EEC 134 RF/MICROWAVE DESIGN-WINTER 2017

Professor. Xiaoguang “Leo” Liu

Team TEAM

Lap Hoang

RF System Design on KiCad and Soldering

Abstract

This paper presents the process of how we transform our RF system design on paper into a Printed Circuit Board (PCB) and discusses the process of soldering components on the board as well. Also, the report will provide some advices that might be helpful for the future student who takes RF/Microwave Design course.

RF PCB Design using KiCad

In this class, we used KiCad as the software to create the PCB design. The basic functions of the software were introduced in the first quarter and we also had chances to practice on the software by doing three PCBs throughout the last quarter. The TAs provided us with materials such as video links and some documents that helped us get familiar with KiCad. The links could be easily found on Youtube and will be provided in the References section of this report.

<u>Design on paper:</u>

A detail design on paper is what we need to begin with KiCad. The paper design provides what components are used and how these components are connected to others. Our design on paper was mainly taken care of by another member of our group. The design is presented in figure 1:

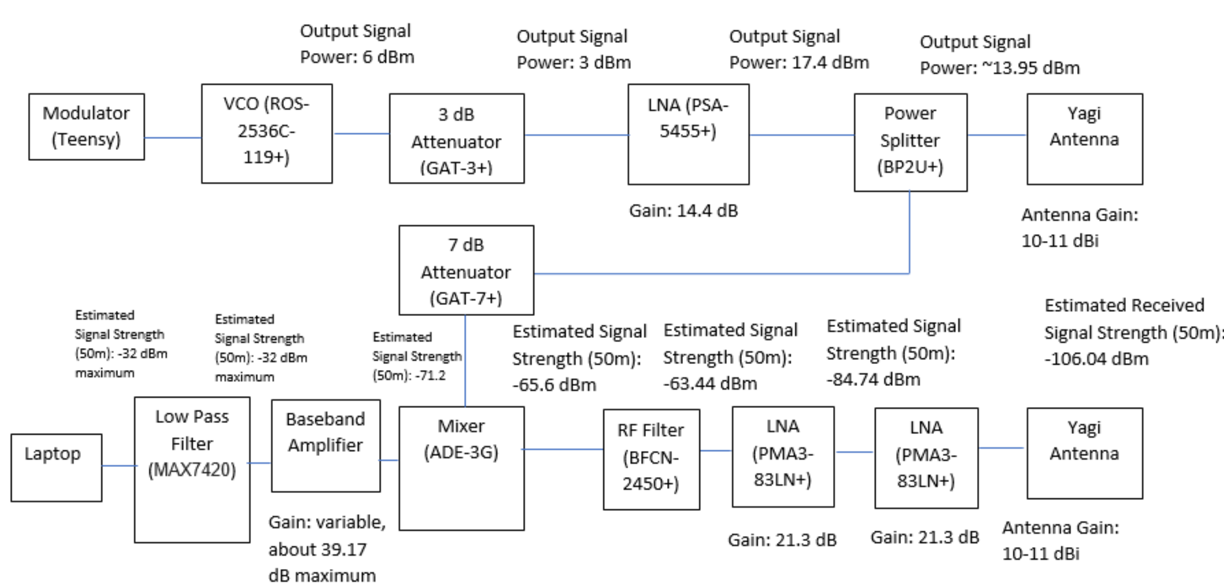


Figure 1. The block diagram of the radar system

KiCad Main Interface:

The main interface of KiCad is presented in figure 2:

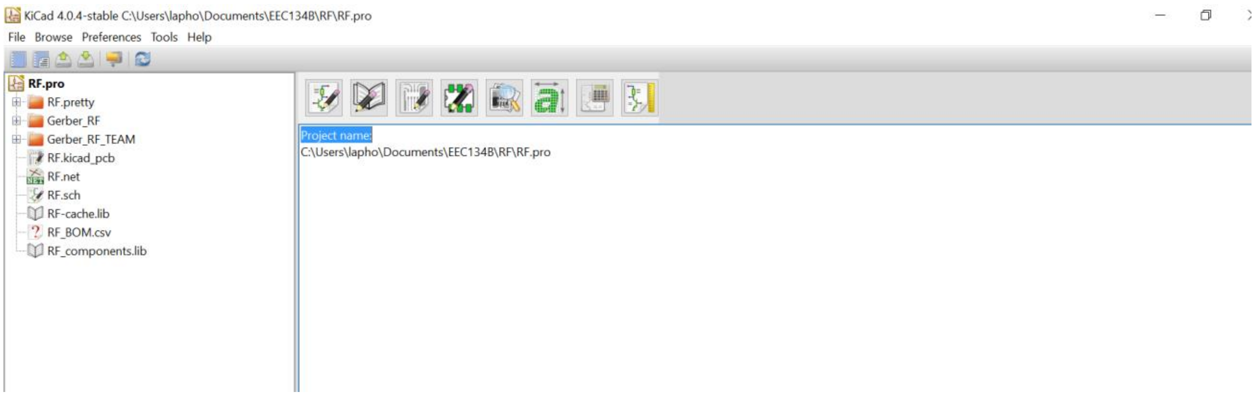


Figure 2. KiCad main interface

On this interface, there are eight icons which lead to eight different windows. Each window performs different function of KiCad. Figure 3 is the close-view of the icons and list of the functions that each icon provides.

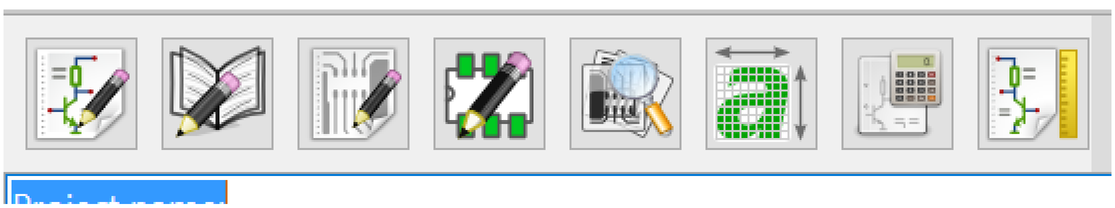


Figure 3. Close-view of Icons on KiCad interface

Going from left to right:

- Eschema-Schematic Editor: to create the schematic of the design showing the connection between components.
- Schematic Library Editor: To create and edit symbol for components.
- PCBnew- Printed Circuit Board Editor: To create the actual layout for the circuit design, displaying how the PCB looks like.
- Footprint Editor: to create the footprint of components.
- Gerview- Gerber viewer: to display each layer of the PCB (Gerber files).
- Bitmap2Component: to convert bitmap images into Eschema or PCBnew elements (not used)
- PCB Calculator: to calculate width track, traces.
- PI Editor: to edit worksheet layout (not used)

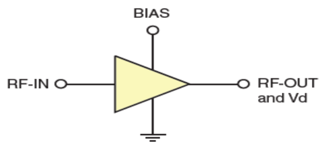
Schematic of RF system:

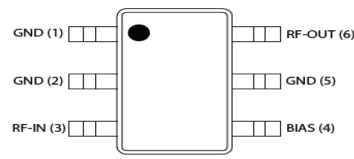
The next step is to create the schematic of the design and it begins with generating symbols for all the components and storing these components in a separate library for your own project. All the common components such as resistor, capacitor, inductor, power source and ground are available in the main library, there's no need to create those components. To begin, first thing you need is the pin description of the component, it could be found in the datasheet. Below is an example of how pin description looks like:

General Description

PSA-5455+ is an advanced wideband, high dynamic range, low noise, high IP3, high output power, monolithic amplifier. Manufactured using E-PHEMT* technology enables it to work with a single positive supply voltage.

simplified schematic and pin description





Function	Pin Number	Description *(See Application Circuit, Fig. 3)*
RF IN	3	RF input pin (connect to RF-IN via blocking cap C1 and Pin 4 via L2)
RF-OUT & Vd	6	RF output pin (connected to RF-out via blocking cap C2 and supply voltage Vd via RF Choke L1)
BIAS	4	Connected to Vs via Rbias. (Connect to ground via C4 & R1)
GND	1,2,5	Connections to ground

* Enhancement mode pseudomorphic High Electron Mobility Transistor.

Figure 4. Pin Description of PSA-5455+

The symbol of the component (PSA-5455+ in this case) will be generated based on this description with pin numbers and corresponding functions as in figure 5:

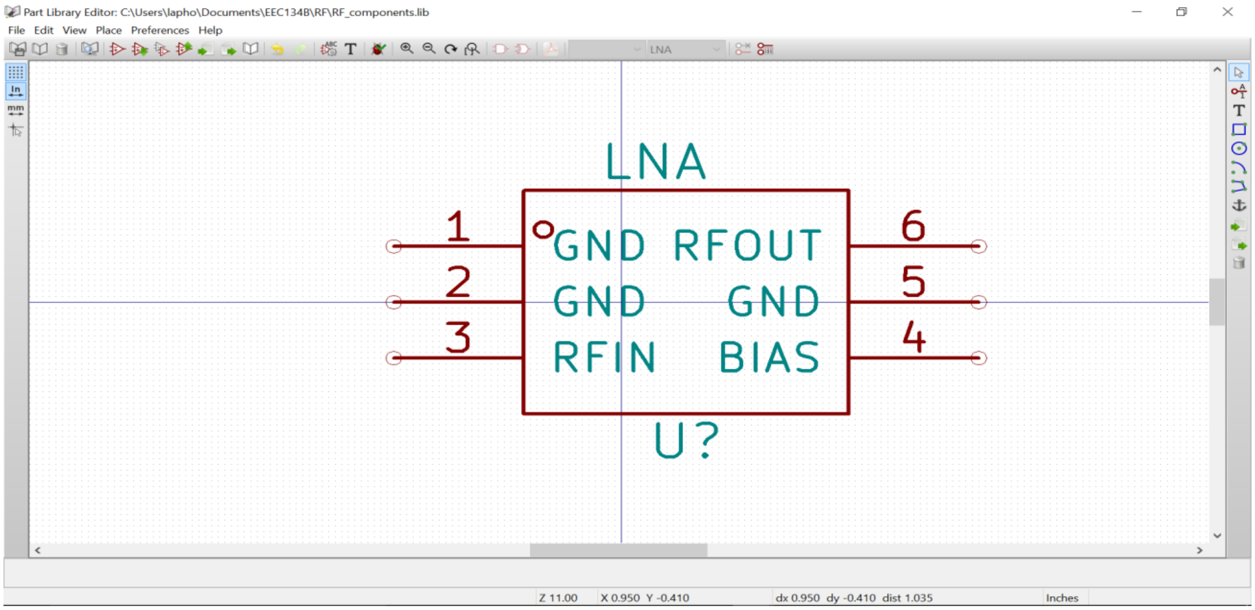


Figure 5. Symbol of PSA-5455+

Only few steps are needed to create the symbol: making a shape, adding pins and pin indicator (the little circle-used to determine the direction of the pin) and saving the symbol to a library. These steps are done using the tools provided in the toolbar which is located on the right edge of the Part Library Editor's interface. It's important to create a separate library for your own design because the library of KiCad is not regularly updated. The remaining components are similarly created and saved in the project's library.

The next step is to connect all the components together based on the design on paper and this step is done using the Schematic Editor. As similar as the Part Library Edit, the toolbar of the Schematic Edit is located on the right edge of its interface. The schematic is generated by placing the component on the platform and wiring them together using the tool provided on the toolbar. The main tools are placing component, placing wire and placing power port tools. Fortunately, these tools have hotkeys: a, w, and p respectively, so it helps us moving forward much faster. The complete schematic is created by repeatedly adding components and wiring following the paper design. Below is the complete schematic of our RF design:

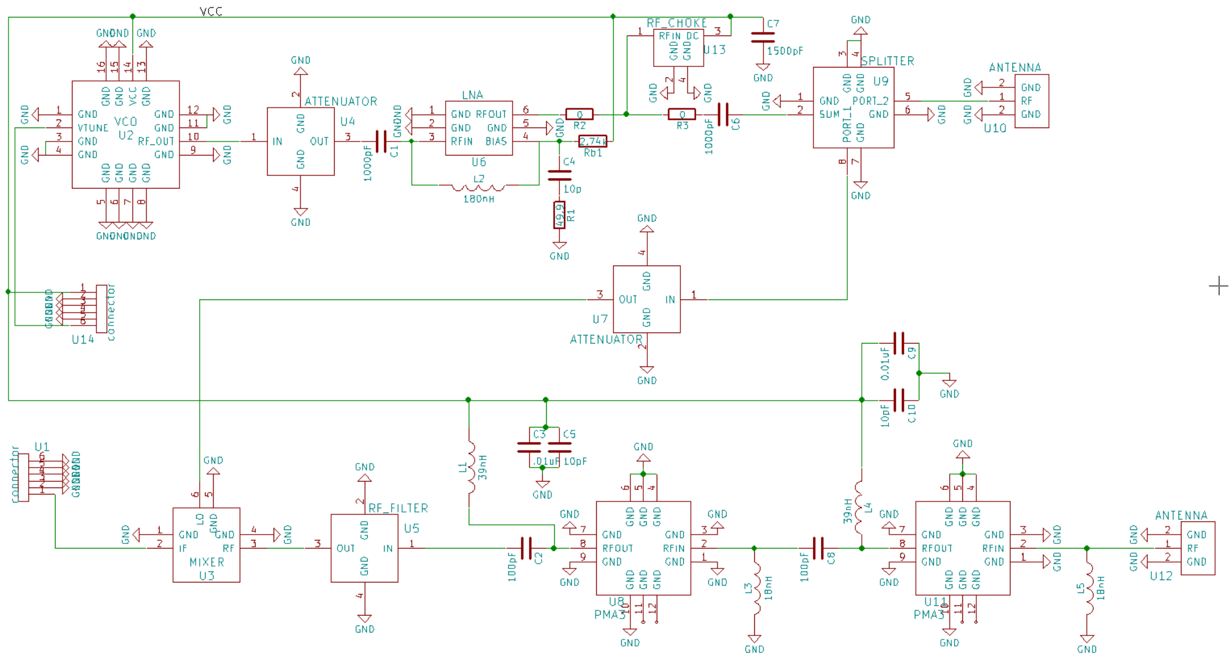


Figure 6. Schematic diagram of the RF board

One important thing should be noticed is that some components require a recommended application circuit. This circuit ensures the components work properly and it is provided on the datasheet. It is necessary to check on every components' datasheet to acknowledge if they have a recommended circuit to follow.

Recommended application circuit for PSA-5455+:

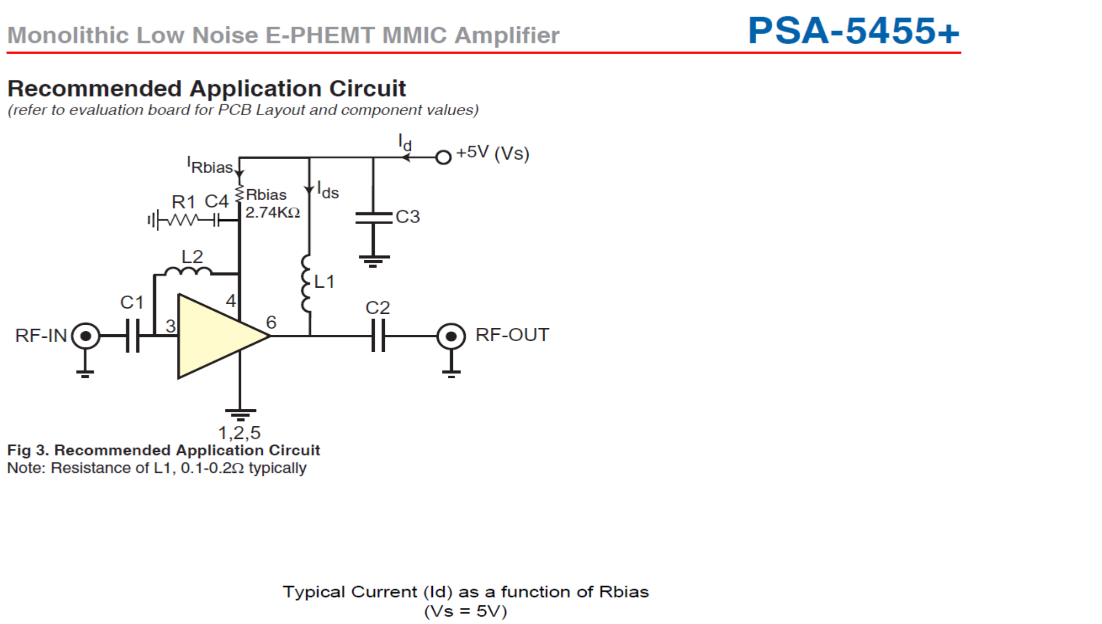


Figure 7 shows the recommended circuit of PSA-5455+:

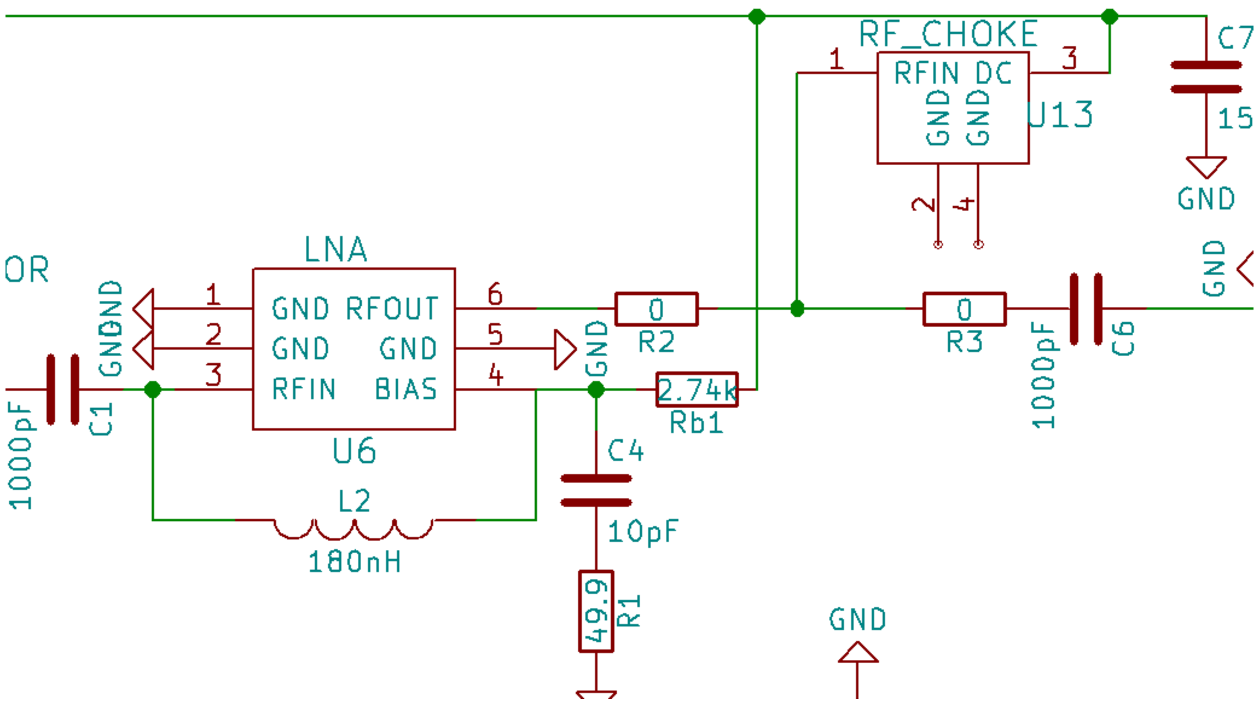


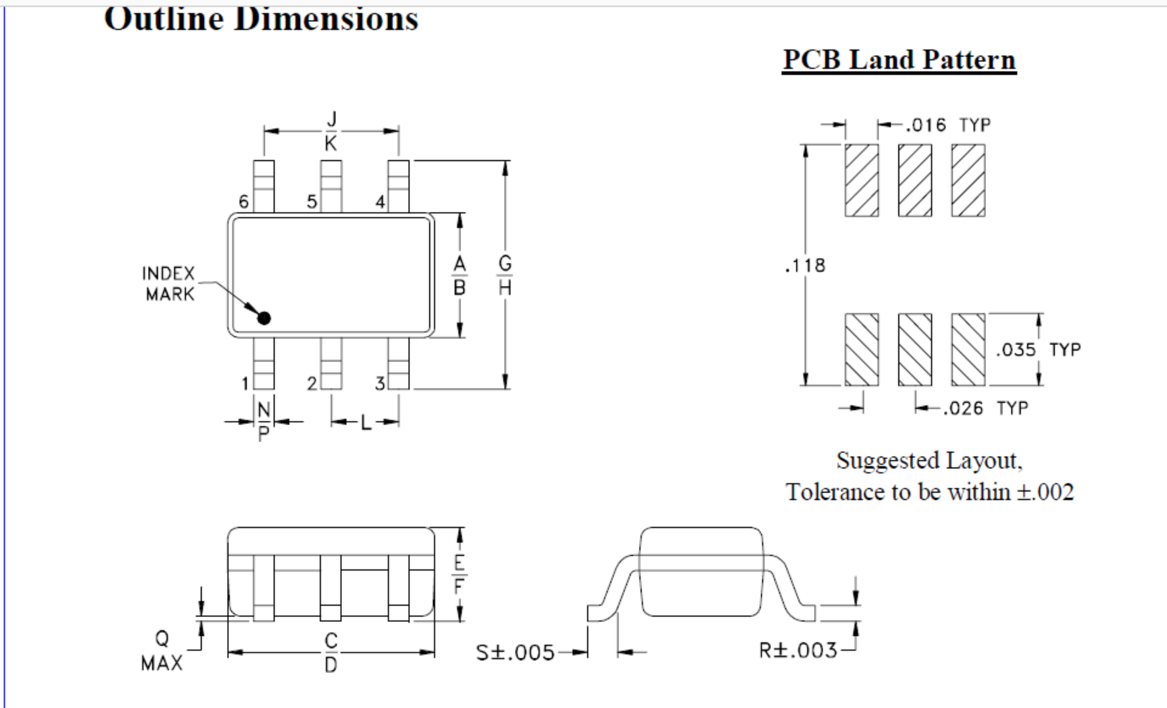
Figure 7. Schematic of recommended circuit for PSA-5455+

The final step to complete the schematic is to check for any error using the DRC tool (the Bug icon on the top of Schematic Editor). The DRC shows unconnected pins or pins without driving signals. They could be an errors or may just be a pin which is intentionally left unconnected. It is better to compare the schematic and the paper design to see if they are similar. If everything is fine, then we can generate the netlist for the schematic by clicking on the netlist icon on the top toolbar. Now, it is time to move to the next step, making footprint for components.

Creating Footprint for Components:

A footprint shows arrangement of the pads that physically and electrically connect the components to the PCB. In KiCad, the footprint is generated using the Footprint Editor. Similar as creating symbols of components, it is also important to create a separate library for the footprints. The toolbar is located on the right edge of the Footprint Editor as well. The first thing we need to know is the dimension of the footprint which is provided on the datasheet or on a separate document depends on the vendor. If the footprint dimension is not on the datasheet, it could be easily obtained by searching online.

Here is an example of a footprint (PSA-5455+):



CASE #	A	B	C	D	E	F	G	H	J	K
CA1389	.045 (1.15)	.053 (1.35)	.073 (1.85)	.089 (2.25)	.031 (0.80)	.039 (1.00)	.079 (2.00)	.091 (2.30)	.051 (1.30)	.051 (1.30)

CASE #	L	M	N	P	Q	R	S	T	WT. GRAM
CA1389	.026 (0.65)	- -	.006 (0.15)	.012 (0.30)	.004 (0.09)	.007 (0.165)	.012 (0.31)	- -	.010

Dimensions are in inches (mm). Tolerances: 2Pl. ± .01; 3Pl. ± .005

Figure 8. Footprint dimension of PSA-5455+

The footprints of resistor and capacitor are available, so it is not necessary to create footprints for these parts. After creating all the components footprint and saving them in the same footprint library of the project, we are ready to move to next step, associating the components with its corresponding footprint.

Associate Footprint:

Figure 9 shows the tools needed to associate the footprints to the components (located on the top of Schematic Editor)

Figure 9. Tool used for annotation

The first thing to do is to annotate the components on the Schematic Editor using the first button from the left. Next step is to generate the netlist by clicking on the third button. The last step is to open the CvPcb window using the second button from the right, the CvPcb looks like as figure 10 below:

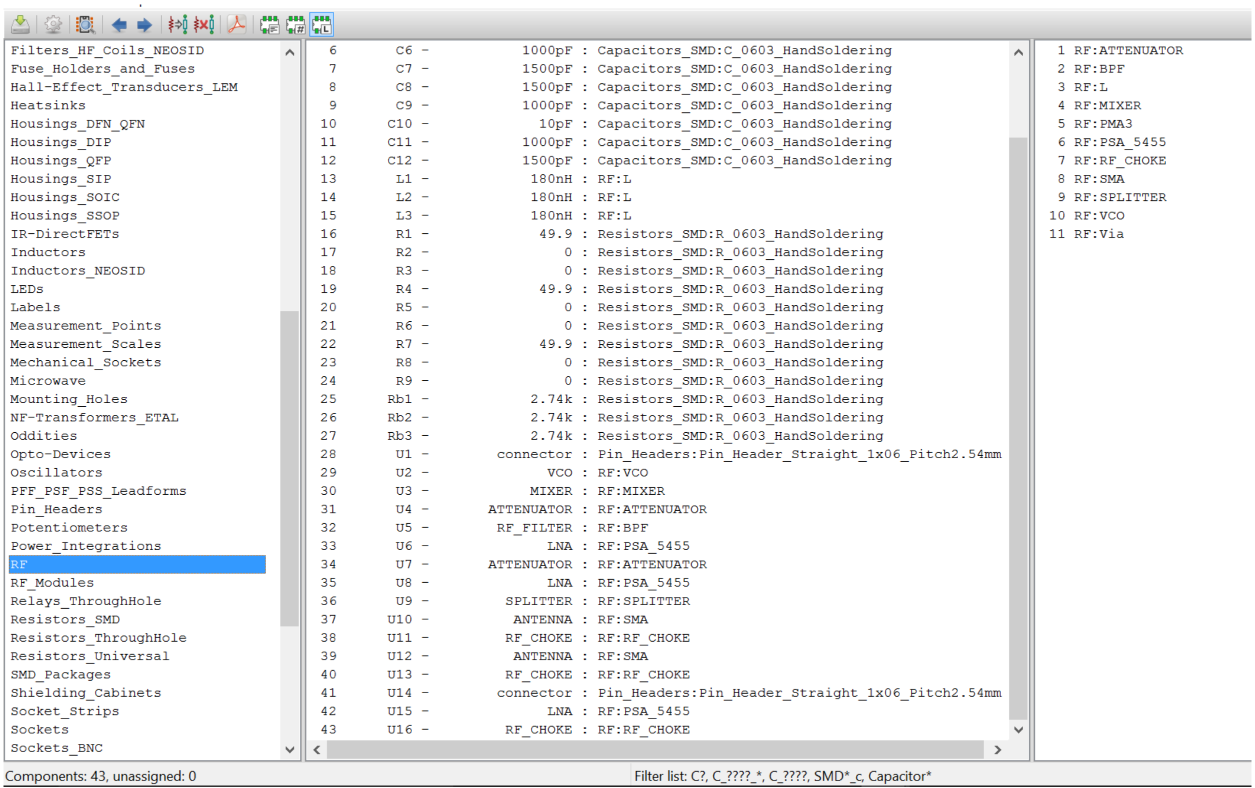


Figure 10. The interface of CvPcb (Component association tool)

The left column is the list of footprint library and the right column is the list of footprint in the selected footprint library (RF is selected). The middle column lists all the components which need to be associated with a footprint. This is an easy task. What need to do are to choose a library on the left column, then click on a component on the middle column, and finally click on the corresponding footprint on the right column. This step could be done quickly. However, it is necessary to check back again to see if any component is associate with a wrong footprint because any mistake at this point could make great impact on the next step, creating layout for the PCB.

<u>PCB Layout:</u>

The last step of design PCB is to create the layout for the circuit and this step is done using PCBnew platform. When the PCBnew platform opens, first thing to do is to read the netlist of the circuit. PCBnew will generate a randomly organized network of components based on the netlist. It is totally a mess, especially if the circuit have so many components. However, we can spread out this mess using the provided tools and separate the components for easy access. After all the components are spread out, they need to be placed on the board in an organized fashion that makes it easy to run traces around the board to connect the components together. Placing traces is really important in the RF board since high frequency signal has several limitations that have to be concerned. For example, the traces should be place on the top layer only or the transmission line track width needs to be calculated to satisfy the 50 Ohm impedance matching. The track width calculation for transmission line can be done using the PCB Calculator of KiCad or other calculator with similar function which can be found online such as Wcalc. The interface of the PCB Calculator of KiCad is shown in figure 11:

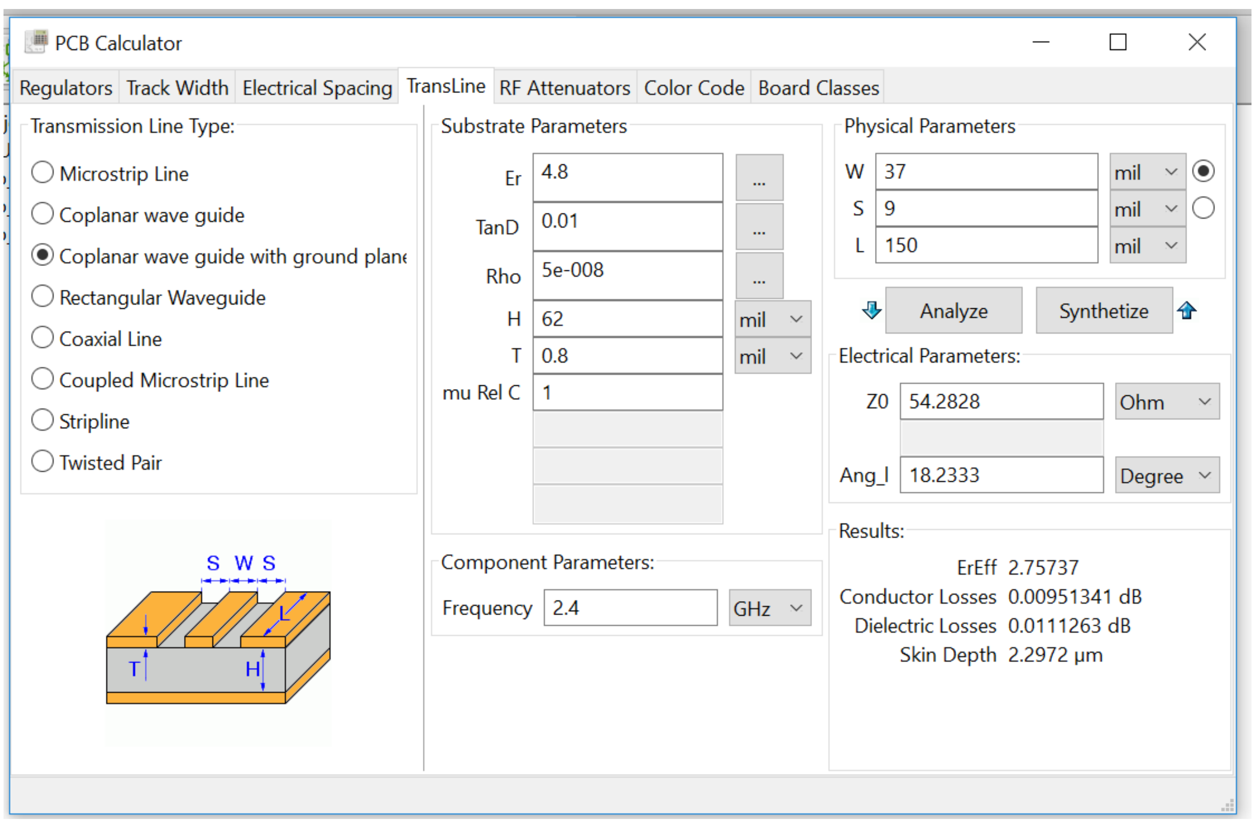


Figure 11. PCB Calculator interface

The substrate column lists the properties of the PCB material which we do not have control because these numbers depend on the vendor's criteria. However, we do have control over the dimension of the trace width (W) and separate distance or clearance (S). Changing the trace width and clearance will change the characteristic impedance of the transmission line. Besides the transmission line, the RF board requires a via-fence to prevent any interfered signals coming from the surrounding environment. The via-fence is created by placing a lot of vias along the transmission line. These vias connect the top and bottom layers which is both grounded and create a shield to protect the RF signal in between.

Running traces is a time-consuming process and one should pay more attention on doing that. One note is that not to connect ground pads together since there are a lot of them and these pads will be connected in the next step, so connecting them just wastes more time.

After connecting all the components together, next step is to fill up both top and bottom layers with copper as a ground network. At this point, we basically finish the layout of the PCB. There are several extra steps that we can do to make the board look nicer such as making the values of components invisible or adding some graphic or text on the edge.cut layer. The final step is to generate the gerber file and drill file in order to send out to the vendor for manufacturing.

The complete layout of our RF board:

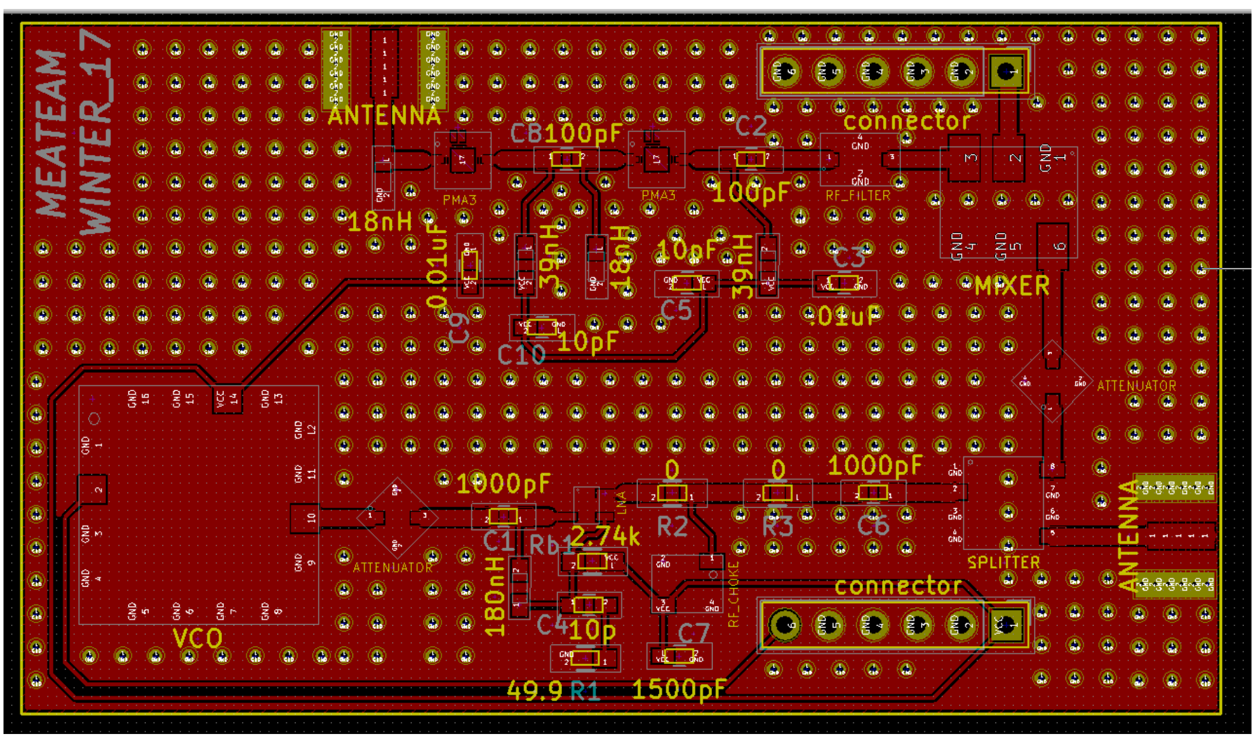


Figure 12. The RF board layout

Soldering the PCB

Equipment:

Soldering components on the PCB board requires several equipment such as soldering iron, hot plate, and magnify glass. Fortunately, the Engineering Fabrication Laboratory (EFL) just finished innovation at the beginning of the quarter and we had a chance to do the soldering job at well-equipped stations in the EFL. The soldering stations in EFL have soldering iron, hot-air flow, solder-sucker along with a Protoflow machine which is ideal for soldering small parts.

Soldering and desoldering:

Since the RF board contains all the SMD parts, it is more effective to do the soldering using the Protoflow machine rather than hand-soldering. There are three basic steps to solder the PCB. The first step is to place the components on the board with applied solder paste and flux. Next, we set the Protoflow machine to proper mode and put the board on the tray inside the machine and let the machine do its job. Finally, we wait till the cool down and then check the board for any misplaced components and bad connection. The machine usually does a very good job, but sometimes it flow the components out of place and we need to fix it. The hot-air tool is really effective for desoldering SMD components, but remember to lower the air flow to prevent the parts be flown away. To check for bad connection, there is an electronic multi meter that could be used, but you need the TA to log in the computer since EEC134 students do not have access to this computer.

Notes:

- Start placing with small components since the big components could block the access to others.
- If possible, place all the component at once and then use the Protoflow machine, it prevents burning the components due to pre heat so many times even though it just rarely happens.
- Use the magnify lamb to place components if do not have a clear vision. I used it most of the time.
- Ask the TA at the EFL if you have any confusion, it is better to ask rather than make a mistake.
- Do not apply too much solder paste, the extra solder at the pin could draw more current, create solder beads and thus reduce the efficiency of the circuit.
- Use solder flux to make the good connection and prevent creating solder beads in the soldering process.

References:

KiCad Program

Minicircuit. Datasheet of PSA-5455+ Low Noise Amplifier.
Web. https://www.minicircuits.com/pdfs/PSA-5455+.pdf

Minicircuit. Landing Pattern of PSA-5455 Low Noise Amplifier.
Web. https://www.minicircuits.com/pcb/WTB-534-5+_P02.pdf

Youtube. Contextual Electronics. KiCad Tutorial 2016 Series.
https://www.youtube.com/watch?v=EJJXThcSyKo&list=PL5iUxv3Op2fOpVASHvcpM2O4UO0yMfKJi&index=2

All the figures were generated by taking screenshot in KiCad or of the datasheet of Minicircuit PSA-5455 Low Noise Amplifier