• Stable version 8 released January 31st, 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 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.
<table>
<thead>
<tr>
<th>Pin</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>IOB_13B</td>
</tr>
<tr>
<td>2</td>
<td>IOB_16A</td>
</tr>
<tr>
<td>3</td>
<td>IOB_18A</td>
</tr>
<tr>
<td>4</td>
<td>IOB_20A</td>
</tr>
<tr>
<td>5</td>
<td>IOB_22A</td>
</tr>
<tr>
<td>6</td>
<td>IOB_13b_LEDG</td>
</tr>
<tr>
<td>7</td>
<td>IOB_18a</td>
</tr>
<tr>
<td>8</td>
<td>IOB_18a</td>
</tr>
<tr>
<td>9</td>
<td>IOB_20a</td>
</tr>
<tr>
<td>10</td>
<td>IOB_22a</td>
</tr>
<tr>
<td>11</td>
<td>IOB_23B</td>
</tr>
<tr>
<td>12</td>
<td>IOB_24a</td>
</tr>
<tr>
<td>13</td>
<td>IOB_24a</td>
</tr>
<tr>
<td>14</td>
<td>IOB_25b_G3</td>
</tr>
<tr>
<td>15</td>
<td>SPI_HLD-IOT</td>
</tr>
<tr>
<td>16</td>
<td>SPI_WP-IOT</td>
</tr>
<tr>
<td>17</td>
<td>SUPPLY</td>
</tr>
<tr>
<td>18</td>
<td>CDONE</td>
</tr>
<tr>
<td>19</td>
<td>VCC</td>
</tr>
<tr>
<td>20</td>
<td>VCC_PLL</td>
</tr>
<tr>
<td>21</td>
<td>VPP_2V5</td>
</tr>
<tr>
<td>22</td>
<td>CRESET</td>
</tr>
<tr>
<td>23</td>
<td>CRESET_B</td>
</tr>
</tbody>
</table>
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Common Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.
Version 8 Schematic Editor Features

- 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.**
Version 8 Schematic Editor Features

- SVG and DXF import in schematic and symbol editors.
- Cadence Allegro PCB Designer netlist exporter.
Version 8 Schematic Editor Features

- SVG and DXF import in schematic and symbol editors.
- Cadence Allegro PCB Designer netlist exporter.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

- 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.
Version 8 Board Editor Features

• 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.
Version 8 Board Editor Features

• 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.
Version 8 Board Editor Features

- Footprint editor library tree preview.
- IPC-2581 export.
Version 8 Board Editor Features

- Footprint editor library tree preview.
- IPC-2581 export.
• 3D viewer improvements.
## Version 8 Supported Third Party Tools

<table>
<thead>
<tr>
<th>Tool</th>
<th>Project</th>
<th>Schematic</th>
<th>PCB</th>
<th>Symbols</th>
<th>Footprints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Altium Designer</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>Altium Circuit Maker</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>Altium Circuit Studio</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>CADSTAR</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>EAGLE</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>EasyEDA / JLCEDA Std</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>EasyEDA / JLCEDA Pro</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
<tr>
<td>Fabmaster</td>
<td>N/A</td>
<td>N/A</td>
<td>⬤</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>gEDA</td>
<td>⬤</td>
<td></td>
<td></td>
<td></td>
<td>⬤</td>
</tr>
<tr>
<td>LTspice</td>
<td>N/A</td>
<td>⬤</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>P-CAD</td>
<td></td>
<td>⬤</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solidworks PCB</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
<td>⬤</td>
</tr>
</tbody>
</table>

**Key:**
- ⬤ Supported in KiCad 7.0
- ⬤ Supporting in KiCad 8.0
Version 8 Statistics

- 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%.
<table>
<thead>
<tr>
<th>Component</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>IOB_32a_SPI_S0</td>
<td></td>
</tr>
<tr>
<td>IOB_33b_SPI_SI</td>
<td></td>
</tr>
<tr>
<td>IOB_34a_SPI_SCK</td>
<td></td>
</tr>
<tr>
<td>IOB_35b_SPI_CSN</td>
<td></td>
</tr>
<tr>
<td>IOB_13B</td>
<td></td>
</tr>
<tr>
<td>IOB_16A</td>
<td></td>
</tr>
<tr>
<td>IOB_18A</td>
<td></td>
</tr>
<tr>
<td>IOB_20A</td>
<td></td>
</tr>
<tr>
<td>IOB_22A</td>
<td></td>
</tr>
<tr>
<td>IOB_23B</td>
<td></td>
</tr>
<tr>
<td>IOB_24A</td>
<td></td>
</tr>
<tr>
<td>IOB_25B_G3</td>
<td></td>
</tr>
<tr>
<td>IOB_29B</td>
<td></td>
</tr>
<tr>
<td>IOB_31B</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>IOB_13b_LEDG</td>
</tr>
<tr>
<td>9</td>
<td>IOB_16a</td>
</tr>
<tr>
<td>10</td>
<td>IOB_18a</td>
</tr>
<tr>
<td>11</td>
<td>IOB_20a</td>
</tr>
<tr>
<td>12</td>
<td>IOB_22a</td>
</tr>
<tr>
<td>21</td>
<td>IOB_23b</td>
</tr>
<tr>
<td>13</td>
<td>IOB_24a</td>
</tr>
<tr>
<td>20</td>
<td>IOB_25b_G3</td>
</tr>
<tr>
<td>19</td>
<td>SPI_HLD_IO</td>
</tr>
<tr>
<td>18</td>
<td>SPI_WP_IO</td>
</tr>
<tr>
<td>7</td>
<td>CDONE</td>
</tr>
<tr>
<td>8</td>
<td>CRESET</td>
</tr>
</tbody>
</table>

**Legend:**
- **VCC:** SUPPLY
- **VCCPLL:**
- **VPP_2V5:**
Version 9 Common Features

- 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.
Version 9 Schematic Editor Features

- Sheet pin and hierarchical label synchronization tool.
- Replace shared schematics with reusable schematics.
- Design blocks.
- Schematic variants.
- Table editing tool.
- Bezier curve editing tool.
Version 9 Board Editor Features

- 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.