

FTM Interface Card Details
Jeff Andresen 6/11/2002

This document has design information for the FTM Interface card to be fabricated by Metro Circuits quote number 0205167JB and Fermilab PRN 21817. The order is for 20 tested JPC Interface cards to be delivered to Fermilab with a 2 week delivery time. This is a 4 trace layer FR4 printed circuit board. The traces can be ½ oz. or 1 oz. copper with a 5 mil minimum width and a 5 mil minimum clearance. The finished board thickness should be .062" + or - what it takes to get the correct controlled impedance.

This is a controlled differential impedance design. The differential impedance is to be 75 ohms for the 5 mil differential pair side by side traces on the inner layers to the GND plane on the outer layers.

According to our calculations, a 4 mil dielectric between the top layer and inner 1, a 52 mil dielectric thickness between inner 1 and inner 2, and a 4 mil dielectric thickness between inner 2 and the bottom layer will provide a differential impedance of 75 ohms with 1 oz. copper and 80 ohms with ½ oz. copper. Please confirm this as your dielectric constant may be different.

The following is the board stackup with the Gerber file extension.

Layer 1, top trace layer, trace layer 1 + GND pour	*.TOP	-----
Layer 2, inner 1, trace layer 2,	*.IN1	-----
Layer 3, inner 2, trace layer 3	*.IN2	-----
Layer 4, bot trace layer, trace layer 4 + GND pour	*.SMB	-----

There are the following additional files.

- *.SST Top silkscreen layer
- *.TAP Drill tape files
- *.ASC Netlist file from the schematic.
- COMPS.TXT Components on board
- CONN.TXT connections on board
- *.DRD Drill drawing Gerber file
- *.DTS Drill tape summary
- *.LIS post processing report including apertures
- *.GTD GERBTOOL design file.

Please contact me if there are any technical questions and deliver the PCBs to me.

Jeff Andresen 630-840-3178

Fermilab, Kirk and Wilson Sts., FCC368, Batavia, IL 60510.