Michelle Lee
998332385
EEC134
Application Note

Circuit Design

In designing the radar, first do some calculations to see how much power you need to transmit to detect an object at whatever range you are aiming for and to have an idea how much power you should be getting at each stage of your radar circuit. Once you have that, you can pick out whatever components you need according to what you have learned about RF circuits to accomplish that power transmission requirement. Then use a program like ADISimRF to check if the components you chose will do their job on the transmitting and receiving ends. You may have to change, add, or remove components to meet specifications. If it checks out, then you can start putting together a schematic for PCB design.

PCB Design

In designing the PCB with the components you chose, it is important to first read up on the data sheets on the parts you are using. The data sheet should list the components' maximum ratings and typical performance specifications. From this information you will know how much voltage should be supplied to the circuit and where you may need to put attenuators or voltage dividers for components that cannot handle the voltage supplied to the higher powered components. For some components there is a page on the data sheet that shows the basic connections for the component and recommends which parts to use to correctly bias, power, or protect the component. So unless you are doing something pretty different with the component, it is usually best to go with the recommended parts and connections.

Once you know the maximum ratings and what other components you need for the amplifiers and more complicated components to properly function, you could then start drawing a schematic of the circuit using mostly what you find recommended on the data sheet and whatever else you find necessary for the whole circuit. For laying down the footprints of the components, I usually arrange the more complicated connections first because it is harder to rearrange a complicated cluster of 10+ components than it is to rearrange the connections of a part that is only connected to a couple things. You also want to arrange the whole circuit so that it is as compact as possible, but also in a way where you can draw the traces to all applicable places and not accidentally have isolated, unconnected ends. Once you are done with the initial arrangement, you draw the traces. Look at the PCB fabricator's website to see the minimum size requirements for traces, via holes, pads, spacing and rings and try to have traces and via holes that are a bit bigger that the minimum requirements to minimize getting fabrication error. The trace width should be whatever that is appropriate for the circuit, for example it should be the thickness of a 50 ohm line when connecting to the transmitting and the receiving antenna, and it may have to gradually get thinner in order to connect to the tiny components. You may have to rearrange some of the components in order to make all the necessary connections with traces. Some general guidelines to placing traces is to avoid making sharp corners, try to keep traces as short as possible, and try not to do too much weaving around components. Once everything is connected, place ground fill at least on the back side of the PCB and do some via stitching to connect the ground layers if you have multiple ground layers to stop signals from going into certain areas and to have better heat dissipation. It is also important to put some test points to see

if you are getting the right signal and/or the right amount of power at certain locations of your circuit, but note that the oscilloscope cannot read high frequencies, so do not put test points in the RF side of the circuit.

Sometimes if the library of the PCB CAD program does not have the footprint of the part you are using, you would have to draw your own footprint. To do this open up the footprint editor of the CAD program and start a new file. Then look at the datasheet and look for the dimensions. Sometimes the dimensions are not on the datasheet, but the datasheet should list what the package the component comes in, so search the package instead. Then on the footprint editor, draw some pads and enter the dimensions of the pads and spacing. Try to make the pads a little bigger than the dimensions. If the component is through-hole, then make the hole wider than the contacts.