NOTE

Wear survives a power cut even unsaved. It is held in battery-backed memory and restored at start-up, because dialling in ten microns and losing it to a brownout means measuring the part again. Geometry is not: it is a deliberate qualification and comes only from the saved table.

9.4 Tool Changes and the Z Retract

CHANGE MODE tells the control what a tool change is on your machine, and it changes what a T word does.

Mode	A T word	M6
MANUAL	Selects the offset	Stops, prompts, and waits for Start
GANG	Selects the offset, and retracts Z first	Does nothing — nothing physically moves
TURRET	Selects the offset and requests an index	Stops for the index

Gang mode inserts a move because on a gang plate the tools sit in a row along X. Switching offsets means the next move crosses the work in X, and it does so at whatever Z the last cut finished at. Nothing in the program is wrong — the traverse is a side effect of the offset the program asked for, and the program never mentions it. So the control retracts Z to TC POS Z before the offset changes. It is a real move: it appears in the toolpath plot, it has a feed-rate profile like any rapid, and it is highlighted against the T line that caused it.

MIN SAFE Z on the tools page exists because the retract has to clear the longest tool on the plate, not the one being selected. A traverse drags everything between the two tools across the work, and a 58 mm boring bar is what hits when you change between two short outside-diameter tools. So the control derives the minimum from the table — the largest Z geometry plus wear of any tool within TOOL COUNT, plus 3 mm standoff — and shows that number next to your TC POS Z, marked OK or TOO LOW.

CAUTION

The shipped table puts MIN SAFE Z at 75 mm, because TOOL COUNT defaults to 12 and slot 11 in the default template is a 72 mm drill. A gang program is refused until TC POS Z clears 75. Where your Z travel cannot reach that, reduce TOOL COUNT to the tools you actually have, or correct slot 11's Z geometry — the minimum follows the table and drops as soon as the long tool does.

Three things the retract deliberately will not do:

- TC POS Z left at zero is inert. That is the shipped default, and a retract to Z0 is a retract to nowhere in particular — on most setups it is a move toward the chuck. No retract is inserted at all.
- TC POS Z below MIN SAFE Z blocks the run and names both numbers. It is not quietly raised to the safe value, because that would send the slide to a position you never configured and cannot see in the settings.
- Nothing is inserted when the offset does not change — re-selecting the tool already cutting, or switching only the wear register — nor when Z is already at or past TC POS Z, because a retract to a Z the slide is past would drive toward the work.

TC POS X is not used by the retract. Z is what has to clear before X may traverse; where X ends up is the program's business.

9.5 Loads But Will Not Run

Some problems are not the program's fault. A T word past TOOL COUNT, or a TC POS Z that does not clear the longest tool, means this machine is not set up for this program. The file may be perfectly good and the fix is a setting.

Those programs load. The plot draws, the G-code view lists them, and Edit-File opens them. What refuses is Start, and the toolpath screen says why in amber before you get there:

```
WILL NOT RUN: T5 - MACHINE HAS 4 TOOLS - LINE 78
```

So you can look at the program, find the line, and decide whether to change the digit in the file or the count in the settings. Refusing the load instead would leave nothing on screen to work from — no listing, no editor, no plot, and no idea how far off the setup was.

A syntax error is still a load refusal. Where the parser cannot work out what a program means — a thread with no pitch, a cycle with no profile — there is no move list to plot and nothing to show, so those report on the file pane and do not load at all.

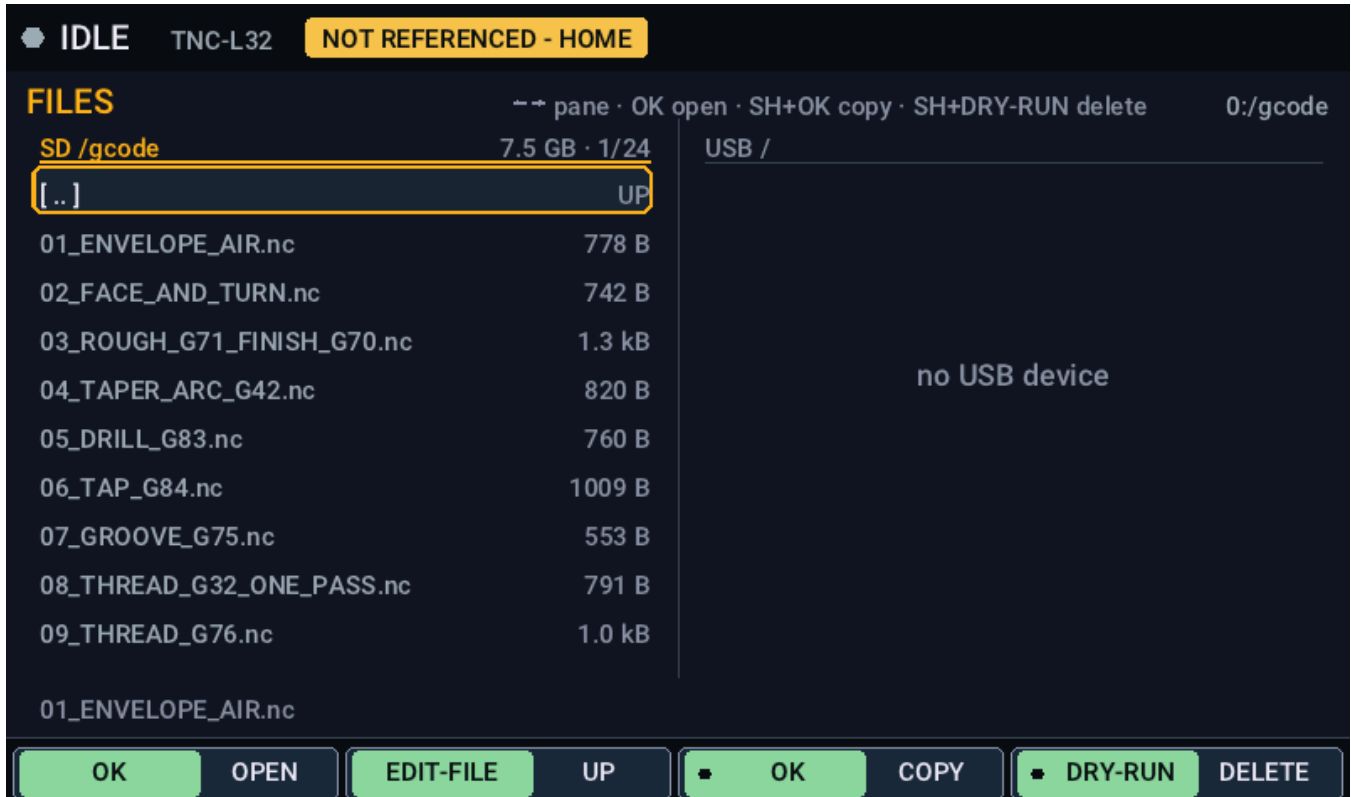
NOTE

A blocked tool plots in the master's frame. A T word naming a tool the machine does not have contributes no offset, exactly as T0 does. The alternative was to plot against the geometry of a table row the tools page will not draw.

Chapter 10 — Files

Two panes: the SD card on the left and USB on the right. Both are read when you open the screen.

Figure 10.1 — The Files Screen



Press	Does
Step- / Step+	Switch pane.
Cont+ / Cont-	Move the selection.
OK	Enter a directory, or load a program.
Edit-File	Open the file in the viewer or editor (Chapter 13, Viewing, Editing, Manual Entry and Macros).
Shift + OK	Copy the selected file to the other pane.
Shift + Dry-Run	Delete the selected file.

Folders sort first and are shown in brackets. Above the drive root there is an entry at the top marked UP, which is the way out rather than an unlabelled function key. The pane header shows which folder you are in, not just which drive.

Long file names are supported. A name longer than 63 characters is listed as not shown rather than truncated, because a truncated name cannot be opened: the path would be rebuilt from the shortened copy and miss.

Files the parser will not accept are dimmed rather than hidden. You need to see that a file is present and will not run, because hiding it looks like a failed copy.

Programs may carry any of these extensions: .nc, .ngc, .gc, .tap, .cnc, .g and .txt, plus .mac for macros. Other dialects are deliberately excluded, because they would parse into something that runs but means something else.

WARNING

Programs load from the SD card only. Pressing OK on the USB pane refuses with COPY TO SD FIRST. A stick is a transfer medium, not a run source: pull it mid-program and the read that feeds the parser fails with the spindle turning. Shift + OK copies it across.

Copying will not overwrite. A file of the same name on the destination is reported rather than replaced. Folders are refused for both copy and delete rather than recursed.

Chapter 11 — G-code Reference

Lathe conventions apply throughout: G18, the XZ plane, is the only plane; X words are diameters by default, switched with G7 and G8; and U and W are incremental X and Z.

11.1 Motion

Code	Meaning
G0	Rapid positioning.
G1	Feed move.
G2 / G3	Arc, clockwise or counter-clockwise. I and K give the centre, or R the radius.
G4 P—	Dwell, in seconds.
G28 / G30	Return to a reference position.
G32 / G33 K—	Threading pass. K is the pitch.
G53	Move in machine coordinates for this line only.

11.2 Modal Groups

Group	Codes
Plane	G18 only. G17 and G19 are refused — this is a lathe.
Units	G20 inch, G21 millimetres.
Radius and diameter	G7 diameter, G8 radius.
Distance	G90 absolute, G91 incremental.
Feed	G94 per minute, G95 per revolution.
Spindle	G96 constant surface speed, G97 fixed rpm.
Work coordinates	G54 to G59.
Path	G61 exact stop, G64 blended.
Offsets	G92 set position, G50 spindle speed clamp.
Retract	G98 initial plane, G99 R plane.

11.3 Canned Cycles

Cycles expand at parse time into ordinary moves. This is why the toolpath plot shows every pass of a cycle and the extents are right — a cycle expanded while the job ran would plot as one straight line.

Code	Does
G70	Finish the profile a roughing cycle worked from.
G71	Rough a profile, turning.

Code	Does
G72	Rough a profile, facing.
G74	Face peck grooving; chip-break peck drilling when X is omitted.
G75	Outside-diameter peck grooving.
G76	Threading cycle, constant-area infeed.
G83	Peck drilling on the centreline, full retract (section 11.4, Peck Drilling — G83 and G74).
G84	Rigid tapping on the centreline. Requires the spindle set to step drive (section 11.5, Rigid Tapping — G84).
G80	Cancel. Accepted, and does nothing — every cycle here is one-shot, so there is nothing to cancel. Programs written for a modal control carry it anyway and must not be rejected over it.

G71 and G72 name their profile by sequence number — P is the first, Q the last — so the shape is written once and G70 runs the same lines as the finish pass. Change the shape in one place and both follow.

Every cycle carries the cycle's own source line on all its moves, so the listing highlights that one line for the cycle's whole duration. That is what you want when watching a twenty-pass thread.

A cycle refuses rather than clamps when it is given:

- a profile with no motion, or one that is a single point;
- a profile that reverses in the cut axis;
- a first profile block that moves the cut axis — that block is the approach;
- a missing sequence number for P or Q;
- no depth of cut, no feed, or a depth so small the cycle would need thousands of passes. The pass count is bounded, and the bound is named.

11.4 Peck Drilling — G83 and G74

Both peck. The difference is the retract, and on a deep hole it is the difference between a cycle that works and a snapped drill.

Cycle	Retract	Right for
G83	All the way out, every peck	Anything past about three diameters deep. The flutes empty, coolant reaches the point, and the next peck starts in a clean hole.
G74 with X omitted	Backs off by R and feeds straight on	Shallow holes, and grooving, where there is nowhere for a chip to go anyway and the point is to break it. Faster.

G83 takes these words:

Word	Means
Z / W	Hole bottom. W is incremental.
Q	Peck depth, in microns — Q3000 is 3 mm, as every other cycle depth here is. Q3 is

Word	Means
	3 microns and is refused as a runaway.
R	Clearance left above the last depth on re-entry. Optional; 1 mm if omitted.
P	Dwell at the bottom, in thousandths of a second. P500 is half a second.
F	Feed. Under G95 that is millimetres per revolution, which is what a drill wants.

Two things about it are deliberate. The last millimetre into the hole is fed rather than rapided, so the drill never arrives at the bottom of an existing hole at rapid rate even where the depth bookkeeping is off by a rounding; R sets that gap. And the retract plane is wherever the tool is when the cycle starts, not Z0 — position 2 mm off the face and every retract goes to 2 mm off the face.

DANGER

G83 has no X word and X never moves. This is a face drilling cycle: position on the centreline first. The control cannot check that for you, because where the centreline lies in work coordinates depends on where you set X0. A G83 run off-centre drills an arc-bottomed slot and will probably break the drill.

11.5 Rigid Tapping — G84

Z is slaved to the spindle rather than to time, by the same follower that cuts threads, so the tap advances exactly one pitch per revolution whatever the spindle actually does. A tension holder is not needed to absorb a feed error, because there is no feed error.

Word	Means
Z / W	Hole bottom, as commanded. W is incremental.
K / F	The pitch, in millimetres per revolution.

There is no separate feed — the pitch is the feed, exactly as for a threading pass. There is no X, Q or R either: like G83 this is a centreline cycle that runs where the tool already is, and the retract plane is wherever the tap is when the cycle starts.

NOTE

G84 requires the spindle set to step drive. A program using it on an analog spindle loads, lists and plots, and then refuses to run, naming the reason: WILL NOT RUN: G84 NEEDS DRIVE = STEP - A VFD CANNOT REVERSE IN TIME. The mechanism itself would work, but a variable-frequency drive takes seconds to reverse and cannot be trusted to have done so within a revolution.

The spindle must already be turning, M3 or M4, or the file is refused at load. Either direction will do: M4 taps a left-hand thread with no other change, because the reversal is a property of the pass rather than a direction the cycle has to know. The cycle issues no M code of its own and restores the direction itself.

The hole ends up deeper than the Z you wrote. At the bottom the spindle has to stop before it can come back, and it is cutting thread the whole way down. The overshoot grows with the square of the spindle speed:

Spindle speed	Hole deeper by
100 rpm	0.17 mm

Spindle speed	Hole deeper by
200 rpm	0.67 mm
300 rpm	1.50 mm
500 rpm	4.17 mm

Those figures are for a 1.0 mm pitch at the shipped spindle acceleration of 500 rpm per second. Because the relationship is quadratic, tapping slowly is what buys the depth back: 100 to 200 rpm costs seconds and saves millimetres of blind-hole depth. Raising the spindle acceleration helps too, and is only linear in its effect, but the drive has to achieve it or the encoder disagrees and the run aborts.

CAUTION

The control does not correct for the overshoot, deliberately. Correcting means reversing early on a prediction of the drive's real deceleration: predict low and the thread is short of the depth asked for, predict high and the tap arrives at the bottom of a blind hole under power. Drill the clearance hole deep enough instead — a tapping hole is drilled deeper than the thread anyway.

Where the tap ends up is exact. The return finishes when Z has come back out by the depth it went in, measured from the commanded bottom, so the tap lands where it started whatever the overshoot was. Only the hole is deeper.

Then it waits. Coming out leaves the chuck running backwards, so the cycle inserts a dwell for it to come to rest and get back up to speed the right way — about 0.8 s at 200 rpm and 2.0 s at 500. Without it, a second tapping cycle or any threading pass would register on an index while the chuck was still turning the wrong way and drive the tap out of the hole. That is a second reason to tap slowly. Feed hold behaves differently inside a synchronised cycle. A hold requested during a tapping cycle, a G76 or a threading pass is deferred until the tool is clear: the panel reads HOLD AFTER THIS PASS, and the machine keeps cutting for up to one pass after the key.

WARNING

That is not the control ignoring you. A threading pass is emitted as one segment which the chuck times, so it cannot be decelerated part-way; an ordinary hold would take effect on the next block, which is the retract, and park the tool at thread depth in the groove. For a tap it is worse — a hold between the two halves would stop Z with the chuck already reversing against an engaged tap, which snaps it.

Stop does not wait. It is the emergency, and it stops Z while the chuck coasts, so a tap at depth will be strained and may snap. That is the correct trade for an emergency, and it is the argument for a tension holder on the first few holes even though the cycle itself does not need one.

11.6 M Codes

Code	Meaning
M0 / M1	Program stop, optional stop.
M2 / M30	Program end.
M3 / M4 / M5	Spindle clockwise, counter-clockwise, stop.
M6	Tool change (section 9.4, Tool Changes and the Z Retract).
M7 / M8 / M9	Mist, flood, coolant off.

Code	Meaning
M13 / M14	Spindle and coolant together.
M48 / M49	Feed override enable, disable.
M62 – M66	Digital output and input, P selecting pin 0 to 7.

Any other M code is skipped rather than treated as an error. An M code this control does not have is almost always an accessory it does not have, and refusing a whole program over one is worse than ignoring it — but the panel says so (Chapter 12, When the Control Disagrees With Your Program).

CAUTION

G codes are treated the other way round. An unrecognised G code carrying coordinates is a motion the program expected and will not get, so it blocks the run and names itself. Nothing crashes, because the line is simply not executed, but the work does not happen either, and a program that carried on would tap into the hole that was never drilled.

G87, side drilling, is the one a CAM post is most likely to emit: it needs a driven tool, which this machine does not have. A program carrying it loads and plots, and then refuses to run, naming the code and the line.

11.7 Nose Radius Compensation

G40 off, G41 left of travel, G42 right of travel.

Tool geometry is qualified to the imaginary tip point — the corner where the two axis-parallel tangents to the nose arc meet — because that is the point you can touch off against a face and a diameter. Driving that point along the program is exactly right on any surface parallel to an axis: a diameter comes out on size, a face comes out square, and compensation would change nothing. It is wrong everywhere else. On a taper or a radius the material is removed by whatever part of the nose arc happens to touch, and that contact point walks around the arc as the direction changes.

Nose radius	45° taper left proud by
0.2 mm	0.083 mm
0.4 mm	0.166 mm
0.8 mm	0.331 mm

The error scales with the angle and falls to zero on the straight sections either side. That is why no wear offset fixes it: wear is a constant and this error is not.

NOTE

The direction surprises people. Uncompensated, the tip point sits on the programmed surface, which puts the nose centre one radius away in each axis — further than the single radius that touching needs. The centre is too far out, so the arc never reaches the surface and the taper is left proud. Compensation drives the tool in.

Which side: G42 is right of travel. Turning an outside diameter toward the chuck travels in negative Z, so right is positive X, away from the centreline and out of the material. G42 is therefore outside-diameter turning, and G41 is boring and facing outward.

CAUTION

G42 means the same thing here whatever the toolpost is set to. On this control X is always a radius from the centreline, and TOOLPOST mirrors the picture rather than the arithmetic. On a control where X points toward you rather than away, the sense inverts, so habits carried from one are worth checking against this rule.

Put G41 or G42 on the approach, not on the first cut. The offset ramps on over the length of the move that follows the code, and comes off over the move after G40. A rapid into the work is the right place for both.

Corners are handled two ways, and both are correct. On an inner corner the path turns toward the compensated side, the offsets overlap, and they are trimmed to their intersection, so the nose centre lands one radius inside both surfaces and leaves a fillet of that radius in the internal corner. This is unavoidable with a round nose. On an outer corner the path turns away, so the nose rolls around the programmed vertex and the machine sweeps an arc about it.

Arcs cost nothing extra: they are broken into straight segments before compensation sees them, so offsetting a curve is offsetting a polyline. Facet-to-facet turns are a fraction of a degree and stay plain intersections, so only real corners grow a fillet.

Four situations load and block the run rather than being compensated quietly, so you still get the listing and the plot:

Situation	Why
G41 or G42 with no nose radius on the active tool, or no tool selected	Compensation that silently does nothing is the worst outcome, because you would believe the taper was right.
Left on across a canned cycle	A cycle generates its own path from a profile; offsetting that segment by segment is not what either was asked for.
Left on across a threading pass	A thread is cut by the form of the tool, not by the path of a nose arc.
Left on across G28 or G53	Those mean the machine coordinate, which an added radius would defeat.

In every case the fix is a G40 before it and G41 or G42 after. Compensation is then dropped for the rest of the program, deliberately, so the plot you are shown is coherently uncompensated rather than half offset.

WARNING

There is no interference check. The compensator looks one segment ahead, which is enough to settle every corner and not enough to notice that a feature is smaller than the nose radius. A 0.5 mm groove programmed for a 0.8 mm nose cannot be cut by that insert, and the offset paths cross over and the tool gouges past the far wall. On any profile with a feature near the nose radius, dry-run it and read the plot: the compensated path is what is drawn, so a crossover shows as a loop or a spike before the tool moves.

11.8 What Is Refused at Load

The parser refuses the whole program rather than running part of it when it finds:

- G17 or G19 — this is a lathe;
- a threading pass with no pitch. An F or a K satisfies it, because for a thread the pitch word is the feed;

- a tapping cycle with no pitch, with no depth, or with the spindle stopped. The cycle reverses whatever is running, so there has to be something to reverse, and a tap driven into a stationary chuck is the first thing that breaks;
- a feed move with no F ever set;
- an arc whose R is too small to reach;
- an axis word that means nothing here — Y, A, B or C;
- G73, which this control does not have;
- G38 probing inside a program. The probe cycle is reachable from a macro, and a silently skipped probe would leave the program running as though contact had been made.

Chapter 12 — When the Control Disagrees With Your Program

Everything the parser decides has to reach the panel, because there is nowhere else for it to go. There are three levels, and which one you get says how serious it is.

Level	Where it appears	What happens to the program
Refused	The files pane, on the load	Does not load — there is no move list to show.
WILL NOT RUN	The toolpath header, in amber	Loads and plots; Start refuses (section 9.5, Loads But Will Not Run).
NOTE	The toolpath header, in cyan	Loads and runs — just not exactly as written.

A note counts the rest, so a message reads like NOTE: M27 NOT SUPPORTED - SKIPPED - LINE 12 (+2 more). One is a note; several mean the program was written for another machine, and that is the case worth opening the listing for. The line number is in the message so you can go straight to it.

What raises a note:

Cause	What the control did instead
An M code it does not have	Skipped the line. M8 on a machine with no coolant pump is a dry cut.
A G code it does not have, with no coordinates	Ignored it. It is a mode the program expected, so the cut then happens under a mode nobody chose.
G76 with no parameter block	Used its own defaults for infeed angle, spring passes and minimum cut. A good thread, and not the one the program specified.
An arc given by R whose start and end coincide	Read it as a full circle and placed the centre one radius away in Z. On a lathe that is almost always a typo.

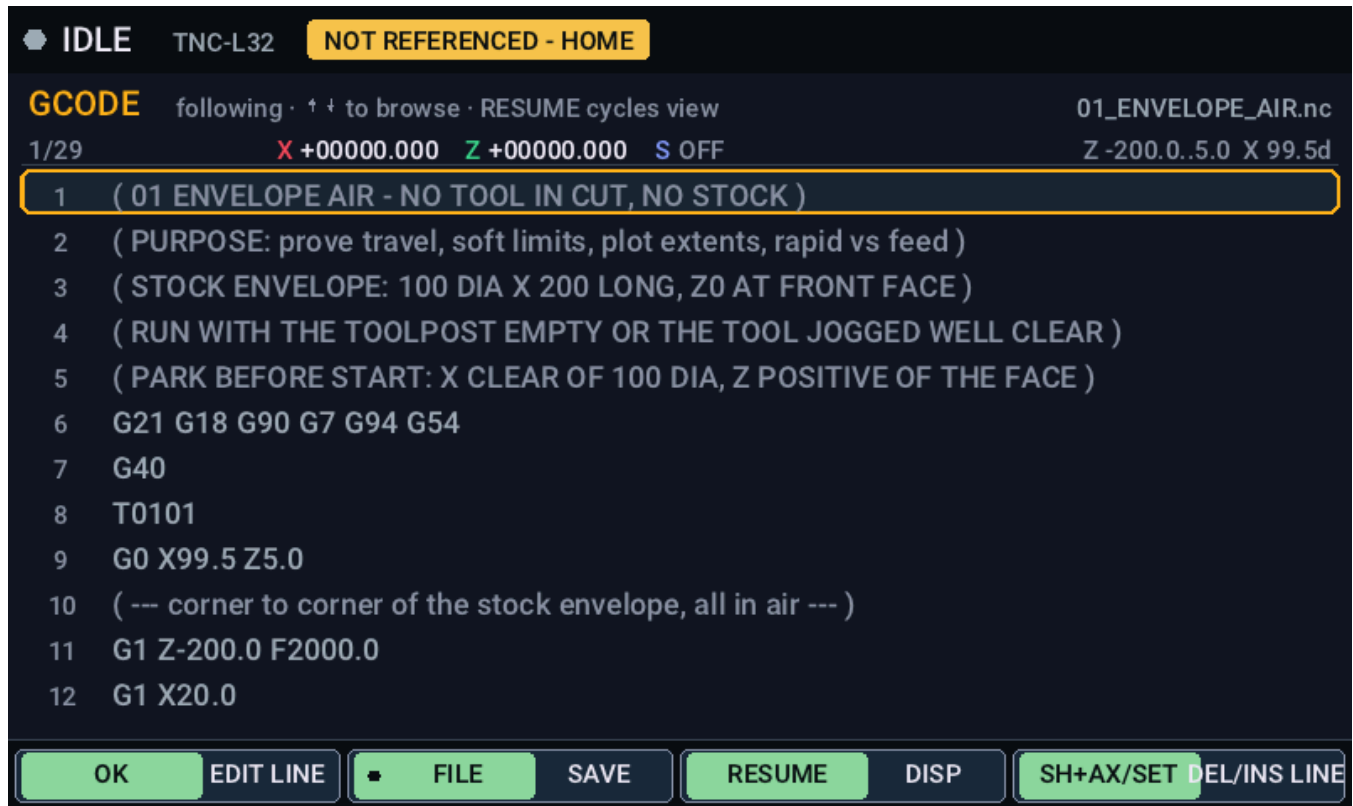
An alarm names itself too. The top bar reads ALARM and the executing-block strip says which of the four it was — emergency stop, hard limit, step queue underrun or drive fault. They want different responses, and ALARM alone sends you to check everything.

Chapter 13 — Viewing, Editing, Manual Entry and Macros

13.1 Viewing and Editing a Program

Edit-File on the files screen opens the selected file. Cont+ and Cont- move through it, and comments are dimmed so the shape of the program reads at a glance.

Figure 13.1 — The GCODE View



There are three tiers, and the differences are not cosmetic:

Tier	What it is
GCODE	The loaded program, following the machine.
EDIT	The loaded program, with the cursor yours.
VIEW	Anything else, including macros. Read only.

The title in the top left says which one you are in. The loaded program is the one with a parsed move list behind it, so it is also the only one with a toolpath to show. A file merely being viewed was never parsed; there is no move list to plot, and the OK and save chips are blank because there is nothing to edit and nothing to write.

In EDIT, OK opens the line for editing, Shift reaches the letters, Shift + Scale deletes a character, and Esc cancels the line. Hold File to save, which writes the file and re-parses it, so a line edited into something illegal is caught there rather than at cycle start. Leaving with Esc discards, and the card is untouched until you save.

NOTE

Editing is refused unless the machine is idle, a held program included. Re-parsing rebuilds the move list the planner is stepping through, and resuming replans against the blocks it was planned with. The two chips go blank and the header says the program is running, so you can see the refusal before you press anything. Stop the program, then edit.

On the cockpit, Edit-File opens the loaded program directly without walking the file browser to find it again. Esc goes back the way you came — to the files screen if you browsed there, to the cockpit if you arrived by cycling views.

13.2 Manual Entry

Hold Edit-File on the cockpit. The strip at the bottom of the screen arms: it grows an accent border and a caret, and the keypad switches to its character tier, so the machine keys underneath are inert while you type.

Shift reaches the letters and a long press on Shift latches, shown as SHIFT LOCK. OK runs the block and leaves the strip armed for the next one; Esc exits. The last block run stays on the strip.

13.3 Macros

Macros are .mac files, run from the file browser like a program but handed to the macro interpreter rather than the parser. They issue the motion commands that already exist — manual-entry blocks, spindle commands, outputs — rather than planning a toolpath. Use them for tool changes, park routines and touch-off.

Verb	Does
MSG text	Put text on the macro banner.
PROMPT text	Stop; Start continues.
CONFIRM text	Stop; Start means yes, Esc aborts.
ALARM text	Abort with a reason.
DWELL secs	Wait.
WAITIN n lvl [timeout]	Wait for input n, 1 to 8, to reach the given level.
SET #n = expr	Arithmetic into a variable.
IF expr GOTO label	Branch.
PROBE X Z target feed [ERR]	Seek until the probe input trips.
WORKZERO X Z expr	Set what this position reads as.
TOOLLEN n X Z expr	Write tool n's geometry.
END	Finish.

Anything that is not a verb is passed to the G-code parser unchanged. The variables available are the requested and current tool, the eight inputs, the last probe result in X and Z, and thirty-two general-purpose registers.

NOTE

Comments in a macro are a semicolon to end of line, not parentheses. Parentheses group expressions instead. This differs from G-code on purpose — a macro is not a G-code file.

A running macro owns the machine: Start resumes it at a prompt, and Esc aborts both the macro and the motion under it.

Chapter 14 — Watching a Job Run

Resume cycles three views of the same job, at any time, including mid-cut:

COCKPIT --> TOOLPATH --> GCODE --> COCKPIT

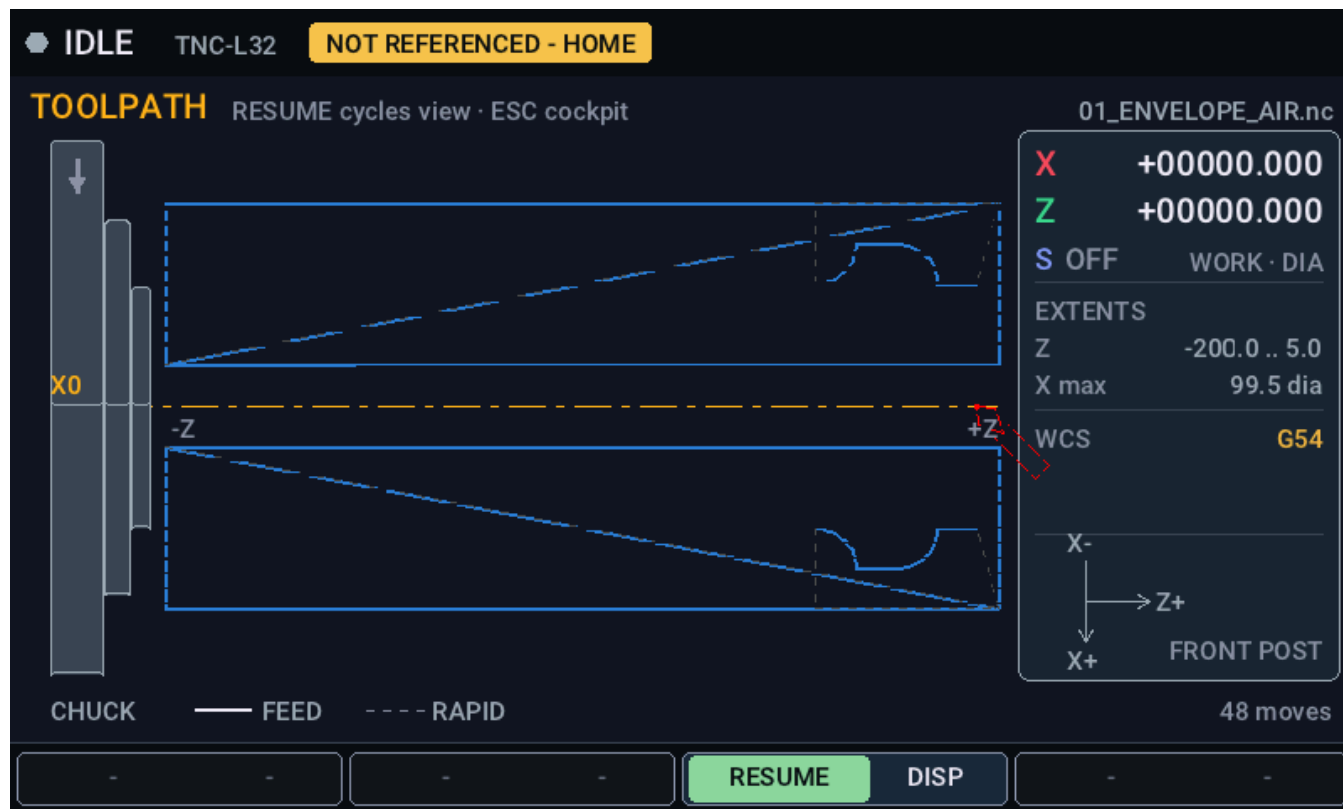
There are always three stops, loaded or not. With no program the last two say so rather than being skipped, because a key that silently drops a screen is indistinguishable from one that has stopped working.

WARNING

Pause and Stop work on all three views, and on every other screen. While anything is moving, the red key is the machine's stop wherever you are standing; it goes back to meaning Esc only once everything is idle. The keys that stop a cut must not depend on what is displayed.

14.1 Reading the Toolpath Plot

Figure 14.1 — The Toolpath Plot



The plot draws the part, not the slide. Tool offsets are taken back off before anything is plotted, so two tools cutting the same diameter land on the same line and a multi-tool program draws as one coherent part. Plotted the other way it would be several copies of the part, each displaced by its tool's offset — which is what the axes actually do, and useless for answering whether this is the program you meant. The extents panel is in the same space, so those numbers describe the part above them and are directly comparable with what you would measure.

CAUTION

The plot answers whether this is the right part. It is the soft-limit check that answers whether the slide will reach it, per axis and by name, when you press Start. The plot is not travel protection.

Line style says what kind of move it is:

Appearance	Means
Solid, thick, in the tool's colour	A cutting move.
Dashed, thin, grey	A rapid. Not cutting, so it deliberately takes no tool colour — colouring the air moves turns a readable outline back into a web of lines.
Orange	Already cut, this run.
Solid orange, thick	The block executing right now.
Faded cyan comb	A thread form. A representation of pitch and length, not the real flank geometry.

Colour says which tool, and the key is on the tools page, where each row carries a chip in its colour. There is no colour legend under the plot, because the legend band there spends its width on the line styles, which apply to every tool at once. Six colours are cycled, so a seventh tool repeats the first. The chuck is drawn at the most negative Z with a stepped jaw profile, and the centreline is a chain-dash. Both are there because a plot of bare toolpath has no orientation: without them a Z sign error and a correct program look identical. The drawing runs from the left with the chuck against the margin, because the spindle does not move and neither does its picture. Empty space on the right means the part ends there, and nothing else.

The panel down the right carries everything that is about the plot rather than part of it: a small position display in work coordinates, the same frame the plot is drawn in; the program's extents in the units being displayed; the work coordinate system and a cue when it needs one; and the axis rose with the configured toolpost. Both view screens carry the small position display — on the G-code listing it sits on the header row.

14.2 The G-code View

The executing line is marked with an amber bar down its left edge, and the window scrolls to keep it centred so you see what is coming rather than only what is happening. A green LIVE dot next to the line counter says the cursor belongs to the machine.

Press Cont+ or Cont–, or turn the handwheel, and you take the cursor back. Following stops, and the LIVE dot is replaced by a note saying which line the machine is running and whether it is above or below, so you still know where the machine is while you read elsewhere. Cycling into the view again re-arms following.

The handwheel scrolls this listing exactly as it scrolls the tool table, the file browser and the menus. It is the natural scroll control, and your hand is already on it.

14.3 The Work Coordinate System and the Plot

A G54 to G59 word in a file is resolved when the file is parsed, against whichever system was active at that moment, and the resulting shift is baked into every coordinate. The plot, and the cut, are correct only while that is still the active system.

So the control re-parses the program by itself whenever you change the work coordinate system. That is also why the change is refused while a program is running or held: re-parsing rebuilds the very move list the planner is stepping through.

Cue on the panel	Meaning
The file selects a different system, shift applied	The file asked for a different work coordinate system than the panel shows, and it has been handled correctly. Worth knowing, because the position reads in one datum while the drawing was dimensioned from another.
PLOTTED IN G54 — RELOAD THE FILE	The binding is broken. The system moved and the program could not be re-parsed — no file behind it, a manual-entry block, or the card is gone. Every coordinate is displaced by the difference between the datums and the plot shows the old geometry. Load the file again.

WARNING

The second cue is amber for a reason. Nothing else on the panel would tell you, and the machine would cut a correct-looking shape in the wrong place.

Chapter 15 — Running a Program

1. Load the program from the files screen.
2. Reference both axes.
3. Jog clear — the first block runs from where the slides stand.
4. Press Start.

Pause requests a feed hold, and the machine decelerates along the programmed path rather than straight to a stop. Start replans from the hold point and continues. Stop aborts and turns the spindle off. The status card shows tool, feed, spindle, surface speed, override, line and elapsed or estimated time. The panel beside the executing-block strip shows the block, or the reason for a hold in amber. Feed is shown in the units the program is using. A G95 block is millimetres per revolution, and displaying it as millimetres per minute would be a wrong number rather than a rounding.

15.1 Overrides

Feed-Ovr: tap opens the entry and takes 10 to 200%; hold returns to 100%. It is typed rather than stepped on purpose, because a key that stepped through a ladder would have to wrap, and one press taking the machine from 200% to 10% is a lurch nobody asked for. The percentage appears on the feed chip only when it is not 100%, so the cue keeps meaning something.

NOTE

Overrides are not remembered across a power cycle. A machine coming back at 50% feed because of where it was left last week is a trap.

Spin-T: tap stops the spindle, hold starts it. Asymmetric on purpose — stopping should never need a deliberate gesture, and starting always should.

Chapter 16 — Threading

A threading pass is not a feed move timed off the spindle. It is a position follower: Z is a function of chuck angle, and the whole pass is emitted as one move that the chuck times.

16.1 Index Registration

Every pass waits for the spindle index before it begins, so pass two starts at the same angular position as pass one. Where the chuck is not turning, the pass waits.

WARNING

A threading move that appears to have hung is almost always a stopped spindle. The pass is armed and will begin the instant the chuck turns. SYSTEM → Spindle source reports that it is waiting for the index while this is happening.

It waits for the next index after Z has come to rest at the start point, never one that has already gone by. So every pass begins from the same condition: stationary, at the index, ramping from zero. Expect a dwell of up to one revolution at the start of each pass, which is the price of registration and every control that threads by index pays it.

CAUTION

Every pass must start from the same Z. The helix is anchored to the start point at the index, so a pass beginning from a different Z is a different helix unless the difference is a whole number of pitches. Two millimetres at a 1.5 mm pitch is a third of a turn, and it will cut a separate groove. Check every start line when you change the lead-in, or drive them all from one value.

16.2 Lead-In

A thread needs three to five pitches of lead-in. The carriage has to be up to speed and locked to the spindle before the tool reaches the work, and the reason is arithmetic rather than caution.

Every pass starts with Z at rest and waits for the index, while the chuck is already at full speed. So from the first instant the chuck is asking Z for its full synchronous velocity — a 1.5 mm pitch at 500 rpm is 12.5 mm/s — and no stepper goes from standstill to that in one step. The follower therefore ramps Z at the axis acceleration, lets the chuck get ahead for the first few tenths of a millimetre, then closes the gap at a bounded rate and locks. That opening stretch is cut at the wrong pitch, so it has to land in air. Ramp plus catch-up comes to roughly twice the square of the synchronous feed divided by the acceleration:

Pitch × speed	Z ACCEL 100 mm/s ²	200 mm/s ²	400 mm/s ²
1.5 mm × 400 rpm = 600 mm/min	2.0 mm	1.0 mm	0.5 mm
1.5 mm × 570 rpm = 858 mm/min	4.1 mm	2.0 mm	1.0 mm
2.0 mm × 400 rpm = 800 mm/min	3.6 mm	1.8 mm	0.9 mm

The tool must not touch the work before that distance is up. Where the program's lead-in is shorter, the first part of the thread is cut with Z still accelerating — invisible in air, and in the cut it is where a stepper under load drops steps, a different number every pass.

The controller supplies that room for you. MACHINE → THREADING → LEAD-IN backs the start of every pass out by the distance you set, so it need not be added by hand. It is a fact about the machine

and the speed rather than about the part, which is why the control knows it and you do not have to remember it.

NOTE

The lead-in is extended by a whole number of pitches, and that is the whole trick. The follower latches Z at the index and drives the carriage from there, so moving the start shifts the entire helix by that much. Move it by a whole pitch and crests land on crests: the same thread, entered earlier. Any other amount would cut a second thread beside the first. So LEAD-IN 3 at a 1.5 mm pitch backs the start out by 3.0 mm, two pitches; LEAD-IN 2 also gives 3.0 mm, because it rounds up rather than leaving less air than you asked for.

Extending outward never cuts more material, because the start moves away from the endpoint and the extra travel is air. What it can reach is a tailstock, which is why the distance is yours to set. Soft limits check the extended move like any other.

CAUTION

This changes programs written before the setting was live. A file that already leaves 4 mm of its own now starts a further 3 mm out. The thread is identical — that is the point of whole pitches — but the pass is longer and begins further from the chuck. Set LEAD-IN to 0 to get exactly the old behaviour, where the lead-in is only what the program writes.

16.3 Run-Out

Leave run-out space too, and know how much. The pass decelerates to rest inside its own last stretch rather than stopping dead at synchronous speed, and that stretch is half the square of the feed divided by the acceleration — about 1 mm at 570 rpm, a 1.5 mm pitch and 100 mm/s². Program the threading endpoint at least that far past where the thread must be good: into a groove, or off the end. For a thread that must be good to Z-25, end the pass at Z-26.

Raising the axis acceleration shortens the lead-in and the run-out by the same ratio.

A proper pull-out — holding synchronisation while X retracts, then decelerating in air — is available on G76. Its own words ask for it per cycle, and MACHINE → THREADING → PULL-OUT supplies it where a G76 carries no parameter block. A bare threading pass cannot have one, because the parser has no way to know which diameter is clear to withdraw to, so on those, program the endpoint past the thread as above.

Neither the axis acceleration nor the axis maximum rate is a threading setting; they are the same limits every other move obeys. Raising the maximum rate does not raise the catch-up speed — the follower never exceeds one and a half times synchronous, whatever the axis allows, because asking a stepper for its unloaded maximum while it is in the cut is how steps get lost. The maximum rate only has to clear one and a quarter times synchronous, or the pass holds before it starts and says so.

16.4 Verifying Threading Without Cutting

SYSTEM → Spindle source selects where the follower takes its angle from. The same follower runs in all three cases, so what you check at the bench is what runs on the machine rather than a model of it.

Source	What it is for
ENCODER	The real chuck encoder — cutting.
MPG (hand)	The handwheel becomes the chuck. Turn it one revolution and Z must advance exactly one pitch.
DRY RUN	A virtual chuck at the speed set by DRY-RUN RPM.

The hand-generate check is the useful one, and it needs no motor. With a dial indicator on the carriage, one turn of the wheel proves sign, scale, re-registration between passes, and smoothness at the index — the four things that ruin a thread and are otherwise invisible until you cut one. MPG DET/REV is how many wheel detents make one chuck revolution. It is deliberately a setting, because a coarse wheel is a coarser master than a real encoder and exaggerates tracking error, which is exactly what this check wants.

NOTE

A synthetic source is refused unless the machine is idle, and any program start forces the source back to the encoder. The setting cannot survive into a cut, and it is not kept across a power cycle either.

Chapter 17 — Commissioning the Threading Sync

Work through these in order. Each one fails in a way the next cannot diagnose, so a skipped step costs more than it saves. Nothing here cuts metal until the last step, and the first four need no tool in the holder.

17.1 Prove the Encoder Is Counting

Open SYSTEM → Diagnostics → LINK and turn the chuck by hand. The count must move, and turning the chuck the other way must make it count down. Where it counts up both ways, one channel is dead and the encoder is reporting direction from noise.

17.2 Prove the Speed Reading

Start the spindle at a known speed and read the S card on the cockpit. The large number is the measured speed from the encoder; the line below it is what was commanded.

- Where the large number reads zero with the chuck turning, the encoder is not reaching the follower. Go back to section 17.1, Prove the Encoder Is Counting.
- Where it reads a steady multiple or fraction of the true speed, the encoder ratio is wrong. Set ENC CNT/IDX and ENC IDX REVS together, remembering that counts are after decoding, so a 5,000-count encoder delivers 10,000.

Check it against a tachometer if you have one. Everything below is scaled by this ratio, so an error here becomes a pitch error later and will look like a follower fault.

17.3 Prove the Index

Still on the diagnostics link page, turn the chuck slowly through several full revolutions. The index is derived from the count rather than from a separate pulse, so section 17.2, Prove the Speed Reading, has to be right first: where the count is wrong, the index arrives at an arbitrary angle and threading registers to a different place every pass.

NOTE

A wrong count shows up as the wrong pitch, not as a shift, and that distinction is what makes this diagnosable. The index is what every pass registers to, so all passes register consistently even when the count is wrong; what changes is how much Z advances per counted turn. A clean thread at the wrong pitch means this pair. Passes that do not land on each other means the index, or the encoder direction.

17.4 One Turn, One Pitch, by Hand

This is the check worth doing, and it needs no motor and no tool.

1. Put a dial indicator on the carriage, reading along Z.
2. Set SYSTEM → Spindle source to MPG (hand).
3. Set MACHINE → SPINDLE → MPG DET/REV to your wheel's detents per turn.
4. Load a single-pass thread file and press Start. The pass waits for the index, which under the handwheel means it waits for you to turn it.
5. Turn the wheel exactly one revolution and read the indicator.

Z must advance exactly one pitch — 1.5 mm for an M20 × 1.5.

What you see	What it means
The right distance	Sign and scale are correct.
The right distance, wrong direction	The axis direction inversion, or the thread is cutting the wrong hand.
Exactly twice or half the pitch	Decoding counted twice or not at all. ENC CNT/IDX is the count after decoding, so a 5,000-count encoder is 10,000.
A multiple that is not two	A belt ratio is missing. Set ENC IDX REVS rather than rounding ENC CNT/IDX.
Slightly out, drifting over the length	The true count per turn is fractional and was rounded. Use the ratio — both fields together.
Drifts a little each turn	The follower is not tracking. Stop and report it.

Then turn the wheel backwards. Z must retrace. A follower that only works one way will cut a thread that will not repeat on the second pass.

17.5 Two Passes Must Register

Run the same file twice without touching anything, watching the indicator each time. Pass two must start at the same angular position as pass one — that is what the index wait is for, and it is the single thing that distinguishes a thread from a helical groove beside it. Where the two passes start at different places, the index is not landing at a consistent angle; go back to section 17.3, Prove the Index.

NOTE

Where Z feels lumpy but the pitch is right, look at the chuck rather than the controller. The follower tracks the chuck faithfully by design, so a drive that cogs at low speed makes Z cog with it. Watch the measured speed on the S card: if it wanders, that is what you are feeling.

17.6 Dry Run at Speed

Set the spindle source to DRY RUN and set MACHINE → SPINDLE → DRY-RUN RPM to the speed you intend to thread at. This runs the follower against a virtual chuck with no encoder and no motor, so it exercises the timing rather than the geometry. Watch that the carriage reaches full speed within the lead-in you allowed, and that the pass ends cleanly.

NOTE

The synthetic sources resolve at about one millisecond rather than at step rate. That is enough for sign, scale and registration, which is all sections 17.4 to 17.6 are checking, but it is not a test of phase smoothness. Only the real encoder is.

17.7 Cut Air, Then Cut Metal

1. Set the spindle source back to ENCODER. A program start would do this anyway.
2. Mount the tool, set X well clear of the work, and run the file with the spindle at threading speed. Watch the carriage, not the screen.
3. Then cut a real thread in scrap, at the same speed you tested at, and gauge it.

Check the first two threads particularly. A malformed lead-in is the signature of too little run-up: the carriage was still accelerating when the tool reached the work. Add lead-in rather than reducing the speed.

Chapter 18 — Commissioning a Machine

The order matters: each step depends on the one before it, and getting the first two wrong makes everything after them meaningless.

1. Display. Set the backlight and theme so the panel is readable where the machine actually stands, then set the pendant's contrast and ratio.
2. Steps per millimetre. Command a known move from the manual-entry strip, measure what the slide actually did, and scale the setting by the commanded distance over the measured one. Do this before anything else, because every rate, offset and soft limit below is expressed in millimetres and means nothing until a millimetre is a millimetre.
3. Direction. Jog each axis one step and confirm it moves the way the display says. Where it does not, use the axis direction inversion. Never compensate in G-code.
4. Pulse width. Opto-isolated drive inputs generally want 5 μ s and several ask for 7.5 or 10; differential line receivers are content at 1 or 2. Too short and the drive misses steps at speed, and the symptom is position that drifts only on fast moves.
5. Rates and acceleration. Raise the maximum rate until the axis stalls, then back off comfortably, and do the same for acceleration. A stall under acceleration sounds like a knock rather than a squeal.
6. Soft limits. Reference the machine, jog to each hard end, read the machine position, and enter those numbers less a margin. Then turn soft limits on.
7. Spindle. Set the analog span to the speed the drive reaches at full scale, command a mid speed and compare the S card against a tachometer. Then the encoder ratio: the count is after decoding, so a 5,000-count encoder is 10,000, and any belt reduction is expressed as the ratio rather than by rounding the count.
8. Tools. Qualify each tool and record its geometry. Leave wear at zero until the first part is measured.

CAUTION

Save at each stage by holding File inside the settings menu. Nothing is written until you do, and an unsaved change is flagged in amber.

Chapter 19 — The Pendant

Seven screens on a 128 × 64 display, six keys and a handwheel.

Screen	When it appears
JOG	Idle.
RUN	A program is running.
HOLD	Feed hold, tool change, or program pause.
THREAD	A threading pass is executing.
SPINDLE	Opened with the Spindle key.
ALARM	Any alarm.
ABOUT	Firmware and link status.

The header carries the machine state, the active jog multiplier — or the tool while running — and the units or line number. The row under the position display carries the work coordinate system and a diameter or radius chip.

Key	Does
Jog Axis	Cycle the selected jog axis, X or Z.
Jog Scale	Cycle the jog step: 1, 10, 100 and 1000 microns.
X-Y-A Zero	Zero the selected jog axis.
Z Zero	Zero Z.
Toggle Spindle	Open the spindle screen; press again to start or stop.
Start Cycle	Start, hold or resume, according to the machine state.
Handwheel	Jog on the cockpit; scroll the selection on any list screen.

NOTE

The key is printed X-Y-A Zero and it zeroes whichever axis Jog Axis has selected. On this machine that is X or Z, so the key zeroes X when X is selected and Z when Z is selected — the same as Z Zero does in that case.

NOTE

Toggle Spindle needs two presses on purpose. The first shows you the speed, direction and current X; the second acts. A spindle should never start blind.

The two zero keys are dedicated caps with no second tier, so pressing a key marked zero zeroes. There is no tap-and-hold distinction to confuse, unlike the panel where one cap does both (section 7.6, Setting a Work Offset).

Contrast and ratio are per unit and set from CONTROLLER → PENDANT on the main panel. They are pushed to the pendant again whenever it re-identifies, because a pendant that browns out comes back on its own defaults rather than yours.

Chapter 20 — Updating the Firmware

20.1 What Happens at Every Power-On

A loader runs ahead of the main firmware every time the machine is powered on. It brings up the panel, starts the USB host, mounts a stick if one is fitted, and looks in the root of it for an update file. Where there is no stick, or no file, it hands over to the main firmware immediately — you will not notice it is there, and it costs a fraction of a second.

Where it does find one, it checks the file before it touches anything: that the update is for this product, for this hardware revision, that it is not older than the machine will accept, and that it is authentic.

Nothing is erased until all of that passes. Only then does it erase, program, and read every part of the new firmware back out again to verify it, before restarting into it.

It takes about twenty seconds and the phase is named as it goes — `ENTERING BOOTLOADER`, `ERASING`, `WRITING` with a progress bar, `VERIFYING`, `PROMOTING`, and finally `CONFIRMING VERSION`.

20.2 Making a Stick

- Format the stick FAT32.
- Put the update file in the root, not in a folder.
- The name must be exactly L32FWUPD.TNC, in upper case. The loader mounts without long-filename support on purpose, so a stick formatted on some machines can present a name that never matches; short upper case is what always matches.
- Leave it in and power-cycle. There is no key to hold.

20.3 What It Refuses, and Why

It refuses	Because
A file for another product	The product is checked and the signing key is per product. A file for a different controller is not a TNC-L32 file even where the board is identical.
A file that fails its authenticity check	An image that is not authentic must not be written, or even unpacked. This check happens before the first erase.
A version below the accepted floor	Downgrading past that floor is how a fixed safety defect comes back.
A file for a different hardware revision	Same firmware, different board, and silently wrong behaviour at the pins.

20.4 If an Update Is Interrupted

Power loss part-way through a write is the case the whole scheme is built around. A marker is set before the first erase and cleared only after every part of the new firmware has been read back and verified. A start-up that finds that marker set refuses to hand over to the main firmware, because there may not be one, and waits for a stick.

NOTE

The recovery is to put the stick back in and power on again. The marker is deliberately kept in permanent storage rather than in battery-backed memory, because putting it in memory would make the protection depend on the battery, which is the failure it exists to prevent. It is the USB stick specifically: the loader reads the stick, not the card.

20.5 The Tool Table After an Update

The tool table lives on the card, in the configuration folder, alongside the two settings files (Appendix B, B.5, On-Card Files).

CAUTION

A machine updated from an earlier build will not see its old tool table. Earlier firmware held the table somewhere this firmware does not read, so all sixteen slots come up as the shipped table and every qualified length and nose radius is gone. Qualify the tools once and hold File to save; the page reports SAVED - 16 tools to card. Do it before the first job after an update, not after a part has been cut to the wrong length (Chapter 9, Tools).

Chapter 21 — Settings Reference

This chapter explains the settings. Appendix C, Complete Settings Menu Tree, is the complete tree, every field with the range the editor enforces and the value it ships with, for looking a figure up rather than reading.

21.1 How the Menu Is Organised

The menu is three levels deep. The top level offers three branches; each branch holds groups; each group holds the fields. Eighty-three fields are settable in all — fourteen under CONTROLLER and sixty-nine under MACHINE — and SYSTEM holds no settings.

Figure 21.1 — SETTINGS, the Top Level

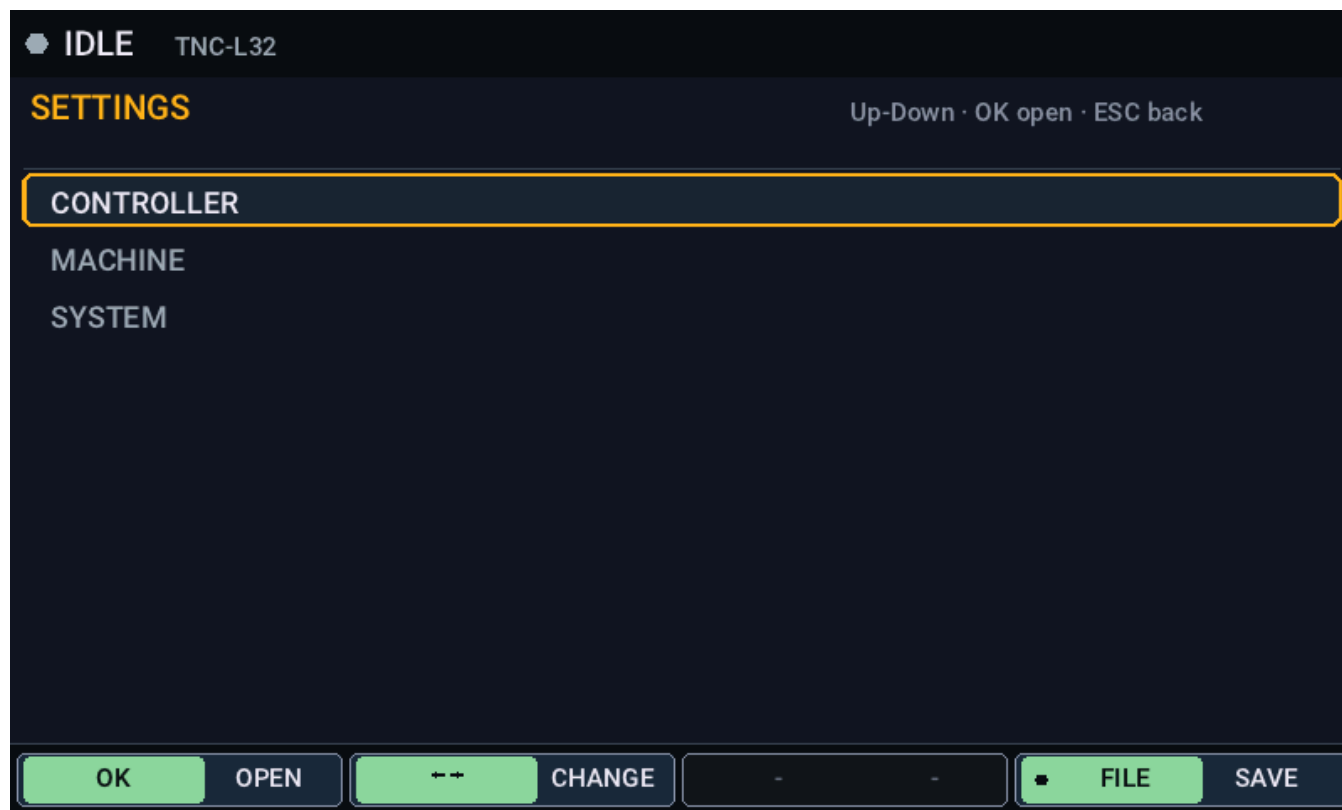


Figure 21.2 — SETTINGS, MACHINE

Branch	What it describes	Groups
CONTROLLER	How this unit behaves — what the screen looks like, what units it works in, how the keys respond, how fast it jogs. Nothing here depends on the lathe it is bolted to, so it is the branch you set once and leave alone.	5
MACHINE	What the lathe is — how far it travels, how fast it may move, what its spindle, switches and toolpost are. Every field here is a fact about the mechanics, and a wrong one is a wrong part or a crash.	9
SYSTEM	Four information and service pages: what firmware is loaded, what the inputs and outputs are doing, the pendant update page, and the spindle source used for the checks in Chapter 17, Commissioning the Threading Sync.	—

NOTE

The two settings branches are saved as two separate files on the card, which is the point of the split: a second identical lathe is set up by copying the machine file alone, without carrying the first one's screen colours across (section 21.12, Where Settings Live).

CONTROLLER holds five groups:

Group	Covers	Fields	Detail
DISPLAY	The panel's theme, highlight colour and backlight.	3	21.3

Group	Covers	Fields	Detail
	Set these first when commissioning, so everything after them is readable where the machine stands.		
PENDANT	The pendant's backlight, contrast and bias ratio. Separate from the panel because it is a different screen with different needs, and it may be stowed.	3	21.3
UNITS & FORMAT	Millimetres or inches, how many decimal places the position shows, and the two independent diameter and radius choices.	4	21.3, 21.14
KEYPAD	How long a hold gesture takes to register, and how fast a held key repeats.	2	21.3
JOG	The step-jog and continuous-jog rates. The jog step size is not here: it is operator state on the cockpit, cycled with Scale.	2	21.3

MACHINE holds nine groups:

Group	Covers	Fields	Detail
GENERAL	The drive-timing and motion-smoothing settings that are not per axis. The first two add together to set the machine's top step rate.	4	21.4
AXES	The same seven fields for X and then Z. Steps per millimetre is the field to set before any other in the whole menu, because every distance the machine moves is that number.	14	21.4
SPINDLE	The largest group: how the spindle is driven and its speed limits, the encoder ratio threading depends on, the synthetic sources used for checking, and on step drive the pulse resolution and acceleration.	15	21.5
HOMING	Whether homing is enabled, the two approach rates, the pull-off distance, which axis goes first, the switch contact sense, the seek direction per axis, and how far to look before giving up.	9	21.6
LIMITS & E-STOP	What a limit does when it trips, whether soft limits are enforced, whether jogging off a tripped switch is permitted, and whether the pendant emergency stop is wired.	5	21.7
TOOLPOST	Front or rear post, how many tools the machine has, the tool-table nudge step, what a tool change means here, and the tool-change position.	6	21.7
PLC	The serial link to a controller or turret. Off by default, and costs nothing on a machine that has none.	7	21.8
THREADING	The lead-in added ahead of every pass, and the default chamfered exit for a G76 with no parameter	5	21.9

Group	Covers	Fields	Detail
	block.		
PROBE	The probe contact sense, its approach rate, how far it backs off after contact, and whether motion that would overtravel it is refused.	4	21.10

CAUTION

Three MACHINE settings ship in a state that is safe to power up but not safe to cut with, and each is deliberate: homing is off, soft limits are off, and the tool count is 12 against a default table whose eleventh slot holds a long drill. Chapter 1, Safety Information, and section 9.4, Tool Changes and the Z Retract, give what each costs until it is set.

21.2 Changing a Value

Cont+ and Cont- move, OK descends into a group, and Esc goes back up one level. The list scrolls, so a group with more fields than fit on screen is not a group with unreachable fields.

On a field there are two gestures, and they are for different jobs.

Press	Does	Use it for
Step- / Step+	Nudge by one step of the field's own resolution — a three-decimal field moves 0.001, a whole-number field moves 1	Trimming a backlight or a backlash while watching the result
Shift + Step- / Step+	Nudge by ten steps	Getting across a range without a hundred presses
OK	Cycles an on-off or list field; opens the value entry on a number	A steps per millimetre you already know — type it rather than nudge to it

The value box refuses rather than adjusts. Out of range, or a decimal on a whole-number field, and it says so while you type; OK does nothing until you fix it. Esc is the way out.

NOTE

Changes apply immediately. Theme, accent, backlight and direction inversion all take effect as you turn them, with no save and no restart. Saving is about surviving a power cycle, not about taking effect.

21.3 Controller

Figure 21.3 — SETTINGS, CONTROLLER

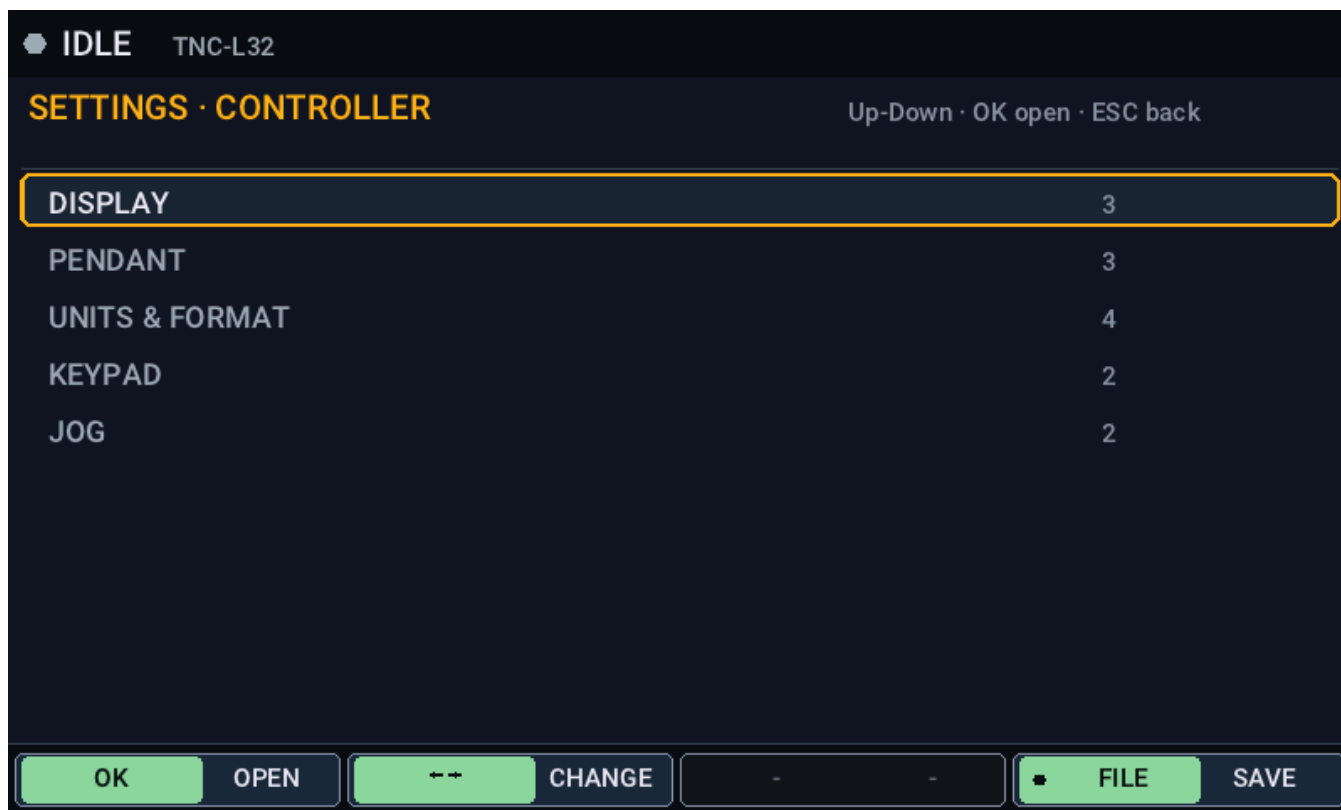


Figure 21.4 — CONTROLLER, DISPLAY

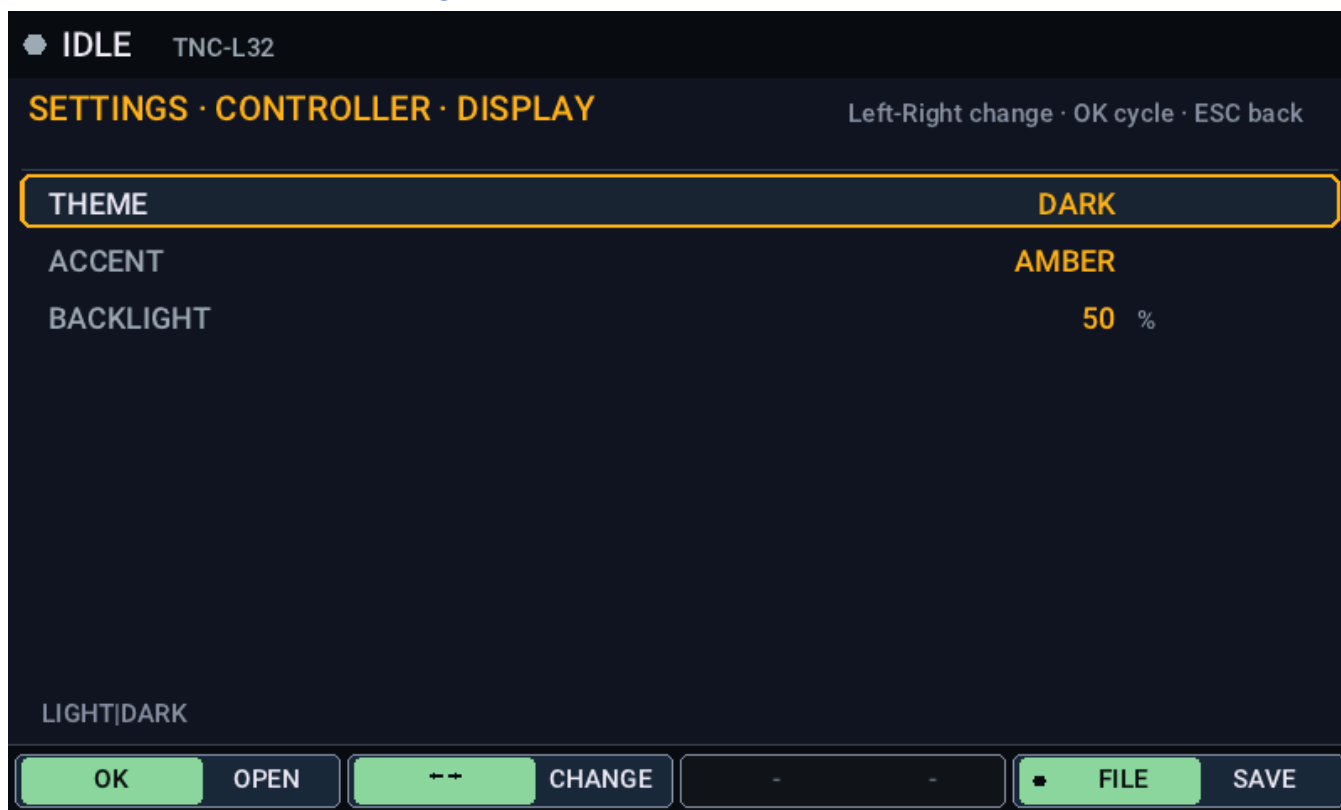


Figure 21.5 — CONTROLLER, PENDANT

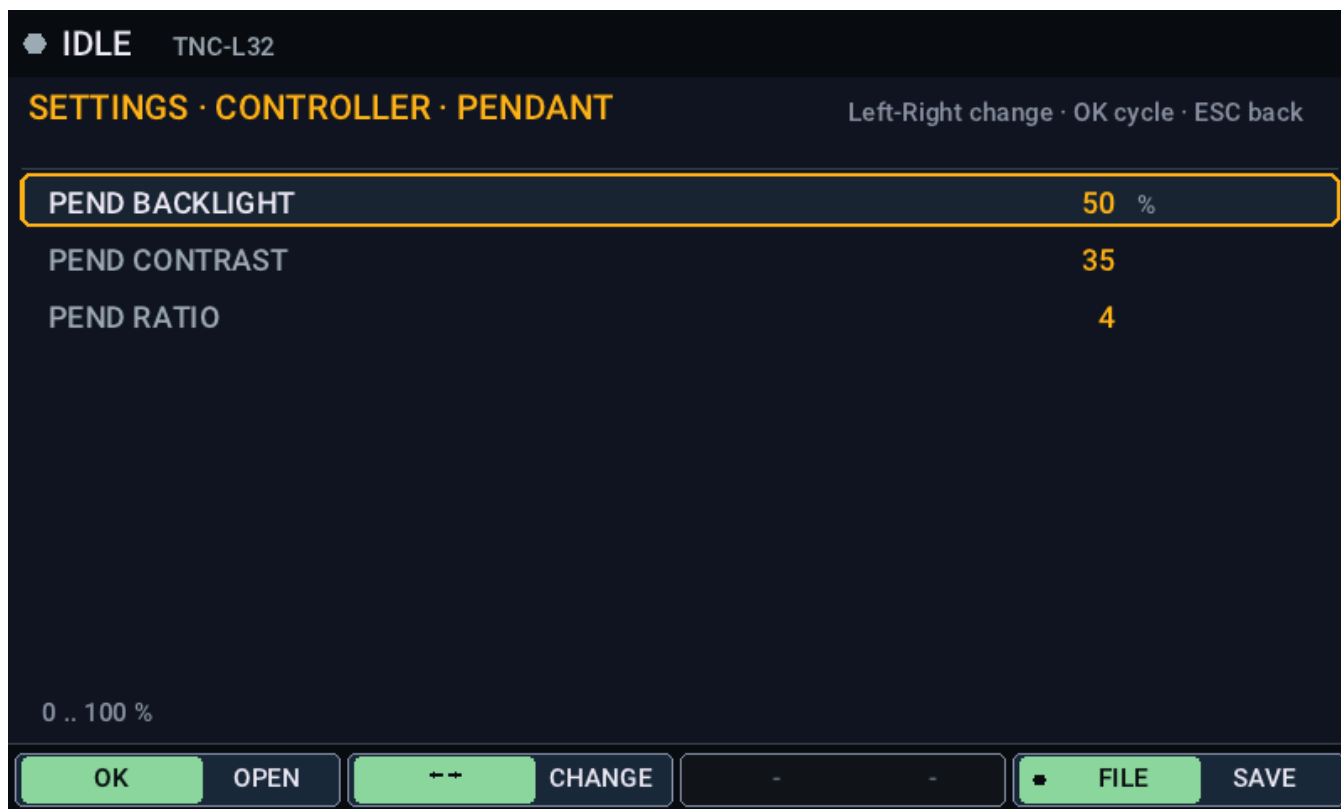


Figure 21.6 — CONTROLLER, UNITS & FORMAT

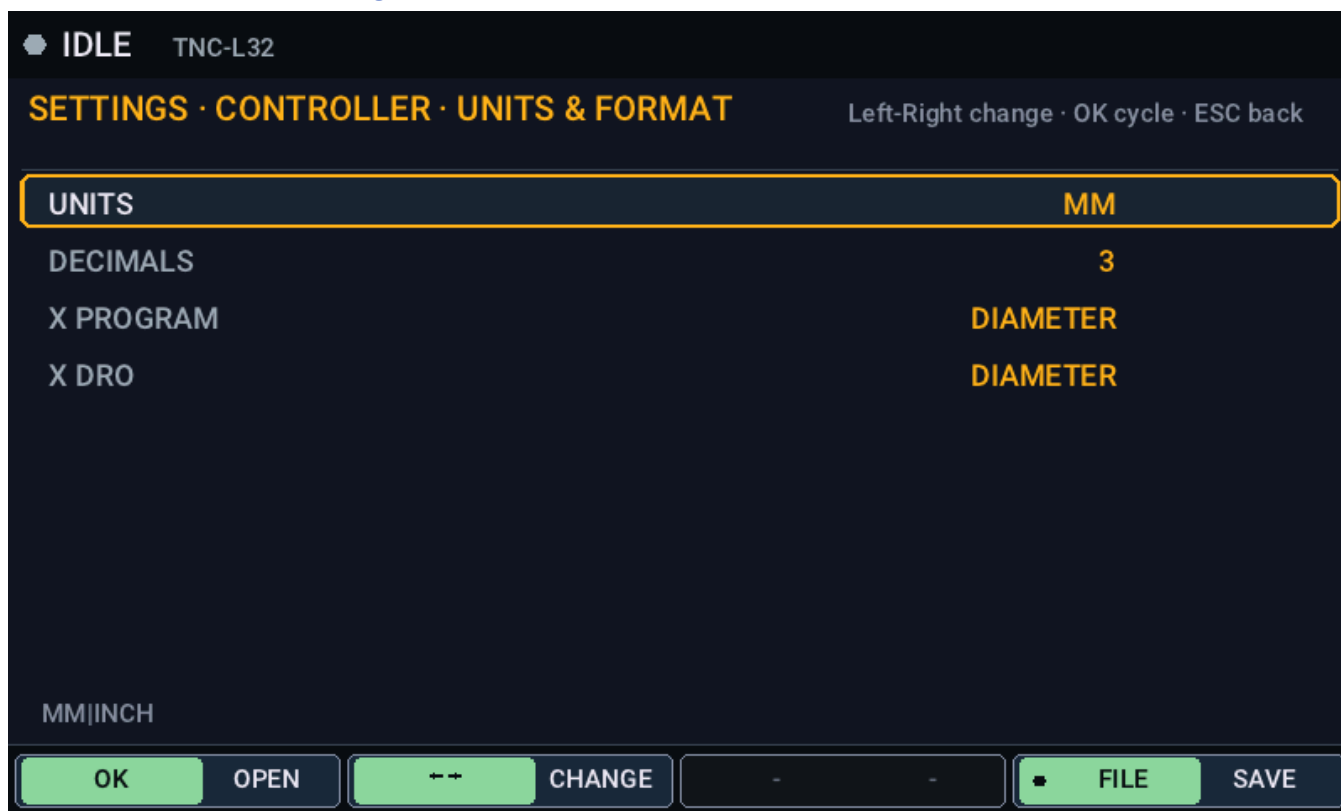


Figure 21.7 — CONTROLLER, KEYPAD

The screenshot shows the 'SETTINGS · CONTROLLER · KEYPAD' menu on a TNC-L32 device. The status bar at the top indicates 'IDLE' and 'TNC-L32'. The menu title is 'SETTINGS · CONTROLLER · KEYPAD' in yellow. The right side of the menu shows navigation options: 'Left-Right change · OK cycle · ESC back'. The 'LONG PRESS' setting is highlighted with a yellow border and shows a value of '800 ms'. Below it, the 'KEY REPEAT' setting shows a value of '2 /s'. At the bottom left, a range '250 .. 800 ms' is displayed. At the bottom right, the status 'UNSAVED · HOLD FILE saves' is shown. The bottom navigation bar contains buttons for 'OK', 'OPEN', 'CHANGE', a minus sign, a plus sign, 'FILE', and 'SAVE'.

● IDLE TNC-L32

SETTINGS · CONTROLLER · KEYPAD

Left-Right change · OK cycle · ESC back

LONG PRESS 800 ms

KEY REPEAT 2 /s

250 .. 800 ms

UNSAVED · HOLD FILE saves

OK OPEN CHANGE - + FILE SAVE

Figure 21.8 — CONTROLLER, JOG

● IDLE

TNC-L32

SETTINGS · CONTROLLER · JOG

Left-Right change · OK cycle · ESC back

JOG VEL	400 mm/min
CONT JOG	800 mm/min

1 .. 10000 mm/min

UNSAVED · HOLD FILE saves

OK

OPEN

↔

CHANGE

-

-

FILE

SAVE

Field	Notes
THEME	Light is the shop default, because a bright panel reads under work lighting.
ACCENT	Selection and highlight colour only. It never carries a warning.
BACKLIGHT	The floor is 10%, not 0 — a panel you cannot see is not a setting.
PEND BACKLIGHT	0 is allowed here, because the pendant may be stowed.
PEND CONTRAST / RATIO	Ratio picks the coarse band and contrast trims inside it, so a pendant that will not come good at any contrast usually wants the neighbouring ratio.
DECIMALS	Places on the position display. Three is one micron on a metric machine.
X PROGRAM	What an X word in a program means — the G7 and G8 default. See section 21.14, Two Settings That Are Easily Confused.
X DRO	What the display shows. Separate from the above on purpose.
LONG PRESS	How long a hold gesture takes.
KEY REPEAT	Auto-repeat rate when a key is held.
JOG VEL / CONT JOG	Step-jog and continuous-jog rates. Both are clamped to the axis maximum rate when used, and the defaults sit inside the shipped axis rates deliberately, because a configuration asking for 2000 on a 1000 mm/min axis lies to whoever reads it.

21.4 Machine — General and Axes

GENERAL holds the drive and mechanics settings that are not per axis: PULSE WIDTH, DIR SETUP, JUNCTION DEV and SMOOTH TIME. Section 7.3 gives which symptom points at which. The first two add, and their sum sets the step-rate ceiling (Appendix B, Integrator Technical Specifications).

Figure 21.9 — MACHINE, GENERAL

The screenshot shows the 'SETTINGS · MACHINE · GENERAL' menu on a TNC-L32 device. The status bar at the top indicates 'IDLE' and 'TNC-L32'. The menu title is 'SETTINGS · MACHINE · GENERAL'. The right side of the screen shows navigation instructions: 'Left-Right change · OK cycle · ESC back'. The settings list includes:

- PULSE WIDTH**: 5 us (highlighted with a yellow border)
- DIR SETUP**: 8 us
- JUNCTION DEV**: 0.0100 mm
- SMOOTH TIME**: 0 ms

At the bottom left, a range '1 .. 20 us' is displayed. At the bottom right, a status indicator shows 'UNSAVED · HOLD FILE saves'. The bottom navigation bar contains buttons: 'OK', 'OPEN', a left-right arrow, 'CHANGE', a minus sign, a plus sign, a file icon, 'FILE', and 'SAVE'.

AXES carries the same seven fields for X and then Z — fourteen rows in all, each shown on screen with its axis prefix: X STEPS/MM, X MAX RATE, X ACCEL, X BACKLASH, X SOFT MIN, X SOFT MAX, X DIR INVERT, then the same seven for Z. They are listed once below because the meaning does not change between them; only the defaults do.

Figure 21.10 — MACHINE, AXES

● IDLE

TNC-L32

SETTINGS · MACHINE · AXES

Left-Right change · OK cycle · ESC back

X STEPS/MM

400.000

X MAX RATE

1000 mm/min

X ACCEL

150 mm/s2

X BACKLASH

0.000 mm

X SOFT MIN

-1.000 mm

X SOFT MAX

100.000 mm

X DIR INVERT

NORMAL

Z STEPS/MM

400.000

Z MAX RATE

1500 mm/min

1.000 .. 100000.000

UNSAVED · HOLD FILE saves

OK

OPEN

← →

CHANGE

-

-

⚙

FILE

SAVE

Field	Notes
STEPS/MM	Set by calibration (Chapter 17, Commissioning the Threading Sync), not by calculation.
MAX RATE	X is the cross slide: short travel, finer resolution, slower by default. Z is the carriage: long travel, faster rapids by default.
ACCEL	Higher on Z than X by default, matching the longer travel.
BACKLASH	Lost motion taken up on reversal. Zero by default on both axes; measure it before entering a non-zero value.
SOFT MIN / SOFT MAX	Machine coordinates, and mean nothing until the axis is referenced. Appendix C, Complete Settings Menu Tree, has the full range and the shipped defaults.
DIR INVERT	Flips the direction output. Set this rather than swapping motor leads.

X is a radius here, as it is everywhere below the display. Appendix C, Complete Settings Menu Tree, carries the range and default for every row above.

WARNING

Soft limits are machine coordinates and mean nothing until the machine is referenced. They also do nothing until soft limits are turned on, which ships off.

21.5 Machine — Spindle

If you are using a closed-loop analog drive, the PID loop parameters ``RPM HOLD P``, ``RPM HOLD I``, and ``RPM HOLD D`` govern how aggressively the controller corrects the spindle speed against the encoder feedback.

Figure 21.11 — MACHINE, SPINDLE

● IDLE

TNC-L32

SETTINGS · MACHINE · SPINDLE

Left-Right change · OK cycle · ESC back

DRIVE	STEP
MAX RPM	20000 rpm
MIN RPM	100 rpm
ANALOG SPAN	3000 rpm
ENC CNT/IDX	2000
ENC IDX REVS	1
ENC INVERT	NORMAL
DRY-RUN RPM	300 rpm
MPG DET/REV	100
ANALOG STEP	UNSAVED · HOLD FILE saves

OK

OPEN

↔

CHANGE

-

-

⚙

FILE

SAVE

Field	Notes
DRIVE	Analog 0–10 V to a variable-frequency drive, or the step channel.
MAX RPM	Ceiling, on both drives, and the reference for the G50 clamp.
MIN RPM	A floor, on both drives — an S word below it turns at the floor instead. The default suits a variable-frequency drive, which will not pull below about 50 rpm. A servo will, so on step drive set this to your lowest genuinely useful speed.
ANALOG SPAN	Analog drive only — the speed that full scale means. Where commanded and measured disagree by a constant ratio, this is the field. The step path drives that output to zero and never reads it.
ENC CNT/IDX	Counts between index pulses, after decoding.
ENC IDX REVS	Chuck revolutions per index. This is what makes a geared encoder exact.
ENC INVERT	Encoder count direction.
DRY-RUN RPM	Speed of the synthetic spindle used by the dry-run source.
MPG DET/REV	Handwheel detents per revolution, for the hand-generate check.

Field	Notes
EN RELAY	Enable-relay contact sense.
STEPS/REV	Step drive only. What one pulse means. The default of 2000 suits a servo, whose own encoder is usually 2000 or 4000 counts.
SPIN ACCEL	Step drive only. How fast the commanded speed may change, in rpm per second.
RPM HOLD P / I / D	On the screen, but read by nothing on either drive. All three at zero and all three at fifty behave identically.

On step drive an S word is a target rather than a setting. A chuck with a bar in it has far more inertia than the cross slide, so the commanded speed ramps at SPIN ACCEL rather than stepping — the shipped 500 rpm per second is two seconds to 1000 rpm. It is conservative on purpose: asking for more than the drive can deliver produces a stall, and a stalled spindle with Z advancing is a broken tool.

CAUTION

A direction change goes through zero. An M4 while running M3 ramps to a stop, flips the direction, and ramps up the other way. Flipping direction at speed asks a servo to reverse a loaded chuck instantly, and it faults, or the work leaves the jaws.

21.6 Machine — Homing

Hardware switch states and limits must be configured to match your machine's wiring (NO / NC). This includes `HOME SWITCH`, `PROBE SWITCH`, `LIMIT SWITCH`, and the `PANEL ESTOP` (ABSENT / PRESENT).

21.7 Machine — Limits, E-Stop and Toolpost

Figure 21.13 — MACHINE, LIMITS & E-STOP

● IDLE

TNC-L32

SETTINGS · MACHINE · LIMITS & E-STOP

Left-Right change · OK cycle · ESC back

LIMIT INVERT	NORMAL
LIMIT ACTION	E-STOP
SOFT LIMITS	OFF
LIMIT OVR	ALLOW
PENDANT ESTOP	ABSENT

NORMAL|INVERT

UNSAVED · HOLD FILE saves

OK

OPEN

← →

CHANGE

-

-

● FILE

SAVE

Figure 21.14 — MACHINE, TOOLPOST

The screenshot shows the 'SETTINGS · MACHINE · TOOLPOST' screen on a TNC-L32. The top status bar indicates 'IDLE' and 'TNC-L32'. The main title is 'SETTINGS · MACHINE · TOOLPOST'. Below the title, there are instructions: 'Left-Right change · OK cycle · ESC back'. The 'TOOLPOST' menu is highlighted with a yellow border. The current selection is 'FRONT'. The settings listed are: 'TOOL COUNT' (12), 'TOOL NUDGE' (10 um), 'CHANGE MODE' (MANUAL), 'TC POS X' (0.000 mm), and 'TC POS Z' (0.000 mm). At the bottom, there are buttons for 'FRONT|REAR', 'UNSAVED · HOLD FILE saves', and a row of control buttons: 'OK', 'OPEN', 'CHANGE', a button with two arrows, a button with a minus sign, a button with a plus sign, a button with a file icon, and 'SAVE'.

Field	Notes
LIMIT INVERT	On the screen, but read by nothing. Continuous limit monitoring is not fitted, so there is no contact sense to describe.
LIMIT ACTION	An emergency stop drops the drive enable; a controlled stop decelerates.
SOFT LIMITS	Turn this on. Off, nothing stops a program running the slide into the chuck.
LIMIT OVR	Whether jogging off a tripped limit is permitted.
PENDANT ESTOP	Set PRESENT only where the pendant emergency stop is wired.
TOOLPOST	Mirrors every X direction on screen — tool marker, tip glyphs, toolpath. Get it right first.
TOOL COUNT	How many tools the machine has. Only this many rows are drawn, and a T word past it blocks the run.
TOOL NUDGE	Step size for Step- and Step+ on a tool-table field.
CHANGE MODE	What a tool change is (section 9.4, Tool Changes and the Z Retract).
TC POS X	Tool-change position in X. Not used by the gang retract.
TC POS Z	The gang retract target. Zero switches the feature off.

21.8 Machine — PLC

Off by default, and costs nothing on a machine that has none. PROTOCOL, BAUD, PARITY, ADDRESS, I/O COUNT and TIMEOUT describe the link; INDEX TMO is how long to wait for a turret to report that it has indexed.

Figure 21.15 — MACHINE, PLC

● IDLE

TNC-L32

SETTINGS · MACHINE · PLC

Left-Right change · OK cycle · ESC back

PROTOCOL	OFF
BAUD	19200
PARITY	NONE
ADDRESS	1
I/O COUNT	0
TIMEOUT	200 ms
INDEX TMO	8000 ms

OFF|MODBUS RTU|ASCII

UNSAVED · HOLD FILE saves

OK

OPEN

← →

CHANGE

-

-

FILE

SAVE

21.10 Machine — Probe

Figure 21.17 — MACHINE, PROBE

● IDLE

TNC-L32

SETTINGS · MACHINE · PROBE

Left-Right change · OK cycle · ESC back

PROBE INVERT

NORMAL

PROBE FEED

100 mm/min

PROBE RETRACT

2.000 mm

PROTECTION

ON

NORMAL|INVERT

UNSAVED · HOLD FILE saves

OK

OPEN

← →

CHANGE

-

-

■

FILE

SAVE

Field	Notes
PROBE INVERT	Probe contact sense.
PROBE FEED	Approach rate.
PROBE RETRACT	How far to back off after contact.
PROTECTION	Refuse motion that would overtravel the probe.

21.11 System

Figure 21.18 — SETTINGS, SYSTEM



Figure 21.19 — SYSTEM, About

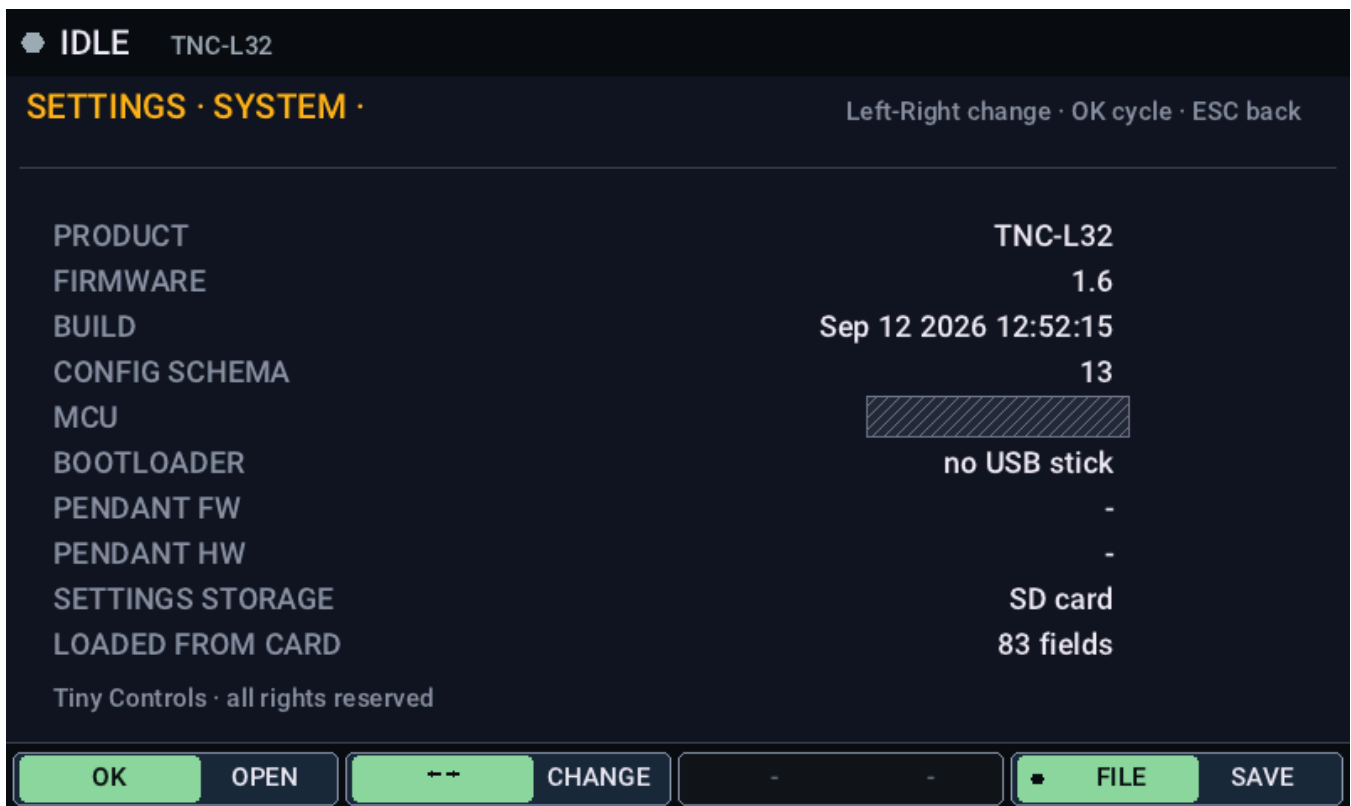


Figure 21.20 — SYSTEM, Diagnostics

www.tinycontrols.com Page 74 of 87

21.12 Where Settings Live

On the SD card, as text you can read, and nowhere else.

WARNING

A machine with no card in the slot comes up on factory defaults. There is no second copy inside the unit. The card is not optional equipment on this control, because programs live on it too.

What the trade buys is that you can see your settings, edit them, compare them and carry them to another machine. The card carries three folders, created automatically the first time it is used: one for part programs, one for macros, and one for the two configuration files. Nothing already on the card is moved into them, the browser still opens at the root, and programs kept there keep working. The settings are split the way the menu is. One file is this unit — theme, units, key behaviour — and the other is the lathe it drives.

- Saving is a separate, explicit act: hold File. Until you do, changes are live but not stored, and the screen shows UNSAVED.
- The menu says what the save did, in the bottom right, for a few seconds — how many fields went to the card, or that there was no card to write to.
- SYSTEM → About reports where the settings are stored and how many fields this start-up took from the card. With no card, that row says so and the footer turns amber.
- Each file is one setting per line, using the labels the menus show, with a version line at the top, and is editable on a PC. A file from older firmware loads and anything it leaves out keeps its current value; a file from newer firmware is refused whole rather than half applied.
- Saves are written alongside the file and renamed over it, so a power cut mid-save leaves the previous file intact.
- To edit settings on a PC, pull the card, edit the file, put it back and restart. There is no import step, because start-up reads the card.

21.13 What Survives a Power Cycle

Machine configuration is held in the controller once saved. Position, work offsets, tool wear, the loaded program and its line, the current tool, the jog axis and step, the diameter or radius choice and the coordinate frame are all kept in battery-backed memory and restored at start-up. Deliberately not restored: feed, rapid and spindle overrides, spindle and coolant state, alarms, and the spindle source.

WARNING

Position comes back so the display reads sensibly, but the axes are not referenced and the cockpit says so. Home before running.

21.14 Two Settings That Are Easily Confused

- X PROGRAM is the G7 and G8 default — how an X word in a file is read. It changes where the tool goes.
- X DRO is only how the number is displayed.

CAUTION

They are separate fields on purpose. One switch driving both would let a display preference silently change how G-code is interpreted, and the result is a part cut at twice or half the intended diameter.

Chapter 22 — Messages and Alarms

The controller refuses an action rather than performing a partial or unsafe version of it, and names the reason on screen when it does. Each entry below is phrased as the situation you are in.

22.1 Alarms

Message	Meaning and what to do
E-STOP	The emergency-stop chain has opened. Clear the cause, release the switch, and press Stop to clear the alarm.
HARD LIMIT	A limit input activated. Clear the alarm with Stop, then jog the axis clear. LIMIT OVR governs whether that is permitted.
DRIVE FAULT	A drive has reported a fault of its own. Read the drive, not the controller.
STEP QUEUE UNDERRUN	A controller fault rather than anything you did. Note what was running and report it.
STALL 500 CMD 12 ACT	On step drive, the spindle is not following the commanded train, and the job is aborted. Both numbers are named so you can see how far off it is (section 21.5, Machine — Spindle).
NO SPINDLE FEEDBACK	The encoder is reading nothing at all. A different fault from a stall, with a different fix — check the encoder before the drive.
SPINDLE 280 NOT 400 - CHECK	The pass started but after ten seconds the spindle never reached the target RPM; the machine holds. Check the drive and the encoder.
BUSY - MACHINE NOT IDLE	The controller refuses to start an action because it is already running one.
STOPPING - TRY AGAIN	The axes are decelerating from an abort. Wait a fraction of a second for them to come to rest before trying a new action.
T7 ABOVE TOOL COUNT 6	The requested tool exceeds the configured tool count limit.
SETTINGS FILE REFUSED	A settings file written by newer firmware is refused. Every setting remains at its default.
SAVE REFUSED - NEWER FILE, HOLD AGAIN	An attempt to save settings when a newer settings file was already refused. Holding again forces the save and keeps the refused file as a backup.

22.2 Homing and Reference

Message	Meaning and what to do
NOT REFERENCED	The axes have not been homed since power-on. Normal at start-up. Tap Homing, or hold it to reference in place.
NOT REFERENCED - HOME	Position was restored from the last power cycle but not measured since. A program will refuse to start until you home.

Message	Meaning and what to do
NO SWITCH IN nnn mm	The seek ran its whole allowance without seeing anything; the distance it covered is named. Check the switch on SYSTEM → Diagnostics → INPUTS, then the home direction, then SEEK MAX.

22.3 Loading and Running

Message	Meaning and what to do
COPY TO SD FIRST	OK was pressed on a file in the USB pane. Programs run from the card only. Use Shift + OK to copy it across, then load it from the SD pane.
WILL NOT RUN: ...	The program loaded and plotted, but Start is refused. The message names the reason and the line — usually a tool the machine does not have, or a tool-change retract that does not clear the longest tool (section 9.5, Loads But Will Not Run).
NOTE: ... (+n more)	The program loaded and will run, but not exactly as written. One is a note; several usually mean the program was written for another machine (Chapter 12, When the Control Disagrees With Your Program).
HOLD AFTER THIS PASS	Pause was pressed inside a threading or tapping cycle. The hold is deferred until the tool is clear, and the machine keeps cutting for up to one more pass (section 11.5, Rigid Tapping — G84).
running - no edits	Editing was attempted while the program was running or held. Stop the program first (section 13.1, Viewing and Editing a Program).
PLOTTED IN ... - RELOAD THE FILE	The work coordinate system moved and the program could not be re-parsed, so the plot shows the old geometry. Load the file again (section 14.3, The Work Coordinate System and the Plot).

22.4 Settings and Storage

Message	Meaning and what to do
UNSAVED	Settings or tool-table changes are live but not stored. Hold File to save them.
SAVE FAILED - NO CARD	There is no card in the slot to write to. Fit one and save again.
NO CARD - defaults	The machine started with no card and is running on factory defaults. Fit the card and restart.
out of range - OK will refuse	The number typed is outside the field's limits. Correct it; OK does nothing until you do.
whole numbers only - OK will refuse	A decimal was typed into a field that counts rather than measures.
TOO LOW	On the tools page, the tool-change Z does not clear the longest tool in the table (section 9.4, Tool Changes and the Z Retract).

Chapter 23 — Warranty and Support

23.1 Warranty

Tiny Controls warrants the TNC-L32 against defects in materials and workmanship for 12 months from the date of invoice. Cover is repair or replacement at Tiny Controls' option. Products resold rather than manufactured by Tiny Controls carry their original manufacturer's warranty, passed through.

23.2 Exclusions

The warranty does not cover:

- incorrect supply voltage or reverse polarity;
- wiring errors;
- water ingress, physical impact, lightning or surge damage;
- unauthorised modification;
- use outside the published ratings.

23.3 Returns

Return the unit carriage paid, with the invoice number and a description of the fault. Where the fault is covered, Tiny Controls pays the return carriage.

NOTE

Out-of-warranty repair remains available at cost for as long as parts are obtainable.

23.4 Support

Office	B-17/A, Nishat Park, Kakrola, New Delhi – 110078
Works	Khewat 863-Killa 103/8/2(1-18), Village Barhana, Tehsil Beri, District Jhajjar, Haryana – 124107
Telephone	Sales and support +91 99111 93212 · Office +91 99111 93210
Email	anil@tinycontrols.com
Web	www.tinycontrols.com

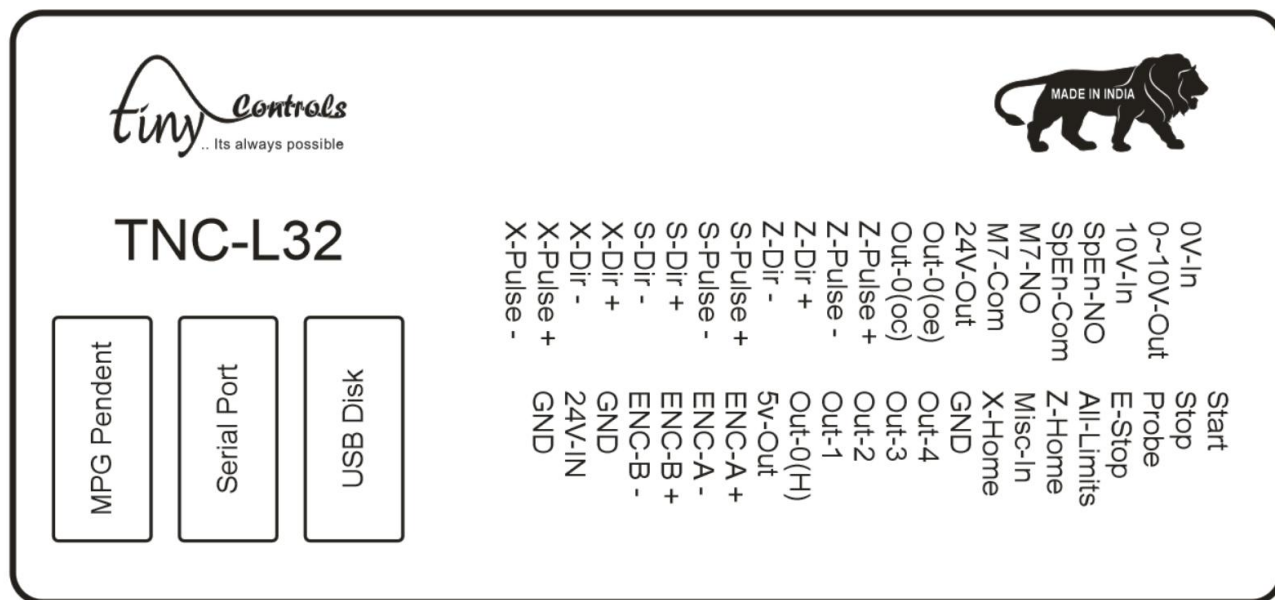
NOTE

This chapter restates the published Terms of Sale, which are the contract. Where the two differ, the Terms of Sale govern.

Appendix A — Connection Diagram

Every terminal as it is labelled on the unit. Section 3.1 names the same terminals in text and gives the function of each.

Figure A.1 — TNC-L32 Terminal Layout

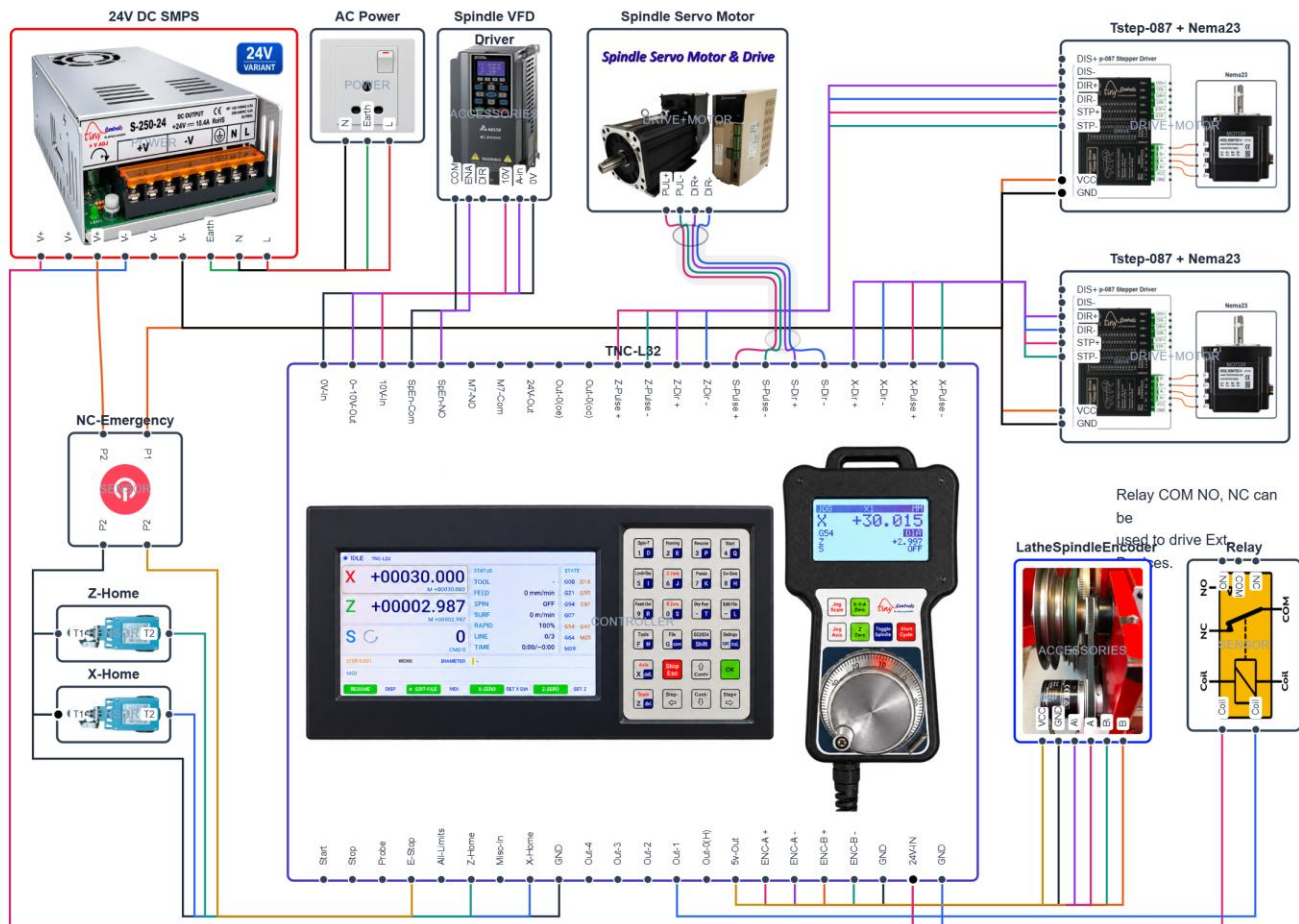


CAUTION

Disconnect the supply before making or changing any wiring (Chapter 1, Safety Information).
Confirm supply polarity before applying power.

Figure A.2 shows the controller wired into a complete machine as one worked example: a 24 V supply, an emergency-stop switch in a normally-closed loop, home switches on both axes, a variable-frequency drive taking the analog speed reference, a stepper drive and motor on each axis, the chuck encoder, and a relay driving an external device. It is an example rather than the only valid arrangement — connect whatever drives, switches and loads the installation needs to the terminals named in section 3.1, Terminals.

Figure A.2 — Worked System Wiring



NOTE

The relay contact is shown with both its normally-open and normally-closed sides. Either can be used to drive an external device.

Appendix B — Integrator Technical Specifications

Derived limits: figures that follow from a combination of settings rather than being set directly. Each is stated with the setting that governs it, because no other setting can raise it.

B.1 Step Rate Ceiling

The controller's own ceiling is 500 kHz per axis, and it is not the number that will bound your machine. The drive is. Every pulse has to be wide enough for the drive to latch, and the direction line has to be stable before the edge the drive latches on. Those two times add, and their sum has to fit inside half a step interval:

$$\text{ceiling} = 1 / (2 \times (\text{DIR SETUP} + \text{PULSE WIDTH}))$$

The right-hand column converts each ceiling to a feed rate at 400 steps per millimetre, a common figure for a direct-driven ballscrew. Scale it for your own STEPS/MM: half the steps per millimetre, twice the feed.

DIR SETUP + PULSE WIDTH	Ceiling	At 400 steps/mm
1 + 1 = 2 μ s	250 kHz	37,500 mm/min
1 + 5 = 6 μ s	83.3 kHz	12,500 mm/min
8 + 5 = 13 μ s (shipped default)	38.5 kHz	5,769 mm/min
2 + 10 = 12 μ s	41.7 kHz	6,250 mm/min
8 + 10 = 18 μ s	27.8 kHz	4,167 mm/min
20 + 20 = 40 μ s (slowest)	12.5 kHz	1,875 mm/min

Why half an interval and not the whole one: if a pulse has not finished before the next one is due, the drive sees one long pulse where two were sent, and the axis quietly loses half its steps at speed. Slowing a step is a velocity error and the machine catches up. Merging two is a position error, and nothing downstream can detect it or recover from it — the part is simply the wrong size. This is also why the defaults are what they are. 5 μ s suits most opto-isolated drives, and several ask for 7.5 or 10. 8 μ s of direction settle is what an optocoupler needs, and it is the field that fixes position lost specifically on reversals — a facing pass, a threading retract — rather than at speed. Both fields sit on the same screen, so the arithmetic is in front of you when you change either. A differential drive changes the picture completely. The microsecond figures above are optocoupler times. A drive with a line-receiver input latches in nanoseconds, so 1 + 1 μ s is realistic and the ceiling moves to 250 kHz — at which point the controller's own 500 kHz cap and the drive's maximum input frequency become the real limits. Read the drive's datasheet rather than raising these fields hopefully.

CAUTION

No axis rate or acceleration setting can raise this ceiling. An axis configured for a maximum rate that would need more steps per second than the ceiling allows will not reach it, and the shortfall appears as lost position rather than as an error. SYSTEM → Diagnostics shows the pair and the resulting figure.

NOTE

None of this applies to the spindle under DRIVE = STEP. That train is a free-running 50 % duty square wave, and it carries no direction-setup penalty because a spindle only reverses at rest. The controller imposes no ceiling on it at all; what bounds the top end is the drive's maximum input frequency. SYSTEM → Diagnostics → S TRAIN reports the train in kilohertz (Chapter 21, Settings Reference).

B.2 Threading Follower Limits

Limit	Governed by
The follower never exceeds 1.5 times synchronous speed while closing its starting deficit, whatever the axis allows.	Fixed. Asking a stepper for its unloaded maximum while it is in the cut is how steps are lost.
The axis maximum rate must clear 1.25 times synchronous speed, or the pass holds before it starts and says so.	Z MAX RATE against pitch × spindle speed.
Lead-in distance, approximately $2 v^2 / a$ where v is the synchronous feed and a the axis acceleration.	Z ACCEL. The controller adds the configured LEAD-IN, rounded up to whole pitches.
Run-out distance, approximately $v^2 / 2a$.	Z ACCEL. Program the endpoint past the thread by at least this much.

B.3 Tapping Depth Overshoot

A tapping cycle cuts thread while the spindle decelerates at the bottom, so the hole finishes deeper than the commanded Z by:

$$\text{overshoot} = \text{pitch} \times \text{rpm}^2 / (120 \times \text{SPIN ACCEL})$$

The relationship is quadratic in spindle speed and linear in spindle acceleration. It is reported rather than compensated (section 11.5, Rigid Tapping — G84). Where the tap itself ends up is exact.

B.4 Other Enforced Limits

Limit	Value and governing setting
Jog rates	Clamped to the axis maximum rate at the moment of use, whatever JOG VEL or CONT JOG is set to.
Tool wear	±2 mm per axis. A correction larger than this is a measurement error, not a worn tip.
Minimum safe tool-change Z	Derived from the table: the largest Z geometry plus wear of any tool within TOOL COUNT, plus 3 mm standoff.
Peck depth	Expressed in microns. A value small enough to imply a runaway pass count is refused.
Canned cycle pass count	Bounded. A depth of cut small enough to exceed the bound is

Limit	Value and governing setting
	refused and the bound is named.
Spindle stall detection	Divergence over 25%, or 60 rpm, held for half a second, on step drive, above 30 rpm.
Feed override range	10 to 200%.

B.5 On-Card Files

File	Holds
0:/config/controller.cfg	Display, units, keypad and jog settings. One setting per line, using the labels the menus show, with a version line at the top.
0:/config/machine.cfg	Axes, spindle, homing, limits, toolpost, threading and probe settings. Copy this one alone to set up a second identical lathe — it carries nothing about this box, so the first lathe's screen colours do not travel with it.
0:/config/tools.cfg	The tool table, saved from the Tools page by holding File rather than by the settings save (section 20.5, The Tool Table After an Update).
L32FWUPD.TNC	The firmware update container, in the root of a USB stick, upper case (Chapter 20, Updating the Firmware).

Appendix C — Complete Settings Menu Tree

Every settable value on the panel, in menu order, with the range the editor enforces and the value it ships with. These are the figures the controller holds a value inside, so nothing printed here can disagree with what the panel will accept. Chapter 21, Settings Reference, carries the reasoning.

```

SETTINGS
|
|-- CONTROLLER -- how the control behaves
|   |-- DISPLAY
|   |   |   THEME ..... LIGHT | DARK                LIGHT
|   |   |   ACCENT ..... TEAL|AMBER|VIOLET|GREEN      TEAL
|   |   |   BACKLIGHT ..... 10 ... 100 %              80
|   |-- PENDANT
|   |   |   PEND BACKLIGHT ... 0 ... 100 %            70
|   |   |   PEND CONTRAST .... 0 ... 63               32
|   |   |   PEND RATIO ..... 0 ... 7                  4
|   |-- UNITS & FORMAT
|   |   |   UNITS ..... MM | INCH                     MM
|   |   |   DECIMALS ..... 2 ... 4                    3
|   |   |   X PROGRAM ..... RADIUS | DIAMETER          DIAMETER
|   |   |   X DRO ..... RADIUS | DIAMETER              DIAMETER
|   |-- KEYPAD
|   |   |   LONG PRESS ..... 250 ... 800 ms           450
|   |   |   KEY REPEAT ..... 2 ... 20 /s              8
|   |-- JOG
|   |   |   JOG VEL ..... 1 ... 10000 mm/min           400
|   |   |   CONT JOG ..... 1 ... 20000 mm/min          800
|
|-- MACHINE -- what the machine is
|   |-- GENERAL
|   |   |   PULSE WIDTH ..... 1 ... 20 us              5
|   |   |   DIR SETUP ..... 1 ... 20 us                8
|   |   |   JUNCTION DEV ..... 0.0001 ... 1 mm         0.01
|   |   |   SMOOTH TIME ..... 0 ... 200 ms             0
|   |-- AXES (the same seven rows for X then Z)
|   |   |   STEPS/MM ..... 1 ... 100000                X 400    Z 400
|   |   |   MAX RATE ..... 1 ... 60000 mm/min           X 1000   Z 1500
|   |   |   ACCEL ..... 1 ... 10000 mm/s2              X 150    Z 200
|   |   |   BACKLASH ..... 0 ... 5 mm                  X 0      Z 0
|   |   |   SOFT MIN ..... -9999 ... 9999 mm            X -1     Z -300
|   |   |   SOFT MAX ..... -9999 ... 9999 mm            X 100    Z 10
|   |   |   DIR INVERT ..... NORMAL | INVERT            X NORMAL Z NORMAL
|   |-- SPINDLE
|   |   |   DRIVE ..... ANALOG | STEP                  ANALOG
|   |   |   MAX RPM ..... 1 ... 20000 rpm               3000
|   |   |   MIN RPM ..... 0 ... 5000 rpm                50
|   |   |   ANALOG SPAN ..... 1 ... 20000 rpm            3000
|   |   |   ENC CNT/IDX ..... 4 ... 65536               4096
|   |   |   ENC IDX REVS ..... 1 ... 16                 1
|   |   |   ENC INVERT ..... NORMAL | INVERT            NORMAL
|   |   |   DRY-RUN RPM ..... 0 ... 10000 rpm            300
|   |   |   MPG DET/REV ..... 1 ... 10000               100
|   |   |   EN RELAY ..... NO | NC                      NO
|   |   |   STEPS/REV ..... 1 ... 1000000               2000
|   |   |   SPIN ACCEL ..... 10 ... 20000 rpm/s          500
|   |   |   RPM HOLD P I D ... 0 ... 100 each           0 0 0
|   |-- HOMING
|   |   |   HOMING ..... OFF | ON                      OFF
|   |   |   FEED (FAST) ..... 1 ... 20000 mm/min        600
|   |   |   SEEK (SLOW) ..... 1 ... 5000 mm/min         100
|   |   |   PULL-OFF ..... 0 ... 50 mm                  2
|   |   |   ORDER ..... X FIRST | Z FIRST | BOTH        X FIRST
|   |   |   SWITCH INV ..... NORMAL | INVERT            NORMAL
|   |   |   X HOME DIR ..... POS | NEG                  POS
|   |   |   Z HOME DIR ..... POS | NEG                  POS
|   |   |   SEEK MAX ..... 0 ... 2000 mm                0

```

```
| +- LIMITS & E-STOP
| |   LIMIT INVERT ..... NORMAL | INVERT          NORMAL
| |   LIMIT ACTION ..... E-STOP | CTRL STOP       E-STOP
| |   SOFT LIMITS .....  OFF | ON                OFF
| |   LIMIT OVR .....    DENY | ALLOW             ALLOW
| |   PENDANT ESTOP .... ABSENT | PRESENT          ABSENT
| +- TOOLPOST
| |   TOOLPOST .....    FRONT | REAR              FRONT
| |   TOOL COUNT .....  1 ... 16                  12
| |   TOOL NUDGE .....  1 ... 1000 um             10
| |   CHANGE MODE ..... MANUAL | GANG | TURRET     MANUAL
| |   TC POS X .....    -9999 ... 9999 mm          0
| |   TC POS Z .....    -9999 ... 9999 mm          0
| +- PLC
| |   PROTOCOL .....    OFF | MODBUS RTU | ASCII    OFF
| |   BAUD .....        9600 ... 460800            19200
| |   PARITY .....      NONE | EVEN | ODD          NONE
| |   ADDRESS .....     1 ... 247                  1
| |   I/O COUNT .....   0 ... 64                   0
| |   TIMEOUT .....     10 ... 5000 ms             200
| |   INDEX TMO .....   100 ... 60000 ms           8000
| +- THREADING
| |   LEAD-IN .....     0.5 ... 50 mm              3
| |   PULL-OUT .....    0 ... 50 mm                1
| |   SPRING PASS .....  0 ... 5                  1
| |   MIN RPM .....     10 ... 5000 rpm            100
| |   SYNC SOURCE ..... ENCODER | S AXIS           ENCODER
| +- PROBE
| |   PROBE INVERT ..... NORMAL | INVERT          NORMAL
| |   PROBE FEED .....   1 ... 2000 mm/min         100
| |   PROBE RETRACT .... 0 ... 50 mm              2
| |   PROTECTION .....  OFF | ON                  ON
| +- SYSTEM -- four pages, not settings
|   About .....         firmware, storage, fields loaded
|   Diagnostics .....   LINK / INPUTS / OUTPUTS, S TRAIN, last key
|   Update pendant fw
|   Spindle source ..... ENCODER / MPG / DRY RUN
```

NOTE

Three defaults mean "off" rather than a value: SEEK MAX at 0 is no distance limit, TC POS Z at 0 disables the gang retract, and LEAD-IN at 0 leaves a threading program exactly as written.

Appendix D — Quick Reference Card

D.1 Cockpit Keys

Key	Tap	Hold
Axis	Select the jog axis	Switch diameter and radius display
Scale	Cycle the jog step	—
Step– / Step+	One jog step	Repeat
Cont+ / Cont–	Continuous jog while held	—
X-Zero	Type what X reads as	Make X zero
Z-Zero	Type what Z reads as	Make Z zero
G53/G54	Swap work and machine display	Pick G54 to G59
Homing	Run the homing cycle	Reference in place
Spin-T	Stop the spindle	Start the spindle
Feed-Ovr	Type an override, 10 to 200%	Return to 100%
Edit-File	Edit the loaded program	Open manual entry
Resume	Cycle cockpit, toolpath, G-code	—
Start	Start, or resume a hold	—
Pause	Feed hold, from any screen	—
Stop	Abort motion and stop the spindle, from any screen	—

D.2 Shifted Combinations

Combination	Does
Shift + Step– / Step+	Tool table: nudge the field. Settings: change by ten steps
Shift + OK	Files: copy to the other pane
Shift + Dry-Run	Files: delete the selected file
Shift + Scale	Any text entry: delete a character

D.3 Before the First Cut

1. Set steps per millimetre on both axes and confirm a measured move.
2. Confirm each axis moves the way the display says.
3. Set the soft limits and turn them on.
4. Enable homing and prove each switch on the diagnostics inputs page.
5. Set the toolpost and the tool count.

6. Qualify the tools, master first at zero.
7. Choose the work offset, then touch off X and Z.
8. Load the program, read the plot, dry-run, then cut.

D.4 Three Things That Bite

- The large position number is work, the small one is machine.
- The first block of a program runs from wherever the slides are standing.
- Pause inside a threading or tapping cycle is deferred until the tool is clear; Stop is not.

TC-TNC-L32-OM-001 · R1.0 · September 2026