

Tutorial on Schematic Design and PCB Artist

By SeanVanHaren

A40484030

Team 7

Introduction

The printed circuit board, or PCB, is an integral part of electronics fabrication in this era. PCBs are used to connect electronic components mechanically using conductive signal traces etched from copper sheets set onto a non-conductive substrate. The architecture of the board is specific to the desired circuit and planned using specialized PCB design computer software. Multiple layers of traces are used when wires would overlap to give the desired output and performance, and not destroy the integrated circuitry. This is a huge change from the time before PCB in which physical connections between ICs had to be done with wires in a floating fashion. This took up much more space and the circuit was much more vulnerable. With the PCB the ICs are soldered into place on the substrate making them more secure, protected, and conduct the leads to the correct peripherals and other ICs. The process of designing a schematic and PCB is fairly simple with the PCB artist software available for free on the internet. The most difficult of the design process is in the creation of the initial schematics. Just as in many processes, getting started is the most difficult step.

Step 1: Elements of Schematic Drawing

Before starting the schematic, we must know what we will be making and the parts we plan on using. The entire circuit may not need to be designed in one swoop. Segmentation can help the whole process especially if more than one person is involved in the initial designing.

To start the design process there are some materials that will make the procedure much easier.

- Graph/Vector paper to keep things straight, neat, and organized. A ruler is also
- A good mechanical pencil to start to get ideas and first drafts out
- A three color pen (black/blue, green, red) for the final drafts to easily identify Vcc, ground, and normal interconnections.

When finished with the first few drafts, check it over for any possible small mistakes in normal circuit convention. Things like pull down/up resistors, capacitive smoothing (capacitor between ground and Vcc), diodes and electrolytic capacitors in the correct orientation, possible floating components, and short/open circuits that may damage the ICs. The best way to fix this is meticulously label everything so it is easier for someone else to read it, and have multiple people check it over for mistakes.

Once the schematic is finished and revised to top working order in the most efficient manor, it is time to lay out the circuit on a breadboard and test. It is important to make sure the circuit works before proceeding to PCB Artist because any mistakes on the PCB after it is sent in are not possible to fix. This will set back the project a few days to weeks.

Step 2: Transferring written schematic to PCB Artist

First, PCB Artist must be downloaded onto the computer. The best place to find it is at 4pcb.com. A link to the free download is easily visible on the homepage. The steps to installation are easy enough to follow from the website.

After the program is installed and opened, go to file new, and create a new project. On the schematic design page, go to the Add Component tab in the lower right corner of the window (adding components by clicking on the thumbnail of the chip on the upper left corner of the window or by hitting F8). The library selection bar is at the top of the side bar with the list of components listed beneath that. Layout pictures of the component are beneath that and it shows the PCB layout with the schematic layout under that. Searching for specific components may be easier by using the add components thumbnail (F8). This has a searchable aspect to it which can look through all the existing libraries. Many large parts, like microcontrollers and accelerometers in our team's case, will not be available in any of the libraries. In this case, we found components with the same layout and package. Relabeling of these parts is easy enough, and there is really no other way around this with the limiting factor of the libraries. It is best to add the biggest ICs into the schematic first to have a good starting point.

Most important libraries to note:

- Discrete (resistors, capacitors, LEDs, diodes, switches, inductors)

- Schema (Ground, Vcc, Vdd, Vss)

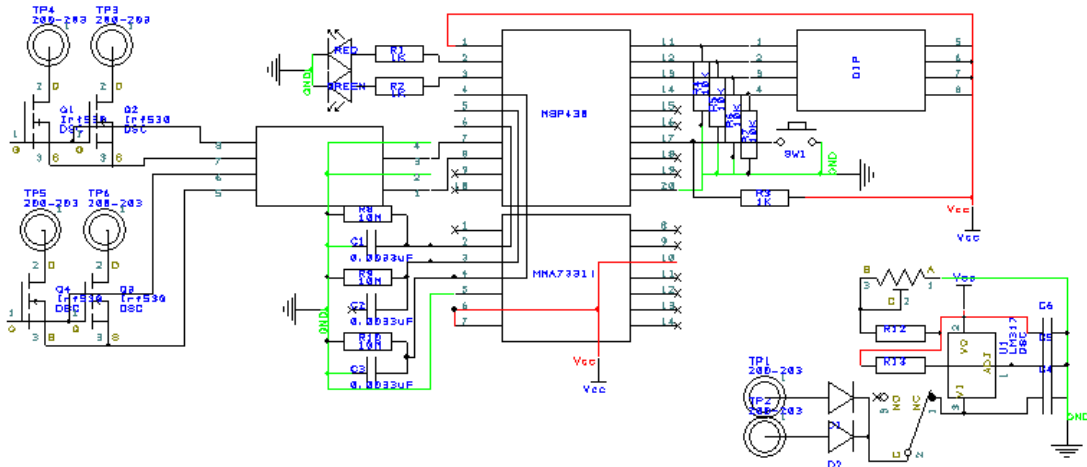
Labeling of parts can be done by double clicking on the component and adding/removing/changing the labels and values to the desired settings.

Once the desired part(s) have been found, just click and drag the picture of the component in the bottom of the side bar. The component can be rotated by the R key, and flipped by the F key. Other hot keys (Ctrl C for copy, Ctrl V for paste, Ctrl Z for undo, and Ctrl Y for redo) are incredibly helpful and streamline the schematic process.

Connections are made between two components by clicking the x of the desired lead and dragging it to the desired endpoint in the connection.

Nodes (or as PCB Artist calls them: Nets) can be kept track of in the Goto tab in the sidebar and selecting Net in the dropdown bar. It is best to label the nets as easily identifiable names to keep the schematic organized and easily readable. Labeling the nets can be done by right clicking on a wire in the net, selecting Change Net..., and adding the name of the net. The net color can similarly be changed by right clicking the net, selecting Change Net Color..., and selecting the desired color for the traces. It is best to have Vcc as red and Ground as green for easy identification.

Once done, have others in the project group look over it for errors.



Here is what our finish schematic came out looking like. Obviously, every circuit schematic will end up looking different. This is just a visual example to help.

Step 3: Transfer to PCB and finishing

Transferring from schematic to PCB can be fairly tricky and annoying at times. It is free software, so this is somewhat expected. Just be patient with the programming and it won't be so difficult

Before transferring, we need to know the dimensions of the board so it can fit into its casing. Some estimation will be needed with limiting factors on the board size, and this can be done with trial and error in the PCB wizard.

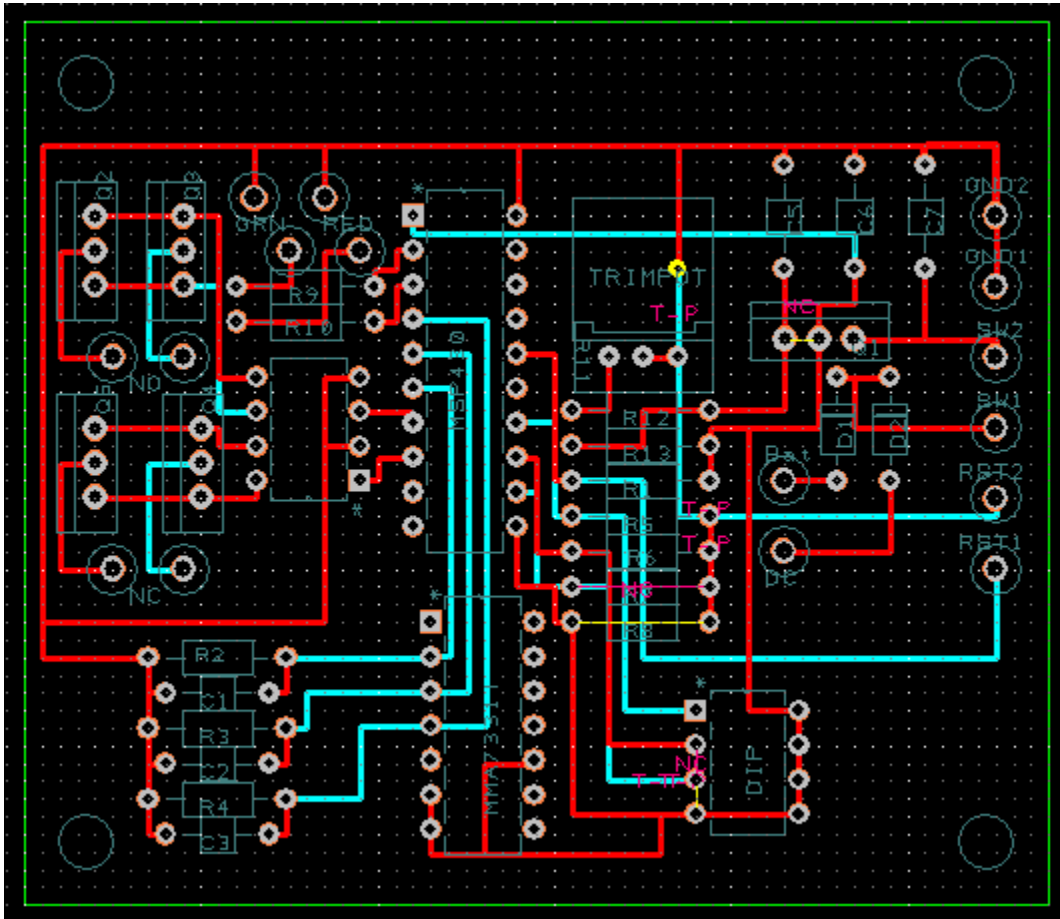
To transfer to PCB, first go into Tools on the menu bar, go down to Schematic <--> PCB, then over to Transfer to PCB. This will bring up the PCB wizard window. Go through with the desired preferences (board dimensions, number of layers, ect.) and hit finish when done. The output will more than likely be somewhat of a jumbled mess with yellow lines everywhere. These are not the actual traces of the circuit. Everything must be arranged in a fashion that uses the least amount of space and the best use of the available layers for traces.

A few things to note on the PCB design:

- Layers of traces can be changed by the L key.
- Give enough space for components that overflow over the allotted area (DIP switches, micro controllers, and other various ICs).
- Figure out the required trace thickness for constraints (current and temp) of such traces. Size can be changed by selecting the net and hitting the S key.
- Leave enough space for the pads of the ICs with traces. Errors will flag this on the design rule check.
- A ground plane can be added by going to the Add button on the menu bar, select Copper, select the desired shape, then click and drag to the desired size.

Do a compatibility check with the Design Rule Check button midway down on the left hand side of the window. Any errors will appear in a prompt and on the PCB design.

Have the entire group check over the PCB design schematic for errors. Any errors discovered after the design is sent in for fabrication will cause major setbacks.



This is our nearly finished PCB design. Some errors have to be fixed with trace and pad spacing.

Conclusion

As an electrical engineer, it is good to know how to create schematics on software like PCB Artist. Many companies use similar software in product design, and PCBs are not likely to change little in the many years of electronics advancements in the years to come. Fortunately these programs are intuitive and the learning curve is fairly shallow. Help is all over the web in user manuals and video tutorials.