

Derbot Design Information

Please check the site <http://www.embedded-knowhow.co.uk/> for updates to Derbot design information, and any errata.

Printed Circuit Board Layouts

Derbot printed circuit board (pcb) layouts were designed using EasyPC, Version 7 <http://www.numberone.com/index.asp>. They are designed to be two-sided, non plated-through-hole (pth), so that they can be easily made using a simple etching system. This has certain implications for the layout, e.g. all tracks are terminated on the solder side of the integrated circuit (i.c.) pads. If made in this way, vias have to be added in all holes that are not used for components. Component leads that have a track running to both top and bottom sides of the pcb must be soldered on both sides. The exception to this is the battery pack, which is single-sided only.

The layouts are supplied here as jpeg scans. They can be printed out on translucent film, scaled appropriately, in order to make pcbs. Gerber files may be made available to institutions recommending the book, and companies wishing to manufacture the pcb for sale. Please contact the author via the web site.

Further versions of the pcbs are likely to be made available, for example for pth, with a reduced board area. This will reduce cost of pcb material. Please check the web site for any such updates.