Alternate KiCad Library version 1.0

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. Additionally 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.
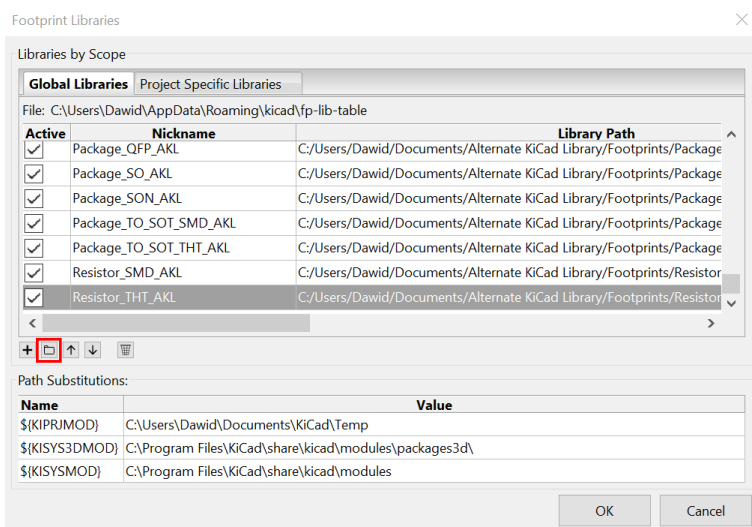
Installation (KiCad 5.1):

Place the library files in any folder – preferably located on an SSD to reduce load times.
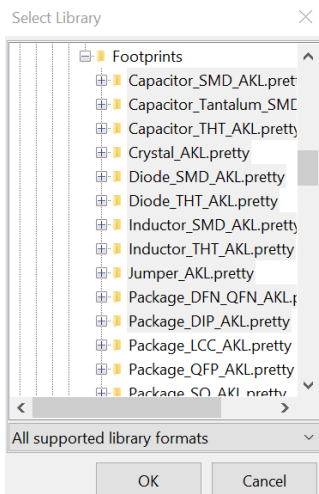
Open KiCad and then under the 'Preferences' tab open 'Manage Footprint Libraries...'
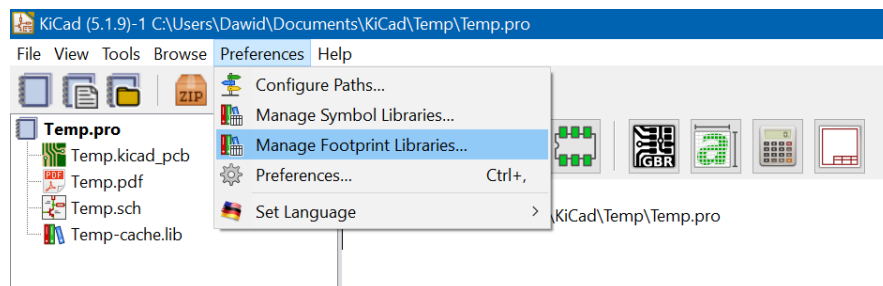
Make sure the 'Global Libraries' tab is selected. Click the folder icon and locate the new library folder

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.