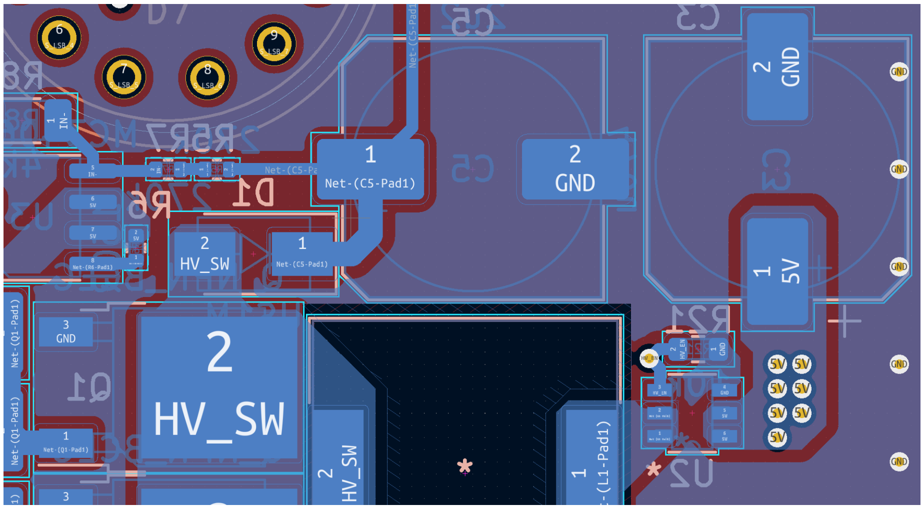


Guide to basic PCB design using KiCAD

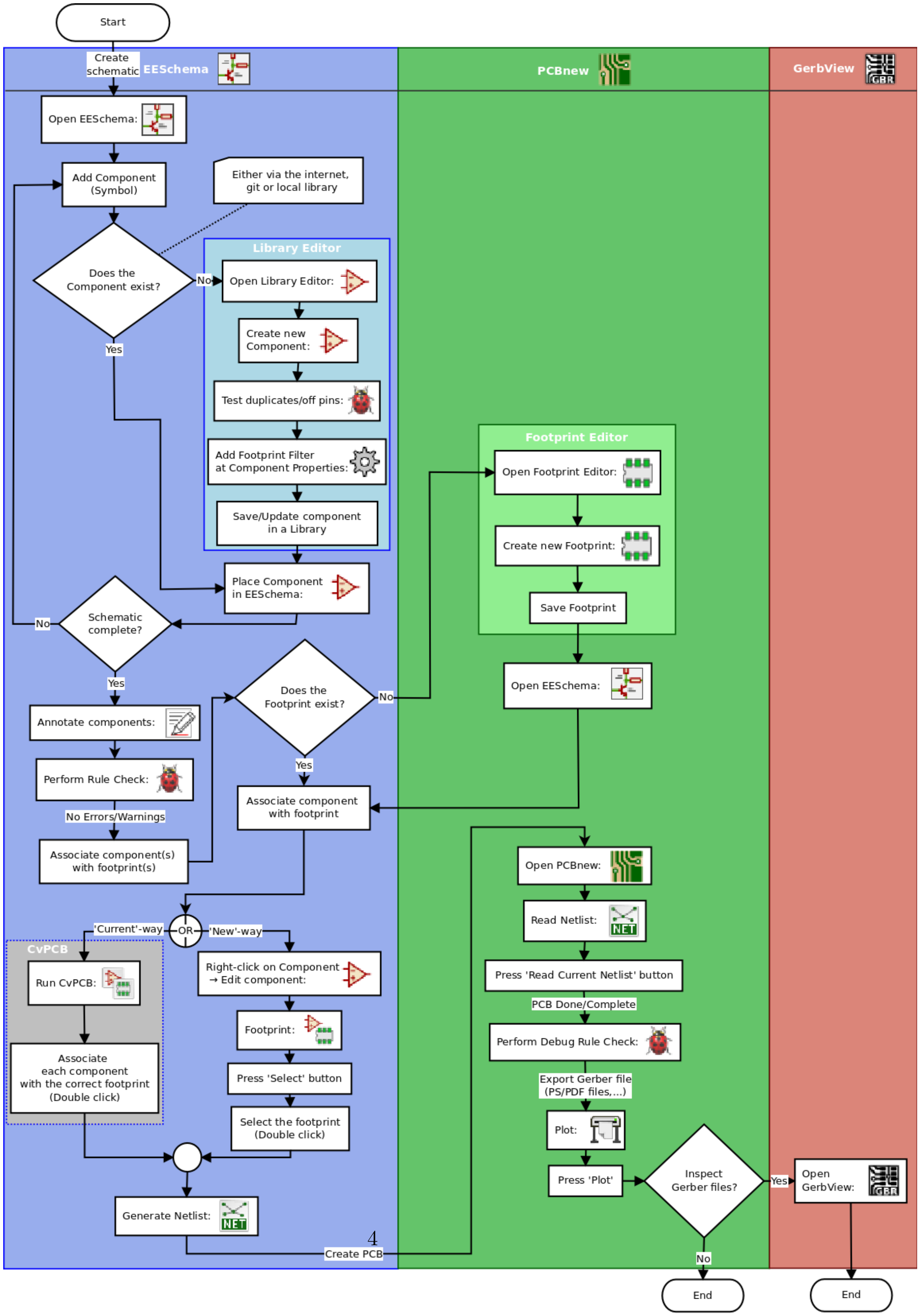
B. Groenen

January 2023

Contents

1 Introduction

In this guide, the free open-source software KiCAD 7.0 is used. KiCAD is a powerful solution which offers a 3D viewer and support for layouts with 32 copper layers. Furthermore, this guide will show step by step how to start designing basic PCB's. Fig. 1 displays the general steps you should take in designing a PCB using this software.



4

Figure 1: Flowchart with steps

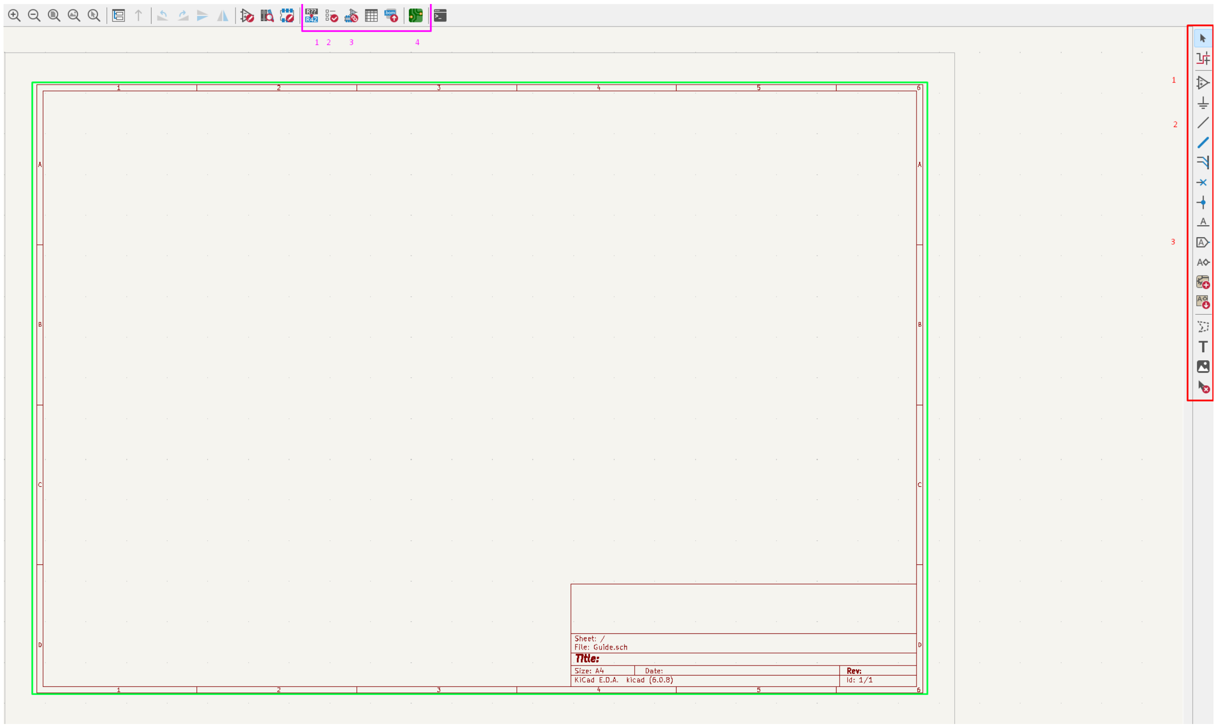


Figure 2: Schematic layout editor

2 Schematic layout editor

When you open KiCAD, you are greeted with a screen that shows your last edited project. Here you can create a new project and open the "schematic layout editor". This is where you create your electronic/electrical circuit of which you want to build a PCB. The following colors depict what options are of importance:

- **Green**: This is the area in which you layout your schematic.
- **Red**: The tools for designing your circuit are placed here. If you click the Op-amp symbol (marked by the red 1) followed by a click on where you want to place a component, you can choose from a wide variety of components that can be placed. In case your component is unavailable, libraries can be loaded in, or existing footprints can be edited. Importing footprints is shown in this guide.

  To connect components, click the single gray diagonal wire (hotkey 'W') and click on the terminals you want to connect together.
  In case a junction is wanted, either click the green dot and place it, or click with the green wire from the start location to the wire. All terminals

should have a connection, even if the pin is to be left unconnected. This "No connection flag" can be placed by clicking the blue cross (or pressing hotkey 'Q') followed by placing it on top of the pin that should remain unconnected.

In case you have a lot of connections on the same line, it might be easier to use global labels (hotkey 'CTRL+L'), which is the black with the letter 'A' in it. With this, names for nets can be defined, with nets being defined as connections to the same node.

Say that for example, in a circuit 5V is needed on a lot of connections. It would be easier to interconnect everything through a global label labelled +5V and connect that label to every point that needs 5V. If you do not do this, everything will have a separate netname.

- **Pink**: In the pink area, the tools are located for preparing the PCB for layout. To start off, you need to annotate the components and give each individual component a name. Luckily, this doesn't have to be done by hand, but this is done automatically with the annotate function. If you click the R??$->$ R42 icon (marked by the pink 1), the annotating menu pops up. In here you can just click annotate and it will do it for you automatically. Of course, you can also do this by hand if you want.

  Next, to see if the circuit isn't oddly defined, you need to do a ERC, short for Electrical Rules Check (marked by the pink 2). This menu checks whether there are any faults in the circuit like unconnected terminals or things that do not make sense for the program. Once the menu is open, press run to see if there are any errors. Any errors occurring should be displayed in the dialog. In case any errors are present: fix them, annotate the circuit again, and then check if the errors still persist. After all errors have been fixed, move on to the next step.

  Since a lot of components can be acquired in multiple packages/types, compatible footprints have to be assigned so the program knows which component package/type you will be using exactly. To do this, press the Op-amp symbol with an IC in front of it (marked by the pink 3). Here you can select a footprint for every component, also for connectors and battery-holders. In case they are wrong, changed or don't fit: you can change these later, even while laying out the PCB. Make sure to give each and every component a footprint. It can be that the required footprint for your project is not listed or wrong. If that is the case, we import footprint libraries. Editing a footprint is not discussed in this guide.

  To design our PCB, a netlist is needed. This draws blue lines (ratnests/rats) between footprints that should be connected together when we design a PCB. This can be generated by clicking File/Export/Export Netlist. A menu pops up with different options so you can pick which PCB design suite you want to export it for. Since we use KiCAD for this as well, keep it on KiCAD en press Export Netlist.

Now we can run PcbNew, the actual PCB designer of the software. It can be found by clicking the small PCB icon (marked by the pink 4). This opens a black screen which will be talked about in the next chapter.

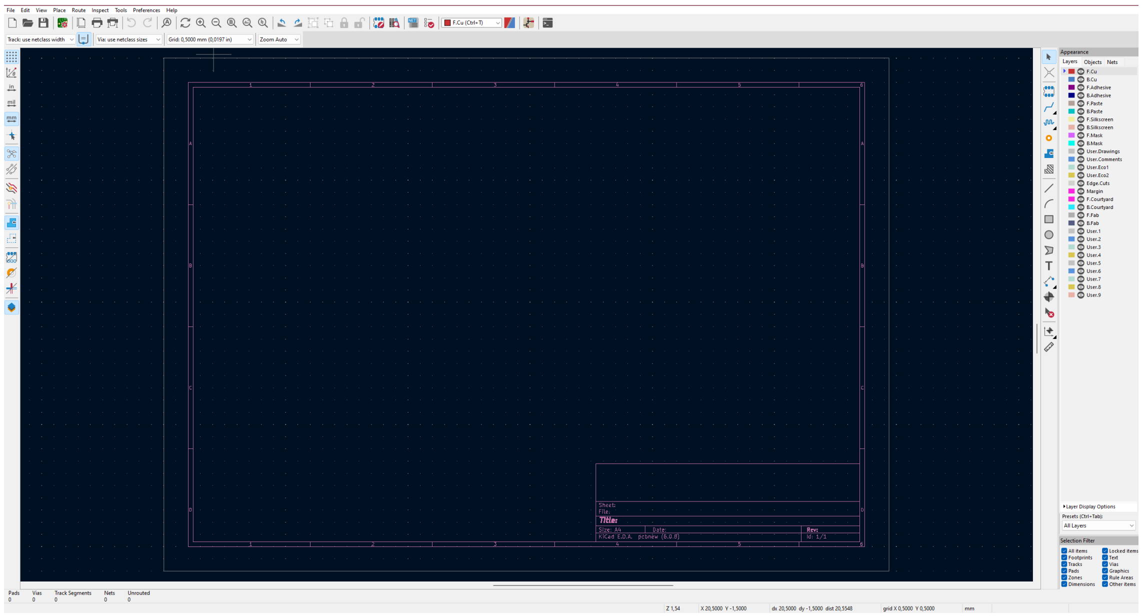


Figure 3: PCB layout editor

3 Pcbnew

Now we are in the actual PCB design part of the software. It has a wide variety of tools that can be used. In this guide, we will be using only a few of these tools.

3.1 Loading in your circuit

Loading in your circuit is very easy. To do this, press the grey block with 'NET' written in it in the top toolbar. Then select the netlist that you just generated in the schematic layout editor and press "Load and test Netlist" first followed by "Update PCB" if no errors pop up. If errors pop up, fix these first.

3.2 Laying out your circuit

Now that your circuit is loaded in, you can drag it around within the red sheet that you see in front of you. This is the area you work in. You can select individual components to drag them on a place you would like it to be. Take enough time to lay out your circuit efficiently to so it fits on a smaller surface plus meets all your requirements (bypass capacitors close to IC, Feedback voltage trace short, nothing under switching node, etc.). Also make sure that you follow the PCB manufacturers capabilities. For example, dont lay traces 10 micrometer from eachother, this is hardly produceable.

3.3 Layers and tracing

Of course, you want to use all sides and layers of the PCB. Standard, KiCAD generates a 2-layer PCB. This can be changed in the board setup under File/Board setup/Physical stackup and changing the amount of copper layers. Keep in mind: More layers is more expensive at the PCB manufacturer! Of course, you cannot place components on inside layers of the PCB, it is limited to F_Cu(Front copper) and B_Cu (Back copper).

You can select which layer you want to work on on the right hand side where all the layers are listed and clicking on the desired layer. You can also disable/enable viewing a certain layer. Every component is still at the top for now, but we can flip sides by right clicking a component and press Flip/Change sides or pressing F. To make a certain selection, select what you want to be able to highlight in the bottom right corner under "Selection".

3.4 Making an outline

When you are satisfied with the layout of your circuit, you can make an outline with the tools provided in the right toolbar. In that area, select the "Edge.cuts" layer and select "Add graphic lines" (if you want rounded corners select "Add graphic arc") and select the area you want your PCB to be. Standard, the grid is in 0.5 mm, this can be changed in the top toolbar.

3.5 Laying traces and copper pours

3.5.1 Traces

In order to connect the components together, traces or copper pours should be laid down. Traces are the "wires" connecting two or more components/pins together. Traces can be laid using the "Route track" tool seen in the right toolbar in figure 3. To lay a track, click the point you want to route from and where you want to route to. Sometimes, you have to switch between layers. This can be done amidst laying a trace by pressing 'V'. This places a via. A via is a small round pinhole that connects all layers of the board together for a specific net. If you need to carry high current, consider making the track-width bigger in the top toolbar or placing a copper pour. Calculating the width of a high current trace can be done by searching up the IPC-2221B standard online.

3.5.2 Copper pours

A copper pour is an area on the PCB filled with copper. Copper pours can be used for: heatsinks (ineffectively however), ground/voltage planes ,EMI reduction and as a high current carrier. To lay a copper pour, first click the "Add filled zone" tool in the right toolbar and click where you want the pour to start. Then a menu will pop up and you select the layer and net you want the pour to connect to. If you want the pour to be "floating", select the "no net" option.

Then, you can select an area where you want your copper pour. It is worth noting you have to manually fill the zones by right clicking the menu.

3.5.3 No-fill zone

Say that you need an area where nothing can be poured, then you add a no-fill zone. No tracks, copper pours or vias can be routed in there by standard, but you can change this in the settings. To place this, click the ”Add a rule area” tool in the right toolbar click where you want the area to start. You can select in the menu which layer you want the zone to be and what to keep out.

3.5.4 Adding text to your PCB

You can add text or images on the silkscreen layer or the copper layers on the board. Click the ”Add a text item” tool, click where you want the text to be. Then a menu will pop up where you can configure how you want the text and on which layer. If you want text in the copper layers, select either the layers ”F.Cu” or ”B.Cu”. If you want the text on the silkscreen layers use either the ”F.Silks” or the ”B.Silks” layer.

3.5.5 Soldermaskless areas

Sometimes, you need some bare copper on your board (to mount a heatsink for example). To do this, select either the ”F.Mask” or the ”B.Mask” layer, select the ”Add filled zone” tool and select the area you don't want a soldermask to be present. This works only for top and bottom layers.

3.6 Exporting your PCB for production

If you want to export your PCB for production, Gerber files have to be created to be sent to the PCB fabricator. First, see if your PCB has any DRC errors (Design rules check). To do this, press the same tool as used for the ERC in the schematic layout editor and press run DRC. If no errors are here, cou can continue. If not, solve them before continuing. Next, press File/Fabrication outputs/Gerbers(.gbr). Here, you select which layers have to be fabricated. You need to at least have the following layers for a 2-layer PCB to be able to fabricate:

- F.Cu - Front copper layer
- B.Cu - Back copper layer
- F.Silks - Front silkscreen layer
- B.Silks - Back silkscreen layer
- F.Mask - Front soldermask layer
- B.Mask - Back soldermask layer

- Edge.Cuts - Borders of the PCB

Once you have these layers or more, you can select an output folder and then press "Generate Drill files" and then press "Generate Drill files". This lets the drillmachine in the factory know where holes should be. Next press "Close" and press "Plot". Now all the files are generated in the output folder you selected. Place all these generated files in a .zip file and now it is ready to be sent to the PCB fabricator!

4 Loading in libraries

4.1 Footprints

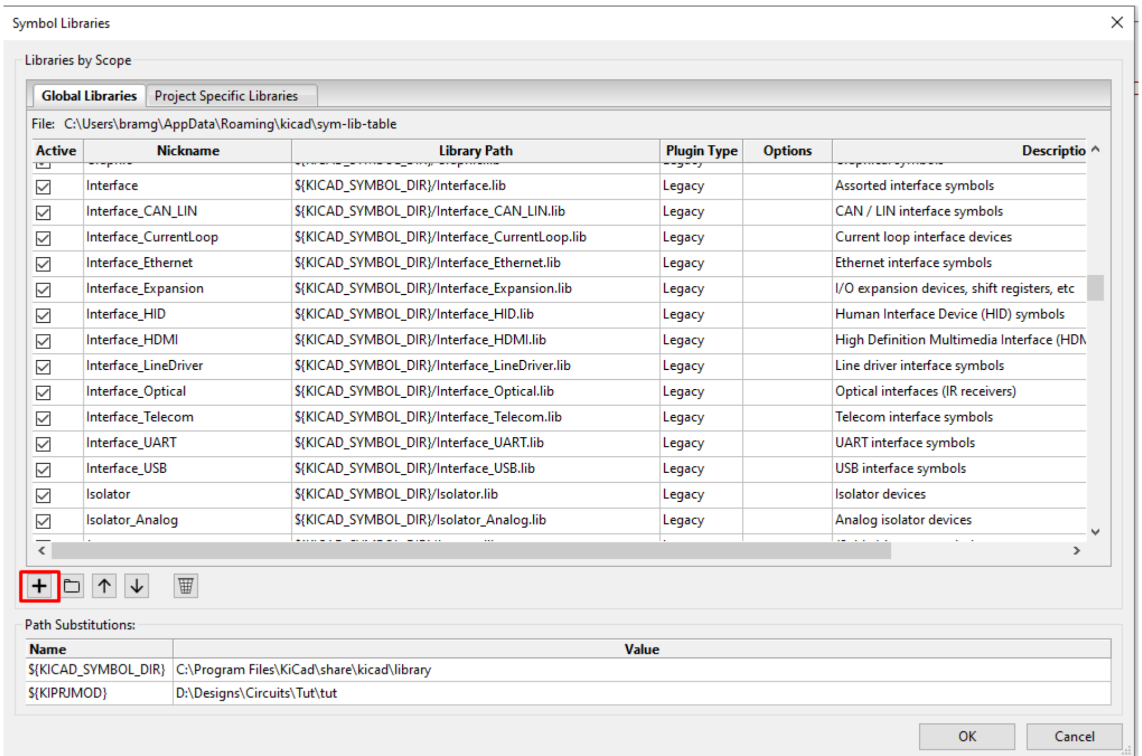


Figure 4: Footprint management area

4.1.1 Preparing the files

To import a footprint library into KiCAD, files have to be prepared first. Make a folder in the directory where you want your library to be stored. Tip: Make this a permanent location easy to find, or project specific. Then, name your newly created folder INSERTLIBRARYNAME.pretty. In here, you drop the files for your footprints, footprint files you drop in here have the ".kicad_mod" file extension.

4.1.2 Importing the created library

To import a footprint libary into KiCAD, you need to go to the "Assign PCB footprints to schematic symbols" tool seen in the pink area in figure 2, and then select "Manage footprint libraries" in the "Preferences" menu. Then press the + symbol highlighted in the red area in figure 4. Here, you can give your library a name and path to the library. This path will look something like: C:/Users/USER/PATH/INSERTLIBRARYNAME.pretty. Now the library is loaded in and is ready for use.

4.2 Symbols

4.2.1 Preparing the files

To import a symbol library into KiCAD, the contents of the library download have to be dragged into a folder where you want the library to be. Tip: Make this a permanent location easy to find or project specific. Next, press "Manage symbol libraries" in the "Preferences" menu. Here, although file extensions are different of the ones used in figure 4, press the + button seen in figure 4 and name your library and put the path of the library in. The path will look something like: C:/Users/USER/PATH/INSERTLIBRARYNAME.lib. The library is now imported and ready for use.