

EEC 134
Application Note
Introduction to PCB Design
Cameron Vossoughi

Introduction

Being fluent in PCB design is essential for electrical engineers regardless of their discipline focus. This application note will focus on creating footprints, schematics, and PCB layouts. However, PCB design may complicate quickly if one takes into account for trace bends, multi-layer design, signal integrity, and other constructs. These design issues are better addressed in area specific textbooks and papers.

In today's market, the industry standard CAD tools for PCB design are Altium, Allegro, and Eagle. In this application note, we will be using CircuitMaker, a free PCB design tool created by Altium that allows for multiple designers to work concurrently.

Creating a Project

Once you have made a CircuitMaker account and downloaded the software, you are ready to create your own project. Create a new project by clicking the "My Projects" tab and click new. This will take you to a window that will have basic tags for the project. On completion, the project will show up on the left side of the window with the project name. Right click the project, and add a new schematic and PCB document. Save the project and compile.

Footprint Design

CircuitMaker contains an online library called Octopart which stores every user's designed components. This means that the component you need to put on to your schematic is most likely already designed by another user. However, the designer should always be cautious of using someone else's design since there is no guarantee that component was designed correctly.

The first step in designing your own component is to look at the component's datasheet. For this example, we will be designing the Mini-Circuits ROS-2536C-119+ VCO.

Pin Connections

RF OUT	10
VCC	14
V-TUNE	2
GROUND	1,3,4,5,6,7,8,9,11,12,13,15,16

Outline Drawing

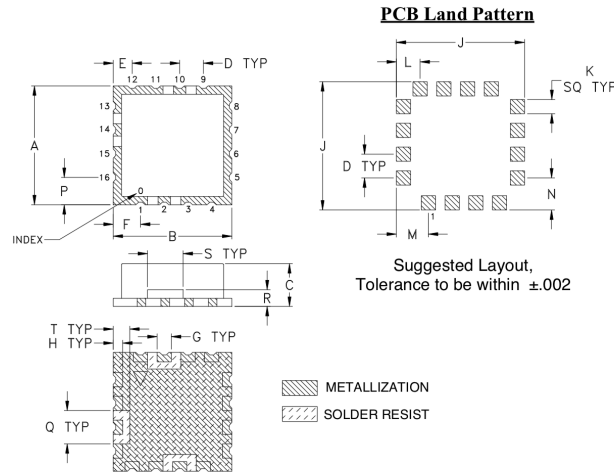


Figure 1 VCO Pinout and Land Pattern

Outline Dimensions ($\frac{\text{inch}}{\text{mm}}$)

A	B	C	D	E	F	G	H	J	K	L	M	N	P	Q	R	S	T	wt.
.500	.500	.220	.100	.080	.115	.060	.040	.540	.060	.100	.135	.135	.115	.140	.070	.150	.070	grams
12.70	12.70	5.59	2.54	2.03	2.92	1.52	1.02	13.72	1.52	2.54	3.43	3.43	2.92	3.56	1.78	3.81	1.78	1.2

Figure 2 Dimensions

There are a few items to account for when looking at this schematic. The diagram indicates that the layout is for the PCB land pattern. This is the view that one would have if they were seeing the component from the top down. Some diagrams indicate that it is a 'bottom view' which means that the designer would be seeing the component from the bottom up. The dimension chart indicates that the units used are in inches.

To begin creating your component, click on the libraries tab on the right of the window. On the bottom side of that screen, there is a blue link "Can't find the component you're looking for? Make your own!". Click this link to move into the component work station. Edit the part name and number so that the component is easily identifiable.

Clicking on the edit schematic symbol will take you a blank schematic document. Click on basic shapes and draw in a rectangle that is large enough for all of your connections. Refer to the pin connections in the diagram from the datasheet and match the pins you create on your schematic. It is crucial that you match your designators (pin numbers) with the correct net label (connection). For example, pin 10 on the diagram indicates that the net label is RF OUT, therefore should match pin 10 on the schematic you are designing.

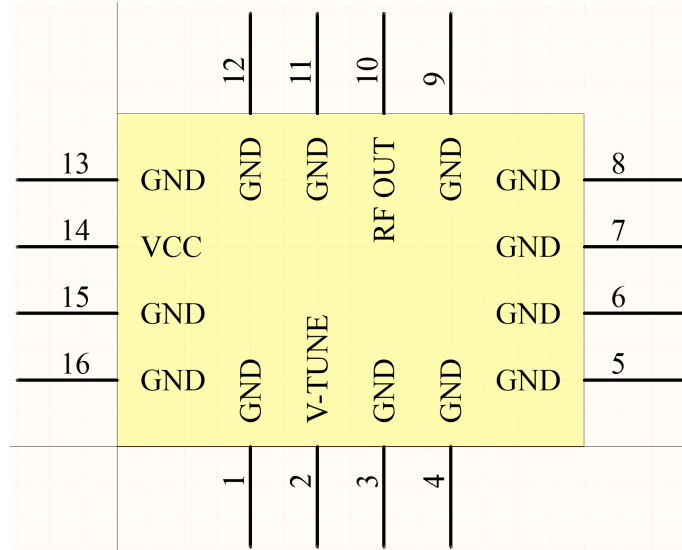


Figure 3 VCO Schematic Symbol

Once the schematic symbol is created, exit the tab. In the component work station, click on edit footprint window. This will show an empty black window. First, click on set>origins. This will set your location for the pad of the component. Click on the place pad option and click 'tab'. Clicking tab while using a tool will open the options of the tool. A popup should show up containing the specifications of the pad. After typing in the dimensions you require, click ok. Once the window has closed, click 'g' to open up grid properties. Enter the distance from center to center of each pad. This will allow for easy placement of each pad, without the need to measure the distance each time.

Once the pads have been placed, verify that the pin outs match the designators from the schematic and datasheet. Clicking 'm' will measure distances. Measure the distance between major dimensions to determine if the pads were placed correctly.

Click on the draw line tab and draw in the border of the component, this does not need to be exact. This will give an idea for the person soldering the PCB which pads correspond to which component.

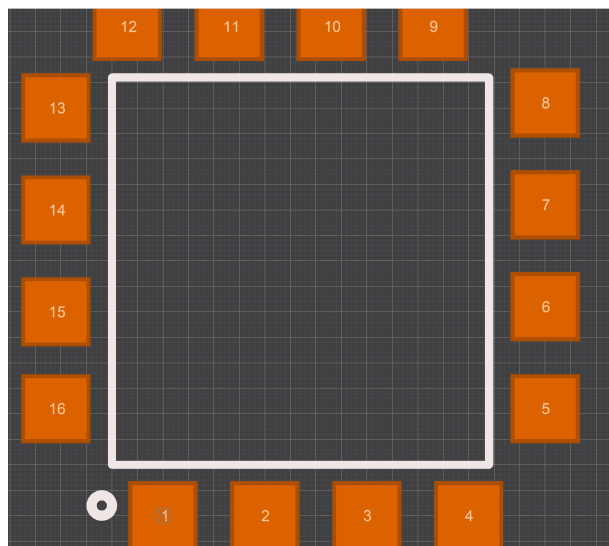


Figure 4 Footprint

After finishing the footprint, you may opt to have a 3D body for the component. This may be done by saving the STP file (usually on the product website) and importing this file to your component. Once the component is imported, match the pads to the STP file. For brevity, this application note will not show the process since it is not essential to this design.

Schematic Design

Once you have created all of your components, open your schematic document. All of your designed components will be in the libraries tab, under the “favorites” folder. Populate your schematic document with the components you require. Wires are created by clicking the wire button or clicking ‘w’ on the keyboard. Since schematic designs are much more complicated than having a few components, it is best to not have all the wires connecting to all the other components. “Net labels” allow for components to be connected without having to physically draw the wire. This shortcut is done with ‘n’, and editing the label is done by double clicking and changing the name. Note that the net labels must be one work, or use an underscore for spacing.

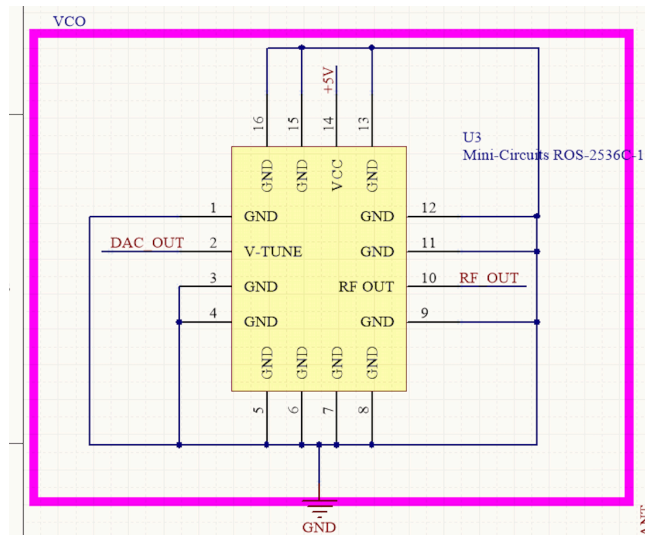


Figure 5 VCO Schematic

With smaller designs, such as this one, it is best to have all the components on one schematic document. The components may be blocked off with the pink border by right click>edit>rectangle which allows the user to draw a pink rectangle. Text may be inserted to demonstrate the purpose of each block.

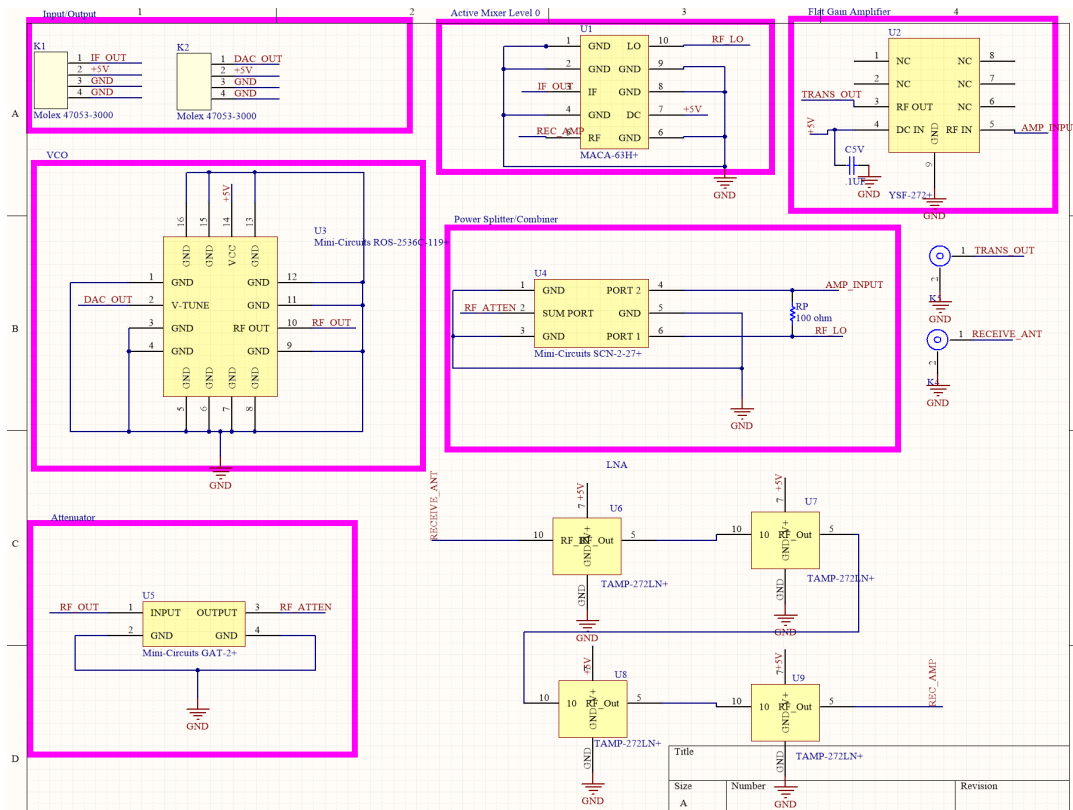


Figure 6 Completed Schematic

It is up to the designer what the best way to split the schematics are. Only experience will determine what the designer is most comfortable with.

After verifying that the schematics are created correctly, they are ready to place their components to the PCB document.

PCB Layout

Be sure to read the standards and requirements set by the PCB manufacturer you use. The PCB layout is started by entering the PCB document and clicking on file>import schematics. This will populate the PCB schematic with all of your components and fly wires (white lines). Do not be daunted by this, as you place your components in the desired positions, the mess will clean itself up.

Once the components are inside the PCB document, draw your PCB outline. In this case, the outline was created in SolidWorks and imported into CircuitMaker. This is an advanced feature and will not be addressed in this document. Once the outline is created, highlight the entire outline. This is done by pressing shift and click for all of the lines. In CircuitMaker, highlighting multiple objects requires the use of a shift and click. Next, click on tools>PCB layout> define from selected objects. This will make your PCB workspace be contained in the outline.

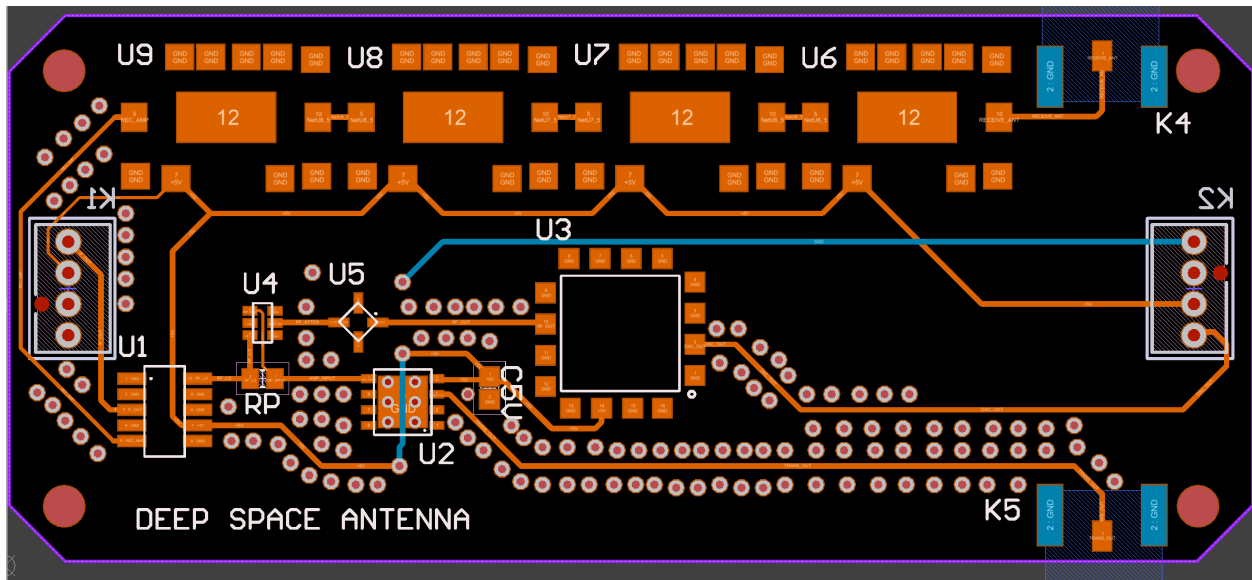


Figure 7 PCB Layout

After placing all of the components in the locations that you desire, it is time to create the traces. Click on the trace button or 'n' button to open the tool. While using the tool, click on the tab button to open the properties to change the trace width. Trace width is determined by many factors and is better left to a focused paper.

After all the traces are completed, make sure that the components have no fly wires in between them. The "cross probe" tool allows for the designer to click on one item in the PCB document and see what it corresponds to on the schematic. The opposite is true as well for using the cross probe in the schematic documents. Verify that all of the components are properly connected.

The polygon pour is made so that the PCB may have a comment net. In most cases, the net label for the polygon pour is set to ground. Click on the polygon pour tab, and click in the area you desire. With Bay Area Circuits, they require that the polygon pour is at least 10mil away from the outline.

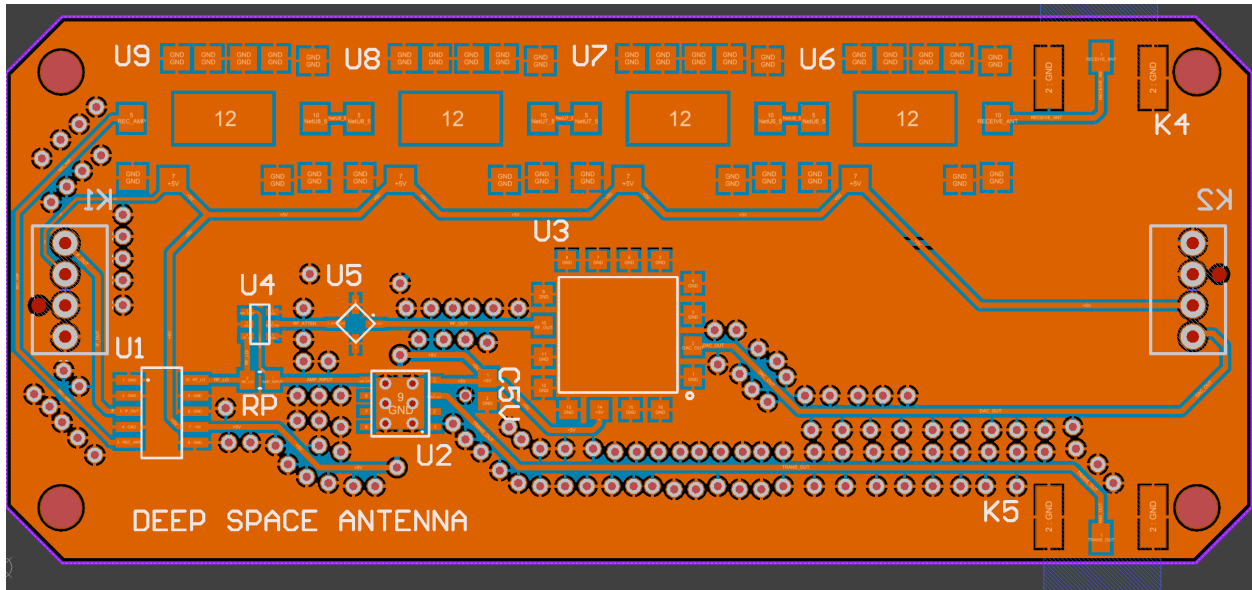


Figure 8 PCB Layout with Polygon

PCB DFM and Manufacturing

Once the PCB has been finished, export the Gerber files as well as the NC drill hole file. These are the files you require to submit to the manufacturer to produce. Typically, the manufacturer has a DFM to check your design for printing. Read their instructions and submit your files accordingly. If the DFM lists an error, edit your PCB and re-submit.

Conclusion

PCB design may seem complicated, however only gets simpler with practice. Different applications of the PCB may require much more complicated designs, however, you now have the fundamentals to improve your design.