

How to order circuit boards designed in Eagle
(from Peter Schury)

If you need a few boards, the best deal is the \$33 each service. If you need one or two boards, type "student" in the in the comment section of the order and it will let you order less than 3 boards. If you need many small boards, they have a second site, www.barebonespcb.com, which charges \$42 + \$0.5/sq.in.

Here is the process in detail:

- 1 Load board design.
- 2 Go to board page (not schematic)
 - a. select file -> run
 - b. choose drillcfg.ulp
 - c. choose inches
- 3 goto Eagle Control Panel
- 4 expand CAM Jobs
- 5 load excellon.cam
 - a. file→open board
 - b. process job
- 6 in CAM Jobs, choose gerb274x.cam
 - a. file→open board
 - b. process job
- 7 create txt file and copy and paste this into the text file
included files are
 - .cmp Component side non-mirrored
 - .sol Solder side mirrored
 - .plc Silk screen non-mirrored
 - .stc Solder stop mask CMP non-mirrored
 - .sts Solder stop mask SOL mirrored
 - .dri,.drl,.drd Drill files
- 8 create a .zip file with the following files:
 - .cmp
 - .drd
 - .dri
 - .drl
 - .plc
 - .sol
 - .stc
 - .sts
 - .txt
- 9 edit part number, layers, soldermask and silkscreen sides, and x and y dimensions
- 10 have files checked at www.freedfm.com
- 11 send zip file to Advanced Circuits for ordering
<http://www.4pcb.com/index.htm>