**KiCAD Database Library Setup Instruction**

**Step 1: Setup your folders**

I have created folders for my KiCAD Database library like this:

A screenshot of a computer

Description automatically generated

I like to separate folders by number sequence, so I can very easily find any folder I need by knowing where exactly in a folder list it is. Another thing is that when the whole library is in one repository it is easy to use it across several personal computers when I store all the library in GitHub repository.

Of course, you can create the folders whenever you like.

**TIP:** *If you are using GitHub or GitLab, I recommend to separate Library Setup File separately from Library repository because the link to your database library can wary between different PCs, so it can be changed locally and not committed to global repository:*



A computer code on a white background

Description automatically generated

**Step 2: Create database library using SQLiteStudio**

Of course you can use any database management tool, I just used **SQLiteStudio** to setup my component database. **SQLiteStudio** is free open-source software.

Take database **TestDatabase.sqlite** file, rename file as you like (this will rename your database) and open it.

A screenshot of a computer

Description automatically generated

When you open the database in SQLiteStudio this will look like this:

A screenshot of a computer

Description automatically generated

When you open your database, you can setup or modify your Component Libraries.

To create a new library, just press Right mouse key on your database Tables and press **Create a table**:

A screenshot of a computer

Description automatically generated

Then write a name for your library in Table Name field:

A screenshot of a computer

Description automatically generated

After that add at least one column to the library:

1. Add a column
2. Create a name for your column
3. For a Key column which is my component PartID I select **Primary Key**. This is your internal component part number.
4. For Key column I also select **Not NULL** constrain so this would not let me save database table if any of the components do not have Key set to them
5. Save the column
6. Then save the database table

A screenshot of a computer

Description automatically generated

Now we have the first table in our library:

A screenshot of a computer

Description automatically generated

Now you can add all the other columns to your library with the same actions as described above. For the data type I select **TEXT** as it fits all the columns except few specific ones.

A screenshot of a computer

Description automatically generated

**TIP:** *I use the same columns for all of my libraries (tables) so I just use the function “Create similar table” and changing name of the table and saving it:*

A screenshot of a computer

Description automatically generated

This way you can create any libraries and any component fields for your KiCAD Database Library

**Step 3: Setup KiCAD Database Library**

If you go to the Database\_Setup folder, there will be a text file DB\_Test.kicad\_dbl. Open this file you can setup your KiCAD database libraries and what fields are represented directly in KiCAD:

A screenshot of a message

Description automatically generated

Firstly, to setup your KiCAD database library, you should add a direct path to the database file you created in previous step

A computer code on a white background

Description automatically generated

After that you will need to setup database fields:

There is an example of how I had added all the fields from my SQLite database to KiCAD component fields for one component library:

{

"name": "Misc\_Artwork\_1",

"table": "Misc\_Artwork\_1",

"key": "Key",

"symbols": "Symbol",

"footprints": "Footprint",

"fields": [

{

"column": "Key",

"name": "PartID",

"visible\_on\_add": false,

"visible\_in\_chooser": true,

"show\_name": false

},

{

"column": "StoreBox",

"name": "StoreBox",

"visible\_on\_add": false,

"visible\_in\_chooser": true,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Name",

"name": "Name",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false

},

{

"column": "MPN",

"name": "MPN",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Type",

"name": "Type",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Tags",

"name": "Tags",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Value",

"name": "Value",

"visible\_on\_add": true,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Value2",

"name": "Value2",

"visible\_on\_add": true,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "MinTemp",

"name": "MinTemp",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "MaxTemp",

"name": "MaxTemp",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Mount",

"name": "Mount",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Package",

"name": "Package",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Manufacturer",

"name": "Manufacturer",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "LQ\_Price",

"name": "LQ\_Price",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "HQ\_Price",

"name": "HQ\_Price",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "JLC\_Code",

"name": "JLC\_Code",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Project",

"name": "Project",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Alternatives",

"name": "Alternatives",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Notes",

"name": "Notes",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Datasheet",

"name": "Datasheet",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Link1",

"name": "Link1",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Link2",

"name": "Link2",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Link3",

"name": "Link3",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Link4",

"name": "Link4",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

},

{

"column": "Link5",

"name": "Link5",

"visible\_on\_add": false,

"visible\_in\_chooser": false,

"show\_name": false,

"inherit\_properties": false

}

],

"properties": {

"description": "Description",

"footprint\_filters": "footprint\_filters",

"keywords": "Keywords",

"exclude\_from\_bom": "exclude\_from\_bom",

"exclude\_from\_board": "exclude\_from\_board"

}

},

As we can see there is only few special lines to configure, and everything else is copy paste 😊

1. **Configure main KiCAD Component Fields**

"name": "Misc\_Artwork\_1",

"table": "Misc\_Artwork\_1",

"key": "Key",

"symbols": "Symbol",

"footprints": "Footprint",

"fields": [

***name*** – KiCAD Library name, this one will be displayed in KiCAD

***table*** – Table name from the SQlite database

***key*** – Is a key identification number of component in the database. In other words internal component part number

***symbols*** – is to name the column name in the database table where is the symbol library reference for that component is.

***footprints*** – the same as with a symbol

***fields*** – there you add additional fields for your component in KiCAD which will reference to the SQLite database specific column

1. **Adding fields**

There is an example of added 2 additional fields for your KiCAD component which reference to the SQlite database:

"fields": [

{

"column": "Key",

"name": "PartID",

"visible\_on\_add": false,

"visible\_in\_chooser": true,

"show\_name": false

},

{

"column": "StoreBox",

"name": "StoreBox",

"visible\_on\_add": false,

"visible\_in\_chooser": true,

"show\_name": false,

"inherit\_properties": false

},

***column*** – is the column name, which KiCAD DB library references to in SQlite database

***name*** – is the name of the field in KiCAD

***visible\_on\_add***– does the parameter is visible when added to schematics (true or false)

A diagram of a usb cable

Description automatically generated

***visible\_on\_chooser*** - does the parameter is when choosing component to add to chematics schematics (true or false)

A screenshot of a computer

Description automatically generated

***show\_name*** – show field name when component is added to schematics

A diagram of a usb cable

Description automatically generated with medium confidence

***inherit\_properties*** – does the field inherit the field properties from symbol library (on selecting false it should be default field settings)

This way you can setup your database library how you like. If you spend some time you can easily create your own style of components.

1. **Adding properties**

Also, in your database library you can change your component properties, like:

* Description
* Keywords
* Footprint Filters
* Exclude from bom
* Exclude from board

"properties": {

"description": "Description",

"footprint\_filters": "footprint\_filters",

"keywords": "Keywords",

"exclude\_from\_bom": "exclude\_from\_bom",

"exclude\_from\_board": "exclude\_from\_board"

}

***description*** – match with component description in your database

***keywords***– match with component keywords in your database

A screenshot of a computer

Description automatically generated

***footprint filters*** – this is optional column. It can be used if for example you have several footprints for your THT radial capacitor, then instead of adding one footprint in the library, you can add a filter, which when selecting footprints will show only suitable footprints for it. To read more about footprint filters you can proceed to this ink: <https://docs.kicad.org/6.0/en/eeschema/eeschema_assigning_footprints.html>

***exclude from bom*** – Boolean type. 1 means that component is excluded from BoM.

***exclude from board*** - Boolean type. 1 means that component is excluded from board.

A screenshot of a computer

Description automatically generated

**Step 4: Add your database to KiCAD**

To be able to use your database library, you should add all the required symbol and footprint libraries

A screenshot of a computer

Description automatically generated

After symbol and footprint libraries are added you can add your database library in “Manage Symbol Libraries” option:

A screenshot of a computer

Description automatically generated

After that you can test the database by adding new components to your schematics:

A screenshot of a computer

Description automatically generated

**Step 5: Adding new components in the database**

To add a new component to the database simply open your SQlite database -> select table, in which you want to add component -> select Data:

A screenshot of a computer

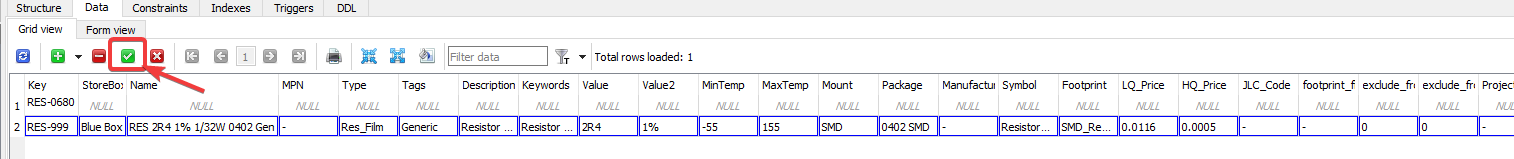
Description automatically generated

After that press green plus sign to add additional 1 row (1 row equal to 1 new component):

A screenshot of a computer

Description automatically generated

After adding new component add Unique component Key and fill all the other information and press save the table:



To add correct Symbol reference, simply add [Symbol Library Name]:[Symbol Name]. The same is for footprints.

A close-up of a computer screen

Description automatically generated

When the new component to the database is added, simply close and reopen KiCAD and you can add a new component to the library.

Now you have all the data of your components directly in KiCAD 😊

