

# KiCad Project Status

IOB\_32a\_SPI\_S0  
IOB\_33b\_SPI\_SI  
IOB\_34a\_SPI\_SCK  
IOB\_35b\_SPI\_CSN

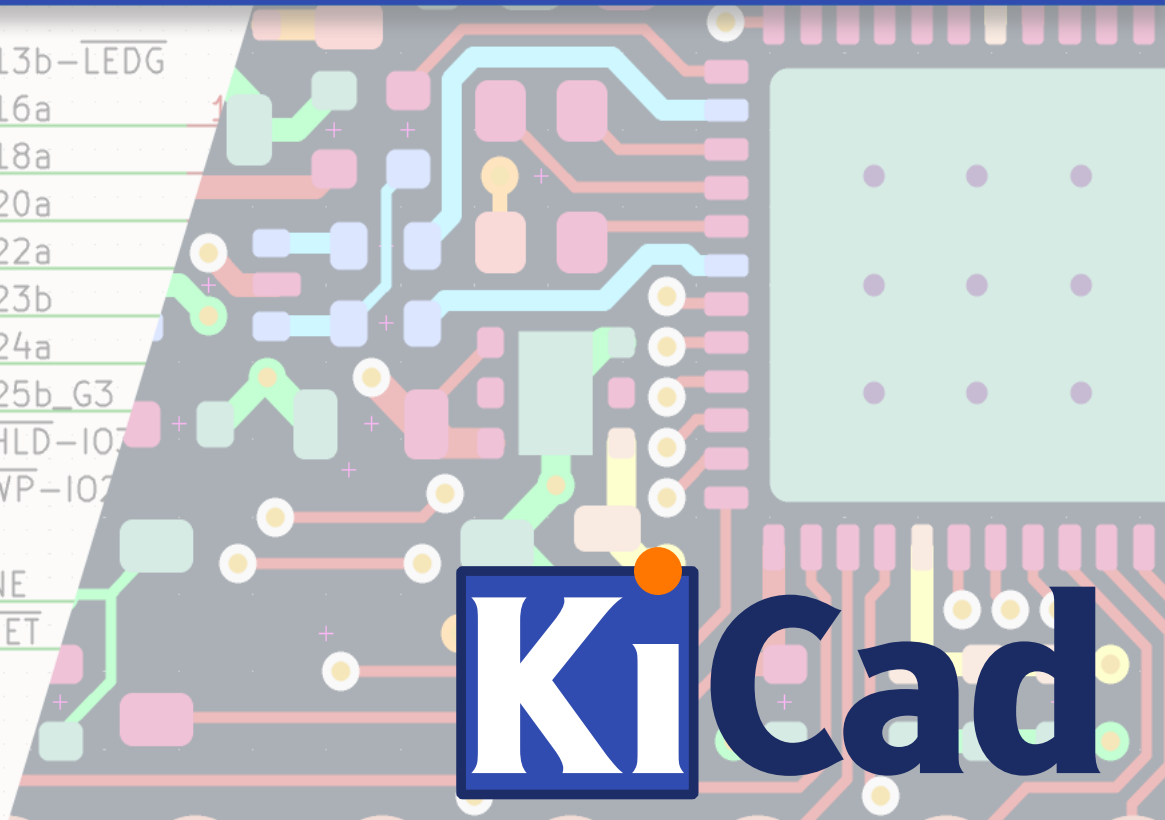
IOB\_13B  
IOB\_16A  
IOB\_18A  
IOB\_20A  
+ DPair IOB\_22A  
- L---IOB\_23B  
+ DPair IOB\_24A  
- L IOB\_25B\_G3  
IOB\_29B  
IOB\_31B

6 IOB\_13b-LEDG  
9 IOB\_16a  
10 IOB\_18a  
11 IOB\_20a  
12 IOB\_22a  
21 IOB\_23b  
13 IOB\_24a  
20 IOB\_25b\_G3  
19 SPI\_HLD-I07  
18 SPI\_WP-I07

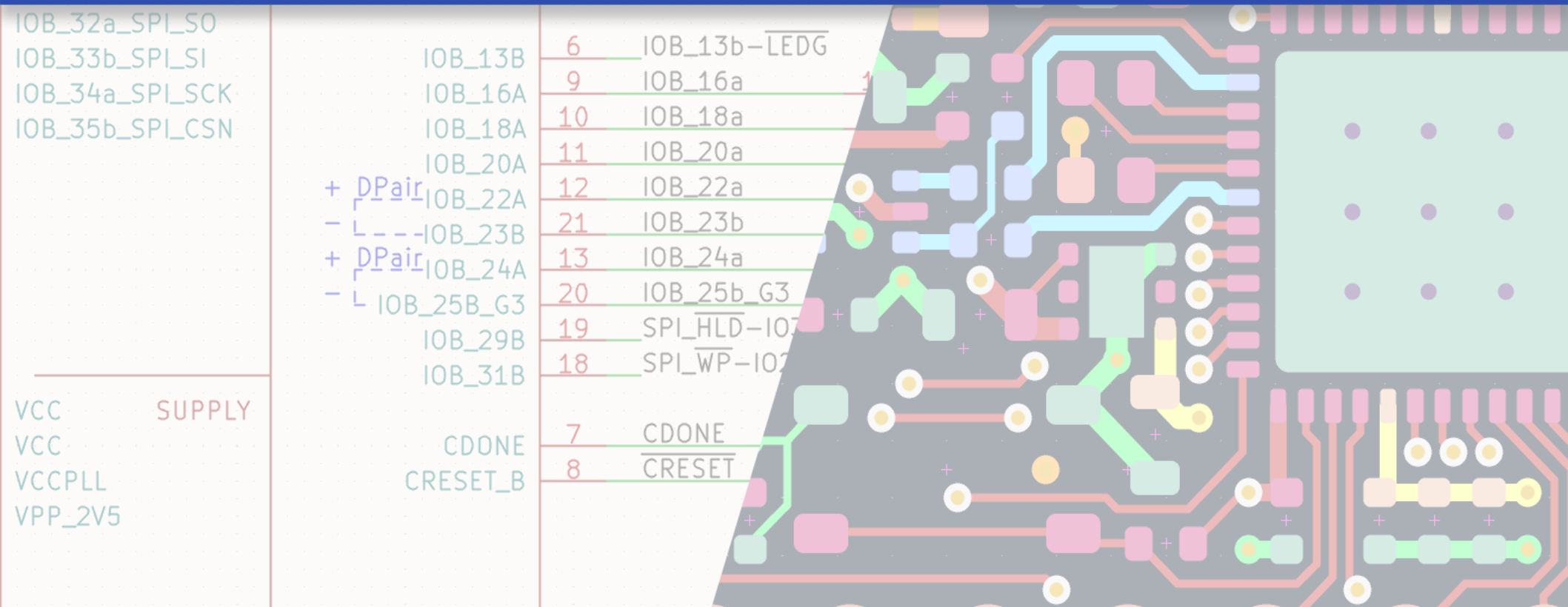
CDONE  
CRESET\_B

7 CDONE  
8 CRESET

VCC SUPPLY  
VCC  
VCCPLL  
VPP\_2V5



# Project News



- Stable version 8 released January 31<sup>st</sup>, 2024!
- Version 8 end of year donation campaign raised over \$200K.
- First KiCon Europe in A Coruña, Spain.
- First KiCon Asia in Shenzhen, China on November 12, 2023.
- Add three new members to the lead development team.
- [Huaqiu NextPCB](#) providing paid full time developers.
- New head librarian and six new library team members.
- Technical writing team member.
- [Würth Elektronik](#) commits to adding entire product line to KiCad libraries.

- Significant increase in usage in Asia.
- Five platinum sponsors.
- Possibly three KiCad conferences annually.
- [KiCad store](#) is now open.



**AISLER** provides fabrication and assembly services with a portion of each order going to support the development of KiCad



**DAI** is an international development consulting firm that works to empower people and institutions in developing countries to build a more a prosperous and equitable future.



**Digi-Key Electronics** offers the world's largest selection of electronic components in stock and available for immediate shipment. From prototype to production, Digi-Key fuels innovation all over the world.

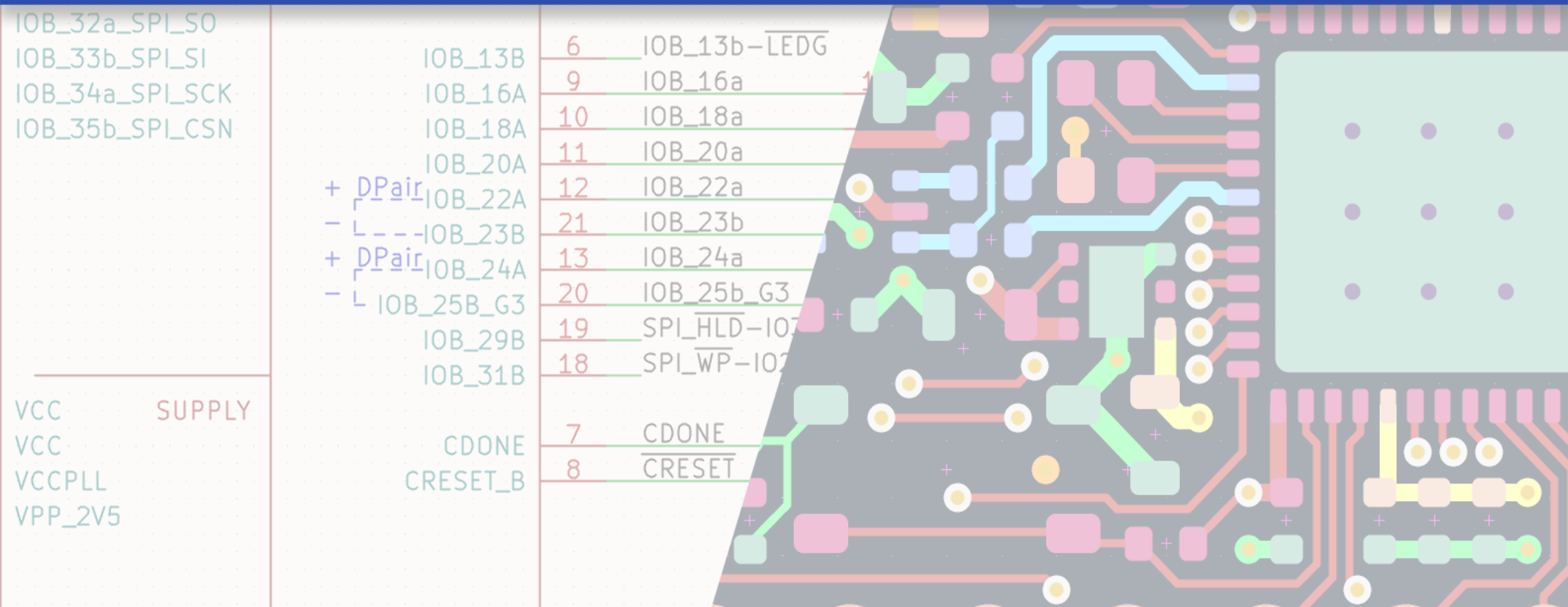


**HQ NextPCB** specializes in reliable multilayer PCB manufacture and assembly. NextPCB is working with KiCad to provide smart tools to simplify the progression from design to physical product.



[KiCad Services Corporation](#) provides direct support to professional KiCad users. They also donate directly to KiCad via the year-end matching grant.

# Version 8





- [SVG export improvements.](#)
- Start up splash screen.
- Alternate hot keys.
- Windows arm64 support.
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- Git integration.

- SVG export improvements.
- Start up splash screen.
- Alternate hot keys.
- Windows arm64 support.
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- Git integration.



- SVG export improvements.
- Start up splash screen.
- [Alternate hot keys.](#)
- Windows arm64 support.
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- Git integration.

- SVG export improvements.
- Start up splash screen.
- Alternate hot keys.
- [Windows arm64 support.](#)
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- Git integration.

The screenshot shows the 'KiCad Downloads' page. At the top, there is a navigation bar with the KiCad logo and the text 'Downloads'. Below this, there is a banner image featuring a colorful PCB layout with various components and traces. The main content area is titled 'Windows Nightlies' and contains a list of download links under the heading 'File'. The links are as follows:

| File  |
|---|
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-x86_64-vcpkg-pdbs.zip</a> |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-arm64-pdbs.zip</a>        |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-x86_64-pdbs.zip</a>       |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-arm64-vcpkg-pdbs.zip</a>  |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-arm64.exe</a>             |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-x86_64.exe</a>            |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-x86_64-lite.exe</a>       |
| <a href="#">kicad-nightly-7.99.0.3493.g1b4e77d53c-arm64-lite.exe</a>        |
| <a href="#">kicad-nightly-7.99.0.3482.gab4d99bcb14-arm64-vcpkg-pdbs.zip</a> |

- SVG export improvements.
- Start up splash screen.
- Alternate hot keys.
- Windows arm64 support.
- [EasyEDA project importer](#).
- ERC and DRC from command line interface (JSON output).
- Git integration.

- SVG export improvements.
- Start up splash screen.
- Alternate hot keys.
- Windows arm64 support.
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- Git integration.

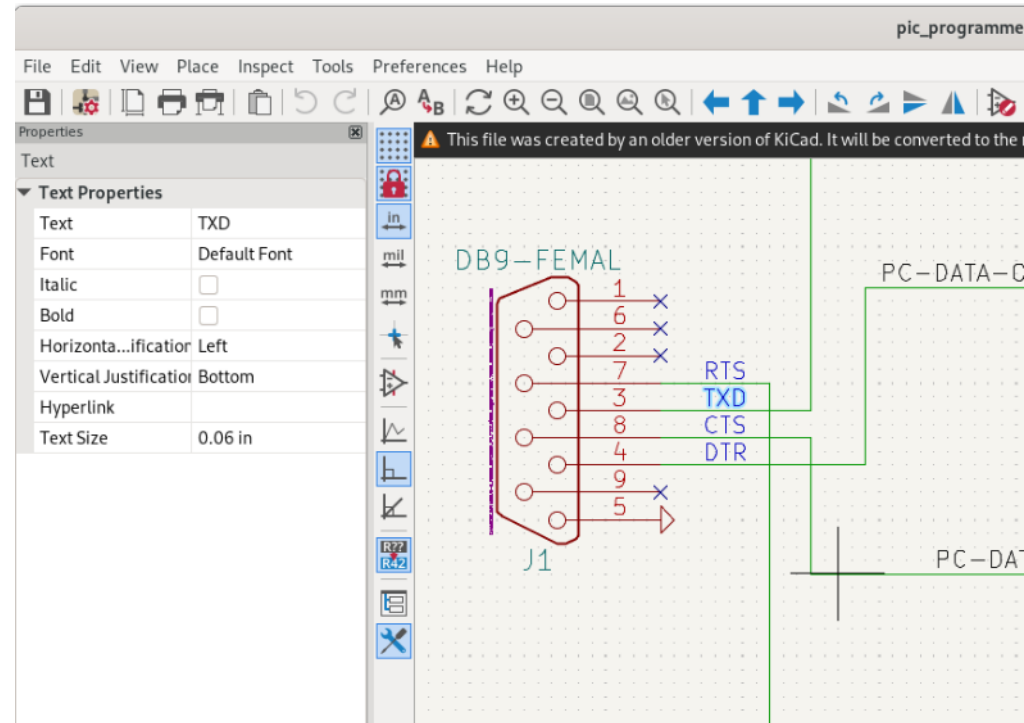
```
$ kicad-cli pcb drc --severity-error ./pic_programmer.kicad_pcb -o pic-programmer-drc.json
Loading board
Running DRC...
Found 16 violations
Found 0 unconnected items
Found 0 schematic parity issues
Saved DRC Report to pic-programmer-drc.json
```

```
** Drc report for ./pic_programmer.kicad_pcb **
** Created on 2023-11-02T14:20:35-0400 **

** Found 16 DRC violations **
[lib_footprint_issues]: Footprint 'DIP-8_W7.62mm_Socket_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(179.0700 mm, 120.6500 mm): Footprint U6
[lib_footprint_issues]: Footprint 'DIP-8_W7.62mm_Socket_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(179.0700 mm, 109.2200 mm): Footprint U1
[lib_footprint_issues]: Footprint 'DIP-8_W7.62mm_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(110.4900 mm, 57.1500 mm): Footprint U4
[lib_footprint_issues]: Footprint 'DIP-28_W7.62mm_Socket_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(207.6450 mm, 50.8000 mm): Footprint P2
[lib_footprint_issues]: Footprint 'DIP-18_W7.62mm_Socket_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(207.8990 mm, 107.9500 mm): Footprint U5
[lib_footprint_issues]: Footprint 'DIP-14_W7.62mm_LongPads' not found in library 'Package_DIP'.
  Local override; warning
  @(115.5700 mm, 119.3800 mm): Footprint U2
[lib_footprint_issues]: Footprint 'DIP-8_Female_Horizontal_P2_77x3.04mm_EdgePinOffset7.70mm_Housed_MountingU2'
```

- SVG export improvements.
- Start up splash screen.
- Alternate hot keys.
- Windows arm64 support.
- EasyEDA project importer.
- ERC and DRC from command line interface (JSON output).
- [Git integration](#).

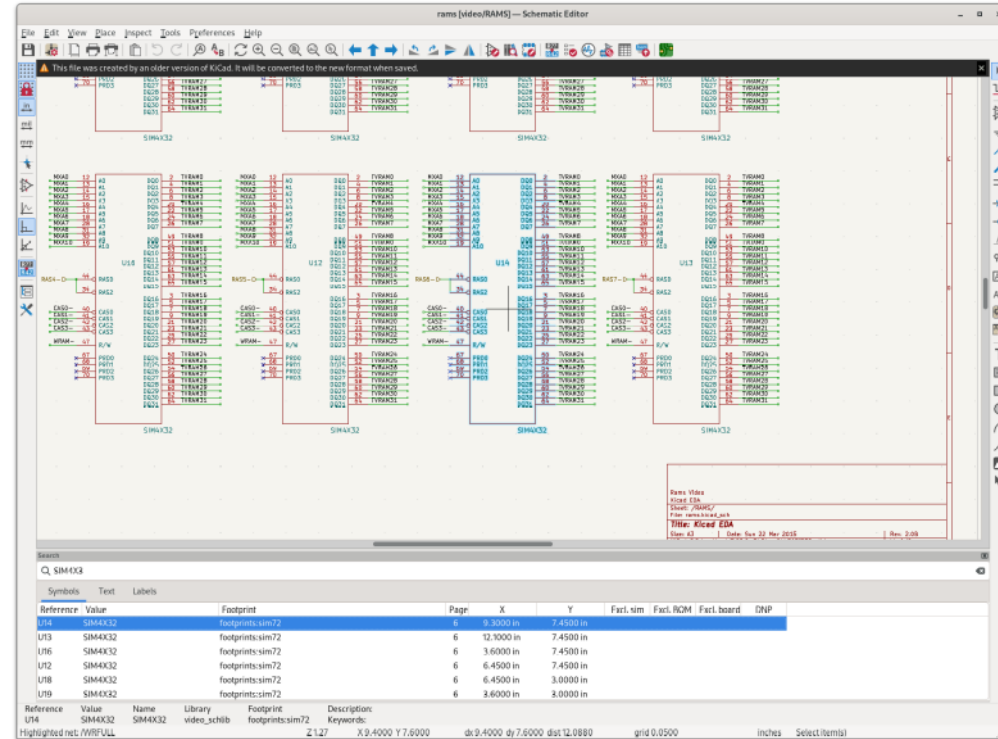
- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.



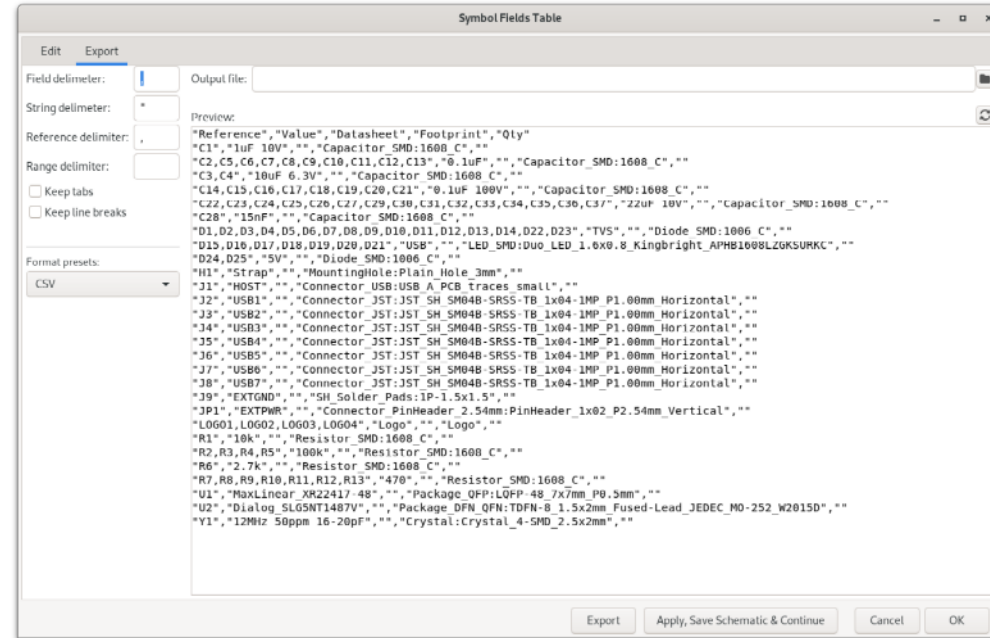


- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.

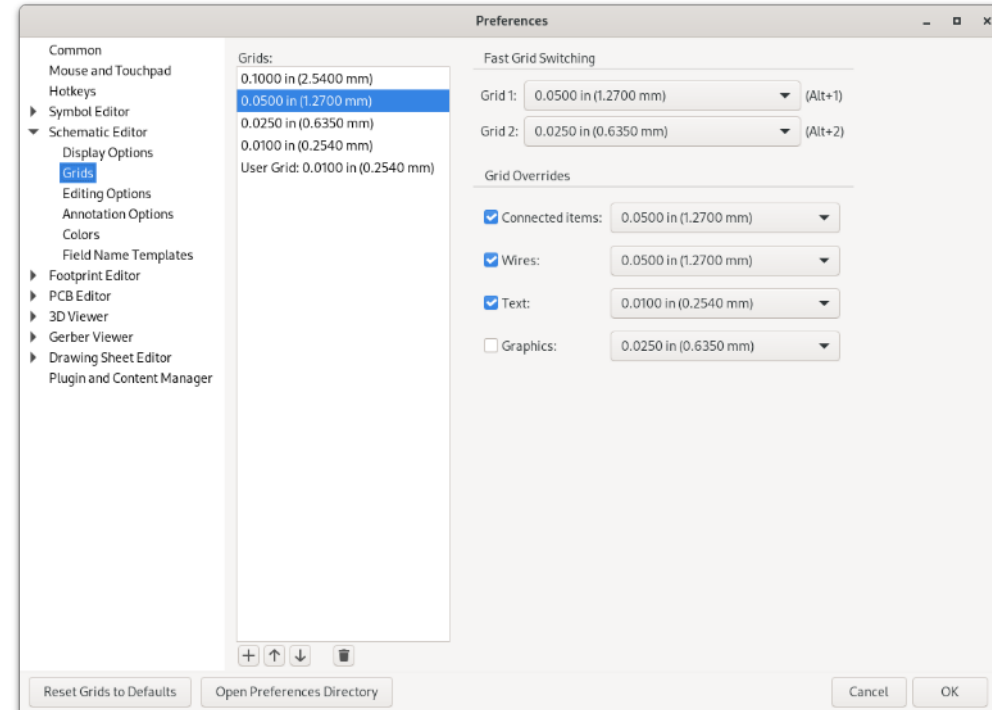
- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.



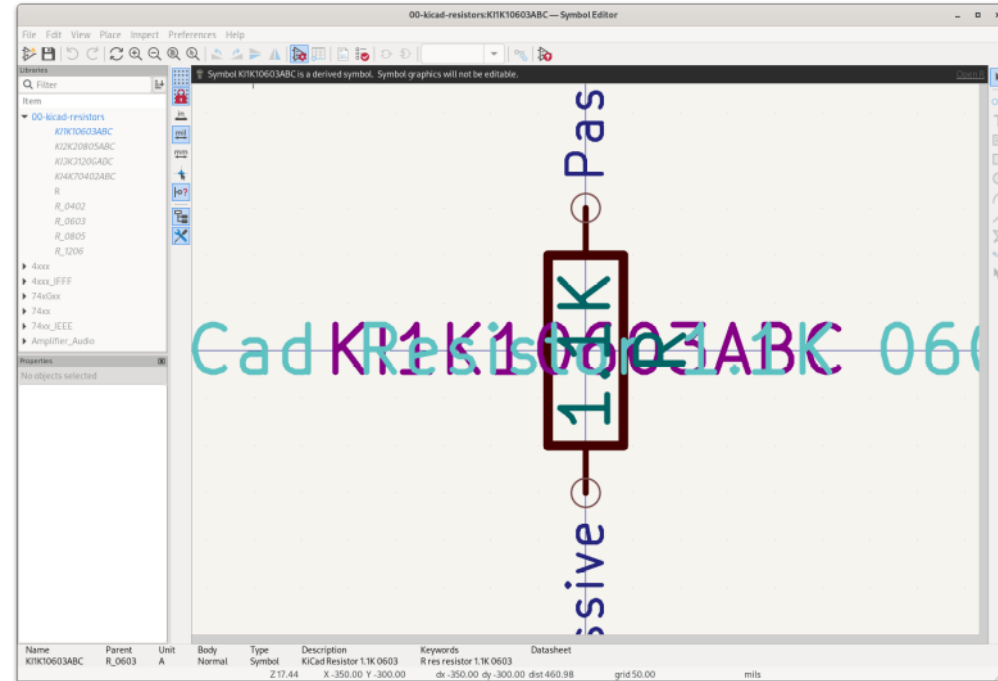
- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.



- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.



- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- **Nested symbol inheritance.**
- Tool to check for symbol library differences in schematic.



- Object property panel for schematic and symbol library editors.
- Highlighted net navigator panel.
- Search panel.
- Internal bill of materials (BOM) tool.
- Object grid alignment in schematic and symbol library editors.
- Nested symbol inheritance.
- Tool to check for symbol library differences in schematic.

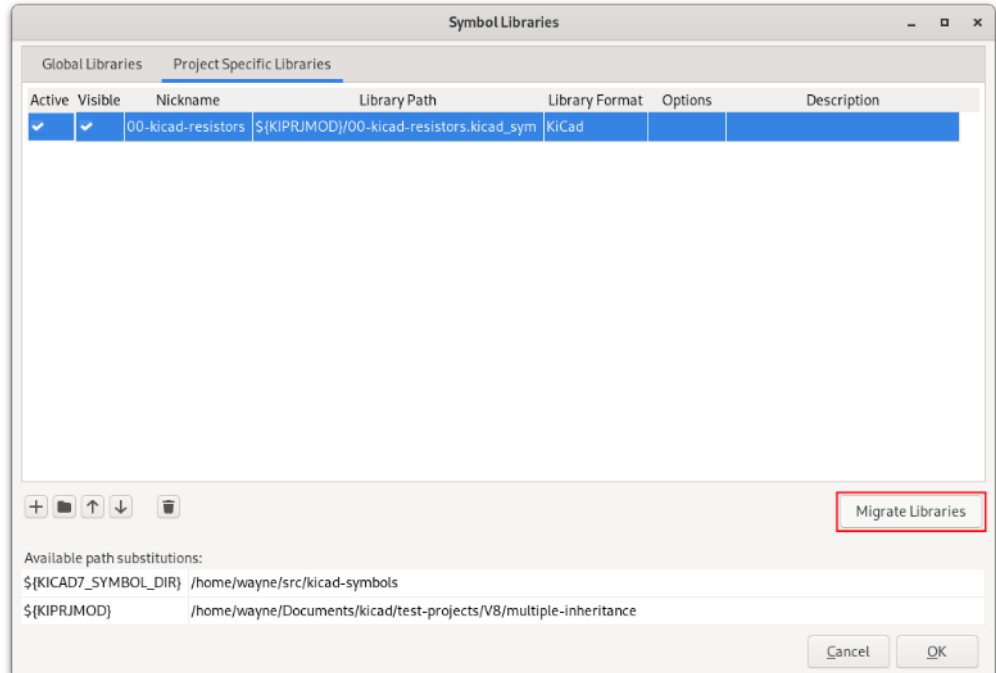
- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

- CADSTAR, Altium, and Eagle symbol library importer.
- [Symbol editor library tree preview.](#)
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

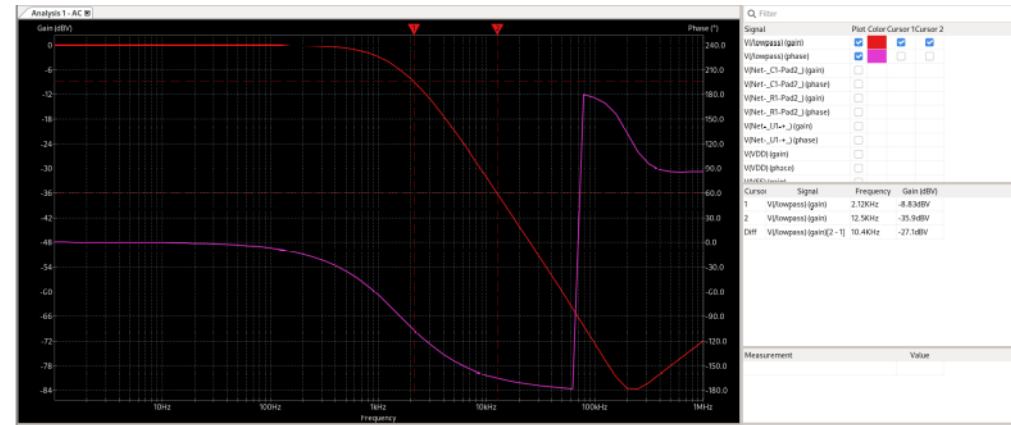


- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- [Symbol library file change watcher.](#)
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

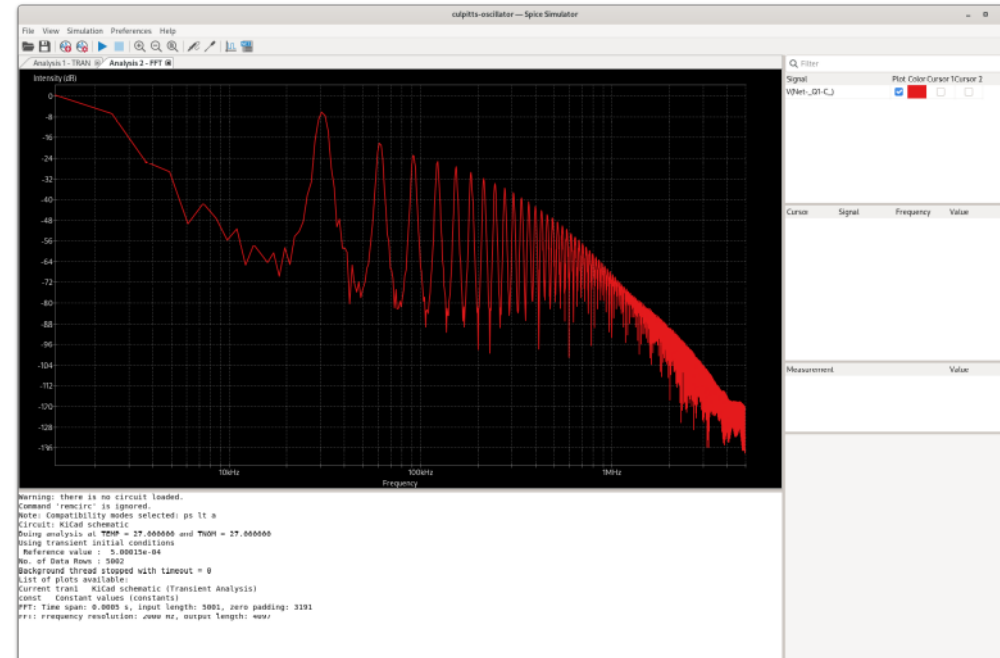


- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.



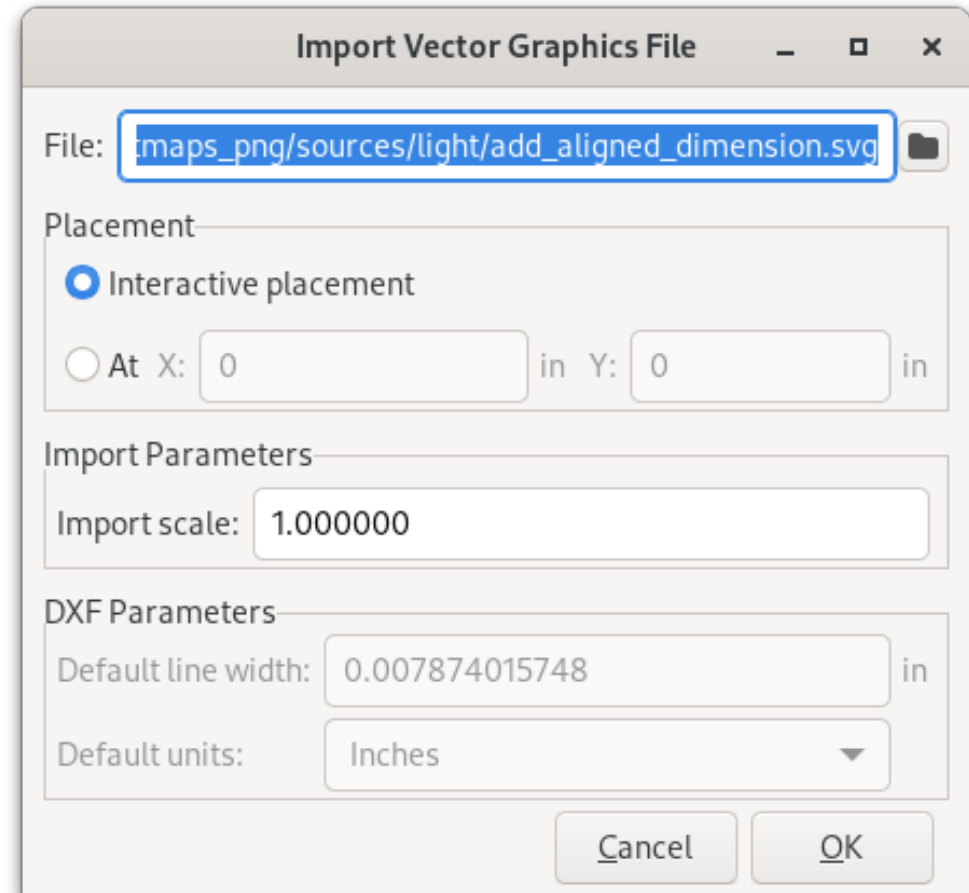
- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- [Import LTSpice schematics.](#)
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- SPICE simulator improvements (FFT, S-parameters, Fourier, etc.).
- Editable power symbols.

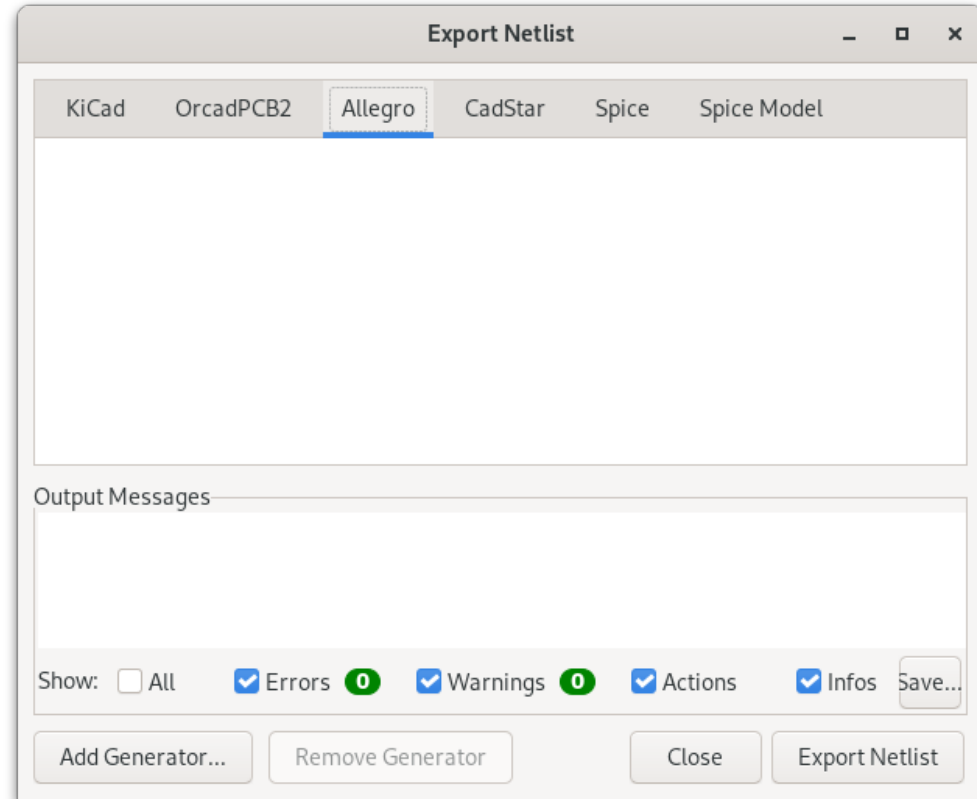


- CADSTAR, Altium, and Eagle symbol library importer.
- Symbol editor library tree preview.
- Symbol library file change watcher.
- Symbol library editor migrate third party and legacy libraries.
- Differential cursors in simulator.
- Import LTSpice schematics.
- Simulator improvements (FFT, S-parameters, Fourier, etc.).
- [Editable power symbols.](#)

- SVG and DXF import in schematic and symbol editors.
- Cadence Allegro PCB Designer netlist exporter.



- SVG and DXF import in schematic and symbol editors.
- Cadence Allegro PCB Designer netlist exporter.



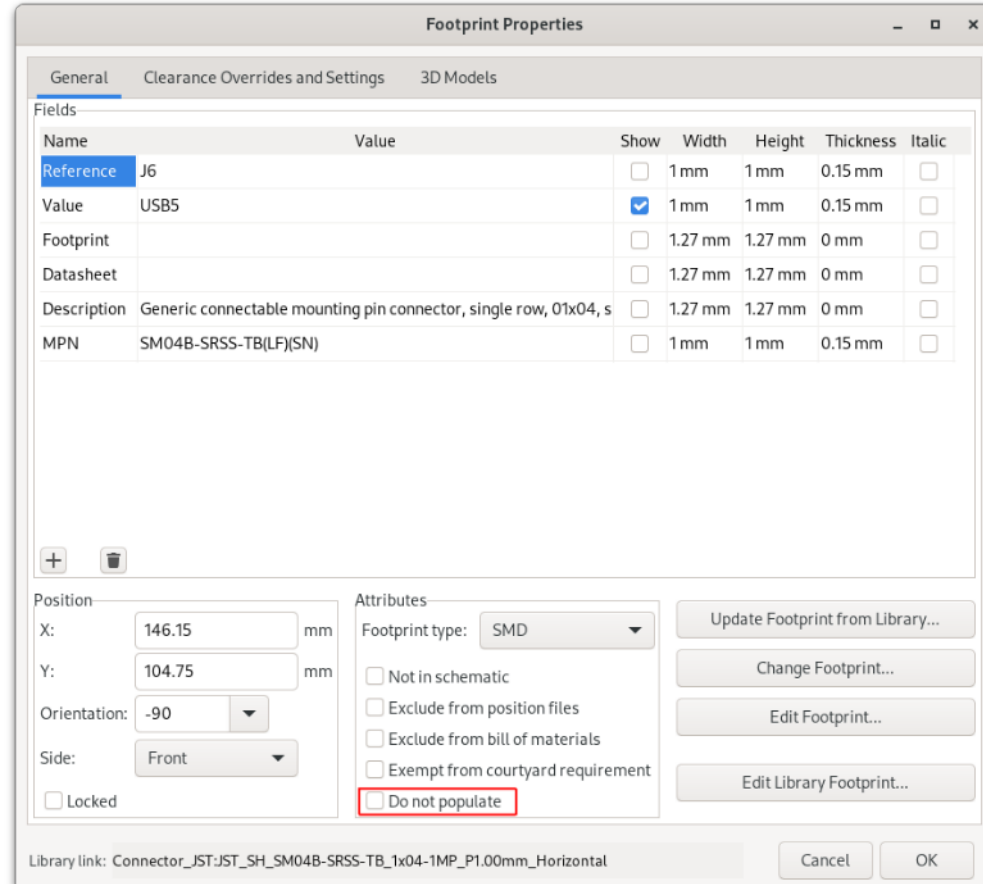


- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.

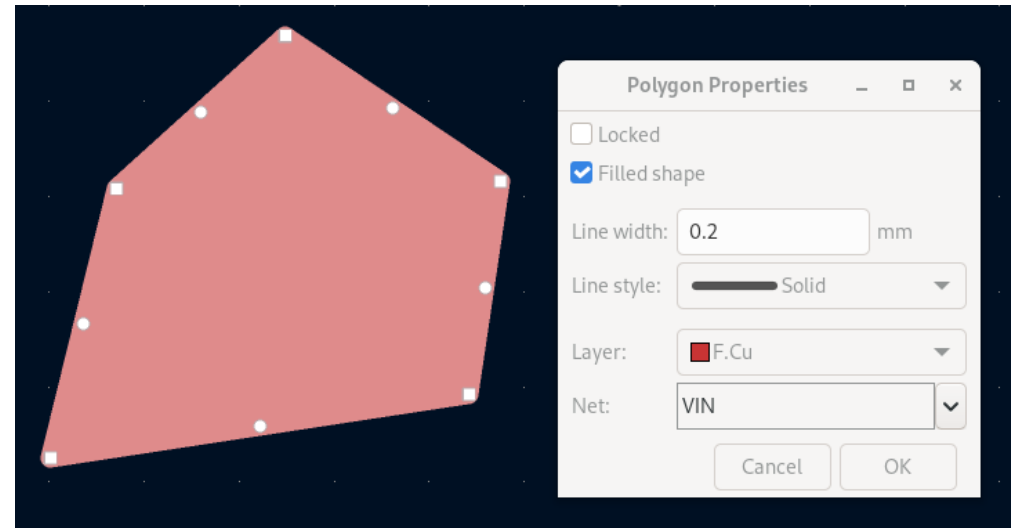
- Tool to check for footprint library changes in PCB editor.
- [Altium footprint library importer](#).
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.

- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- [Import SolidWorks PCB files.](#)
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.

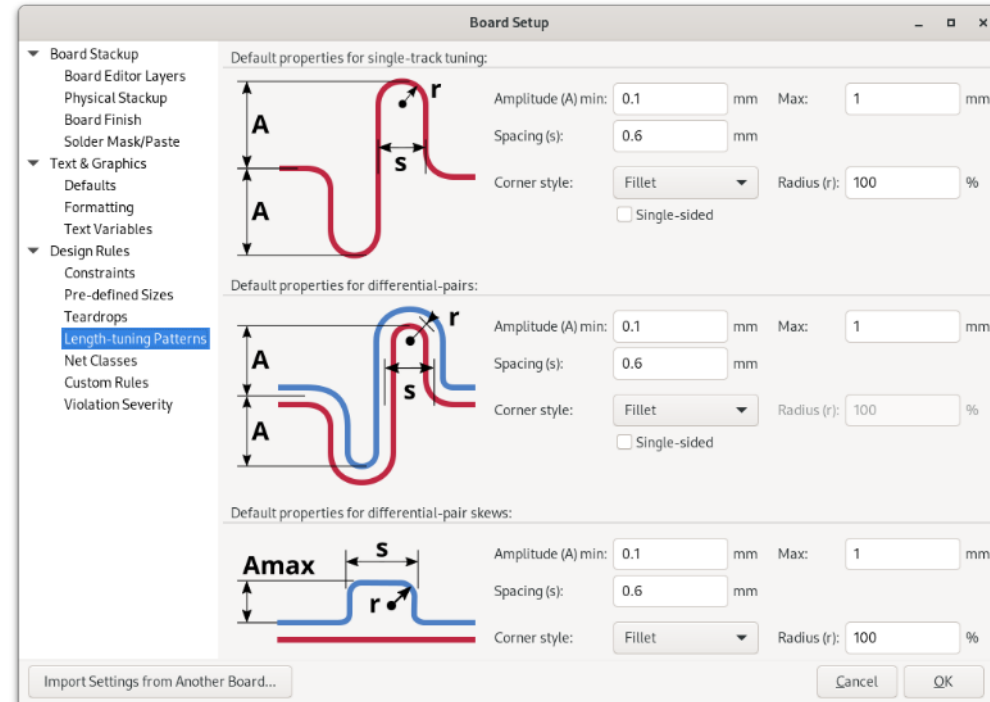
- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.



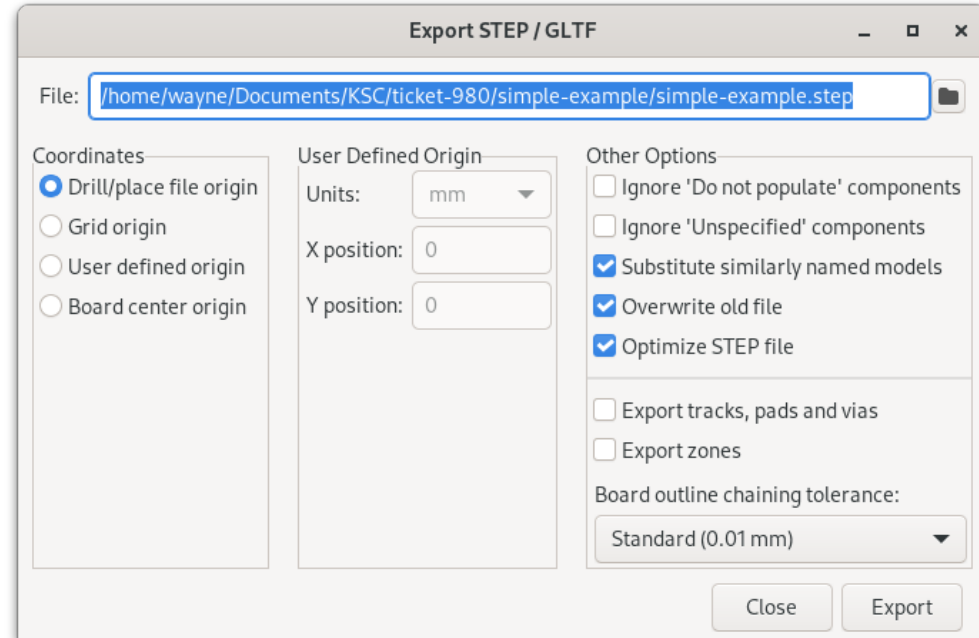
- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.



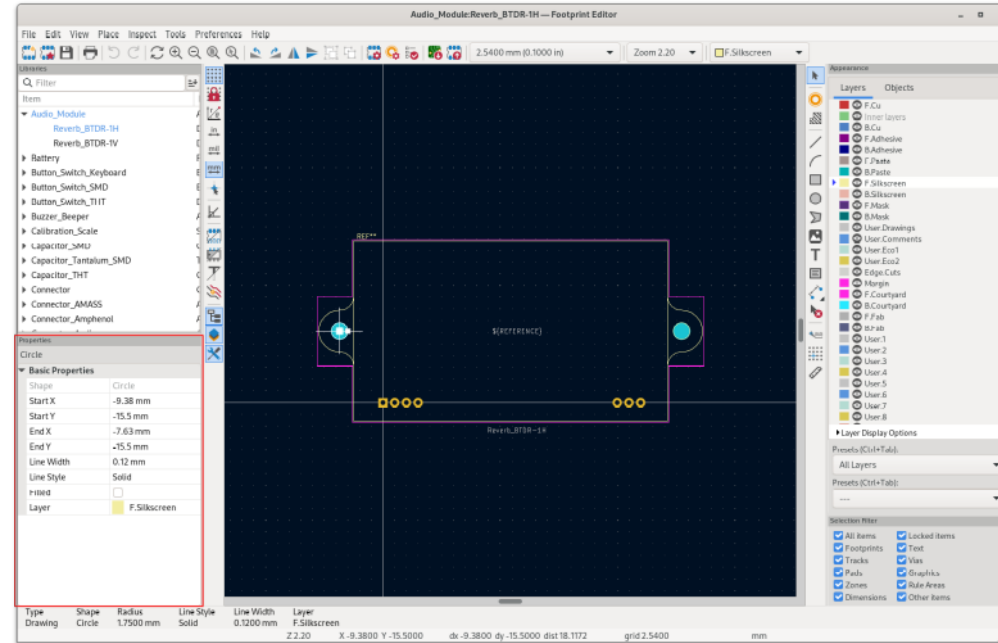
- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- **Interactive meander tuning.**
- STEP export improvements.
- Footprint editor properties panel.



- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- [STEP export improvements.](#)
- Footprint editor properties panel.



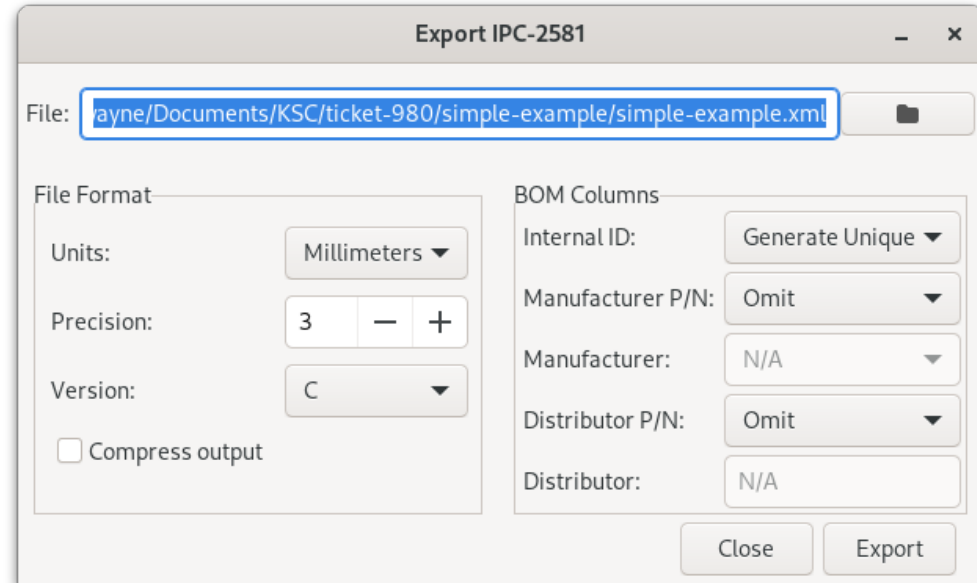
- Tool to check for footprint library changes in PCB editor.
- Altium footprint library importer.
- Import SolidWorks PCB files.
- Do not populate flag for position file export.
- Allow connectivity to graphic shapes on copper layers.
- Interactive meander tuning.
- STEP export improvements.
- Footprint editor properties panel.



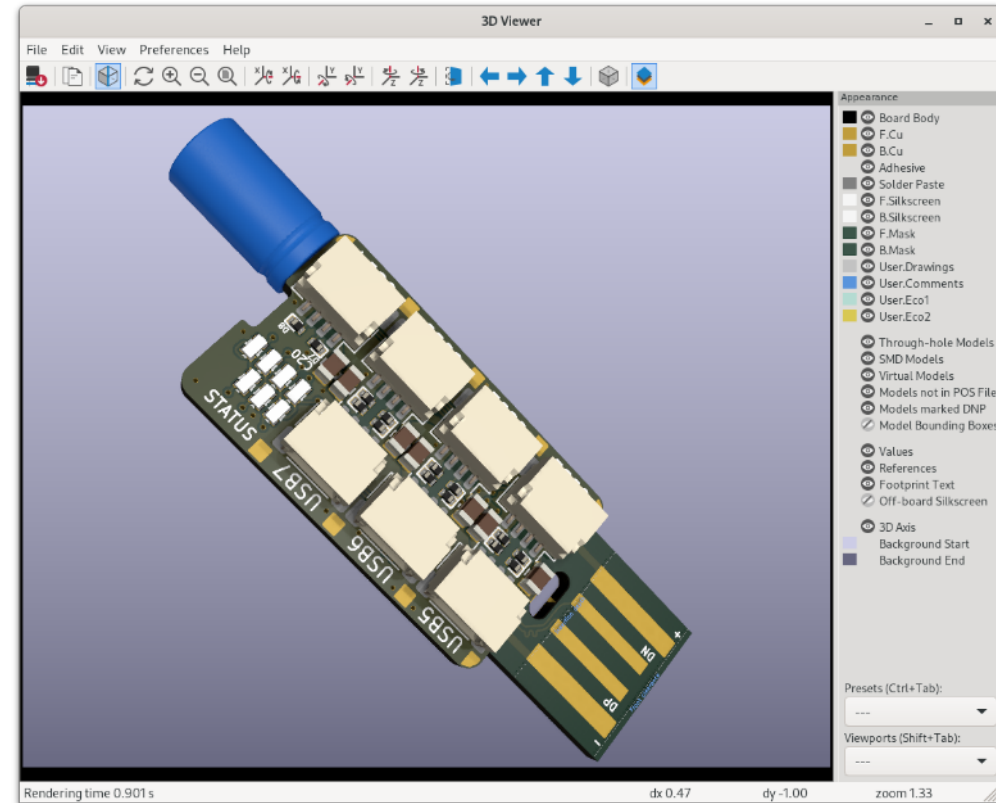


- Footprint editor library tree preview.
- IPC-2581 export.

- Footprint editor library tree preview.
- IPC-2581 export.



- 3D viewer improvements.



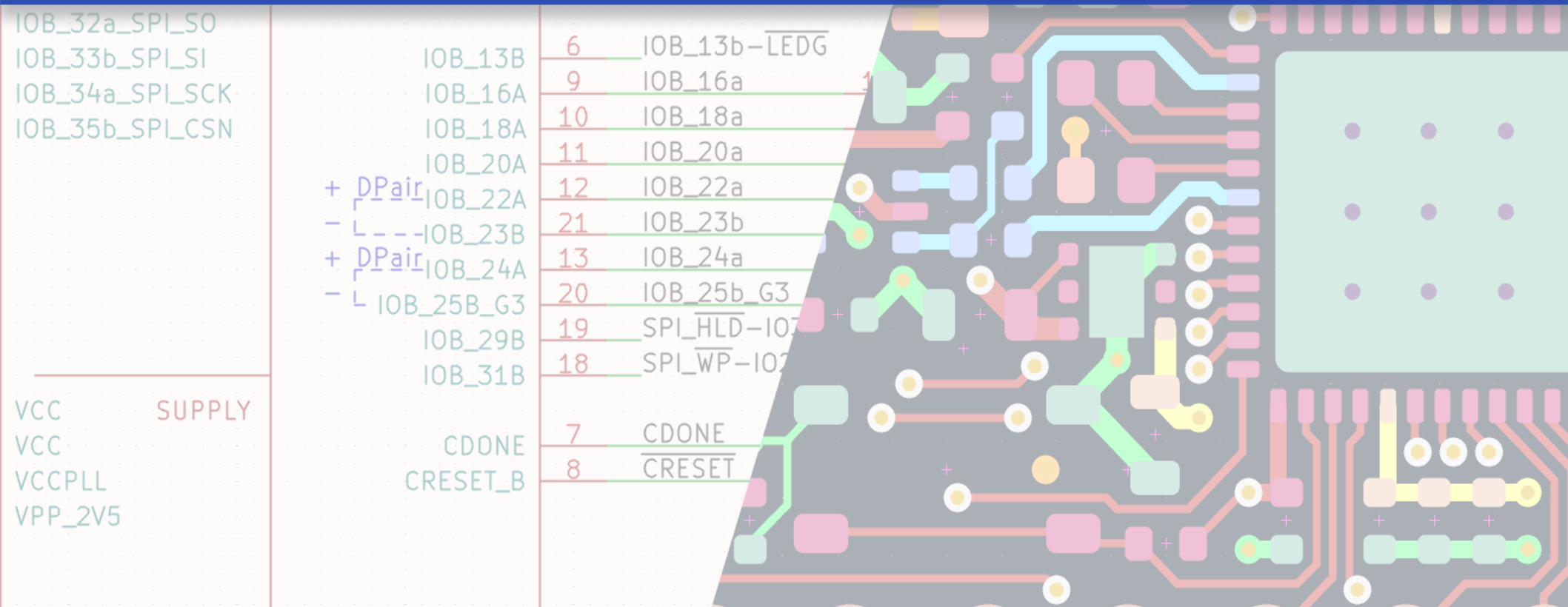
# Version 8 Supported Third Party Tools

|                       | Project | Schematic | PCB | Symbols | Footprints |
|-----------------------|---------|-----------|-----|---------|------------|
| Altium Designer       |         | ●         | ●   | ●       | ●          |
| Altium Circuit Maker  |         | ●         | ●   | ●       | ●          |
| Altium Circuit Studio |         | ●         | ●   | ●       | ●          |
| CADSTAR               | ●       | ●         | ●   | ●       | ●          |
| EAGLE                 | ●       | ●         | ●   | ●       | ●          |
| EasyEDA / JLCEDA Std  | ●       | ●         | ●   | ●       | ●          |
| EasyEDA / JLCEDA Pro  | ●       | ●         | ●   | ●       | ●          |
| Fabmaster             | N/A     | N/A       | ●   | N/A     | N/A        |
| gEDA                  |         |           |     |         | ●          |
| LTspice               | N/A     | ●         | N/A |         | N/A        |
| P-CAD                 |         |           | ●   |         |            |
| Solidworks PCB        |         | ●         | ●   | ●       | ●          |

Key: ● Supported in KiCad 7.0      ● Supported in KiCad 8.0

- Source repository: 4500 commits by 15 authors.
- 1.63M lines of code (without translations), 176K lines comments.
- Symbol library: 358 commits, added 1077, total 20233.
- Footprint library: 288 commits, added 507, total 13355.
- 3D model library: 62 commits, added 215, total 6465.
- Translations: 9 languages 99%+ translated, 17 languages 99%+ in V7 :(.
- Aisler monthly usage stats for December 2023 shows KiCad at 42%.
- OSHPARK 2023 usage was 37%.

# Version 9



- IPC support for schematic and board editor interface.
- Customizable user interface (including toolbars) layouts.
- Visual diff/merge tool for Git.
- Embedded licenses for library and project files.
- Barcode support.
- Multi-user editing support.
- PADS project importer.
- gEDA project importer.

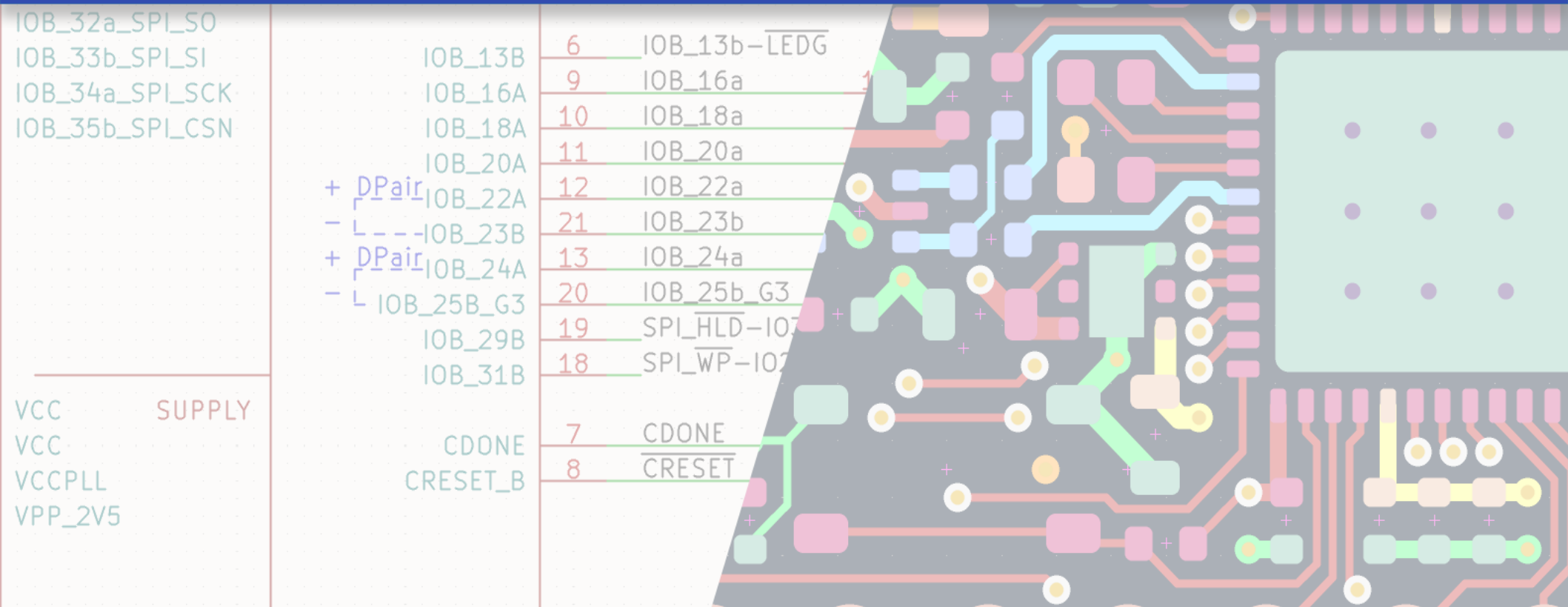
- Sheet pin and hierarchical label synchronization tool.
- Replace shared schematics with reusable schematics.
- Design blocks.
- Schematic variants.
- Table editing tool.
- Bezier curve editing tool.



- [Zone manager](#).
- Multi-channel designs.
- Pad stacks.
- Guard rings.
- Flexible grid system.
- Bezier curve editing tool.
- Table editing tool.
- Embedded 3D models.
- ODB++ export.

- Electromagnetic (EM) simulation.

# Wrap Up



**Thanks to all of the developers who contribute their valuable time and talent to the KiCad project.**

**Thank you to all of our sponsors and everyone who has generously donated to KiCad.**

**Thank you for your interest and continued support of the KiCad project.**

**Special thanks to Seth Hillbrand for organizing this FOSDEM developers room.**

**Hope to see you all at a KiCad conference in 2024.**