Alternate KiCad Library version 1.1

License:

Alternate KiCad Library by Dawid Cisło is a derivative of KiCad Library made by KiCad community (see: KiCad library GitLab), used under Creative Commons CC-BY-SA 4.0 License, with the following exception:

To the extent that the creation of electronic designs that use 'Licensed Material' can be considered to be 'Adapted Material', then the copyright holder waives article 3 of the license with respect to these designs and any generated files which use data provided as part of the 'Licensed Material'.

Additional information can be found here: KiCad libraries license

What does this mean?

You can freely use Alternate KiCad Library data for commercial, closed and non-commercial projects without any restrictions. There is no need to attribute this library or original KiCad libraries within your design and no obligation to share any project files under this or any other license agreement.

If you wish to redistribute the Alternate KiCad Library, or its parts (including in modified form) as a collection you need to share it under the same license agreement. Libraries must also retain attribution information and license documents which are distributed with the library files.

About:

The aim of this library is to provide a different way of utilizing the silkscreen layer to maximize the amount of information per component. This library consists mostly of modified footprints from KiCad library and some brand new footprints that match the new style and philosophy.

- Different types of components can now be distinguished more easily (e.g. axial capacitors and resistors).
- Directional components have more pronounced polarity (diodes, tantalum capacitors etc.).
- Integrated circuits have marks that make counting pins easier (for example when you need to test a specific pin on a large IC during troubleshooting).
- Different visual style – SMD parts have a rectangular outline and a dot marking pin 1.

Parts of this library deliberately violate the original KiCad Library Convention by including silkscreen drawings under some of the components. Additionaly some SMD footprints might take more physical space that original KiCad ones.

This should make the PCB easier to ‘read’ and reduce errors during hand-soldering and troubleshooting at a cost of lower layout density.

Some SMD part libraries have 2 variants:

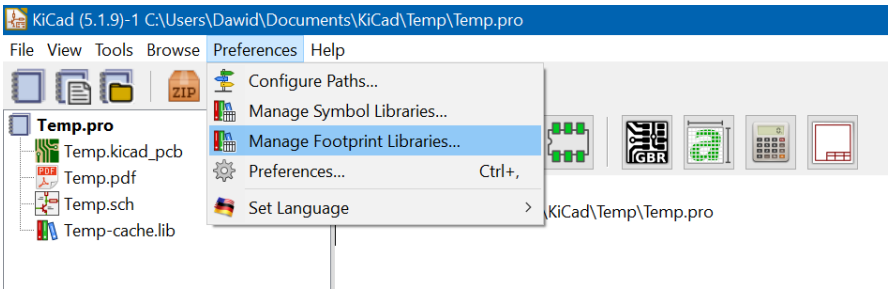
- Standard libraries have no silkscreen under the parts to avoid problems during machine soldering.

- Hand soldering libraries have part symbols under the part. This could help prevent errors during manual placement of the parts by making it easy to distinguish different types (diodes from capacitors or inductors etc.). Not recommended for machine soldering as small SMD parts will swing around on the silkscreen and cause ‘tombstoning’.

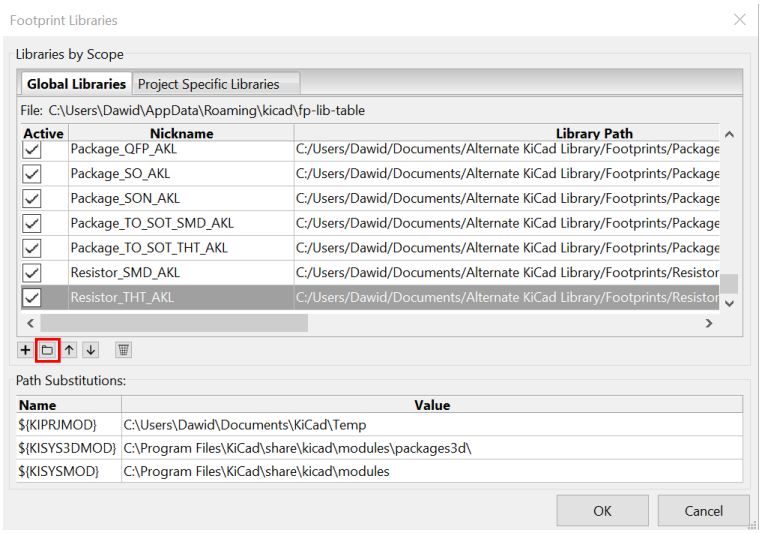
Installation (KiCad 5.1):

Place the library files in any folder – preferably located on an SSD to reduce load times.

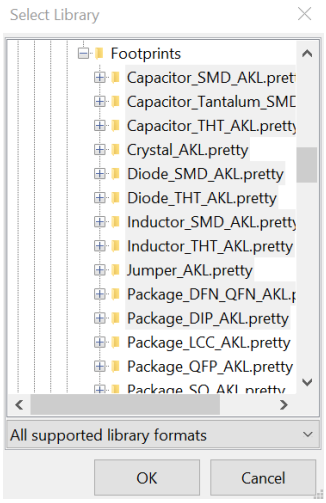
Open KiCad and then under the ‘Preferences’ tab open ‘Manage Footprint Libraries…’



Make sure the ‘Global Libraries’ tab is selected. Click the folder icon and locate the new library folder



Select the ‘Footprints’ folder if you want to use all libraries, or hold **Ctrl** and click to select libraries that you would like to use and then click ‘OK’



You can find more information about library management here:
https://forum.kicad.info/t/library-management-in-kicad-version-5/14636

Compatibility:

Alternate KiCad Library 1.0 is known to work with KiCad 5.1.9 but should be cross-compatible with all KiCad 5 versions.

Footprint library should be forward-compatible with the upcoming KiCad 6 (tested on nightly build r22263).

Disclaimer:

Alternate KiCad Library as a modification of KiCad Libraries is a work of a single person and is not guaranteed to be correct.

If you find any errors please send me an e-mail with the details (footprint name, link to datasheet, screenshots showing the error) here: dawid_cislo@o2.pl

It might take months between updates, because I'm making this project in my spare time, but more features are coming.

Changelog:

Version 1.1

- SMD Diode, Capacitor and Inductor footprint libraries have been split into standard and hand soldering variants.

Capacitor_SMD_Handsoldering_AKL
Capacitor_Tantalum_SMD_Handsoldering_AKL
Diode_SMD_Handsoldering_AKL
Inductor_SMD_Handsoldering_AKL

- Standard libraries are suitable for machine soldering and have no silkscreen under the parts.

- Hand soldering libraries have silkscreen under the part to improve readability of the PCB and reduce component placement errors.

- Footprints (included in those libraries) with bigger pads that used to be called the 'HandSoldering' variants have been renamed to 'BigPads' to reduce confusion.

- Footprint names between standard and hand soldering libraries are consistent. You can easily swap the footprint by using 'change footprint' option and renaming the library in PCBNew.

- Minor improvements to SMD inductor library.

- Some SMD TVS diode footprints had unnecessary polarity marks.

- Crystal and SMD resistor libraries have no silkscreen under the SMD parts.

Version 1.0

- Modified KiCad Libraries:

Capacitor_SMD_AKL
Capacitor_Tantalum_AKL
Capacitor_THT_AKL
Crystal_AKL
Diode_SMD_AKL
Diode_THT_AKL
Inductor_SMD_AKL
Inductor_THT_AKL
Package_DFN_QFN_AKL

Package_DIP_AKL
Package_LCC_AKL
Package_QFP_AKL
Package_SO_AKL
Package_SON_AKL
Package_TO_SOT_SMD_AKL
Package_TO_SOT_THT_AKL
Resistor_SMD_AKL
Resistor_THT_AKL

These libraries inherit KiCad’s pad layout and footprint names so you can easily swap footprints using ‘change footprint’ option and renaming the library in PCBNew.
Some new footprints have been added, see the showcase folder for details.

- New footprint library:

Jumper_AKL

Adds THT jumpers with multiple pin pitch and two wire diameters variants. Helpful during single-sided PCB layout.