VeeCAD Notes for KiCad

These notes cover how to use VeeCAD with a KiCad generated schematic. This is a rehash of my original notes for interfacing VeeCAD with Eagle.

VeeCAD is a stripboard / Veloboard editor which allows you to lay out such boards.

I would like to say that while I don't use VeeCAD regularly, I have used it occasionally for years now. It has been a GREAT tool. If I want to do a one-off board of easy to moderate complexity, VeeCAD and stripboards have worked every time I've attempted a design.

Note: Updated text in this document is from a different project than the original text; therefore, schematics and pictures are not all of the same project.

Contents

General Information <u>1</u>	Select Board Size <u>8</u>
Create the Schematic <u>6</u>	Layout the Components
Export Net List <u>6</u>	Building the Stripboard 9
Import Net List <u>6</u>	Test the Board
Defining New Components 7	
Initial Component Layout	

General Information

VeeCAD's help is extensive. You can also find an RTF file of the manual at their download site, <u>https://veecad.com/downloads/</u>

I'm going to assume you have already installed KiCad and VeeCAD.

VeeCAD Hot Keys

Knowing these will make your life easier:

🕖 Settir	ngs				
Cursors	Display	Graphic Copy	Hot Keys		
Sel	ect Mode	Esc	~	Redraw	F6 ~
Bre	ak Mode	F2	~		
	(Mode	F3	~		
Wir	e Mode	F4	~		
Te	A Node	F5	~		

Other Keys you will want to know:

ESC ESC	Sometimes ESC by itself is not enough to get back to select mode, so
	I always press it twice.
Space	Rotates the selected object.
Shift Drag Lead	This will lengthen the wire/lead or move the component diagonally.

VeeCAD Components for KiCad

The easiest way to use VeeCAD with KiCad is to **install the VeeCAD components into KiCad**. Instructions for doing this are in the help section, *Getting Started* | *KiCad* & *VeeCAD*. These instructions are slightly out of date for KiCad 8, but essentially move the VeeCAD LBR file to a location in KiCad and add that library to KiCad.

Once you do that, you will find a library of common components in KiCad beginning with "V_":

0	- vl
lte	m
>	V_adc_dac
>	V_Battery
>	V_Connector
>	V_Diode
>	V_Linear
>	V_Logic
>	V_Microcontroller
>	V_Passive
>	V_Regulator
>	V_Relay
>	V_Transistor
~	MCU_ST_STM32F0

This library works by assigning known VeeCAD footprints to the VeeCAD schematic symbols. For example, if you select the common resistor symbol *R*, when you select it you will see at the bottom the footprint AX3_1 is assigned:

POLYSW	тсн	
R		
RESISTOR	<u>L</u> 0.5	
RESISTOR	0.25	
RESISTOR	<u>L</u> 1	
RESISTOR	L_2	
RESISTOR	<u>_</u> 5	
RESISTOR	L_PULLUP	
RT		
DV		
R		
Reference	R?	
Footprint	AX3_1	
Datasheet		
Description		

VeeCAD knows what AX3_1 is and will use that footprint.

If you want **to use an existing schematic** (or add symbols that aren't part of VeeCAD), then things are a little trickier.

In this schematic designed with default KiCad symbols, I want to use VeeCAD, so I edit

the symbol and assign the VeeCAD resistor symbol, AX3_1:

1	2 3
+VDC	+VDC
PWR_FLAG	A
SW1 UIA	Symbol Properties
A SW_Push IK 10K	General Pin Functions
	Name Value
PWR_FLAG	Reference R2
	. Value 1K
GND Continous HIGH PB-1	Footprint AX3_1
+VDC	Description Resistor
$\square \qquad \qquad$	
SW5 C5 U1E SW Burb 0.001/JE (102) 40106	

One has to do this for EVERY symbol in the schematic.

I have done this and it isn't as cumbersome as I was expecting. I DO copy the KiCad project (using File | Save) and convert the copy so I can use the KiCad version to produce a normal PCB if necessary.

What if you have a **schematic designed using VeeCAD** symbols and you want to **use KiCad footprints for a PCB**? I have not done this yet, but the process appears to be to clear out the footprint fields as shown above, then use the normal *Assign Footprints* tool.

What if you have a schematic symbol for which there is **no matching VeeCAD footprint**? In that case, type in your own, made up (custom), footprint name. Then in VeeCAD you will see that footprint unassigned and you can build your own custom footprint.

Logic components present a slightly different issue. VeeCAD has some designed, but it is missing many, and, for some, I prefer the KiCad version. In this case, go ahead and use the KiCad symbol. Then change the footprint to DIPnn, where nn is the number of pins in the IC:



What **footprint names do you have available to you**? If you use VeeCAD to open one of the symbol libraries:

XLDesigner	10/31/2024 4:54 PIM
ダ V_Alphabet.per	3/6/2022 12:07 PM
ダ V_Capacitors.per	3/6/2022 12:07 PM
V_Capacitors_Metric.per	3/6/2022 12:07 PM
🖉 V_Displays.per	5/18/2022 9:00 PM
ダ V_Relays.per	3/6/2022 12:07 PM
🖉 V_SMD.per	3/6/2022 12:07 PM
🕖 V_Standard.per	7/18/2022 6:29 PM

For example, V_Standard shows these footprints:



Create the Schematic

With an explanation of schematic symbols out of the way, you can create the schematic for your project in KiCad as usual using the appropriate VeeCAD or KiCad symbols. Once the schematic is done and passes ERC, you are ready to export the netlist to VeeCAD.

Export Net List

Exporting the netlist from KiCad is easy. Press File | Export | Netlist, select the OrcadPCB2 tab and click on the Export Netlist button. The netlist is saved in the same directory as the rest of the KiCad project.

Import Net List

Start VeeCAD, and click on Netlist | Import.

Make sure the format is Orcad PCB2 and specify the .net file you just created.

Netlist Format Orcad PCB2 Netlist Filename C:\Users\danh\Desktop\kicad projects\DebouncersVeeCad\DebouncersVeeCad.net Libraries C:\Program Files (x86)\WeeCAD\Library\V_Standard.per C:\Program Files (x86)\WeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\WeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\WeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\WeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\WeeCAD\Library\V_Relays.per C:\Program Files (x86)\WeeCAD\Library\V_SMD.per	🖋 Import Netlist	×
Orcad PCB2 Netlist Filename C:\Users\danh\Desktop\kicad projects\DebouncersVeeCad\DebouncersVeeCad.net Libraries C:\Program Files (x86)\VeeCAD\library\V_Standard.per C:\Program Files (x86)\VeeCAD\Library\V_Alphabet.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_SMD.per	Netlist Format	
Netlist Filename C:\Users\danh\Desktop\kicad projects\DebouncersVeeCad\DebouncersVeeCad.net Libraries C:\Program Files (x86)\VeeCAD\Library\V_Standard.per C:\Program Files (x86)\VeeCAD\Library\V_Alphabet.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per	Orcad PCB2 V	
C: \Users\danh\Desktop\kicad projects\DebouncersVeeCad\DebouncersVeeCad.net Libraries C: \Program Files (x86)\VeeCAD\Library\V_Standard.per C: \Program Files (x86)\VeeCAD\Library\V_Alphabet.per C: \Program Files (x86)\VeeCAD\Library\V_Capacitors.per C: \Program Files (x86)\VeeCAD\Library\V_Capacitors.per C: \Program Files (x86)\VeeCAD\Library\V_Capacitors.per C: \Program Files (x86)\VeeCAD\Library\V_Relays.per C: \Program Files (x86)\VeeCAD\Library\V_Relays.per C: \Program Files (x86)\VeeCAD\Library\V_Relays.per	Netlist Filename	
Libraries C:\Program Files (x86)\VeeCAD\ibrary\V_Standard.per C:\Program Files (x86)\VeeCAD\Library\V_Alphabet.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors_Metric.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per	$\fbox{\c:\content} C:\content\conten\content\content\content\content\content\content\con$	
C:\Program Files (x86)\VeeCAD\library\V_Standard.per C:\Program Files (x86)\VeeCAD\Library\V_Alphabet.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors_Metric.per C:\Program Files (x86)\VeeCAD\Library\V_Displays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per	Libraries	
C: \Program Files (x86) \VeeCAD\Patterns \777.per C: \Program Files (x86) \VeeCAD\Patterns \Atarado ATmegaDIL-104x83.per C: \Program Files (x86) \VeeCAD\Patterns \Atarado ATmegaTQFP44-79x79.per	C:\Program Files (x86)\VeeCAD\library\V_Standard.per C:\Program Files (x86)\VeeCAD\Library\V_Alphabet.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors.per C:\Program Files (x86)\VeeCAD\Library\V_Capacitors_Metric.per C:\Program Files (x86)\VeeCAD\Library\V_Displays.per C:\Program Files (x86)\VeeCAD\Library\V_Relays.per C:\Program Files (x86)\VeeCAD\Library\V_SMD.per C:\Program Files (x86)\VeeCAD\Library\V_SMD.per C:\Program Files (x86)\VeeCAD\Patterns\777.per C:\Program Files (x86)\VeeCAD\Patterns\Atarado ATmegaDIL-104x83.per C:\Program Files (x86)\VeeCAD\Patterns\Atarado ATmegaTQFP44-79x79.per C:\Program Files (x86)\VeeCAD\Patterns\Patter	^

From my schematic, I end up with this:



VeeCAD did a pretty good job assigning footprints. I used a custom footprint for the resistors and capacitors, so those came in as the default footprint. The rest are correct.

Now go to **Netlist** | **View and select the Validation tab**. This should be clean or at least the errors make sense. If not, you will have to fix .net and import again.

🥖 Ne	etlist							×
Nodes	Con	nponen	t Pins	Valio	lati	ion	Project	
Node Node Node Node Node	has has has has has	only only only only only only	one one one one one	pin pin pin pin pin		"u "u "u "u	nconnected-(U1-Pad12)" nconnected-(U3-Pad10)" nconnected-(U3-Pad12)" nconnected-(U3-Pad8)" nconnected-(U4-CV-Pad5)" nconnected-(U5-CV-Pad5)"	^

Look at the **Project View**. This lists components for which there are not yet a defined footprints:

🥖 Ne	etlist					×
Nodes	Compo	nent Pi	ns	Validation	Proje	ct
Net p	in :	l not	on	Compone	nt :	C1 ^
Net p	in :	2 not	on	Compone	nt :	C1
Net p	in :	l not	on	Compone	nt :	C2
Net p	in :	2 not	on	Compone	nt :	C2
Net p	in :	not	on	Compone	nt :	C3
Net p	in :	2 not	on	Compone	nt :	C3
Net p	in :	l not	on	Compone	nt :	C4
Net p	in :	2 not	on	Compone	nt :	C4
Net p	in :	l not	on	Compone	nt :	C5
Net p	in :	2 not	on	Compone	nt :	C5
Net p	in :	l not	on	Compone	nt :	C6
Net p	in :	2 not	on	Compone	nt :	C6
Net p	in :	not	on	Compone	nt :	C7
				-		

Defining New Components

If you use the VeeCAD library, hopefully you won't need to define a bunch of footprints. If you do need to create a footprint, take a look under Help for *Using VeeCAD* | *Custom Outline Editor*.

Once you build a custom outline, you can still re-import the netlist and it will now correctly assign the footprint to the custom footprint. This is how I handled the resistors and capacitors in my schematic.

Initial Component Layout

If you haven't read it already, read the Help section on laying out the stripboard (*Using VeeCAD* | *Layout Hints*) has lots of good tips. My favs are:

Start with a large donut board. This creates a rats nest like I'm used to in KiCad and EagleCad. With schematic and the rats nest, unaffected by tracks, I can position parts into a roughly correct area pretty quickly.



Once I have the components roughly where I want them, then I select an actual stripboard.

Creating a **holding area** is a great tip too (Layout Hints | Creative Tips). This allows you to focus on adding each component.

Because I prefer starting with a donut board and getting everything into rough position, the holding area isn't as useful for me. Instead, once I get the subsections of the circuit laid out, I wall off each subsection with breaks, work on the subsection, then join subsections. This too is covered in the help (*Layout Hints* | *Large Stripboard Layouts*).

Select Board Size

I don't keep a very large selection of stripboards on hand. I do try to keep some fairly large ones (6x3"). It's easier to build a circuit on a big board and then cut it down to size than on a board too small.

Also make sure the proposed board size is going to fit the enclosure you are planning.

Downsizing the board after laying it out is possibly a do-over.

🗡 TextPtr - VeeCAD
<u>File E</u> dit <u>S</u> elect <u>B</u> oard <u>N</u> etlist <u>T</u> ools <u>H</u> elp
▶ ▼ ▲ 1

Layout the Components

You are ready to go! Again, the Help section has lots of tips to help lay out components on the board. If you've never done this or it's been a long time, don't be surprised if you rip it all up and start over again once or twice.

Don't forget ALT-DRAG lengthens wires and leaded components

As you lay out the board, click on component pins o view all of the wires/pins on the same network. This makes it easy to see where you need to add a wire.

Once the board is laid out it will look something like this:



Building the Stripboard

Once I'm ready to build the board, I print the front diagram and the back. Use the front to layout the components and the back to ream the holes to break the tracks. It is critical

that you triple check the placement of these templates so top and bottom match or you've wasted a board, a lot of time, and perhaps some components.

Recently I started using 8.5x11 vinyl sticker paper to print the top of the board. I can then stick that on the board for component layout:



I *think* this is safe for low-voltage low-wattage circuits. For safety, don't use vinyl for the top, use paper. Layout the board, then tear out the paper.

For cutting the tracks, I always use a paper guide and remove it after the tracks are reamed:



I built my [spot face cutter] track cutter using

http://www.instructables.com/id/Stripboard-track-cutter/

VeeCAD points out a purpose built Farnell spot facecutter. I was able to purchase one of these, recently, at Newark:

https://www.newark.com/vero/22-0239/cutter-spot-face/dp/10WX4170

Before Cutting Tracks

- Very carefully align the top template and stick (or tape) into place. Make sure the template aligns with the holes by sticking a needle thru a few holes to check.
- Hold the bottom template in place. Flip between top and bottom of the board to verify all of the components line up properly.
- Align the bottom template perfectly and tape into place. Again stick a pin thru some holes to make sure the templates and the board all match.

To cut the tracks:

- Using the track cutter tool, cut out all of the 'X' marks in the template.
- Remove the bottom template.
- Stick a needle thru the hole in the center of each and every reamed circle.
- Now look at the top and verify every 'X' has a pin pick (and there are no unexpected pin pricks indicated an improperly placed 'X').
- Use a continuity tester to verify every cut track is fully cut. It is guaranteed there will be some tracks not fully cut and you will need to ream the 'X' a bit further.
- Finally, from the top side, use a pin to punch a hole in every single component leg so the component can be inserted into place.



Installing and Soldering Components

As with other boards, you want to start w/ the lowest profile components and work your way up. If wires go across components, do those last. So I started with the jumper wires:



I then worked my way thru all the components:



here is the bottom. Not pretty, but not the worst solder job I've done:



All of the cheap components were soldered in. The more expensive stuff is socketed. I also socket CMOS chips. I've been rather unlucky with destroying a lot of CD40106 ICs with static.

Test the Board

Before inserting ICs into the sockets, I test the board to verify voltages are where they should be, especially for MCUs and other socketed chips.

- Connect continuity tester to Vcc. Test all of the points on the board you expect to find Vcc such as Vcc pins on ICs, etc.
- Due the same with the Ground connection.
- With the ICs still not installed, power up the board.
- Test you have proper voltage on all Vcc pins of the ICs and any other place you would expect. It would be a good idea to also verify GND is oV on all ICs.

Once I believe everything is in order, I insert the socketed chips and proceed with final testing.

Reworking the Board

Mistakes will undoubtably be made. I have found it pretty easy to unsolder pins from veloboard with solder wick so I can replace components.

If you find an actual mistake in the design, look at the board in VeeCad and figure out where you can install a jumper wire to correct the problem.

Here is my first KiCad to VeeCad project.



I wasn't sure I would get all of these components to fit onto a single 6x3" board. Note my use of 1/8W resistors to minimize their size.