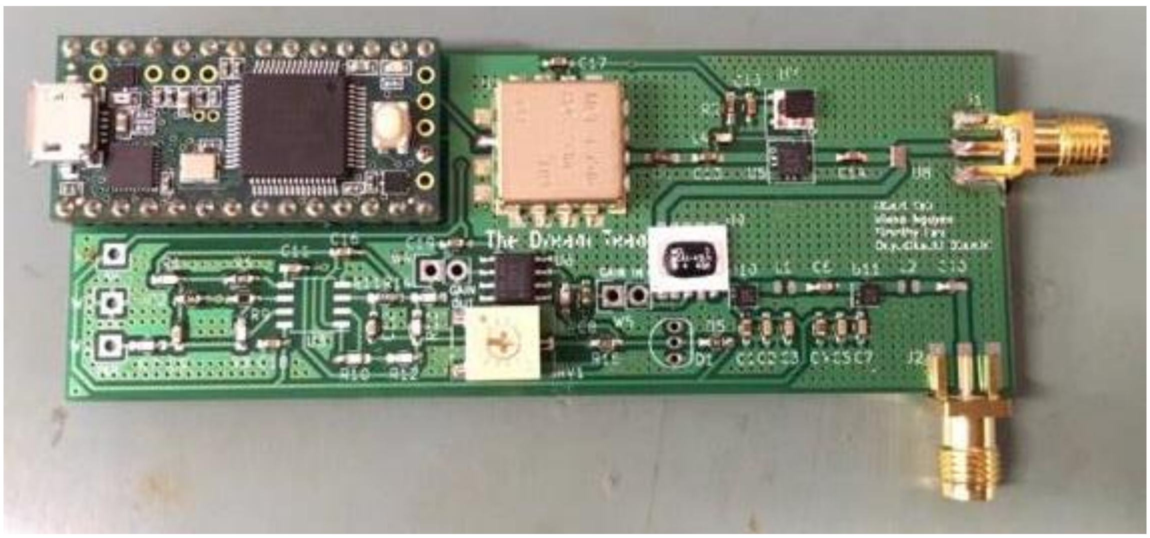
# EEC 134AB

*Application Note*

Printed Circuit Board Design for High Frequency Systems

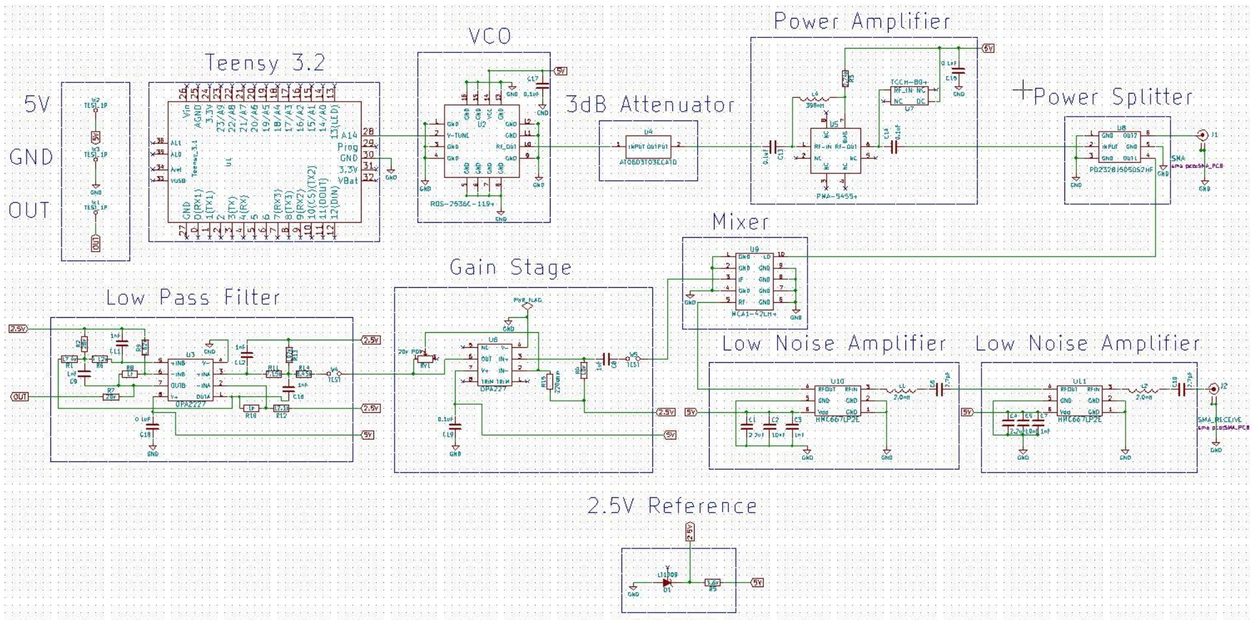
By: Timothy Lau



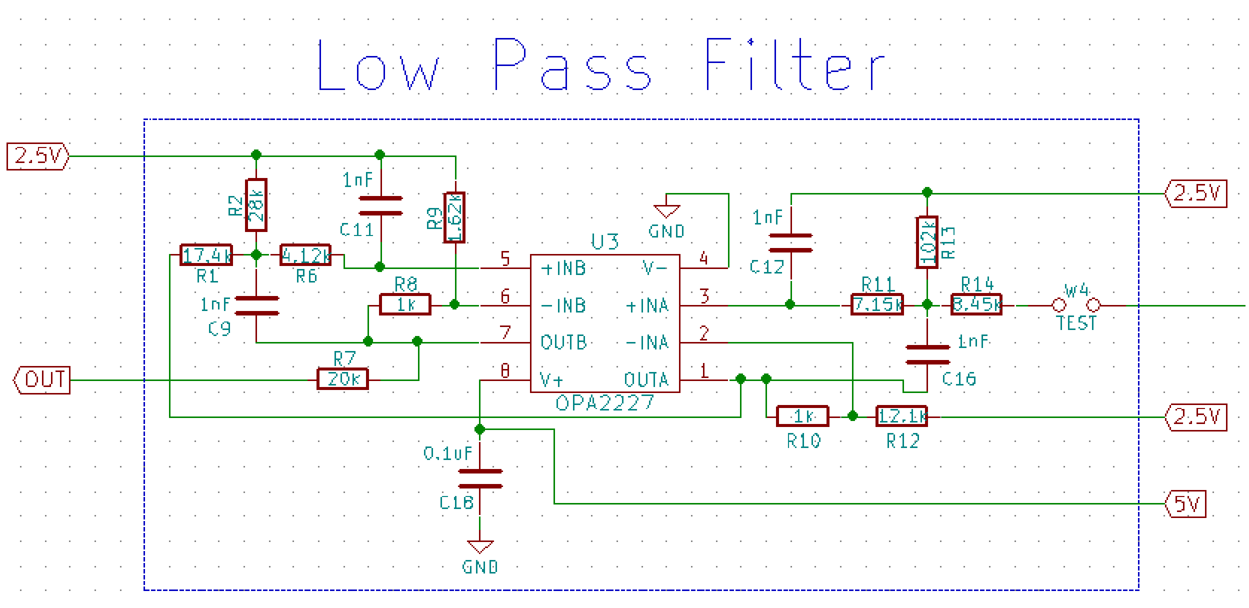
## Introduction:

Printed circuit boards (PCB) are typically used in almost all modern electronic products and devices. They allow for circuits to be easily manufactured and assembled using automation. Furthermore, developing circuits with PCBs are cheaper and faster to manufacture than any current methods of circuit design. PCBs can be singled sided, double sided, or even multi-layered to efficiently use space to save on costs, lower power consumption, and provide better performance.

## Schematic Design:

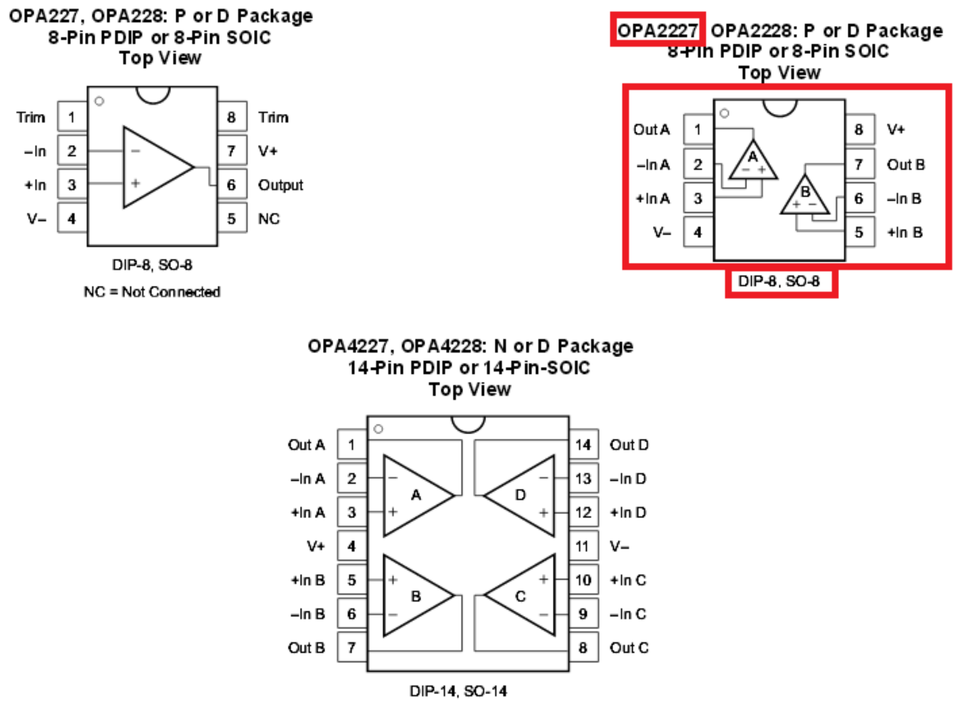



The first step to designing PCBs is to develop a circuit schematic. One of the most important aspects of schematic design is keeping the circuit clean and easy to read as this will help a great deal when it comes to PCB layout and debugging. Also, sectioning off circuit designs into modules will allow for transparency and even instantiation of modules as seen with the Low Noise Amplifiers in the image above. The schematic above was designed in KiCad 4.0.4.

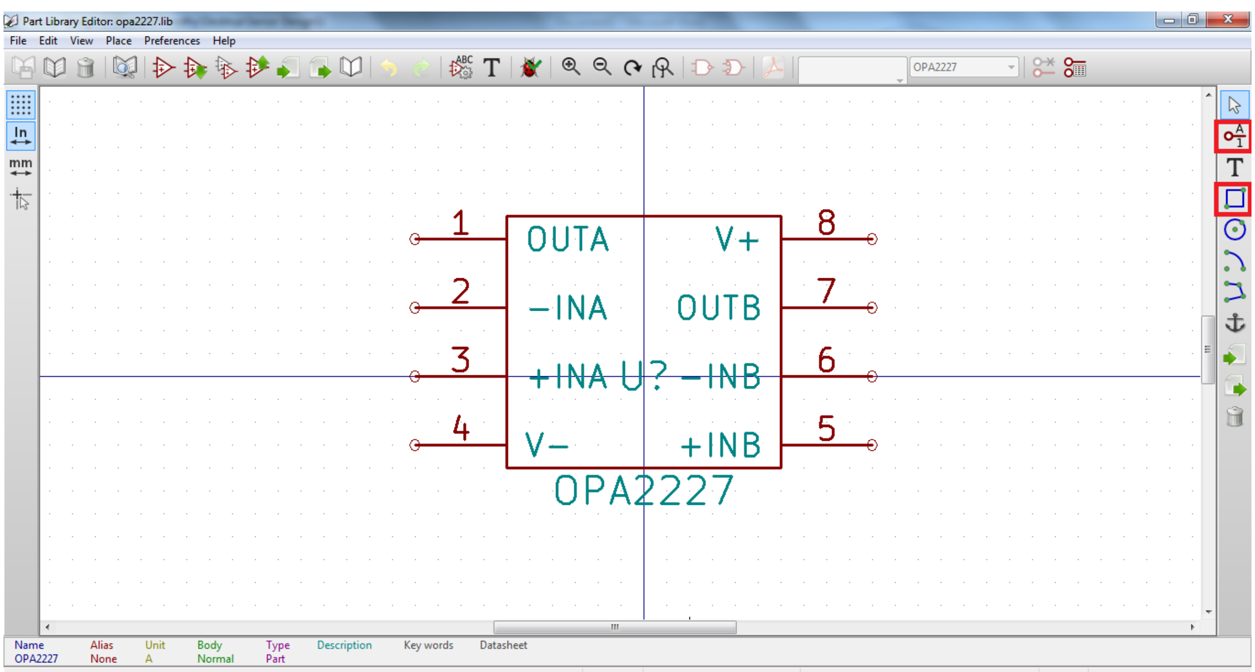



To start schematic designing, lets take a closer look at the Low Pass Filter module. Here, the main IC in this module is the OPA2227 High Precision, Low Noise Operational Amplifier.

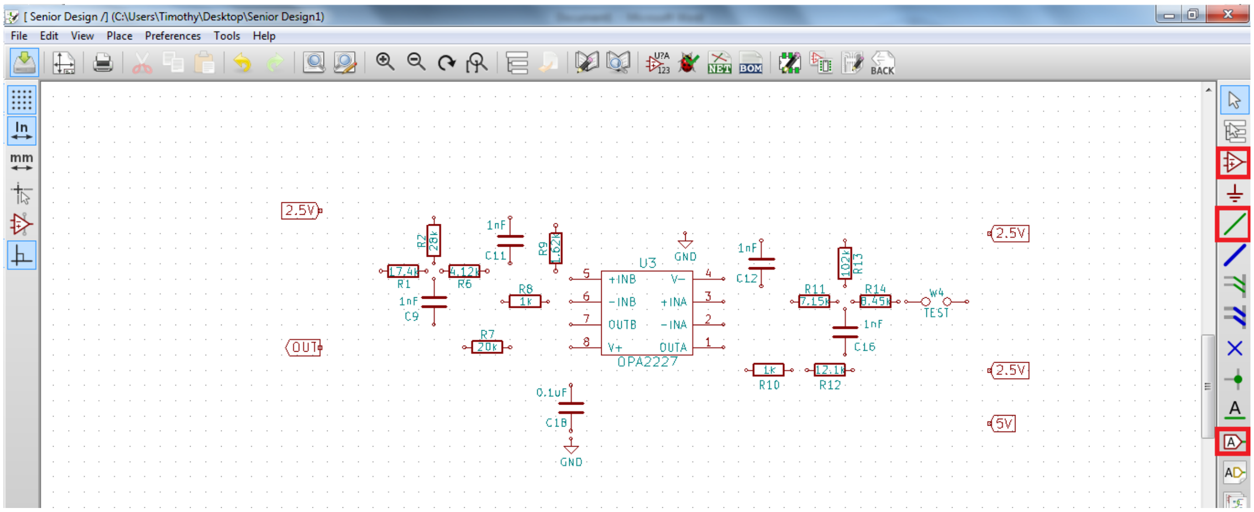
## 5 Pin Configuration and Functions



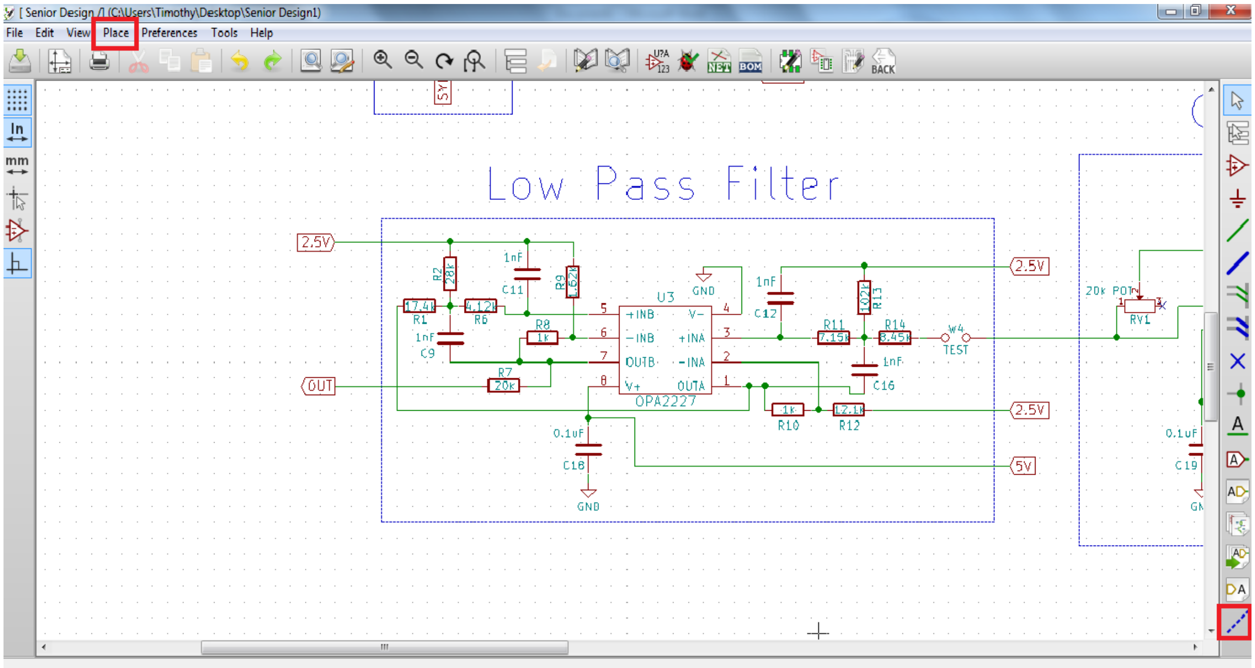

To get the pin configurations, we first consult the OPA2227 online datasheet. As seen above, we find the number of pins, pin configuration, and PCB footprint package. This allows us to create an OPA2227 block on our schematic.



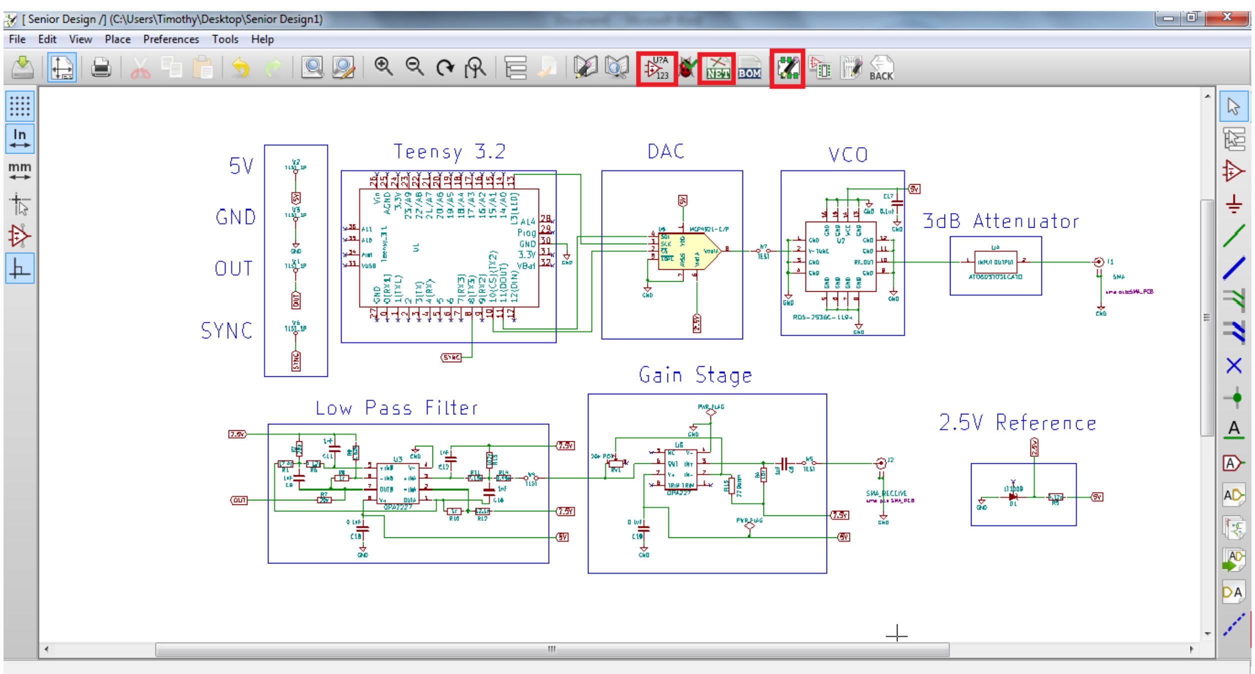
From KiCad, we enter the library editor and begin drawing our IC. From the datasheet, we follow the general pinout diagram and recreate it in our block. In the top right square is the ‘Add pins to the component’ option which allows for the user to enter input or output pins to their block diagram. After the correct number of pins is entered, they should be labeled accordingly. Once the pins have been placed, another option is the ‘Add graphic rectangle to component body’. With this, the block is done and ready to be included in the library for use in our schematic.



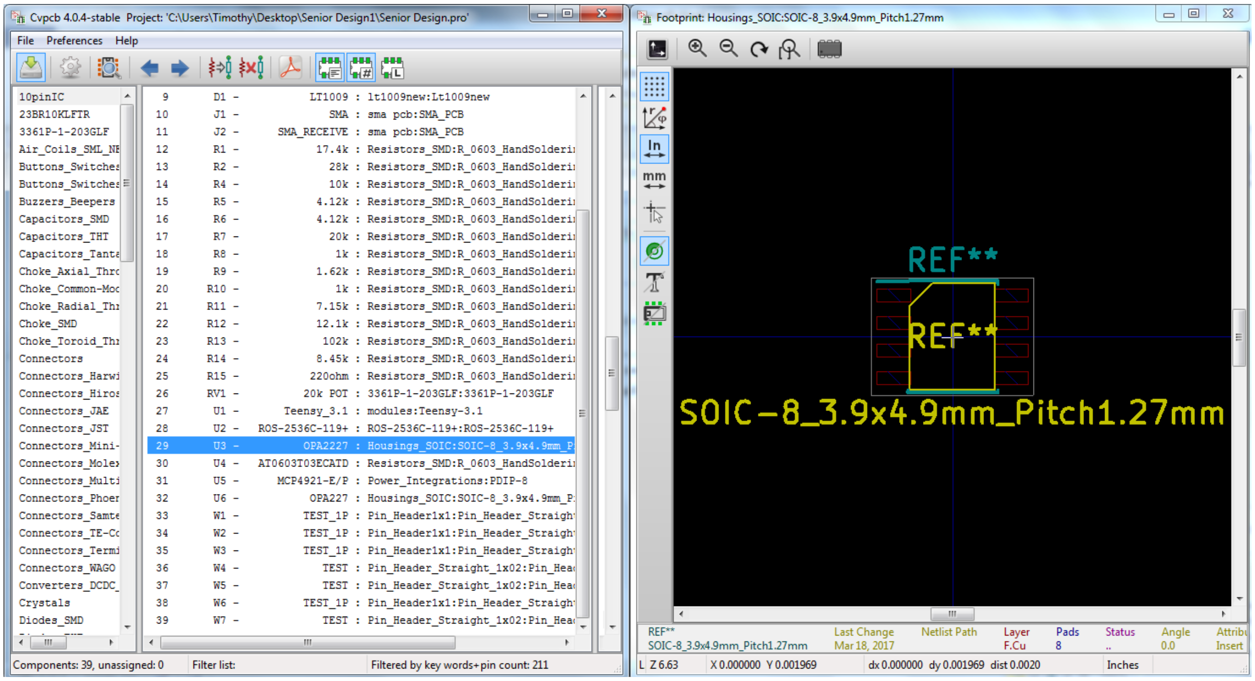

Going back to our schematic diagram, other circuit components of the Low Pass Filter module can be placed using the option in the right bar called ‘Place Component’. Additionally at the bottom of the right bar is an option for ‘Place global label’ which allows for wireless connections. For my schematic, I used these wireless connections to connect a global 5V power rail and 2.5V reference voltage. Once the component placing is complete, we can begin to connect them all using the ‘Place wire’ option located also in the right bar.



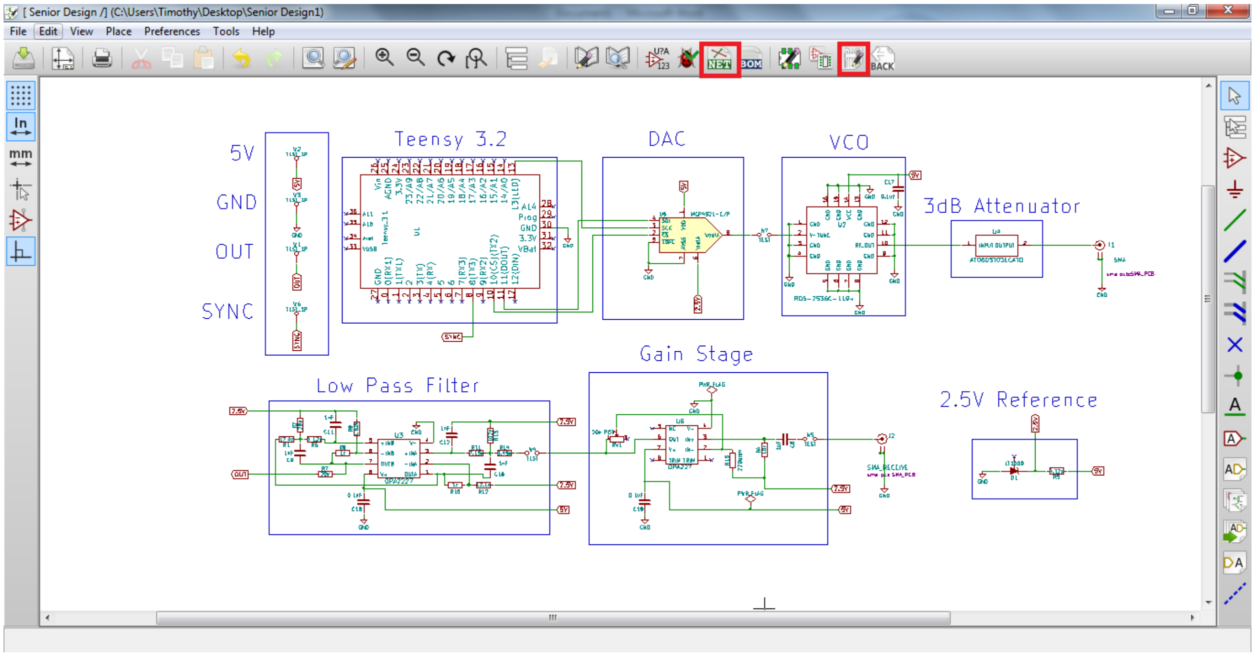

With the module wired up, we can then place a graphic enclosure using the ‘Place graphic lines or polygon’ option on the bottom right. Once done, a graphic text can be placed in the top right under the ‘Place’ tab. With this the module is complete and the rest of the modules can be created to complete the system.



Once the entire circuit schematic is complete, we will press ‘Annotate schematic components’ in the top to label our components. Next, we will ‘Run CvPCB’ to associate each component in our netlist with a footprint for PCB layout.

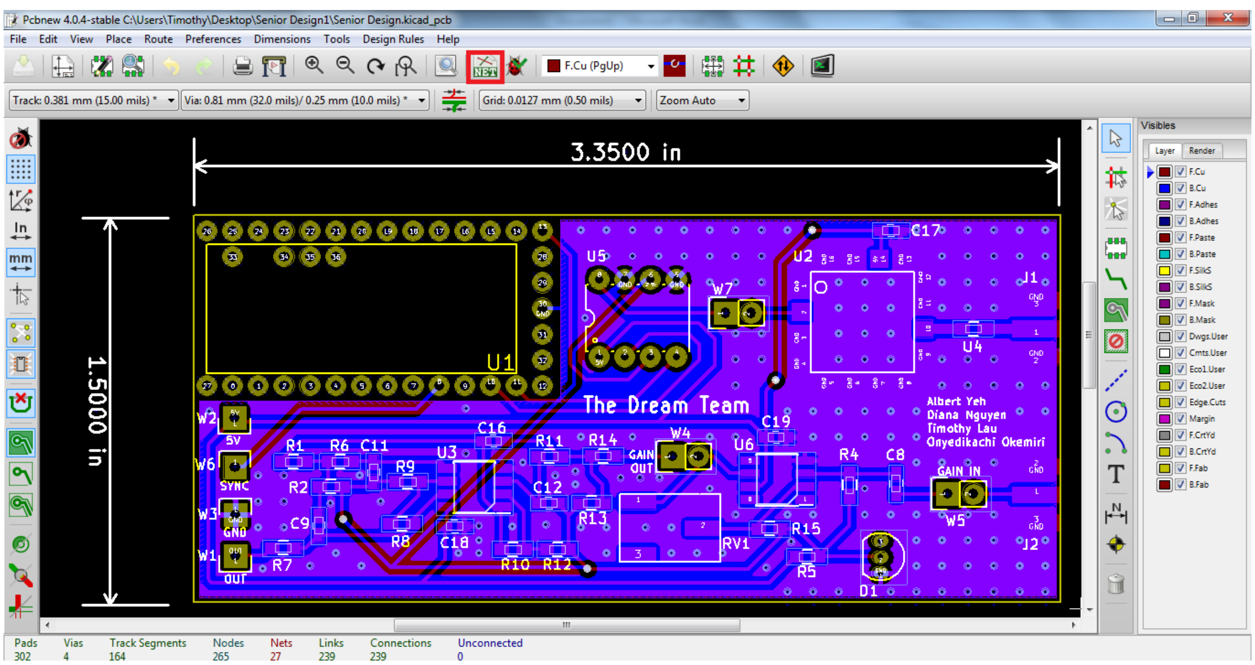



In CvPCB, we simply find the footprint in the footprint library and associate them with our circuit components. Referring back to our OPA2227 datasheet in the Low Noise Amplifier module, we find that the PCB footprint can either be DIP-8 or SOIC-8. In this instance, I have chosen the SOIC-8 package to associate my component with.

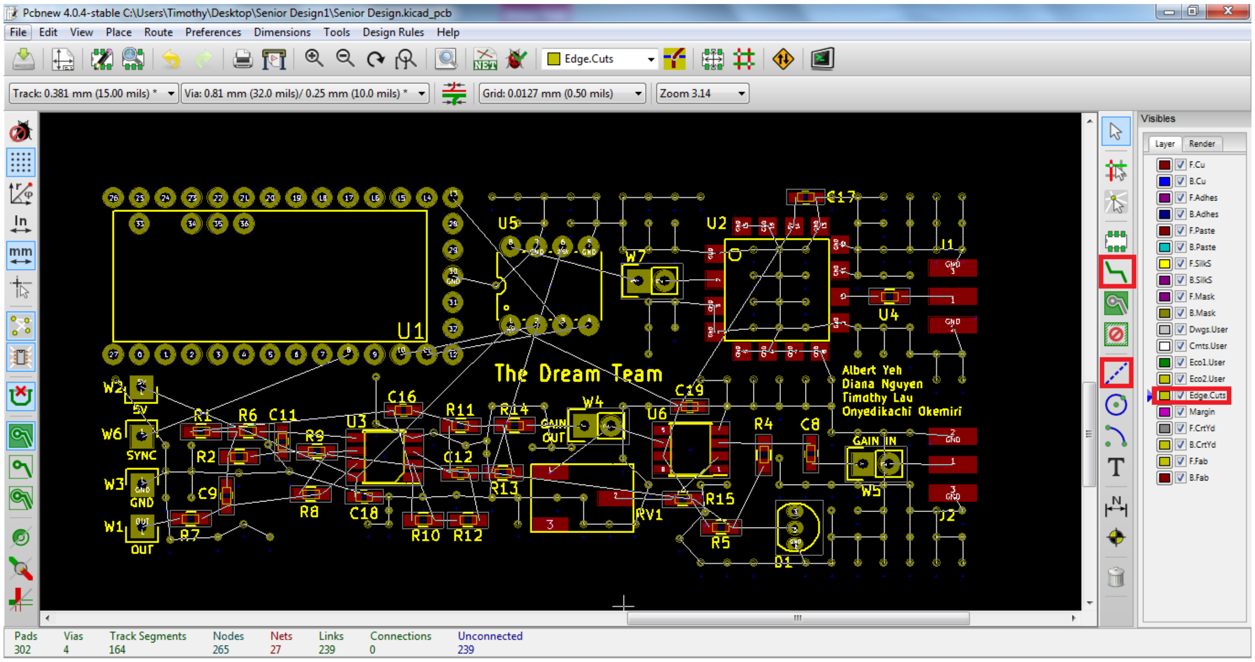



Once done with CvPCB, we return to our circuit schematic and choose ‘Generate netlist’ at the top and we can begin the PCB layout by clicking the ‘Run PCBnew’ option.

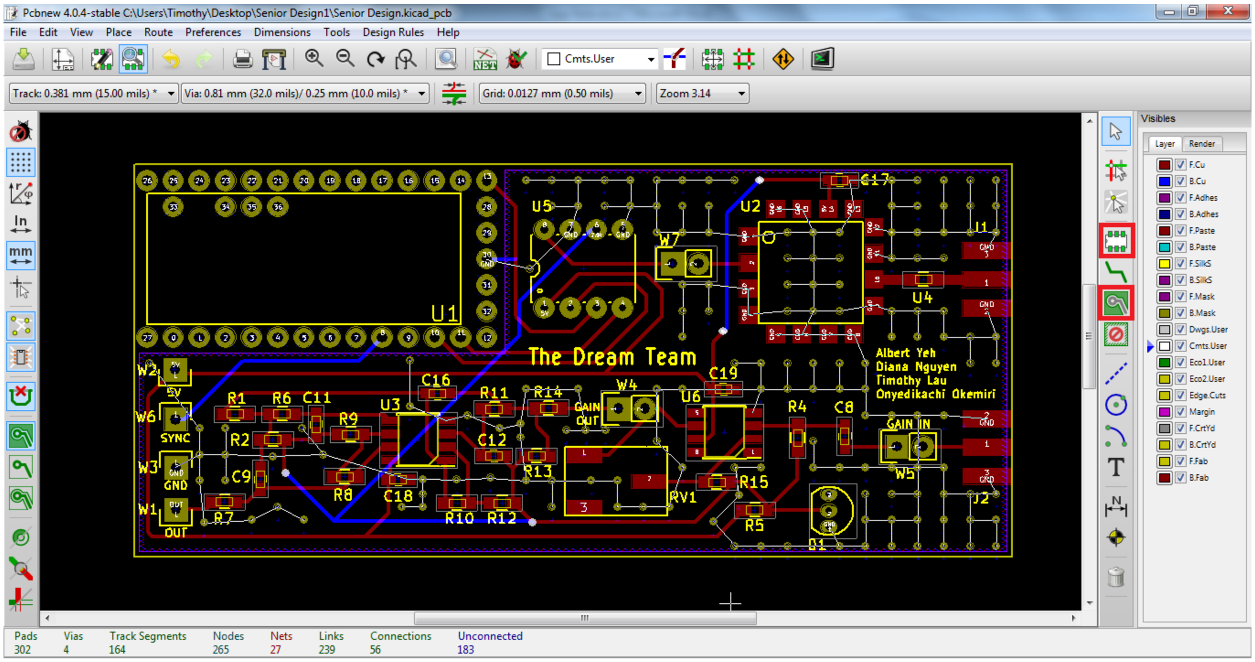
## PCB Layout:



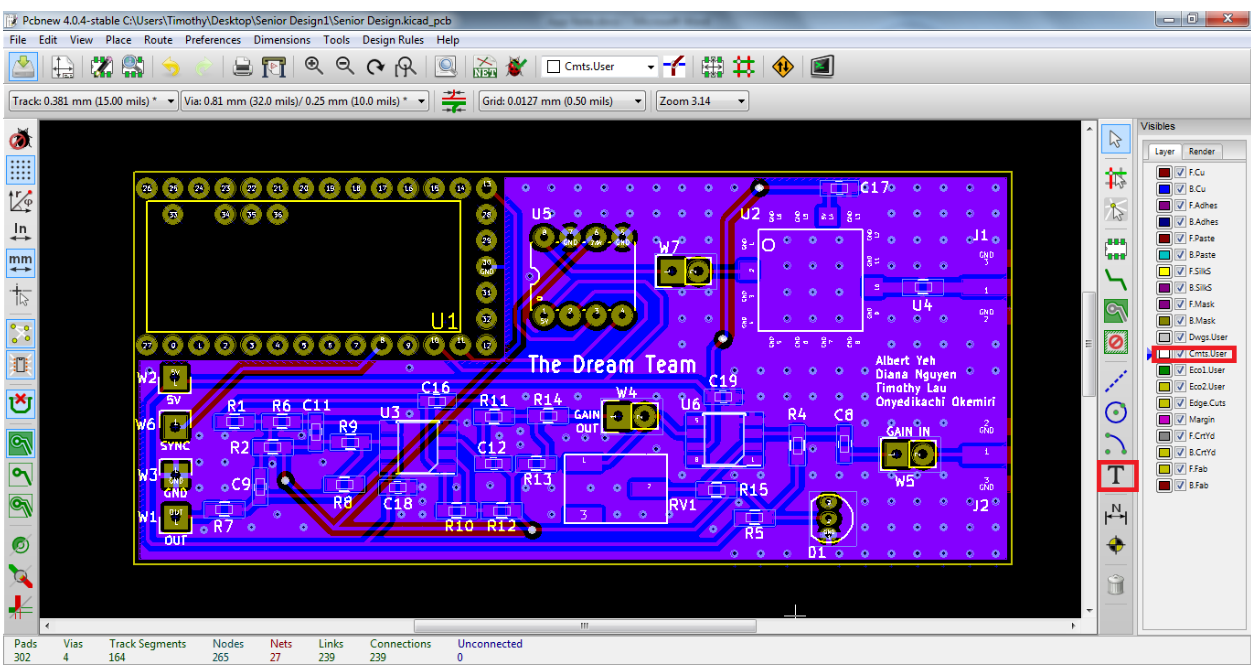
In PCBnew, we first begin by choosing the option ‘Read netlist’ at the top. This will drop all of the circuit components into one spot. After the circuit components are in one spot, they can be repositioned and rotated if necessary. For the PCB layout, it is best to reposition all the components into the general area of where they are in the circuit schematic. This will ensure that components go where they are supposed to and allow for easier debugging in the future.




After the components are positioned, click on the Edge Cut layer and choose the ‘Add graphic line or polygon’ feature to add a physical edge to the PCB. With the physical edge of the board defined, we can then select ‘Add tracks and vias’ to connect our components. Be sure to have no 90 degree turns for traces but rather have then at 45 degrees instead to minimize loss.

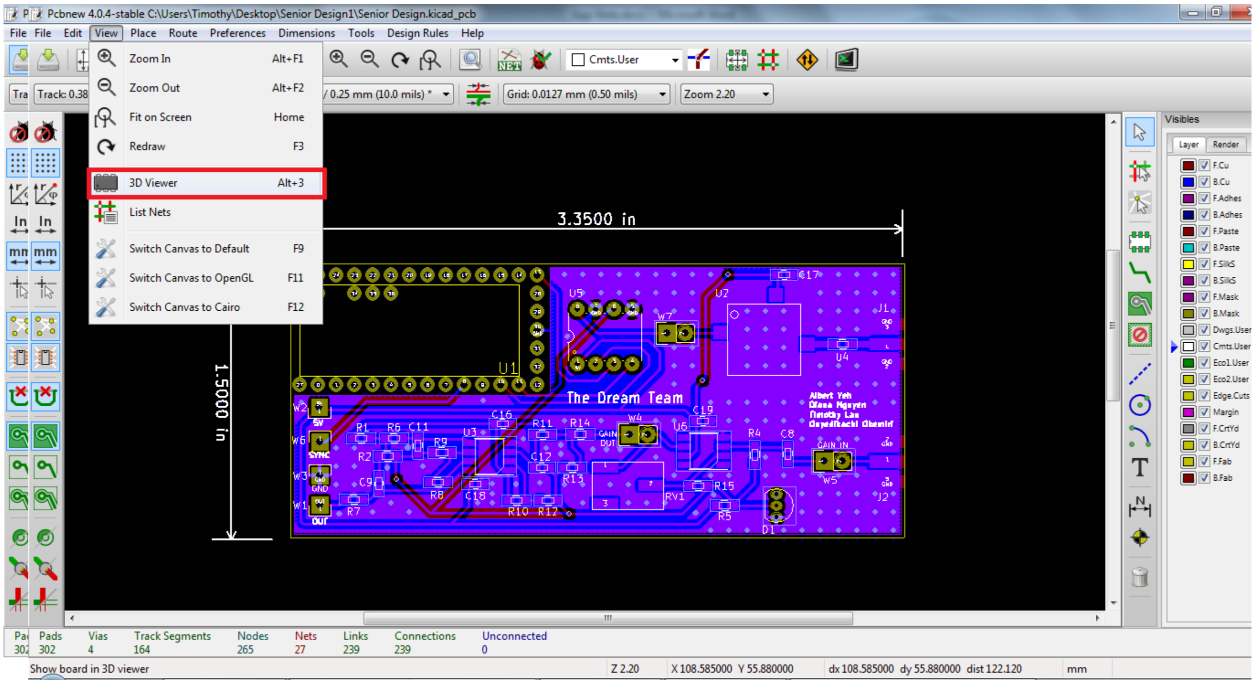



After the traces and vias have been placed, we can then begin to place our Via Holes. Click the 'Add Footprints' feature on the right and add drill holes connecting both the front and bottom of the copper ground planes. This is to connect the two ground planes and prevent any unwanted resonations in the circuit. Once the Via Holes are placed, the feature 'Add filled zones' can be used to flood the circuit and ground the areas between the circuit components.

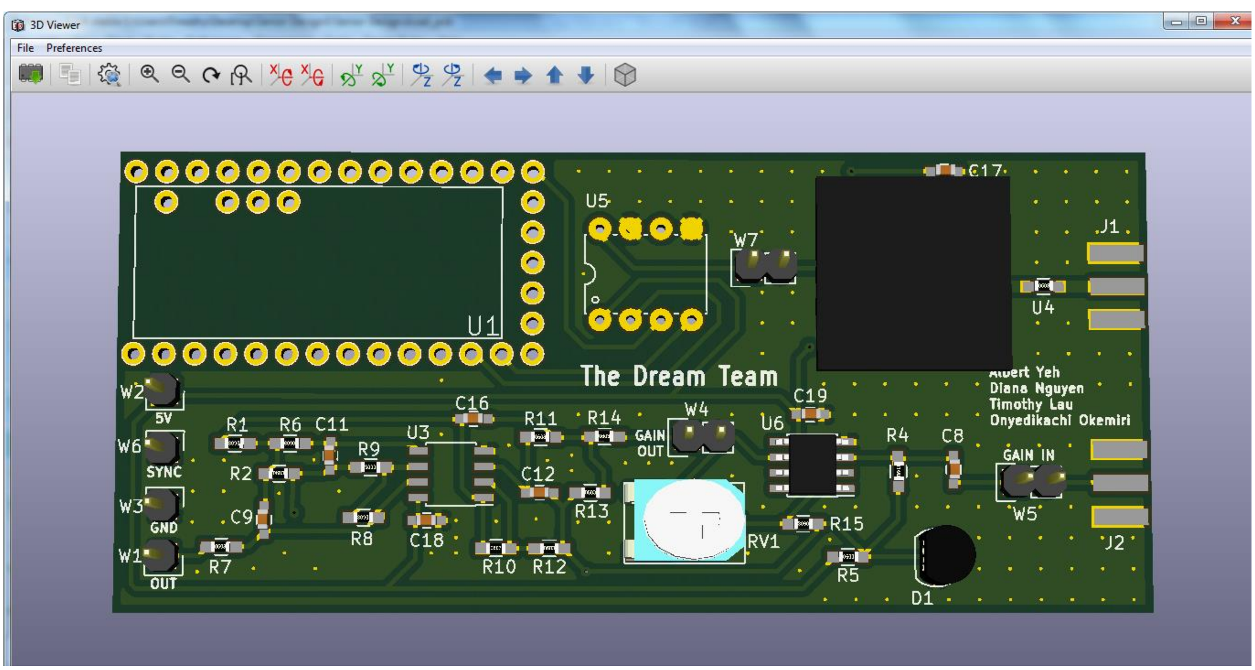


With this, the overall layout of the PCB is mostly complete and graphic text can be added as necessary in the 'User Comments' layer on the right using the 'Add graphic text' feature. The PCB layout is complete and we can now begin our 3D model rendering.

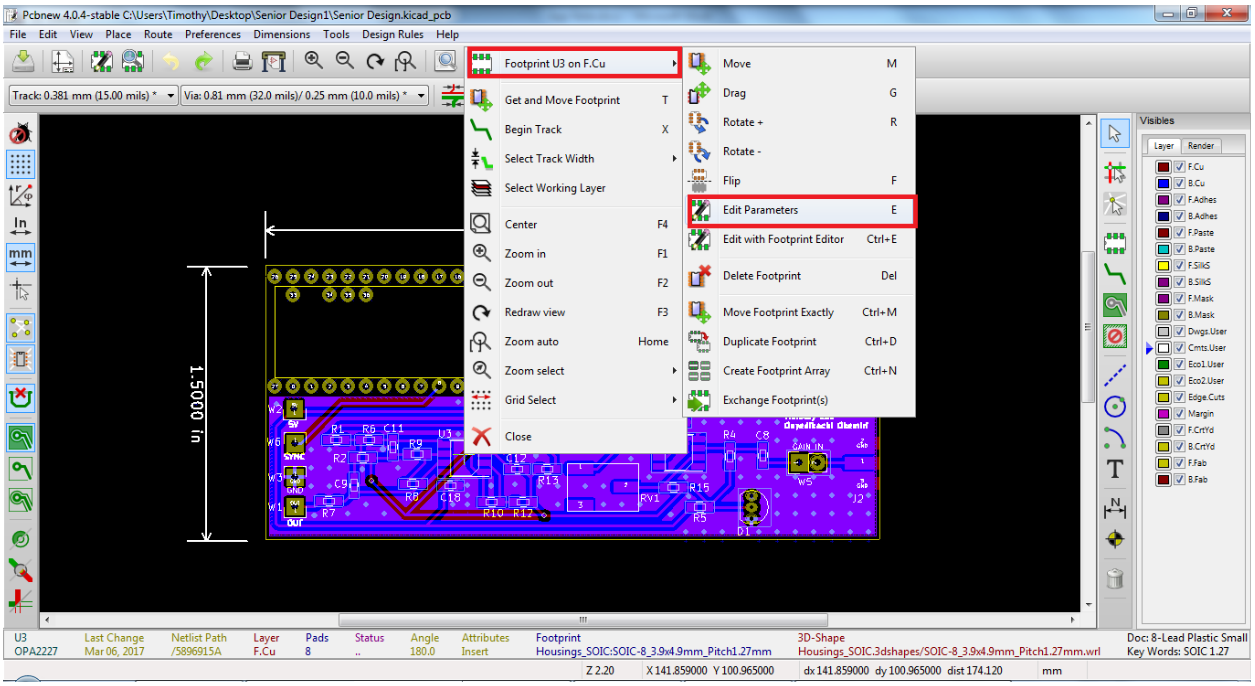
## 3D Model:



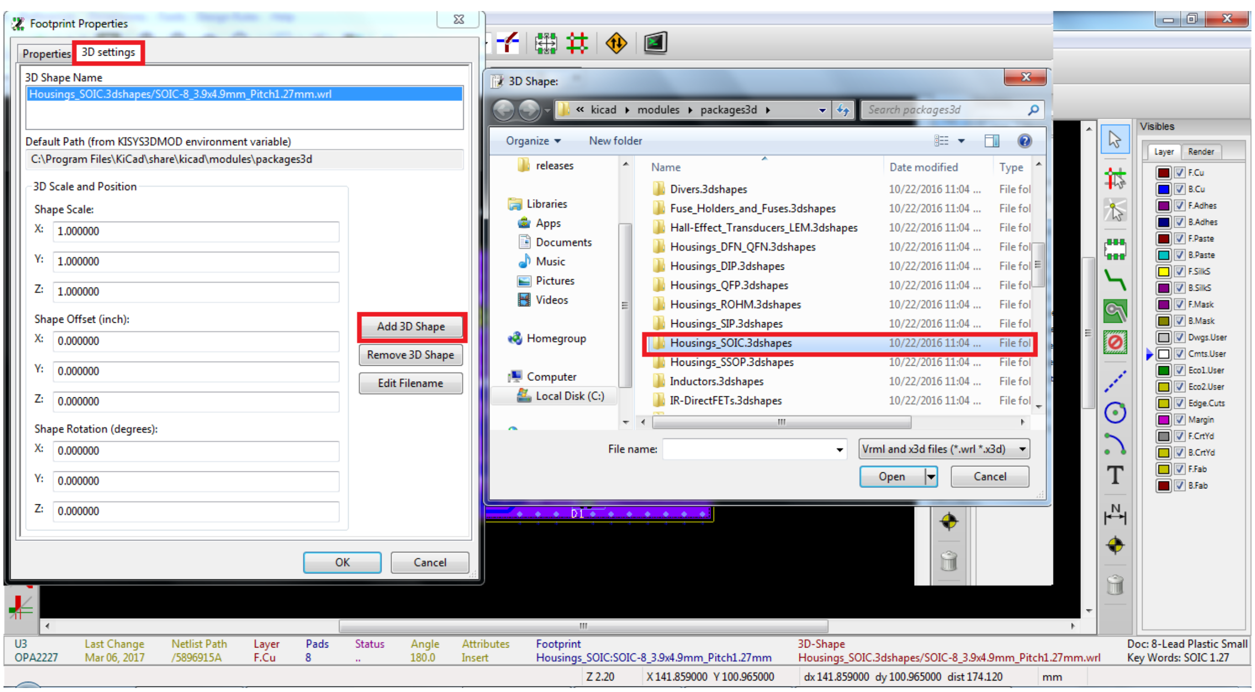

To do a 3D model rendering, simply go to the 'View' tab at the top and select the '3D Viewer' option.



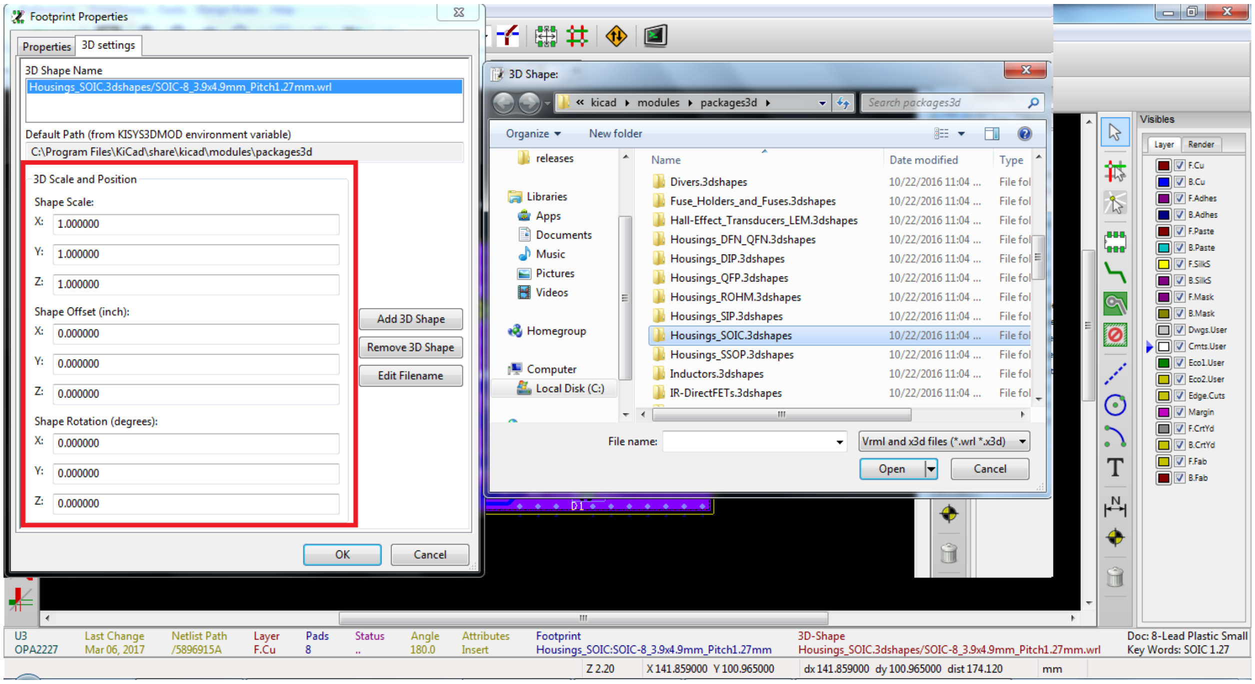
3D Rendering with no OPA2227



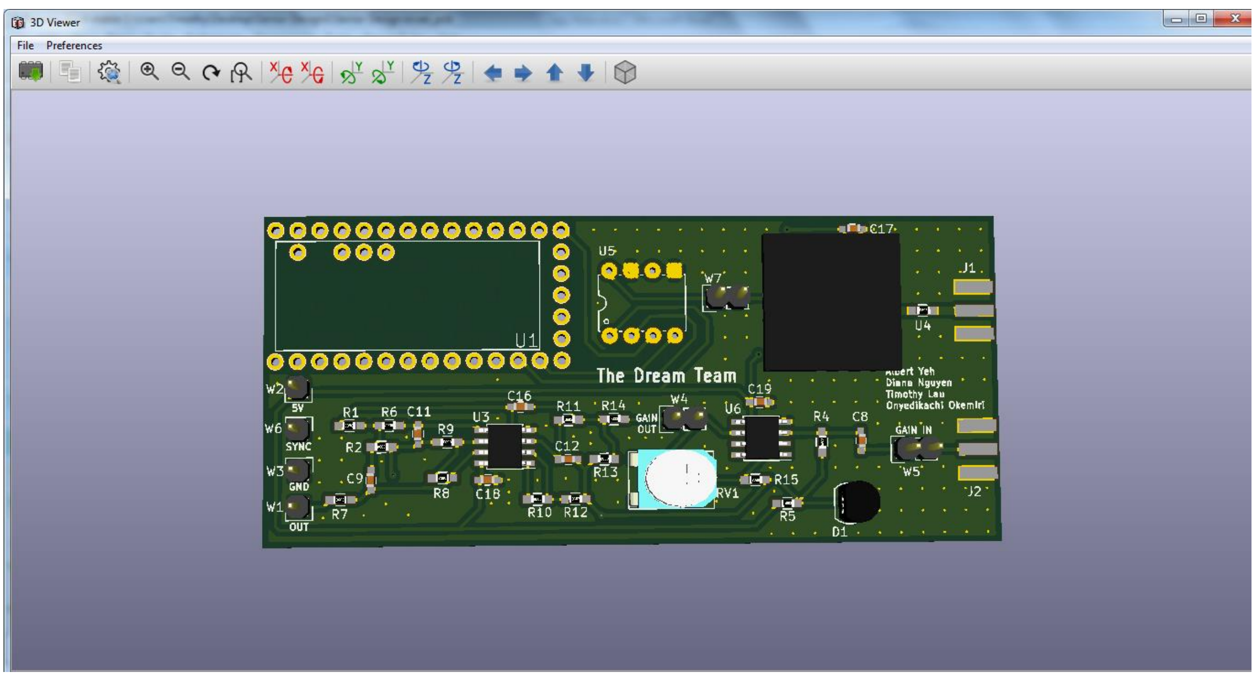

As in our example above, we will now add a 3D model to the OPA2227. To do so, we go back to our PCB layout and select the OPA2227 footprint and choose the ‘Edit Parameters’ tab.



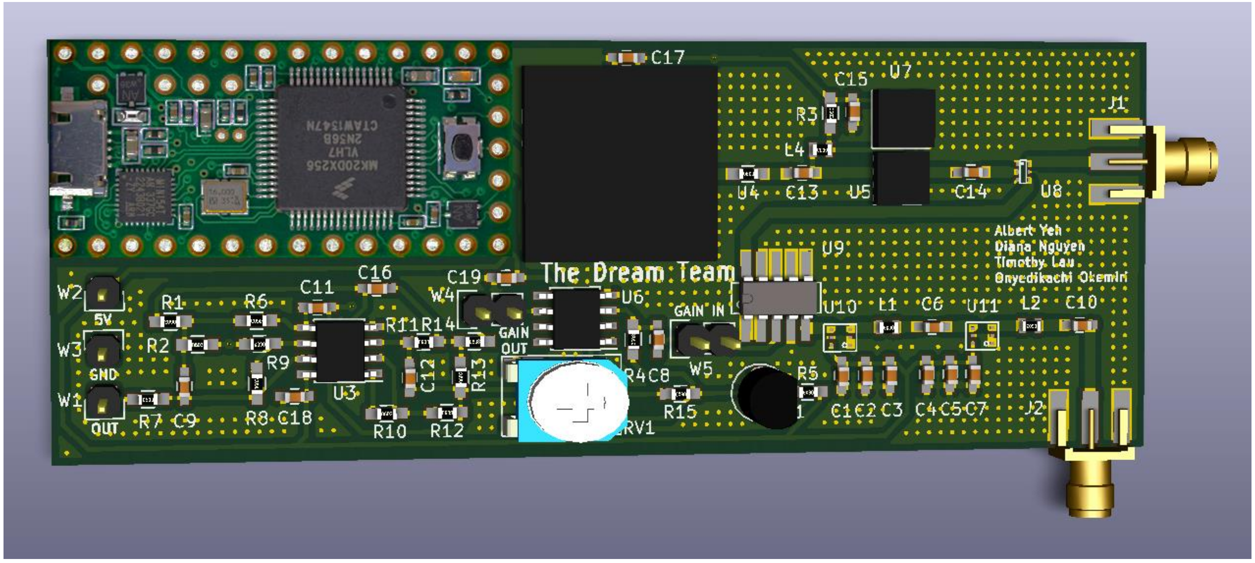

From this window, we select the ‘3D Settings’ tab and choose ‘Add 3D Shapes’ below. This will open the KiCad 3D model library. If a model does not exist, they may be found online.

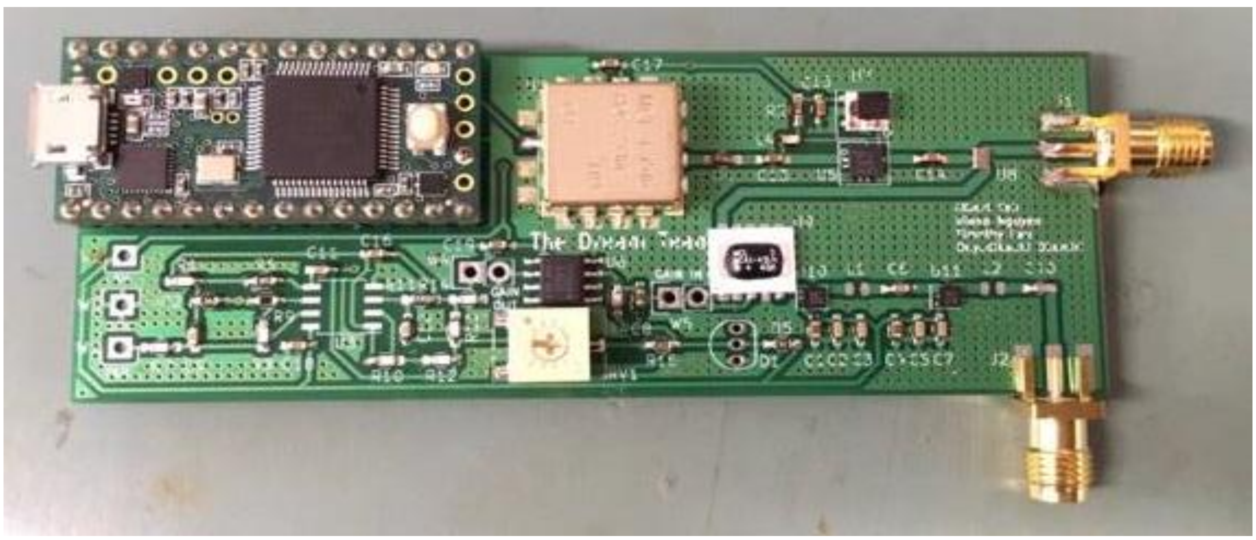



After a model is imported, it can be positioned accordingly and scaled to fit the 3D rendering.




OPA2227 with OPA2227






Afterwards, the rest of the 3D models can be placed onto the rendering to complete the design and be used as a guide for PCB assembly.