

PCBAtlas User Handbook



A workspace for viewing, organizing, and exporting PCB design and manufacturing data

The current version of PCBAtlas is built around a practical loop: bring board data into the library, inspect it through **2D / 3D / Data**, manage revisions, and export what you need from one consistent workspace.

Library Management

2D Main Workflow

Board-Level 3D Review

Original-Source Export

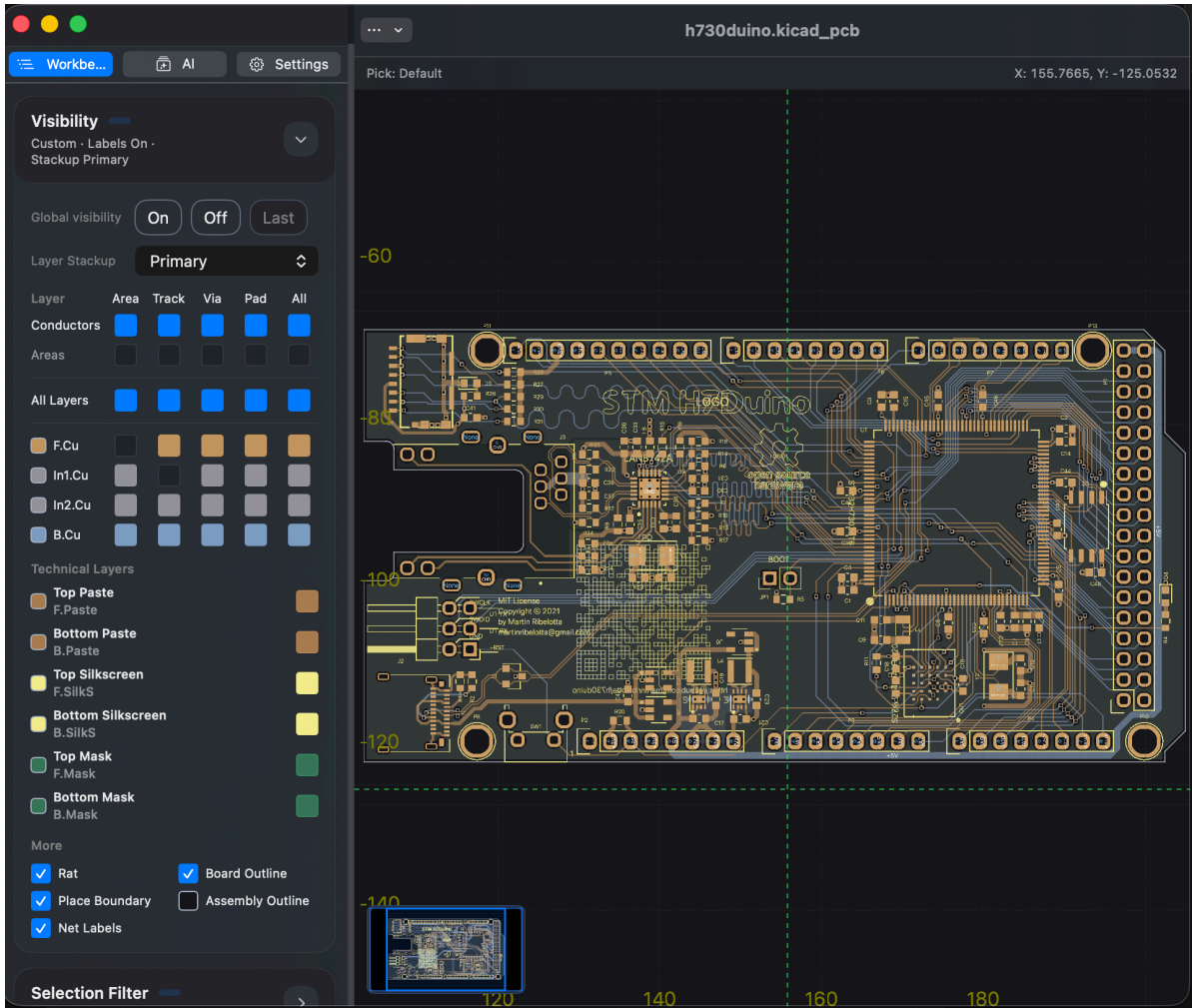
Converted Export

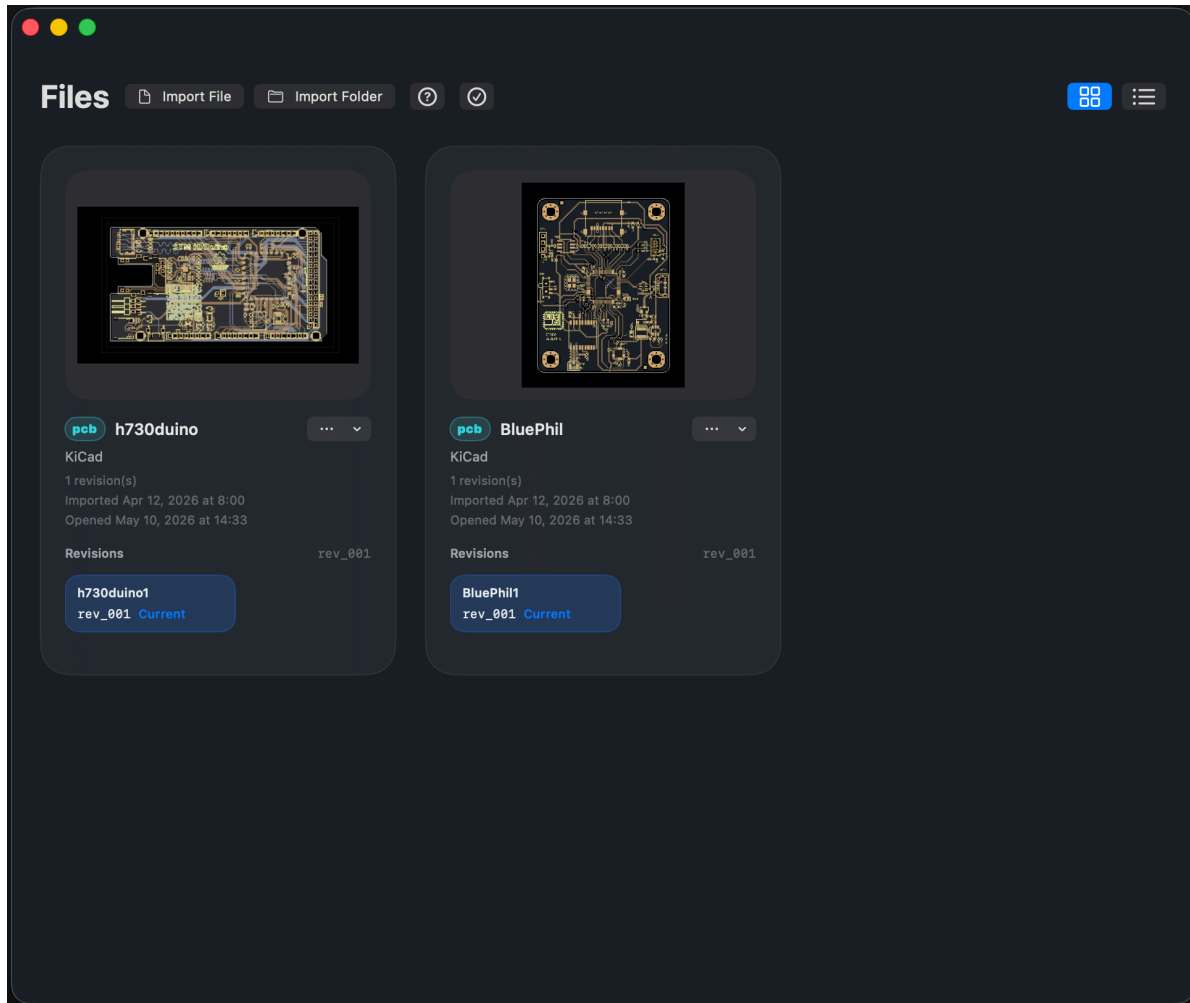
Project / schematic

AI Workspace

Three things worth knowing before you dive in

- **Data** is not only a summary page. It also carries **Quick Estimate / Revision Quote**, revision details, and issue summaries in the current release.
- Export still has to start from **2D**, and export access must already be unlocked. If it is still locked, the app opens the purchase center first.
- **AI** depends on your configured AI service setup, network access, and sharing settings. It helps with diagnosis and summaries, but does not replace engineering review or manufacturing validation.
- PCBAtlas currently supports Apple platforms: **iOS / iPadOS 17.0 or later** and **macOS 14 or later**.





What You Can Do Today

- **Library management:** import design files or manufacturing data and maintain multiple revisions of the same project.
- **2D inspection:** pan, zoom, box-select, inspect object details, and use Workbench to narrow down what matters.
- **3D review:** open documents that form a board session in 3D and inspect board body, surfaces, pads, vias, silkscreen, and component bodies. When imported component models are available and renderable, PCBAAtlas prefers those model bodies; missing or unsupported models stay visible through proxy bodies or issue records.
- **Data-page review:** check document, revision, statistics, status summaries, and `Quick Estimate / Revision Quote` in `Data`.

- **Export and sharing:** export `PNG / PDF` , or export the current revision's original project files and converted `DSN / KiCad 8` outputs.
- **Settings and AI:** search settings, change terminology preferences, configure AI services, and review support and open-source information.
- **Project and schematic review:** after importing `KiCad .kicad_pro / .kicad_sch` , browse the source tree in `Project` , switch sheets in the schematic Workbench, search `symbol / net / label / text / marker` , and pick `symbol / pin / wire / label` directly on the canvas.

Quick Entry Points

Install and launch

Make sure the app starts cleanly and the library home page is ready.

First launch and import

Walk through first-run samples, import, and opening the workspace end to end.

Five-minute tour

Take the shortest path through the current library, 2D, Workbench, Data, 3D, and export flows.

Library and revisions

Understand how documents, revisions, duplication, pinning, and sharing fit together.

2D Viewer

Inspect objects, open [View Info](#), and prepare exports in the current main working view.

Quick Estimate

Understand the local manufacturing estimate, Revision Quote status, and quote results shown on the Data page.

Export and sharing

Understand the difference between capture export, original-source export, and converted export.

Recommended Reading Order

1. [Install and launch](#)
2. [First launch and import](#)
3. [Five-minute tour](#)
4. [Library and revisions](#)
5. [2D Viewer](#)
6. [Quick Estimate](#)
7. [Workbench](#)
8. [Export and sharing](#)
9. [Settings](#)

How This Handbook Is Organized

- **Quick Start:** first launch, import, and the shortest end-to-end walkthrough.
- **Guides:** task-based pages for the library, viewers, Workbench, export, and settings.
- **Troubleshooting:** the most common issues around import, permissions, and loading.

- **QA Appendix:** practical checklists for test and regression work.

Scope of This Handbook

- It documents the current code-backed user-visible behavior and does not explain internal architecture.
- When platforms differ, the handbook explicitly separates `iOS` / `iPadOS` and `macOS`.

Install and Launch

Goal

Confirm that the app launches cleanly and lands on the library home page so you can move on to real work.

Supported Platforms

- **iOS / iPadOS:** 17.0 or later.
- **macOS:** 14 or later.

Intended Audience

- New users launching PCBAAtlas for the first time
- Testers confirming that the app reaches the library successfully

Prerequisites

- PCBAAtlas is installed on the current device
- The app can be launched normally from the system

Steps

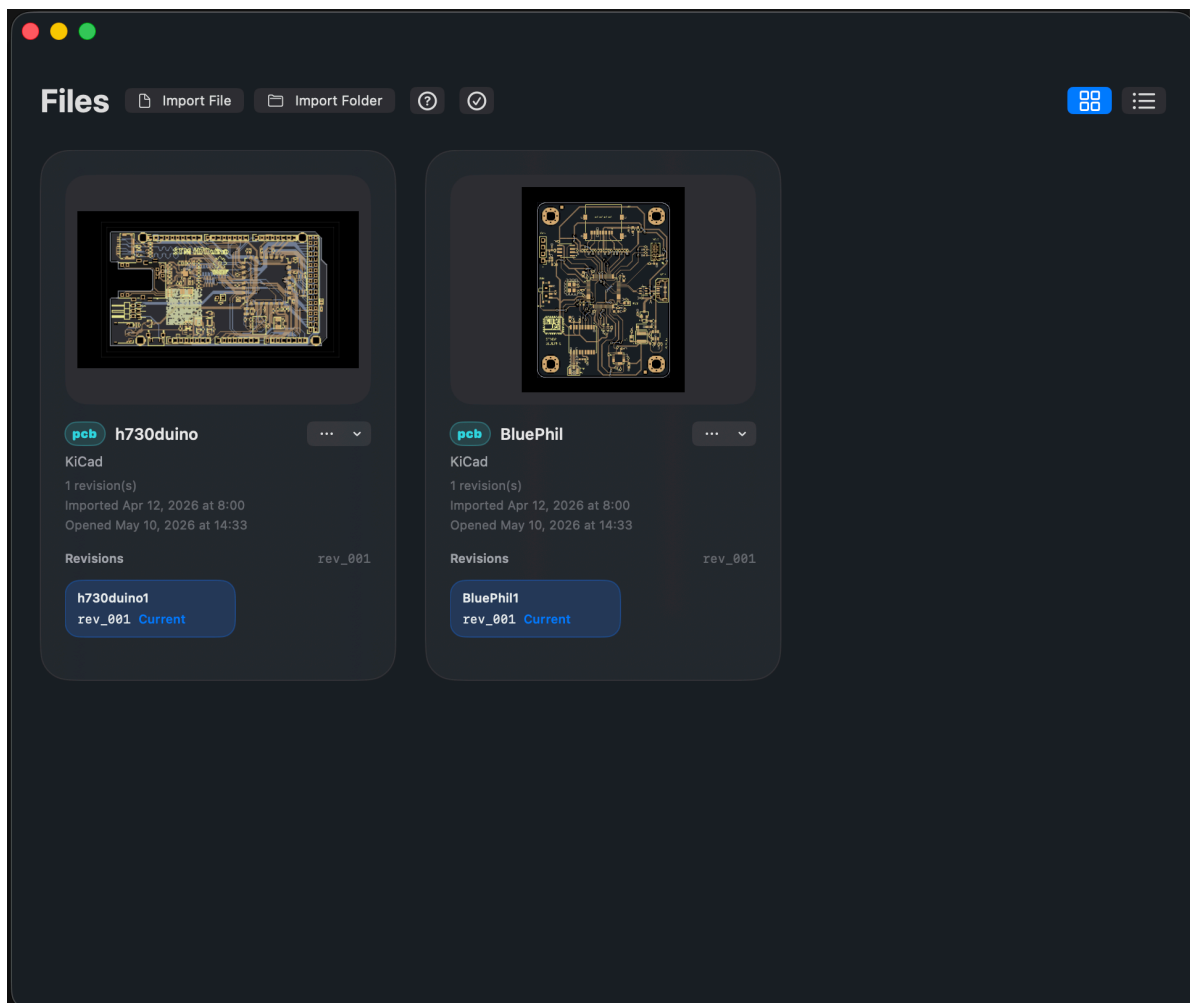
1. Launch PCBAAtlas and wait for the library home page to appear.
2. If the library starts empty, confirm that the built-in sample documents are seeded automatically.
3. Check that layout switching, import entry points, and basic home-page interaction all work.

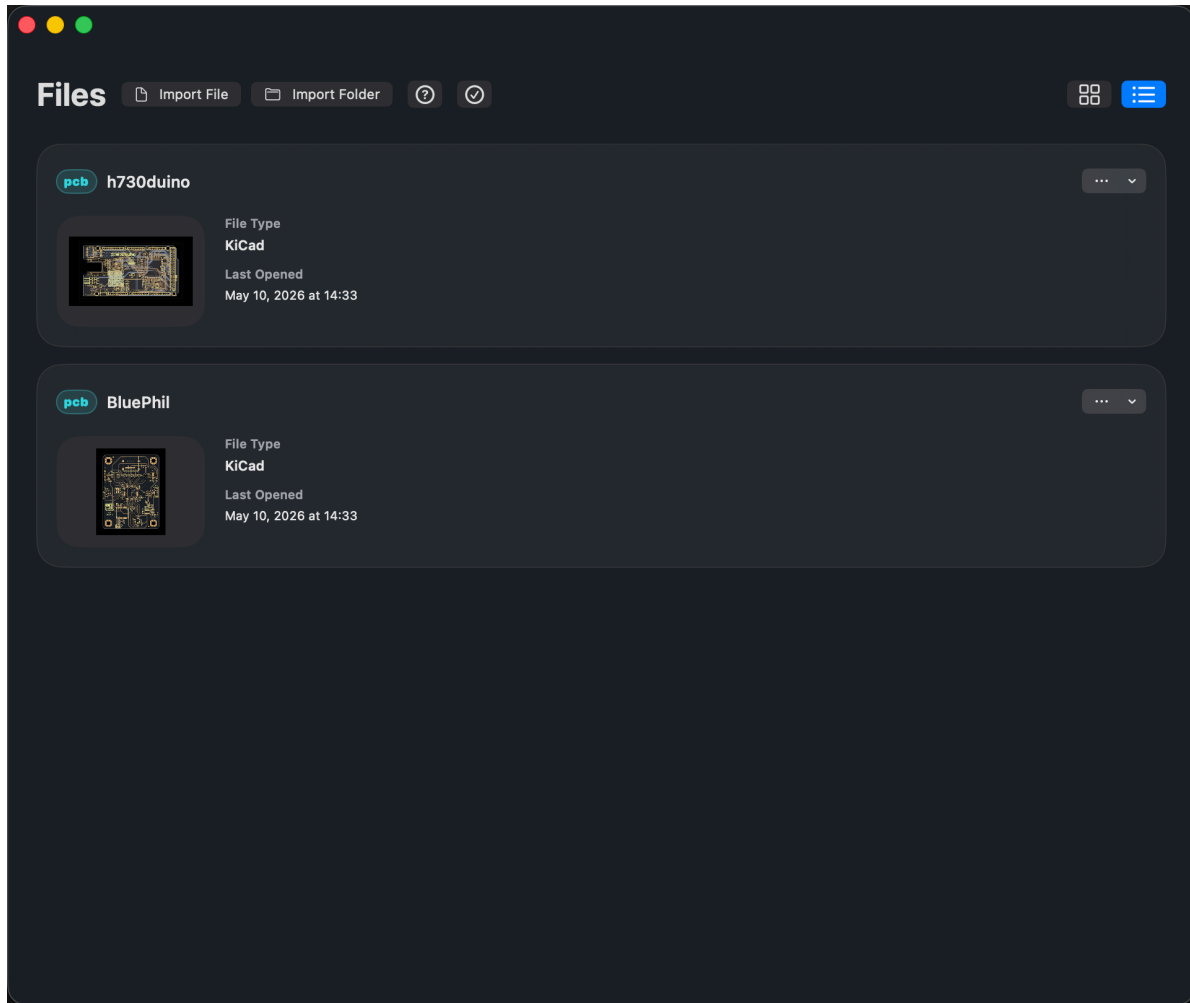
Result

- The app reaches the library home page and is ready for import or document opening.

What You See the First Time

1. The app opens to the **Library** home page instead of jumping straight into a board.
2. If the library is completely empty, the app automatically seeds it with three built-in sample documents: `h730duino`, `BluePhil`, and `WooKey Mainboard`.
3. The library supports both card and list layouts, and it exposes entry points for import, feedback, help, and batch selection. The ladybug button at the top explains feedback and data boundaries before opening a prepared bug-report email; the app does not automatically upload projects, usage information, or diagnostics.





Suggested Checks

1. The app launches reliably without repeated error dialogs.
2. The home page responds to clicks, scrolling, and layout switching.
3. You can see the library title and the import entry points.
4. If the library was previously empty, the three sample documents are visible.

Success Criteria

- You can reach the library home page.
- The home page is interactive and lets you continue with import or document opening.

- In an empty-library case, the sample documents appear; in an existing-library case, your existing document list appears.

Platform Differences

- **iOS:** compact layouts are more common, top space is tighter, and panel switching inside the workspace often happens from the bottom tab area.
- **macOS:** both the library and the workspace benefit from wider side-by-side layouts that show more information at once.

Common Questions

What if the app launches to a blank screen?

Quit the app completely and reopen it. If it still cannot reach the library, record any system error prompt and check [Permissions and file access](#).

Is an empty library abnormal?

No. As long as you can import files or see the built-in sample documents, the initial state is normal.

First Launch and Import

Goal

Help you complete a full first-use loop: start in the library, import something, open it, and return to the library again.

Supported Platforms

- iOS
- macOS

Intended Audience

- New users who want to complete their first import loop
- Testers validating import behavior from the library into the workspace

Prerequisites

- PCBAtlas can already launch into the library
- You have either one of the built-in samples or your own design/manufacturing project files ready

Steps

1. Start from the library and choose `Import File` or `Import Folder` for the source.
2. If the source is `IPC-2581 XML`, choose the import intent when prompted.
3. Wait for the library progress banner to complete, then open the resulting document.

4. Confirm that standalone PCB sources open in PCB Viewer; schematic sources and KiCad projects with a schematic open in Schematic Workbench, and revisions are updated as expected.

Result

- The source is imported into the library as a new document or revision, and the workspace opens the corresponding PCB Viewer or Schematic Workbench.

Default Content on First Cold Launch

When the library is empty, the app automatically seeds it with three sample documents that can be opened right away:

- `h730duino` (ODB++ manufacturing data)
- `BluePhil` (KiCad design board)
- `WooKey Mainboard` (a KiCad project containing a PCB and hierarchical schematics)

If you just want to learn the interaction flow, open one of these samples first. If you want to validate your own data, continue with the import flow below.

Current Import Paths

Import File

- a standalone `DSN`, `KiCad .kicad_pcb`, `KiCad .kicad_sch`, or `IPC-2581 XML`
- an ordinary KiCad `.zip`, or an ODB++ / Gerber / Drill manufacturing archive recognized by the app
- a compatible `.epcbdoc.zip` document package

Do not import `.kicad_pro` as an isolated file. A project needs its sibling schematics, libraries, and settings, so use `Import Folder`. A standalone `.kicad_sch` can be imported directly only when it does not reference child sheets. A multi-sheet schematic must also be imported through its project folder; the app does not present a second file picker to obtain directory permission.

Import Folder

- a KiCad project directory with a valid `.kicad_pro`, including schematic-only projects

Source Selection in Folders and Archives

`Import Folder` accepts KiCad projects only: PCBAtlas must find a valid `.kicad_pro`. If there is no project file or the project cannot be reopened, import fails directly rather than falling back to a `.kicad_pcb` or `.kicad_sch` import. If multiple projects are found, it asks you to choose explicitly rather than selecting one silently. After a successful import, every regular file in the selected directory is preserved in the same revision at its original relative path, including notes or manufacturing output outside the project closure. An unreadable file, symbolic link, or another unsupported directory entry fails the whole import rather than saving only part of it.

When you choose a project directory from Files, iCloud Drive, or a third-party file provider, the app first prepares the complete directory as a local temporary snapshot and then checks the project entry. Historical directories such as `.history` stay in the revision at their original paths, but a PCB or schematic inside them remains historical supporting material and does not compete with the current design beside `.kicad_pro` for the opening entry.

For a project that still uses KiCad 5 `.sch` schematics, if the `.kicad_pro` points to a same-name `.kicad_sch` that no longer exists but an exactly same-name `.sch` is present in the directory, PCBAtlas recognizes that `.sch` and its child sheets as the project schematic and permits a dual PCB-and-schematic canvas. It does not select an unrelated `.sch`. Legacy project directories with only `.pro` and no `.kicad_pro` remain outside the `Import Folder` support boundary.

When a KiCad project opens, the app first opens the schematic with the same name as the project; if none has that name, it selects an available schematic by path order. A project with both PCB and schematic prepares two canvases in one progress stage and shows dual canvases by default when preparation completes. A project with only one design source shows a single canvas and has no canvas-switch button.

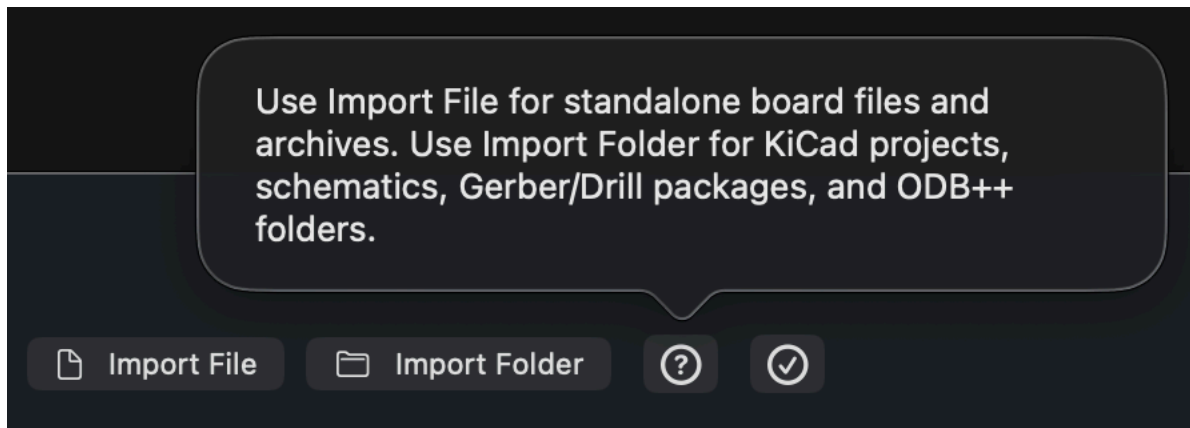
Special Behavior for IPC-2581

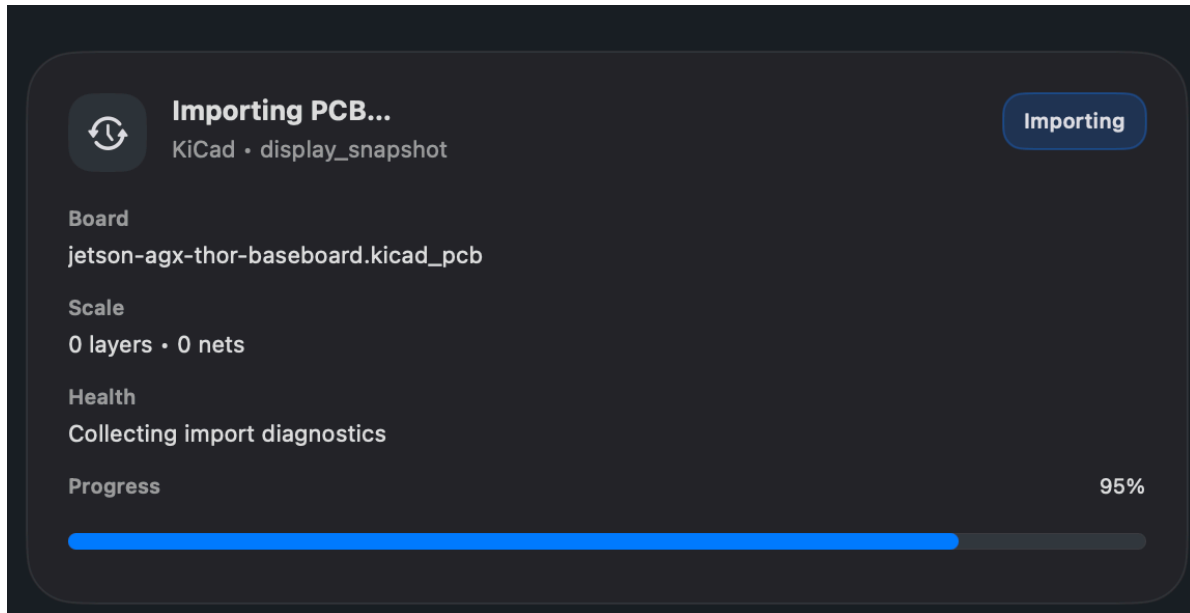
When you select an `IPC-2581 XML`, the app does not assume whether it should be treated as design-side or manufacturing-side data. Instead, it shows a confirmation dialog so you can choose:

- import as **design data**
- import as **manufacturing data**

Recommended Steps

1. Click `Import File` or `Import Folder` on the library home page.
2. Use `Import File` for standalone files and archives; use `Import Folder` for a complete KiCad project directory containing `.kicad_pro`.
3. After selecting a file, if it is an `IPC-2581 XML`, choose the import intent.
4. Wait for the import progress banner at the top of the library to advance.
5. When import finishes, a new document card appears, or the revision count of the target document increases.
6. Open the document. Confirm that a project with both PCB and schematic shows dual canvases by default; a document with only one design source shows a single canvas. Also confirm complete `Project / Data` content.





What Feedback You Will See

- The top of the library shows an in-progress banner with a title, stage detail, and percentage.
- When opening a project, portrait or compact layouts stay on the Viewer progress page. Interactive content is shown directly only after canvas preparation completes; the UI does not switch early to `Project`.
- If you manually open another document during import, the task still completes, but it does not forcibly switch focus back to the newly imported document.
- If you drop a new board file into the workspace while another document is open, the app first asks whether to replace the current content.

After Importing a KiCad Project or Schematic

After importing `.kicad_pro` through `Import Folder` or an ordinary KiCad `.zip`, PCBATlas stores the related KiCad project files inside the library revision and exposes a `Project` page in the workspace. `Import Folder` also retains other regular files from the selected directory. It groups content under `Source Documents / Libraries / Settings` so you can confirm which board files, schematics, library files, and settings were captured. A standalone `.kicad_sch` enters the schematic workspace directly through `Import File` without requesting its parent directory.

When you open a schematic, the workspace still uses the `2D` canvas, but the right Workbench switches to the schematic-specific structure:

- **Sheets** : inspect hierarchy, breadcrumb, and openable child sheets.
- **Search** : search `Symbols / Nets / Labels / Text / Markers` in `Current Sheet` or `Whole Project`.
- **Selection / Details** : show the object selected from search results or canvas pick.
- **Problems** : show import issue details, unsupported content, and clear issue messages when the schematic workbench is unavailable.

The schematic canvas supports picking for `symbol / pin / wire / bus / label / text / junction / marker`. Selection, details, and highlight stay synchronized, overlapping hits can be cycled, and a selected non-net object is framed by an orange locator. When a PCB route is available, single-canvas mode offers `View on PCB`; it switches to the PCB, selects the matching placement or pad when possible, otherwise highlights the connected net, and fits the view to the target. Split view keeps cross-probe linked across both canvases. Layout edits stay in dedicated EDA tools.

Revision-Related Actions

From the action menu on a document card, you can:

- **append a design revision**
- **append a manufacturing revision**

That means import is not only for creating new documents. It is also how ongoing revision history is maintained for the same project.

Current Rule for Re-importing the Same Name and Format

When a source comes back with the same document name and the same format, the current version treats it as another input round for the same document rather than automatically creating a new top-level card:

- if the newly imported original source is **byte-identical** to the current revision, the app reports that nothing changed and ignores the import
- if the original source is **different**, the app stores it as a new revision under that document and switches to the latest revision
- revision labels are displayed dynamically as `A1 / A2 / ...` based on the document name

Recommendations

- For a first validation pass, start with a single `DSN` or `KiCad` file.
- For manufacturing data, confirm the archive or directory is complete before importing.
- For iterative updates to the same project, prefer appending revisions instead of creating multiple nearly identical documents.

Common Questions

Why does the app ask again after I pick `IPC-2581` ?

Because the same `IPC-2581 XML` may be treated as design-side or manufacturing-side data, and the current version asks you to choose explicitly.

Why do the design and manufacturing intents for the same `IPC-2581` sometimes look almost identical after import?

That is normal in the current version. For `IPC-2581` , the main difference today is the library-side import intent and document classification, not an immediately obvious visual difference in the workspace.

- If you choose the design intent, the document is managed as design-side material.
- If you choose the manufacturing intent, the document is managed as manufacturing-side material.
- That classification affects later management behavior, such as the default append-revision type.
- Importing the same `IPC-2581` with both intents usually creates two separate documents instead of auto-merging them.

At the same time, the current display behavior does not yet express the design-vs-manufacturing distinction very strongly once the file is open, so the visible result can still look quite similar.

Why doesn't the import picker list every manufacturing-package suffix?

Manufacturing import accepts common archives, directories, and general data inputs. The final result depends on what the app can actually recognize from the selected source.

Five-Minute Tour

Goal

Take the shortest possible path through the app, experience the most important current value, and quickly judge whether this version is ready for day-to-day review and communication.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users who want a quick confidence check before deeper use
- Testers who need a short smoke path through the current UI

Prerequisites

- A sample or real board document is available in the library
- If you want to validate 3D, use a sample that can form a board session
- If you want to complete export end to end, export access should already be unlocked

Steps

1. Open one document from the library and move through 2D , Data , and, when applicable, 3D .
2. Use Workbench , View Info , and AI once each so the main workspace paths are exercised.
3. Return to 2D , open the export configuration from the bottom floating toolbar, and confirm one export. If export access is still locked, confirm that the purchase center opens.

Result

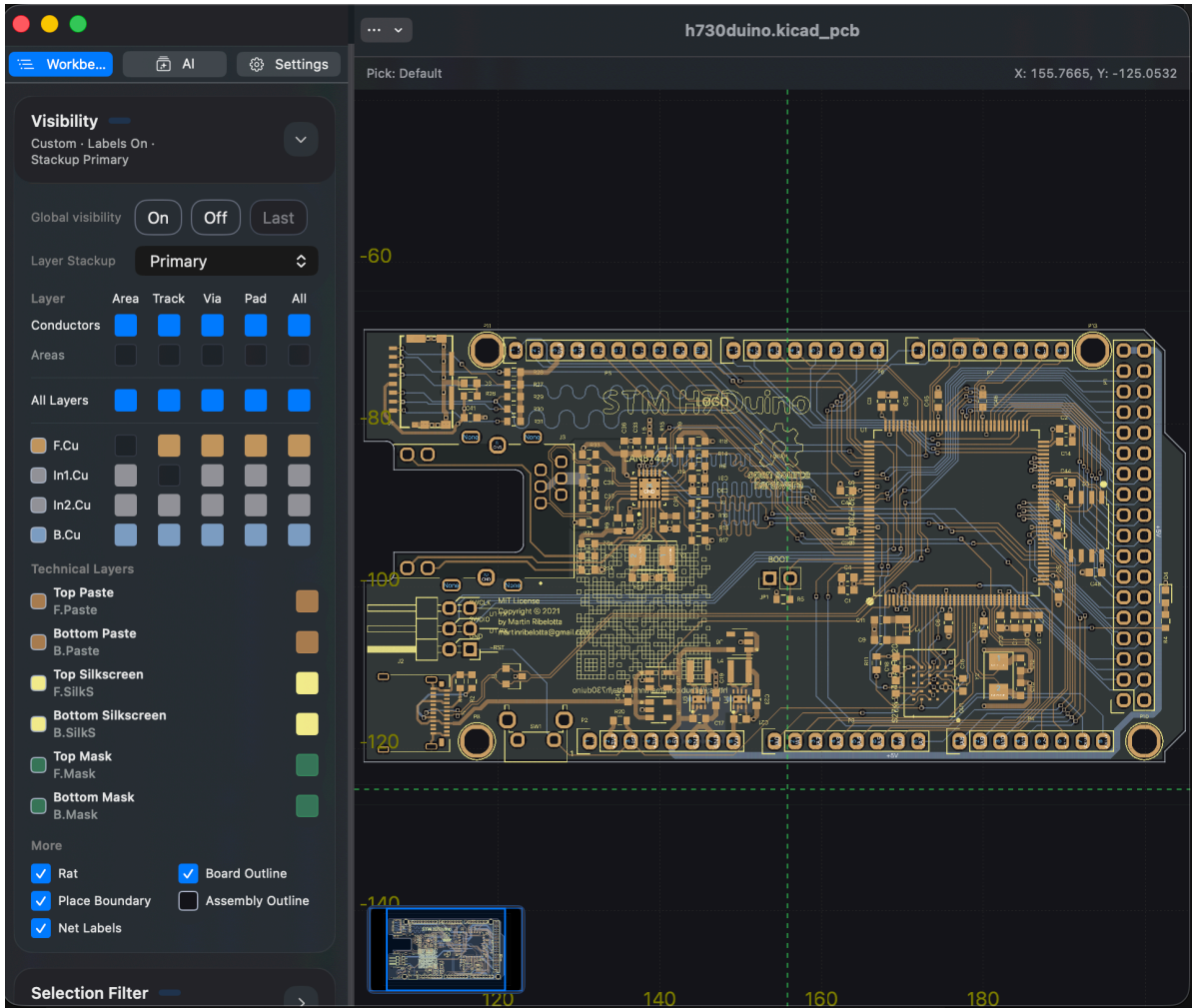
- You can quickly confirm whether the current build is usable for routine inspection, navigation, and export-related workflows.

Recommended Samples

- If you want to validate `2D / 3D / Data` together, start with `h730duino` or `BluePhil` from the library.
- If you mainly care about the import path, bring in one of your own `DSN` or `KiCad` files.

Five-Minute Route

1. Open a document from the library.
2. In `2D`, do one pan, one zoom, and one object selection.
3. In `2D`, right-click or long-press a target, choose `View Info`, and confirm that the info card appears and can be copied.
4. Open `Workbench` and confirm the default sections are `Visibility`, `Selection Filter`, and `Objects`.
5. In `Objects`, search for one item and try at least one action from `Locate / Highlight / Isolate / Clear`.
6. Switch to `Data` and confirm that the current document, revision, statistics, issue summaries, and `Quick Estimate / Revision Quote` are visible.
7. If the current document has formed a board session, switch to `3D` and confirm that the board-level view finishes loading.
8. Open `AI`. If no AI service is set up yet, confirm that the setup card points you to `Settings > AI`. If an AI service is already configured, confirm that `Diagnose Selection` and `Chat` are available, and that the advanced AI details card can be opened.
9. Return to `2D`, open the export configuration from the bottom floating toolbar, and confirm one export. If the purchase center opens afterward, export access is not unlocked on this device.
10. Go back to the library and reopen the same document.



Revision Quote

Ready

Quick Estimate

JLPCB Prototype

COMMERCIAL CONTEXT

BASE CURRENCY REFERENCE

CNY 29.21 / CNY 5.84

Reference FX: 1 CNY = 0.1380 USD
(Bundled FX snapshot, Apr 18, 2026 at 8:00).

SERVICE TIER / ETA

Standard / 4-6 business days

QUANTITY

5

Tier 5-9 pcs

DETECTED BOARD INPUTS

LAYERS

2

Confirmed from Runtime copper layers

BOARD SIZE

66.05 x 50.05 mm

Confirmed from Edge cuts/profile geometry

ASSUMPTIONS AND SOURCE

DETECTED SOURCE

Layer Confirmed from Runtime copper layers

Size Confirmed from Edge cuts/profile geometry

ASSUMPTIONS

Display currency USD, Quantity 5, Material FR-4, Thickness 1.60 mm, Surface finish HASL, Service tier Standard, Copper 1.00 oz, Mask Green, Silkscreen White, Panel mode Single, Electrical test Included

PRICE BREAKDOWN

Setup

One-time setup fee

USD 1.93

Board Area

66.05 x 50.05 mm, billable 3305.80 mm²

USD 2.10



Notes

Pricing model: JLPCB Prototype. Detected copper layers from Runtime copper layers. Detected board size from Edge cuts/profile geometry. Current quantity tier: 5-9 pcs. Reference FX: 1 CNY = 0.1380 USD (Bundled FX snapshot, Apr 18, 2026 at 8:00).



Catalog Disclaimer

Estimate only. Final price must be confirmed with the manufacturer's official online quote.

Edit Inputs

Archive Quote JSON + Text



14:43



AI Unavailable

Set up an API key before using AI workspace

No AI provider is configured

AI chat and diagnose actions are unavailable until a provider credential is saved.



Open AI Settings

Diagnose Selection

Clear

Diagnose Selection

Evidence-first di...

Chat

General board-a...

Evidence-first diagnosis will keep typed artifacts and next-step actions in view.

Skeleton ready.

0 runs

0 results

Result

Primary response and typed artifacts

No results yet

Run Diagnose Selection to publish facts, hypotheses, unknowns, and next steps.

Compose

Optional instruction for the current selection

If You Open a KiCad Project or Schematic

In the five-minute route, also confirm:

- Open `Project` and check that project source documents, libraries, and settings are grouped.
- Switch from `Project` or `Sheets` to a schematic sheet, and confirm the breadcrumb and current-sheet content refresh together.
- In schematic Workbench `Search`, switch between `Current Sheet / Whole Project` and search for a symbol, net, or label.
- Click a result and confirm the sheet switches and the target becomes selected; after `Locate / Highlight / Clear`, the canvas and details panel stay in sync.
- Pick a symbol, pin, wire, or label directly on the schematic canvas and confirm `Selection / Details` and canvas highlight update together.

What You Should Be Able to Confirm

- Navigation between the library and the workspace works normally.
- The 2D, object-info-card, and Workbench paths are all usable.
- The `Data` view provides document and status summaries.
- Board-level design documents can enter 3D.
- The `AI` panel shows the expected setup gate when no AI service is configured, or supports `Diagnose Selection / Chat` plus advanced cards when configuration is ready.
- The 2D export path works, or in a locked state correctly redirects to the purchase center.

Things Worth Watching

- During import or first open, the viewer may briefly switch to `Data` to show preparation status. This is normal.
- The first 3D load shows explicit stages and percentages instead of silently hanging.
- Export starts from `2D`; in `3D` or `Data`, return to `2D` before exporting.
- If export access is still locked, you can still view the export configuration. Confirming export opens the purchase center instead of starting an export.

- If you need a support-ready snapshot, **More > Copy Support Bundle** should include issue details, Quick Estimate state, AI activity summary, and recent reproduction clues without including AI credentials.

Common Questions

Why do I sometimes see **Data** first when opening a document?

During import or view preparation, the app may temporarily prioritize the **Data** page and its loading card. Once the first 2D frame is ready, normal browsing resumes.

Do I always need to validate 3D?

No. 3D requires the current document to form a board session with enough board-level geometry. For samples that do not meet that condition, focus on 2D, Data, and export.

Why did tapping export open a purchase page instead?

That is the current expected behavior. When export access has not been unlocked yet, the workspace opens the purchase center first. After unlocking, return to **2D** and export again.

Document Version

v2026.08.31.0

Library and Revisions

Goal

Help you turn loose project files into board workspaces that can be reopened, extended with new revisions, and shared.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users organizing imported boards into a reusable library
- Testers validating document, revision, and share behavior

Prerequisites

- The app is already open on the library home page
- At least one document is available, or project files are ready to import

Steps

1. Review the document list in grid or list mode and open the action menu for a target document.
2. Use the document-level actions to inspect details, append revisions, duplicate, pin, or share as needed.
3. Reopen the document after changes and confirm that revision and details updates are reflected in the library.

Result

- Documents remain organized as reusable project entries, and revisions can be maintained without creating unnecessary top-level duplicates.

What the Library Home Page Can Do

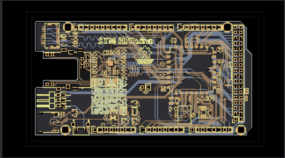
- switch between **grid** and **list** layouts
- import new design or manufacturing documents
- enter batch selection and batch deletion
- view each document's thumbnail, format, last-opened time, and revision count
- rename, duplicate, pin, delete, share, or inspect a single document

For a project that contains both a PCB and schematics, the library thumbnail prefers the PCB canvas. It uses the schematic canvas only when the project has no PCB.

Files

Import File

Import Folder



pcb h730duino

...

KiCad

1 revision(s)

Imported Apr 12, 2026 at 8:00

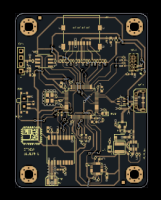
Opened May 10, 2026 at 14:33

Revisions

rev_001

h730duino1

rev_001 Current



pcb BluePhil

...

KiCad

1 revision(s)

Imported Apr 12, 2026 at 8:00

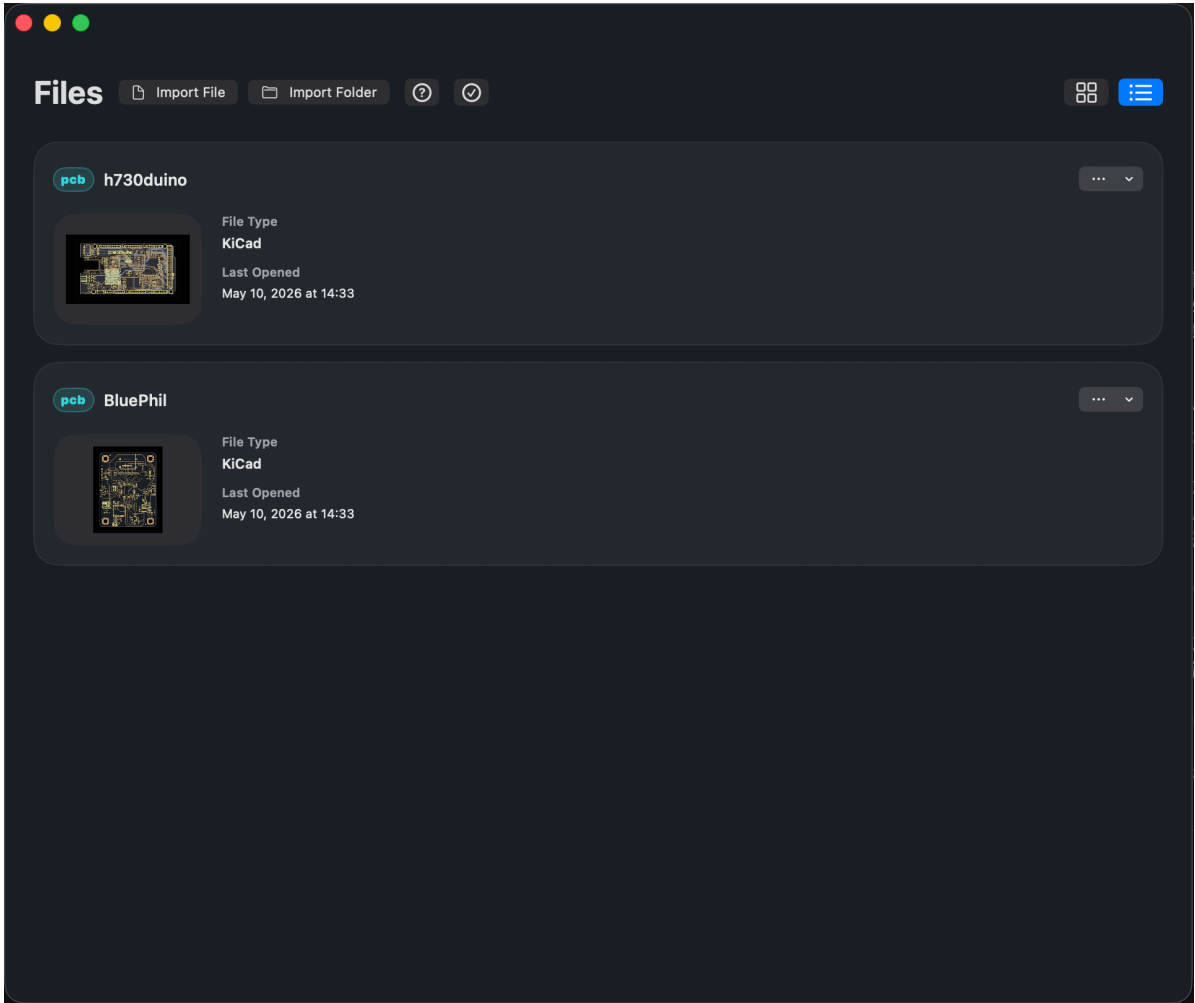
Opened May 10, 2026 at 14:33

Revisions

rev_001

BluePhil1

rev_001 Current



Files

Import File

Import Folder



pcb h730duino



File Type

KiCad

Last Opened

May 10, 2026 at 14:33

pcb BluePhil

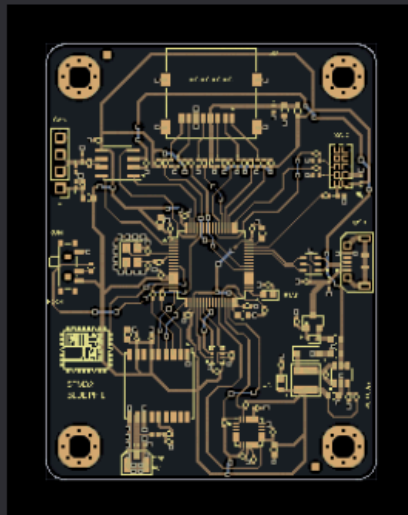


File Type

KiCad

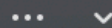
Last Opened

May 10, 2026 at 14:33



pcb

BluePhil



KiCad

1 revision(s)

Imported Apr 12, 2026 at 8:00

Opened May 10, 2026 at 14:33

Revisions

BluePhil1

rev_001 **Current**

+ Add Design Revision

Y Import SES

/ Rename

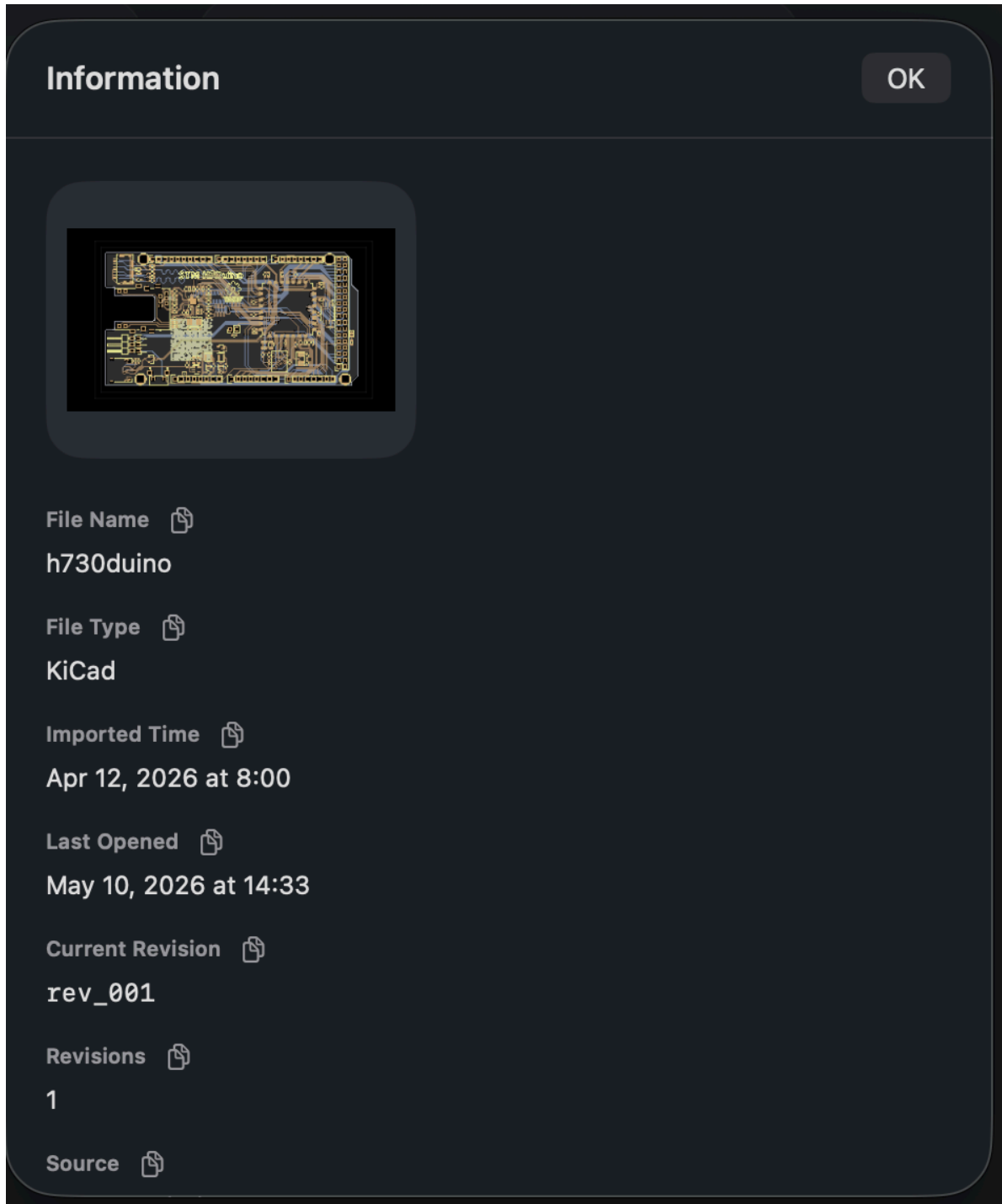
📄 Duplicate

📘 Information

🔗 Share

📌 Pin

🗑 Remove



Relationship Between Documents and Revisions

- **Document:** one project entry in the library
- **Revision:** a historical version under that document

- **Current revision:** the revision currently opened, displayed, and exported

When you append a revision, the app does not create a separate top-level card. Instead, the card's `revisionCount` increases, and you can switch the current revision within the document.

Source Trees for KiCad Project Documents

When a document comes from `KiCad .kicad_pro`, the library stores more than a single entry file: it keeps the project project files associated with that revision. After opening it, the workspace `Project` page groups content into:

- `Source Documents`: PCB files, schematic sheets, and other openable design documents.
- `Libraries`: symbol, footprint, and other library files shipped with or referenced by the project.
- `Settings`: `.kicad_pro` and project configuration files.

This keeps the PCB and schematics of one project inside the same library document and revision context. When appending revisions, prefer importing the complete project or related project files so later sheets, libraries, or settings are not missing.

Key Actions in the Document Menu

From each document card menu, you can:

- **append a design revision**
- **append a manufacturing revision**
- **rename**
- **duplicate**
- **view information**
- **share**
- **pin / unpin**
- **delete**

What the Information Sheet Shows

When you click `Information`, you can inspect:

- document name

- file type
- import time
- last-opened time
- current revision ID
- total revision count
- full revision list

All of this is copyable for testing notes and issue reproduction.

Recommendations

- For ongoing evolution of the same project, prefer **append revision**.
- If you want to preserve an experimental branch, use **duplicate** to create a separate document.
- Pin important documents to reduce the risk of accidental deletion or mis-selection.
- After import completes, open the current revision once to confirm the content, thumbnail, and last-opened time were updated correctly.

Common Questions

Why does one document contain multiple revisions?

Because the current library model is designed around continuous project evolution. Multiple revisions let you track change history within a single project context.

If I open another document while import is still running, will the newly imported document steal focus later?

No. Import continues and completes, but it does not force the UI back to the newly imported document.

Does sharing send the whole document package?

The library share action prepares a share item from the current document's shareable root path. For grouped or directory-based sources, the shared result is based on that source root rather than a text summary alone.

Basic PCB editing

Page goal

Use a limited, undoable set of KiCad PCB editing operations in the PCBAtlas 2D workspace and save the result back to the current library revision.

Supported platforms

- **iOS / iPadOS:** 17.0 or later.
- **macOS:** 14 or later.
- The first editing release is available only when the board actually opened by the workspace is a KiCad `.kicad_pcb` source.

Intended audience

- Reviewers who need to move a via quickly
- Users who need to Slide an existing track segment, adjust its width, or remove an entire track or via
- Testers validating the editing state machine, Undo, and save flow against the current product boundary

Prerequisites

1. Prepare a disposable KiCad `.kicad_pcb` test board; keep an original copy of important projects.
2. Open the board from the library and remain in `2D`.
3. If a target cannot be selected, check `Workbench Visibility` and `Selection Filter` to see whether placements, tracks, or vias are hidden or filtered.

Steps

1. Enter the editing session

1. Wait for the 2D Viewer to finish its first preparation. For an editable KiCad PCB, PCBAtlas automatically prepares a full editing session.
2. Select a track segment or via on the canvas.
3. Choose the currently available `Slide`, `Move Via`, `Delete Track`, or `Delete Via` action from the floating toolbar below the canvas.

The Viewer remains read-only during preparation. If the source is not a KiCad PCB, a full editing session cannot be created, or the board is stale, the bottom of the canvas reports a clear reason rather than continuing with an approximate editing mode.

2. Select an object and review its actions

After the editing session starts, select a target in the 2D Viewer. Both wide and compact layouts use only the floating action bar at the bottom of the Viewer; Workbench no longer hosts an editing panel.

Selected object	Available actions	Important behavior
Placement	Read-only; no editing action	Its information remains available, but basic editing cannot change a Placement
Non-fixed via	<code>Move Via</code> , <code>Delete Via</code>	Moving a via does not rewrite adjacent tracks
Supported track segment	<code>Slide</code> , <code>Track Width</code> , <code>Delete Track</code>	Slide applies to the exact segment; deletion applies to the whole Track

Fixed tracks or vias, inexact targets, and unsupported track geometry disable affected actions and show a reason. Placements are currently always read-only and do not provide disabled Move or Rotate buttons.

3. Move a via

1. Select a non-fixed via.
2. Select **Move Via**.
3. Start dragging on the target; dragging from empty space still pans the camera.
4. Review the ghost, original position, ΔX / ΔY , and grid step.
5. Release at the target position to create a pending edit preview.

After release, the ghost and confirmation bar remain. Select ☒ to accept the edit or ☐ to discard it. If committing fails, the app displays an error and the board never retains a partial edit.

4. Slide an existing track

1. Precisely select a supported track segment.
2. Select **Slide**. The canvas shows a blue centerline preview and bottom ☒ / ☐ controls.
3. Start dragging the blue line. The preview track and its actual width update with the blue line in real time.
4. The Slide state stays active after release, so you can drag the same blue line again to refine it.
5. Select ☒ to accept the current valid preview or ☐ to leave Slide and discard this edit.

If you drag the segment beyond its geometric limit, PCBAtlas retains the last valid preview and displays the reason without leaving Slide. Drag the blue line back into a valid range to continue editing.

You cannot add or remove nodes, split or merge a Track, or change its layer, net, or rule class here.

5. Change track width

1. Select a supported track.
2. Select **Track Width**.
3. Enter a millimeter value greater than zero.
4. Select **Apply** to create a preview, then use bottom ☒ to accept or ☐ to discard it.

Invalid numbers, non-positive widths, and values outside the valid range are rejected.

6. Delete a track or via

1. Select any segment of a track, or select a via.

2. Confirm that the entire Track is highlighted or that the correct via is selected.
3. Select `Delete Track` or `Delete Via`, then use bottom `✓` to accept or `×` to discard it.

Deletion does not use an additional modal confirmation. Related teardrops are removed atomically when they can be resolved safely. For an accidental deletion, use `Undo` in the floating toolbar at the bottom of the Viewer.

7. Undo and save

- `Undo` in the bottom floating toolbar is available only after the current command ends and returns to the ready state. Completion messages no longer show a separate Undo button.
- Each accepted via move, Slide, track-width change, or cascading deletion creates one history entry.
- After each edit or `Undo` commits, the app appends its recovery record to the internal `EditingDraftsV4` working copy. If saving still fails after automatic retry, the committed canvas result remains, but further editing is locked and `Retry Saving Draft` appears. Selecting it retries only saving the pending record; it does not perform the edit again. The ready state returns only after saving succeeds.
- A `*` before the title means the current session has unsaved edits.
- Selecting `Save` in the bottom floating toolbar, or pressing `⌘S` on macOS or an iPad with a hardware keyboard, writes the result in place to the currently open library revision. It does not create a revision or change the current canvas or camera.
- PCB-edit saves retain the existing Case Mini Viewer thumbnail. Explicitly select **Regenerate Thumbnail** later if you want the thumbnail to reflect the updated board.
- If unsaved edits exist when you return to the library from the upper-left control, choose one of:
 - `Save` : save to the current revision, then return to the library;
 - `Discard Unsaved Edits` : restore the latest saved checkpoint, then leave;
 - `Continue Editing` : return to the current editing session.
- Importing a same-name file or explicitly selecting `Add Revision` still creates a new revision. Ordinary editing saves do not change the revision count.

Acceptance checklist

- ✓ A non-KiCad PCB reports the explicit KiCad-only boundary when an editing action is requested.

- ✓ After first Viewer preparation, a KiCad PCB automatically creates a full editing session and shows available actions in the bottom floating toolbar.
- ✓ Selecting a Placement shows its information but no Move or Rotate action.
- ✓ A non-fixed via can be moved and deleted, and an accepted edit can be undone from the bottom floating toolbar.
- ✓ A Slide retains the blue line and preview after release and can continue dragging.
- ✓ A Slide outside its limit retains the last valid preview and can continue after returning to a valid range.
- ✓ A supported track can Slide and change to a positive width.
- ✓ Starting deletion from one segment removes the entire Track and can be undone.
- ✓ Fixed or unsupported targets remain unchanged and show a specific reason.
- ✓ `Save` updates the current revision in place; the revision count remains unchanged and the saved result is visible after reopening.
- ✓ When unsaved edits exist, `%S` and bottom `Save` have the same result.
- ✓ The current canvas and camera position remain unchanged before and after saving.
- ✓ Returning to the library with unsaved edits offers the three outcomes described above.

Current boundaries

- Slide changes only the existing topology constraints for the supported current segment. New routing, interactive routing, shove / walk-around, and autorouting are not supported.
- Moving, rotating, creating, or deleting placements is not supported. Neither are footprint replacement, side flipping, or multi-object transforms.
- No Zone, Pad, board outline, text, graphic, or arbitrary shape editing.
- A single Track segment cannot be deleted; `Delete Track` always removes the complete track.
- Editing can introduce ratsnest lines or DRC issues. PCBAtlas keeps an explicitly committed result instead of silently reverting it.
- Saving updates the currently open library revision. Only same-name file import or an explicit `Add Revision` creates a new revision.

FAQ

Why is no editing action shown in the floating toolbar, or why does entering one fail?

Confirm that a PCB 2D view is open, its actual board source is `.kicad_pcb`, and the app can create a full editing session. Manufacturing packages, DSN, SES, ODB++, IPC-2581, and schematics are outside the first editing release. The floating toolbar displays only actions available for the current selected object.

Why are there no editing buttons for the selected object?

Placements are currently inspectable but not editable, so selecting one does not show a Move or Rotate button. Other objects may also have no available action because they are fixed, use unsupported track geometry, or became stale after a scene refresh. Select the object again and read the blocking reason at the bottom of the canvas.

Why does the blue Slide line stop moving at one point?

That position exceeds the current geometric constraints. The UI retains the last valid preview instead of leaving Slide; drag from the retained blue line back into a valid range to continue.

Result

The current KiCad PCB revision contains a saved result that can be reopened, and every in-session change is an undoable atomic transaction.

Last verified

2026-08-22

Compare two board revisions (Board Diff)

Page goal

Start Board Diff from managed revisions, understand the `Before` / `After` direction, inspect object-level changes, and export reviewable JSON or PNG output.

Supported platforms

- **macOS 14 or later:** the primary wide-screen Board Diff experience and first-round acceptance platform.
- **iOS / iPadOS 17.0 or later:** the entry is included in the app; landscape or a wide window is recommended while dedicated narrow-screen refinement continues.

Intended audience

- PCB design, manufacturing, and review users comparing two revisions of the same board
- Testers validating directional changes, layer pairs, objects, and risk hints

Prerequisites

Choose two boards with different content that resolve to the same actual format:

Input	Current status
DSN	Supported
Standalone KiCad <code>.kicad_pcb</code>	Supported
Gerber + Excellon folder or archive	Supported

SES	Supported only when both sides identify the same verified base design
ODB++, IPC-2581, complete KiCad Project	Explicitly unsupported for Board Diff

`Before` and `After` are directional:

- `Added` : present only in `After` .
- `Removed` : present only in `Before` .
- `Modified` : a paired object whose properties or geometry differ.
- `Unchanged` : identical on both sides and shown only as context.

Matching extensions do not prove matching formats. PCBAtlas resolves both sides independently before allowing a comparison.

Steps

Compare two managed revisions

1. Find a library document containing at least two revisions.
2. Open the document card's more-actions menu.
3. Select `Compare Revisions...`.
4. Choose two different same-format revisions in `Before` and `After` .
5. Select `Compare` .

When the current revision has a parent, the page prefers the parent for `Before` and the current revision for `After` . You can replace either selection.

A completed managed comparison stores a verified artifact. Return to the same document's comparison page and use `Reopen Saved Comparison` to open it again.

Confirm ambiguous layer pairing

If PcbCore finds multiple compatible candidates, the comparison pauses at `Layer pairing needs confirmation` :

1. Review the name, role, order, and object count of each `Before` layer.
2. Choose the correct `After` layer .
3. Make a selection for every ambiguous item.
4. Select `Confirm and Compare` .

PCBATlas does not guess when pairing is ambiguous and does not force the boards together with automatic translation or rotation. A whole-board translation or rotation remains a real visible change.

Wait for or cancel a comparison

Progress covers validation, Before / After import, layer pairing, snapshots, geometry matching, risk analysis, the exact Diff scene, and persistence.

After selecting `Cancel` , wait for `Cancelling comparison...` to finish. A safe retry is available only after both native sessions and temporary staging data have been released.

Read the comparison workspace

Display modes

- `Overlay` : combines exact changes in one scene; `Context` controls unchanged context intensity.
- `Split` : shows `Before` on the left and `After` on the right; `Sync Camera` is enabled by default and can be turned off for independent inspection.
- `Blink` : alternates Before and After with the same camera to reveal small movement.
- `Fit` : fits the current scene back into the visible area.

Layers and changes

- `Layer Pairs` lists paired layers and the change count for each pair.
- Select `All layers` for the whole board or one pair to narrow the view.
- In `Changes` , toggle `Added` / `Removed` / `Modified` and filter by object type or Net.
- Selecting a change row locates and highlights it; selecting an object in the Viewer also looks up its change.
- `Previous` / `Next` walk the current filtered results. The macOS shortcuts are `⌘[` and `⌘]` .

- `Risks` filters directional hints by severity. Risk hints help review and do not replace DRC or manufacturing approval.

Swap, export, and reopen

- `Swap` creates and recomputes a reverse request. Added / Removed, statistics, risks, and details change direction.
- `JSON` exports the stable, versioned semantic result; iOS hands the file to other apps.
- `PNG` exports the current Diff view; iOS saves it directly to Photos, while macOS continues through the system file-delivery interface.
- A managed revision comparison can be reopened through `Reopen Saved Comparison`.
- If revision content, format, policy, or layer pairing changes, the old artifact becomes invalid instead of being displayed silently.

Acceptance checklist

- ✓ `Compare Revisions...` is disabled for a document with only one revision.
- ✓ Two same-format managed revisions run and reopen through `Reopen Saved Comparison`.
- ✓ Identical inputs, different formats, unsupported formats, and mismatched SES bases block `Compare` with an actionable reason.
- ✓ Ambiguous layers require manual pairing before comparison continues.
- ✓ The camera position remains consistent across `Overlay / Split / Blink`.
- ✓ `Split Sync Camera` works in both states, and `Fit` restores the whole-board view.
- ✓ Added / Removed / Modified, object type, Net, and risk filters narrow the results.
- ✓ List selection, Viewer pick, and `Previous / Next` locate the same changes in both directions.
- ✓ `Swap` reverses direction, colors, statistics, details, and risks correctly.
- ✓ Cancelling a large comparison waits for cleanup and then permits another comparison.
- ✓ JSON and PNG both export, and a managed artifact is not reused after its inputs change.

Current boundaries

- Inputs must have the same format; cross-format comparison is not supported.
- ODB++, IPC-2581, complete KiCad Project, schematic Diff, and 3D Diff are not supported.

- Board Diff does not merge, accept, or write changes back to source files.
- There is no automatic alignment; whole-board translation and rotation remain visible differences.
- Image pixels, bounding boxes, and file text are not used as a fallback for exact geometry Diff.

FAQ

Why is `Compare` disabled?

Check that both sides are selected, contain different content, and resolve to the same supported format. For Gerber / Excellon, provide the complete package folder or archive on each side rather than two arbitrary individual layer files.

Why do matching filename extensions still produce a format mismatch?

The format comes from actual resolution, not just the filename. Directories, archives, SES base designs, and the resolved board entry are part of validation.

Why can I not retry immediately after cancelling?

The page waits for PcbCore to release both Before / After sessions and temporary data so a new task cannot overlap the old one. Retry after `Cancelling comparison...` disappears.

Why did the change count or risks change after Swap?

Added / Removed and risk hints are directional. Swap recomputes with reversed Before / After; it does not merely exchange labels.

Result

You can locate exact bilateral changes, review directional risks, and export JSON / PNG in one workspace. Managed revision comparisons can also be stored, verified, and reopened.

Last verified

2026-07-30

Document Version

v2026.08.31.0

2D Viewer

Goal

Help you handle the most important day-to-day work in 2D: inspect, select, locate, verify, and prepare exports.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users doing day-to-day board inspection in the workspace
- Testers validating 2D interaction, object info, and export readiness

Prerequisites

- A board document is already open in the workspace
- The viewer has finished its initial preparation and published the first 2D frame

Steps

1. Use 2D to pan, zoom, and select objects in the current board view.
2. Select an object, then open **Info** from the floating toolbar below the canvas to review the object details you need.
3. Use the bottom floating toolbar to switch among **2D** / **3D** / **Data** and to open export configuration in **2D**.

4. Adjust visibility or filtering through Workbench, then return to 2D for final confirmation and export preparation.

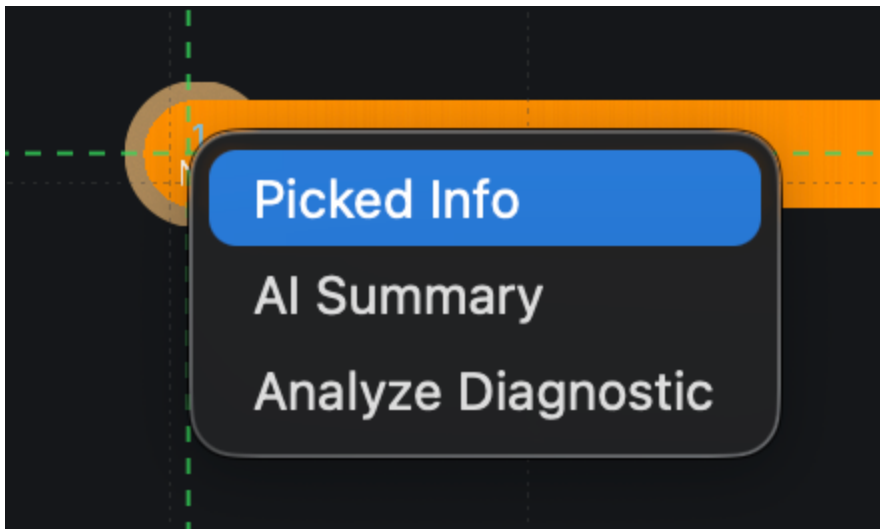
Result

- 2D serves as the primary working view for precise inspection, object lookup, and export preparation.

2D Is the Main Working View in the Current Version

Inside the workspace, 2D handles most of the high-frequency work:

- pan, zoom, box-select, and point-select
- open **Info** from the floating toolbar below the canvas
- work with Workbench for visibility, filtering, and object targeting
- review the current selection, coordinates, or display status in the status bar
- switch among 2D / 3D / Data and launch every export flow from the bottom floating toolbar



Pad

 Copy



P5 · F.Cu · /I2C5_SCL

Identity

Type: Pad

Category: Pad

Object ID: pad/P5/P5/2

Component: P5

Place: P5

Footprint:

Socket_Arduino_Mega:Socket_Strip_Arduino_1x10

Pad: 2

Net: /I2C5_SCL

Layer: F.Cu

Origin / Placement

Place Side: front

Place Origin X: 122.174

Place Origin Y: -70.866

Place Rotation: 0.000

Pin Local X: 2.540

Pin Local Y: 0.000

Pin Rotation: 0.000

Topology

Granularity: Object

Span Layers:

B.Cu;B.Mask;F.Cu;F.Mask;F.SilkS;In1.Cu;In2.Cu

Scene info

 Copy



Info

Camera

```
PCBAtlas 2D Location Info
sourcePath: /Users/ethan/Library/Containers/
com.henrykang.pcbatlas.uitests.xctrunner/Data/
tmp/PCBAtlas-UITest-89E071CB-7825-407B-
B4A6-091D2B686849/ui-smoke-board.dsn
sceneRevision: 1
sceneMode: compact
objects: compact layers=1, objects=3, strokes=5,
padVias=1, texts=0, tiles=8
parseSummary: layers=1, nets=1, tracks=1, vias=0,
pads=1
camera.center: (2.77575, 2.73311)
camera.zoom: 4.39998
viewportSize: (779, 813)
pointer.world: (2.50142, 2.53614)
pointer.worldScreenRatio: 0.002795
pickTolerance.pixels: 10
pickTolerance.world: 0.027954
visibleWorldBox.min: (0.66776, 0.53312)
visibleWorldBox.max: (4.88373, 4.9331)
visibleWorldBox.size: (4.21597, 4.39998)
bounds.min: (0, 0)
bounds.max: (25.4, 25.4)
bounds.size: (25.4, 25.4)
```

1

Scene info



Info

Camera

Ready to apply shared camera.

center: (2.77575, 2.73311)

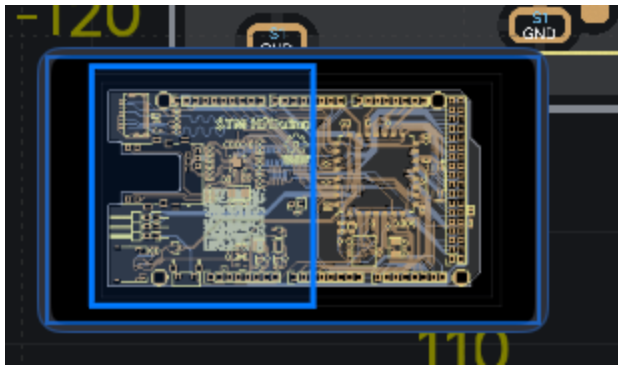
zoom: 4.39998



Paste



Apply



14:43



BluePhil.kicad_pcb

Pad(U200) · F.Cu

X: 108.2962, Y: -79.7999

Pad



U200 · F.Cu · /Microcontroller/MCU_HSE_OUT

Identity

Type: Pad

Category: Pad

Object ID: pad/U200/U200/6

Component: U200

Place: U200

Footprint:

Package_QFP:LQFP-64_10x10mm_P0.5mm

Pad: 6

Net: /Microcontroller/MCU_HSE_OUT

Layer: F.Cu

Origin / Placement

Place Side: front

Place Origin X: 114.000

Place Origin Y: -80.500

Place Rotation: 0.000

Pin Local X: -5.675

-60

CAN

1
GND

-80

RUN

BOOT

-100

STM32

BLUE-PHIL

STAT

VBUS

VBUS

+5V

AS+

+3V3

EXN+

V0.1 3/3/21

Supporting Capabilities You Will See

- **Object info card:** after selecting a PCB or schematic object, open it with `Info` in the bottom floating toolbar. PCB objects that support local footprint preview can switch between `Info` / `3D` in the card.
- **Mini viewer:** shown at the upper left of the canvas by default, compares the current viewport with the global board range, and temporarily makes room when the object info card opens
- **Crosshair guideline:** when enabled in settings, follows the pointer
- **Status bar:** the left side summarizes the current selection; the right side defaults to resource status and can switch to coordinates, frame metrics, or off
- **DRC markers:** after the board finishes first-frame preparation, DRC runs in the background. When complete, locatable results appear as red circles, crosses, and optional evidence connector lines. Workbench's 2D Display Grid controls their visibility globally or by layer.

What the Bottom Floating Toolbar Can Do

The persistent floating toolbar below the canvas concentrates viewing and editing actions:

- shows the available `2D` / `3D` / `Data` modes and highlights the current mode
- shows an export entry in `2D`
- shows `Info` for an exact selected object in a PCB or schematic, opening the matching in-canvas object-details card
- shows `Save` and `Undo` when an editing session is available
- after selecting a supported via or track segment, shows the applicable move, Slide, track-width, or delete actions

Buttons that cannot run in the current context are hidden. Available actions update when the selection, view, or editing state changes.

The map collapse menu on the left side of the toolbar hosts global viewer actions:

- `Fit All`
- toggle axis visibility
- regenerate thumbnail
- `Scene info`

`Scene info` opens as a centered card. The `Info` tab shows and copies the current scene, import source, camera, and viewport data. The `Camera` tab can paste a teammate's shared `Scene info` or `PCBAtlas 2D Camera` text, parse `camera.center` and `camera.zoom`, and apply them to the current 2D view. It does not validate that source paths match, which keeps it useful for manually sharing screen positions on the same device model and the same case.

Inside the mini viewer, pressing and dragging the current viewport pans the viewport. Pressing outside the viewport and dragging draws a new viewing range. The main 2D canvas right-click / long-press menu still keeps only context actions for the current object and chat capability; it no longer hosts `Info` or the mini viewer's global viewer actions.

When a KiCad project opens both a PCB and schematic, 2D Viewer shows dual canvases. It splits vertically when the available canvas aspect ratio is below `1.5:1`, horizontally when it is above `1.5:1`, and retains the current split orientation between those thresholds so resizing near the boundary does not make it flip repeatedly.

How Schematics Behave in 2D

Schematic documents are also viewed through the `2D` canvas, but their interaction model is different from PCB review:

- Pick hits come from the schematic document data, not visual guessing.
- Current pick coverage includes `sheet / symbol / pin / wire / bus / label / text / junction / marker`.
- A single candidate selects and highlights immediately. Non-net objects also show a high-contrast locate frame, and point-like pins retain a clear minimum visible extent. Multiple candidates let you cycle overlapping objects.
- Clicking empty space clears schematic selection and highlight.
- Schematic object details synchronize to Workbench `Selection / Details` and can also be opened in the schematic canvas through toolbar `Info`. PCB objects continue to use the equivalent card in their PCB canvas.

Schematic review supports sheet navigation, object search, details, locate, highlight, and canvas pick. Layout edits, ERC fixes, and property writeback stay in dedicated EDA tools. When a schematic opens for the first time, it fits the full paper range rather than mistaking the paper's lower-left `(0, 0)` for the center of the view.

PCB-to-Schematic Cross-Probing

KiCad projects that contain both a PCB and schematics use exact project-object relationships for cross-probing:

- Selecting a schematic symbol or pin in dual canvases locates and highlights the corresponding footprint or pad in the PCB canvas. Selecting a PCB footprint or pad does the reverse for the corresponding symbol or pin.
- After a PCB-to-schematic symbol or pin jump succeeds, the target locate frame expands twice and then stays static. Ordinary schematic picks do not replay the animation, and a system **Reduce Motion** setting shows only a static locate frame.
- Selecting a net, track, via, copper graphic, or schematic net object highlights the exact related net in the other canvas instead of arbitrarily selecting one object on a sheet.
- The other canvas has only two states: **exact associated object highlighted** or **not selected**. Selecting empty space, or selecting an object with no jump relationship, clears prior cross-canvas selection and highlighting instead of dimming the entire schematic.
- Single-canvas mode keeps the current local selection and offers **View related design object** from the floating toolbar, Workbench, or the `Board` command. After a successful jump, a return button restores the original canvas, camera, and selection.
- Selecting empty space, changing projects, or closing dual canvases clears an unfinished cross-probe request.

Cross-probing accepts only unique, verifiable object relationships; it does not guess from coordinates, displayed text, or similar names. BOM-only components, symbols without PCB footprints, board footprints without an associated schematic, and missing or ambiguous relationships do not create a jump. The UI keeps the local selection and reports that the associated target is unavailable.

Division of Responsibility Across Views

- **2D**: precise inspection, object info review, and export preparation
- **3D**: overall structure review for board sessions, with board-level geometry and renderable imported 3D models
- **Data**: document, revision, statistics, status, and issue summaries

How Export Relates to 2D

Export starts from `2D` :

- the export button is unavailable in `3D` and `Data`
- `PNG` / `PDF` are capture exports
- board-file export is split into two paths:
 - **Original source export:** returns the original `DSN` / `KiCad` / `IPC-2581` / `ODB++` / `Gerber-Drill` data stored for the current revision
 - **Converted export:** currently keeps `DSN` and `KiCad` 8

Recommendations

- Narrow the visible scope with Workbench first, then use 2D for local confirmation.
- When checking design-rule issues, use the `DRC` column in Workbench to narrow marker scope by layer. Hiding markers does not change the board or DRC result.
- When you need to record an issue, select the target in 2D first and then copy its details from the info card.
- When you need a support-ready snapshot, use `More > Copy Support Bundle`. It summarizes the current document, `Board Problems`, Quick Estimate state, AI activity summary, and recent reproduction clues without including AI credentials or full design file contents.
- Before exporting, return to 2D and confirm that the visible layers, visibility state, and framing are correct.

Common Questions

Why can't I interact with 2D immediately after opening a document?

During import or render preparation, the app shows the Viewer progress page. A project presents interactive content directly after its canvases are ready.

Why isn't object detail shown in Workbench anymore?

The current design moves object detail into the floating info card in 2D. Workbench now focuses on visibility, filtering, and object browsing.

Why is the export button unavailable?

Because export starts from 2D . If you are in 3D or Data , return to 2D ; if another export task is running, wait for it to finish.

Document Version

v2026.08.31.0

3D Viewer

Goal

Help you understand the valid scope, supported operations, and current limits of the 3D view.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users reviewing the overall 3D structure of boards
- Users checking board shape, main surfaces, pads, vias, silkscreen, imported 3D models, and spatial relationships

Prerequisites

- A board document is already open in the workspace
- The current task really requires 3D rather than 2D-only inspection

Steps

1. Switch to 3D from the workspace and wait for the staged view build to finish.
2. Use the 3D camera actions and, if needed, adjust the 3D options in Workbench.
3. If you find a rendering or asset issue, copy View Summary before returning to 2D for deeper inspection.

Result

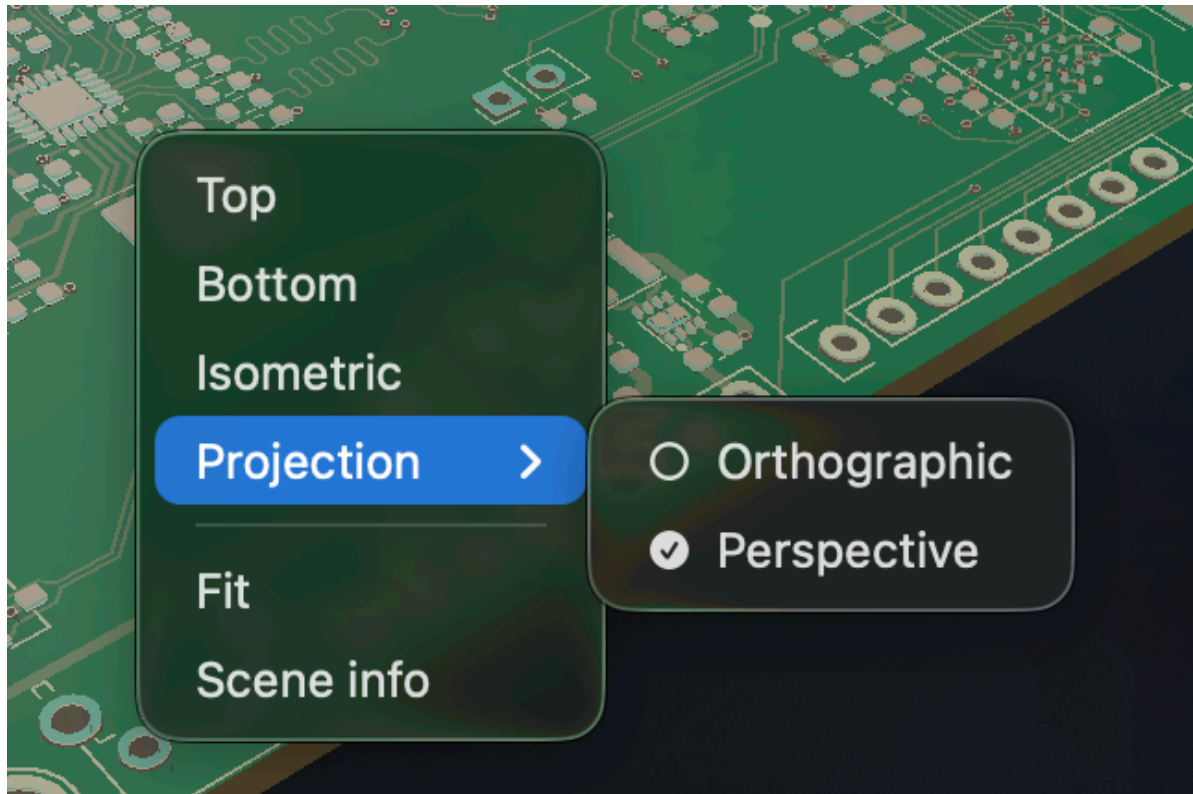
- You can confirm whether the board's 3D view loads, or whether a clear issue state is shown instead.

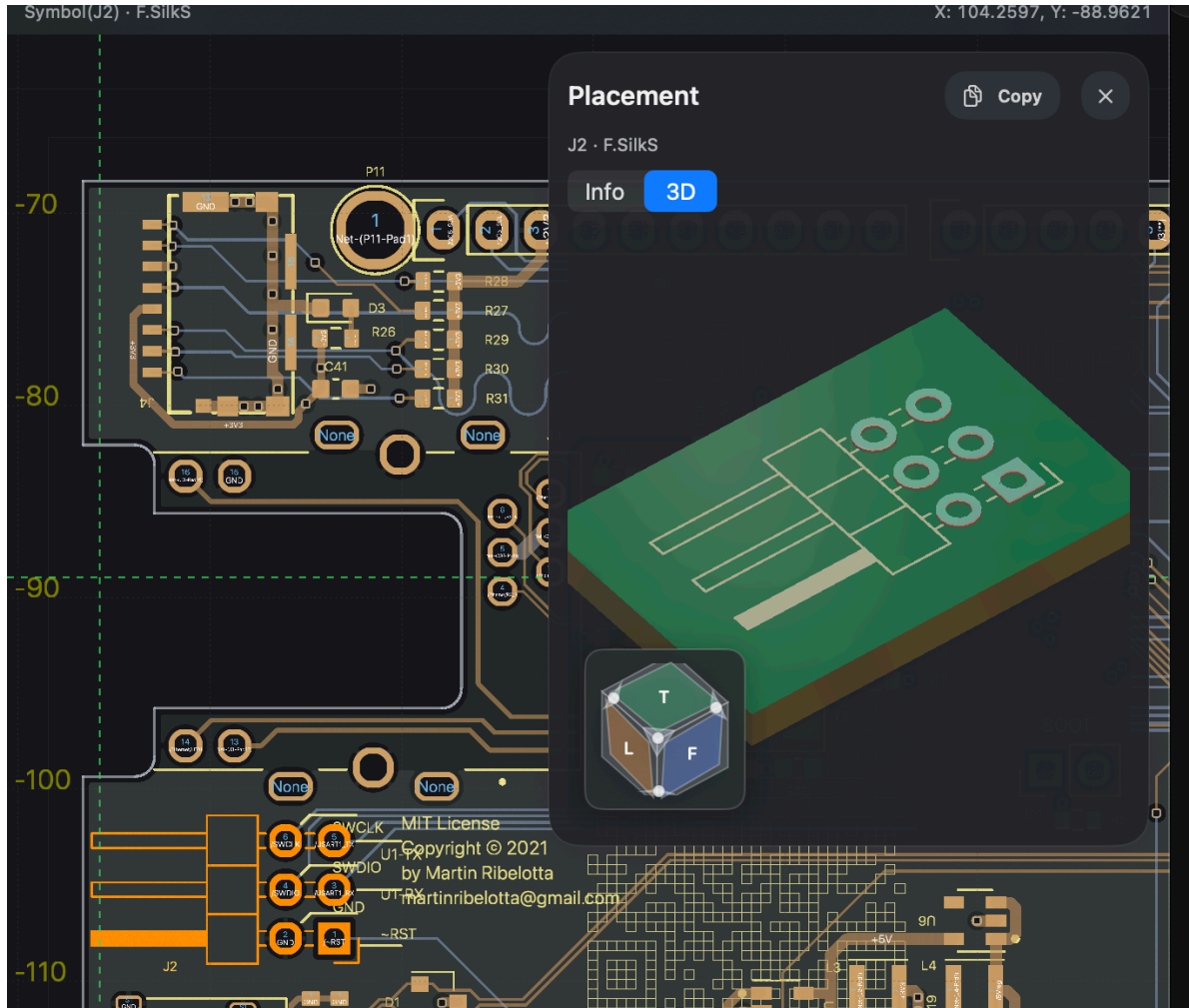
Current 3D Scope

`3D Viewer` is intended for board-level design data that contains enough information for a spatial view. If the currently open document does not provide enough 3D information, the app shows a clear issue message instead of an empty view.

After `Show 3D models` is enabled, renderable imported 3D models are preferred. Only when a model file is missing, its format is unsupported, its path cannot be resolved, or rendering fails will the app show a fallback/proxy body or issue so you can continue spatial review while seeing where the model problem is.







What Happens After Entering 3D

1. The app starts building the 3D view.
2. You see staged loading feedback such as Loading / Preparing / Drawing .
3. When the view is ready, the app automatically fits a suitable camera view to the board bounds.

Current Supported Operations

- switch to Top / Bottom / Isometric
- Fit
- copy View Summary

These actions are available from the context menu and directly update the current 3D camera state.

Workbench Integration

When you are in **3D**, the **Visibility** area in Workbench adds an extra 3D section for board-surface and auxiliary display settings, including **Show 3D models**. Renderable models are preferred; models that cannot be rendered are exposed through fallback/proxy bodies or issues so they can still be reviewed and recorded.

Recommendations

- Use 3D for overall orientation, then return to 2D for precise review and export.
- If you notice issue counts in 3D, copy **View Summary** first so you have an issue snapshot to share.
- 3D is good for shape, layer, and spatial understanding; it is not the final export view in the current version.

Common Questions

Why do some files not have a 3D view?

3D needs enough board-level information to build a spatial view. If a file only contains partial manufacturing data or lacks the necessary geometry, the app shows a clear message.

Why do some components show proxy bodies or issues?

The current 3D view prefers renderable imported 3D models. Only when a model file is missing, its format is unsupported, its path cannot be resolved, or rendering fails will the app show a fallback/proxy body or issue.

Can I export directly from 3D?

No. Export does not start from 3D in the current version. Return to **2D** first.

Document Version

v2026.08.31.0

Workbench

Goal

Help you use one side-panel area for visibility control, selection filtering, object browsing, and local focusing.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users who need visibility control, filtering, and object targeting
- Testers validating Workbench behavior across 2D and board-level 3D paths

Prerequisites

- A board document is already open in the workspace
- Workbench is visible and the status view is available

Steps

1. Start with `Visibility` to narrow the visible scope of the board.
2. Use `Selection Filter` to restrict what can be picked.
3. Use `Objects` to search, select, and apply actions such as `Locate`, `Highlight`, `Isolate`, or `Clear`.

Result

- Workbench helps you reduce visual noise, find the right target quickly, and drive focus actions without taking over object-detail display.

Current Workbench Structure

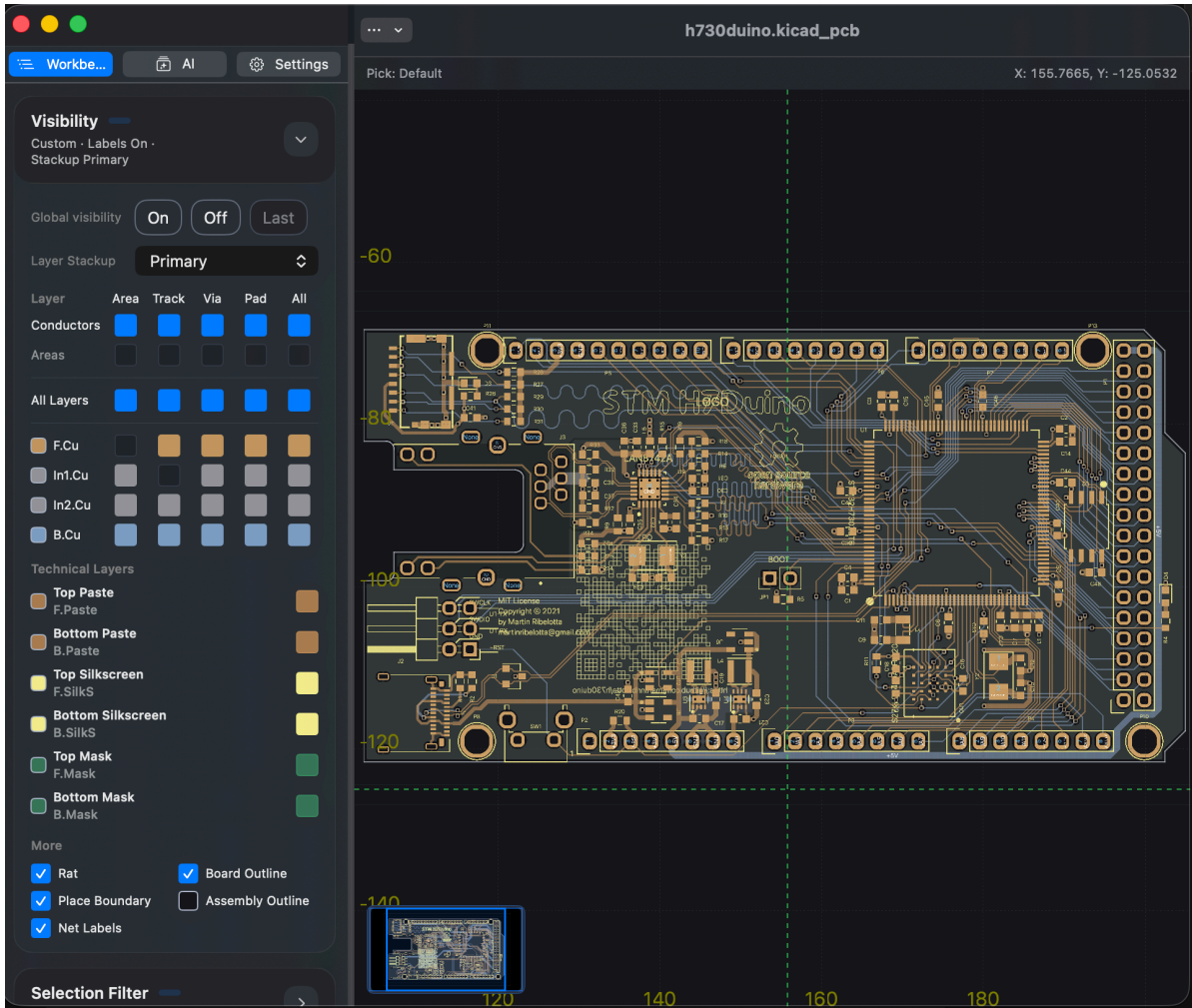
By default, Workbench contains three main sections:

1. `Visibility`
2. `Selection Filter`
3. `Objects`

Their default states are:

- `Visibility` expanded
- `Selection Filter` collapsed
- `Objects` expanded

Workbench now routes by status type: PCB documents show `Visibility / Selection Filter / Objects / Board Problems`, while schematic documents show `Sheets / Search / Selection / Details / Problems`. The PCB Workbench is described first, followed by the schematic Workbench.



Visibility

Custom · Labels On ·
Stackup Primary



Global visibility

On

Off

Last

Layer Stackup

Primary



Layer

Area

Track

Via

Pad

All

Conductors



Areas



All Layers



 F.Cu



 In1.Cu



 In2.Cu



 B.Cu



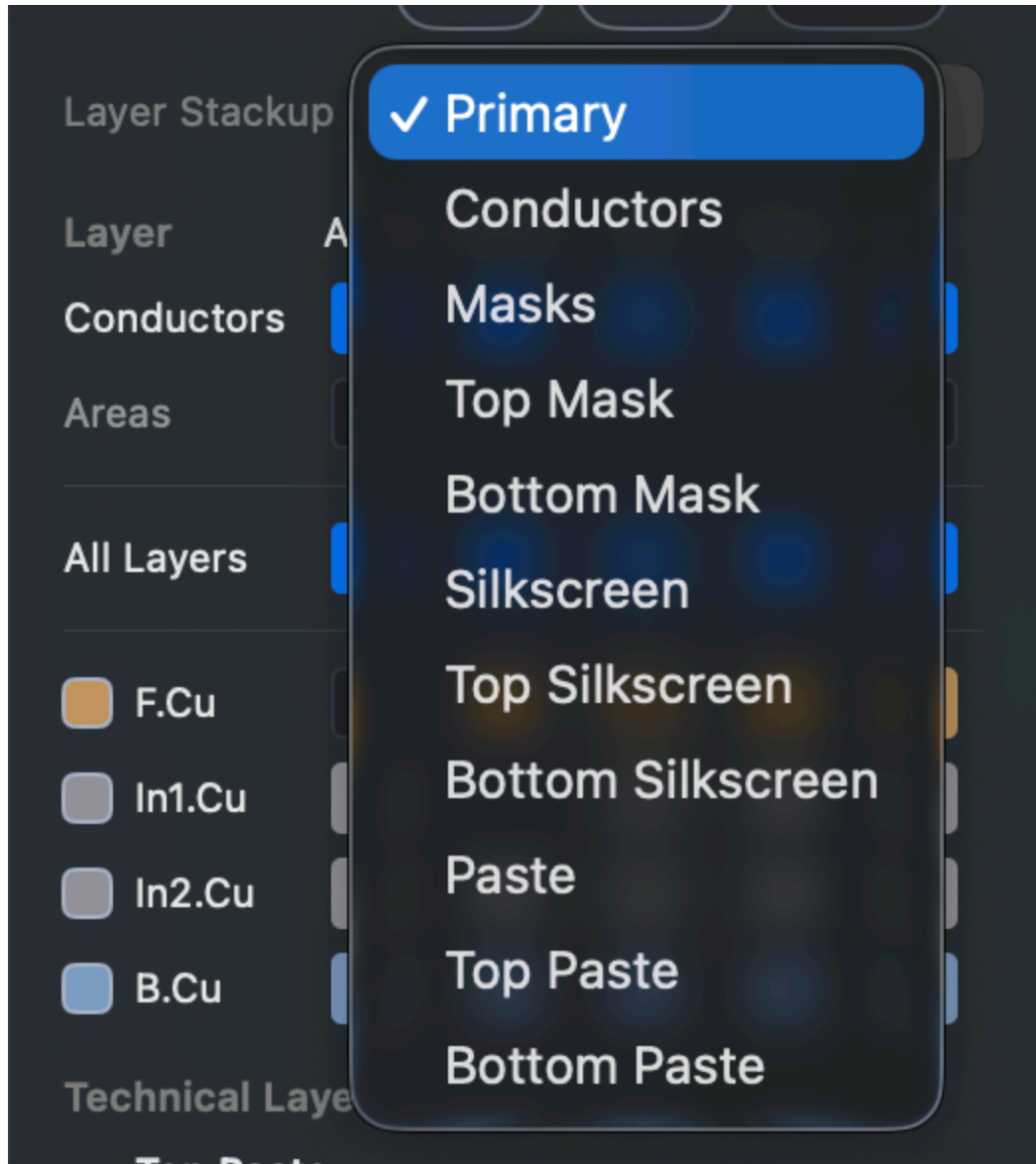
Technical Layers

 **Top Paste**
F.Paste



 **Bottom Paste**
B.Paste





Selection Filter

Pad, Via, Track



Default 

All

None

Default

Routed Copper



Track



Segment



Other Objects



Pad



Via



Symbol



Net



Line



Copper Area



Text



Ratsnest



Objects

Nets · 169 objects



Facet

Nets



Filter

169 objects

Type to filter objects

Details

No object selected

Locate

Highlight

Isolate

Clear

Board Problems

0 diagnostics · 0 issues



No board problems detected.

Import Diagnostics 0

No import diagnostics.

Runtime Issues 0

No runtime issues.

☒ Net Labels

14:43

Selection Filter

Pad, Via, Symbol, Track · 10 candidates

Default ?

All

None

Default

Routed Copper ?

- ☒ Track ?
- ☐ Segment ?

Other Objects ?

- ☒ Pad ?
- ☒ Via ?
- ☒ Symbol ?
- ☐ Net ?
- ☐ Line ?
- ☐ Copper Area ?
- ☐ Text ?
- ☐ Ratsnest ?

Pick Candidates

1. U200 · Package_QFP:LQFP-64_10x10mm_P0.5mm
F.Cu · /Microcontroller/MCU_HSE_OUT

2. U200 · Package_QFP:LQFP-64_10x10mm_P0.5mm

What the Schematic Workbench Can Do

The schematic Workbench is built for view-first KiCad schematic / project workflows. It does not show PCB-only visibility and rule controls.

A standalone `.kicad_sch` that does not reference child sheets can open directly in the Schematic Workbench through `Import File`. A schematic that references child sheets must instead use `Import Folder` with a valid `.kicad_pro`. When opening a `.kicad_pro` project, the app first opens the schematic with the same name as the project; if none has that name, it selects an available schematic by path order. Only a project without an available schematic opens in PCB Viewer, while its schematic, library, settings, and other regular files from the selected directory remain under `Project`.

Sheets

- Shows the current project sheet tree, breadcrumb, and top / child sheets.
- Clicking an openable sheet switches the current 2D schematic canvas.
- Missing or unopened sheets keep issue state instead of failing silently.

Search

- Supports `Current Sheet` and `Whole Project` scopes.
- Facets include `Symbols / Nets / Labels / Text / Markers`.
- Clicking a result selects the object; locate switches sheets when needed, then centers or fits the target bounds.

Selection / Details

- Shows the schematic entity selected from search, sheet navigation, or canvas pick.
- Common actions include `Locate / Highlight / Clear`.
- When the same KiCad project contains a PCB and the current selection has an exact associated object, this area shows **View in PCB**. A single canvas jumps and offers a return action; dual canvases link automatically through selection.
- Canvas pick and Workbench selection share the same state, so there are not two competing selections.

Problems

- Collects schematic import issue details, unsupported content, and clear issue messages when the required view data is unavailable.
- If the complete required view data is missing, the UI reports the unavailable issue instead of guessing object lists in the app.

What Visibility Can Do

- switch display presets
- change the stackup viewing mode
- show or hide layer groups, individual layers, semantic columns, and technical layers
- control DRC markers globally, by layer group, or by individual layer in the 2D Display Grid
- toggle Rat / Board Boundary / Place Boundary / Assembly Outline
- edit layer colors or reset them to defaults

DRC in the 2D Display Grid

After a PCB finishes first-frame preparation, the app runs DRC for the current board in the background. While it runs, the status area reports that DRC is being prepared or shows progress. When it completes, findings with locating evidence appear as red circles and crosses on the 2D canvas; paired evidence also has connector lines.

- The DRC column appears only in the PCB 2D Visibility Display Grid, not in schematic or 3D controls.
- Select the DRC column heading to show or hide every DRC marker.
- Select a DRC cell in a layer-group or individual-layer row to control only that layer's markers; cells with no markers are unavailable.
- The All preset includes DRC marker state. Reopening a board or switching runtime sessions recalculates DRC for the new board instead of reusing old results.

DRC markers are a display layer for the current complete analysis result. Hiding markers does not modify board data or reinterpret or remove a DRC finding. If analysis is incomplete or returns an unknown layer, the app reports a clear error instead of publishing partial markers.

Live DRC runs only for native, DSN, and KiCad sources that retain design-level copper objects, nets, and rule semantics. Manufacturing outputs such as Gerber, Excellon, ODB++, and IPC-2581 contain already synthesized layer graphics; the app does not reinterpret valid overlaps of those graphics with pads, tracks, or vias as shorts. These sources clearly report that no copper model is available for design-rule evaluation and do not publish DRC markers.

When you are in `3D`, this area also shows an extra 3D section for board-surface and auxiliary display settings, including `Show 3D models`. When enabled, renderable imported 3D models are preferred; missing, unsupported, or failed models show fallback/proxy bodies or issues instead.

What `Selection Filter` Can Do

- switch quickly to `All / None / Default`
- limit which object categories are pickable
- show `Pick Candidates` when track/segment disambiguation is needed

The default filter keeps the most common surface targets and routed copper objects available, which fits most everyday browsing.

What `Objects` Can Do

`Objects` is the most direct object browser in the current version. You can:

- switch between facets
- type text to filter objects
- select a target from the result list
- apply actions to the current selection

Current facets include:

- `Nets`
- `Layers`
- `Via Definitions`
- `Rule Classes`

The main Objects panel keeps lightweight actions available directly:

- `Locate`

- Highlight
- Clear

Runtime-backed details are opened explicitly with `Details`. When the sheet finishes loading and the current object type supports it, `Isolate` appears there.

Current Responsibility Boundary

Workbench is responsible for:

- visibility
- filtering
- search
- local object browsing
- focus / highlight / clear actions

`Details` is responsible for:

- building or reusing the runtime query session on demand
- session-backed object details and net analysis
- isolate actions for supported runtime objects

Workbench no longer creates runtime sessions just because selection changes.

Recommendations

- Tune `Visibility` first, then `Selection Filter`, and then use `Objects` search for the best hit rate.
- If you cannot find a target, first check whether visibility or selection filtering is too strict.
- When recording an issue, use Workbench to narrow the target, then return to 2D for the actual detail and copying step.

Common Questions

Why is there no object-detail panel here anymore?

The current design moves the detail card back into 2D so Workbench can stay focused on filtering and targeting.

Why is `Isolate` not in the main Objects panel?

The main panel is a local browser and should not create a runtime session implicitly. Open `Details` ; if the loaded runtime object supports isolation, the sheet shows `Isolate` .

Why do I see extra visibility controls while in 3D?

Because Workbench exposes additional board-level 3D display options while the workspace is in `3D` .

Quick Estimate

Goal

Help you understand what `Quick Estimate / Revision Quote` on the `Data` page actually does, which inputs it uses, when manual confirmation is still required, and why it is not a live vendor quote.

Use Cases

- You want a fast estimate of how the current revision's layer count, board size, and basic manufacturing settings affect a prototype quote
- You want to archive the current local quote assumptions as `report` results
- You need estimate context next to revision details and issue details instead of re-entering everything on a vendor website

What You See on the Data Page

- the main `Quick Estimate` card
- the `Revision Quote` status and any action-needed prompts
- detected layer count, board size, and input-source captions
- local catalog switching between `JLCPCB` and `PCBWay`
- `Archive Quote Report` for snapshot-style reporting
- `Notes` and the `Catalog Disclaimer`

Where the Inputs Come From

The current local estimate mainly uses three sources:

- revision details and the current project files

- viewer-derived layer and board-size information
- built-in local vendor catalogs and quote rules

If layer count or board size is still uncertain, the page marks that input as inferred or manually required instead of silently treating it as final.

Recommended Flow

1. Open the workspace and switch to `Data` .
2. Check whether `Revision Quote` is currently `Ready` or `Needs Manual Confirmation` .
3. Review `Layers` , `Board Size` , `Detected Source` , and `Assumptions` .
4. If something is missing, open `Advanced Manufacturing Settings` and confirm the required fields.
5. When you need a snapshot for handoff or support, use `Archive Quote Report` .

Important Limits

- This is a local estimate based on bundled rules and catalogs, not a live manufacturer quote.
- Price, lead time, and capability assumptions still need final confirmation from the manufacturer's official quote flow.
- `Quick Estimate` is useful for early review, handoff summaries, and troubleshooting context. It is not a purchasing guarantee.

Troubleshooting Tip

- If the estimate looks wrong, check `Board Problems` , `Issue Summary` , and the current revision source first.
- When you need to report an issue, copy the `Support Bundle` . It includes document details, issue summaries, Quick Estimate status, quote report paths, and AI activity summary, but does not include AI credentials or full design file contents.

Export and Share

Goal

Help you turn the current 2D view or the current board into something you can deliver, share, or continue processing elsewhere.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users exporting board views or board-file outputs for delivery and sharing
- Testers validating capture export, original-source export, and converted export behavior

Prerequisites

- A managed board document is already open in 2D
- If full export is expected, export access should already be unlocked
- If export access is still locked, you can still view the export configuration; confirming export opens the purchase center

Steps

1. Return to 2D and confirm the current visibility state, framing, and export target.
2. Select the export button in the bottom floating toolbar, then choose the format and destination that match the task.

3. Confirm the export and open the result, or confirm that the purchase-center redirect appears when export access is still locked.

Result

- The current board view or board data is delivered through the appropriate capture, original-source, or converted export path.

Currently Supported Export Formats

Capture exports

- PNG
- PDF

Original-source exports

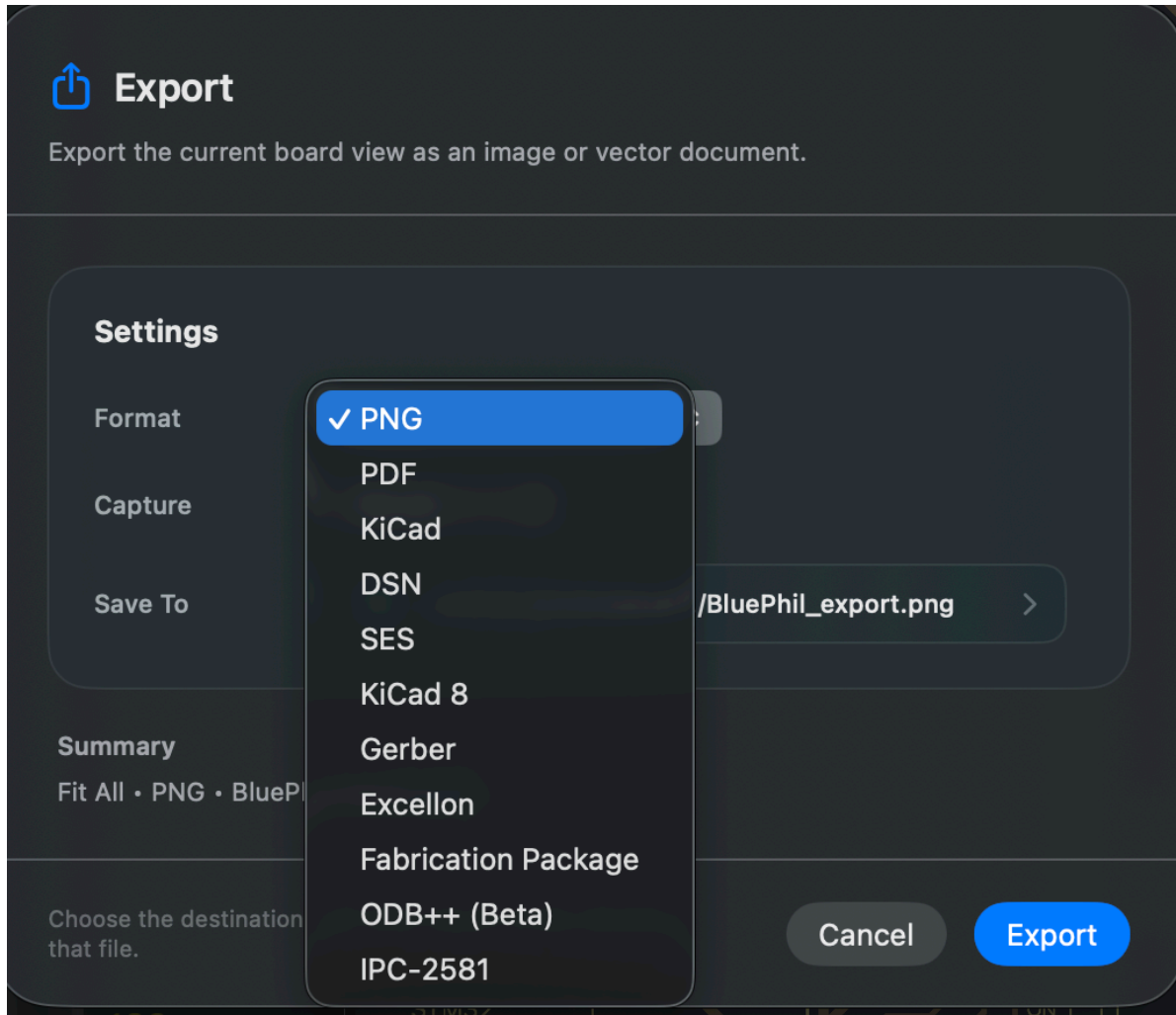
- DSN
- KiCad
- IPC-2581
- ODB++
- Gerber/Drill

This path returns the original project files or design-file set stored with the current revision:

- single-file formats are returned as the original file, without regeneration
- multi-file formats are returned as a ZIP, with the original paths and bytes preserved

Converted exports

- DSN
- KiCad 8



Current Export Preconditions

- export can only be started from **2D Viewer**
- a board file must already be open
- the export configuration can be opened directly; when export is confirmed, access must be unlocked on the current device or account or the purchase center opens
- a new export request is blocked while another export is in flight

Export Scope

PNG / PDF

You can choose:

- Viewport
- Fit All

DSN / KiCad 8

These are converted exports. They are not screenshots, and they are not simply returning the original project files. The app exports the whole board and does not expose a viewport-based partial scope here.

If a KiCad board has dynamic copper zones (zones / dynamic planes) whose editable semantics must be retained, conversion to DSN fails explicitly rather than silently flattening the current filled outline and losing dynamic semantics. Use KiCad 8 converted export or original-source export instead.

Original-source exports

Original-source exports also do not use viewport scope:

- if the current revision is already stored in that format, the app returns the original source first
- for example, if the current revision is KiCad .kicad_pcb, exporting KiCad returns the original KiCad file
- if the current revision is DSN, exporting DSN returns the original DSN file
- if the current revision is a multi-file manufacturing package, export returns a ZIP containing the original project files

Platform Differences

iOS

- PNG : saves only to Photos
- PDF / original source / DSN / KiCad 8 : delivered only through Other Apps

Notes:

- images are not written to Files first; saving requests the system's photo-library **Add** permission

- files do not attempt to write to a directory the app cannot access; `Other Apps` uses the system share sheet

macOS

- all formats export only to `Files`
- selecting the path field opens the save dialog so you can change the destination path
- if the path is unchanged, confirming export asks the system for access to the prefilled location; after approval, it exports directly instead of reopening the save dialog
- the save dialog prefills a default file name, and the default directory is `~/Documents`

File Naming

The default file name uses the current source name and purpose, followed by a local-time `_MMddHHmm` suffix:

- `board_export_12020430.png`
- `board_export_12020430.pdf`
- `board_dsn_export_12020430.dsn`
- `board_kicad_export_12020430.kicad_pcb`

For example, `_12020430` means December 2 at 04:30. A filename changed in the macOS save dialog is the final filename; export does not generate another name afterward.

If you are using original-source export:

- single-file exports prefer the original file name
- multi-file exports prefer the current document name and produce a `.zip`

Recommended Workflow

1. return to `2D`
2. if export access is still locked, complete the purchase or restore flow first
3. adjust visibility, framing, and zoom
4. choose the format and destination
5. open the result immediately after export and verify it

Common Questions

Why can't I export from 3D or Data?

Because the current version only allows export to start from **2D**.

Why did export open the purchase center first?

Because export is currently gated behind a separate export unlock. The workspace lets you open the export configuration; if access is still locked when you confirm export, it redirects to the purchase center.

Why do image exports and board-file exports offer different options?

Because **PNG / PDF** are viewport captures, while board-file export is now split into two paths:

- **Original source export:** returns the original files stored in the current revision
- **Converted export:** generates a new target format from the current workspace content

Why can an image go to Photos on iOS, but PDF cannot?

Because iOS image delivery is connected to the photo library; the other export formats are files and are delivered only through the **Other Apps** system share sheet.

Settings

Page Goal

Help you quickly find adjustable options and understand how the current Settings page is organized.

Current Settings Categories

The Settings page currently has six top-level categories:

- General
- Viewer
- PCB Display
- AI
- Support PCBAtlas
- About and Reset



Search



General

Appearance, language, and logging.



Viewer

2D and 3D display, camera, rendering, and appearance.



PCB Display

Layer colors, object styles, labels, and interaction behavior.



AI

Providers, routing, privacy, and service options.



In app purchase

Manage app purchases.



About & Reset

Version info and reset tools.

What Each Category Covers

General

- Interface appearance mode
- Terminology preference
- Status toast toggle
- Issue Details log level
- Language information entry

Viewer

- 2D Viewer display, camera, status bar, and interaction preferences
- 3D Viewer projection mode, board-surface display, auxiliary display mode, and `Show 3D models`
- Renderable imported 3D models are preferred; missing, unsupported, or failed models show fallback/proxy bodies or issues instead
- Examples include whether to show `axis / grid`, what appears on the right side of the status bar, and whether 3D uses orthographic or perspective projection; the mini viewer is always shown
- `axis` and `grid` are independent settings: the axis action in the mini viewer menu only affects axes and does not also change grid
- **Camera behavior on open** can restore the last saved position for that case (fitting the whole board when no valid position exists), or fit the whole board every time it opens

PCB Display

- Styles and colors directly related to board display
- Includes layer palette, board outline, track / via / pad, silkscreen, and assembly-line display parameters
- Net label display also lives here because it describes how board objects are presented
- PCB element opacity is currently clamped to `0.1 . . . 0.9`: silkscreen is the most visible by default but is no longer fully opaque, while shape / plane defaults keep the lowest visible opacity

AI

- AI service setup
- Secret key
- Model list
- AI service options
- Request model and sharing settings
- Secrets are stored securely by the system and are not written into normal settings files

Support PCBAtlas

- The current purchase page presents a single primary item: `UnLock Plus`
- It is a one-time purchase, not a subscription
- Plus unlocks unlimited board workspaces, `PNG / PDF` captures, original-source delivery, converted board outputs, and share workflows
- Restore purchases

About and Reset

- Version and information
- The `Find a bug?` feedback entry opens a prepared email after confirmation and does not automatically upload projects, usage information, or diagnostics
- Open-source licenses
- Reset by section
- Full reset

Important Capabilities on the Current Settings Page

- **Search:** supports searching by display name, description, category, and some terminology aliases
- **Language page:** reachable from General, but mainly explains how to follow the system language setting
- **Open-source licenses page:** reachable from About and Reset

- **Reset confirmation:** resetting a single section or all settings always asks for confirmation first
- **Full-row click:** setting categories and drill-in rows support full-row click / touch, not only the text or icon

Terminology Preference

The current version keeps terminology preferences and supports these styles:

- `generic`
- `kicad`
- `allegro`
- `altium`

This switch only affects:

- Interface labels
- Settings search
- Some terminology aliases

It does not change:

- Import results
- The documentation itself

Real-World Limits of AI Settings

The AI category supports multiple AI service setups, but actual availability depends on:

- Whether you entered a usable secret
- Whether the corresponding AI service is reachable
- Whether the current device and network allow the request to succeed

Three additional boundaries are important:

- AI-related features may send board context, selected objects, net names, component names, and chat text to the AI service you configure. Use stricter sharing settings for sensitive designs.

- AI summaries, issue details, and suggestions are assistive outputs only. They do not replace engineering review or manufacturing validation.
- `Support Bundle` , normal export, and sharing flows do not include AI service credentials.

Real-World Limits of the Language Page

The current language page is more of an explanation page. It tells you to change the app display language at the system level:

- On `iOS` , use the system App Language setting in most cases
- On `macOS` , use the system language or app-language settings in most cases
- You may need to restart the app before the interface fully reflects the change

Usage Suggestions

- Change only a few key settings at a time so it is easier to understand what caused a visible difference.
- If you mainly want a better reading experience, start with General and Viewer.
- If you mainly want to adjust colors, layers, and board-object visuals, start with PCB Display.
- When you need to restore default behavior, prefer resetting one section instead of doing a full reset.

FAQ

Why do some settings only become visible after returning to the Viewer?

Because many settings are Viewer behavior or display parameters. You can only see their effect in the corresponding view.

Can Settings search find EDA terminology aliases?

Yes. Search covers entry names and also includes some terminology aliases, such as different EDA names for the same concept.

Can reset take effect without message?

No. Reset actions always show a confirmation dialog first.

Why is there no button on the language page to switch immediately to Chinese or English?

Because the current version does not have an independent in-app instant language switch. The language page mainly explains the system-side switching path and restart note.

Import Failures

Goal

Help you quickly decide whether the current failure is an unsupported input, an import-path mismatch, or a proactive status protection stop.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users whose import path did not behave as expected
- Testers triaging import failures, ignored imports, or degraded import outcomes

Prerequisites

- You have a specific import case, message, or reproduction path to inspect
- If possible, keep the source file and the exact import path that triggered the issue

Steps

1. Identify whether the failure happened before file selection, after file selection, or after entering the workspace.
2. Check which import path was used and whether the source should be supported there.
3. Compare the observed behavior against the failure patterns below, including unchanged re-imports and partial DSN degradation messages.

4. Record the exact message and source context before retrying.

Result

- You can narrow the issue down to unsupported input, path mismatch, protective stop, unchanged re-import, or a local degradation case.

Look at These Three Things First

1. Did the failure happen **before file selection**, **after file selection**, or **after entering the workspace**?
2. Did the app show a clear error message?
3. Was the action a **new import** or **append revision**?

Files



Import File



Import Folder



Free plan

0/3 workspaces used

Unlock

pcb

BluePhil



Failed to Open File

Import Folder requires choosing a folder. Use Import File for standalone board files and archives.

OK

Copy



KiCad

Last Opened

Never opened

Most Common Failure Types Right Now

1. Unsupported input type

Typical signs:

- the app directly says the extension is unsupported
- the flow stops after you pick the file

Suggested actions:

- first confirm whether you used the design or manufacturing path
- then confirm whether the input belongs to the currently supported set

2. Wrong IPC-2581 import intent

Typical signs:

- the file can be selected, but the resulting import behavior is not what you expected

Suggested actions:

- import it again
- reselect whether it should be treated as design-side or manufacturing-side data

3. Large IPC-2581 opens slowly or fails during real import

The current version no longer stops very large IPC-2581 XML inputs with a conservative memory stability check.

If this kind of source still fails, the failure now happens in the real import path instead of being rejected before opening.

Suggested actions:

- record the exact error text or last visible loading stage
- record the file size, platform, and device model
- note whether the app stayed responsive, stalled, or was terminated by the OS

4. Another import task is already running

Only one PCB import task is allowed at a time in the current version. If a previous import has not completed yet, the next import request is rejected with a status message.

5. Same-name, same-format input is judged as unchanged

The current version compares original source bytes when you re-import a source with the same document name and the same format:

- if the newly imported original source is identical to the current revision, the app reports that the import was ignored because nothing changed
- this is not treated as a failure; it is how duplicate revisions are avoided
- if you expected a new revision, confirm that the file contents really changed

6. `DSN` local geometry degradation after a successful import

Typical signs:

- import ultimately succeeds and the document opens
- but the some local details do not appear as expected

What it means:

- some local details could not be shown, while the rest of the board remained available
- this behavior is meant to avoid failing or crashing the whole board import, and it usually affects only local rendering rather than the entire document

Suggested actions:

- first confirm whether the affected area is still usable for the task you care about
- if the missing geometry matters, record the message text, the original `DSN`, and the reproduction steps for follow-up investigation

KiCad-Specific Note

Even if a KiCad board imports successfully in 2D terms, missing or unreadable project-local 3D assets may still surface as issues later in 3D or issue details. That does not always block 2D

opening, but it can reduce 3D completeness.

What to Record

- platform and device
- import path type: design / manufacturing
- original file type and size
- the original error text
- whether it reproduces consistently
- whether another import task was already running

Common Questions

Why can some manufacturing inputs be imported even though they are folders?

Because the manufacturing path accepts directory-style inputs. Whether import succeeds depends on whether the app can recognize the directory as a supported manufacturing package.

Why is there no new card in the library after a failed import?

That usually means the failure happened before document creation, such as unsupported input, stability check rejection, or app detection failure.

Why didn't I get a new revision after importing the same file name a second time?

If the app says the import was ignored because nothing changed, that means the newly imported original source is identical to the current revision. The current version skips it instead of creating a duplicate revision.

It worked on the second try. Should I still record it?

Yes. Intermittent failures still matter for stability evaluation.

Permissions and File Access

Goal

Help you distinguish between application logic problems and system file-access restrictions.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users blocked by file visibility or destination-access problems
- Testers separating permission issues from app-logic failures

Prerequisites

- A concrete import, export, or drag-and-drop path has already failed or behaved unexpectedly
- You can retry the same path and observe whether the system shows a permission prompt

Steps

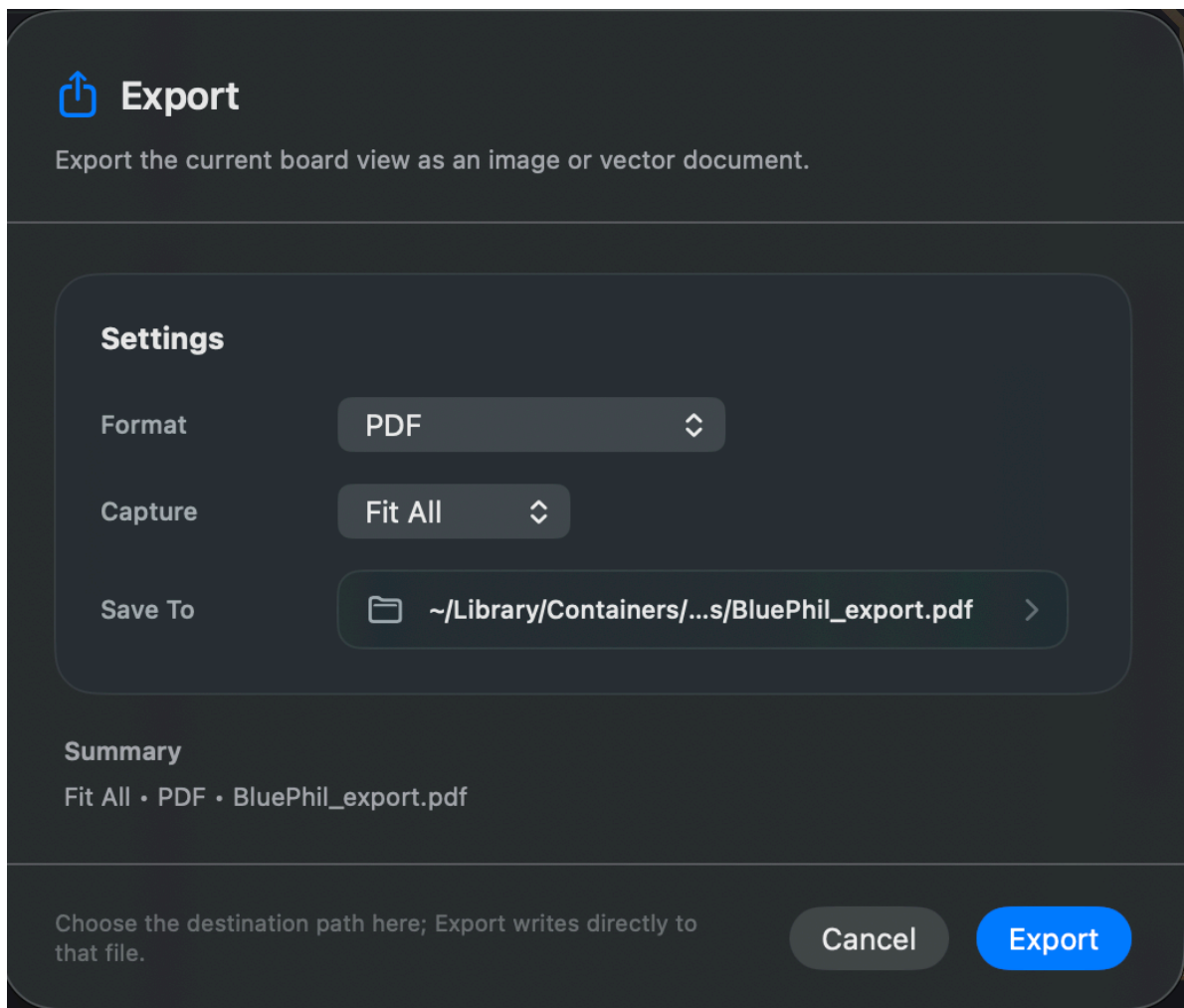
1. Retry the same path and watch for system permission prompts.
2. Confirm that the source or destination location is actually accessible to the app.
3. Check whether the failure matches a platform-specific rule such as PNG -> Photos on iOS.
4. Record the exact error text and the rough location involved.

Result

- You can tell whether the problem comes from system access restrictions or from a different application-level issue.

Most Common Permission Scenarios

- the file picker cannot see the target data
- the system refuses access after a file is selected
- export fails when writing to the destination
- saving to Photos fails on iOS
- drag and drop is blocked by the system



Recommended Troubleshooting Order

1. Check whether the system showed a permission prompt.
2. Confirm the target data is stored in a location the app can access.
3. On iOS, remember that the Photos destination only applies to `PNG`.
4. On macOS, verify that the save-panel destination is writable.
5. Retry the exact same path and confirm whether the problem reproduces.

A Few Behavior Facts That Matter

- If the user cancels file picking intentionally, the app does not treat it as an error alert.
- Drag-and-drop import depends on the system providing a valid `fileURL`; without permission, the app cannot resolve one.
- On iOS, exporting to `Photos` is a distinct path and is not the same as a normal `Files` export.

What to Record

- platform and system version
- rough destination location
- whether the system showed a permission prompt
- whether the failure happened during import, export, or drag and drop
- the exact error text

Common Questions

The file exists. Why can't the picker see it?

Most of the time, this is caused by system access-scope restrictions rather than the app being unaware of the file. Move the file to a more directly accessible location and try again.

If export fails, does that always mean export is broken?

No. A destination that is not writable, missing permissions, or a mismatch between format and destination can also cause export failure.

Why does saving to Photos only support PNG?

Because iOS image delivery is connected to the photo library. Other export formats are files and are delivered only through the Other Apps system share sheet.

Performance and Loading

Goal

Help you decide whether the app is still progressing normally or is actually stuck.

Supported Platforms

- iOS
- macOS

Intended Audience

- Users waiting on import or view preparation and unsure whether the app is still progressing
- Testers triaging loading stalls, apparent hangs, or perceived performance regressions

Prerequisites

- A document is currently importing, opening, or building a 3D view
- You can still observe the visible progress state, stage text, or UI responsiveness

Steps

1. Check whether the stage text, percentage, or loading card is still changing.
2. Confirm whether the UI still responds to basic interaction.
3. Record the exact stage and how long the app remains there before deciding that it is stuck.
4. Compare the behavior against the known normal-wait and suspicious-stall patterns below.

Result

- You can distinguish between normal staged progress, a protective stop, and a likely real loading problem.

Common Waiting Stages in the Current Version

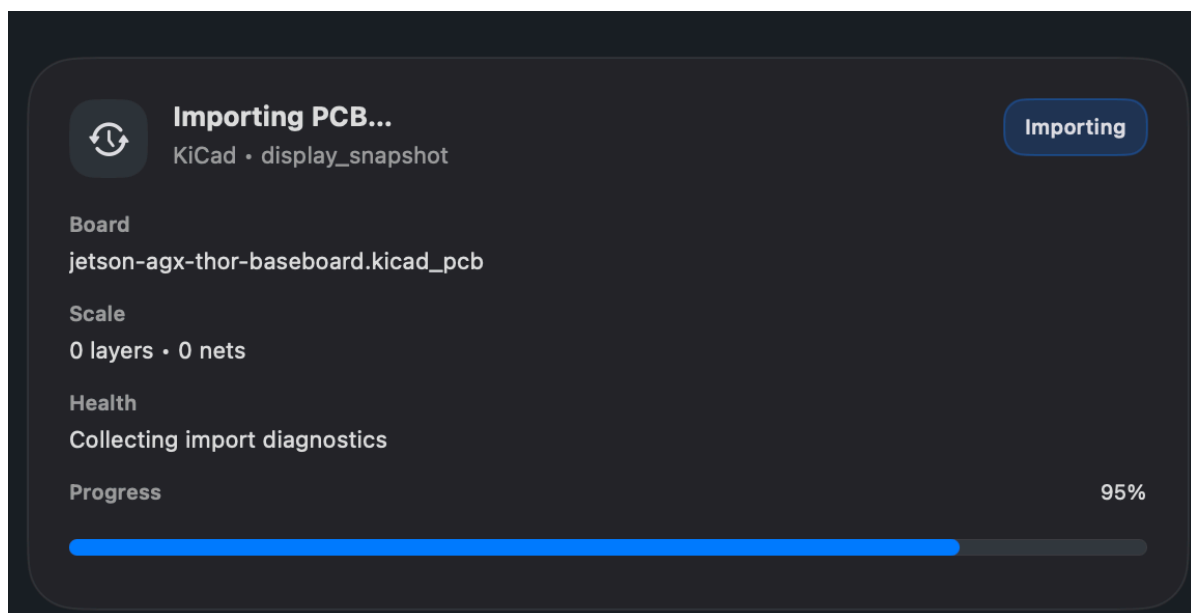
Import to 2D preparation

- the top of the library shows an import banner
- the workspace may temporarily switch to `Data`
- the 2D viewer may remain blocked until the first projected frame is published

3D view construction

3D shows explicit stage and progress text, such as:

- `Loading`
- `Details`
- `View`
- `Finishing`



How to Tell That Progress Is Still Normal

Focus on these signals:

1. the progress number or stage text is changing
2. the UI still responds to basic interaction
3. the import banner or 3D loading card keeps updating

If all three are true, the app is usually still progressing normally.

When to Suspect an Actual Problem

- the app remains on the same stage for a long time with no text change
- the UI becomes unresponsive
- the same input repeatedly stalls at the same point on the same device
- the behavior only appears in this version and not in previous stable ones

A Few Current Behavior Facts

- During import, the viewer may automatically switch to `Data`; this is a normal transition and does not mean opening failed.
- 3D requires a board session with enough board-level geometry. If that input is missing, a clear issue message is not a performance problem.
- Huge `IPC-2581` inputs are no longer proactively stopped by an opening-time memory stability check; if one is very slow or eventually fails, record it as a real import-path problem.

What to Record

- device and platform
- input type and size
- which stage the app stops at
- how long it stays there
- whether the UI still responds

- whether the issue only appears in one view

Common Questions

If it waits for a long time, does that always mean something is wrong?

No. Large boards, first-time 3D view construction, and manufacturing-package recognition can all produce noticeable wait time. What matters is whether the state is still advancing.

Why is it important to record the exact stage?

Because the current version already breaks import and 3D loading into explicit stages. Recording the stage makes reproduction and diagnosis much faster.

Glossary

Goal

Help you align quickly with the concepts that appear most often in this handbook.

Common Terms

Library

The home entry that manages all documents.

Document

One project unit in the library, usually representing one maintained board project.

Revision

One historical version under a document.

Current Revision

The revision currently opened, displayed, and exported.

2D

The main working view in the current version, responsible for inspection, object info, and export.

3D

The board-level 3D view for documents that can form a board session; it shows board surfaces, primary board geometry, and component bodies, preferring imported 3D models when available.

Data

A summary page for document, revision, statistics, capability, status, and issue information.

Workbench

The side-panel area used for visibility, filtering, search, and object actions.

Selection Filter

The control that decides which object types are pickable.

Objects

The Workbench section used to browse and search objects by facet.

Proxy Body

A fallback component body used in 3D when an imported component model is missing, unsupported, or cannot be rendered.

Terminology Profile

The settings switch that changes UI terminology style only. It affects wording and search, not underlying data.

Manufacturing Package

A directory- or archive-style manufacturing input handled by the manufacturing import path.

Export

Outputting the current 2D view as an image or PDF, or exporting the current revision's original project files and converted board files.

Common Questions

Why does the handbook distinguish between “document” and “revision”?

Because the library model is “one document for one project context, many revisions for its evolution.” That structure is better for ongoing maintenance and traceability.

What is the difference between Data and Workbench ?

Data shows summaries and statistics. Workbench is used for filtering, search, and actions.

Smoke Test

Goal

Use the shortest set of steps to validate the main path from the library to the workspace, viewer, and export.

Recommended Minimum Samples

- start with the built-in `h730duino` or `BluePhil`
- if you also want to validate import, add one of your own `DSN` or `KiCad` files

Minimum Smoke Path

1. Launch the app and enter the library.
2. Confirm the library home page is interactive.
3. Open a document and enter the workspace.
4. In `2D`, complete one pan, one zoom, and one object selection.
5. Right-click or long-press an object and open `View Info`.
6. Open `Workbench` and perform one visibility or object-search action.
7. Switch to `Data` and confirm the summary page appears.
8. If the current document forms a board session, switch to `3D` and wait for the board-level scene to finish loading.
9. Open `AI`. If no AI service is configured, confirm that the setup card points to `Settings > AI`.
If an AI service is configured, confirm that `Diagnose Selection / Chat` are available and that the advanced cards can be expanded.
10. Return to `2D` and try one export. If export is still locked, confirm that the purchase center opens first.
11. Go back to the library and reopen the same document.

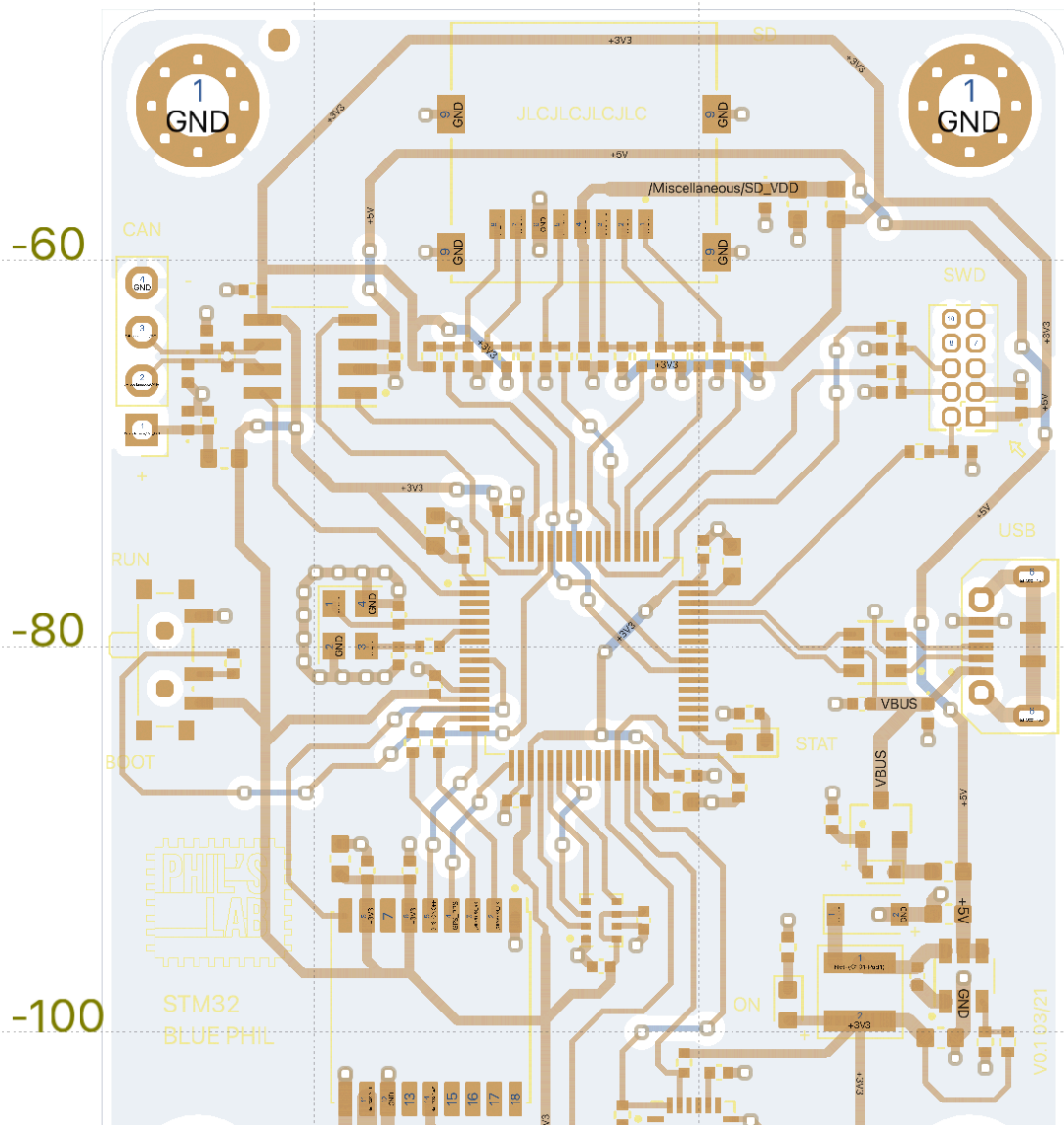
14:42

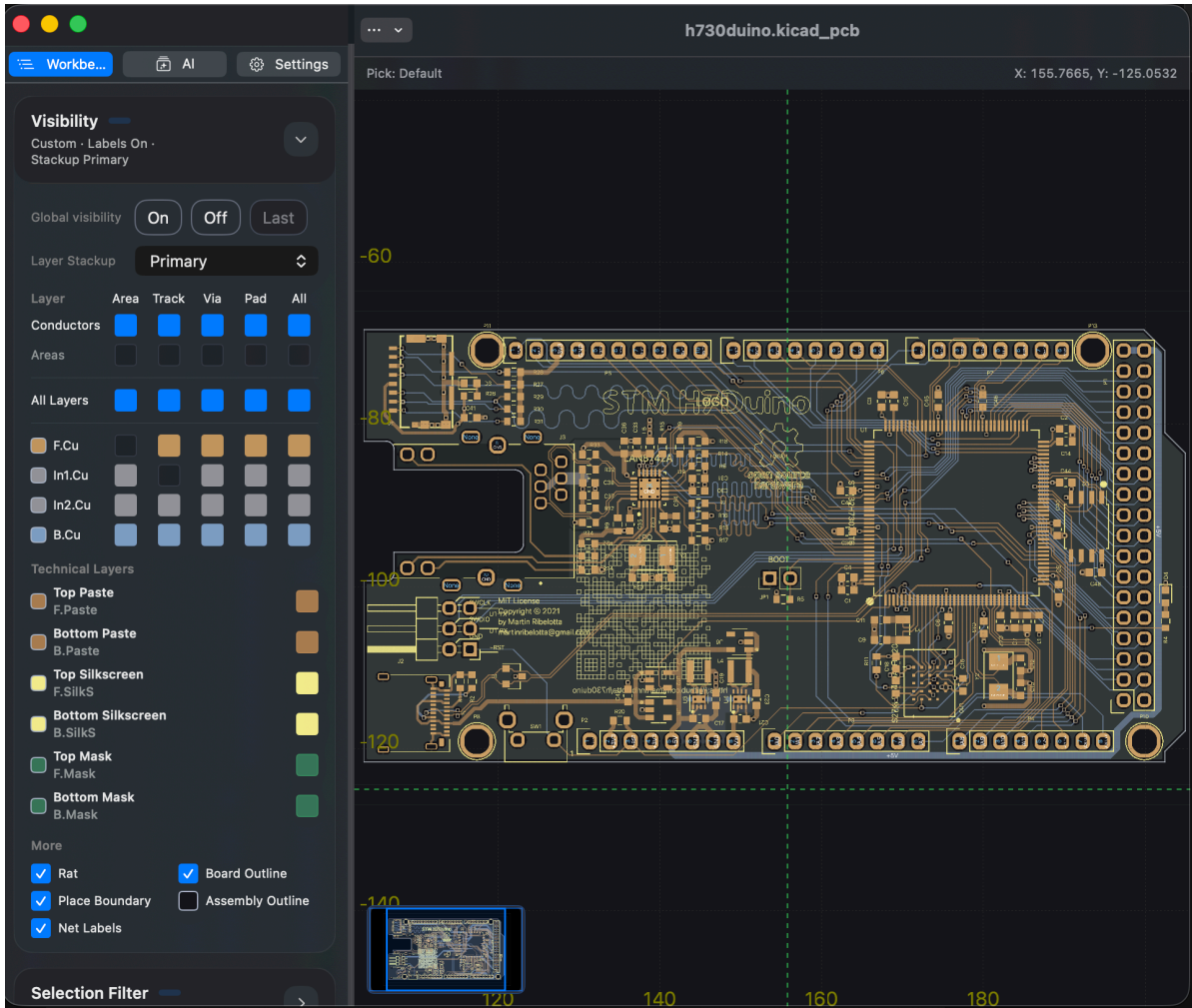


BluePhil.kicad_pcb

Pick: Default

X: --, Y: --





Revision Quote

Ready

Quick Estimate

JLPCB Prototype

COMMERCIAL CONTEXT

BASE CURRENCY REFERENCE

CNY 29.21 / CNY 5.84

Reference FX: 1 CNY = 0.1380 USD
(Bundled FX snapshot, Apr 18, 2026 at 8:00).

SERVICE TIER / ETA

Standard / 4-6 business days

QUANTITY

5

Tier 5-9 pcs

DETECTED BOARD INPUTS

LAYERS

2

Confirmed from Runtime copper layers

BOARD SIZE

66.05 x 50.05 mm

Confirmed from Edge cuts/profile geometry

ASSUMPTIONS AND SOURCE

DETECTED SOURCE

Layer Confirmed from Runtime copper layers

Size Confirmed from Edge cuts/profile geometry

ASSUMPTIONS

Display currency USD, Quantity 5, Material FR-4, Thickness 1.60 mm, Surface finish HASL, Service tier Standard, Copper 1.00 oz, Mask Green, Silkscreen White, Panel mode Single, Electrical test Included

PRICE BREAKDOWN

Setup

One-time setup fee

USD 1.93

Board Area

66.05 x 50.05 mm, billable 3305.80 mm^2

USD 2.10



Notes

Pricing model: JLPCB Prototype. Detected copper layers from Runtime copper layers. Detected board size from Edge cuts/profile geometry. Current quantity tier: 5-9 pcs. Reference FX: 1 CNY = 0.1380 USD (Bundled FX snapshot, Apr 18, 2026 at 8:00).



Catalog Disclaimer

Estimate only. Final price must be confirmed with the manufacturer's official online quote.

Edit Inputs

Archive Quote JSON + Text



Pass Criteria

- both the library and workspace are reachable
- 2D and basic Workbench interaction are not blocked
- **Data** can show the current document summary
- 3D works for samples that form a board session
- **AI** shows the expected setup gate when unconfigured, or the expected **Diagnose Selection** / **Chat** flows when configured
- 2D export succeeds and can be verified, or in a locked state the purchase center opens correctly

Suggested Notes

- test platform
- sample type
- whether the built-in examples were used
- whether 3D was an applicable path
- failing step and message text

Common Questions

Why include `Data` in the smoke path?

Because the current import and view-preparation flow uses the `Data` view as part of the real user path.

If the sample cannot form a board session, does 3D failure count as a blocker?

No. 3D requires a board session with enough board-level geometry. Samples that do not meet that condition should skip that step.

Format Coverage Matrix

Goal

Help you quickly confirm the real supported scope for current input and output formats.

Import Coverage

Path	New document	Append revision	Notes
DSN	Supported	Supported	Design import path
KiCad .kicad_pcb	Supported	Supported	Supports 2D / Data / 3D; 3D can prefer imported component models when available and uses proxy bodies when models are missing or unsupported
KiCad .kicad_sch	Standalone file only	Standalone file only	A schematic that references child sheets must be imported through Import Folder with a valid .kicad_pro; a second file picker does not appear
KiCad .kicad_pro project folder	Supported	Supported	Import Folder must read the project file and retain every regular file in the selected directory; it does not fall back to a PCB or schematic import

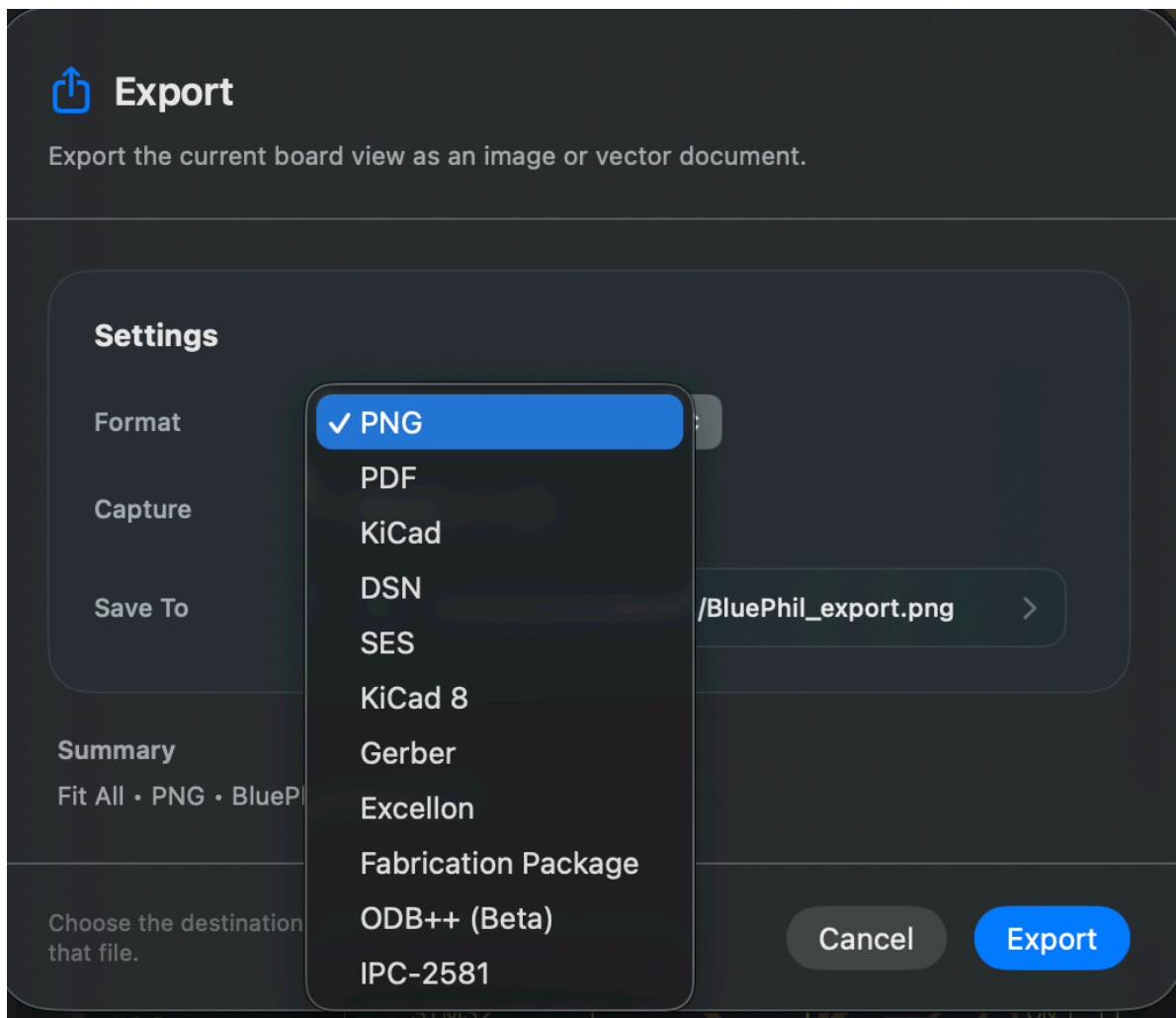
IPC-2581 XML	Supported	Supported	Import asks you to choose design vs manufacturing intent
ODB++ archive	Supported	Supported	Manufacturing import path; a new document uses an archive
Manufacturing-package directory data	Not supported	Supported	The Import Folder entry for new documents is reserved for KiCad projects; a manufacturing revision of an existing document can still select a directory
SES	Not supported	Not supported	The current version offers no SES import entry

View Coverage

View	Applies to	Notes
2D	All open documents	Main PCB review and export view; schematic sheet viewing, search locate, and canvas pick also happen here
Data	All open documents	Used for summaries, statistics, and import-stage transitions
Project	KiCad .kicad_pro	Browse project source documents, libraries, and settings, and switch between PCB and schematic entries
3D	Design inputs that form a board session	Shows board surfaces, pads, vias, silkscreen, and component bodies; imported 3D models are preferred when available, with proxy bodies or issues when model assets are missing or unsupported

Export Coverage

Format	Type	iOS	macOS	Notes
PNG	Capture export	Photos	Files	2D only; iOS images save only to the photo library
PDF	Capture export	Other Apps	Files	2D only
DSN	Original-source or converted export	Other Apps	Files	Returns original DSN first when the current revision is already DSN
KiCad	Original-source export	Other Apps	Files	Returns the original .kicad_pcb when the current revision is KiCad
KiCad 8	Converted export	Other Apps	Files	KiCad 6 is no longer offered
IPC-2581 / ODB++ / Gerber-Drill	Original-source export	Other Apps	Files	Multi-file sources are delivered as a ZIP



Regression Recommendations

- Cover at least one design input and one export format in every regression pass.
- If this round touches Project / schematic behavior, cover one `.kicad_pro` project, one standalone `.kicad_sch`, and verify sheet switching, search, locate/highlight/clear, and symbol/pin/wire/label pick.
- If this round touches 3D, validate with a sample that forms a board session.
- If this round touches revision management, include one `append revision + switch current revision` path.
- If this round touches manufacturing import, add one directory-based or archive-based input.

- If this round touches original-source export, include one single-file return path and one multi-file ZIP export path.

Document Version

v2026.08.31.0

Regression Checklist

Goal

Provide a regression checklist aligned with the current version.

Library

- ✓ The library home page opens correctly
- ✓ In an empty-library case, `h730duino / BluePhil / WooKey Mainboard` sample documents are visible
- ✓ Grid/list layout switching works
- ✓ A new document can be imported
- ✓ Re-importing the same name and same format with byte-identical original source is ignored with a clear message
- ✓ An existing document can be opened
- ✓ A design or manufacturing revision can be appended
- ✓ Re-importing the same name and same format with changed original source appends a new revision and switches to the latest revision
- ✓ `SES` import can be launched from the document menu
- ✓ Rename, duplicate, pin, delete, share, and view-information actions work from the document menu

Workspace

- ✓ Entry points for `Workbench / AI / Settings` are available in the workspace
- ✓ Mode switching across `2D / 3D / Data` works
- ✓ The app can enter the `Data` transitional state during import preparation
- ✓ 2D can pan, zoom, and select objects

- ✓ The `View Info` card in 2D can open and copy text
- ✓ `AI` shows the setup gate when no AI service is configured
- ✓ `AI` supports `Diagnose Selection / Chat` plus advanced cards when configured
- ✓ `Visibility` is expanded by default in Workbench
- ✓ After a PCB 2D first frame is ready, DRC completes in the background. For a sample with locatable findings, red markers are visible, and the Display Grid `DRC` column controls visibility for all, layer groups, and individual layers.
- ✓ Manufacturing outputs such as Gerber, Excellon, ODB++, and IPC-2581 do not publish design-level DRC markers and clearly report that the source lacks the copper model needed for design-rule evaluation.
- ✓ `Selection Filter` is collapsed by default in Workbench
- ✓ `Objects` can search and apply actions
- ✓ If the sample forms a board session, the 3D board-level scene finishes loading

Export

- ✓ The export button is available in 2D
- ✓ If export access is still locked, tapping export opens the purchase center first
- ✓ After export access is unlocked, the export configurator opens and export completes normally
- ✓ `PNG` or `PDF` succeeds at least once
- ✓ If this round involves board-file exchange, original-source export succeeds at least once
- ✓ If this round involves format conversion, `DSN` or `KiCad 8` converted export succeeds at least once
- ✓ On iOS, if Photos export is part of validation, `PNG -> Photos` succeeds

Stability

- ✓ Import progress text or percentage changes over time
- ✓ 3D loading stage text changes over time
- ✓ Failure cases show clear messages rather than failing silently
- ✓ A sample without a board session or required board-level geometry shows a clear message when switched to 3D

- ✓ The language page explains system language switching and the possible relaunch requirement

Suggested Note Template

- Regression version:
- Platform:
- Sample:
- Is KiCad:
- Passed items:
- Failed items:
- Notes:

Capability Boundaries

Goal

Clarify how current capabilities are meant to be used and which behaviors are product boundaries rather than regression bugs.

Current Capability Boundaries

1. Board-level 3D display scope

- 3D supports design inputs that form a board session and shows board surfaces, pads, vias, silkscreen, and component bodies
- imported 3D component models are preferred when available in a renderable format
- missing, unsupported, or failed component models fall back to proxy bodies or issue records rather than blocking the whole 3D view

2. Export starts from 2D

- the export button is unavailable in 3D and Data
- 3D and Data remain review views; return to 2D before export to confirm visibility, camera, and export scope

3. Export and delivery are unlocked by Plus

- if export access has not been unlocked yet, the workspace does not start export directly
- tapping export opens the purchase center first
- after purchase or restore succeeds, export becomes available again

4. There is currently no SES import entry

- the file picker does not list `SES` as an importable document
- the document menu offers no `SES` entry for new documents or appended revisions

5. iOS Photos is the PNG destination

- `PNG` saves only to Photos; it is not handed off to Files or other apps
- `PDF`, original-source export, and converted export are delivered only through Other Apps and do not save to Photos

6. macOS export uses file destinations

- macOS uses the save dialog to choose the destination path

7. Original-source export depends on the current revision being library-managed

- the original-source export option only appears when the currently open revision is managed by the library
- original-source export does not read directly back from a temporary external path

8. AI capability depends on external AI service setup

- the settings page can configure multiple AI services
- actual usability depends on AI credentials, network access, and AI service response

9. The language page is not an in-app instant language switcher

- it mainly explains the system-level language path
- after changing it, the app may need to be relaunched

10. Directory-style manufacturing inputs go through recognition

- folders and archives go through app recognition before entering the matching manufacturing import path

11. Schematics support viewing and targeting

- The schematic Workbench supports sheet navigation, object search, details, locate, highlight, and canvas pick
- Schematic editing, ERC fixes, and property writeback stay in dedicated EDA tools
- Symbols, pins, footprints, pads, and nets with an exact object identity in the same KiCad project support bidirectional cross-probing. Dual canvases link automatically; a single canvas provides an explicit jump and one-level return.
- BOM-only components, symbols without a footprint, board footprints without an associated schematic, and missing or ambiguous relationships do not guess a target. Use details to inspect them instead.

How to Use This Page

1. Before filing a bug, check whether the case falls inside one of these boundaries.
2. If product behavior conflicts with this page, report it as a documentation-vs-version mismatch first.
3. If a capability boundary changes, update the handbook and QA pages together.

Cases to File as Bugs

- reproducible crashes
- blocking failures in a clearly supported path
- regressions from previously stable behavior

Changelog

Recent Updates

- The library home page and Settings now share a feedback entry. After confirmation, it opens a prepared email and does not automatically upload projects, usage information, or diagnostics.
- The library no longer offers an `SES` import entry, and the file picker no longer lists `SES` as an importable document.
- Library import has been organized around file import and folder import, making it clearer how to bring in design files, project folders, and manufacturing packages.
- KiCad project and schematic review have been updated with project browsing, sheet switching, search, locate, and canvas picking.
- The 2D, 3D, Data, Workbench, AI, and export handbook pages now match the current user interface.
- Settings documentation now covers the current categories, search, display preferences, terminology preferences, AI setup, and purchase / restore entries.
- The documentation site now includes multilingual handbook entries and current-language PDF download links.

This page records user-visible changes only and omits internal internal details.