

1. Two general PCB fabrication methods are available. Thru-hole is easy for an amateur to build, in small quantities using normal tools and a decent soldering station. SMT is way more compact but the small detail soldering takes more time and a very steady hand. This is likely best left to machine fabrication by a custom board house, especially if a large number of boards is needed.
2. It's generally a good idea to breadboard a new design to check functionality and fine tune component selections. You will need to download and install DipTrace free, either 64 or 32 bit depending on your system. A quick pass through the Tutorial is time well spent, though tedious.
3. To start your pcb project, create and save the schematic in DipTrace Schematic. Then from File click on Convert to PCB. PCB Layout will open with a haphazard layout of your components connected by (thin blue) ratlines according to your schematic. Adjust the components in logical order best way possible, rotating and moving as needed to minimize/rectangularize the footprint. (Another option is to open PCB Layout and from File, open Renew Layout From Schematic, and select By Component. Then enter the schematic file name and click Open.)
4. If minimum board size is important, you can fit axial lead components such as resistors, diodes, capacitors etc vertically where possible. Packages that may need heat sinking are best mounted at board edge. All connectors and headers should be at board edge. Larger boards especially should include mounting holes at the corners if possible.
5. Pay close attention to the component pin numbers because DT follows these numbers for routing trace connections. Make the board as compact as is reasonable because the board builders charge by area. Most often a 2 layer (double-sided) rectangular board is best.
6. When the layout looks about right; from Route, select Run Autorouter for trace generation. Once complete run all Verifications, including the compare to schematic. Fix any conflict or clearance issues. You can drag trace segments to improve clearances as needed. If necessary DT will put part of a trace on the other side of the board, with "vias" through the board. This looks a little odd but is normal. DipTrace defaults to 13 mil (.33 mm) wide traces. This is fine for low power boards < 100ma. Increase default trace sizes for load carrying traces (up to 1 amp) up to 20-25 mils wide, or more. Select in Route, Route Setup.
7. Run the 3D view and check for any conflicts. You can print the board top layout in full size and check that the components actually fit.
8. You can move components by dragging with the mouse; DipTrace will adjust the traces to suit. To edit the location of component ID's for the top silk, press F10 **once** and drag each ID to where you want it. With the top or bottom silk layer open, use Place Text to add any other text needed.
9. If modifying an existing board using a revised schematic; first open the board file in PCB Layout. Then select Unroute All to delete all previous traces. Then import the new schematic as per 3 above. Re-used components will stay put, and new ones will be off to the side of the board, with ratline connections. Simply drag them to where you want them, rotating as needed. Once done run Autorouter again to create the new traces.
10. You can print your schematic directly from the File/Print menu. It will print as dark lines on a white background. For printing the board layout go to File/Preview menu. You can upsize a small board using Print Scale setting, in increments of 100/200/300/400%. Center the board on the sheet using the centering tools. In Show, select Current or All Layers as desired. Once done select Print. Select Portrait or Landscape from your printer properties menu.
11. Once all checks are done and you are happy with the board layout, it's time to make the Gerber files. Go to the File menu and select Export, then select Gerbers. Select Export All and export to a new destination folder. You **must also** separately open Export and **export the N/C Drill file** to the same folder. It will be called Through.drl; all the rest are .gbr files. Next right click on the new

folder and Send it to a compressed zip folder. This is the zip file you will send out for board fabrication. There should be 14 files total in the zip folder.

12. For only a few boards, or for a few trial boards before placing a large order, try OSH Park (US) or JLCPCB or SEEED (China). Cost per board will be high, but you should get them within 2 weeks. Once you get them, do a test build, check operation, and fix any problems in DT before placing a larger order. It's almost certain that you will find a few things to tweak or add for the final run.
13. For a larger run of say more than 20 boards, consider an Asian fabricator such as AllPcb or PCBWay. Your unit board cost will be a fraction of domestic pricing, and the quantity discount breaks are attractive. Allow 3-4 weeks for delivery however, depending on their backlog.
14. For boards up to about 2" in either dimension, 0.8 mm thick boards should be fine. For larger size boards use 1.6 mm thick for more strength. Consider 2 oz copper where available, though it will usually add more cost. (2 oz is standard for OSH Park). For a large lot of smaller boards, panelization in lots of 3 x 5, or 4 x 6, or more works quite well. The panels are scored along the board edges and will separate very easily.
15. Usually for smaller boards, populating and soldering is easier done in a strip of 4 to 6 boards separated from the multi-board panel.