

EEC134
Team Diode Hard 3

Application Note for PCB Design

Introduction:

This will be a guide on making footprints and creating a PCB using CircuitMaker. The RF PCB will be used in this example as it requires more planning than the baseband PCB.

Footprints:

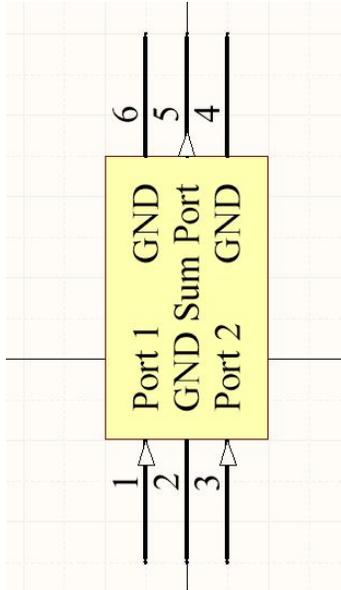
Many components that will be used most likely do not have schematic symbols or footprints created for them, so to begin I would recommend making the footprints so that once you get started designing the schematic and layout everything is available. In CircuitMaker go to the View tab and click libraries. From there search for your part and if it is there and made by a reputable company such as SparkFun you could use it, but I recommend creating your own. Click build your own component. From the window that pops up put the name and any other info. After that click add a new symbol.

Once you begin creating your own symbol you have to look at the pin descriptions of your component which is on the data sheet of the component. I will be using the SP-2U1+ splitter as an example.

Pin Connections	
SUM PORT	5
PORT 1	1
PORT 2	3
GROUND	2,4,6

SP-2U1+ Pins

There are a total of six pins for this component so we will need six connections. I recommend just using a simple rectangle the symbol. From the home tab place a rectangle down. Once a rectangle is down we need to attach pins to it from the home tab. We will need six pins. Place a pin and double click the pin. Change the pin display name to the name of the connection, for example "Sum Port". Next change the designator the pin number. This is very important. The designator is how all connections are made to it from schematic to design is made. Do this for all pins and then attach it to the rectangle so that the display name is in the rectangle is in it. It should look like this:

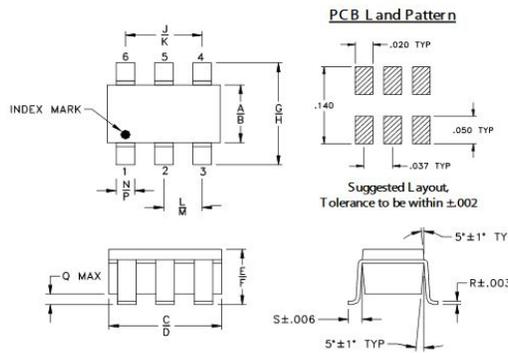


SP-2U1+ Symbol

Next up we will create the footprint which is where the physical component will sit on your finalized PCB. Make sure you triple check this part and have partners check it as well, this part can make your PCB not work if not done correctly.

From the SP-2U1+ tab click “create new footprint”. Now look at your data sheet again and look for the outline drawing, which will look like this:

Outline Drawing

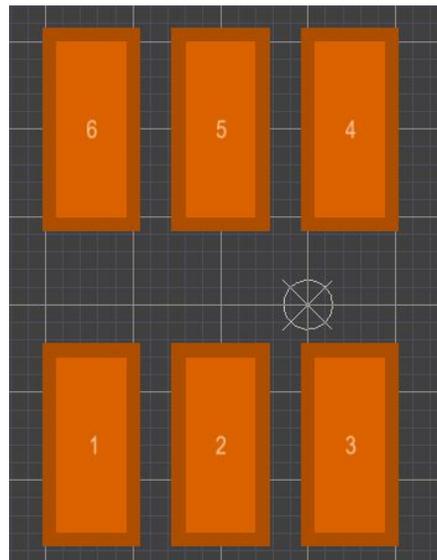


Outline Dimensions (inch/mm)

A	B	C	D	E	F	G	H	J
.052	.067	.106	.122	.035	.064	.087	.118	.067
1.32	1.70	2.69	3.10	0.89	1.63	2.21	3.00	1.70
K	L	M	N	P	Q	R	S	wt
.083	.033	.042	.012	.020	.012	.006	.018	grams
2.11	0.84	1.07	0.30	0.51	0.30	0.15	0.46	0.020

SP-2U1+ Outline Drawing

This is a straightforward outline. All six pads are the same size so we will begin by creating the pads. The measurements for the pad sizes are in inches. So we will be using mils. We can see the pad is .02x.05 inches which is 20 mils by 50 mils. To begin, right click in CircuitMaker and click place->pad. Place one pad and then double click it. On the right side change the sizes to 20 and 50 mils respectively and change the shape to rectangular. On the left side by layer, change to top layer. Now make 5 more of those and you're almost done. The designators here have to match the designators of the schematic symbol, aka 1 to 6. Next up we need the vertical and horizontal spacing. We can see the horizontal spacing is .037 inches (37 mils). Align one of the pads and note down the x and y coordinates (by double clicking it). Next pick the next two pads and make the y coordinate the same but change the x coordinate 37 mils to the right (or left). For the other three keep the x coordinates the same as the ones above them but change the y coordinate 140-50 mils (since the data sheet gives from top of one to bottom of other. Now the footprint is done! It should look like this:

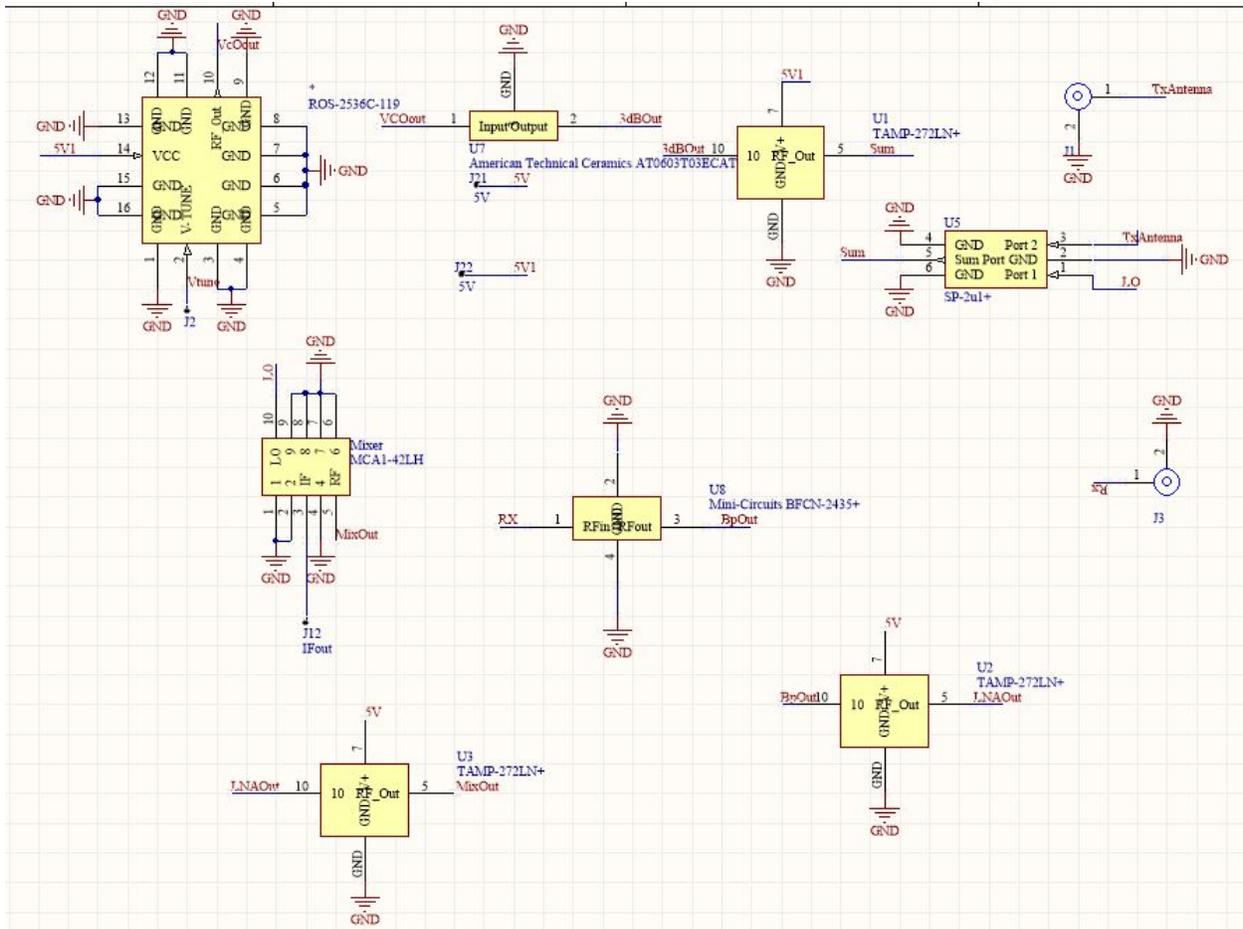


SP-2U1+ Footprint

Schematic Creation:

This is the design of your PCB. All connections and components will go here, it's the most critical part of the PCB process. To begin you need to know all the components that your PCB will need. To start in CircuitMaker go to the projects tab and click create new project and save it. Then open the design and right click the name on the left and click "add new to project"->"schematic". From here place all the components down. From here I usually begin by connecting all the grounds to ground as they have nowhere else to go. I highly recommend using a netlist rather than physically connecting the wires from component to component. To do this press "n" on your keyboard and create a netlist. Name it something that makes sense like "sum" for the input of the

splitter. Now we need to connect all the components together. For example I have an LNA going into the input of the splitter. So for the output port of the LNA I named it “sum” with the netlist and named the sum port of the splitter “sum”. They now have a connection. Do this for all your components and then double and triple check all connections. Also keep in mind if something needs an output to connect to another PCB. For this one the baseband connects to the teensy so we need a pin to connect those. Also don't forget a ground pin. When all connections are done it should look something like this:



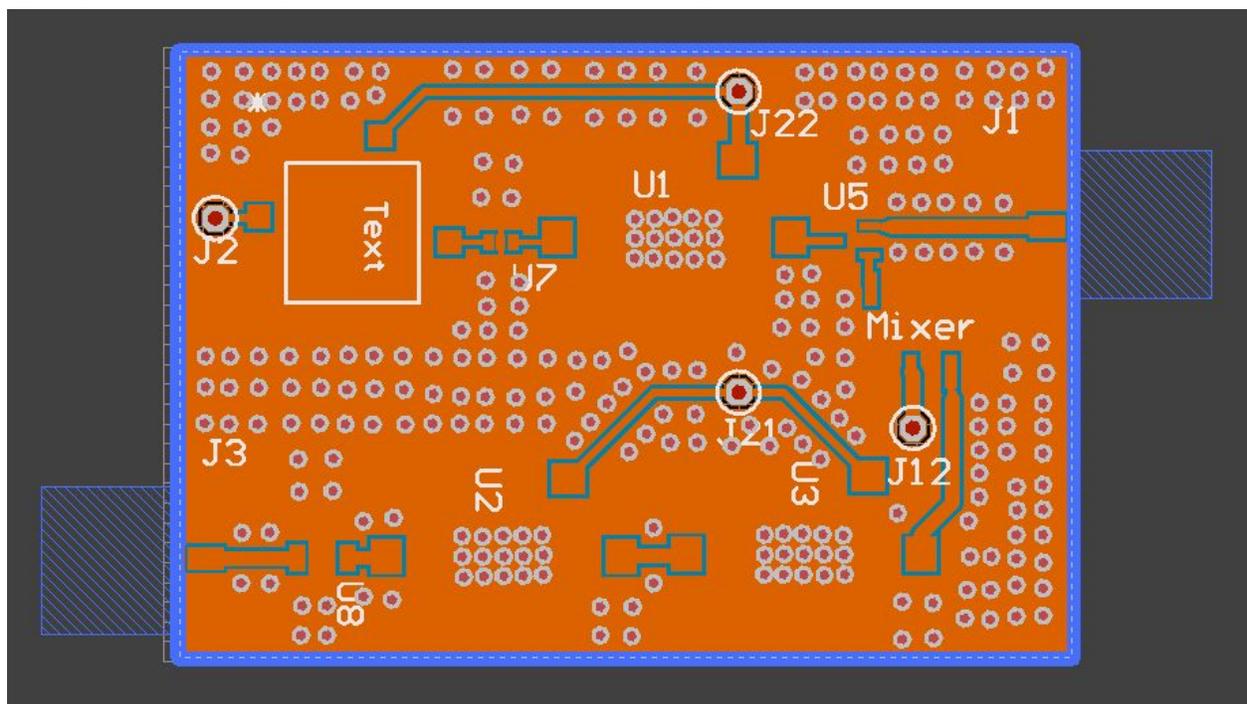
RF Board Schematic

Now save and compile the project. Now we can import this schematic into a PCB design file. Right click the project name on the left and click “add new to project”->”PCB”.

PCB:

To begin click project and import changes from your schematic. This will place all your footprints on the layout. Place the components in roughly the order they should be in. We can see floating lines that are the netlist, this is how you know where to put your components, the lines represent electrical connections. Place them in order so that they

can be connected. For the RF board they either have to be very close together or have you need 50 ohm traces for impedance matching. Also for the RF board do not put traces on the bottom layer of the board that overlap components or top layer traces, as this can cause interference. Now we can make the board outline so that we can pour the copper layer. Click board shape from the home tab and redefine board shape and then outline your components. Then click the board shape again and create primitive from board shape. After this is done we can pour the top and bottom copper layers. To do this click polygon pour. In the settings click connect to net: GND. Change the layer to top or bottom layer. Then pour the copper for both layers. Now your PCB is practically done. Only part left is adding vias since this is the RF board. I placed many vias, but it is up to you. The final product should look something like this:



RF Board PCB

Conclusion:

Now we need to run it through the DFM report to see if our PCB will work. To do this click project and generate outputs and select the gerber and drill files. You must configure the gerber files. Go to the layers tab of the gerber files and select these layers:

Layers To Plot			
E...	Layer Name	Plot	Mirror
—	C Top Overlay	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Top Paste	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Top Solder	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Top Layer	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Bottom Layer	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Bottom Solder	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Bottom Paste	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Bottom Overlay	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Outline	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	A Assembly Text Top	<input type="checkbox"/>	<input type="checkbox"/>
—	A Assembly Text Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	A Assembly Top	<input type="checkbox"/>	<input type="checkbox"/>
—	A Assembly Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	C Dimensions Top	<input type="checkbox"/>	<input type="checkbox"/>
—	C Dimensions Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	3 3D Top	<input type="checkbox"/>	<input type="checkbox"/>
—	3 3D Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	C Courtyard Top	<input type="checkbox"/>	<input type="checkbox"/>
—	C Courtyard Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	T Test Points Top	<input type="checkbox"/>	<input type="checkbox"/>
—	T Test Points Bottom	<input type="checkbox"/>	<input type="checkbox"/>
—	C Keep-Out Layer	<input checked="" type="checkbox"/>	<input type="checkbox"/>
—	C Top Pad Master	<input type="checkbox"/>	<input type="checkbox"/>
—	C Bottom Pad Master	<input type="checkbox"/>	<input type="checkbox"/>
+	Component Layers	<input type="checkbox"/>	<input type="checkbox"/>
+	Signal Layers	<input type="checkbox"/>	<input type="checkbox"/>
+	Electrical Layers	<input type="checkbox"/>	<input type="checkbox"/>

Run these files together with the drill files through the DFM report and if everything comes back green you are done! This may have looked straightforward and simple, but making everything connected and compact is very time consuming and you will make many errors. It will take a long time so do not procrastinate on this, I know from experience. If mistakes are made your entire design will not work so make sure you have your PCB checked by multiple people.