# Document Control

## Title Box

|  |  |
| --- | --- |
| Title | KiCAD User Guide |
| Project Name | User & Admin Guides |
| Document Revision | 00.01 |
| Date | 06/07/2023 |
| Author | Farren, Kitty |

**Contents**

0. Document Control 1

0.1 Title Box 1

KiCAD User Guide 1

1. Preamble 1

1.1 List of Provided Files 1

2. KiCAD Project Workflow 2

3. Add Ons 4

4. Symbol and Footprint Creation 4

4.1 Using Existing CT-Footprints 4

4.2 Using External Footprints 5

5. Panelisation 5

6. Common Errors 6

6.1 KiKit 6

6.2 STEP Export 6

7. References 6

# Preamble

This guide shows how to use KiCAD to produce a PCB that can be populated and constructed in the factory. The aim to produce and equivalent DFM manpack like that used by Fablink. This is for **prototype boards only.**

**Out of Scope**: Installation of any necessary software. Please refer to the official websites or the relevant getting started guides. All software required has been referenced.

## List of Provided Files

|  |  |
| --- | --- |
| **Name** | **Description** |
| PanelConfig.json | KiKit example panelisation file following CT standard |
| ManPackExample.zip | An example ManPack that can be used for DFM review for basic boards |
| KiCADFix.zip | A small program to convert the KiCAD generated BOM and GenCAD files into a procurement, DFM and SMT machine acceptable format. See the README.md for instructions. |

# KiCAD Project Workflow

Similarly to an Xpedition project, a KiCAD project is made up of a schematic and a layout. This file does not go into the basics of creating a KiCAD project as this is gone over in their official documentation [1]. The following steps make up a typical workflow.

1. **Add Symbols**

See section 4 Symbol and Footprint Creation. A footprint and symbol are linked together through the symbol editor and then the symbol properties dialog . 3D models are linked to the footprint in the footprint editor and then the footprint properties dialog .

1. **Create the Schematic**

Place all required components and connect as necessary. This section of the report is underdeveloped for creating complex schematics) so additions are welcome. Hierarchical, multi-sheet schematics, bussed signals and custom symbols are all possible.

1. **Board Outline**

* Create a contiguous board outline using drawing elements on the edge.cuts layer.
  + THIS SHOULD NOT BE A POLYGON to avoid STEP export and kikit panelisation errors.
* The board outline can be checked by going though toolbar: View -> 3D Viewer
  + If this view does not work, there is a discontinuity in the board outline. Figure 1 shows a 3D viewer output.

A close-up of a circuit board

Description automatically generated

Figure : An example of the 3D viewer output

It should also be noted that where a more challenging mechanical design is being considered, a board outline can first be created in FreeCAD [2] (another excellent opensource program) and pushed to KiCAD using the *KiCADStepUp* workbench. This is not described here but the documentation on a typical workflow can be found here [3] . This should be considered when external mechanical elements need to be modelled with the PCB (e.g. panel support, screw demonstrations) which may change the PCB outline.

1. **Place Components**

No issue with this were found.

1. **Create Basic Fab Files (or other)**

The easiest way to do this is to get the PCBWay KiCAD add on. Once installed just press the PCB Way button in the toolbar to generate all files and automatically load them into PCBWay. Other add-ons are available for other producers (e.g. NextPCB, PCB GOGO)

Otherwise, Gerbers can be drawn individually in the *Plot*  menu and should be drawn for the following layers:

B\_Cu, B\_Mask, B\_Paste, B\_SilkS, Edge\_Cuts, F\_Cu\_F\_Mask, F\_Paste,F\_SilkS

And the following drill files produced:

NPTH.drl, PTH.drl

**THE NEXT TWO STEPS ARE NECESSARY FOR FABRICATION AT U79 (optional)**

1. **Panelise (Optional)**

See Section 5 Panelisation for this workflow but the CT standard config file provided does a lot of the heavy lifting if this seems like a daunting task.

1. **Create Full ManPack (Optional)**
   1. Drawings

The drawings can be created by adding measurements to the user.drawings layer (see toolbar: Place -> Add Orthogonal Measurement for a useful measurement tool). It may also be useful to see the drawing in the example manpack provided for what measurements are recommended. Once the measurements have been added to this layer, the board2pdf add-on can be used to generate the pdf drawings by turning off and on different layers.

* + 1. PANELTEST141\_Iss0000\_\_DWG: a drawing of the whole panel
    2. TEST141\_Iss0000\_D1, TEST141\_Iss0000\_D2: individual drawings of both sides of the board
    3. TEST141\_Iss0000\_\_DWG: a basic drawing of the cut-outs and outlines
  1. Gerbers

These are the Gerbers provided to the PCB manufacturer and a Gerber of the user-drawings layer containing measurements- this is useful for DFM. Make sure to use the **drill origin** to line up with the PCBWay auto generated Gerbers.

* 1. STEP file

The STEP file is of a circuit only, not a panel, and is useful for both the mechanical and manufacturing departments. This can be created through toolbar: File -> Export -> STEP. The outline chaining tolerance should work with standard and the coordinated **must use the drill origin**.

* 1. .cad file

Create the basic GenCAD file using toolbar: File -> Export -> GenCAD. This file however, isn’t in a state appropriate for the SMT machine in U79 so please use the KiCADFixer program provided to update this file. Annoyingly, at the time of writing this report, there is a bug in KiCAD that only allows the gencad to be generated in inches, but the provided program converts these measurements to mm which the SMT machine uses. It also removes library names (leaving only CT-Pt Number)

The .cad file requires the use of the GenCAD fixer tool that is provided separately. This tool converts the GenCAD into a form acceptable to the SMT machine. See the README file in the .zip for further instructions on this.

* 1. .bom file

Create the basic BOM file using toolbar: File -> Fabrication Outputs -> BOM. This file however, isn’t in a state appropriate for procurement and MPI scrubs so please use the KiCADFixer program provided to update this file.

The .bom file also requires the use of the KiCADFixer fixer tool that is provided separately. See the README file in the .zip for further instructions on this.

* 1. .gwk

To create the .gwk file used by DFM the software GCPrevue is required which can be found in the software shared area. In GCPrevue, import all the Gerbers and drill files in the Gerber file (this includes the user.drawings plot). Save this as the .gwk output. This does not produce layers with the same name as Fablink but it is an acceptable alternative for small boards.

* 1. Build requirements document.

The build requirements document in the example manpack should be altered to include any information required about what your test board is and why you are making it. Please also check the shared TEMPLATES area to see if there had been included there.

# Add Ons

The following addons are required to follow the process. Addons are added in the Plugin and Content Manager shown in Figure 2.

A screenshot of a computer

Description automatically generated

Figure : KiCAD homepage

### KiKit [4]

If your design is required to go through the factory it will need to be panelised. This add-on allows you to use the provided config file that automatically panelises your project. Documentation for this library can be found at the reference given. [4]

### Board2Pdf

This is needed to produce the drawings the factory needs to check placements. It is important to include component outlines in the drawing layer so that these can be included in the final pdf.

### PCBWay Plug-in for KiCAD

This automatically generates Gerbers in the format required for PCBWay.

# Symbol and Footprint Creation

The symbol libraries where parts are stored are managed in toolbar: preference -> manage libraries. Directories into which parts are downloaded can be added into the path through this option. Once a library is added for a symbol and footprint, they can be accessed in the place symbol/footprint tool.

## Linking a CT-Part Number

No matter what method you use from the two suggested here (either drawing a custom part based on a CT-footprint or importing a new footprint from Mouser or SnapEDA etc.) a CT-Pt number should be linked to that A close up of a number

Description automatically generated with medium confidencepart so the GenCAD SMT placement file can tell the SMT machine what part to place. To do this the **name of the part must be set to the CT part number.** An example of what a basic library called “ImportedParts” would look like is shown in Figure 3. The name of a component can be changed in the footprint properties in the “footprint name” box.

## Using Existing CT-Footprints

To “use” at CT footprint is a long process and requires access to the Siemens Library Manager software. The following is the process for copying measurements from a CT part.

1. Select the cell you require from the central library through the Library Manager and double click to edit.

Figure : Library Layout

1. In the toolbar: file -> export -> DXF
2. Export each layer that you require SEPERATLEY and save locally (NOT IN THE CENTRAL LIBRARY LOCATION)
3. In KiCAD footprint editor, create a new footprint and toolbar: file -> import -> graphics.
   1. Do this for each layer ensuring the correct measure (mm or mils/tho) has been selected based on the CT footprint.
4. It will then be necessary to measure pad sizes and add them as proper KiCAD pads, or reoutline cutouts to create a zero width line filled polygon (see 6.1.1 Tabs added incorrectly.)

## Using External Footprints

A recommendation made by this report is to get *Library Loader* [5]. This program integrates KiCAD with the Mouser website.

1. Install library loader.
2. Run library loader in KiCAD mode.
3. When a footprint is available on Mouser which has symbol on the right in the ECAD column in the search function, click the symbols and choose download CAD models in the popup window.
4. If you do not have an account create one at this point when prompted to login, otherwise login.
5. Once the download completes (almost instant), the parts will be autoloaded into KiCAD.
6. These symbols and footprints are linked automatically.

Other footprints can be found at SnapEDA [6] and can be integrated into KiCAD using the method outlined in the KiCAD guide. [7]

# Panelisation

To panelise a design, the KiKit addon should be used.

An example config file is included with the document and an example of the panelisation workflow using this file is included here. The comments in the provided config file explain what each section does, and further information is available in the KiKit documentation [8].

Workflow:

1. Place the location of your tabs on the board.
   1. These can be found in the kikit library installed automatically when the kikit addon is installed.
   2. The arrow should point towards the edge of the board.

A screen shot of a computer

Description automatically generatedA screen shot of a computer screen

Description automatically generated

1. Put the config file (PanelConfig.json) into your KiCAD project file.
   1. This provided file is a good starting place as it works around some difficult things to do in KiKit like adding a frame around the edge and between circuits.
2. In the KiCAD command prompt enter the following command:
   1. YOURPROJECTNAME: TEST141.kicad\_pcb – this is the file you want to panelise.
   2. YOURPROJECTLOCATION: C:\KiCADProjects\TEST141\PANELTEST141\– where the project is on your system.
   3. NEWPANELFILENAME: PANELTEST141\_TEST.kicad\_pcb- the new file you want to save your panelised file into (this does not already have to exist)

kikit panelize -p PanelConfig.json**^**

{YOURPROJECTNAME}.kicad\_pcb**^**

{YOURPROJECTLOCATION}{NEWPANELFILENAME}.kicad\_pcb

* 1. KiCAD command prompt can generally be found here- (C:\Program Files\KiCad\7.0\bin\kicad-cmd.bat) but will depend on you installation.

1. Open the new panelised board **through the file explorer** (not through the KiCAD homepage). As the panelised file is no longer linked to the schematic so should be opened in the standalone PCB viewer.
   1. Check mouse bites and fiducials are where you would expect

# Common Errors

## KiKit

### Tabs added incorrectly.

This error looks like no tabs have been added to your panelised board.

This could be because the incorrect library is being used for the tab. Try a tab from a library called “kikit” rather than “PCM\_kikit” which is also installed. This will depend on your installation method for kikit (pip install vs through the KiCAD add-on manager)

### TopologyException: unable to assign free hole to a shell at -900000 -1200000

This error can occur when running the kikit panelisation command.

Fix:

* Make a copy of your file.
* Remove all through hole components and board in board joints in a new file
  + Leave a reference component like a resistor that can be used to line up copying these components back into the panellised design.
* panelise this new file.
* On the finished panel copy and paste the missing components back onto the board using other components as a guide.
  + All components can be copied in a group from one board to another very easily in KiCAD.

See Section 5 Panelisation, for an example of what this looked like on an example board (in this case R1 was left in as a reference)It will then be necessary to measure pad sizes and add them as proper KiCAD pads, or reoutline cutouts to create a zero width line filled polygon (see 6.1.1 Tabs added incorrectly.)

## STEP Export

### Incorrect edge cut sizing

The step export in KiCAD defines the centre of a cut-out as the edge of the cut-out so if you are using a thick line to provide a cut-out, this will not show up in the STEP export. The fix for this is:

1. Draw an outline with the centre of each line at the point you require from draw elements.
2. Select all draw elements in this outline and right click -> create from selection -> polygon -> use centre lines.
3. Then make this a filled shape with 0-line width.

### Exporting STEP failed with BRepAdaptor\_Curve::No geometry in KiCad 7

This error is produced when attempting to export a STEP model of your board.

This was my fix for this error: This error is cause by using a polygon as the outline of the board, so I redrew my outline with lines and curves.

Another suggested fix on the issue is to make sure all edges of the PCB are aligned perfectly (check the (x,y) values in the properties).

# References

|  |  |
| --- | --- |
| [1] | KiCAD, “KiCAD Docs,” KiCAD, [Online]. Available: https://docs.kicad.org/. [Accessed 22 Aug 2023]. |
| [2] | FreeCAD, “FreeCAD,” FreeCAD, [Online]. Available: https://www.freecad.org/. [Accessed 14 Sep 2023]. |
| [3] | FreeCAD, “KicadStepUp Workbench,” FreeCAD, [Online]. Available: https://wiki.freecad.org/KicadStepUp\_Workbench. [Accessed 14 Sep 2023]. |
| [4] | J. Mrazek, “KiKit,” 22 Aug 2023. [Online]. Available: https://github.com/yaqwsx/KiKit. |
| [5] | Mouser, “The easy way to get symbols, PCB footprints & 3D models,” SymacSys, [Online]. Available: https://www.mouser.ie/electronic-cad-symbols-models/. [Accessed 14 Sep 2023]. |
| [6] | SnapEDA, “Design electronics in a snap,” SnapEDA, [Online]. Available: https://www.snapeda.com/. [Accessed 14 Sep 2023]. |
| [7] | KiCAD, “KiCAD Custom Symbols and Footprints,” KiCAD, [Online]. Available: https://docs.kicad.org/6.0/en/getting\_started\_in\_kicad/getting\_started\_in\_kicad.html#tutorial\_part\_4\_custom\_symbols\_and\_footprints. [Accessed 2023 Aug 22]. |
| [8] | J. Mrazek, “KiKit- Automation for KiCAD (Documentation),” GitHub, [Online]. Available: https://yaqwsx.github.io/KiKit/v1.3/installation/intro/. [Accessed 2023 Aug 22]. |