

PCB DESIGN: CREATION, TRACES, AND VIAS

We are now going to learn how to design a PCB using Altium. This section assumes that you've already created your schematic in the previous section so if you haven't done that please complete it before moving forwards.

1. Open the example project that you created in the previous section.
2. Right click on your project, click “Add New to Project” and then “PCB”

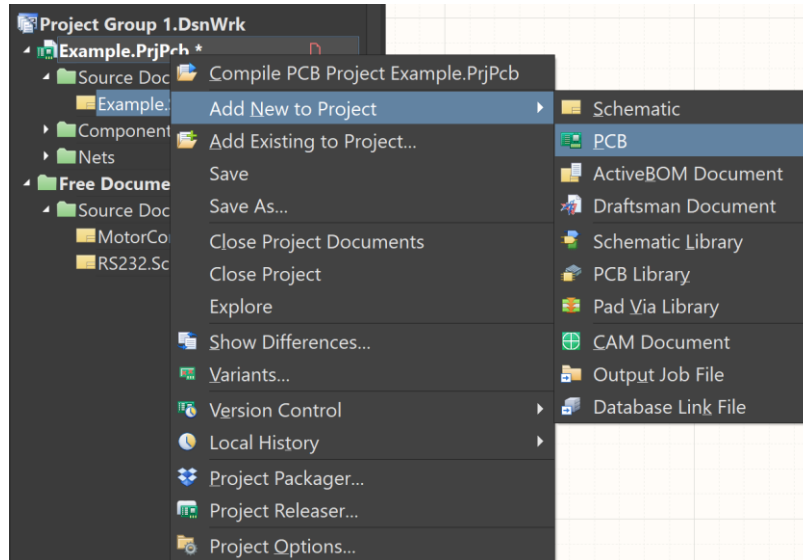


Figure 1: Path to New PCB

3. Click on Project > “Compile PCB Project Name_of_Your_Project.PrjPcb”

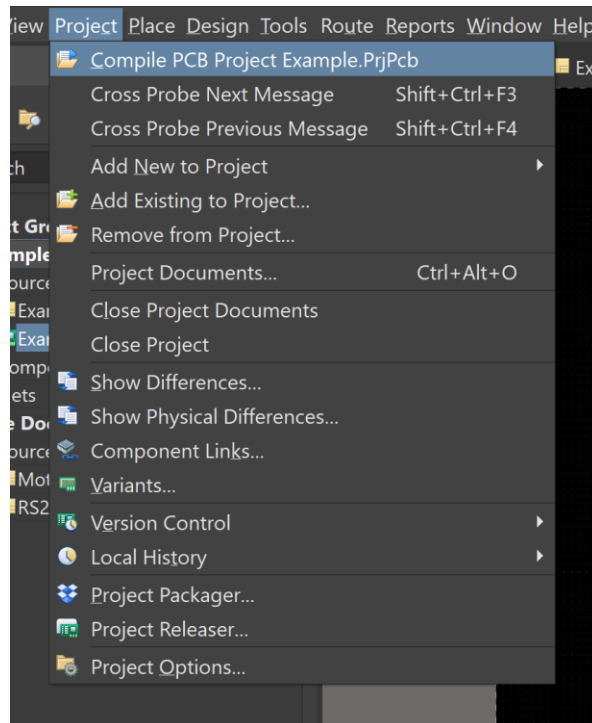


Figure 2: Path to Compile

4. Go to Design > “Import Changes From Name_of_Your_Project.PrjPcb”

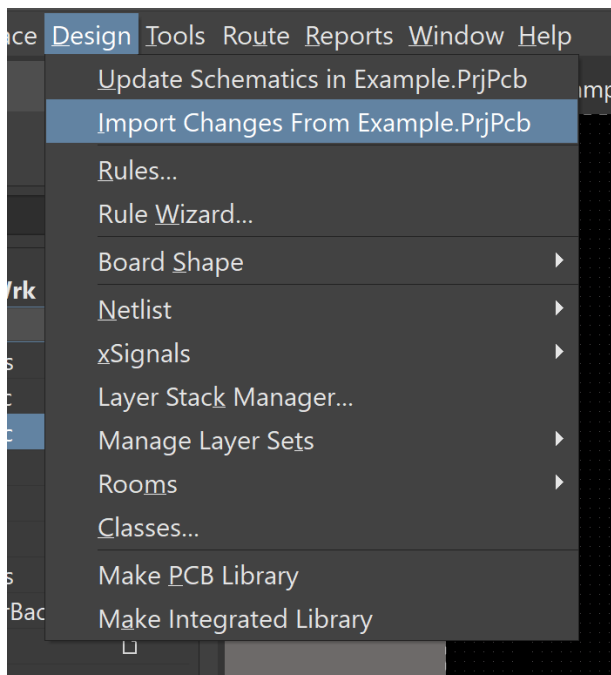


Figure 3: Path to Import Changes

5. You should see the screen in Figure 4.
6. Uncheck “Add Rooms”. While rooms can be useful under certain circumstances, they are typically unnecessary and often annoying to work with.
7. Click on “Validate Changes”.
8. Execute changes and make sure there are all check marks on the side of the screen
9. Close the pop up.

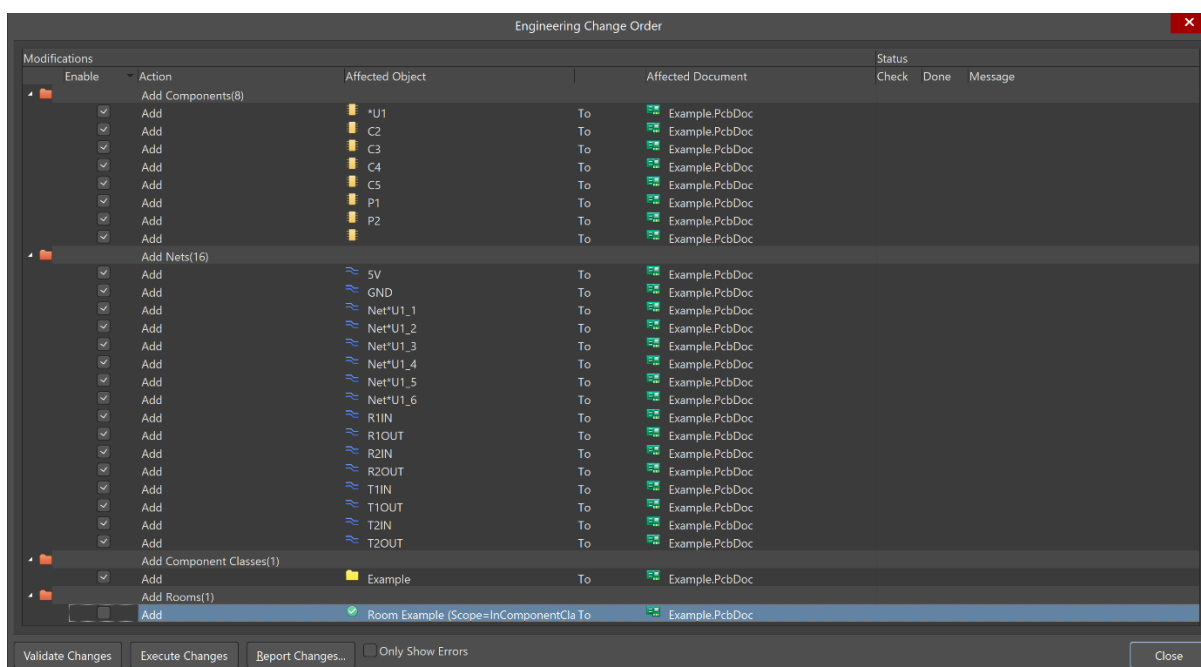


Figure 4: Pop-Up Screen Showing List of Changes to PCB

10. You should now see a screen similar to Figure 5.

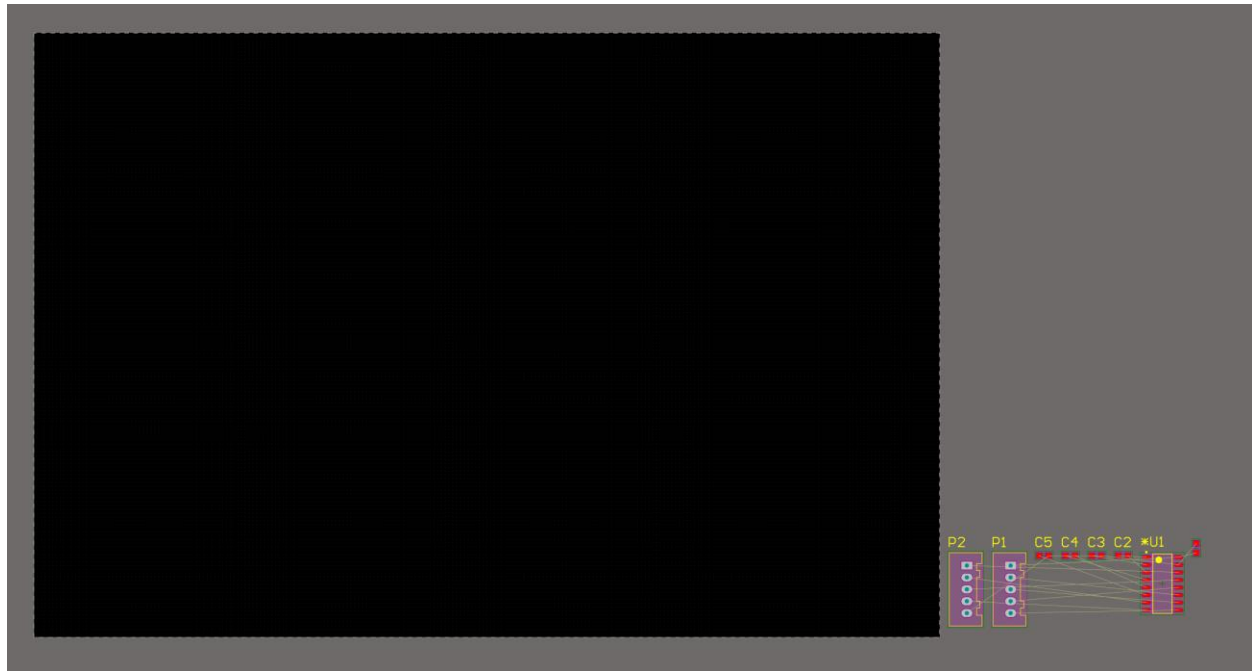


Figure 5: Default Layout of New PCB

11. Move your components to the bottom left corner and organize them in a similar style as Figure 6.
12. The lines represent nodes that will eventually need to be connected. Altium immediately finds the closest node to connect to so, as you move your components around, the lines will most likely change. You can ignore them for now.

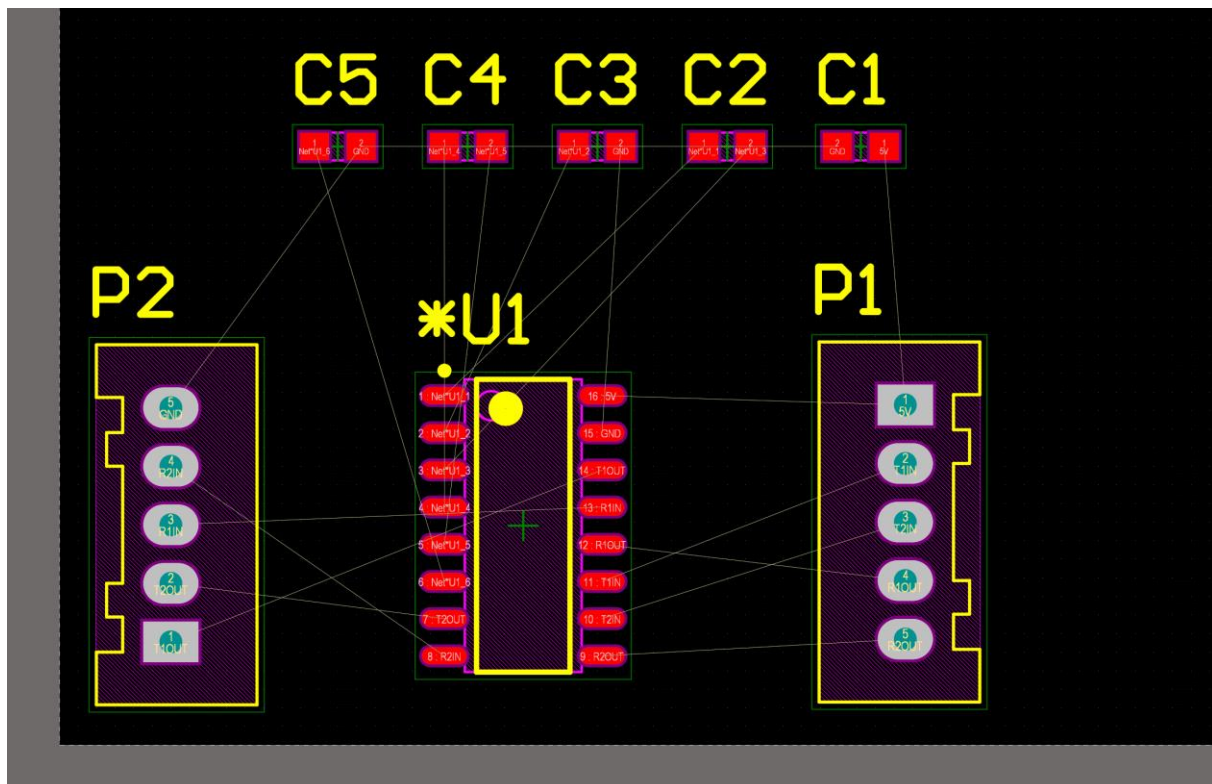


Figure 6: Initial Organization of PCB Layout

13. Now we want to organize our parts to be more compact together. Typically, you want to put your parts as close to their respective nodes as possible to prevent noise, reduce space, and make routing simpler.
14. Sometimes you will want to put capacitors on the bottom of the board (currently all are on the top layer). To do this, select a components properties (double-click) and click on the drop-down list called “Layer” then select “Bottom Layer” as shown in Figure 7.

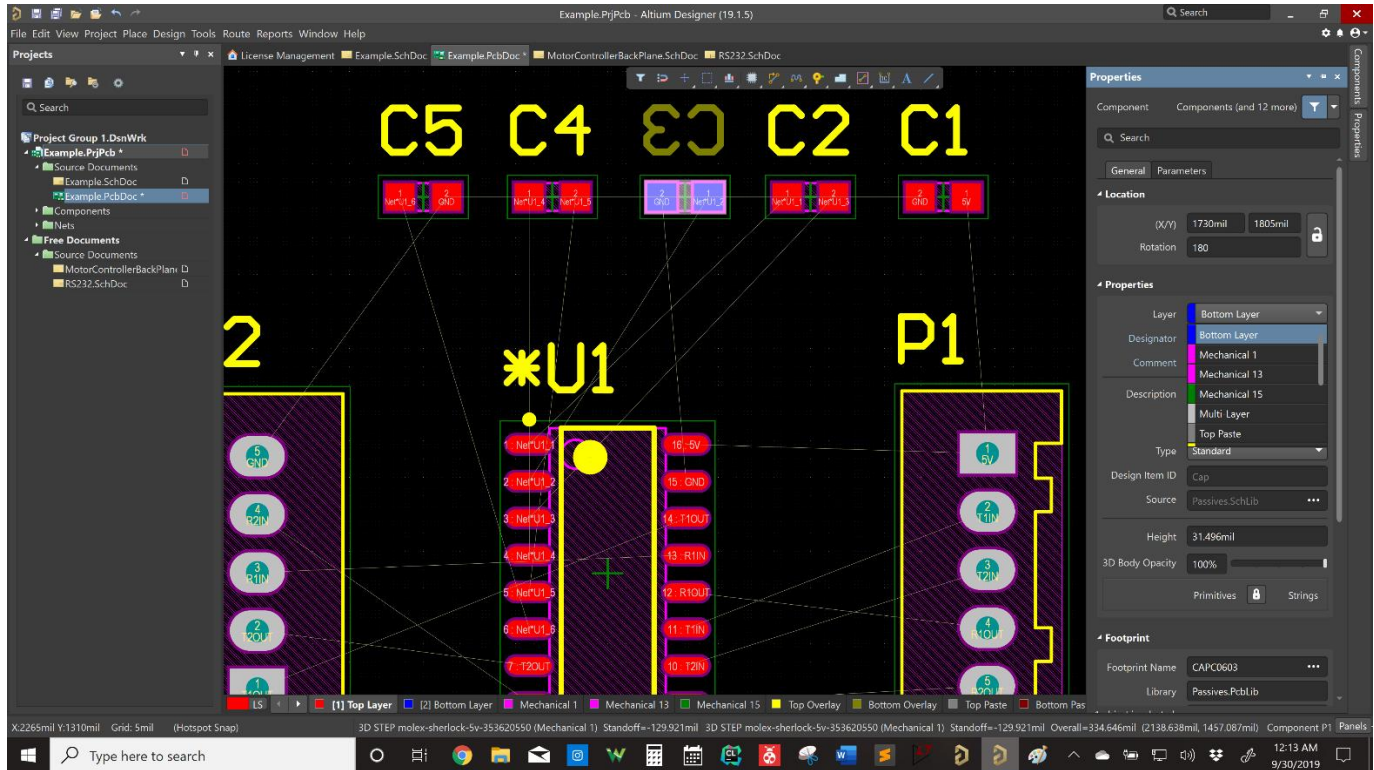


Figure 7: Properties Screen to Move Between Layers

15. To specifically work on a layer, simply select it on the bottom list as shown in Figure 8.

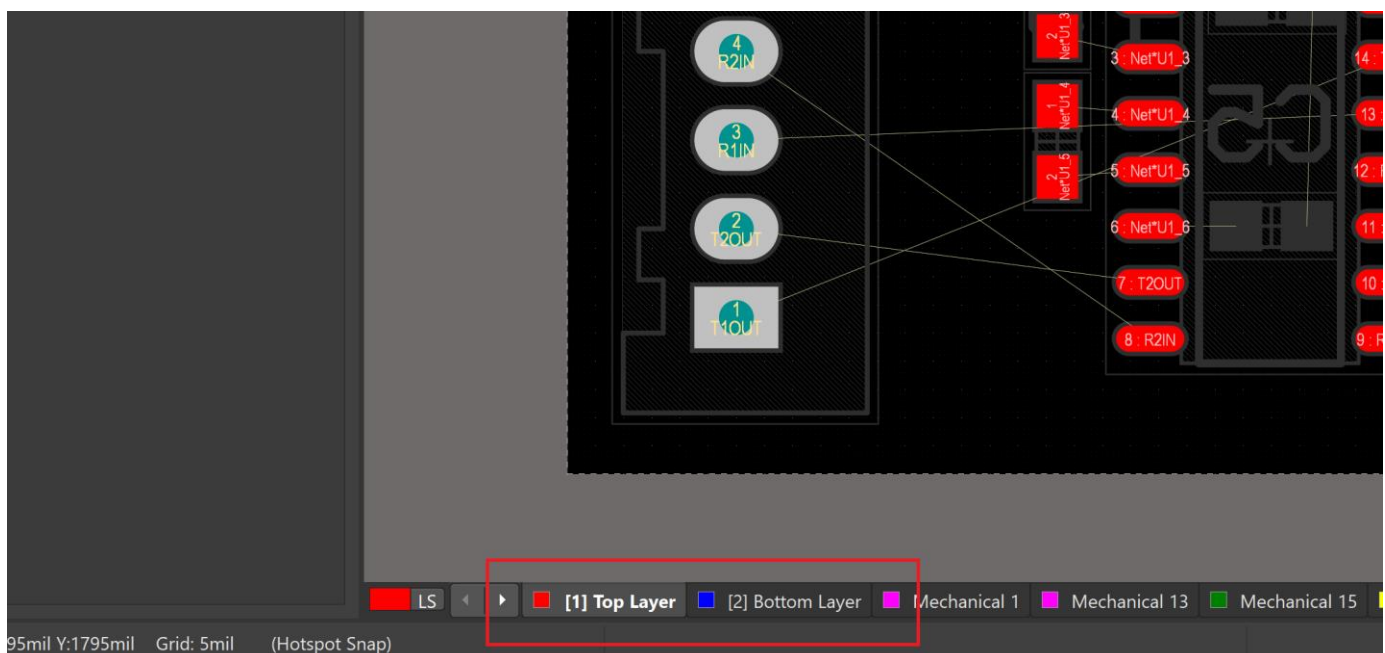


Figure 8: List to Select Between Layers to Edit

16. Using the information from steps 14 and 15, organize your capacitors as shown in Figure 9. Remember that you can rotate your components using the space bar when holding them.

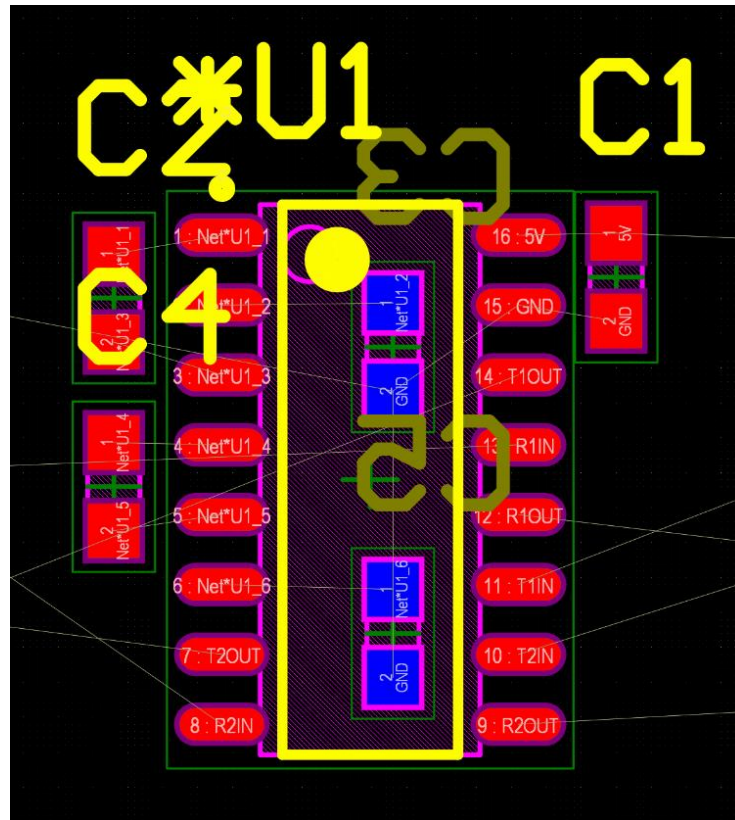


Figure 9: Ideal Layout of RS-232 and Capacitors

17. Your PCB should now look similar to Figure 10.

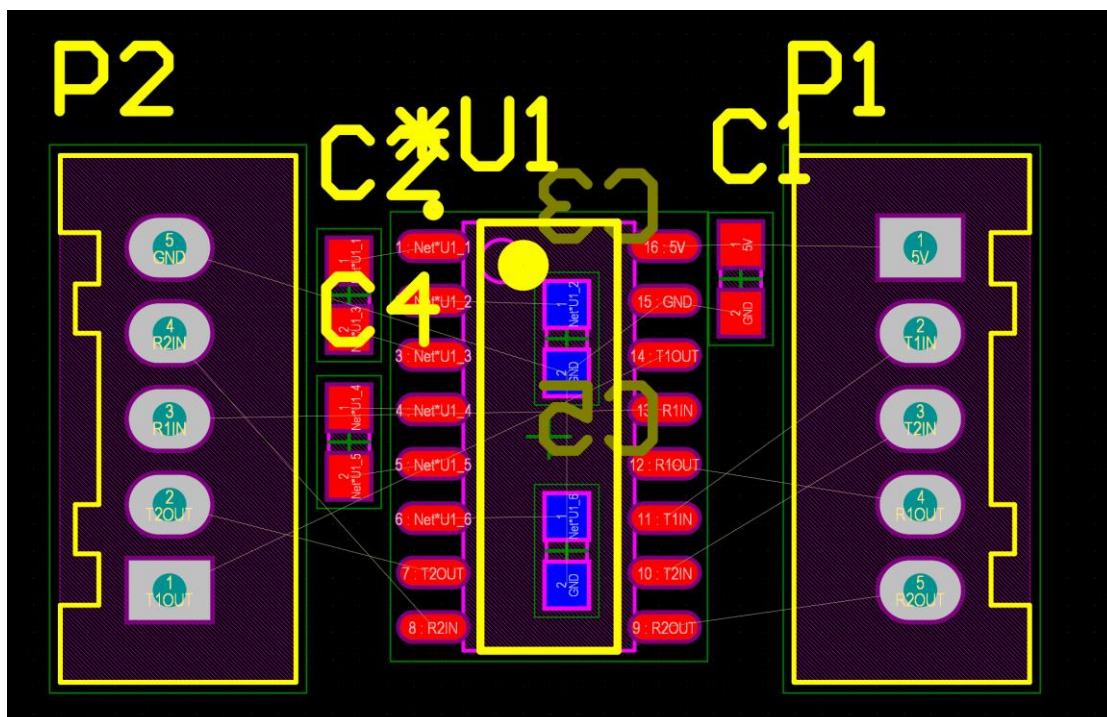


Figure 10: Full Component Layout of PCB

18. You can switch between three different views of your PCB by pressing Shift-S.
 - a. The first view shows all layers at once. This view is useful for an overall view of your board and for demonstration purposes but isn't very efficient for editing.
 - b. The second view highlights a single layer and dulls all the others. This is useful for editing a single layer while still understanding how it fits in the whole PCB.
 - c. The final view solely shows your current layer. This is useful when you don't care about the rest of the PCB and just need to focus on your current layer.

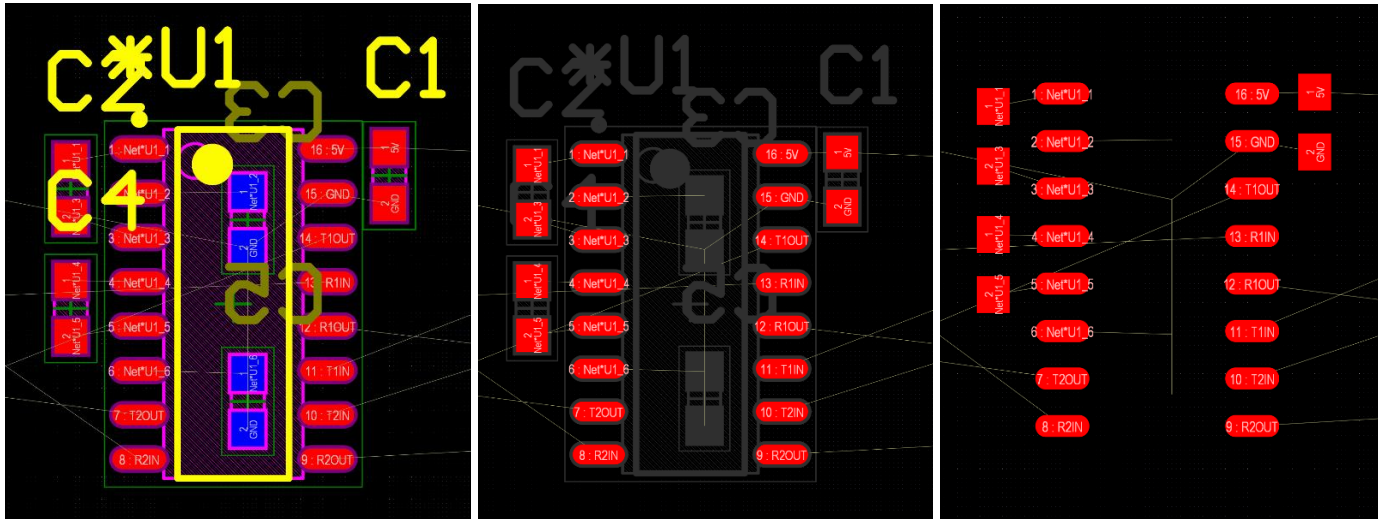


Figure 11: Full View, Enhanced View, and Isolated View

19. Press P then T on your keyboard. This selects the tracing tool for Altium, which allows you to connect your nodes together.
20. Click on your top pad of your top left capacitor and connect it to the top pin.
21. Next click on the bottom pad of the capacitor and connect it to the third from top pin as shown in Figure 12. (Note that Altium will not allow you to connect it to any other pin since it is not on the same node)

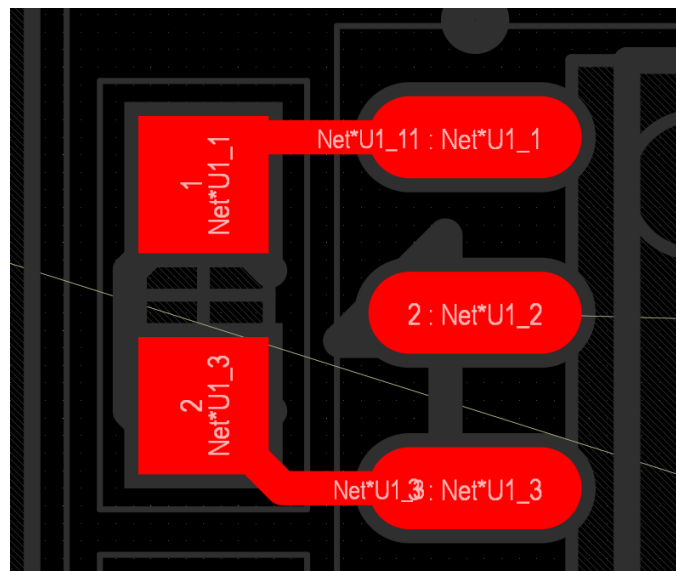


Figure 12: Example of Basic Tracing (Add macro with box around this cap)

22. Complete the traces shown in Figure 13. Notice that as you make certain traces, the paths of other traces get cut off. (Note: we will ignore connecting the GND nodes together in this part since there is a method to connect them that will be covered later) (Note #2: no angles should ever be less than or equal to 90 degrees unless absolutely necessary)

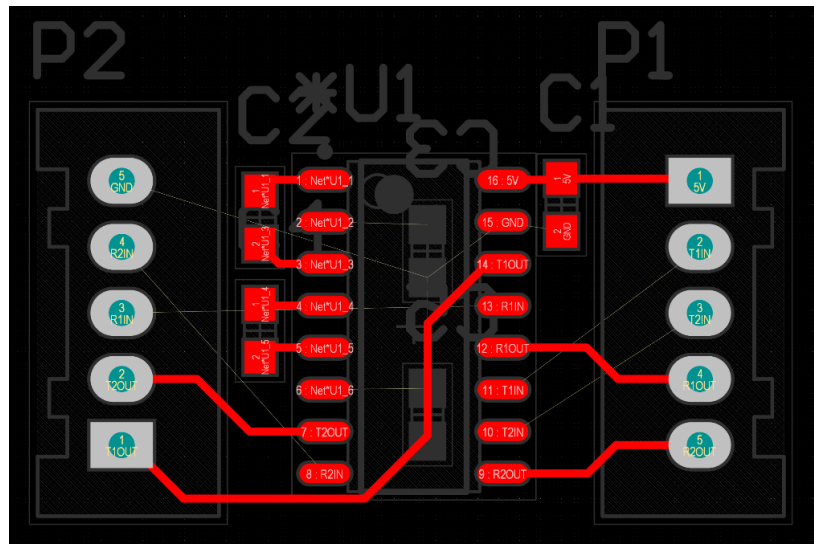


Figure 13: Initial Tracing of Top Layer

23. To connect traces from one layer of the board to the other layer, we must use vias.
24. Click on the circle icon and select “Via” as shown in Figure 14.



Figure 14: Path to Via

25. Place your vias in the following locations.

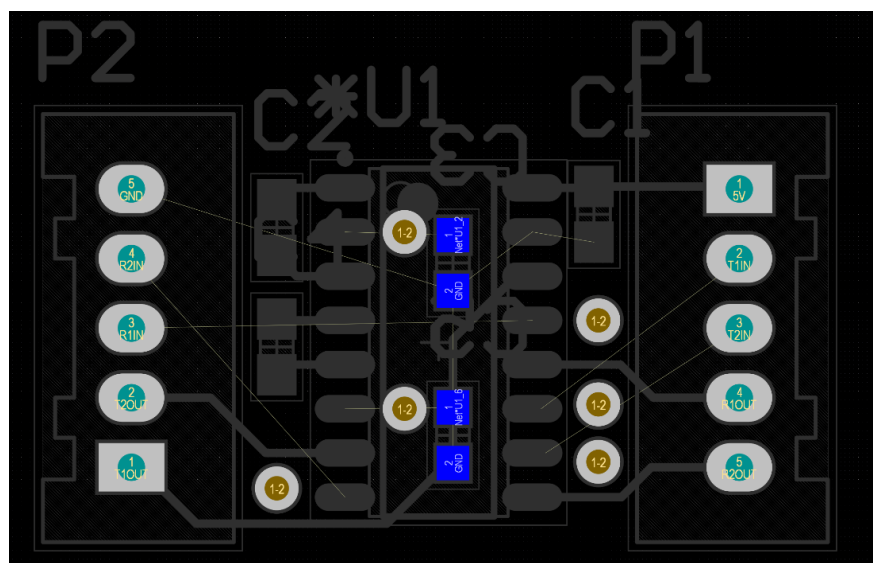


Figure 15: Positioning of Vias on Bottom Layer

26. Now we must label our vias so that they can connect to their desired nodes.
27. Select the properties of your node and click on “Net”.
28. Select the node that is closest to your via, if confused refer to Figure 17.

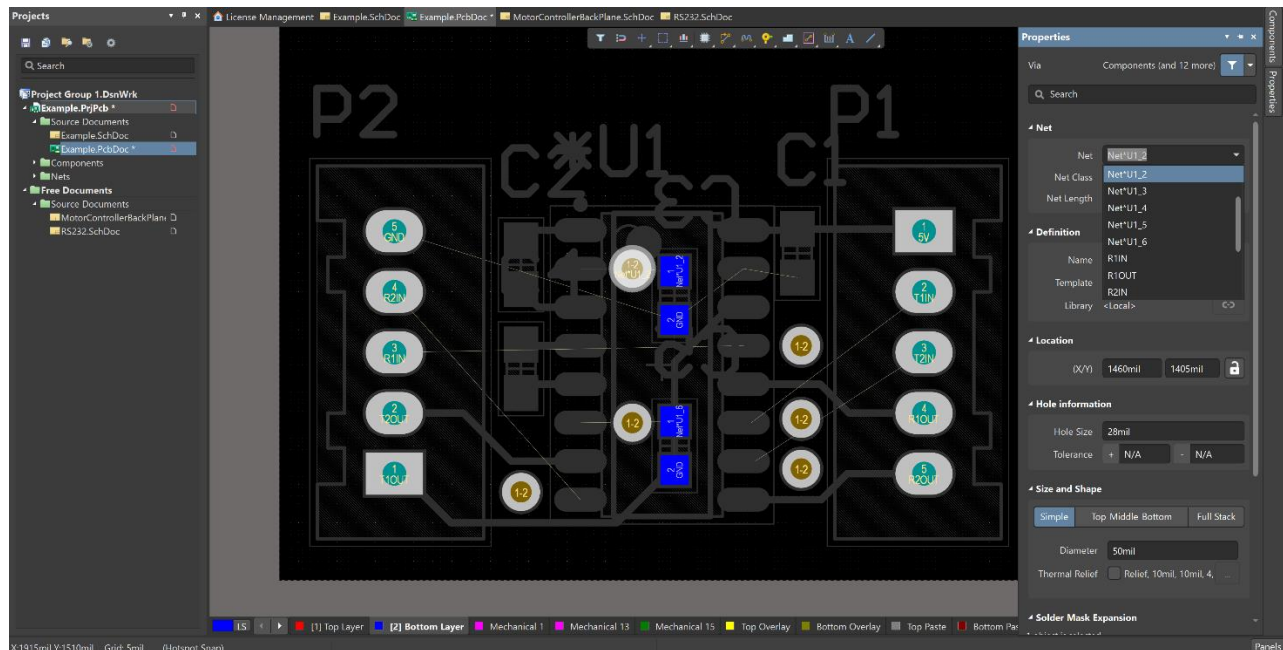


Figure 16: Properties Panel of Via to Select Node

29. If you labelled all your nodes correctly, you should have a layout similar to Figure 17. Notice how the net connections have now changed to include the vias.

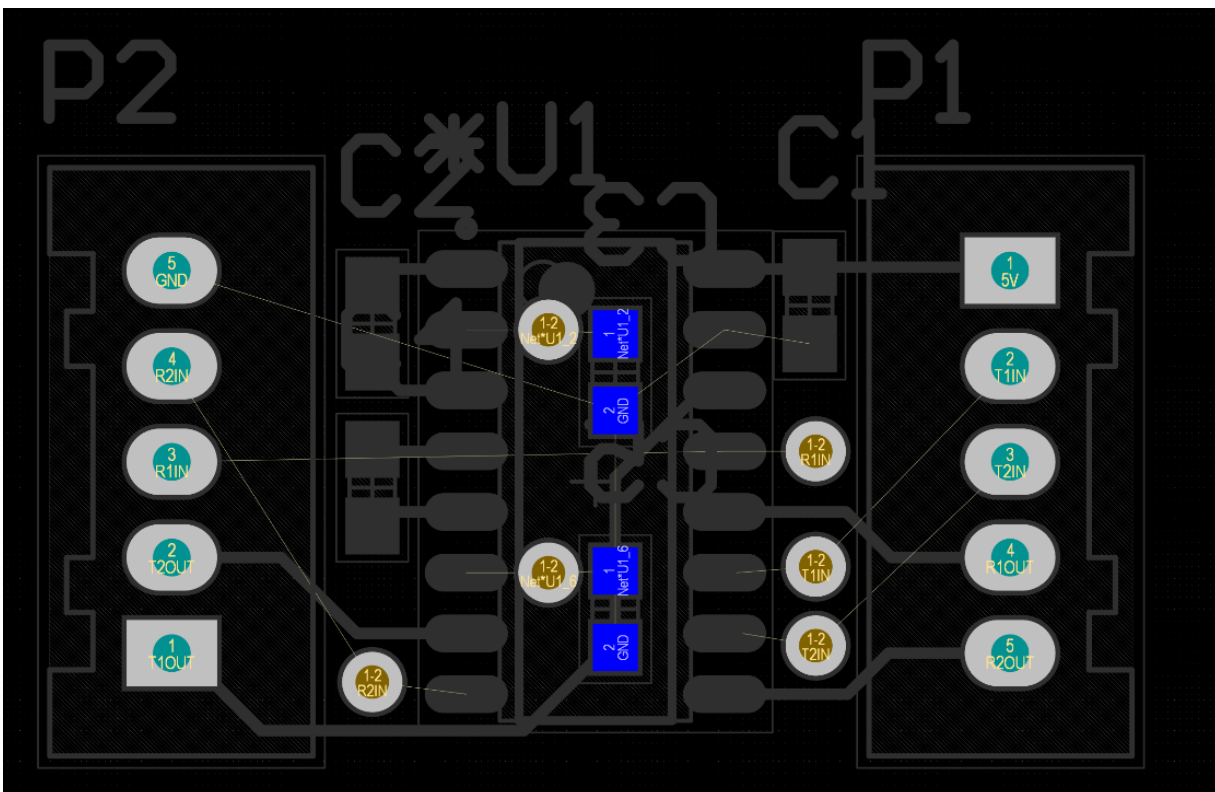


Figure 17: Bottom View of Fully Labelled Vias

30. Now connect your nodes to their respective vias on the using traces on both the top and the bottom. (Note: vias are present on both the top and bottom layers, that's their whole purpose)
31. You should get layouts similar to Figures 18 and 19.

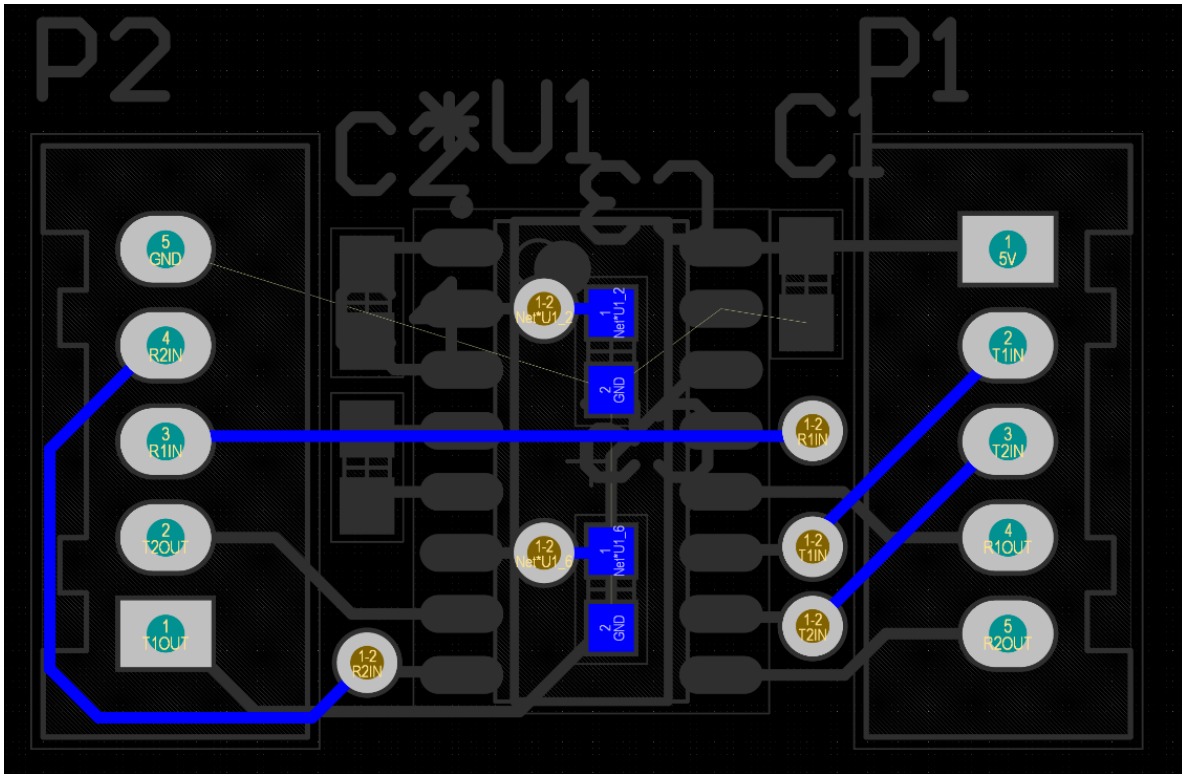


Figure 18: Bottom View of Via Connections

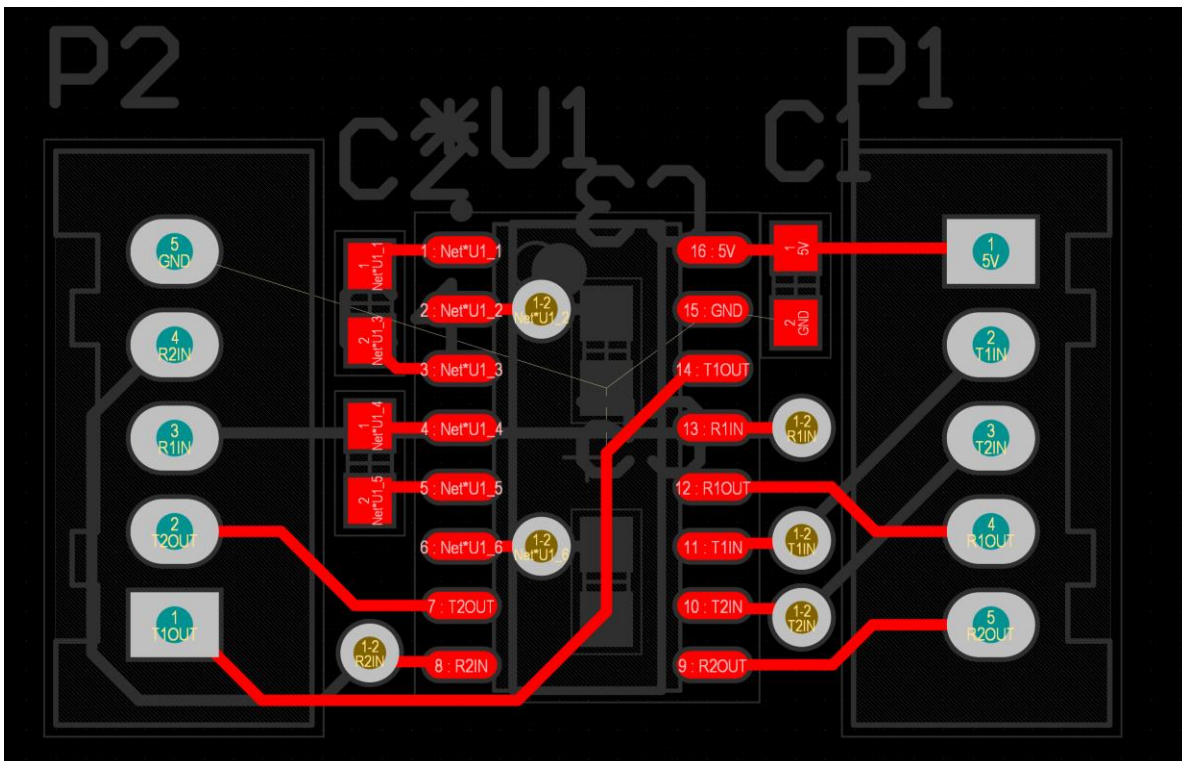


Figure 19: Top View of Via Connections

32. You have now completed all of the traces you will need for this PCB.
33. If you did it right, you should have a layout similar to Figure 20.

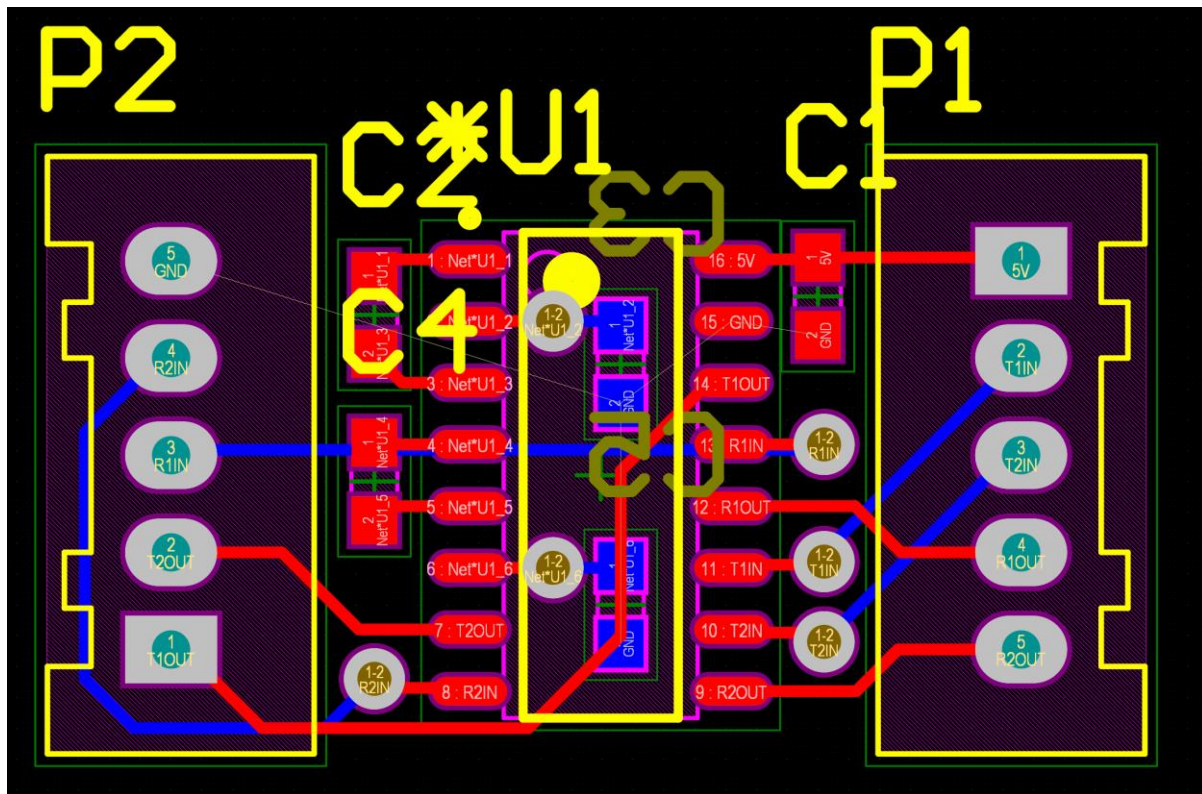


Figure 20: Full Trace View of Current PCB Layout

This ends Part 1 of our PCB tutorial. The next part will include more advanced techniques for connecting your nodes, cleaning up your labels, and defining your board outline.