

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. Refer back to it whenever needed, as you gain experience with DipTrace.
3. From the DipTrace start up icon, open Schematic Capture (red) and start building your circuit. You can find most any component you may need in one of the many included libraries. You can search for any component by going to Object, Find Component. In Search Area select All Libraries, then in Name enter the component name/number that you need. If found, several possibles may show up listed. Select the correct one and put it in the drawing. Play around with adding wires, flipping and rotating components etc. After a while it gets pretty easy. Make sure connections in your circuit are correct, otherwise it may not function correctly. When done save the schematic. To print your circuit go to File, Print, or Preview, and follow the toolbar icons.
4. **PCB Layout: A)** 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 really no wrong way to lay them out, but usually flowing logically from left to right like the schematic works best, and will be easier for trace routing. To rotate a component, highlight it and tap R repeatedly in 90 degree increments CCW.  
**B)** Another option is to open a blank 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 quite so haphazard.
5. Make the board as compact as is reasonable because most 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 like TO-220 chips 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 or near the corners if possible.
6. Make sure the component pin numbers are correct for the component pattern because DT follows these numbers for routing trace connections.
7. **Board Outline:** When the layout looks about right 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. From Place Board Outline you can also specify dimensions for a rectangular starting board.
8. **Dimensioning:** 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 vertical dimension line. Now when you drag a board corner to move it, the dimension arrow automatically moves and the dimension corrects itself. How neat is that!
9. **Routing:** When the layout looks OK; 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. Often dragging a trace segment around a little can eliminate a via, or allow moving a trace segment to the other side of the board. Highlight the segment, right click, click Trace Layer, and select either Top or Bottom. Then to delete a needless via, highlight it, right click, Delete. You can also drag a via to another location. Rerun compare to schematic again.
  10. **Trace Widths:** DipTrace defaults to 13 mil (.33 mm) wide traces. (1 mil = .0254 mm) This is fine for low power boards up to 500ma. Increase default trace sizes for load carrying traces (up to 1 amp) up to .50 mm 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.
  11. 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. To add board text click on “ABC”, then open the box to the right, and select Top Silk. Now text you enter will be colored green and will be silk-screened on the top surface. You can add component values, project name & rev number, date, connection header pin names, etc.
  12. **Grouping:** Use to move, rotate several items together as one. Drag a rectangle around the items, right click on one of them, and select Group. Feint blue borders show what is grouped. After re-positioning them, right click on any one and select Ungroup. Note that groups can be a part of a larger group, and an item within a group can still be edited. Grouping works in schematics also.
  13. **3D Preview:** From Tools run the 3D Preview and check for any obvious conflicts. You will have to first load the latest 3D file (1.5 GB) from the DipTrace site. The 3D view can be rotated in all 3 axes, so it is great for board design review. You can also print the board top layout in full size and check component fit.
  14. If there are components that are not fully seated in 3D view, fix them this way. Back in board layout, right click the item and select 3D Model. Now rotate the x axis until the part is in full frontal view. Now use the Z Shift to move it into the board; Eg for a TO-220, use  $-.4$ ”; for a TO-92 use  $-.2$ ”. Note that for either you can select Apply To: Similar Name Components, if there is more than one. The changes will be saved in layout, and your 3D view will now show them properly.
  15. **Revisions:** 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 4.B) 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.
  16. To print 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.
  17. **Ordering Boards:** Once all final Verifications are done and the board layout is OK, 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. When done, right click on the new folder and Send it to a compressed zip folder. This is the file you will send out for board fabrication. There should be 14 files total in the zip folder.

18. For only a few boards, or for a few trial boards before placing a large order, try OSH Park (US) or JLCPCB or SEED (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.
19. For a larger run of say more than 20 boards, consider an Asian fabricator such as AllPcb or JLCPCB or SEED. 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. Fab time is typically only 2-3 days, but regular shipping is very slow. They offer expedited shipping at additional costs.
20. JLCPCB offer a fixed price of \$5 for 5 boards, up to 100 x 100mm size. You can fit multiple smaller boards into this size, so as to get more boards for the same price. Simply group the entire board outline, copy it, and paste above or beside the current board. You can do this any number of times as long as the 100mm dimensions are not exceeded. Leave a thin space (1mm) between borders to allow for cutting them apart. And note that each copy will have component ID numbers revved up, a minor inconvenience, but very cost effective.
21. For boards up to about 2" (50mm) 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 panels 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.
22. Usually for smaller boards, populating and soldering is easier done in a strip of 4 to 6 boards separated from the multi-board panel. A good time to allow for board assembly is about 2 minutes average per component, less if you are building several at the same time.