



David Cussans <phdgc@bristol.ac.uk>

JTAG Mux Gerbers

Magnus Loutit <Magnus.Loutit@bristol.ac.uk>
To: David Cussans <David.Cussans@bristol.ac.uk>

7 June 2017 at 16:40

Gerbers are attached. All in Gerber_RS274X format except drill file which is Exellon


- .gto Top Layer Silkscreen
- .bto Bottom Layer Silkscreen
- .gtl Top Copper
- .g1 Inner Copper 1 (GND plane)
- .g2 Inner Copper 2 (Vcc Plane)
- .gbl Bottom Copper
- .gm1 Mechanical
- .gts Top Solder
- .gbs Bottom Solder
- .gdd Drills


.mnt text file containing location data for SMD parts for pick & place & reflow soldering. I'm assuming that given the small number of boards we are making, we can do through-hole components manually rather than have them machine stuffed and wave-soldered. If that's not the case I can extract data for TH parts as well.

Also attached is the bill of materials

Magnus.

2 attachments

 **EL1034v3 BoM.xls**
38K

 **EL1034v3.zip**
195K