

Designing a Printed Circuit Board for Haptic Interface with PCB Artist

Bin Tian

April 4, 2013

ECE480 Design Team 6



Summary

This application note is intended to guide the reader through the process of using the PCB artist software to create a circuit schematic and generate a printed circuit board (PCB) for our Haptic Interface Project.

Keywords:

PCB, Design, Schematic, Layout, PCB artist

Contents

Introduction	3
Objective	3
Getting Started	4
Creating a schematic.....	5
Converting the Schematic to Layout	9
Design Rule Check.....	10
Order the PCB from PCB artist	11
Recommendation and Tips	11

Introduction

The **Printed Circuit Board (PCB)** is a professional way to integrate the electronic components into a single board. The prototype design of a breadboard can be transferred into the PCB design. It allows surface mount components to save more space and make the circuit more secure. Since physical wires are eliminated, ICs on the PCB are protected and lead to the correct peripherals to each other. Computer aided software is the tool used to design a PCB and create fabrication files. PCB Artist is one of many options available. PCB Artist is an easily available software and a student version can be downloaded online for free.

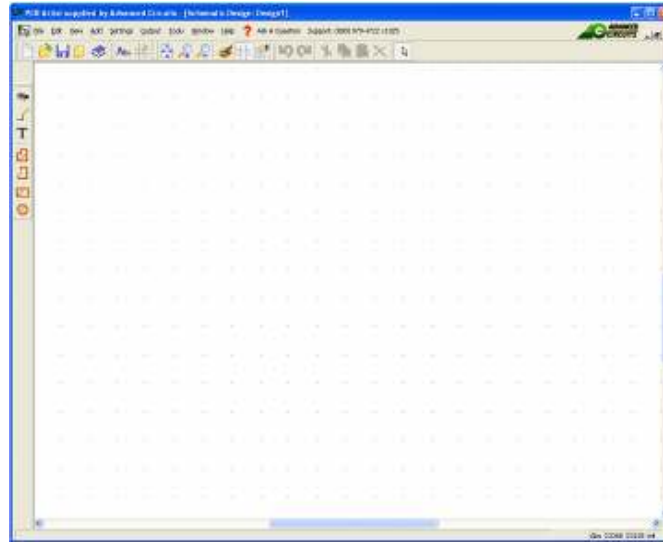
Objective

This application note will enable the reader to obtain a basic knowledge of using the PCB Artist to design a schematic and create a layout. Furthermore, it allows the reader to understand our haptic interface circuit design. Our design is used as an example for this application note. The steps of how to make a PCB for a haptic interface PCB board are:

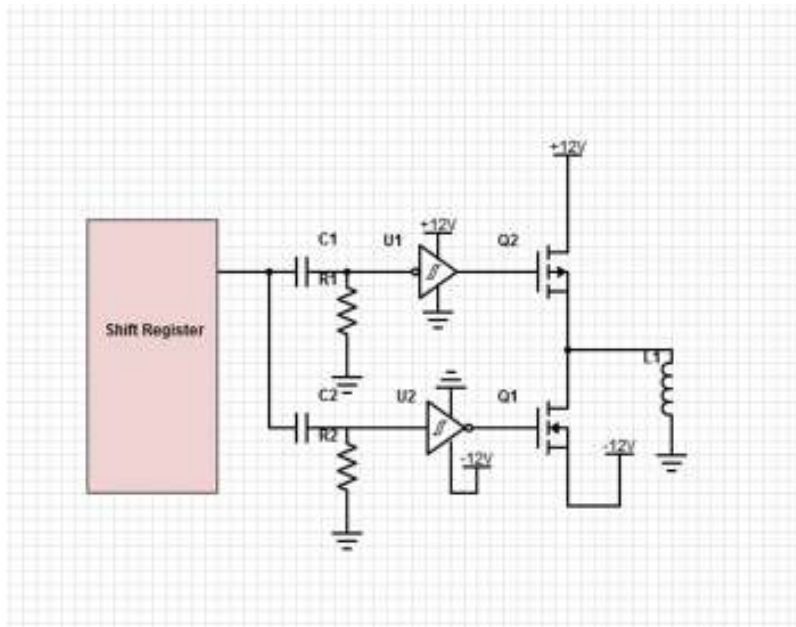
- Begin a design in PCB Artist
- Create a circuit schematic
- Modify a schematic
- Convert the schematic into the PCB
- Run a design rule check
- Submit fab (fabrication) file and order the board

Getting Started

Run the PCB Artist software. Select File->New to open a blank schematic window. Most electronic parts can be found in the libraries since the PCB Artist's default libraries include common components as well as libraries for parts made by other manufactures.

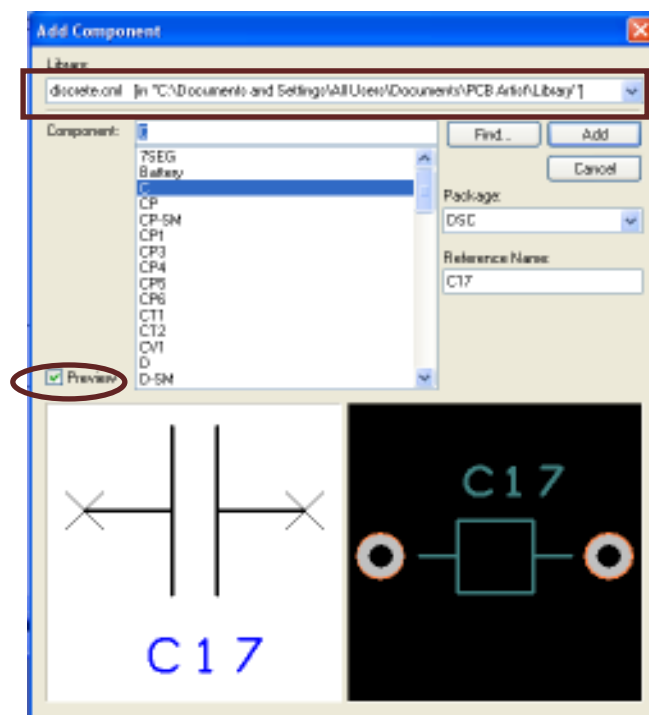


Before designing a schematic, we must know what the circuit will look like and parts we plan to use. The entire circuit may be designed on a breadboard or sketched on paper. The virtual ground circuit of the haptic interface shown below is used for a PCB design example.



Creating a schematic

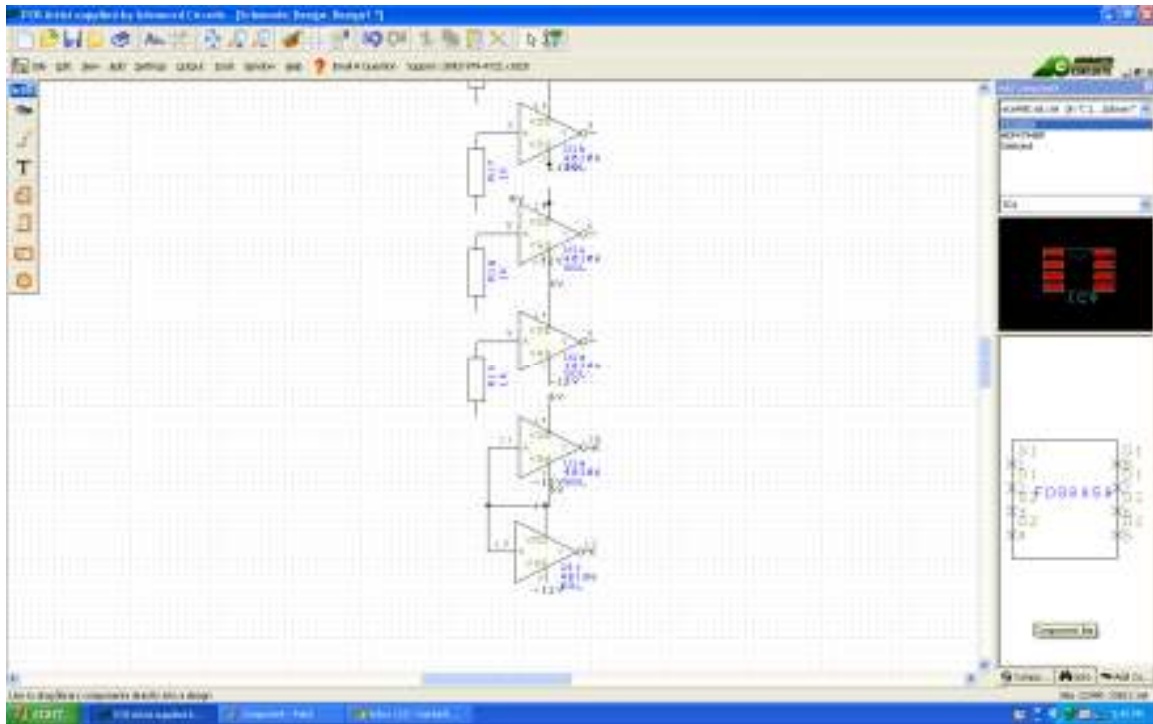
Now it is time for us to work on the schematic design page. Go to the Add Component tab in the lower right corner of the window and select a library we want to use. If we want to look for a component from a different library, we could click the small “down” arrow on the upper right corner to overview the full libraries list. The graph below is an example I applied for our circuitry design. To view the PCB footprint, click the Preview button to display previews of the Schematic symbol.



After we choose components what we want, we can click the add button and place them on the schematic design page. We can adjust the position of the component on the schematic properly to make it easy to connect the nets. Left click the pins of the components and connect to others following the sketch of the virtual ground circuit. If the component has to be rotated, right click the component and click rotate button.

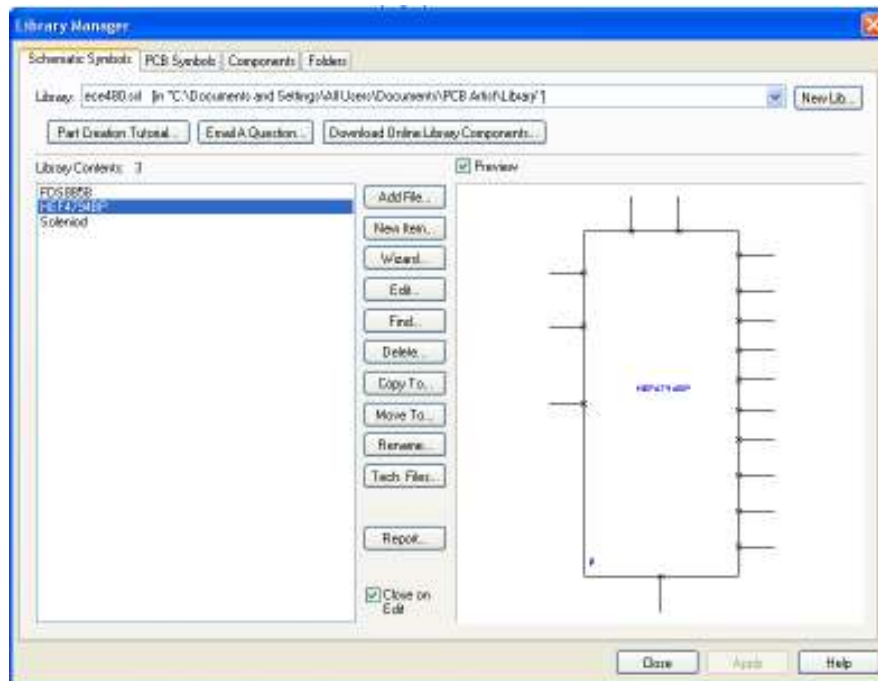
Using “NET” or “Label” idea is a very efficient way to build a schematic. It also allows us to reduce the connection wires on the schematic, which leads to a neat schematic. We do not have to connect every single pin on the schematic by visual wiring. For example, draw a piece of the short wire from the power supply pin on the inverter. Right click the wire and select

properties. Define a general name for the net and click “ok” button. The power label does not have to be connected to the inverter. However, they have been connected logically. Later on the layout it will automatically display the logic connection between the power supply and the inverter. This technique is particularly useful for larger schematics with many component or ICs with many pins.

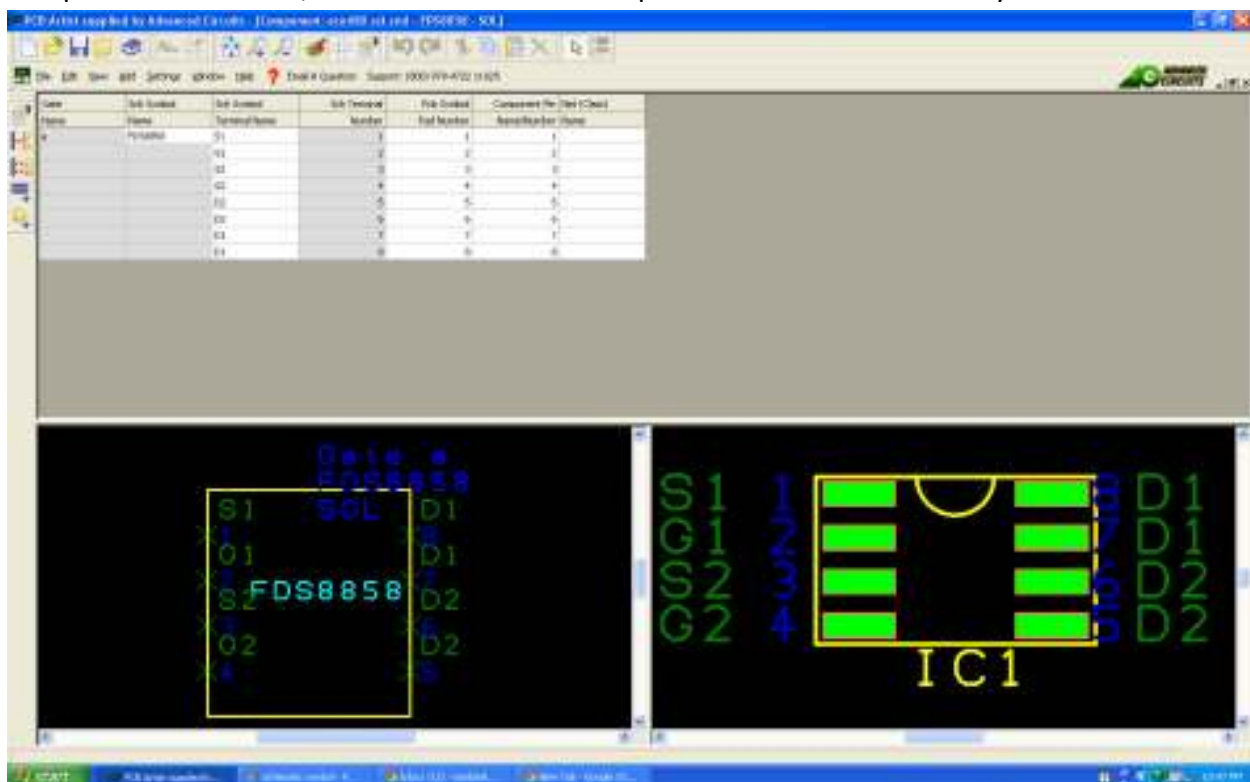


Since the “+12v”, “-12v” and “GND” are logic display in the PCB artist, they do not have a physical layout. We should add through-hole pads from the library and place them on the schematic page to connect with power nodes. Therefore, the actual PCB is able to connect with the external power supply, ground and signal inputs.

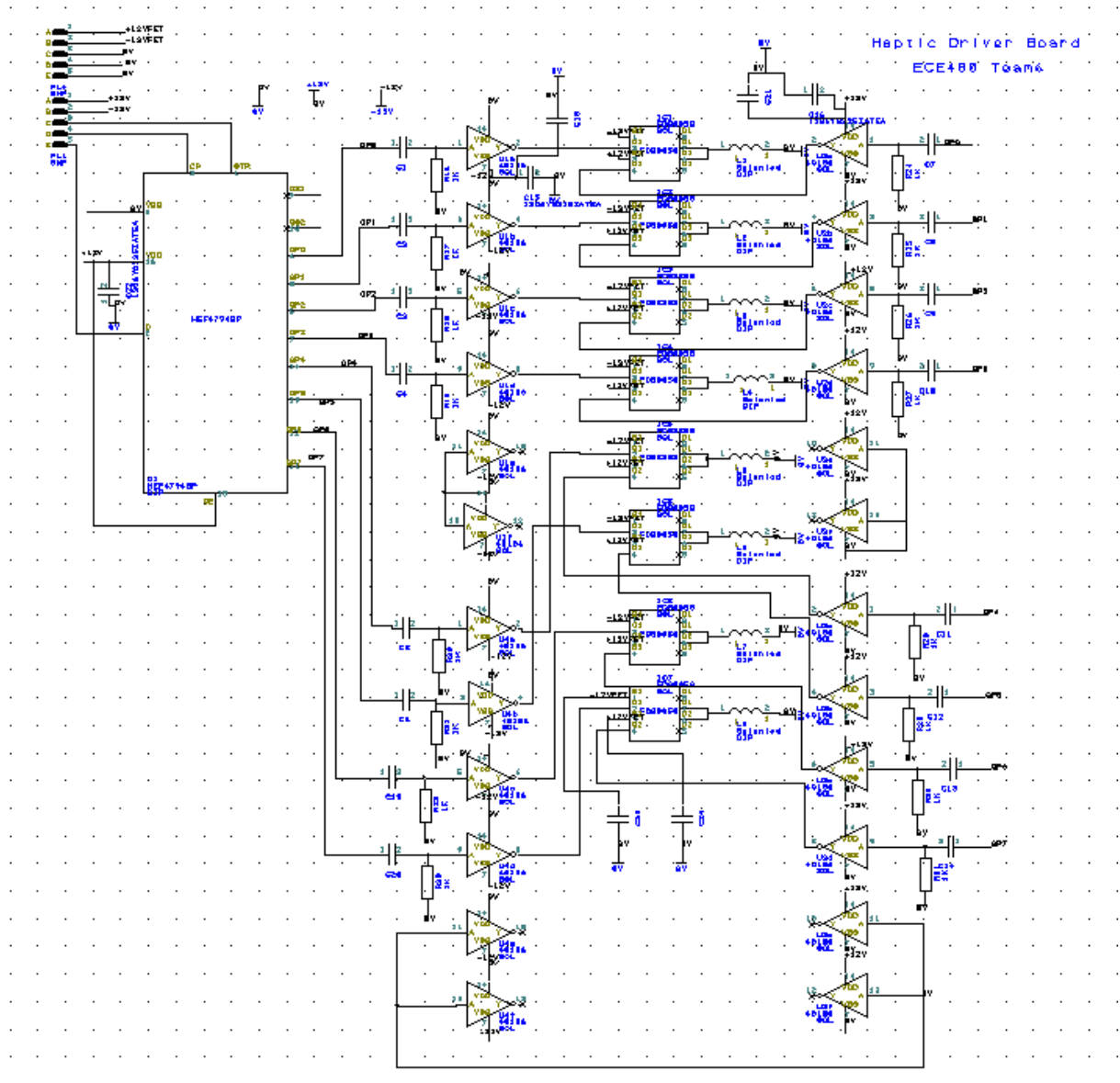
When we create our schematic, we figure out we have some larger components in our design that could not be found from given libraries such as FDS8858 (MOSFET), HEF4794BP (Shift register) and solenoid. Creating a new library is necessary to clarify the design. Left click library manager on the upper tool bar and a window will pop up. In the window, four categories are shown and they are schematic symbols, PCB symbols, components and folders. For this project, we only have to use first three categories. The graph attached below shows the schematic symbols category. Click New Item that we can design a schematic symbol. Follow the same way; we can design a PCB symbol corresponding to the schematic symbol. We can choose the surface mount or through hole. Lastly, the sizing could be adjusted.



When we finish the schematic symbol design and the PCB design, the last step of building a component for our project library is to use Component category. Click New Item button. A New Component window will pop up. Define name and Select schematic symbol and PCB symbol from specific libraries. Finally click OK and everything will be integrated into the new component. Therefore, we can use the new component from our own library.



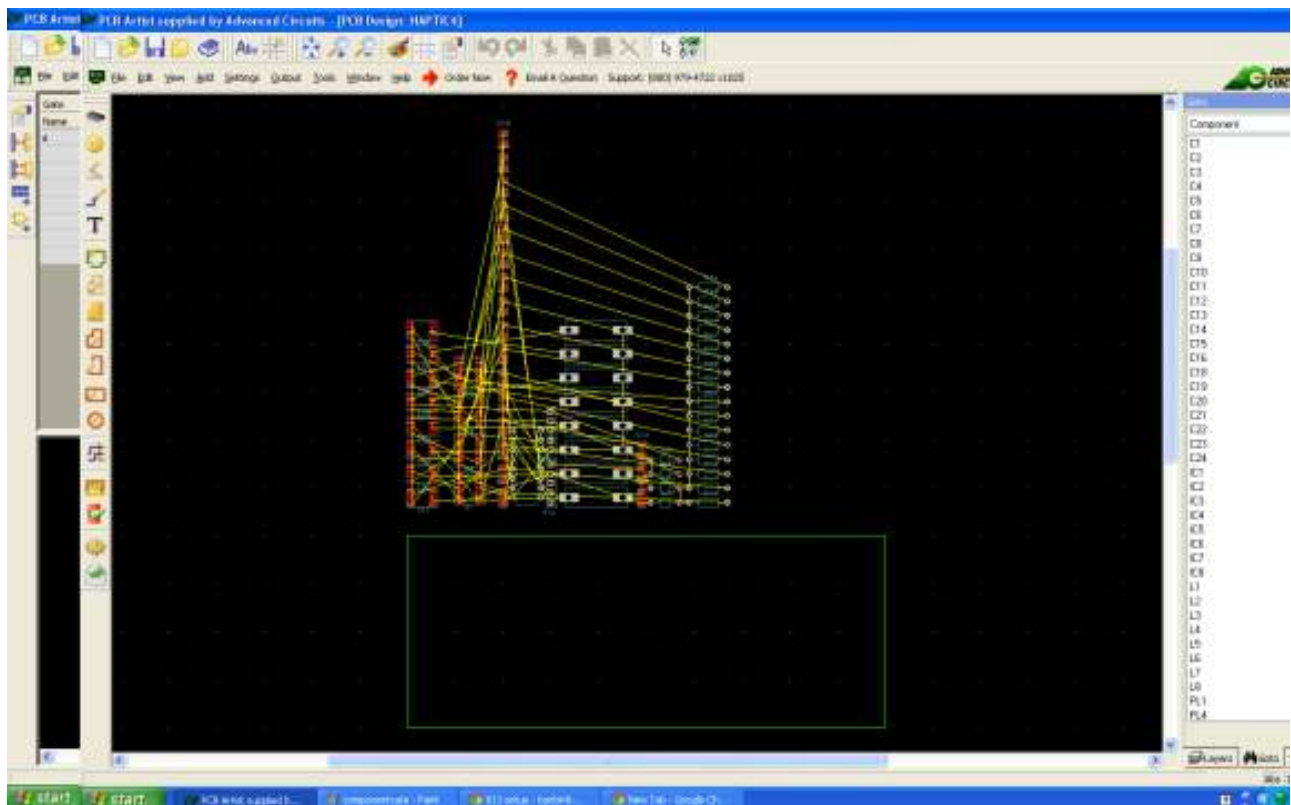
Open the component file; we can see the picture shown above. Symbol names and pad numbers of the PCB symbol can be matched by labeling the names and numbers based on the real components. After this is done, we have finished the component design.

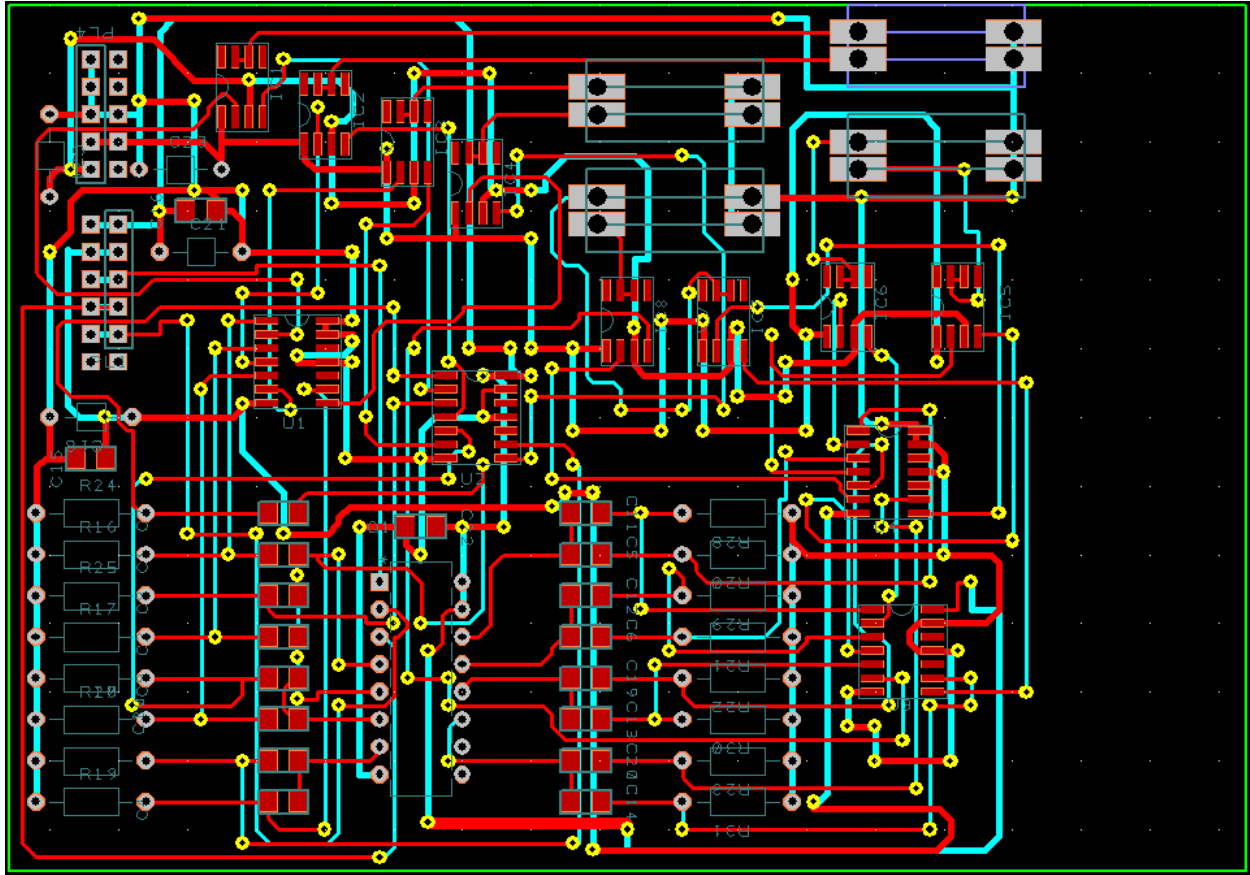


Here is what our final schematic design looks like. Half of the process of designing a PCB has been completed.

Converting the Schematic to Layout

After finishing the schematic design, click the tools button on the top bar->Schematic to PCB->Translate to PCB. This will bring up the PCB wizard window that we work on the layout design. The footprints appear outside of the green outline with yellow air wires representing the net connections.





This shows actual layout for our PCB design.

Design Rule Check

Design rule check (DRC) is a vital step before finishing the PCB design. It is a useful tool to avoid the design errors. Errors have to be fixed with trace and pad spacing. Click tools->consistency check. The DRC will check the consistency between PCB design and schematic design.

Meanwhile, short circuits and spacing violation will be found by DRC. If the schematic design changes such as adding a new component, we can click forward design changes button and a new component will show up in the layout page. We can adjust the layout until finalizing it.

Order the PCB from PCB artist

A button called “order now” can be clicked for ordering PCB boards from what we designed. This function is built into the software and makes the process easier. Click place order button and upload the fab file automatically created by the program to the website. Fill the ordering information such as numbers, size tolerance and so on. Make sure all information is correct and submit the order. The company, Advanced Circuits, will receive the order and manufacture PCBs for us.

Recommendation and Tips

- Give enough space for ICs before layout
- Traces on the top side could be designed horizontally and on the back could be designed vertically to avoid the wiring cross
- Ground and power traces are usually designed in wider shape