

# LibrePCB

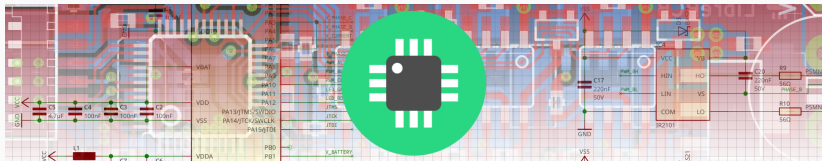
A new, powerful and intuitive EDA tool for everyone

---

Urban Bruhin

February 1, 2020

# About LibrePCB



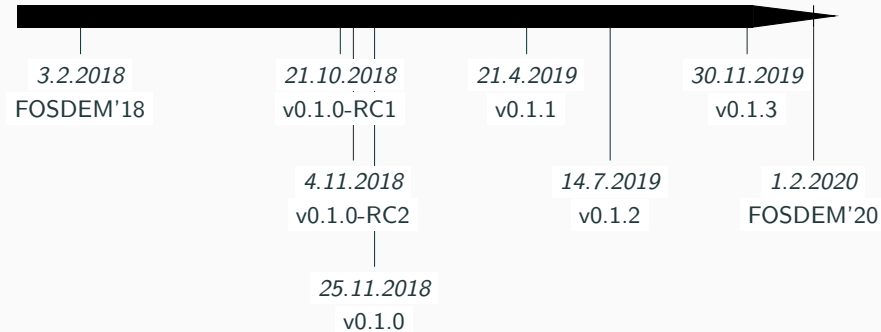
## Free/OpenSource EDA Suite

- Multiplatform   
- Written from scratch in C++11/Qt5
- Development started in 2013
- Website: <https://librepcb.org/>
- GitHub: <https://github.com/LibrePCB/LibrePCB>

# Timeline

2018

2020



# Library Editor Base Rule Check

Like a DRC, but for library elements (symbols, footprints, ...).

The screenshot shows the LibrePCB Base - LibrePCB Library Editor interface. A yellow warning banner at the top reads: "WARNING: This library element contains errors, see exact messages below. You should fix these errors before saving it, otherwise the library element may not work as expected." Below the warning is a footprint diagram with a red triangle indicating an error. The right panel shows the element's properties, including Name, Description, Pitch, Lead span, Height, Lead size, Thermal tab size, Keywords, Author, Version, and Categories. The Messages panel at the bottom right lists several errors: "No categories set", "Missing text '({VALUE})' in 'default'", "Clearance of pad '1' in 'default'", and "Author not set".

File Edit View Tools Help

LibrePCB Base \*SOT223-4P230\_700X180L175X75T175X300

**WARNING: This library element contains errors, see exact messages below. You should fix these errors before saving it, otherwise the library element may not work as expected.**

Name: 223-4P230\_700X180L175X75T175X300  
Description: SOT-223-4 package (EIAJ SC-73)  
Pitch: 2.3mm  
Lead span: 7.0mm  
Height: max 1.8mm  
Lead size: 1.75x0.75mm  
Thermal tab size: 1.75x3mm  
Keywords: sot223,sot-223,sc73,sc-73  
Author:  
Version: 0,1  
Deprecated:  Package should no longer be used.  
Categories:

Add category Remove selected

Messages: No categories set  Fix ?  
 Missing text '({VALUE})' in 'default'  Fix ?  
 Clearance of pad '1' in 'default'  Fix ?  
 Author not set  Fix ?

X: 6.842964mm Y: 5.753537mm

← Warning

← Messages

## Use-Cases

- Use CI to make sure all library elements are valid
- Check pull requests for library convention violations (ToDo)

## Usage

```
$ librepcb-cli open-library --all --strict MyLibrary.lplib
Open library 'MyLibrary.lplib'...
Process 86 component categories...
Process 44 package categories...
Process 37 symbols...
Process 492 packages...
Process 34 components...
Process 37 devices...
SUCCESS
```

# CLI for Libraries

LibrePCB-Libraries / LibrePCB\_Base.lplib

Unwatch 4

Star 2

Fork 6

Code Issues 2 Pull requests 2 Actions Security Insights Settings

## Add sym/cmp for solder jumpers 2P/3P #52

Edit

Merged ubruhin merged 1 commit into master from solder-jumper on 8 Dec 2019

Conversation 4 Commits 1 Checks 1 Files changed 6

+145 -0

✓ Add sym/cmp for solder jumpers 2P/3P f8bc024

CI  
on: push

✓ check

CI / check

succeeded on 5 Dec 2019 in 40s

Search logs

```
▶ ✓ Set up job 1s
▶ ✓ Initialize containers 14s
▶ ✓ Run actions/checkout@v1 3s
▶ ✓ Run librepcb-cli open-library --all --strict 21s
  1 ▶ Run librepcb-cli open-library --all --strict .
  4 Open library 'LibrePCB_Base.lplib'...
  5 Process 87 component categories...
  6 Process 52 package categories...
  7 Process 59 symbols...
  8 Process 603 packages...
  9 Process 47 components...
 10 Process 38 devices...
 11 SUCCESS
▶ ✓ Stop containers 1s
▶ ✓ Complete job 0s
```

## Use-Cases

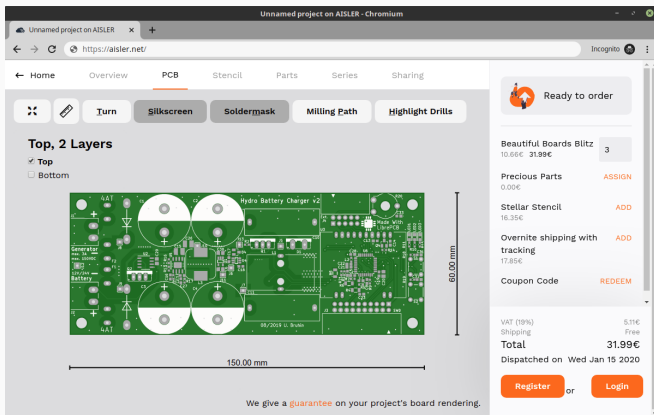
- Use CI to make sure there are no ERC/DRC issues
- Automatically generate output files (Gerber, BOM, PDF, ...)

## Usage

```
$ librepcb-cli open-project --erc --export-pcb-fabrication-data MyProject.lpp
Open project 'MyProject.lpp'...
Run ERC...
  Approved messages: 7
  Non-approved messages: 2
    - [WARNING] Net signal connected to less than two pins: "CAN_RX"
    - [WARNING] Net signal connected to less than two pins: "JTCK"
Export PCB fabrication data...
  Board 'default':
    => 'output/v1/gerber/MyProject_DRILLS-PTH.drl'
    => 'output/v1/gerber/MyProject_OUTLINES.gbr'
    => 'output/v1/gerber/MyProject_COPPER-TOP.gbr'
    => 'output/v1/gerber/MyProject_COPPER-BOTTOM.gbr'
    => 'output/v1/gerber/MyProject_SOLDERMASK-TOP.gbr'
    => 'output/v1/gerber/MyProject_SOLDERMASK-BOTTOM.gbr'
    => 'output/v1/gerber/MyProject_SILKSCREEN-TOP.gbr'
    => 'output/v1/gerber/MyProject_SILKSCREEN-BOTTOM.gbr'
Finished with errors!
```

# AISLER Supports LibrePCB

- You can upload LibrePCB projects to <https://aisler.net>
- For every order of a LibrePCB project, AISLER makes a donation to LibrePCB!



The screenshot displays the AISLER website interface. The top navigation bar includes links for Home, Overview, PCB (selected), Stencil, Parts, Series, and Sharing. Below this, there are tool icons and buttons for Turn, Silkscreen, Soldermask, Milling Path, and Highlight Drills. The main content area shows a PCB rendering titled "Top, 2 Layers" with a checked "Top" layer. The rendering is a green PCB with various components and dimensions: 150.00 mm width and 60.00 mm height. A shopping cart on the right side contains items like "Beautiful Boards Blitz" (3 items, 10.66€ to 31.99€), "Precious Parts" (0.00€, ASSIGN), "Stellar Stencil" (16.35€, ADD), "Overnite shipping with tracking" (17.85€, ADD), and "Coupon Code" (REDEEM). The cart summary shows VAT (19%) at 5.11€, Shipping as Free, and a Total of 31.99€. The dispatch date is Wed Jan 15 2020. At the bottom, there are "Register" and "Login" buttons.

Unnamed project on AISLER - Chromium  
https://aisler.net/

Home Overview **PCB** Stencil Parts Series Sharing

Turn Silkscreen Soldermask Milling Path Highlight Drills

**Top, 2 Layers**  
 Top  
 Bottom

150.00 mm  
60.00 mm

Hydro Battery Charger v2

4 A1  
Generator  
4 A1  
18/2011 Li-Iu-Ah

Ready to order

Beautiful Boards Blitz 3  
10.66€ 31.99€

Precious Parts 0.00€ ASSIGN

Stellar Stencil 16.35€ ADD

Overnite shipping with tracking 17.85€ ADD

Coupon Code REDEEM

VAT (19%) 5.11€  
Shipping Free  
**Total 31.99€**  
Dispatched on Wed Jan 15 2020

Register or Login

We give a **guarantee** on your project's board rendering.



## Download

<https://librepcb.org/download/>

- Windows: Installer, ZIP archive
- Linux: Installer, Flatpak, Applmage, binary archive
  - Arch Linux: AUR packages
    - NixOS: Nix package
  - OpenPandora (ARMv7): PND package
    - macOS: Installer, Bundle
    - BSD: FreeBSD port

## Documentation




[https://docs.librepcb.org/getting\\_started/](https://docs.librepcb.org/getting_started/)

**Demo Time!**




# Project Status

Library Management		
Library Editor		
Schematic Editor		(except missing copy&paste)
Board Editor		(usability, no 3D viewer, ...)
Export (e.g. Gerber)		
Available Libraries		

## Priority 1: Improve usability

- Add various small features (custom pad shape, blind/buried vias, slotted holes/pads, ...)
- Improve board editor (usability issues, push&shove router, ...)
- Clipboard    in schematic-/board editors
- Extend part libraries

## Priority 1: Improve usability

- Add various small features (custom pad shape, blind/buried vias, slotted holes/pads, ...)
- Improve board editor (usability issues, push&shove router, ...)
- Clipboard    in schematic-/board editors
- Extend part libraries

## Priority 2: Add more functionality

- Part management (MPN, assembly variants, ...)
- 3D models in library / 3D board viewer / MCAD export
- Hierarchical sheets
- ...

# Contributors welcome!

<https://github.com/LibrePCB/LibrePCB/blob/master/CONTRIBUTING.md>

IRC: #librepcb on Freenode

- Participate in issues
  - Open pull requests
- Improve documentation
- Donate (Patreon, GitHub Sponsors, . . . )

# Thank you!

<https://librepcb.org>

# Why not contributing to KiCad?

- **Very different underlying concepts**

- Mindset/opinions often can't be changed by contributors
- Fundamentally changing concepts is *very* hard since an upgrade mechanism needs to be provided

- **Different priorities / target audience**

- KiCad: Feature-rich, professional-grade, flexibility (can do everything)
- LibrePCB: Usability, intuitive UI, portability, stability, version controlability