

HANDOUT

Modular Synthesiser PCB Workshops

Oli Sharratt, Junzhe Chen, and Bhavy Metakar

Introduction

The comprehensive guide to using KiCAD for Fleming Society’s Synthesiser PCB Workshop. This includes instructions for KiCAD project file setup, schematic editing and making PCB layouts.

Contents

1	Project File Setup	4
1.1	Import Footprints	4
1.2	Import Symbol	4
1.3	Import 3D Files	4
2	Schematic Editor	5
2.1	Document Setup	5
2.2	Add Components	5
2.3	Connect Components	6
2.3.1	Wires & Junctions	6
2.3.2	Labels	6
2.3.3	Not Connected	6
2.4	Assign Footprints	6
2.5	Electrical Rule Checker	7
2.6	Multi-Page Documents	7
3	PCB Layout Editor	8
3.1	Document Setup	8
3.2	Import from Schematic Editor	8
3.3	How to use the PCB Layout Editor	9
3.4	Track, Via, & Grid Sizes	9
3.5	Layers & Display Options	10

3.6	Vias	10
3.7	Copper Pours	11
3.8	Rules Area & Exclusion Zones	12
3.9	Board Outline	12
3.10	3D Viewer	12
3.11	Design Rule Checker	13
4	Requirements for UCL In-house Production	14
5	Troubleshooting	15
5.1	I can't click on the application window	15
5.2	None of my symbols show up in the Symbol Library	15
5.3	There isn't a button to switch from Schematic Editor to PCB Layout Editor	15
6	Appendix I: Keyboard Shortcuts	16
7	Appendix II: External Resources	16
7.1	External Footprint/Symbol/3D File Libraries	16

Introduction

Electronic components can be represented in several ways as shown in Figure 1. All of these stages are required when designing and fabricating your own Printed Circuit Board (PCB).

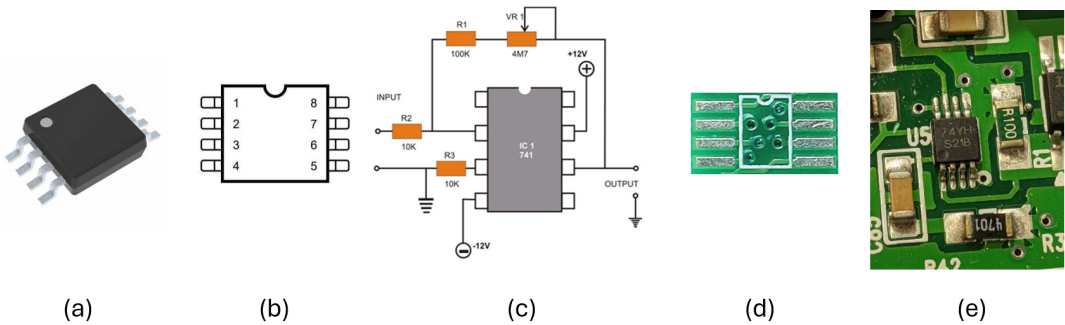


Figure 1. Representations of electronic components.

There are keywords used to describe each type of representation:

- **Component:** This is the physical electronic part as shown in Figure 1a.
- **Symbol:** This is a 2D drawing that represents the component as shown in Figure 1b.
- **Schematic:** This is a collection of symbols showing how they are joined together which represents circuit for the components as shown in Figure 1c.
- **Footprint:** This is the physical arrangement of copper pads / holes which the components joins on to on a printed circuit board (PCB) as shown in Figure 1d.
- **PCB:** This is a printed circuit board where all of the components physically connect to and it connects all of the relevant components together as shown in Figure 1e.

Electronic Design and Automation (EDA) software is used to make a schematic from symbols of physical components. Using such software, the footprints of the physical components can be arranged and connected together according to the schematic. KiCAD is an example of such EDA software, which has the following editors:

- **Schematic Editor:** Used to create the circuit representation of the circuit.
- **Symbol Editor:** Used to create symbols for the Schematic Editor that represent physical components in circuit diagram form.
- **PCB Editor:** Used to place and connect the footprints of components for the physical PCB.
- **Footprint Editor:** Used to create footprints for the PCB Layout Editor that represent the dimensions of the physical components.

1 Project File Setup

Currently, MPEB 6.02 Teaching Lab only has KiCAD 7.0.11 installed on the computers. Hence they cannot open any newer versions of KiCAD (8.0 release or 9.0 beta). For compatibility between personal device and the lab, please use KiCAD 7.0.11.

1.1 Import Footprints

Open the Footprint Editor from the project application window. Click **File - New Library**, and select **Project**. Name the file something like `[project_name].pretty` and save it in the project folder. This will create a `[project_name]` element in the left hand-side list. Locate it, and right-click **Pin Library** if you want. Otherwise, right-click and **Import Footprint**, selecting the `.kicad_mod` files that you wish to import. Save each footprint, then exit the editor.

1.2 Import Symbol

Open the Symbol Editor from the project application window. Click **File - New Library**, and select **Project**. Name the file something like `[project_name].kicad_sym` and save it in the project folder. This will create a `[project_name]` element in the left hand-side list. Locate it, and right-click **Pin Library** if you want. Otherwise, right-click and **Import Symbol**, selecting the `.kicad_sym` files that you wish to import. Save each symbol, then exit the editor.

1.3 Import 3D Files

Add your 3D files (`.stl`, `.step`, `.IGS`, etc) to a folder in the project directory (call it `3d_models`). See Section 3.10 for more information.

2 Schematic Editor

Compared to DipTrace, there are many shortcuts (already configured) and schematics can be made much quicker. However, the cursor position is much more important. Wherever the mouse is – the symbol its hovering over – is typically where shortcuts are applied.

2.1 Document Setup

Set File – Page Settings for the project (name, date, revision, company, comments, etc). Check symbols and footprints are present, if not see Section 5.2.

Search for the symbol you need (if it is not present it will have to be imported) and double click to select, then click to place. If you need multiple of the same component quickly, select Place repeated copies at the bottom of the window.

2.2 Add Components

Click the Add Symbol (🔍) icon, or press <A> to search for the component, the following window will appear:

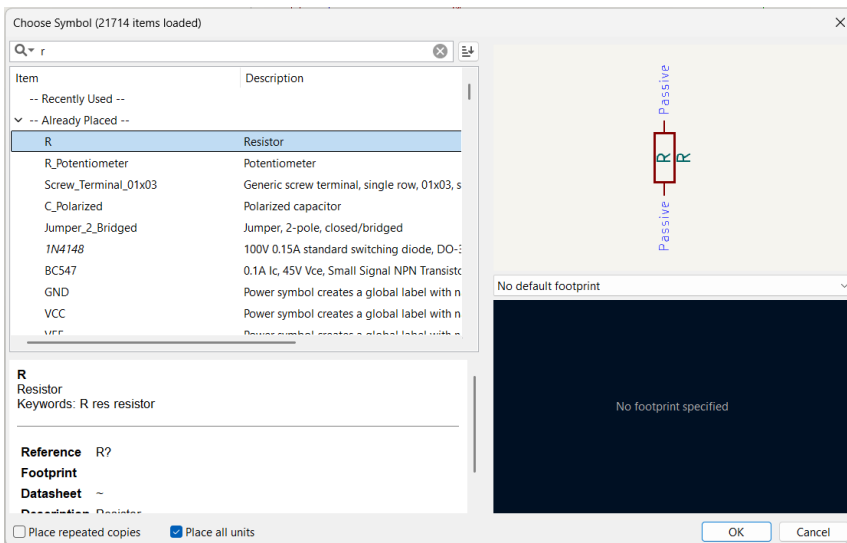


Figure 2. Choose Symbol

Some components come with both its symbol and footprint; make sure that the footprint matches with the physical components. Others, like a resistors, capacitors, inductors, etc. have “No footprint specified”. This will be assigned later in Section ??.

2.3 Connect Components


2.3.1 Wires & Junctions


Components are connected via wires. When moving the cursor close to the connection end, the cursor will change and allow you to click between nodes to join them together. Alternatively, components can be connected using shortcut key <W>. A wire will generated at the cursor location.

2.3.2 Labels

For some connections such as power and components far away from each other, connecting them by wire can make schematics unorganised and less readable. A method to improve that issue is to connect those components by labels. **Any two labels of the same type and same name will be automatically connected.** You can check net connections by using the Highlight Nets tool (right toolbar, below cursor).

Pre-defined Labels: Some labels are generic, such as the GND and VCC power symbols. These are found in the component selection window, see Section 2.2. If you choose to use Net or Global Labels for power connections, the ERC (2.5) will likely give an error because its not defined as a power connection. To fix this add a **Power Flag** anywhere along the net.

Net Labels: Select the  on the right toolbar, or use <L> to make a user-defined label. As mentioned if two labels have the same name, the will be connected.


Global Labels: Besides all the properties of the Net Label, Global Label have two more properties. One being the abilities to connect labels between multiple sheets, hence the name *global*. Another property of the Global Label is it defines the state of the label – if it is an Input, Output, Bi-directional, etc. Access these with the  or use <Ctrl+L>.

For Hierarchical Labels, see 2.6

2.3.3 Not Connected

Extra pins may present in the design. If these pins are left unconnected, it gives an ERC error (see Section 2.5). Using <Q> or by right-clicking an unconnected node – **Pin Helpers – No Connect**, you can add the **No Connect** cross symbol.

2.4 Assign Footprints

Looking at the toolbar of the Schematic Editor Run **footprint assignment tool**  or select **Tools – Assign Footprints...** There are three panes – left is the libraries, right is the content of those libraries, and middle are the symbols that need footprints assigning to. Select the appropriate library to find the footprint. Select the symbol in the middle pane and double-click on the chosen footprint in the right right pane. This will assign the footprint to the respective component. The Schematic Editor will highlight any active element so you can see which is which. There is a search

box at the top which will filter only the right pane. If you accidentally leave text in this box, and change library, its likely nothing will show up in the right window. See an old KiCAD Forum Guide for more information.

2.5 Electrical Rule Checker

Use the Electrical Rule Checker (ERC) to ensure there are no inherent issues with the schematic (it will not tell you if our circuit works or not, just whether its connected properly). In the top toolbar, go to **Inspect - Electrical Rule Checker**, and click **Run ERC**. This may generate a series of errors and warnings.

2.6 Multi-Page Documents

This is unlikely to be required for this project, but here is the information regardless. For larger documents, multiple pages of a schematic may be required. It is good to separate each sheet depending on function. They can be added **Place - Add Sheet** or by pressing **<S>** in the Schematic Editor. You are required to draw out a square, and provide a name. Once edited and saved, it generates a new `kicad_sch` file in the project directory.

As mentioned, Global Labels will be automatically connected between sheets, given they have the same name. However, to connect elements between specific sheets, you have to use Hierarchical Labels.

Hierarchical Labels: When placed within a sheet, they act like Net Labels, but if you return to the page where the sheets have been placed, you can right-click on a sheet and **Import Sheet Pin**. This allows you to selectively connect labels between sheets, and they do not have to have matching names.

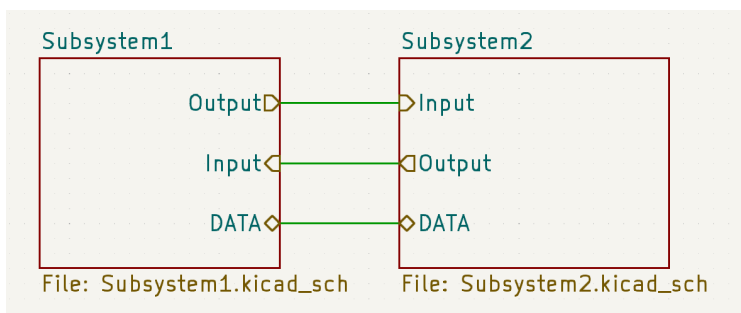


Figure 3. Example connections between sheets using Hierarchical Labels.

3 PCB Layout Editor

Compared to DipTrace, there are many shortcuts (already configured) and PCB layouts can be made much quicker. However, the cursor position is much more important. Wherever the mouse is – the footprint its hovering over – is typically where shortcuts are applied.

3.1 Document Setup

Set **File - Page Settings** for the project (name, date, revision, company, comments, etc).

File - Board Setup - Constraints:

Set constraints as required, where most importantly add for UCL in-house production:

Minimum Clearance: ≥ 0.4 mm

Minimum Track Width: ≥ 0.4 mm

File - Board Setup - Pre-defines Sizes:

Add appropriate track widths and via sizes.

Tracks should be ≥ 0.4 mm, vias should be \geq “1.4 mm / 0.7 mm” for UCL in-house PCB production.

See Section 4 for more details about in-house production.

3.2 Import from Schematic Editor

Before moving forward, please ensure the ERC has been run, and there are no errors in the schematic – see 2.5. From the Schematic Editor is a button in the toolbar to switch to the PCB Editor (🔌). From the PCB Editor there is another button to pull changes from the Schematic Editor into the PCB Editor (🔄), which will pull up a dialogue box. If you cannot find the button, see Section 5.3.

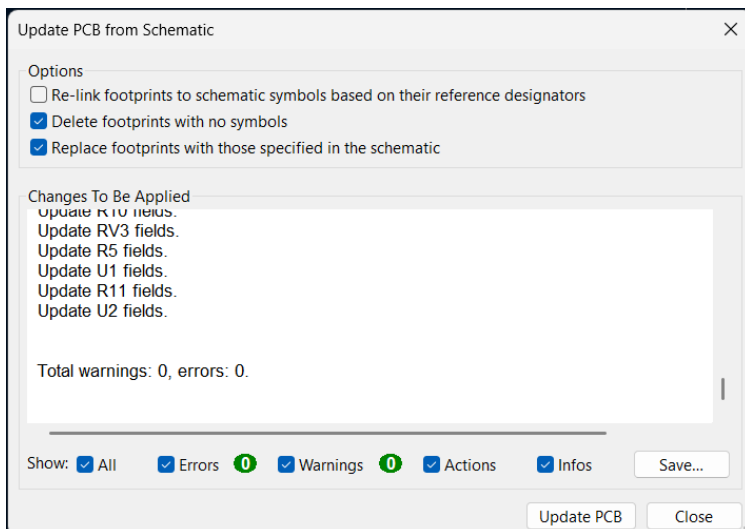


Figure 4. Update PCB from Schematic

From this, click **Update PCB**, let it run, and once you close it, all changes should be pulled through. The first time, it will give you a pile of components (they are not aligned to any grid), and if you change anything in the Schematic Editor in the future, run the tool again and all changes will be updated. ~~Infinitely better than DipTrace...~~

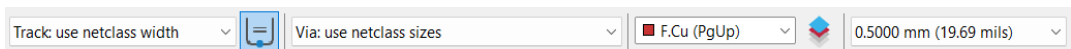
Depending on what you are doing, the options at the top of the window may be required. If symbols have been removed on the schematic, use **Delete footprints with no symbols** to remove all footprints that no longer link to any symbol.

3.3 How to use the PCB Layout Editor

After importing, components are simply placed on the canvas. Use <M> (move) to move footprints individually, or use <D> (drag) to move footprints with tracks. <R> (rotate) when hovering over an element will rotate it. Use <X> to place a tracks wherever the cursor is currently hovering (pin or other track). This will highlight everything that needs to be connected to that net. Use <E> to edit properties such as track width. <V> will switch between copper layers. <F> will send a component to the other side of the board. fills all copper pours.

3.4 Track, Via, & Grid Sizes

To change dimensions of active elements in the PCB Layout, use the toolbar just above the editor – by default it will look like this:



The first box is the track width, followed by the via size, active layer, and then grid size. Select any box and click on the last item to **Edit Pre-defined Sizes...** As mentioned, width sizes > 0.4mm are ideal, and depending on what the trace is for and how much current it is designed to carry, the thickness can be multiple millimetres. Vias should be at least 1.400 mm / 0.600 mm, where the first number determines the diameter of the copper pad, and the second is the diameter of the hole. These numbers allow for a 0.35mm copper surround to the hole, and are the recommended minimum for in-house production.

It is suggested to use larger grid (0.5-1.5mm) to place components (align the components nicely) and a smaller grid (0.1 – 0.5mm) to route tracks and smaller SMD components. The grid position is dependant on the origin. Set an origin (top toolbar: **Place - Grid Origin**) such that everything can be aligned reasonably – usually at the corner of the board outline.

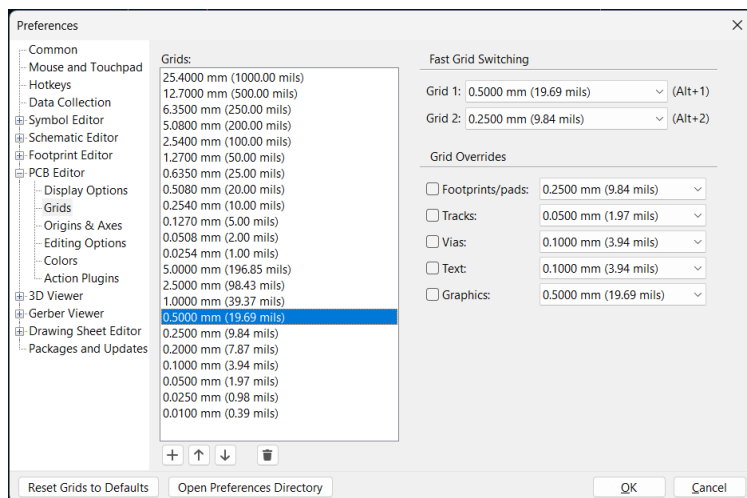


Figure 5. Properties Window (Grid Settings)

You can use <Alt+1> and <Alt+2> to fast switch between two defined sizes, which can be set in Preferences - PCB Editor - Grids - Fast Grid Switching.

3.5 Layers & Display Options

See the bottom right corner – there are selection filters for all elements. Change these as required. If you lock elements, you can select them to be able to unlock them by enabling the Locked Items selection filter.


In addition, if you open the Layer Display Option shade, you can change the view settings of layers. Normal shows all layers at 100% opacity, Dim reduces the opacity of the not-selected layer, and Hide only shows the active layer. When working on the rear side of the board, Flip board view may be useful.

3.6 Vias

Placing vias in KiCAD is far easier and quicker than DipTrace... When actively routing a trace (<X>), press <V> to make a via appear and then left-click to place it. You will automatically be taken to the other copper layer. From here route your trace, and if required, press <V> again to place another via and return to the first side. Once you get the hand of it, it should be very intuitive.

If you want to place a free-standing via, you can select the via (🔴) in the right toolbar, or use <Ctrl+Shift+V> and place as normal. These are usually placed between copper pours on multiple layers to “stitch” them together, assuming they are connected to the same net.

3.7 Copper Pours

To place copper pours, select () and click on the PCB. In the Copper Pour Properties window, select the net, that the pour should be connected to. If the desired net is not shown, try deselecting Hide auto-generated net names.

If you want to make a copper pour on multiple layers of a board with the same dimensions (e.g. GND planes), save yourself time by selecting all desired layers in the top left corner.

Looking at the settings in the bottom-half of the window. You can set a name, and priority level. A zone with a higher priority (bigger number), will take precedence over other zones when they are filled. Add corner smoothing if you're feeling fancy, but the important settings are in Electrical Properties. Ensure Clearance is ≥ 1 mm, and Minimum Width should be at least that of your Minimum Clearance (from Section 3.1). Considering Pad Connections depends on the purpose of the pour. For a ground plane, it is common to use Thermal Reliefs to aid in soldering, where for in-house production, the gap and width should be ≥ 0.7 mm. See Fig. 6 for recommended settings.

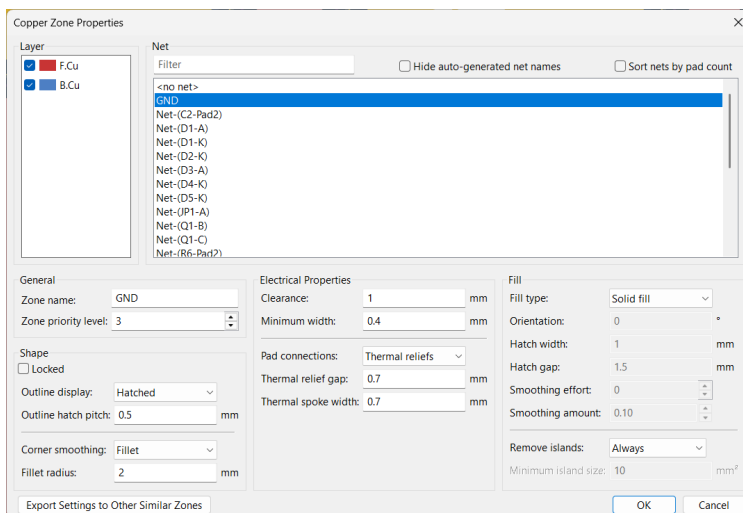


Figure 6. Recommended Copper Pour Properties for a Ground Plane.

When dealing with higher currents, you need wider tracks. Using large tracks can get a bit annoying as their shape is not very customisable, hence you can use a copper pour in its place. For these, you usually won't want Thermal Reliefs, and would opt for Solid pad connections instead.

All other default settings are usually fine. Now, you can select OK and place the outline of the pour. To update/ fill the pour, select Edit - Fill All Zones in the top toolbar or press . You can lock copper pours (so you can't select them), by selecting and Toggle Lock in the right-click menu, or press <L>. So be able to select them again, and unlock them, enable the Locked Items selection

filter in the bottom right corner.

3.8 Rules Area & Exclusion Zones

These work in the same way as copper pours except you define regions where you do not want certain elements – “keep-out zones”. Selecting **Add Rule Area** (🔍). This is fairly self-explanatory.

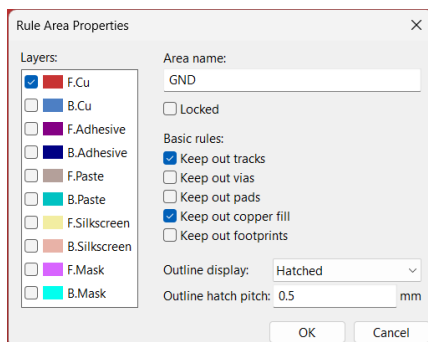


Figure 7. Rule Area Properties Window


3.9 Board Outline

To add a board outline, select the **Edge.Cuts** layer in the right **Appearance** window (it should get a little arrow at the left of the name). Then use the shape tools in the right toolbar to draw an outline. Ideally use the **Rectangular** tool to avoid issues with the lines connecting and the DRC (3.11) getting angry.

If you wish to add a fillet to the shape, right-click the corner **Fillet Lines** and enter a value. Repeat for all corners. If you need a non-standard sized hole/ gap inside the PCB you can add **Edge.Cuts** internally and they can be routed out in production.

3.10 3D Viewer

Access by going to **View - 3D Viewer** or by pressing **<Alt+3>**. This will open a new window which you can move around and view the PCB with components. In KiCAD 8.0, there is an **Appearance** panel on the right side which allows you to easily toggle the visibility of elements. In KiCAD 7.0, you can select **Preferences - Toggle [type] models**, but this is more limited.


If there are missing models, ensure the 3D files can be found by KiCAD, and are assigned to the component. To check, select the footprint, then either right-click **Properties**, or press **<E>**. This opens the **Footprint Properties**, then select the **3D Models** tab. If no model is shown on the preview and the  is shown, then it cannot find the file. This can be a common issue with imported items. It can be an issue with the configured paths in the main project window **Preferences - Configure Paths**, but you can also set your own path on a footprint-by-footprint basis. Using `${KIPRJMOD}` allows you to

make a relative path substitution to the project folder, hence `${KIPRJMOD}/3d_models/model.stp` would be valid, assuming it is setup like Section 1.3.

3.11 Design Rule Checker

Visit **Inspect - Design Rule Checker** to open the DRC window. Run **DRC** to see what it has to say. Most warnings aren't important - if you have silkscreen errors, it doesn't matter for in-house production as we don't have silkscreens, but check them anyway to see if they're an easy fix. Looking at errors, these are most commonly about constraints not being met, or insufficient numbers of spokes on thermal reliefs. Try to fix as required, use the internet if you're unsure, or contact us for help.

4 Requirements for UCL In-house Production

Given that UCL still mainly uses DipTrace, you have to be nice to Martin whenever you design your files. The minimum resolution that PCBs can be made with is 0.3mm, hence any gaps/pads/vias/features that are $< 0.3\text{mm}$ will not be distinguishable. To measure features, locate the measuring tool () in the right toolbar.

If all constraints have been set as mentioned in Section 3.1, then the component placement should have no issues. Firstly, please check all text is thick enough (when made in copper); change the height and width to suit.

For copper pours, as mentioned in Section 3.7, ensure **Clearance** is $\geq 1\text{mm}$, and **Minimum Width** should be at least that of your **Minimum Clearance** (from Section 3.1). When using **Thermal Reliefs**, the gap and width should be $\geq 0.7\text{mm}$.

5 Troubleshooting

If you have any issues, the solution might be explained in the following section. If not, please contact the Fleming Society, and we will try to give assistance.

5.1 *I can't click on the application window*

When switching between editors, you may try to interact but it will (*windows bong sound*) not let you. Make sure no dialogue boxes are open – certain sub-windows on any of the other editors that are open. If you have the symbol placement window open in the Schematic Editor, you won't be able to do anything on the PCB Editor.

5.2 *None of my symbols show up in the Symbol Library*

When opening the Schematic Editor or PCB Layout Editor for the first time, you will be confronted with this pop-up:

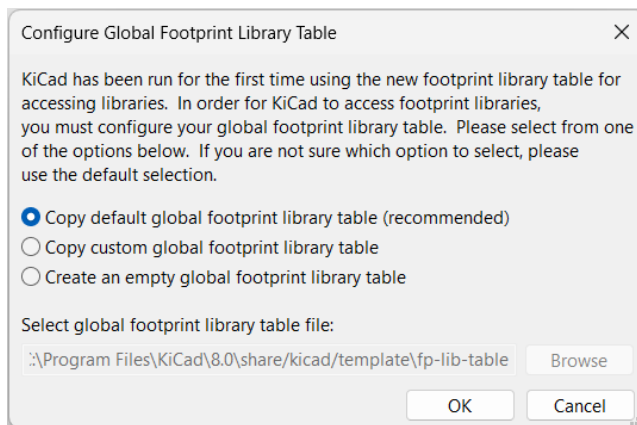


Figure 8. Global Footprint Library Configuration

where you should select the top recommended option. If that is greyed out, then KiCAD has not automatically found the global libraries that it installed with the application. Hence you will have to configure a custom path to the fp-lib-table file. On Windows 10/11 it should be in [installation directory]\KiCad\7.0\share\kicad\template\fp-lib-table.

5.3 *There isn't a button to switch from Schematic Editor to PCB Layout Editor*

You must open the project file (.kicad_pro) first, then open the Schematic Editor file (.kicad_sch) for the button to appear in the top bar. The project file links the Schematic Editor files to the PCB Layout Editor files (assuming they have the same file name).

6 Appendix I: Keyboard Shortcuts

Schematic Editor

Shortcut	Action
M	Move a symbol
R	Rotate a symbol
X	Flip a symbol
E	Edit a symbol
G	Drag a symbol
Del	Delete a symbol
W	Place a wire
P	Place a power symbol
L	Place a label
CTRL + L	Place a global label
F1/F2	Zoom in/out

PCB Editor

Shortcut	Action
M	Move an element
D	Drag an element
R	Rotate an element
E	Edit properties
Del	Delete an element
X	Place a track
V	Change copper layer
V	Add a via (whilst routing)
Ctrl + Shift + V	Add a free-standing via
B	Rebuild copper zones
H	Cycle layer display options
Alt + 3	3D Viewer

General Shortcuts

Shortcut	Action
Left click/drag	Select item(s)
Right click drag	Navigate canvas
Middle click drag	Navigate canvas
Scroll in/out	Zoom in/out
Ctrl + S	Save the document
Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + C	Copy
Ctrl + V	Paste
Ctrl + X	Cut

7 Appendix II: External Resources

7.1 External Footprint/Symbol/3D File Libraries

- <https://componentsearchengine.com/>
- <https://www.ultralibrarian.com/>
- <https://www.snapeda.com/>