

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. Once you have found a schematic that you want to build, it's generally a good idea to breadboard it to check functionality and fine tune component selections. Next 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. From the DipTrace start up icon, open Schematic Capture and start building your circuit. When done save the 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. You have great liberty to move and organize the components as you want them. There is no really wrong way to lay them out, but usually flowing logically from left to right like the schematic works well, and will be easier to route. (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. You should get a decent starting layout to work from; not so haphazard.)
4. 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. 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. Make sure the component pin numbers are correct for the component pattern because DT follows these numbers for routing trace connections.
6. When the layout looks good you should set up the board edge outline. Open Objects, then Place Board Outline. Click in one board corner, then likewise for each consecutive corner, ending at the forth. Here right click, and press Enter. You should now have a nice straight-sided closed purple rectangle. You can drag any corners needed to just enclose board objects, and to make the sides perfectly horizontal and vertical. You can resize it at any time as board changes occur.
7. To add board dimension lines open Objects, click on Place Dimension, then Horizontal. Click on the upper left corner of your board, then move to the upper right corner and double click. You should now have a nice arrowed dimension line showing the board width in inches (or mm). Repeat for the vertical dimension line. Go to Normal Mode and simply drag the red dimension line a short distance away from the board. You should get 4 small orange squares formed with 2 aligned with the board corners. Repeat for the other dimension line. Now when you drag a board corner to move it, the dimension arrow automatically moves and the dimension corrects itself. Pretty neat!
8. 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 it's quite normal. DipTrace defaults to 13 mil (.33 mm) wide traces. (1 mil = .0254 mm) This is fine for low power boards < 250ma. Increase default trace sizes for load carrying traces (up to 1 amp) up to 20-25 mils wide. Select this in Route, Route Setup, Trace Width. For loads over 1

amp go to any one of the handy online pcb trace width calculators, such as that found on the Digikey site under Tools, See All. These are very easy to use.

9. From Tools run the 3D Preview and check for any conflicts. You can also print the board top layout in full size and check that the components actually fit.
10. 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.
11. 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. Resize the board outline as needed.
12. 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.
13. Once all checks are done and you are happy with the board layout, it's time to take the leap and 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.
14. 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.
15. For a larger run of say more than 20 boards, consider an Asian fabricator such as AllPcb or JLCPCB. Your unit board cost will be a fraction of domestic pricing, and the quantity discount breaks are attractive. Allow 2-4 weeks for delivery however, depending on their backlog.
16. 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.
17. Usually for smaller boards, populating and soldering is easier done in a strip of 4 to 6 boards separated from the multi-board panel.