

PCB CAM SOFTWARE

TraceFab User Manual

Structure Preview

Version 1.0

Contents

1 Table of Contents

2 Introduction	1
2.1 About TraceFab	1
2.2 How to Use This Manual	2
2.3 The TraceFab Workflow	2
3 Safety and Responsible Use	3
3.1 General Safety Principles	3
3.2 CNC Machine Safety	4
3.3 Personal and Environmental Protection	4
3.4 G-code and Data Safety	5
4 System Requirements and Installation	6
4.1 System Requirements	6
4.2 Installing TraceFab	7
4.3 First Start	11
4.4 Updating or Removing TraceFab	12
5 Licensing and Activation	13
5.1 License Types	13
5.2 Machine Identification	15
5.3 Activating TraceFab	15
5.4 Activation Troubleshooting	16
6 User Interface Overview	17
6.1 Main Workspace	17
6.2 PCB Preview	20
6.3 Status and Messages	21
7 Importing PCB Data	21
7.1 Preparing an Import	21
7.2 KiCad PCB Import	21
7.3 Gerber and Excellon Import	22
7.4 Validating Imported Geometry	24
8 Contouring and Isolation Toolpaths	24
8.1 Contouring Workspace	24
8.2 Contour Pass Strategy	25
8.3 Layer and Toolpath Position	26

8.4 Calculating and Reviewing Contours	27
9 Excellon Drilling	28
9.1 Drill Setup Workspace	28
9.2 Drill Plan Modes	29
9.3 Previewing Hole Groups	29
9.4 Calculating the Drill Operations.....	30
9.5 Reviewing the Drill Plan.....	30
10 PCB Edge Cutting	31
10.1 Edge Cutting Workspace	31
10.2 Cutting Parameters.....	32
10.3 Bridge Gaps.....	33
10.4 PCB Inner Cuts.....	35
10.5 Calculating and Reviewing the Edge Cut.....	36
11 Double-sided Boards and Alignment	36
11.1 Purpose of Alignment Pins.....	36
11.2 Opening the Alignment Pins Window.....	37
11.3 Placing Alignment Pins	37
11.4 Calculating the Alignment Operation.....	38
11.5 Using an Alignment Pin as the World Origin	39
11.6 Recommended Board-Flipping Workflow.....	40
12 CNC Origin and Coordinate Setup	41
12.1 Coordinate Systems	41
12.2 G-code Output Units	41
12.3 Setting the World Origin.....	42
12.4 Mirroring and Board Orientation.....	43
12.5 Verifying the Coordinate Setup	43
13 Creating the CNC Program	43
13.1 Manufacturing Window Overview.....	44
13.2 Building an NC Sequence.....	45
13.3 Creating Multiple NC Programs	45
13.4 Job and Output Settings	46
13.5 Validating the Sequence	46
13.6 Generating the G-code File.....	47
13.7 Controls Reserved for Future Development.....	47
14 G-code Preview and Verification	48
14.1 Inspecting the Generated G-code	48
14.2 Tool Changes and Tool Length.....	49

14.3 Graphical G-code Preview	49
14.4 First-run Safety Tests	50
14.5 Final Pre-machining Checklist.....	50
14.6 Future Built-in Preview.....	51
15 Settings and File Management.....	51
15.1 Program Settings.....	51
15.2 CAM Parameters	52
16 Typical Workflows	53
16.1 Importing the PCB	53
16.2 Single-sided PCB Workflow	54
16.3 Double-sided PCB Workflow.....	54
17 Troubleshooting	55
18 Updates, Support, and Feedback.....	55
18.1 Software Updates	55
18.2 Getting Support	56
18.3 Reporting Problems	56
19 Disclaimer	57
19.1 Operator Responsibility.....	57
19.2 Mandatory G-code Verification.....	57
19.3 Responsibility and Liability	57
20 Revision History	57

2 Introduction

2.1 About TraceFab

2.1.1 Purpose of TraceFab

TraceFab is an independent desktop PCB CAM application for makers, electronics developers, prototype laboratories, and small workshops that manufacture printed circuit boards on CNC machines. It bridges the gap between PCB design data and the machining operations required to mill, drill, and cut a physical board.

TraceFab was created because the available PCB-to-CNC landscape is fragmented. Some established solutions remain tied to legacy interfaces or older platforms, while other capable tools are strongly associated with specialised or Linux-first workflows. Open-source projects can also become inactive, leaving users dependent on software that no longer evolves with current PCB design tools or modern Windows systems. Even commercial alternatives do not always provide the focused, predictable workflow that small workshops need.

The ambition behind TraceFab is to provide a fresh, modern, and stable Windows desktop environment for this specialised branch of PCB manufacturing. The application focuses on a clear visual workflow, direct inspection of imported PCB data, controlled creation of machining operations, and reliable generation of CNC-ready G-code. TraceFab is not a machine controller: its role ends with preparing and exporting the program that the operator reviews and runs on a suitable CNC system.

2.1.2 Intended Users

TraceFab is intended for makers, PCB designers, repair and prototyping teams, educational environments, and small production workshops that already have basic knowledge of CNC machining. It is particularly suited to users who want to manufacture prototypes in-house without building a CAM workflow from several unrelated utilities.

The software does not replace CNC training, knowledge of cutting tools, or familiarity with the machine and its controller. Users should understand coordinates, work offsets, spindle operation, feed rates, cutting depths, tool changes, and emergency procedures before running generated G-code on physical equipment.

2.1.3 Scope of This Manual

This manual explains how to install and activate TraceFab, import supported PCB design data, inspect the imported layers, define machining operations, organise the CNC process, generate G-code, and verify the result before machining. Later chapters also cover settings, file management, typical workflows, troubleshooting, updates, and support.

Machine-specific operation is outside the scope of this manual. Always follow the documentation supplied by the CNC manufacturer, controller vendor, cutting-tool supplier, and material supplier. Where this manual gives general machining or safety guidance, the instructions for the actual equipment and workplace take precedence.

2.2 How to Use This Manual

2.2.1 Document Conventions

The manual follows the normal TraceFab workflow from installation to final G-code verification. Names of buttons, menu items, fields, and tabs are shown as they appear in the application. File names and extensions, such as `TraceFab.exe` or `.kicad_pcb`, are presented in a distinct technical style where appropriate.

2.2.2 Interface References

Instructions normally identify the form or workspace first, followed by the control to use. Because TraceFab is actively developed, the exact position or wording of a control may change between releases. When the interface and this manual differ, consult the release information and use the labels shown by the installed version.

2.2.3 Screenshot and Figure Conventions

Screenshots illustrate the relevant form, controls, and expected visual result. Numbered callouts may be used when several controls require explanation. Screenshots are representative and may differ slightly from a newer build.

The manual is intended to be reviewed with every TraceFab release. Updated copies will be distributed with the software package and installer and made available through the official TraceFab website. The manual included with a particular release should therefore be treated as the primary reference for that release.

2.3 The TraceFab Workflow

2.3.1 From PCB Design to CNC G-code

A TraceFab job begins by importing PCB design and manufacturing data. Depending on the supported workflow and application version, this can include a KiCad PCB file or a combination of Gerber layer files and Excellon drill data. TraceFab converts the source data into a visual representation that allows the user to confirm the copper, drill, and board-outline geometry before creating toolpaths.

The user then configures the required CNC operations. Typical operations include isolation contours, drilling, board edge cutting, holding tabs, and alignment features for double-sided boards. The CNC origin and machining parameters are defined, and the operations are placed in the required production order. TraceFab finally generates G-code from that configured process.

2.3.2 Recommended Processing Order

A reliable job should be prepared in a deliberate order: import the source files, validate every required layer, create the individual machining operations, verify tool and depth parameters, define alignment and origin settings, organise the operation sequence, and only then generate the CNC program. Keeping these stages separate makes errors easier to detect before they reach the machine.

Future TraceFab development may add reusable job recipes or batch automation. The goal is to allow a frequently manufactured PCB to be associated with a proven CNC process so that the same import, operation sequence, and G-code generation can be repeated with minimal

interaction. Such automation is a planned direction and must not be assumed to be available in the current release.

2.3.3 Verification Before Machining

Generated G-code is an output to be reviewed, not an instruction to start the machine automatically. Before transfer to a CNC controller, inspect the program and preview, confirm the origin, dimensions, units, cutting depths, feed rates, safe heights, spindle commands, tool sequence, and expected machine travel. New or changed jobs should be tested with a dry run or air cut before the tool is allowed to enter the material.

3 Safety and Responsible Use

3.1 General Safety Principles

3.1.1 User Responsibility

CNC machining can cause serious personal injury, fire, equipment damage, and material loss when a machine, tool, workpiece, or program is used incorrectly. The operator is responsible for the complete machining setup and must be trained to use the CNC machine, controller, spindle, cutting tools, workholding system, extraction equipment, and emergency controls.

TraceFab generates G-code only. It does not control the machine, monitor its motion, verify the physical setup, or guarantee that a generated program is suitable for a specific controller or machine. The operator must review and approve every program before execution and remains responsible for deciding whether it is safe to run.

To the maximum extent permitted by applicable law, the authors and publisher of TraceFab accept no responsibility for injury, accidents, machine damage, broken tools, damaged workpieces, lost production, or other loss arising from the use of generated output. Nothing in this manual replaces the machine manufacturer's safety instructions, workplace rules, a suitable risk assessment, or legal obligations that cannot be excluded.

3.1.2 Supervision and Emergency Stop

Never leave a CNC machine unattended while testing a new or modified program. During the first runs, remain at the machine and be ready to use the controller's feed-hold function or emergency stop immediately. Know the location and behaviour of the emergency controls before starting the spindle.

For an unfamiliar job, begin conservatively. Use reduced feed where appropriate, perform a dry run or air cut above the workpiece, and observe the complete motion sequence. Stop the machine if movement is unexpected, if the selected coordinate system is incorrect, or if the tool approaches a clamp, fixture, machine limit, or unsafe depth.

3.1.3 Safe Working Environment

Keep the working area clean, dry, well lit, and free from loose objects. Restrict access while the machine is operating, particularly for children, visitors, or anyone unfamiliar with CNC equipment. Do not operate machinery when tired, distracted, or impaired.

Provide adequate ventilation and suitable extraction for the material being machined. Ensure that cables, extraction hoses, coolant lines, and other equipment cannot enter the machine's travel area. Keep the emergency stop accessible at all times.

3.2 CNC Machine Safety

3.2.1 Moving Parts and Cutting Tools

Keep hands, hair, clothing, jewellery, tools, and measuring equipment away from the spindle and all moving axes. A spindle can remain dangerous after power is removed because the tool may continue rotating. Wait for all movement to stop before opening an enclosure or reaching into the working area.

Use cutting tools that are undamaged, correctly sized, and suitable for PCB material and the selected spindle speed. Install each tool according to the spindle and collet manufacturer's instructions. Never attempt to adjust, clean, or measure a rotating tool.

3.2.2 Workpiece and Tool Securing

Secure the PCB material and any sacrificial backing plate so that they cannot move, lift, or vibrate during machining. Confirm that clamps, screws, adhesive fixtures, alignment pins, and vacuum fixtures do not intersect any programmed toolpath. Check that the cutting tool and collet are clean, properly seated, and tightened before starting the spindle.

A correct software preview cannot detect a loose workpiece, incorrectly installed tool, protruding clamp, or unsuitable fixture. These physical checks must form part of the operator's pre-start inspection.

3.2.3 Machine Limits and Travel

Confirm that the complete program fits within the usable travel of every machine axis. Check the selected work origin, board dimensions, tool length, safe Z height, clearance moves, and maximum cutting depth. Consider the space required for clamps, alignment pins, dust extraction, and tool changes.

Do not rely solely on software limits. Verify the physical position of the machine before starting, and ensure that homing, work offsets, and controller coordinate systems are configured correctly. Unexpected axis direction or an incorrect origin can cause an immediate collision.

3.3 Personal and Environmental Protection

3.3.1 Personal Protective Equipment

Use personal protective equipment appropriate to the machine, enclosure, process, and material. This normally includes suitable eye protection and may include hearing protection, protective footwear, and respiratory protection. Follow the machine manufacturer's instructions and the safety information supplied with the PCB substrate, copper laminate, cutting tools, and any chemicals used during production.

Gloves can be hazardous near rotating tools and moving machinery. Follow the equipment manufacturer's guidance and workplace risk assessment rather than assuming that gloves are appropriate during machine operation.

3.3.2 Dust and Fume Control

PCB milling and drilling can produce fine dust and particles from copper and substrate materials. Prevent this material from spreading through the workspace. Use an enclosed machine and

effective local extraction where possible, and select filtration that is suitable for the material and particle size.

Do not blow machining dust into the air with compressed air. Clean the machine and surrounding surfaces using a method approved for the material. Consult the material safety data and local occupational-health requirements when selecting ventilation, extraction, filtration, and respiratory protection.

3.3.3 Electrical and Fire Safety

Inspect power cables, spindle wiring, controller connections, earthing, and extraction equipment before operation. Do not use damaged equipment or make electrical modifications unless qualified to do so. Keep liquids away from electrical components unless the machine is specifically designed for their use.

Machining faults, stalled tools, damaged wiring, unsuitable extraction, and accumulated dust can create heat or fire hazards. Keep appropriate fire-safety equipment accessible and know how to isolate power safely. Never continue machining when smoke, unusual heat, abnormal noise, or an electrical smell is detected.

3.4 G-code and Data Safety

3.4.1 Previewing Generated Toolpaths

Review every generated program before it reaches the machine. Use the TraceFab preview to inspect toolpath location and sequence, then review the G-code manually or with a trusted controller-specific simulator. Pay particular attention to units, coordinate values, rapid movements, cutting depths, safe Z values, spindle commands, feed rates, tool changes, and the final return movement.

A graphical preview is an aid and cannot prove that a program is safe for every controller. Controller dialects, machine configuration, work offsets, macros, and post-processing behaviour may change how a command is executed.

3.4.2 Testing New Machine Profiles

Treat a new machine, controller, tool definition, origin strategy, or parameter set as unverified until it has completed a controlled test. Use a simple sample job, place the tool safely above the material, reduce operating speeds where appropriate, and run the program step by step or in single-block mode if the controller supports it.

After the dry run, verify the physical zero point and repeat the test with a non-critical workpiece before machining valuable material. Record settings only after the result has been checked and found repeatable.

3.4.3 Protecting Original Design Files

Keep the original PCB design and manufacturing files separate from generated CNC output. Save generated files to a dedicated job folder and avoid overwriting the source data. Before changing a proven job, create a versioned copy so that the last verified program remains available.

Back up important designs, tool definitions, machine settings, and verified G-code to storage that is independent of the working computer. TraceFab output should form part of the workshop's normal revision, backup, and production-approval process.

4 System Requirements and Installation

4.1 System Requirements

4.1.1 Supported Windows Versions

The current TraceFab build is intended for **64-bit Windows 10 and Windows 11**. These are the Windows versions used for development and testing. The application is published as a win-x64 Windows desktop application, and the installer explicitly uses 64-bit installation mode on x64-compatible systems.

Windows 7 has not been tested and is not supported. A 32-bit x86 build is not currently provided or tested. TraceFab should therefore not be expected to install or operate correctly on 32-bit Windows, Windows 7, non-Windows systems, or unsupported compatibility layers.

TraceFab is distributed as a self-contained .NET 9 Windows desktop application. The required .NET runtime components are included with the installed and portable packages, so a separate .NET installation is normally not required.

4.1.2 Hardware Recommendations

TraceFab is a desktop engineering application and does not require workstation-class hardware for normal PCB projects. No formal minimum-hardware certification has been completed, but the following practical baseline is recommended: a modern x64 processor, at least 4 GB of RAM, and approximately 500 MB of free disk space for installation, temporary files, updates, and project output. For larger boards or regular use, 8 GB of RAM or more is preferable.

A display resolution of at least 1280 × 720 is recommended, with Full HD or higher providing a more comfortable workspace for the PCB preview and operation panels. A mouse with a scroll wheel is strongly recommended for efficient zooming and navigation. Very high-resolution or scaled displays should use a Windows scaling configuration that keeps all forms and controls readable.

TraceFab generates files for later transfer to a CNC controller; it does not require a direct machine connection. Provide suitable local, network, or removable storage according to the workflow used to transfer approved G-code to the machine.

4.1.3 Required Permissions

The Windows installer places TraceFab in the standard program-files location and may request administrator approval through User Account Control. The account running TraceFab must be able to read the selected source files and write to the folders used for jobs, generated G-code, settings, and exported data.

The portable package does not require a traditional installation, but it must be fully extracted to an accessible folder before use. Do not run the application directly from inside the ZIP archive. Internet access is required for automatic activation and online update or news functions; an offline or file-based licensing workflow may require separate license files supplied through the official TraceFab process.

4.2 Installing TraceFab

4.2.1 Using the Windows Installer

Download the current installer from the official TraceFab website and close any running TraceFab instance. Start the setup executable and follow the installation wizard. The wizard presents the applicable license information, installs the complete application, and can create a desktop shortcut before optionally launching TraceFab.

Use the default installation folder unless there is a specific administrative reason to select another location. Do not manually remove DLLs or runtime files from the installed directory: the package is self-contained and these files are required by the application.

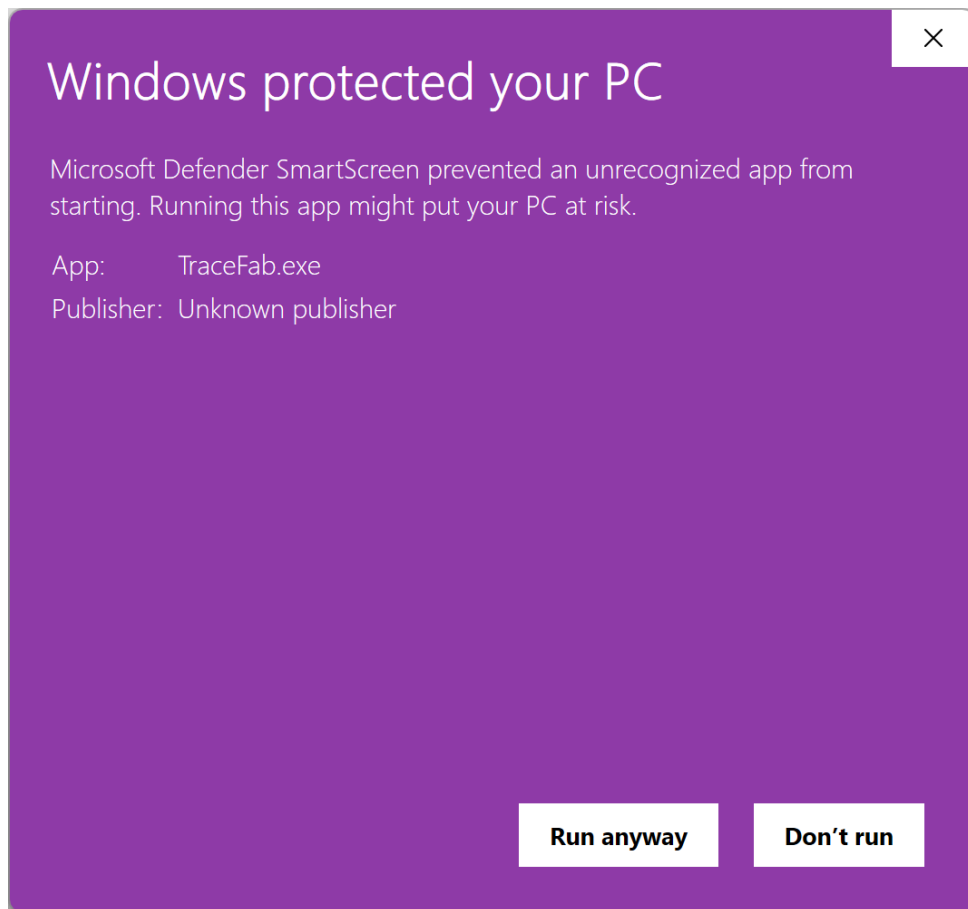


Figure 4-1 – SmartScreen warning, click “More Info” then “Run Anyway”

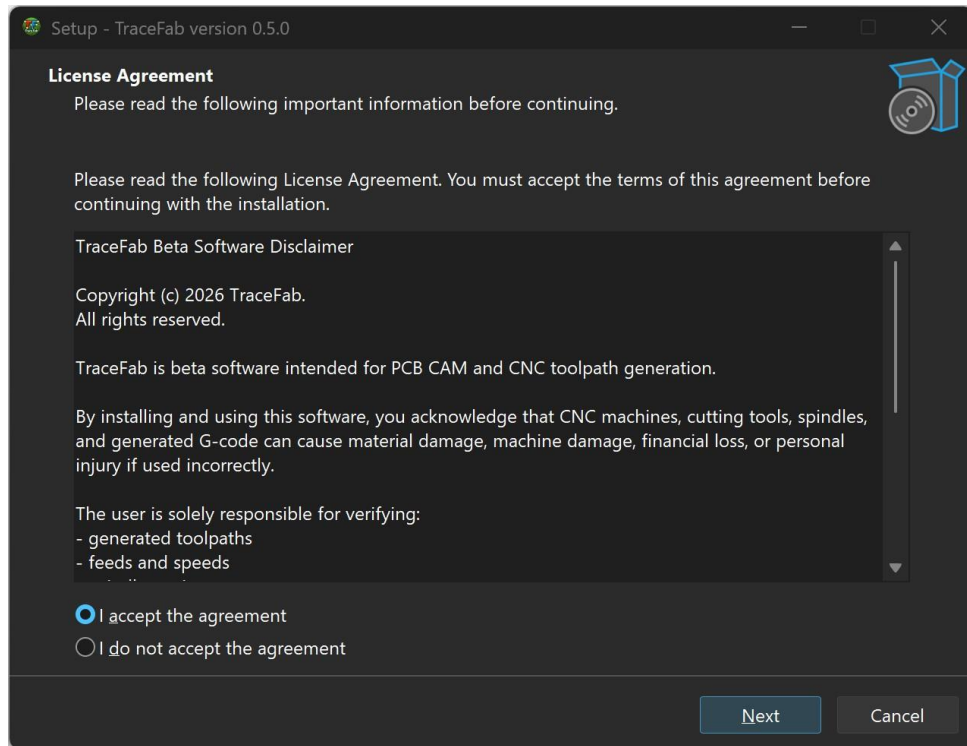


Figure 4-2 - Accept the TraceFab beta license agreement.

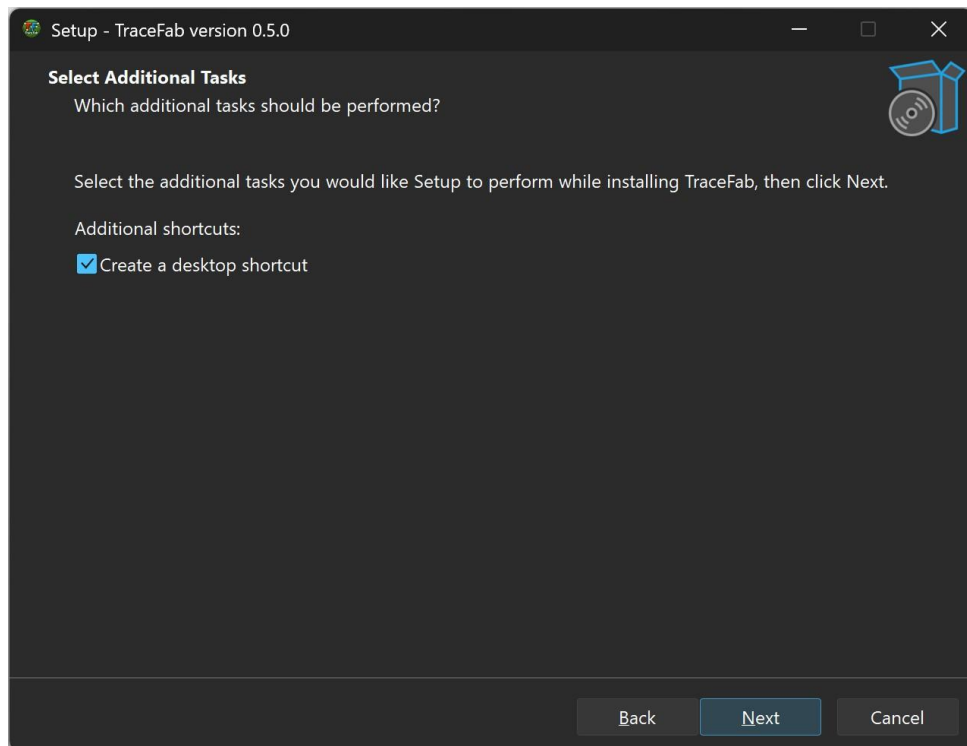


Figure 4-3 - Choose whether Setup should create a desktop shortcut.

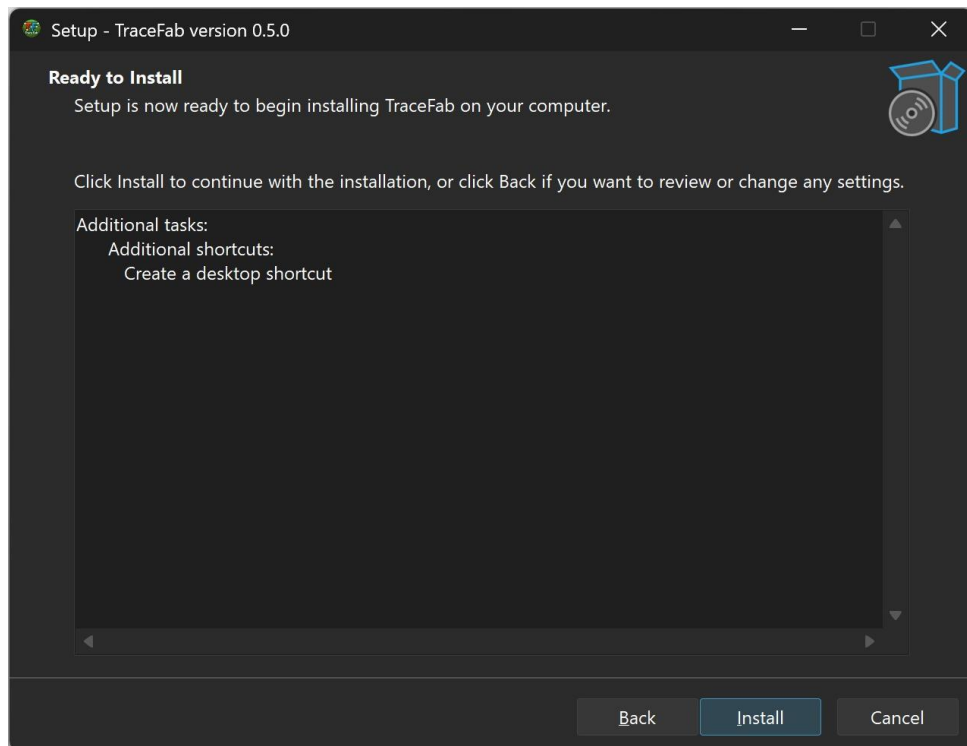


Figure 4-4 - Review the installation options before installing TraceFab.

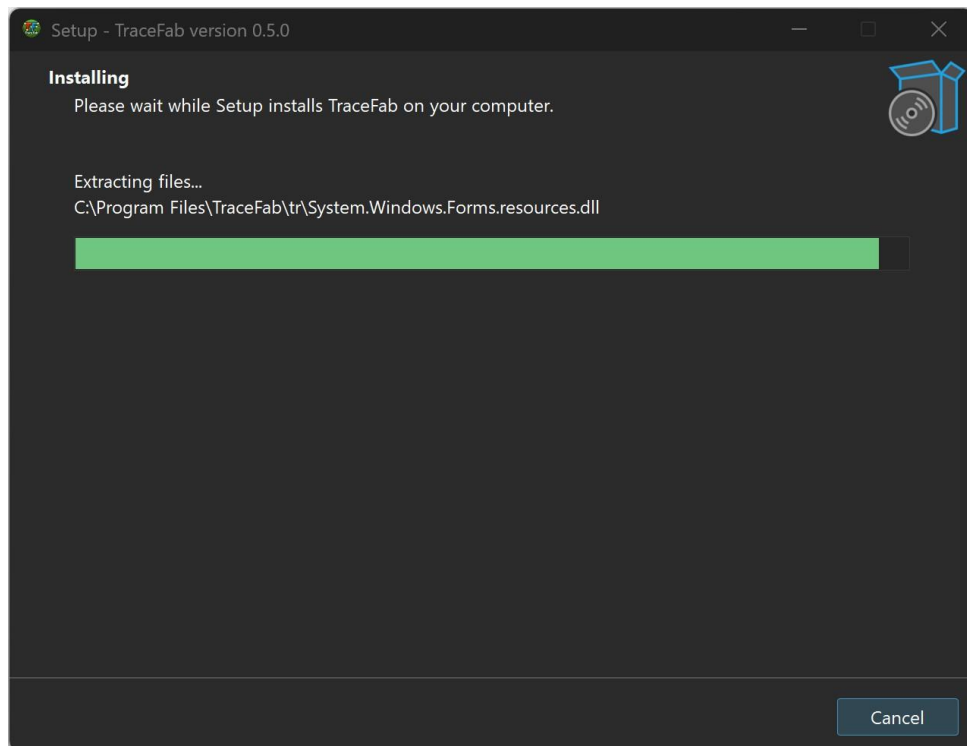


Figure 4-5 - Setup copies the required TraceFab application files.

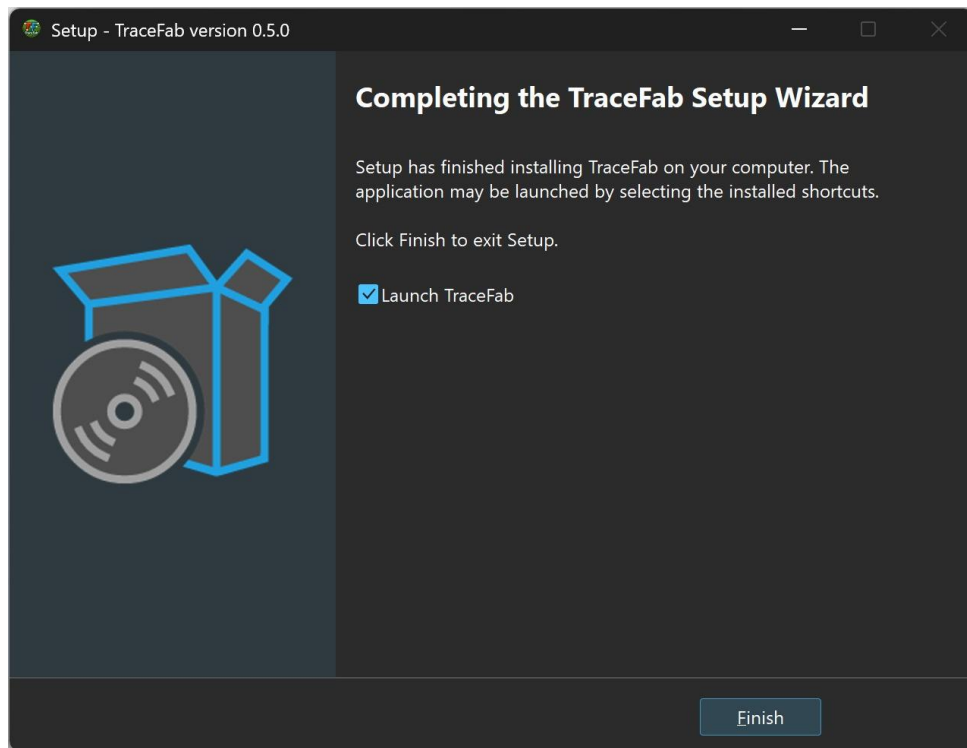


Figure 4-6 - Complete Setup and optionally launch TraceFab.

4.2.2 Using the Portable Package

The portable ZIP contains the same self-contained application in a form that does not use the installation wizard. Download the ZIP from the official TraceFab website, extract the complete archive to a dedicated folder, and start TraceFab.exe from that extracted folder.

Keep the extracted files and subfolders together. Do not move only the executable, and do not launch it from the compressed archive. When replacing a portable version, use a new folder or make a backup first so that settings, license files, samples, or manually stored project data are not accidentally overwritten.

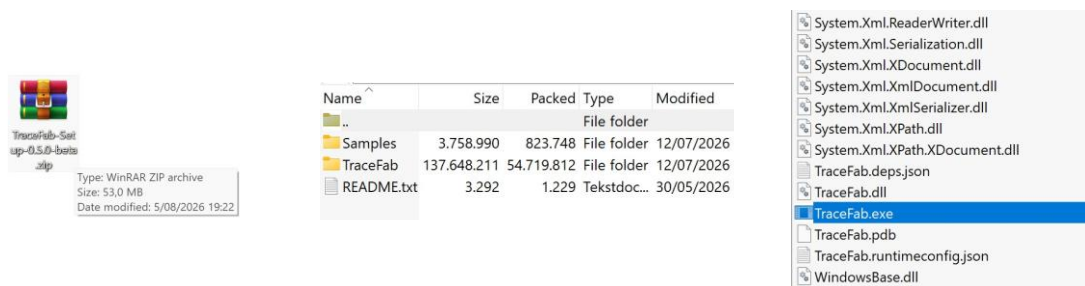


Figure 4-7 - Portable package workflow: download the ZIP archive, extract its contents, and start TraceFab.exe.

4.2.3 Windows Security Messages

Beta builds may be unsigned or may not yet have sufficient download reputation. Microsoft Edge, Google Chrome, Windows SmartScreen, or Smart App Control may therefore display a warning or block the file even when it was obtained from the official website.

Only continue when the file was downloaded directly from the official TraceFab website and its name and source are as expected. Read the complete Windows warning before making a decision. Security controls exist to protect the computer; do not disable them globally merely to

install TraceFab. If company policy blocks unsigned applications, consult the system administrator.

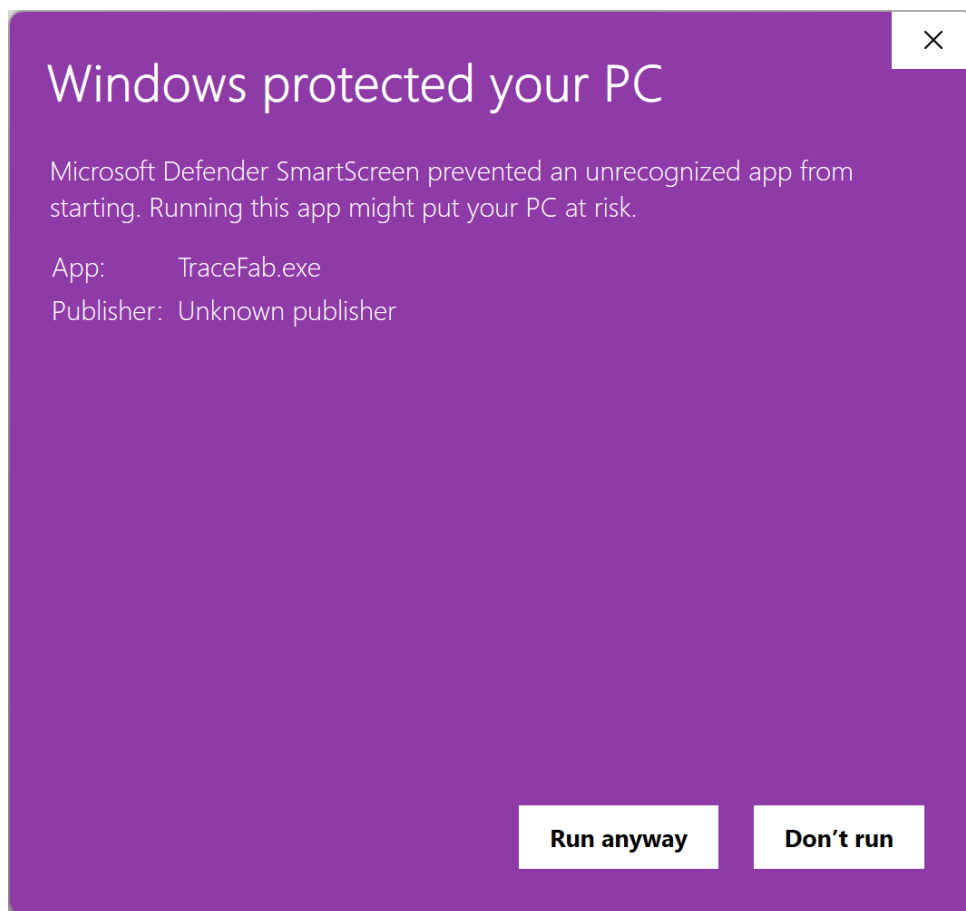


Figure 4-8 – SmartScreen Warning. Click first on More Info and then Run Anyway

4.3 First Start

4.3.1 Launching TraceFab

After installation, start TraceFab from the desktop shortcut or the Windows Start menu. For a portable installation, open the extracted folder and run TraceFab.exe. The application should open in the main workspace and may present activation, news, or update information during startup. Tip: You can create a desktop shortcut for quick access to TraceFab. Open the extracted portable folder, right-click TraceFab.exe, and select Send to > Desktop (create shortcut).

4.3.2 Initial Application Checks

On the first start, confirm that the main workspace, menus, operation controls, and PCB preview area are displayed correctly. If sample PCB files are included with the distribution, open a sample before working with production data. This provides a quick check that importing, preview rendering, and the basic workflow operate correctly on the computer.

Complete the required activation process and review any version-specific release notes or known issues. Do not connect the first generated program to a production machine until the safety and G-code verification chapters of this manual have been followed.

4.3.3 Default Storage Locations

Source PCB files and generated G-code may be stored in user-selected project folders. Use a location where the current Windows account has write permission; do not use the protected application installation folder as a general project workspace.

TraceFab stores its application settings and other persistent data in %APPDATA%\TraceFab.

On a standard Windows installation, this normally resolves to

C:\Users\<username>\AppData\Roaming\TraceFab. The folder is separate from the installed or extracted application files and is used by both the installed and portable versions. This allows persistent data to remain available when TraceFab is updated or when the application files are replaced. Do not manually edit or delete this folder unless TraceFab is closed and a backup has been created.

4.4 Updating or Removing TraceFab

4.4.1 Installing an Update

Close TraceFab before applying an update. Download the new installer or portable package from the official website and read the release information for changes that may affect files, settings, supported input formats, or generated G-code. After updating, open a known sample and verify the workflow before using the new version for production.

Installer builds can normally be installed through the setup wizard. For portable builds, extracting the new version to a separate folder is safer than overwriting the previous folder because it preserves a known working copy for comparison or rollback.

4.4.2 Preserving User Data

Before updating, repairing, moving, or removing TraceFab, back up the complete %APPDATA%\TraceFab folder, together with PCB source files, generated CNC programs, verified parameter records, and any other user-created data. Project files and generated G-code may be stored outside TraceFab in locations selected by the user and must therefore be backed up separately. Do not assume that files manually stored inside an installation or portable application folder will be preserved when that folder is replaced or removed.

Keep verified production programs under revision control or in clearly versioned job folders. After an update, regenerate and review G-code rather than assuming that output from a changed software version is identical.

4.4.3 Uninstalling the Application

An installed copy of TraceFab can be removed through the Windows Installed apps or Apps & features settings. Close the application first, select TraceFab, and follow the uninstall procedure. Review and back up user-created files before removal.

A portable copy can be removed by deleting its extracted folder after all required data has been backed up. Removing local application files does not necessarily remove licensing or activation records maintained outside that folder.

5 Licensing and Activation

5.1 License Types

TraceFab can be used in three licensing states: without an activated license as a limited demo, with a temporary Pro trial, or with an activated Pro license. The current license state controls which CNC operations are available. Licensing does not replace the operator's responsibility to review the generated G-code and verify every machine operation.

5.1.1 Demo Mode (No License)

TraceFab starts without activation, allowing new users to explore the interface and evaluate the basic workflow. When no valid license is installed, the News panel displays a License invalid message and **Help > Activation** reports **No valid license**. CNC functionality is deliberately limited in this state and is intended for demonstration and evaluation only.

A missing license does not prevent TraceFab from opening. To temporarily evaluate all Pro features, request a trial. To keep Pro functionality, activate a Pro license for the computer.

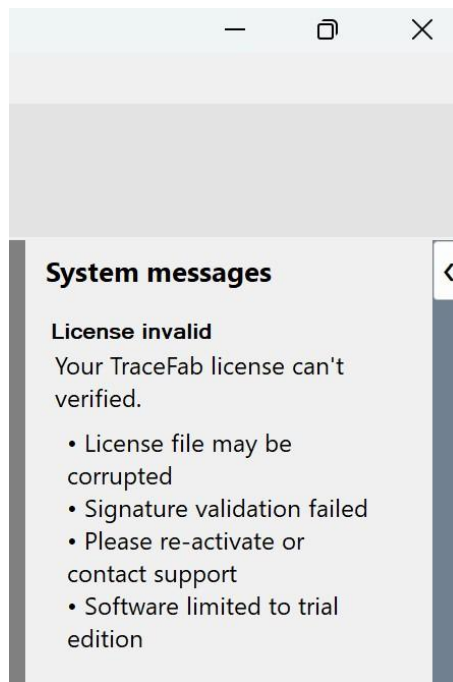


Figure 5-1 - The News panel reports that no valid TraceFab license is available.

5.1.2 Trial Activation

A free temporary Pro trial can be requested directly from TraceFab. The request requires an active internet connection because the application contacts the activation service.

1. Open **Help > Activation**.
2. Confirm that the computer is connected to the internet.
3. Select Request Trial and wait for the activation response.
4. After a successful request, review the displayed license type, status, and trial expiration date.

The trial is free but time-limited. When its displayed period ends, Pro functionality expires and TraceFab returns to the limited demo state unless a Pro license has been activated.

Figure 5-2 - The Activation window provides the machine fingerprint and the Request Trial command.

5.1.3 Pro License

A Pro license enables the complete set of licensed CNC operations. During the TraceFab beta period, a perpetual Pro license can currently be claimed free of charge through Trace-Fab.com. A Pro license issued through this beta offer remains assigned to the machine fingerprint for which it was generated. After the beta period, new Pro licenses are expected to be available for purchase through the TraceFab website. Pricing, checkout conditions, and license terms shown on the website at the time of the request take precedence over this manual. A temporary trial and a perpetual Pro license are different: the trial expires on the displayed date, while the beta Pro license does not expire.

Windows Installer

Recommended for most users.

Download Windows Installer

Includes desktop shortcut and uninstaller

Portable Version

No installation required.

Download ZIP

Extract and run TraceFab.exe

Requirements

- ✓ **Operating system:** Windows 10, 11
- ✓ **Architecture:** 64-bit (x64)
- ✓ **Activation:** Free Pro license during beta
- ✓ **Offline use:** Fully supported

Free during beta

Activate your perpetual Pro license

Install TraceFab, copy the machine fingerprint from the activation window, and use it to claim your Pro license. Activation is free while TraceFab is in beta, and the license you claim is yours forever.

You will need the fingerprint code shown inside TraceFab.

Activate Free Pro License

Attention! We just released beta 0.5.0 with Gerber Support. See the screenshots in the Features section. The download links are now active. For help, please join our Discord community or contact us using the form below.

Figure 5-3 - The TraceFab website provides access to the Pro-license request during the beta period.

5.1.4 License Status

Open **Help > Activation** at any time to review the current Type, Expiration Date, Flag, and Status. A trial shows its remaining validity through the displayed expiration information. A successfully activated Pro license should be reported as valid and should no longer produce the License invalid warning in the News panel.

5.2 Machine Identification

5.2.1 Machine Fingerprint

TraceFab generates a machine fingerprint that identifies the computer for licensing purposes. Pro license data is generated for this fingerprint and should therefore be used only on the corresponding computer. The fingerprint is displayed in **Help > Activation**.

5.2.2 Copying the Fingerprint

Open **Help > Activation** and select Copy beneath the Fingerprint ID. Paste the complete value into the Machine Fingerprint field on Trace-Fab.com exactly as copied. Do not add spaces, remove characters, or manually retype the value. Figure 5-2 shows the location of the fingerprint and Copy command.

5.2.3 Hardware Changes

Significant hardware changes or moving the application to another computer may produce a different fingerprint. A license generated for the previous fingerprint may then fail validation. If the fingerprint has changed, request the appropriate license for the new fingerprint or contact TraceFab support for assistance.

5.3 Activating TraceFab

5.3.1 Automatic Online Activation

The Request Trial command performs the automatic online activation described under Trial Activation. This is the quickest way to evaluate Pro functionality temporarily. Keep TraceFab open while the request is processed and read any response shown in the Messages area before closing the Activation window.

Trial activation requires an outbound HTTPS connection from TraceFab to trace-fab.com over TCP port 443. No inbound firewall rule is required. TCP port 80 is not used for activation. On restricted company networks, the domain and application may need to be allowed by the network administrator.

5.3.2 Activating with Existing License Files

To request a full Pro license, first copy the fingerprint from **Help > Activation**. Visit Trace-Fab.com, select Activate Free Pro License during the beta period, and complete the license request or checkout form. Enter the requested customer details, paste the fingerprint exactly as copied, confirm that the license will be generated for that machine, and submit the request. The checkout is currently free during beta; later releases may require payment according to the conditions published on the website.

Pro License Request

Fill in your details below and paste the fingerprint exactly as shown inside the TraceFab application.

Full Name

John Doe

Email

you@example.com

Company Name

Your company

VAT Number

BE0123.456.789

Recommended for business customers. This will be used for invoicing.

Street + Number

Street and number

Postal Code

1000

City

Brussels

Country

Belgium

Machine Fingerprint

Paste fingerprint from TraceFab

☐ This license will be generated for this machine fingerprint

Get Free Pro License

How it works

This is the current beta license flow for TraceFab Pro.

- ✓ Open TraceFab, go to Help > Activation
- ✓ Copy your machine fingerprint
- ✓ Fill in your name, email and address
- ✓ Paste your machine fingerprint
- ✓ Submit the request for your free Pro license
- ✓ The license will be generated for this machine

Fingerprint

Fingerprint ID

29b696a0ef80c17bb207ea22f5d1d550b81de989ec02820cc3

Export

Copy

Copy the fingerprint from the activation window in TraceFab.

Pro License Benefits

- ✓ Unlimited contour toolpaths
- ✓ Edge-cuts and tabs support
- ✓ Single and double sided boards
- ✓ No output limitations
- ✓ Perpetual license – no subscription
- ✓ Works fully offline after activation
- ✓ No internet needed for activation

Figure 5-4 - Enter the machine fingerprint and requested details in the Pro License Request form.

After completing the website process, follow the supplied delivery instructions for the license data. In Help > Activation, use License and Signature to select the corresponding supplied files, then select Activate. The license and signature belong together and must match the fingerprint shown in the Activation window. Do not modify either file.

5.3.3 Verifying Activation

After activation, review the License Status section in **Help > Activation**. Confirm that the expected license type and a valid status are shown. For a trial, also confirm the expiration date. Close and reopen TraceFab if requested, then verify that the License invalid message has disappeared and that the expected CNC operations are available.

License information and other persistent TraceFab data are stored under %APPDATA%\TraceFab. Do not delete this folder while diagnosing an activation problem unless a backup has been made and TraceFab support has instructed you to reset the local data.

5.4 Activation Troubleshooting

5.4.1 Internet Connection Problems

If Request Trial fails, verify that the computer has internet access and that the system date and time are correct. A firewall, proxy, security product, or company network policy may prevent TraceFab from reaching the activation service. Do not disable security protection globally; allow

the connection only when permitted, or consult the network administrator. The website-based Pro request also requires access to Trace-Fab.com.

5.4.2 Invalid or Blocked Activation

If TraceFab reports an invalid license, compare the current fingerprint with the fingerprint used for the license request. Confirm that the matching license and signature were selected and that neither file was renamed, edited, or corrupted. An expired trial, mismatched files, or a fingerprint changed by hardware replacement can prevent validation. Repeating activation with unrelated files will not correct a fingerprint mismatch.

5.4.3 Contacting Support

If activation still fails, contact TraceFab through the official support channel. Include the TraceFab version, Windows version, license type, exact status or message, and whether the fingerprint recently changed. Share license files and identifying information only through an official private support channel; do not publish them in public forums or screenshots.

6 User Interface Overview

6.1 Main Workspace

6.1.1 Workspace Layout

The TraceFab main workspace keeps project identification, navigation, common manufacturing commands, status information, and the PCB preview within one window. Figure 6.1 identifies the five primary interface areas used throughout this manual.

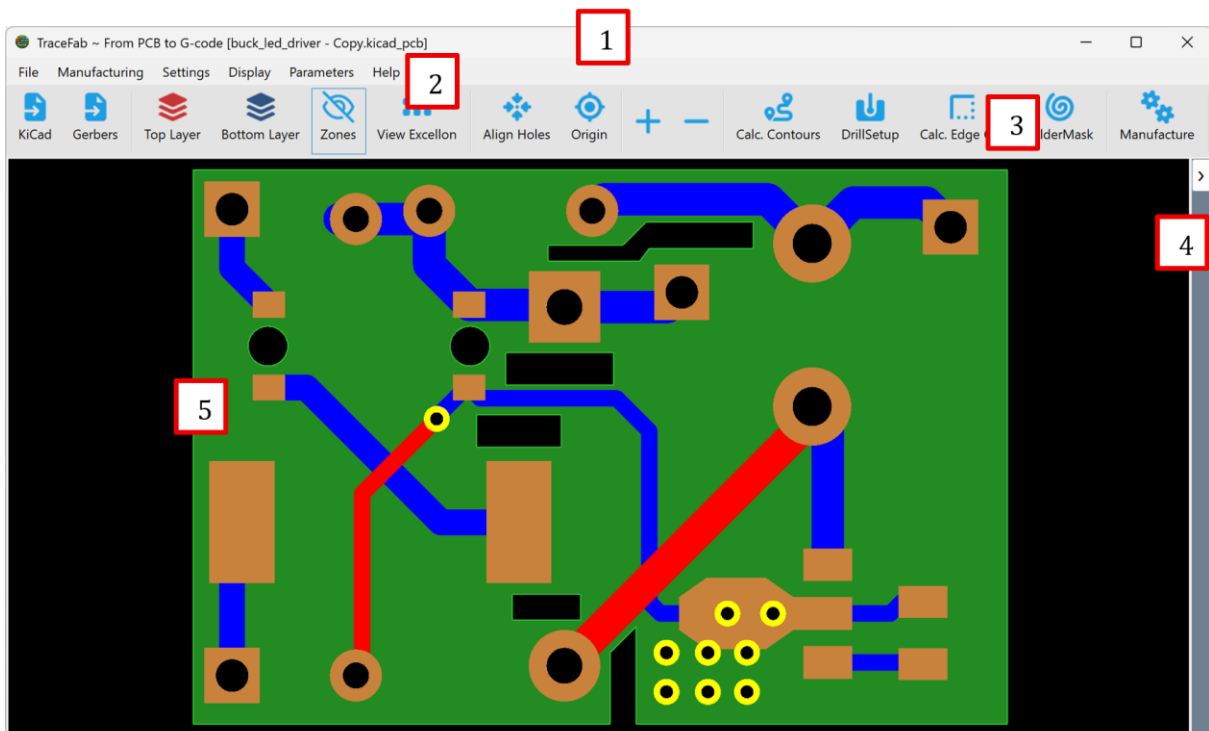


Figure 6.1 - Main TraceFab workspace and its primary interface areas.

The numbered callouts correspond to the following interface areas:

1 - Title bar: Displays the TraceFab application name and the currently opened PCB or project file. The standard Windows minimize, maximize, and close controls are located at the right.

2 - Menu bar: Provides access to file commands, manufacturing functions, settings, display options, parameters, activation, and help.

3 - Main toolbar: Provides quick access to frequently used import, layer-display, alignment, origin, toolpath, drilling, edge-cutting, and manufacturing commands.

4 - News and system messages panel: A collapsible panel for TraceFab news, license information, update notices, and system messages. Use the arrow at the right edge of the workspace to expand or collapse it.

5 - PCB viewport: Displays the imported PCB geometry, visible copper layers, drill locations, board outline, origin, and generated graphical results used while preparing the CNC process.

6.1.2 Application Menu

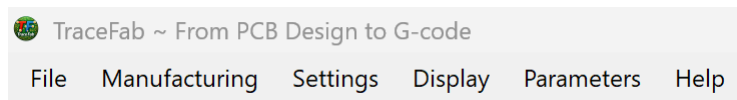


Figure 6.2 - TraceFab application menu.

The application menu provides access to all principal commands. Commands shown with a strikethrough in TraceFab are placeholders for planned functionality and are not available in the current beta.

File menu

- **File > Import Project:** Reserved for a future project-based import workflow; this command is not available in the current beta.
- **File > Board Dimensions:** Shows the width and height derived from the imported PCB boundaries and allows the board thickness to be overwritten, for example with the common value of 1.60 mm.
- **File > Save Project:** Reserved for saving a complete TraceFab project; this command is not available in the current beta.
- **File > Close Project:** Reserved for closing and clearing a saved TraceFab project; this command is not available in the current beta.
- **File > Import Info:** Displays the technical report from the most recent PCB import, or reports that no import information is available. Currently not functional.
- **File > Test Samples:** Provides access to the supplied sample files so that importing and the basic workflow can be tested without using production data.

Manufacturing menu

- **Manufacturing > Calc. Contours:** Opens the contouring setup used to calculate isolation toolpaths around imported copper geometry.
- **Manufacturing > Calc. Edge Cut:** Opens the edge-cut setup for preparing the inner and outer board-outline toolpaths, including holding gaps or tabs where applicable.
- **Manufacturing > Drill Setup:** Groups imported Excellon holes, assigns tools, and prepares the enabled drilling operations for manufacturing.
- **Manufacturing > Solder Mask:** Opens the solder-mask removal setup for generating toolpaths over selected pads and/or vias.

- **Manufacturing > Alignment Pins:** Configures reference holes or pins used to align the PCB when machining double-sided boards.
- **Manufacturing > Hatching:** Reserved for a future hatching operation; this command is not available in the current beta.

Settings menu

- **Settings > Program Settings:** Opens the software preferences for TraceFab, including UI, file, import, user-experience, and logging settings. It does not define CNC machining parameters. Some settings remain under development during the beta period.

Display menu

- **Display > Reset Zoom:** Returns the PCB viewport to its default zoom level.
- **Display > Zoom In:** Magnifies the PCB viewport in incremental steps.
- **Display > Zoom Out:** Reduces the PCB viewport magnification in incremental steps.
- **Display > Top Layer Preview:** Shows or hides the top-copper preview and, when selected, switches off the bottom-layer and drill-only previews.
- **Display > Bottom Layer Preview:** Shows or hides the bottom-copper preview and, when selected, switches off the top-layer and drill-only previews.
- **Display > Excellon Preview:** Switches the viewport to highlight the imported Excellon drill-hole preview.

Parameters menu

- **Parameters > Tool Library:** Opens the tool library used to create and maintain the drills, cylindrical cutters, and conical tools assigned to CNC operations.
- **Parameters > CAM Parameters:** Opens the CNC machining defaults for safe heights, units, spindle speeds, feeds, depths, passes, overlaps, tool choices, edge cutting, solder-mask removal, and alignment operations. This is a separate form from Program Settings.
- **Parameters > G-Code:** Reserved for future G-code configuration; this command is not available in the current beta.
- **Parameters > Post-Processor:** Reserved for future controller-specific output rules; this command is not available in the current beta.
- **Parameters > Job Composer:** Reserved for a future workflow that combines reusable operations into a complete CNC job; this command is not available in the current beta.

Help menu

- **Help > Activation:** Opens the license status, machine fingerprint, trial request, and Pro-license activation controls.
- **Help > Contact:** Provides the TraceFab contact route for support, questions, and product feedback. Contactform en e-mail will be provided in the next release.
- **Help > About:** Displays product and version information for the installed TraceFab build.

6.1.3 Toolbar and Main Actions



Figure 6.3 - TraceFab main toolbar.

The main toolbar provides one-click shortcuts to the commands used most frequently while importing, viewing, configuring, and manufacturing a PCB. The corresponding menu command and later workflow chapters provide additional context where required.

- **KiCad:** Imports a KiCad .kicad_pcb board file.
- **Gerbers:** Starts the Gerber and Excellon import wizard.
- **Top Layer:** Switches the viewport to the top-copper layer.
- **Bottom Layer:** Switches the viewport to the bottom-copper layer.
- **Zones:** Toggles filled copper zones in the viewport. Zones set to Invisible are also excluded from contour calculation and generated G-code.
- **View Excellon:** Switches the viewport to the imported drill-hole data.
- **Align Holes:** Opens the alignment-pin setup for double-sided PCB positioning.
- **Origin:** Currently not yet implemented.
- **Zoom In (+):** Magnifies the PCB viewport.
- **Zoom Out (-):** Reduces the PCB viewport magnification.
- **Calc. Contours:** Opens the isolation-contour calculation.
- **DrillSetup:** Opens the Excellon hole and drilling-tool assignment.
- **Calc. Edge Cut:** Opens the board-outline milling setup.
- **SolderMask:** Opens the solder-mask removal toolpath setup.
- **Manufacture:** Opens the manufacturing workspace after CAM operations have been prepared, allowing the job to be assembled and processed toward G-code output.

6.2 PCB Preview

6.2.1 Layer Display

The Top Layer and Bottom Layer buttons toggle the corresponding copper-layer preview on or off. This makes it easier to distinguish the geometry belonging to each side of the PCB while reviewing an imported board and preparing its manufacturing operations.

The Zones toggle works together with the Top Layer and Bottom Layer controls. When Zones is set to Visible (default), selecting a layer also displays the copper zones belonging to that layer. When Zones is set to Invisible, those zones are hidden from the viewport.

Important: the Zones toggle also controls processing. Zones that are set to Invisible are excluded from contour calculation and will not be included in the generated G-code. This allows complex zones to be excluded as a post-processing choice, for example when a zone produces an unwanted or problematic rendered result. Always verify the visible geometry before calculating contours so that required copper zones are not accidentally omitted from the CNC program.

6.2.2 Zooming and View Reset

Use the plus (+) and minus (-) buttons on the main toolbar, or the corresponding commands under Display, to zoom the PCB viewport in or out. **Display > Reset Zoom** returns the viewport to its default zoom level.

Viewport panning is not available in the current release. It is planned as a future roadmap feature; the present interface is limited to zooming and resetting the view.

6.3 Status and Messages

6.3.1 News and Update Notifications

When TraceFab starts, it queries the online TraceFab news feed when an internet connection is available. Release announcements, important product news, and notifications about newly available updates can then be shown in the collapsible News and system messages panel.

TraceFab remains usable when the news query cannot be completed, but current online announcements cannot be retrieved without internet access. The panel can be expanded or collapsed using the arrow at the right edge of the main workspace.

6.3.2 License Warnings

The panel also displays licensing warnings, for example when no license is installed, a trial has expired, or the current license cannot be validated. At present, this panel is used primarily for online news, update notifications, and license warnings; it is not a complete diagnostic log for every application error. Other workflow-specific warnings are presented in the relevant TraceFab forms or message dialogs.

7 Importing PCB Data

TraceFab supports two PCB import workflows: direct import of a KiCad board file and import of Gerber manufacturing files with Excellon drill data. Both workflows create the geometry used for previewing, CAM preparation, and G-code generation. The best choice depends on the source data available and the level of compatibility required.

7.1 Preparing an Import

7.1.1 Choosing an Import Workflow

- **KiCad (.kicad_pcb):** Use this workflow when the original KiCad board file is available. It provides the most direct route because the board geometry and design information are stored in one structured file.
- **Gerber and Excellon:** Use this workflow for standard PCB manufacturing outputs. Copper, board-outline, and drill information are supplied as separate files and combined during import.

7.1.2 Preparing Source Files

Before importing, work from a copy of the source data and keep the original PCB design files unchanged. Confirm that the board outline is closed and that all required copper and drill data have been exported. For Gerber-based projects, keep the related files together in one folder so that layer assignments can be checked easily.

An import should always be treated as the start of the CAM workflow, not as final proof that the design is ready for machining. Review the resulting geometry in TraceFab before calculating contours or generating G-code.

7.2 KiCad PCB Import

7.2.1 Why TraceFab Supports KiCad

A KiCad board is stored as a vector-based **.kicad_pcb** file. The file contains the board structure and its relevant design parameters in a single source. TraceFab can therefore read and parse the required PCB information on the fly without first asking the user to export a separate file for every layer.

This single-file workflow is convenient during design iterations: update the board in KiCad, save it, and import the updated board into TraceFab for a new CAM pass.

7.2.2 Loading a KiCad Board

5. Click **KiCad** on the main toolbar.
6. Select the required **.kicad_pcb** file and confirm the selection.
7. Allow TraceFab to parse the board. The imported geometry is then displayed in the PCB preview.
8. Use the Top Layer, Bottom Layer, Zones, and View Excellon controls to inspect the imported content.

7.2.3 KiCad Version Compatibility

The KiCad file format can change between KiCad releases. A newly released KiCad version may therefore introduce data that the current TraceFab parser does not yet recognise. TraceFab will be updated when such changes require parser adjustments, but compatibility with a new KiCad release should not be assumed until the imported board has been checked.

Some KiCad features may also use a relative anchor or reference that differs from the coordinate relationship expected by TraceFab. Unsupported anchoring can result in an import error or geometry appearing at an incorrect position. If this occurs, verify the source item in KiCad and consider using the Gerber workflow as a compatibility fallback.

7.2.4 Reviewing Imported KiCad Geometry

Check the following before continuing:

- The top and bottom copper layers appear on the correct side of the board.
- Pads, tracks, vias, holes, and the board outline are present and positioned correctly.
- Copper zones appear as expected, or are intentionally hidden when they must be excluded from contouring and G-code generation.
- The board dimensions and thickness match the physical PCB material.

7.3 Gerber and Excellon Import

7.3.1 Why Use Gerber Files

Gerber RS-274X is an established industry format for transferring PCB manufacturing geometry. It is widely supported by PCB design applications and fabrication services, so Gerber RS-274X import is an essential alternative when a native KiCad board file is unavailable or when a format-independent manufacturing export is preferred.

Unlike the KiCad workflow, a Gerber-based job normally consists of several files. Copper and board-edge geometry use Gerber files, while plated and non-plated drill coordinates are typically supplied as Excellon files.

7.3.2 Selecting Gerber and Drill Files

9. Click **Gerbers** on the main toolbar to open the Gerbers Import Window.

10. Use the Browse command for each available layer and select the corresponding file. The wizard provides entries for Front Copper Layer, Back Copper Layer, Excellon (PTH), Excellon (NPTH), and Edge Cut.
11. Confirm that every file is assigned to the correct layer. Use Clear to remove an incorrect selection before continuing.

7.3.3 Units, Coordinate Format, and Zero Handling

Gerber and Excellon coordinates must be interpreted with the correct units and number format. A mismatch can scale, shift, or otherwise corrupt the imported geometry. TraceFab analyses the selected files and reports the detected Type, Units, Mode, Format, Zeroes, Status, and Message in the lower part of the wizard.

Review these detected values before importing. In particular, verify that millimetres or inches are interpreted correctly and that the coordinate format and leading-zero handling match the exported files. Files that belong to the same board should resolve to compatible coordinate settings and show a valid status.

The screenshot shows the 'Gerbers Import Window' with the title 'Import Gerber / Excellon'. It instructs the user to 'Select copper and drill files. TraceFab validates them before import.' The window contains a table for selecting files and a section for detected file details.

Layer	File Path	Browse	Clear	Status
Front Copper Layer	C:\Users\... \Desktop\gerber\Buck\buck_led_driver-F_Cu	...	X	-
Back Copper Layer	C:\Users\... \Desktop\gerber\Buck\buck_led_driver	...	X	-
Excellon (PTH)	C:\Users\... \Desktop\gerber\Buck\buck_led_driver	...	X	-
Excellon (NPTH)	C:\Users\... \Desktop\gerber\Buck\buck_led_driver	...	X	-
Edge Cut	C:\Users\... \Desktop\gerber\Buck\buck_led_driver	...	X	-
Front Solder Mask		...	X	-
Back Solder Mask		...	X	-

Below the table is a section titled 'Detected file details' with a table of file analysis results. A green button 'Import / Let's Go!' is located to the right of the table.

Layer	Type	Units	Mode	Format	Zeroes	Status	Message
F.Cu	Gerber	mm	Absolute	4.6 / 4.6	Leading	OK	Ready
B.Cu	Gerber	mm	Absolute	4.6 / 4.6	Leading	OK	Ready
Excellon	Excellon	mm	Absolute	-	Unknown	OK	Ready
Excellon	Excellon	mm	Absolute	-	Unknown	X	No drills found
EdgeCut	Gerber	mm	Absolute	4.6 / 4.6	Leading	OK	Ready

Figure 7.1 - Gerber and Excellon import wizard with detected file details and validation status.

7.3.4 Completing the Gerber Import

When the required files show a valid status and the detected settings are consistent, click **Import / Let's Go!** TraceFab imports the selected files and displays the combined board geometry in the main workspace. A missing optional file may be left unassigned, but any required copper, drill, or outline information must be available before the corresponding CNC operation can be prepared.

7.3.5 Solder Mask Support

Direct import of front and back solder-mask Gerber files is not yet available and remains on the TraceFab roadmap. The corresponding entries in the import workflow should therefore not be treated as completed functionality in the current release.

TraceFab does include a manual solder-mask generator. Open **Manufacturing > Solder Mask** or use the **SolderMask** toolbar command to generate the required solder-mask removal geometry. This generates removal toolpaths from the imported board data; it is not the same as importing an existing solder-mask Gerber file.

7.4 Validating Imported Geometry

7.4.1 Copper Layers and Zones

Toggle the Top Layer and Bottom Layer views separately and compare them with the source design or Gerber viewer. For KiCad imports, also verify the Zones state. Hidden zones are excluded from contouring and G-code generation, which can be useful when a complex zone produces a rendering or post-processing problem.

7.4.2 Drill Data and Board Outline

Use View Excellon to check that plated and non-plated holes are present at the expected coordinates. Confirm that the Edge Cut geometry follows the intended board boundary and forms a usable outline. Missing drill data or an incomplete board edge must be corrected before drilling or edge-cutting operations are created.

7.4.3 Recognising and Correcting Import Problems

Geometry that is extremely large, very small, shifted, mirrored unexpectedly, or internally misaligned usually indicates an import or source-data problem. Do not compensate for such an error by changing CNC dimensions manually. Close or clear the incorrect import, verify the source files and coordinate settings, and import the project again.

If a KiCad-specific feature cannot be parsed correctly, export a complete Gerber and Excellon manufacturing set from KiCad and use the Gerber workflow. Always perform a final visual comparison before calculating toolpaths.

8 Contouring and Isolation Toolpaths

Contouring creates isolation toolpaths around copper tracks, pads, vias, and copper areas. The calculated paths remove a controlled channel of copper so that electrically separate features remain isolated after machining. Contours are calculated from the imported PCB geometry and are stored for the later Manufacture and G-code generation stages.

8.1 Contouring Workspace

8.1.1 Opening the Contouring Window

Import and visually validate the PCB before calculating any contours. Click **Calc. Contours** on the main toolbar, or select **Manufacturing > Calc. Contours**. The Calculate Contours Window opens while the PCB remains visible in the main preview.

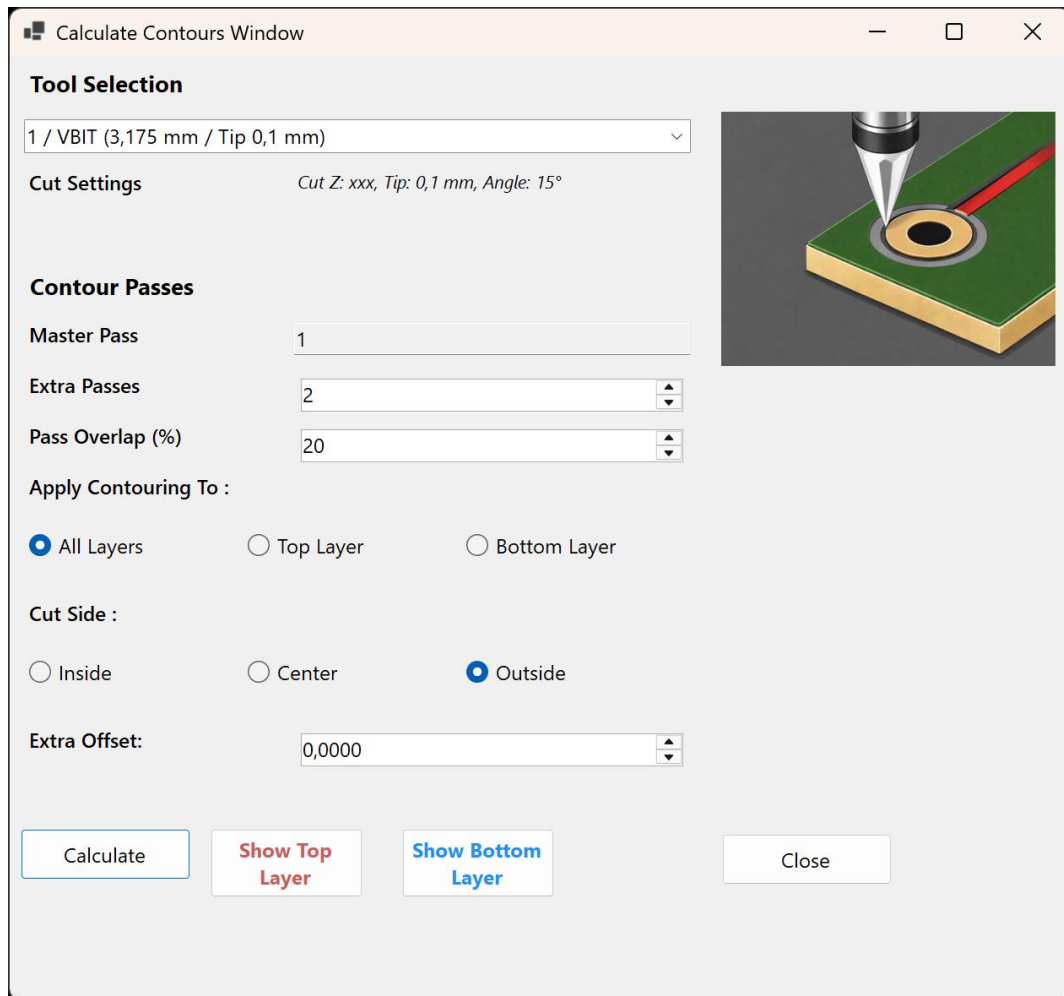


Figure 8.1 - Calculate Contours Window with tool, pass, layer, cut-side, and offset controls.

8.1.2 Selecting a Contour Tool

Select the required contour tool from the TraceFab tool library. The selection list shows the tool identity and its relevant dimensions, while the Cut Settings line reports information such as the tip size and angle.

TraceFab validates whether the selected tool is suitable for isolation contouring. A V-bit or another tool with a valid cutting tip can be used. If no suitable tools exist in the library, add or correct the tool definition before continuing.

Tool geometry is important because TraceFab uses the effective cutting width to position the tool centre and space additional passes. Confirm that the installed physical tool matches the selected library entry.

8.1.3 Preview and Layer Display

Show Top Layer and **Show Bottom Layer** change only the layer displayed in the PCB preview. They are useful for inspecting dense or overlapping two-sided designs.

These preview buttons do not determine which toolpaths are calculated. The actual calculation scope is selected separately under **Apply Contouring To**.

8.2 Contour Pass Strategy

8.2.1 Master Pass

Every contour calculation contains one mandatory Master Pass. This first pass forms the primary isolation path around the copper geometry. Its value is fixed at 1 and cannot be changed because at least one pass is required to create an isolation contour.

8.2.2 Extra Passes

Extra Passes add successive paths beside the Master Pass. The total number of calculated passes is therefore one Master Pass plus the selected number of Extra Passes. Additional passes widen the cleared isolation channel and provide more assurance that neighbouring copper features are electrically separated.

Use only as many passes as the design and machining result require. Every extra pass increases the amount of removed copper and adds machining time.

8.2.3 Pass Overlap

Pass Overlap controls how much each new pass overlaps the cutting width of the preceding pass. TraceFab reduces the spacing between pass centre lines as the overlap percentage increases. This helps prevent thin ridges or copper burrs from remaining between adjacent passes.

A higher overlap creates a more continuous cleared area, but it also increases repeated cutting and machining time. Review the calculated result and choose a value appropriate for the selected tool and PCB material.

8.3 Layer and Toolpath Position

8.3.1 Applying Contours to Board Layers

- **All Layers** is the default and calculates the top and bottom contour toolpaths in one calculation.
- **Top Layer** recalculates only the top-side contour toolpath.
- **Bottom Layer** recalculates only the bottom-side contour toolpath.

Selecting one layer is useful when the two sides require different post-processing or when only one side of the PCB will be machined. Recalculating a selected layer replaces the previous contour result for that layer without intentionally clearing the other layer.

8.3.2 Cut Side

Cut Side determines where the tool centre is positioned relative to the outline of each copper feature:

- **Outside (default):** Places the tool centre outside the copper outline by the effective tool radius. The cutting edge follows the copper boundary while preserving the intended track and pad geometry. This is the normal choice for isolation routing.
- **Center:** Places the tool centre on the copper outline. Part of the cutting width therefore enters the copper feature and brings the contour closer to its track or pad geometry.
- **Inside:** Places the tool centre inside the copper outline. This removes still more copper from the feature and should be reserved for deliberate, specialised post-processing.

Center and Inside can narrow tracks, pads, or other copper areas. Use them only when their effect is understood and always inspect the calculated geometry before generating G-code.

8.3.3 Extra Offset

Extra Offset adds a manual distance to the calculated contour position. With the standard Outside setting, a positive offset moves the contour farther away from the imported copper outline. This leaves additional copper around the original feature and can compensate when tracks were designed too narrowly for the practical limits of the CNC setup.

For example, an operator may deliberately add a small offset when a track imported from the PCB design software is considered too fine for reliable milling. This adjustment changes the manufactured copper geometry and can reduce the clearance to neighbouring features. Apply it conservatively and verify the entire board, especially narrow gaps and fine-pitch pads.

8.4 Calculating and Reviewing Contours

8.4.1 Calculate

After reviewing the settings, click **Calculate**. TraceFab validates the selected tool and inputs, calculates the requested top and/or bottom toolpaths, and updates the PCB preview.

The resulting contour operations are stored in the current TraceFab job and become available in the Manufacture stage. Calculate does not yet generate the final G-code; G-code is created later after the complete manufacturing process has been assembled and validated.

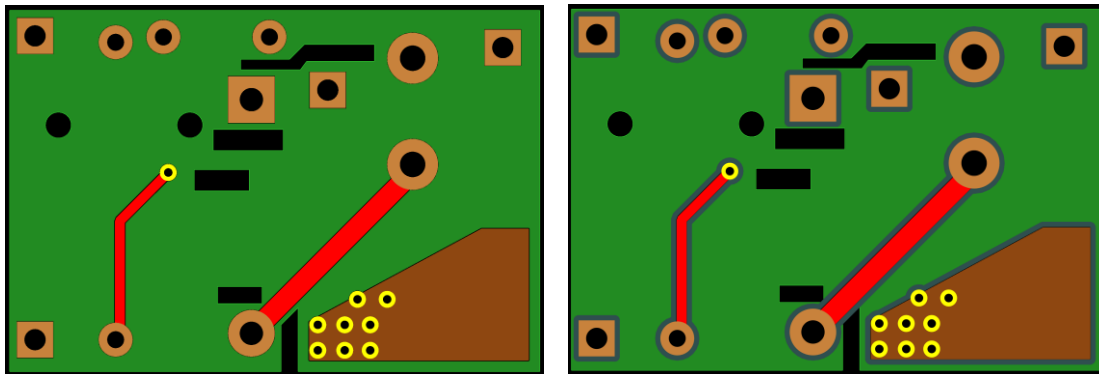


Figure 8.2 - PCB preview before contouring (left) and with calculated isolation contours (right).

8.4.2 Reviewing the Calculated Result

Before proceeding to Manufacture, inspect the result for the following:

- Contours are present around all copper features that require electrical isolation.
- Tracks, pads, vias, and copper areas have not been narrowed or removed unintentionally.
- Additional passes form a continuous cleared channel without narrow copper ridges between them.
- The selected top and/or bottom layers contain the expected contour result.
- Offsets and special Cut Side settings do not create conflicts in narrow clearances or fine-pitch areas.

8.4.3 Recalculating Contours

If the result is not satisfactory, change the tool, pass count, overlap, layer scope, Cut Side, or Extra Offset and click Calculate again. TraceFab replaces the contour result for the selected layer scope. Repeat the review until the displayed toolpaths match the intended machining result.

9 Excellon Drilling

The Drill Setup stage converts imported Excellon hole data into CNC drilling operations.

TraceFab collects the plated through-hole (PTH) and non-plated through-hole (NPTH) apertures found in the imported board, groups holes with the same nominal diameter, and lets the operator decide which physical drill tool will machine each group.

9.1 Drill Setup Workspace

9.1.1 Opening Drill Setup

Import the PCB and verify that its drill data is present in the viewport. Click **DrillSetup** on the main toolbar, or select **Manufacturing > Drill Setup**.

The Drill Setup Window lists the detected hole groups and loads the available drill tools from the TraceFab tool library.

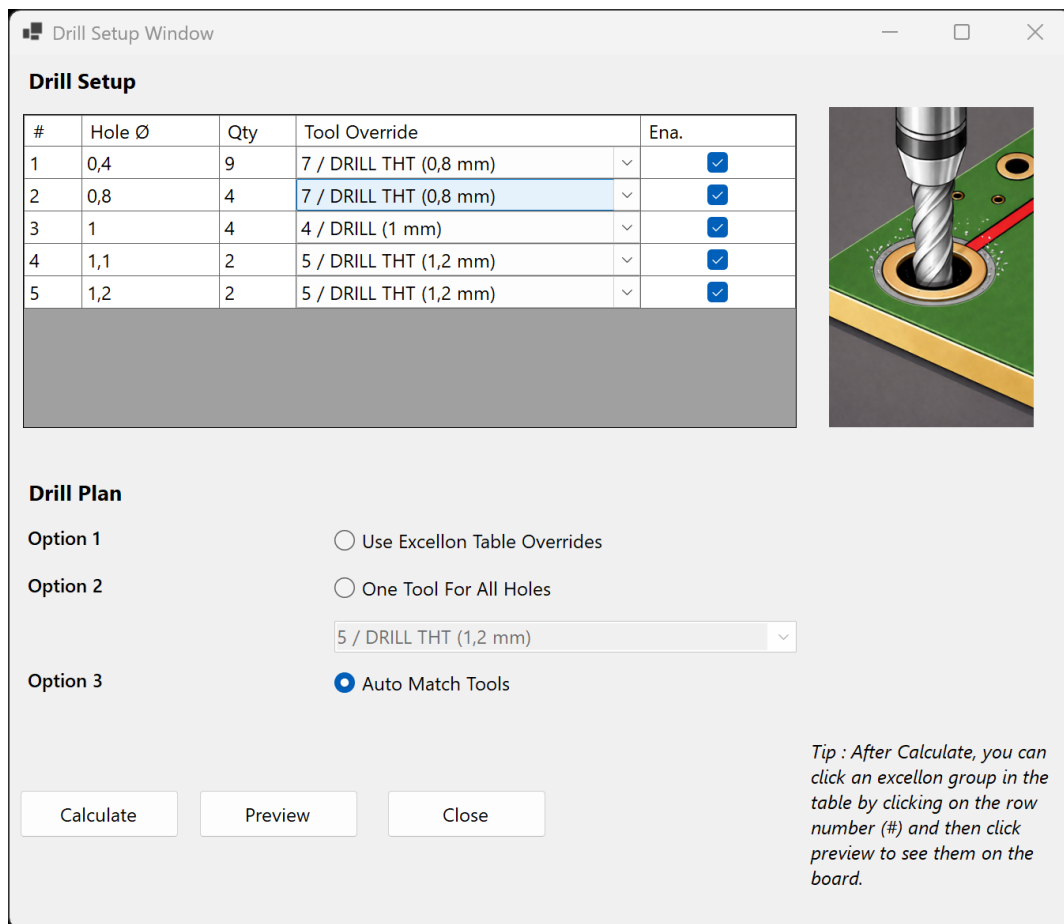


Figure 9.1 - Drill Setup Window with detected hole groups and the three drill-plan modes.

9.1.2 Detected Hole Groups

TraceFab combines the imported PTH and NPTH drill information and creates one row for each exact nominal hole diameter. The rows are sorted from the smallest diameter to the largest. Each row represents an aperture-size group rather than one individual hole.

The drill table contains the following columns:

- #: Sequential number used to select and identify the hole group.

- **Hole Diameter:** Nominal aperture or finished-hole diameter detected in the imported board data.
- **Qty:** Number of holes contained in that diameter group.
- **Tool Override:** Drill tool currently assigned or suggested for the group.
- **Ena.:** Enabled state. Clear this checkbox to exclude the entire diameter group from the calculated drill plan.

All groups are enabled by default. Always compare the diameters and quantities with the source PCB or Excellon data before assigning tools.

9.2 Drill Plan Modes

9.2.1 Use Excellon Table Overrides

Use Excellon Table Overrides gives the operator direct control over every enabled aperture group. Select the required drill from the Tool Override list in each row.

This mode is appropriate when particular aperture sizes require specific physical tools, when a suggested match should be changed, or when the operator wants to minimise or deliberately control tool changes. Every enabled row must have a valid tool assignment before the plan can be calculated.

9.2.2 One Tool For All Holes

One Tool For All Holes applies one selected drill tool to every enabled aperture group. Choose the tool from the selector displayed beneath this option.

This creates a simple drilling plan with a single physical drill and avoids tool changes. However, all enabled holes will be drilled using that tool diameter. Smaller nominal holes may therefore become larger than designed. Use this mode only when one common hole size is acceptable for the finished PCB.

9.2.3 Auto Match Tools

Auto Match Tools is the default mode. TraceFab assigns a drill-library tool to every enabled diameter group automatically.

The matching logic is applied in the following order:

12. Use a drill whose diameter matches the nominal hole within the current 0.02 mm matching tolerance.
13. If no such match exists, use the smallest available drill that is larger than the nominal hole.
14. If no larger drill exists, use the available drill with the smallest overall diameter difference as a fallback.

Automatic matching is a practical starting point, but it remains the operator's responsibility to verify every assignment. In particular, check fallback selections and confirm that the resulting hole diameters suit the components, fasteners, plating allowance, and physical CNC process.

9.3 Previewing Hole Groups

9.3.1 Previewing One Aperture Size

A single diameter group can be highlighted in the main viewport:

Click the required row under the **#** column to select the complete row. Click **Preview**. TraceFab highlights only the holes belonging to that nominal diameter group in the PCB viewport. The highlighted labels make it easier to compare their positions and quantity with the table.

Preview is a visual inspection aid. It does not create drill operations and does not replace the Calculate step.

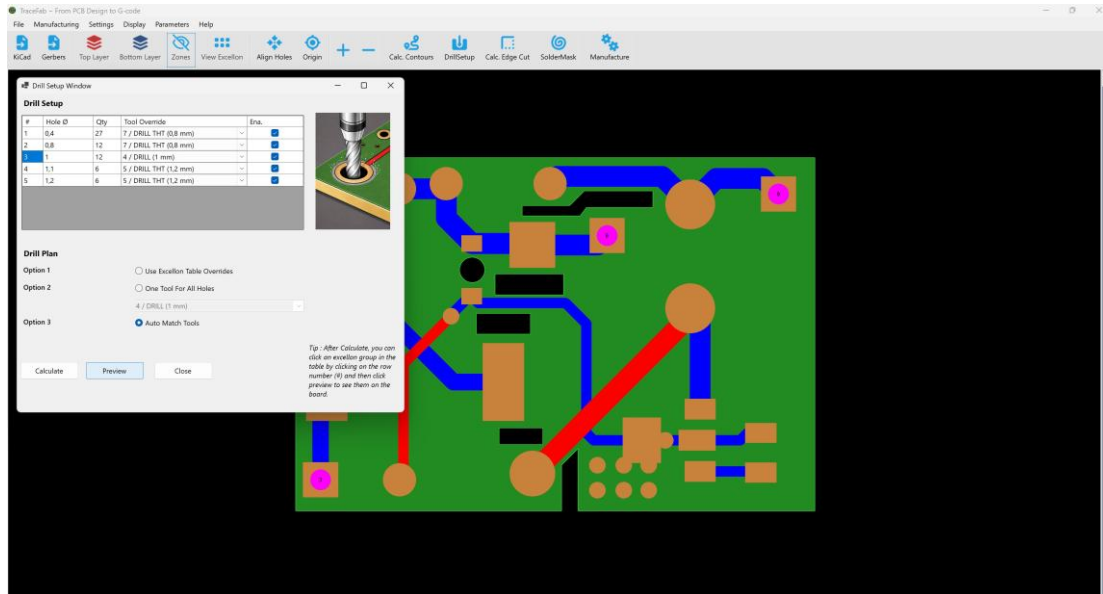


Figure 9.2 - Preview of one selected Excellon aperture-size group in the PCB viewport.

9.4 Calculating the Drill Operations

9.4.1 Calculate

After choosing a drill-plan mode and verifying all enabled rows, click **Calculate**. TraceFab validates the assignments and converts the enabled hole groups into drilling operations.

Groups assigned to the same tool are combined into one drill operation. A plan using several different tools therefore produces a separate operation for each unique drill tool. The previous calculated drill operations are replaced by the new plan.

The calculated operations become available in the Manufacture stage, where they can be added to the machining sequence and later converted to G-code. Calculate does not itself generate the final G-code.

9.4.2 Drilling Depth and Motion Parameters

The Drill Setup Window assigns tools and groups hole coordinates; it does not define every CNC motion parameter. Drilling depth, feed, plunge, retract, safe height, spindle behaviour, and related machining defaults are configured through the CAM parameters and are applied when the manufacturing program and G-code are generated.

9.5 Reviewing the Drill Plan

9.5.1 Pre-calculation Checks

Before clicking Calculate, verify that:

- The listed hole diameters and quantities agree with the imported PCB data.
- Every required PTH and NPTH diameter group is enabled.
- Every enabled row has the intended drill tool assigned.
- No selected drill is too large for nearby pads, annular rings, tracks, or mechanical clearances.
- Any group intentionally disabled can safely be omitted from the manufactured board.

9.5.2 Tool Changes and Operation Count

The number of aperture groups does not necessarily equal the number of drill operations. TraceFab groups enabled rows by assigned Tool ID. Five diameter groups using two physical tools, for example, become two drill operations. Their final order is determined later in the Manufacture sequence.

9.5.3 Recalculating the Drill Plan

If an assignment or enabled state must be changed, update the Drill Setup table and click Calculate again. The newly calculated plan replaces the previous drill operations. Reopen or refresh the Manufacture stage as required and verify the updated operations before generating G-code.

10 PCB Edge Cutting

Edge cutting creates the milling operation that separates the finished PCB from the surrounding board material. TraceFab offsets the imported board outline for the selected tool, optionally divides the cut into several depth passes, and can leave bridge gaps that hold the PCB in place until machining is complete.

10.1 Edge Cutting Workspace

10.1.1 Opening the Edge Cut Milling Window

Confirm that the imported PCB contains a valid, closed board outline. Click **Calc. Edge Cut** on the main toolbar, or select **Manufacturing > Calc. Edge Cut**. The Edge Cut Milling Window opens and loads the edge-cut defaults from the CAM parameters.

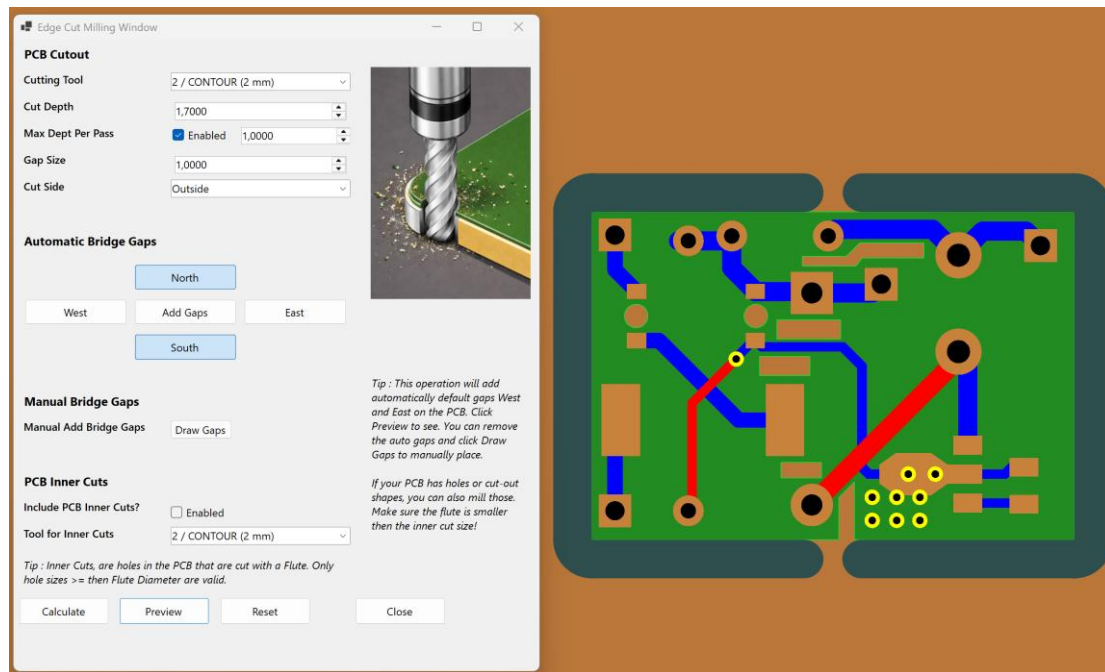


Figure 10.1 - Edge Cut Milling Window with the calculated outer cut and bridge gaps shown in the viewport.

10.1.2 Selecting the Cutting Tool

The Cutting Tool list is limited to cylindrical CNC milling tools defined in the TraceFab tool library. Drill tools and conical V-bits are not offered for this operation because the edge offset and inner-cut checks require a known cylindrical flute diameter.

Choose a tool that can cut the PCB substrate cleanly and that is compatible with the spindle, material, feed rate, and available clearance. The selected tool diameter directly determines the edge-path offset.

10.2 Cutting Parameters

10.2.1 Cut Depth

The default edge-cut depth equals the PCB thickness plus an additional 0.1 mm. This small overcut enters the sacrificial waste board and helps ensure that the PCB is cut through completely. For example, a 1.6 mm PCB produces a default cut depth of 1.7 mm.

The PCB thickness is defined under **File > Board Dimensions**. The additional edge-cut depth can be changed under **Parameters > CAM Parameters**. In the current release, use the CAM Parameters value as the authoritative override rather than relying only on editing the displayed Cut Depth field in this window.

Never use an extra depth that can damage the CNC bed. A suitable sacrificial board must be installed and the programmed Z zero must be verified before machining.

10.2.2 Maximum Depth per Pass

Enable **Max Depth Per Pass** to divide the total cut depth into several step-down passes. The entered value is the maximum depth removed during one pass; TraceFab adds passes until the full cut depth is reached.

Clear the Enabled checkbox to cut the complete depth in one pass. A single full-depth pass can place substantially more load on the tool, spindle, PCB, and workholding. Use it only when the machine, cutter, feed rate, and material are suitable.

10.2.3 Cut Side

Cut Side positions the tool centre relative to the imported outer board outline:

- **Outside (default):** Offsets the tool centre outward by the tool radius so the cutting edge follows the intended external board dimension.
- **Center:** Places the tool centre directly on the imported outline and removes material on both sides of it.
- **Inside:** Offsets the tool centre into the board outline and reduces the finished outside dimensions.

Outside is the normal choice for manufacturing the board perimeter. Use Center or Inside only for deliberate geometry correction or specialised post-processing, and verify the resulting dimensions carefully.

10.3 Bridge Gaps

10.3.1 Purpose and Gap Size

Bridge gaps, also called holding tabs, are short sections of PCB material that remain uncut in the outer perimeter. They keep the PCB connected to the surrounding panel or waste material so it cannot move when the final part of the outline is milled. After machining, the bridges can be cut or snapped and the remaining edges cleaned manually.

Gap Size defines the intended length of PCB material to preserve at each bridge. Internally, TraceFab also accounts for the selected tool diameter when it creates the skipped section of toolpath. This compensates for the cutter entering and leaving the bridge area so the remaining material corresponds to the requested gap size.

Use gaps large enough to hold the board safely, but not so large that removal becomes difficult or risks damaging the PCB. Bridge gaps apply to the outer board outline, not to inner cut-outs.

10.3.2 Automatic Bridge Gaps

The North, East, South, and West selectors determine where automatic bridge gaps are placed. Their initial states and the default gap size are loaded from the Edge Cut settings in the CAM Parameters form. The standard configuration can, for example, provide one bridge on the east and west sides of the PCB.

When the form opens, these default selections are prepared but the edge geometry must still be built. Click Preview, or use Add Gaps, to calculate the outer path and display the selected automatic gaps in the viewport. Changing a direction or gap size requires another Preview before the rebuilt result is visible.

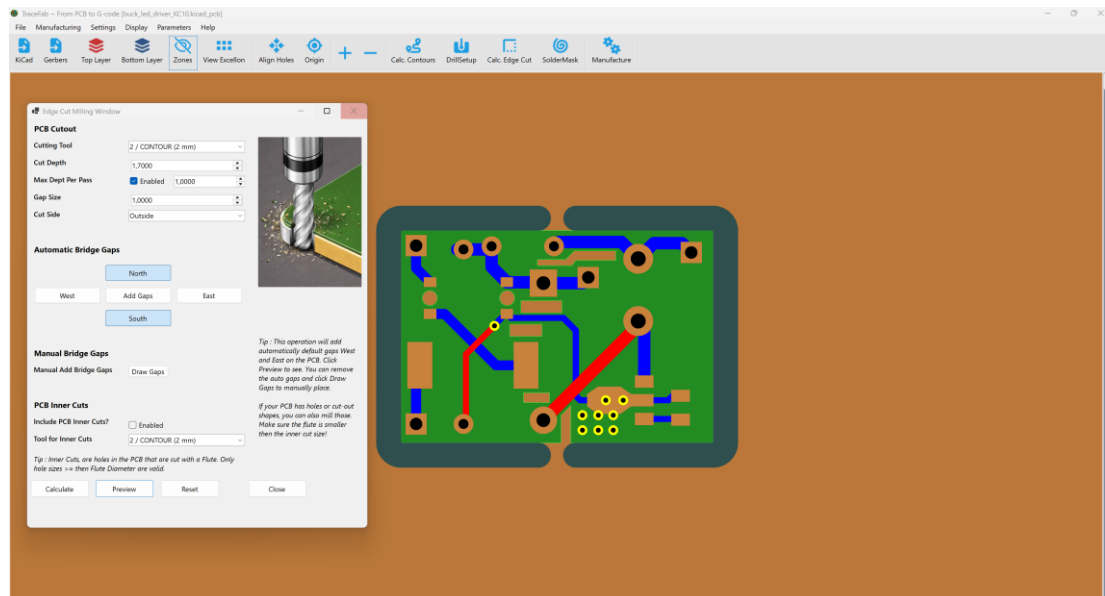


Figure 10.2 - Automatically generated bridge gaps displayed in the edge-cut preview.

10.3.3 Manual Bridge Gaps

Use manual placement when a bridge must avoid a connector, narrow feature, internal cut-out, or another unsuitable location. To create a layout using only manual gaps:

1. **Click Reset** to leave the current edge-cut preview.
2. Clear the North, East, South, and West automatic-gap selectors if no automatic gaps should be regenerated.
3. **Click Preview** to rebuild and display the outer edge path from the current settings.
4. **Activate Draw Gaps** to enable manual gap placement in the viewport.
5. Left-click near the outer edge-cut path where the bridge should be placed. TraceFab projects the click onto the closest point of the outer path and adds a bridge using the current Gap Size.
6. Repeat the left-click action for any additional bridges, then disable Draw Gaps when placement is complete.

Reset returns the viewport from edge-cut preview mode. When Preview is run again, TraceFab rebuilds the working edge path from the current automatic-gap selections; this provides the reliable way to replace the previous visible gap arrangement. Always inspect the rebuilt result before Calculate.

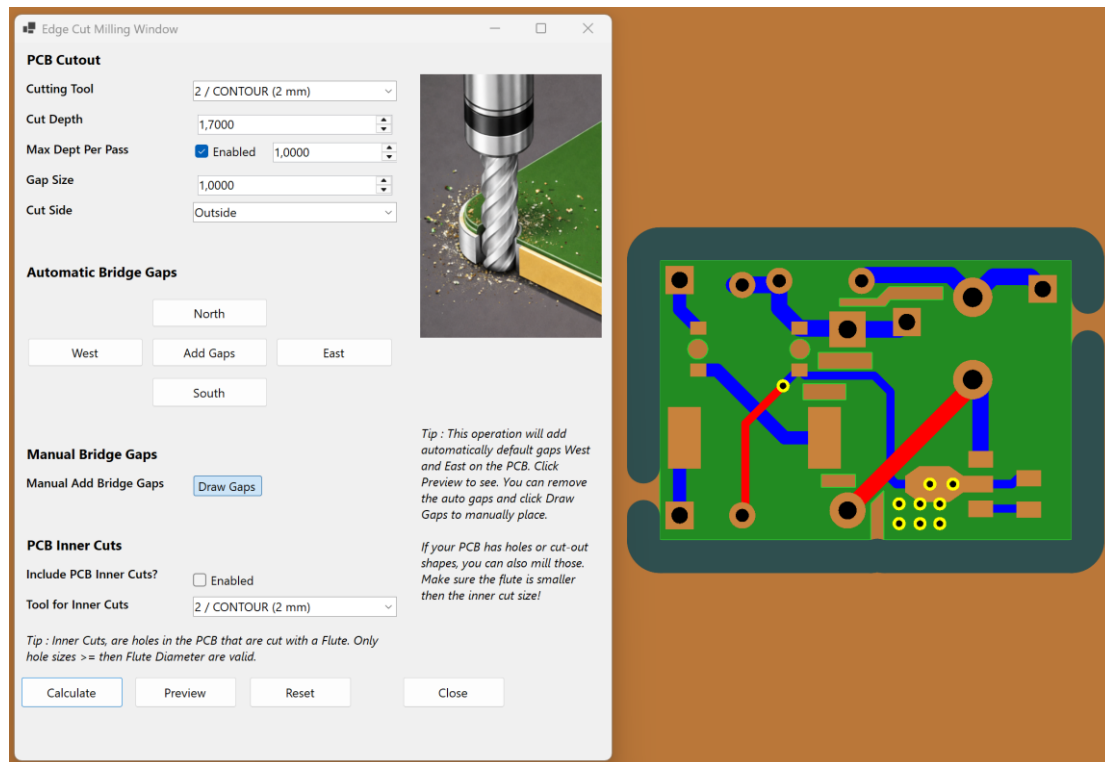


Figure 10.3 - Draw Gaps mode for placing bridge gaps manually on the outer edge-cut path.

10.4 PCB Inner Cuts

10.4.1 Including Inner Cut-outs

Enable **Include PCB Inner Cuts** when the board outline contains closed internal openings that must also be milled. Select a cylindrical tool under Tool for Inner Cuts. TraceFab creates the inner-cut operation separately from the outer perimeter operation.

Inner cuts always use an inside offset so the cutting edge follows the inside boundary of the opening. The tool must physically fit through the intended shape: both the available width and height must be at least as large as the selected flute diameter.

10.4.2 Inner-cut Tool Validation

If the tool is too large, the inward offset can collapse the opening or produce no valid path. TraceFab then rejects the calculation and displays an Invalid Inner Cut warning. Select a smaller cylindrical tool, enlarge the source cut-out, or exclude the inner cuts before trying again.

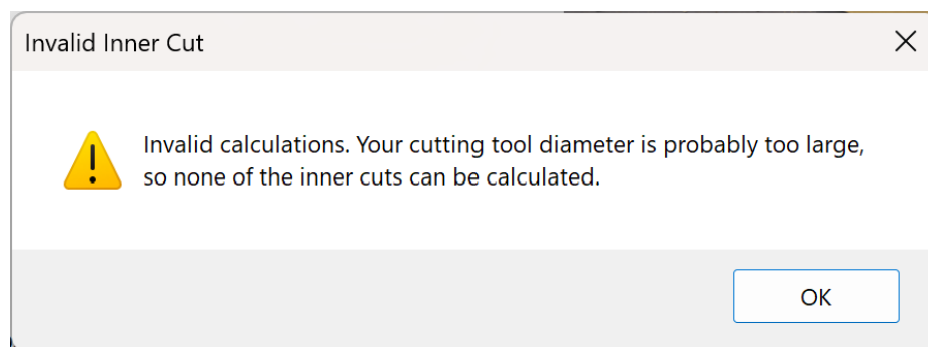


Figure 10.4 - Warning shown when the selected tool is too large to calculate a valid PCB inner cut.

10.5 Calculating and Reviewing the Edge Cut

10.5.1 Preview and Verification

Use Preview before accepting the operation. Verify that:

- The outer path forms one continuous closed loop around the intended board boundary.
- The selected Cut Side produces the required finished outside dimensions.
- Bridge gaps are present, sufficiently strong, and clear of sensitive PCB features.
- All required inner cut-outs are visible and can accommodate their selected tool.
- The selected tools, total depth, and maximum depth per pass are suitable for the physical CNC setup.

10.5.2 Calculate

When the preview and settings are satisfactory, click **Calculate**. TraceFab rebuilds the geometry from the current settings, stores the outer edge cut, and stores an additional inner-cut operation when inner cuts are enabled and valid.

These operations become available in the Manufacture form, where they can be placed in the machining sequence and later converted to G-code. Calculate does not itself generate the final G-code.

10.5.3 Recalculating an Edge Cut

If the result must be changed, adjust the tool, cut side, pass settings, gaps, or inner-cut selection and run Preview again. After verification, click Calculate to replace the stored edge-cut operations with the new configuration.

11 Double-sided Boards and Alignment

Double-sided PCB machining requires the second side to remain aligned with the first side after the board is flipped. TraceFab can generate two or four alignment-pin holes that establish a repeatable mechanical reference between the PCB and the sacrificial waste board. When the same pin pattern, fixture, and work origin are preserved, the board can be flipped without mechanically shifting the machining reference or re-homing the job.

11.1 Purpose of Alignment Pins

11.1.1 How the Alignment Method Works

Alignment holes are machined at symmetrical positions and extend far enough into the waste board to accept locating pins. Depending on the selected positions and PCB design, the holes may pass through dedicated registration areas in the PCB or may be placed outside the board outline directly in the surrounding waste-board area.

After the first side has been machined, the PCB is lifted, flipped around the intended axis, and placed back over the same pins. The pins reproduce the original mechanical reference so the bottom-side geometry remains aligned with the top-side geometry.

Important: Alignment pins do not compensate for loose holes, bent pins, a shifted waste board, an incorrect flip direction, or a changed machine work origin. The complete setup must remain fixed until both sides have been machined.

11.2 Opening the Alignment Pins Window

11.2.1 Starting the Operation

Import and verify the PCB first. Open the Alignment Pins window by clicking Align Holes on the main toolbar or by selecting **Manufacturing > Alignment Pins**. The window loads the current alignment-pin defaults from **Parameters > CAM Parameters > Centering**.

11.2.2 Cutting Tool and Pin Size

The Cutting Tool list contains drill tools and cylindrical milling tools from the Tool Library. The available Pin Size behaviour depends on the selected tool:

- Drill tool: The pin-hole diameter is fixed to the drill diameter. The tool plunges at each calculated pin position.
- Cylindrical milling tool: Helix Cut is enabled automatically. TraceFab can mill a circular hole larger than the flute itself; the current release allows a pin size from the flute diameter up to 1.9 times that diameter.

A helical cut is useful when an accurately sized hole or a deliberate clearance or interference fit is required. Tool runout, pin tolerance, material behaviour, and machine accuracy all affect the real fit, so test the chosen diameter in scrap material before machining the final board.

11.2.3 Pin Depth

Pin Depth is configured under **Parameters > CAM Parameters > Centering**. TraceFab uses this value directly as the total negative Z machining depth from the active Z zero. It does not subtract the PCB thickness and does not derive the depth automatically from the physical length of the alignment pin.

When Z0 is set on the top surface of the PCB, a Pin Depth of 10.0 mm on a 1.6 mm PCB produces 10.0 mm of total depth: 1.6 mm through the PCB and approximately 8.4 mm into the waste board. If Z0 is established elsewhere, this relationship changes accordingly.

Choose a depth that provides stable pin engagement without approaching the machine bed. Confirm the waste-board thickness, available flute length, pin insertion depth, safe Z height, and actual Z-zero reference before generating or running the operation.

11.3 Placing Alignment Pins

11.3.1 Two-Pin Placement

Click 2 pins to start placement mode, then left-click the desired anchor position in the viewport. TraceFab places the first pin at the selected point and creates the second pin at the vertically mirrored position across the horizontal centreline of the board. This produces one matched pin above and one below the board centre.

Two pins normally provide the minimum positive location needed for a repeatable flip while avoiding unnecessary over-constraint. Position them as far apart as practical and clear of copper, clamps, edge-cut toolpaths, and other fixtures.

11.3.2 Four-Pin Placement

Click 4 pins and select one anchor position in the viewport. TraceFab mirrors that point across both board-centre axes to create four symmetrical pin positions. Four pins provide additional

support, but all four holes and pins must match accurately; tolerance errors can make the board difficult to seat or can introduce stress.

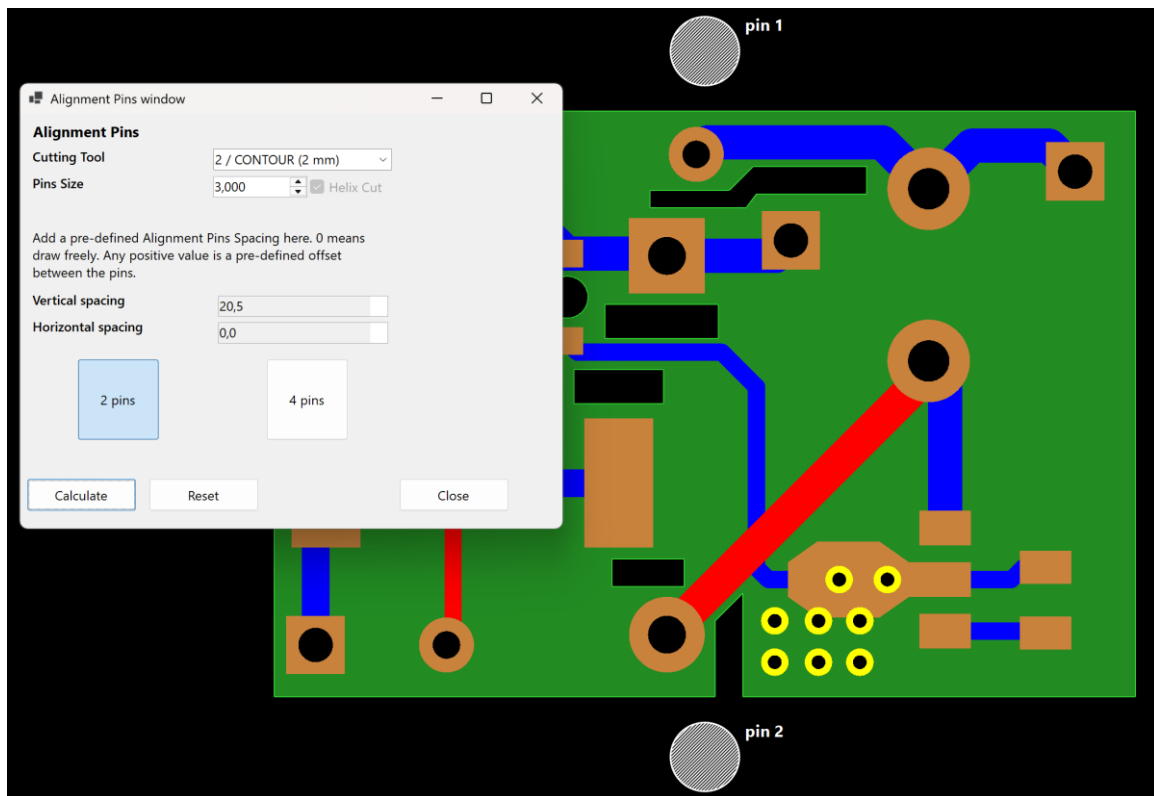


Figure 11.1 - Alignment Pins window with two vertically mirrored pin positions shown in the viewport.

11.3.3 Spacing and Reset

The Vertical spacing and Horizontal spacing fields show the separation produced by the selected anchor and the board-centre mirroring. In the current release, the viewport click determines the pin positions; use the displayed spacing and the viewport preview to verify the result.

Click Reset to clear the working pin positions, remove the stored alignment-pin operation, leave placement mode, and return the viewport to its normal state. Select 2 pins or 4 pins again to create a new arrangement.

11.4 Calculating the Alignment Operation

11.4.1 Calculate

After verifying the tool, pin size, depth, pin count, and visible positions, click Calculate. TraceFab stores the alignment-pin geometry and adds one Alignment Pins operation to the manufacturing plan. Calculate prepares the operation for the Manufacture form; it does not generate the final G-code by itself.

When G-code is generated, a matching drill tool produces direct plunge moves. A cylindrical tool that is smaller than the requested pin hole produces a helical milling path. Review the selected strategy, depth, feed rates, tool diameter, and machine compatibility in the generated G-code before running it.

11.5 Using an Alignment Pin as the World Origin

11.5.1 Selecting the Origin in Manufacture

After the alignment pins have been calculated, the available pin positions are added to the Set World origin list in the Manufacture window. A two-pin arrangement provides Alignment Pin 1 and Alignment Pin 2; a four-pin arrangement also provides Alignment Pin 3 and Alignment Pin 4.

Select the pin that will serve as the physical reference for the complete double-sided job. TraceFab stores the centre of that pin as the CAM world origin and expresses all generated XY coordinates relative to it.

11.5.2 Pin Numbering

The pin numbers follow the order in which TraceFab builds the symmetrical pattern:

- Pin 1: The original anchor position selected in the viewport.
- Pin 2: The position vertically mirrored from Pin 1 across the horizontal centreline of the PCB.
- Pin 3: For a four-pin pattern, the position horizontally mirrored from Pin 1 across the vertical centreline of the PCB.
- Pin 4: For a four-pin pattern, the position mirrored from Pin 1 across both centre axes.

For example, when Pin 2 lies on the horizontal centre of the PCB and below its vertical centre, selecting Alignment Pin 2 makes the centre of that lower pin X0 Y0 in the generated G-code.

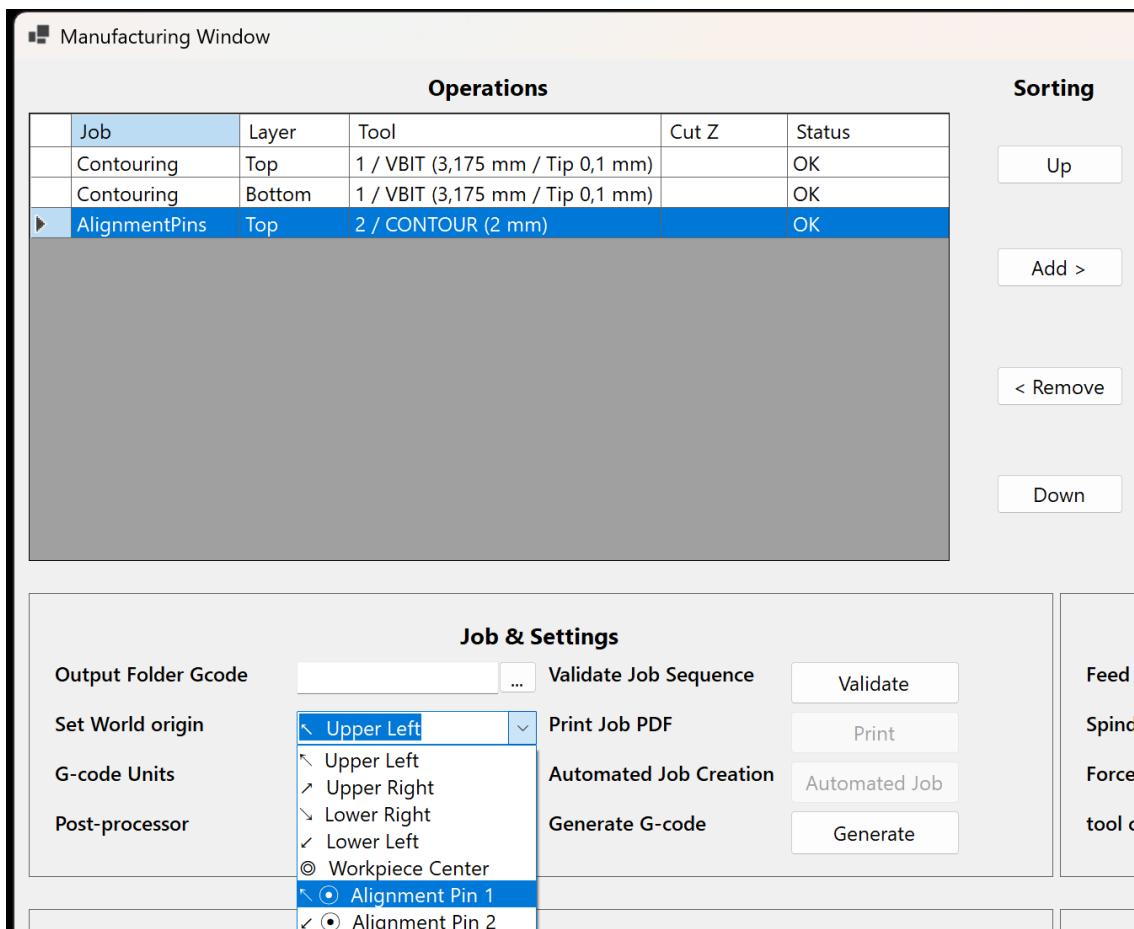


Figure 11.2 - Selecting an alignment pin as the world origin in the Manufacture window.

11.5.3 Using the Same Physical Zero on the CNC

A pin origin is especially useful after flipping the PCB because the reference is a real, repeatable feature in the fixture rather than a board corner that may be difficult to locate again. Both PCB sides can use the same physical pin centre as their XY reference while the waste board and pins remain fixed.

Important: Selecting an alignment pin in TraceFab changes the coordinate origin of the generated G-code; it does not automatically set the work offset in the CNC controller. Before running the job, set the controller's X0 Y0 to the centre of the same numbered physical pin, using the machine's normal probing or alignment procedure. Machine homing and the job's work zero are separate references.

Do not change to another pin origin between the top-side and bottom-side jobs unless the CNC work offset is changed to the corresponding physical pin as well. Always verify the selected pin number and perform a safe dry run or raised-Z check after the board has been flipped.

11.6 Recommended Board-Flipping Workflow

11.6.1 Machining Sequence

7. Secure a flat waste board to the CNC machine and fasten the PCB without obstructing the planned pin positions or machining paths.
8. Establish the intended XY work origin and set Z0 at the reference surface used when the Pin Depth was chosen.
9. Open Alignment Pins, select a suitable tool and pin size, choose 2 pins or 4 pins, and click the anchor position in the viewport.
10. Verify that every pin location is accessible, symmetrical, and clear of copper, clamps, edge cuts, and the machine limits. Then click Calculate.
11. In Manufacture, place the Alignment Pins operation before any step that requires the PCB to be removed or flipped. If a pin-based reference will be used, select the intended Alignment Pin under Set World origin and establish the centre of that same physical pin as X0 Y0 on the CNC controller. Generate the G-code and review it manually.
12. Machine the alignment holes before loosening the PCB. Insert clean, straight pins that seat securely in the waste board without excessive force.
13. Machine the first side. Do not move the waste board, pins, fixture, or work-coordinate origin afterward.
14. Lift the PCB, flip it around the intended axis, place it over the same pins, and seat it flat against the waste board. Secure it again without shifting the reference.
15. Load the corresponding second-side operations, confirm the expected mirrored orientation, and perform a dry run or raised-Z verification before cutting.

11.6.2 Verification and Safety

- The pin-hole diameter matches the real pins and provides a repeatable fit without damaging the PCB.
- The programmed depth reaches the required waste-board engagement but cannot contact the machine bed.
- The selected flip axis and the bottom-layer mirroring convention match the physical way the PCB will be turned.
- The PCB sits completely flat after flipping; debris under the board or around a pin can introduce a Z or XY error.

- The same work origin is still active and no re-homing, probing, fixture movement, or coordinate reset has altered the reference.
- A manual G-code review and a safe first run confirm that all rapid moves, plunge depths, and toolpaths remain within the machine and fixture limits.

12 CNC Origin and Coordinate Setup

TraceFab separates imported file coordinates, its internal CAM coordinates, and the units written to the final CNC program. Understanding this conversion chain helps prevent mirrored boards, incorrect scale, and a work origin that does not match the physical CNC setup.

12.1 Coordinate Systems

12.1.1 TraceFab Internal Coordinates

TraceFab performs its geometry and CAM calculations in a metric Cartesian coordinate system. Internal distances are represented in millimetres, the positive X axis points to the right, and the positive Y axis points upward. Geometry is kept in this world-coordinate system; only the screen renderer converts Y-up world coordinates to the Y-down coordinate direction used by the Windows display.

Because the internal model is normalised, contouring, drilling, edge cutting, alignment pins, and other CAM operations can use one consistent geometry system regardless of the source-file units or source-file axis convention.

12.1.2 KiCad Coordinate Normalisation

KiCad board files use a Y-axis convention that is inverted relative to TraceFab's Cartesian CAM workspace. During import, TraceFab retains the X coordinate, reverses the sign of Y coordinates, adjusts the corresponding rotations, and recalculates the board extents. The visible PCB orientation and physical dimensions are preserved while the imported geometry is converted to Cartesian millimetres.

12.1.3 Metric and Imperial Imports

Gerber and Excellon files may declare either metric or imperial coordinates. TraceFab reads the unit declaration together with the selected coordinate format. Imperial values are multiplied by 25.4 during parsing so the resulting board model is stored in millimetres. Metric values use a scale factor of 1.0.

For Gerber and Excellon imports, the Units, zero-suppression, and coordinate-format settings must still match the exported files. Normalisation can convert a correctly interpreted imperial file to millimetres, but it cannot repair an incorrect decimal position or an incorrectly selected leading- or trailing-zero convention.

12.2 G-code Output Units

12.2.1 Millimetres and Inches

The G-code Units selector in the Manufacture window controls the units written to the CNC program. This output choice is independent of the units used by the imported source file:

- Millimetres: TraceFab writes G21 and outputs coordinates and feed values in millimetres and millimetres per minute.
- Inches: TraceFab writes G20 and converts coordinates and feed values from millimetres by dividing them by 25.4.

Changing the output units does not resize or recalculate the PCB geometry. It changes only the numeric representation used in the generated G-code. A 25.4 mm move therefore becomes a 1.0000 inch move when inch output is selected.

12.2.2 Default CNC Units

The default length unit can be selected on the first, General tab under **Parameters > CAM Parameters**. This default preselects the G-code Units value when the Manufacture window opens. The operator can still change the output unit for the current manufacturing job before generating the program.

The generated header also selects absolute positioning (G90), feed per minute (G94), the XY working plane (G17), and work coordinate system G54. Verify that the selected post-processor and CNC controller support these commands as expected.

12.3 Setting the World Origin

12.3.1 Available Origin Positions

Use Set World origin in the Manufacture window to choose the XY reference used by the generated coordinates. TraceFab currently provides:

- Upper Left, Upper Right, Lower Right, or Lower Left: Uses the corresponding corner of the imported board extents.
- Workpiece Center: Uses the centre calculated between the minimum and maximum X and Y board extents.
- Alignment Pin 1 to Alignment Pin 4: Uses the stored centre of the selected alignment pin. Only pins that were calculated for the current board are listed.

Alignment-pin origins are particularly useful for double-sided machining because the physical reference remains available after the PCB is flipped. Pin numbering and the selection in Manufacture are described in Chapter 11 and shown in Figure 11.2.

12.3.2 How the Origin Offset Is Applied

TraceFab subtracts the selected world-origin X and Y position from every CAM output point. The selected corner, workpiece centre, or pin centre therefore becomes X0 Y0 in the generated G-code. Other coordinates are expressed relative to that point.

A corner origin usually keeps most board coordinates positive. A workpiece-centre origin normally produces both positive and negative X and Y coordinates. Before choosing an origin, confirm that the CNC controller, work offset, probing method, and available travel are suitable for the resulting coordinate range.

12.3.3 Machine Home, G54, and Work Zero

The TraceFab world origin is a CAM reference, not the CNC machine's permanent home position. The program selects G54, but the operator must establish the matching physical G54 work zero on the CNC controller. If Workpiece Center is selected, for example, the controller's

X0 Y0 must be set at that same physical centre. If Alignment Pin 2 is selected, X0 Y0 must be set at the centre of physical Pin 2.

Important: Set World origin changes only the XY coordinate reference. It does not establish the Z-zero surface, Safe Z height, tool length, or machine home. Set and verify Z0 separately for the actual PCB and waste-board setup.

12.4 Mirroring and Board Orientation

12.4.1 Mirror X

Each operation added to the Manufacture sequence has a Mirror X option. When enabled, TraceFab first expresses the point relative to the selected world origin and then reverses the sign of X. The selected world origin therefore acts as the vertical mirror axis.

In the current release, assigning an operation to the Bottom layer does not automatically mirror its G-code coordinates. Enable Mirror X only when it matches the physical way the PCB will be flipped and the origin used on the CNC. Always confirm the resulting orientation in the generated program before cutting.

12.5 Verifying the Coordinate Setup

12.5.1 Import Scale and Board Dimensions

After import, compare the displayed board width, height, PCB thickness, and at least one known hole spacing or feature length with the source design. A factor-of-25.4 error normally indicates a metric/imperial mismatch. A much larger or smaller error in a Gerber job may indicate an incorrect coordinate format or zero-suppression setting.

12.5.2 Pre-run Coordinate Checklist

- The imported board dimensions and a known feature distance match the source design.
- The selected G-code units match the CNC controller and post-processor expectations.
- The physical G54 X0 Y0 on the CNC matches the origin selected in TraceFab.
- The required Mirror X state matches the real board-flipping direction.
- All positive and negative XY coordinates remain inside the machine's usable travel and clear of clamps.
- Z0, Safe Z, cutting depths, and tool clearances have been set independently and verified.
- The generated header, coordinate ranges, and first movements have been reviewed manually before the first run.

13 Creating the CNC Program

The Manufacturing Window is TraceFab's NC program composer. Earlier CAM windows calculate individual machining processes; this window collects those calculated Operations and lets the operator arrange selected operations into the exact Sequence that will be written to one G-code file.

13.1 Manufacturing Window Overview

13.1.1 Opening the Manufacturing Window

Complete the required CAM calculations before opening Manufacturing. Each time Calculate is accepted in Contouring, Drill Setup, Edge Cut, Alignment Pins, or Solder Mask Removal, TraceFab stores the resulting operation data. When the Manufacturing Window opens, it reads the currently calculated data and builds the Operations list.

Only processes that were calculated for the current PCB can appear. Recalculate a process when its tool or parameters change, then reopen or refresh Manufacturing as required so the available operation uses the latest result.

13.1.2 Operations List

Operations is the optional source list of all currently available machining steps. A row can represent a top or bottom contour, one drill-tool group, an outer or inner edge cut, alignment pins, or top or bottom solder-mask removal. The row also identifies the associated layer and tool.

An operation in this list is not automatically included in an NC program. It becomes part of the generated file only after the operator adds it to the Sequence. The current Status values are provisional and must not be treated as a complete safety validation.

13.1.3 Sequence List

Sequence is the ordered list for one NC program. TraceFab writes the selected operations in the displayed order, from the first row to the last row. Each Sequence row shows its sequence number, job type, layer, tool, and Mirror X state.

Important: The Sequence controls real machining order. Tool changes, board flips, workholding, alignment-hole creation, and the final edge cut must be planned deliberately before G-code is generated.

The Manufacturing Window interface is divided into several sections:

- Operations:** A table listing available machining steps.
- Sorting:** Controls for managing the Operations list (Up, Add >, < Remove, Down).
- Sequence:** A table for the ordered list of operations for the NC program.
- Job & Settings:** Configuration options for output, world origin, G-code units, and post-processor.
- CNC Overrides:** Settings for feed rate, spindle RPM, force safe Z up, and tool change pause.
- Job Summary:** Key parameters like lowest/highest Z, min/max feed, and spindle range.
- Validation & Status:** Checkmarks and status indicators for various operations.

Job	Layer	Tool	Cut Z	Status
Contouring	Top	1 / VBIT (3,175 mm / Tip 0,1 mm)		OK
Contouring	Bottom	1 / VBIT (3,175 mm / Tip 0,1 mm)		OK
Drilling	Top	7 / DRILL THT (0,8 mm)		OK
Drilling	Top	4 / DRILL (1 mm)		OK
Drilling	Top	5 / DRILL THT (1,2 mm)		OK
EdgeCut	Top	2 / CONTOUR (2 mm)		OK
SolderMaskRe...	Top	3 / SOLDERMASK (3,175 mm / ...)		OK
SolderMaskRe...	Bottom	3 / SOLDERMASK (3,175 mm / ...)		OK

Seq.	Job	Layer	Tool	Cut Z	Mirror X
1	Contouring	Top	1 / VBIT (3,175 mm / Tip 0,1 mm)		☒
2	Drilling	Top	7 / DRILL THT (0,8 mm)		☐
3	Drilling	Top	4 / DRILL (1 mm)		☐
4	Drilling	Top	5 / DRILL THT (1,2 mm)		☐
5	EdgeCut	Top	2 / CONTOUR (2 mm)		☐

Job & Settings

Output Folder Gcode: Validate Job Sequence: Set World origin: Print Job PDF: G-code Units: Automated Job Creation: Post-processor: Generate G-code:

CNC Overrides

Feed Rate Multiplier (%): placeholder: Spindle RPM override: placeholder: Force Safe Z Up: placeholder: tool change Pause: placeholder:

Job Summary

Lowest Z (mm): Spindle Range (RPM) : Highest Z (mm): XY bounds (mm): Min feed (mm/min): Board Thickness (mm): Max feed (mm/min): Job Sequences:

Validation & Status

Top Contours: ☒ Tool Changes: ☒ Bottom Contours: ☒ Board Flip Required: ☒ Drill Code: ☒ Edge Cut: ☒

Figure 13.1 - Manufacturing Window with available Operations, an ordered Sequence, and the job-generation controls.

13.2 Building an NC Sequence

13.2.1 Adding Operations

Select the required row in Operations and click **Add >**. TraceFab appends it to the bottom of the Sequence and assigns the next sequence number. Repeat this action until the Sequence contains all steps required for that specific NC program.

The same source Operation can currently be added more than once. This can be intentional, but an accidental duplicate will also be generated and executed twice. Review the complete Sequence for duplicate rows before export.

13.2.2 Changing the Order

Select a Sequence row and use **Up** or **Down** to move it one position. TraceFab rennumbers the list after each change. Typical planning considerations include:

- Create alignment holes before the PCB is loosened or flipped.
- Group compatible operations where practical while accounting for every required tool change and Z-zero procedure.
- Keep drilling, contouring, or solder-mask steps that rely on rigid workholding before the board perimeter is released.
- Place the outer Edge Cut near the end of the final program unless the physical process requires a different verified order.

13.2.3 Removing Operations from the Sequence

Select a Sequence row and click **< Remove** to exclude it from the current NC program. Remove affects only the Sequence; it does not delete the calculated source Operation. The same operation remains available on the left and can be added to another program.

13.3 Creating Multiple NC Programs

13.3.1 Splitting a PCB into Separate Jobs

A PCB is often manufactured in several CNC stages. Build and export one Sequence at a time, then remove its rows and assemble the next Sequence from the remaining Operations. For example:

1. Build a top-side Sequence containing Alignment Pins, Top Contouring, the required Drilling operations, and Top Solder Mask Removal.
2. Choose the correct origin, units, and **Mirror X** state, validate the Sequence, and export it as a descriptive file such as Top.nc.
3. Remove the exported rows from the Sequence. The calculated Operations remain available.
4. Build a second Sequence containing Bottom Contouring, Bottom Solder Mask Removal, and the final Edge Cut where appropriate.
5. Verify the physical board-flip procedure and origin again, then export the second program as Bottom.nc or another unambiguous name.

This method lets the operator create as many independent .nc programs as the physical process requires. In the current release, these Sequences are assembled manually and are not yet reusable automated job recipes.

13.4 Job and Output Settings

13.4.1 Output Folder and File Name

Use the browse button next to Output Folder Gcode to select the intended export folder for the current job. When Generate is clicked, TraceFab also opens a save dialog for the .nc file. Choose a clear file name that identifies the board, side, and process stage, and confirm the final path before saving.

Keep separate programs in the same controlled job folder and avoid overwriting a previously verified file unless that replacement is intentional. Files such as BoardName_Top.nc and BoardName_Bottom.nc are less ambiguous than the default job.nc.

13.4.2 World Origin and G-code Units

Select Set World origin before validation and generation. Available choices include board corners, Workpiece Center, and calculated Alignment Pins. The physical CNC work zero must match the selected TraceFab origin. See Chapters 11 and 12 for pin-based origins and coordinate setup.

G-code Units is normally fixed by the CNC controller and workshop convention. Select Millimeters or Inches consistently; TraceFab writes G21 or G20 and converts the coordinate and feed values accordingly. Recheck this setting whenever a job is opened on another machine.

13.4.3 Mirror X

Mirror X is selected per Sequence row. Use it only when the operation must be mirrored around the chosen world origin to match the physical board flip. Bottom-layer assignment alone does not automatically mirror the operation. Review every row because an incorrect checkbox changes the generated geometry.

13.5 Validating the Sequence

13.5.1 Job Summary

Click Validate after the Sequence, origin, units, and Mirror X selections are ready. The current release updates the Job Summary with:

- Lowest cutting Z and highest Safe Z.
- Minimum and maximum feed values.
- Minimum and maximum spindle RPM.
- XY board bounds, board thickness, and total number of Sequence operations.

Use this summary as a reasonableness check. Unexpected depths, feed rates, spindle speeds, dimensions, or operation counts are reasons to stop and correct the CAM setup before generation.

13.5.2 Current Validation Limits

Validate currently builds the summary but does not yet provide a complete manufacturing approval. The Validation & Status indicators and the OK values in Operations are provisional. Generate checks that the Sequence is not empty, but it does not automatically run or enforce the full validation workflow first.

Operator responsibility: Always run Validate manually, inspect the Sequence and summary, and review the generated G-code. A successful export message does not prove that the program is safe or correct for the CNC machine.

13.6 Generating the G-code File

13.6.1 Pre-generation Check

- Every required operation is present and no accidental duplicate operation remains.
- The order matches the real machining, tool-change, board-flip, and workholding plan.
- Edge Cut is positioned so the board remains secure for all earlier operations.
- Every Mirror X checkbox is correct for its operation.
- World origin, physical CNC work zero, G-code units, output folder, and file name are correct.
- The Job Summary contains plausible Z, feed, spindle, dimension, and operation-count values.

13.6.2 Generate

6. Click Generate. If the Sequence is empty, TraceFab stops and displays No operations in sequence.
7. Choose the .nc file name and location in the save dialog and confirm the selection.
8. TraceFab generates one CNC program from the current Sequence in its displayed order and writes the selected origin, units, Mirror X transformations, tools, and operation toolpaths to the file.
9. After writing the file, TraceFab displays a G-code generated confirmation and opens the .nc file automatically in Notepad for immediate inspection.
10. Review the complete file before transferring it to the CNC controller. Pay particular attention to the header, origin, units, first XY movement, Safe Z moves, cutting depths, spindle commands, tool changes, board flips, and final Edge Cut.

13.7 Controls Reserved for Future Development

13.7.1 Roadmap Features

Several controls visible in the Manufacturing Window are placeholders or planned functions and are not yet part of the current production workflow:

- Print Job PDF: A printable NC-program summary and operator checklist, including tool-change and Tool Length Calibration reminders.
- Automated Job Creation: Reusable Sequence recipes and later one-click preprocessing for repeat or batch manufacturing.
- CNC Overrides: Possible last-minute feed, spindle, Safe Z, and tool-change overrides; their necessity and safety model are still being evaluated.
- Validation & Status: Expanded calculated flags, warnings, job summaries, tool-change detection, and automatic validation before export.

- Post-processor selection: The displayed field is not yet an active machine-profile selector in the current release.

14 G-code Preview and Verification

Generated G-code is the final instruction set sent to the CNC controller. TraceFab writes readable comments and opens the resulting .nc file in Notepad, but the operator must still inspect the text, preview the toolpaths in a trusted G-code viewer or controller simulator, and test the program safely before machining a PCB.

14.1 Inspecting the Generated G-code

14.1.1 Program Header

Start by checking the program header. In the current release, TraceFab writes the selected unit command (G21 for millimetres or G20 for inches), absolute positioning (G90), feed per minute (G94), the XY plane (G17), and work coordinate system G54. Confirm that these modes match the CNC controller and the work offset prepared on the machine.

14.1.2 Comments and Operation Markers

TraceFab places comments between parentheses throughout the program. These comments identify the program, generation time, operation type and layer, tool changes, tool descriptions, and selected geometry groups such as individual contours and their point counts. They make the file easier to audit and help the operator prepare the correct tool at each change.

Comments are explanatory only; they do not make a movement safe and they do not replace verification of the following G-code commands. Some CNC interfaces may hide comments even though the controller accepts the surrounding program.

```
( TraceFab - CNC Program )
( Generated: 2026-08-06 19:22:37 )
( ----- Header -----)
G21
G90
G94
G17
G54

(----- Tool change -> 1 / VBIT (3,175 mm / Tip 0,1 mm) -----)
M5
G0 Z10
T1
M6

S10000
M3

(----- Contouring Top -----)
G0 Z3.0000
( Contour 1 - 41 points )
G0 X2.0620 Y-2.0980
G1 Z-0.2000 F100.0000
G1 X2.0680 Y-2.0960 F300.0000
G1 X2.0730 Y-2.0940
G1 X2.0790 Y-2.0900
G1 X2.0830 Y-2.0860
G1 X2.0880 Y-2.0820
G1 X2.0910 Y-2.0770
G1 X2.0970 Y-2.0650
G1 X2.0990 Y-2.0530
```

Figure 14.1 - Generated G-code with header commands, descriptive comments, a tool change, and the beginning of a contouring operation.

14.2 Tool Changes and Tool Length

14.2.1 Generated Tool-change Block

When the next Sequence operation uses a different tool, TraceFab writes a tool-change comment containing the Tool Library display name. It then stops the spindle with M5, moves to Z10, selects the tool with T followed by its Tool ID, issues M6, and restarts the spindle with the configured S value and M3.

M6 behaviour is controller-specific. Depending on the machine, it may start an automatic tool changer, pause for a manual tool change, be ignored, or require a sender-specific macro. Test this behaviour without cutting material before relying on it in a real PCB program.

14.2.2 Tool Length Compensation

Important: TraceFab does not perform automatic tool-length compensation. It currently does not write G43, G43.1, or another tool-length compensation command. After every physical tool change, use the probing, tool setter, Tool Length Calibration, or Z-zero procedure required by the CNC controller before continuing the job.

Test the complete tool-swap workflow in advance: confirm the pause location and Z clearance, install the tool identified in the comment, perform the correct length calibration, verify the resulting Z reference, and resume in a controlled mode. Never assume that M6 alone has corrected the new tool length.

14.3 Graphical G-code Preview

14.3.1 Using an External Viewer

The current TraceFab release does not contain a built-in G-code renderer. Load the generated .nc file into a trusted external G-code viewer, CNC sender preview, or controller simulator. The preview should be based on the actual exported file rather than only on the earlier PCB viewport, because the exported file includes the final origin, units, Sequence order, Mirror X settings, Z moves, and tool changes.

Colour conventions differ between viewers. Identify which colours represent cutting moves, rapid travel, plunges, retracts, and tool changes before interpreting the image.

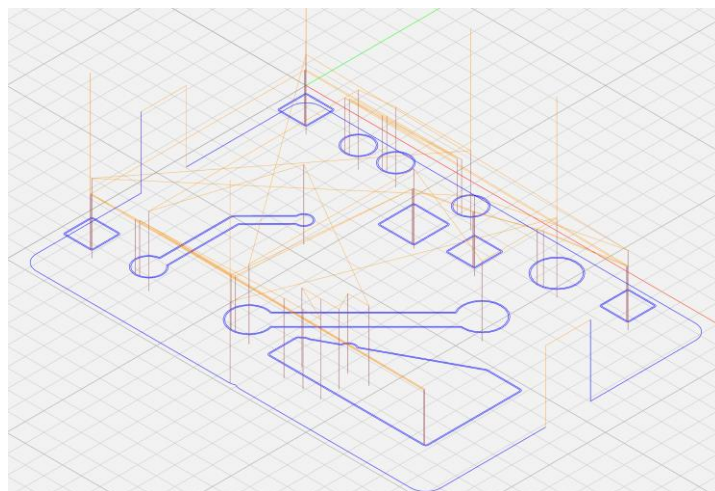


Figure 14.2 - Example external G-code preview showing cutting geometry, rapid travel, and vertical Z movements.

14.3.2 What to Verify in the Preview

- The board outline, holes, contours, and solder-mask paths have the expected size and orientation.
- Top and bottom operations use the intended origin and Mirror X state.
- All machining remains inside the available stock and machine travel.
- Rapid XY movements occur at a safe Z height and do not cross clamps or locating hardware at cutting depth.
- Plunge depths and retract heights are plausible for each tool and operation.
- Alignment-pin holes, board-flip references, and the final Edge Cut appear in the intended program and order.
- Tool changes occur only where expected and no operation silently uses the wrong tool.

14.4 First-run Safety Tests

14.4.1 Dry Run

Perform a dry run for the first PCB on a machine, after a major workflow change, and whenever origin, mirroring, tool-change handling, or post-processing has changed. Use the CNC controller's approved dry-run, simulation, single-block, or raised-Z procedure. Where the controller permits it safely, keep the spindle disabled and run at a reduced feed override while checking the motion.

11. Home the machine and establish the physical G54 work origin and Z0 required by the generated program.
12. Remove the workpiece or raise the tool sufficiently above it using the machine's normal dry-run method. Preserve enough clearance above clamps and alignment pins.
13. Start in single-block or reduced-speed mode and remain at the machine with the emergency stop immediately accessible.
14. Watch the first rapid move, the approach to every operation, all tool changes, the board-flip transition, and the final return or retract.
15. Stop immediately if the direction, scale, origin, Z movement, spindle state, or tool identity differs from the verified plan.

14.4.2 Testing the Board Flip

Before committing a valuable PCB, test the complete flip procedure on scrap material. Drill the alignment-pin pattern, machine a small reference feature on the first side, flip and reseal the board, and verify that a corresponding second-side feature aligns correctly. Keep the waste board, pins, work offset, and clamping reference unchanged throughout the test.

A visually correct G-code preview cannot detect clearance in loose pin holes, debris beneath the board, a bent pin, an incorrect physical flip axis, or a shifted fixture. These errors require a physical alignment test.

14.5 Final Pre-machining Checklist

- The .nc file opened and reviewed is the same file that will be transferred to the CNC controller.
- Units, G54 origin, scale, Mirror X settings, and board orientation are correct.
- The graphical preview shows no unexpected cutting or rapid movement.

- Every tool-change comment matches the required physical tool and the controller's M6 behaviour has been tested.
- A valid tool-length or Z-zero procedure is available after every tool change.
- Safe Z, cutting depths, clamps, waste-board thickness, spindle speeds, and feed rates are suitable.
- The first dry run and, where required, the board-flip test were completed successfully.
- The operator will remain with the machine during initial machining and can stop it immediately.

14.6 Future Built-in Preview

14.6.1 Roadmap

A native TraceFab G-code viewer and simulator is planned for a future release. The intended feature will visualise the actual generated .nc program, distinguish cutting and rapid moves, show tools and operation boundaries, support top and isometric views, and flag suspicious bounds, origin, unit, and Z movements.

Safety note: A future built-in preview will remain an advisory verification tool. It will not replace controller-specific simulation, manual G-code inspection, correct workholding, or a physical dry run.

15 Settings and File Management

TraceFab uses two separate settings forms. Program Settings controls how the application behaves; CAM Parameters defines the default CNC machining values used by the manufacturing operations. Changing a software preference does not change a machining parameter, and changing a CAM default does not change the application interface.

15.1 Program Settings

Open this form through Settings > Program Settings. It contains software, UI, folder, import, user-experience, and diagnostic preferences. The current beta window is still under development, so controls marked as unfinished should not be relied upon until a release note confirms that they are active.

UI / Viewport

- **PCB colors:** Set the display colors for the Top Layer, Bottom Layer, pads, vias, PCB substrate, and copper substrate.
- **Preview colors:** Set the colors for contours, Excellon holes and labels, edge-cut slots, soldermask removal, and alignment pins.
- **Background:** Set the viewport background color.

File / Project

- **Default Project Folder:** Preferred location for TraceFab project files.
- **Default G-code Export Folder:** Preferred destination for generated .nc files.
- **Default Import Folder:** Starting folder for PCB imports.
- **Default Backup Folder:** Preferred location for project backups.
- **Auto Save:** Planned automatic project saving.

- **Auto-Save Interval:** Planned interval between automatic saves.
- **Default Save Location:** Planned default location for saved work.

Import

- **Geometry Match Tolerance:** Tolerance used when matching imported KiCad geometry, in millimetres.
- **Default Units:** Planned default unit selection for Gerber import.
- **Arc Tolerance:** Planned tolerance for converting imported Gerber arcs.
- **Segment Resolution:** Planned resolution for segmented Gerber geometry.
- **Auto-Match Excellon File:** Planned automatic matching of drill data to the Gerber job.

User Experience

- **Reopen Last Project on Startup:** Reopen the most recently used project when TraceFab starts.
- **Show Welcome Panel:** Show startup information and guidance.
- **Invert Mouse Wheel Zoom:** Reverse the mouse-wheel zoom direction.

Logging

- **Enable Logging:** Enable diagnostic log output.
- **Log Folder Path:** Select where log files are stored.
- **Open Log:** Open the current log for inspection or support.
- **Support / Diagnostics:** Display basic counts for active layers, pads, vias, and drills.
- **Dump License:** Export license information for troubleshooting.
- **Dump Tool Library:** Export the tool library for troubleshooting.

Persistent TraceFab data is stored under AppData\Roaming\TraceFab. Back up this folder before manually changing or removing persistent settings, licenses, libraries, or other application data.

15.2 CAM Parameters

Open this form through Parameters > CAM Parameters. These values are CNC defaults that prefill or control the individual machining operations. Where an operation window offers its own setting, the operator can adjust that value for the current job. Controls shown with struck-through labels are marked as unfinished in the current beta and should not be relied upon.

General

- **Safe Z Height:** Clearance height used for non-cutting movement above the workpiece.
- **Safe Z Speed:** Travel speed used for safe vertical movement.
- **Plunge / Retract Feed:** Intended default vertical feed rates; currently marked unfinished.
- **Speed Units / CNC Units:** Default speed and length units for CAM values and G-code output.
- **Spindle Speeds:** Default, minimum, and maximum spindle RPM limits; the default value is currently marked unfinished.

Contouring

- **Default Tool:** Tool initially selected for copper isolation contouring.
- **Passes / Pass Overlap:** Default isolation-pass count and overlap between adjacent passes.

- **Cut Side / Extra Offset:** Default tool position relative to copper and any additional clearance.
- **Cut Depth / Max. Depth:** Default contouring depth and intended per-pass depth limit; Max. Depth is currently marked unfinished.
- **XY Feed / Plunge Feed:** Default horizontal cutting and vertical entry speeds.

Drilling

- **Extra Depth Below PCB:** Additional drilling depth below the board thickness.
- **Pecking / Full Retract:** Planned chip-clearing drill behaviour; currently marked unfinished.
- **Plunge Feed:** Default drilling feed into the board.

Edge Cut

- **Extra Depth / Multipasses / Max. Depth:** Extra depth below the PCB and optional maximum depth per cutting pass.
- **Default Gap Size:** Default bridge width that keeps the milled PCB attached until it is snapped free.
- **Default Gap North / East / South / West:** Board sides where bridge gaps are added automatically.
- **Default Tool / Cutting Direction / Cut Side:** Initial edge-cut tool, travel direction, and tool position relative to the outline.
- **XY Feed / Plunge Feed:** Default horizontal milling and vertical entry speeds.

Solder Mask

- **Default Tool:** Tool initially selected for mechanical solder-mask removal.
- **Pass Overlap:** Overlap between adjacent removal passes.
- **Remove From Pads / Vias:** Choose which feature types are processed.
- **Cut Depth / XY Feed:** Default removal depth and horizontal feed rate.

Centering

- **Default Tool:** Tool initially selected for alignment-pin holes.
- **Pin Size / Pin Depth:** Default alignment-hole diameter and total machining depth from the active Z zero.
- **Helix Cut:** Allow a smaller flute to create a larger alignment hole using a helical path.
- **XY Feed / Plunge Feed:** Default helical movement and vertical entry speeds.

16 Typical Workflows

The appropriate workflow depends on the source files and whether the PCB has one or two copper layers. Import and validate the complete board before calculating machining operations. Confirm the board dimensions, copper layers, drill data, and closed edge outline before building an NC program.

16.1 Importing the PCB

16.1.1 KiCad Import

- Use the **KiCad** button for a one-click import of the .kicad_pcb file. TraceFab reads the board geometry and supported layers from this single vector-based project file.

- After import, inspect the viewport, board dimensions, copper layers, holes, and edge outline before continuing.

16.1.2 Gerber Import

- Use the **Gerbers** button when working with manufacturing files. Import and assign the required files per layer, including the copper, edge, and Excellon drill data.
- In the Gerber wizard, match the units and zero-suppression settings to the source files. Validate the preview before clicking **Let's Go!**
- Once the Gerber job has been imported and normalized, the CAM workflow is the same as for a KiCad project.

16.2 Single-sided PCB Workflow

A single-sided board is the simplest workflow. Alignment pins and mirrored Sequence operations are normally unnecessary because the PCB is not flipped between copper-side operations.

1. Import the PCB with **KiCad** or **Gerbers**, then validate the visible copper layer, drill data, board dimensions, and edge outline.
2. Calculate the copper isolation paths with **Calc. Contours** for the active copper layer.
3. Configure and calculate the holes with **DrillSetup**. If soldermask is used, apply and UV-cure it manually before running the calculated **SolderMask** removal operation.
4. Calculate the board outline with **Calc. Edge Cut**. Keep **Edge Cut** as the final machining operation so the PCB remains fixed during all earlier work.
5. Open **Manufacture**, add the required operations to the Sequence in machining order, select the world origin and G-code units, validate the job, and click **Generate**.

A typical single-sided operation order is Contouring, Drilling, SolderMask Removal, and Edge Cut. When manual coating, curing, cleaning, or tool preparation is required between operations, split the work into separate .nc programs rather than treating the complete process as unattended automation.

16.3 Double-sided PCB Workflow

A double-sided PCB is prepared from the top-layer coordinate view. KiCad and Gerber geometry remains in the normalized Cartesian coordinate system after import. Physically flipping the copper board does not modify that geometry, so every operation executed after the flip must be mirrored explicitly in the Manufacturing Sequence.

1. Import and validate both copper layers. Confirm that Top and Bottom features, drill data, board dimensions, and the edge outline align correctly in the viewport.
2. Create two or four registration holes with **Align Holes** or **Manufacturing > Alignment Pins**. Calculate the Alignment Pins operation before building the first NC program.
3. In **Manufacture**, set **Set World origin** to a numbered lower alignment pin, normally **Alignment Pin 2** or **Alignment Pin 4**. This provides a repeatable X0 Y0 reference that remains physically accessible on the raw copper board.
4. Machine the alignment-pin holes through the PCB and into the wasteboard, install the pins, and confirm that the board seats without play or mechanical shift. You can do this manually or simply adding this sequence in the full NC program.

5. Prepare the first, top-side NC program. A typical sequence contains **Alignment Pins**, Top **Contouring**, and Top **SolderMask Removal** after the soldermask has been applied and UV-cured manually. Leave **Mirror X** disabled for these pre-flip operations.
6. After the top-side work is complete, stop the CNC safely and flip the board on the alignment pins. Do not establish an unrelated new work origin; retain or restore the selected alignment-pin zero according to the controller procedure.
7. Prepare the post-flip NC program with Bottom **Contouring**, **Drilling**, Bottom **SolderMask Removal**, and finally **Edge Cut**. Apply and UV-cure soldermask manually before executing its removal operation.
8. In the **Manufacture** Sequence, enable **Mirror X** for every operation that will run after the physical board flip, including Contouring, Drilling, SolderMask Removal, and Edge Cut. TraceFab does not mirror Bottom Layer operations automatically.
9. Validate and generate the second .nc program. Review the mirrored toolpaths in a G-code viewer, perform a dry run, and confirm the origin, flip direction, board limits, Z clearance, and tool sequence before cutting.

Important: Mirror X is selected per Sequence row. Enabling it on only one post-flip operation can produce a board whose operations no longer align. Verify every row in the second-side Sequence before generating G-code.

17 Troubleshooting

At the time of this manual revision, no recurring errors have been reported that require a documented troubleshooting procedure. Verified issues and their solutions will be added to this chapter in future manual releases.

18 Updates, Support, and Feedback

18.1 Software Updates

18.1.1 Checking for Updates

TraceFab does not require a separate update checker window. Update announcements are distributed through the application and the TraceFab website.

- **News panel:** The panel slides out from the right side of the main workspace. When an internet connection is available, TraceFab requests the latest news through an API call. New releases and important update information are shown here.
- **Trace-Fab.com > Downloads:** The Downloads page shows the latest available TraceFab version and provides the current installer and portable package.

18.1.2 Updating an Installed Version

1. Download the newest installer from **Trace-Fab.com > Downloads**.
2. Close TraceFab and any generated G-code or project files that are still open.
3. Uninstall the previous TraceFab version through Windows.
4. Run the new installer and complete the installation wizard.
5. Start TraceFab and verify the installed version under **Help > About**.

Persistent data: A normal uninstall and reinstall leaves the data stored under **AppData\Roaming\TraceFab** in place. This preserves settings, the license, and other persistent application data. Back up this folder before an update if it contains important custom data.

18.1.3 Updating a Portable Version

1. Close TraceFab.
2. Delete the old portable application folder only. Do not delete **AppData\Roaming\TraceFab**.
3. Extract the new portable ZIP file into a new folder.
4. Open the extracted folder and start **Trace-Fab.exe**.
5. Verify the version under **Help > About**.

18.1.4 Release Information

Release information and this manual are supplied with the installer and portable edition. The installed software also displays version and release information under **Help > About**.

18.2 Getting Support

18.2.1 Support Channels

- **E-mail:** support@trace-fab.com for support questions and issue reports.
- **Website chat:** [Trace-Fab.com](https://www.trace-fab.com) allows you to leave a message. Include valid contact details if you would like a reply.
- **Discord:** <https://discord.gg/wT49aXbFYZ> is useful for community support, bug discussions, and sharing screenshots.

18.3 Reporting Problems

18.3.1 Information to Include

A useful report should contain enough information to reproduce and identify the problem. Where possible, include:

- **TraceFab version:** The exact version shown under Help > About.
- **Package type:** Installed or portable edition.
- **System:** Windows version and relevant display scaling when the issue concerns the UI.
- **Import type:** KiCad or Gerber, including the relevant file type or layer.
- **Reproduction steps:** The actions taken immediately before the problem occurred.
- **Expected and actual result:** What you expected TraceFab to do and what happened instead.
- **Evidence:** Screenshots, warning text, logs, or exported diagnostic information when available.

18.3.2 Screenshots, Files, and Privacy

Screenshots are often the fastest way to explain a UI or toolpath problem, especially through Discord. If the issue depends on a PCB file, support may ask for a reduced sample that reproduces the behaviour.

Before sharing files, remove confidential board data, customer information, credentials, private license files, and any other sensitive content. Only send project or log files when you are authorised to share them.

19 Disclaimer

19.1 Operator Responsibility

TraceFab is a PCB CAM application that generates G-code. It does not control, supervise, or safeguard the CNC machine, and it cannot verify the machine, controller, workholding, tools, material, or operator setup. The operator remains responsible for the safe preparation, verification, and execution of every CNC job, including compliance with the machine manufacturer's instructions and all applicable safety requirements.

19.2 Mandatory G-code Verification

Before running any generated file, the operator must inspect the complete G-code and confirm that it is suitable for the intended CNC controller, machine, tooling, material, and workpiece setup. The exported file should also be checked in a trusted G-code visualizer or controller simulator.

- **Units:** verify that **G20** selects inches or **G21** selects millimetres as intended.
- **Positioning and feed modes:** verify commands such as **G90** for absolute positioning and **G94** for feed per minute.
- **Coordinate setup:** confirm the selected world origin, work coordinate system, board side, and any required mirroring before machining.
- **Machine motion:** check XY limits, safe Z moves, cutting and drilling depths, feed rates, spindle speeds, and all tool-change positions for unexpected or unsafe values.
- **First or unfamiliar jobs:** perform a dry run with the spindle stopped where practical, remain at the machine, and keep the emergency stop within immediate reach.

19.3 Responsibility and Liability

Generated G-code may require controller-specific review or adjustment and must never be treated as automatically safe or machine-ready. To the maximum extent permitted by applicable law, the developers and distributors of TraceFab are not responsible for personal injury, equipment or material damage, production loss, data loss, or other loss resulting from the use of generated output, an incorrect machine setup, modified G-code, or failure to perform the required checks. Nothing in this disclaimer excludes or limits rights or obligations that cannot legally be excluded or limited.

20 Revision History

Current status: Version 1.0

Significant changes to this manual will be recorded here from the first public manual release onward.