

Printed Circuit Board Check-list prior to manufacture

LIGO-E060231-03-C

R. Abbott, Caltech

29 September, 2006

1. Overview

A check-list for PCB manufacturing has been prepared and may serve as a proposed format for other such lists.

2. Check-list for LIGO PCB, document number _____

Item	Description	Check
1	Does design adhere to T060123, Standard LIGO Electrical Interfaces	
2	Labeling should be included on the silkscreen to ease identification of all inputs and outputs using clear and meaningful text. Avoid unusual abbreviations. Inputs and outputs should be identified.	
3	If text has been added, ensure the text matches the net function. Don't accidentally label "+15" as "-15"	
4	Ensure the board has been re-annotated if it's needed (not for multi-channel designs).	
5	Full design rule check done within the PCB CAD software. This should include a review of the Gerber data using the integrated Gerber viewer's analysis capacity	
6	Ensure the DCC number, revision and serial number block are present	
7	Ensure the design has been thoroughly evaluated for thermal management and the inclusion of heat-sinking if needed	
8	Verify all polarized components (diodes, capacitors etc.) are properly oriented, and clearly indicated on the silkscreen	
9	Connectors must have some form of keying to prevent reversal of connections	
10	Verify that sufficient test-points exist to facilitate testing, including power and ground	
11	Verify that there are power indicator LEDs present in the design	
12	Check that the drill sizes correspond to known acceptable manufacturer sizes and that they are sufficiently large to allow removal of parts. Be careful of threaded screws passing through tight drill holes, shorts can be caused by the threads cutting into internal planes	
13	Verify no floating inputs for op-amps and digital switches etc.	
14	Ensure that adequate mechanical supports exist for the boards at the front, rear and middle if needed. Don't use adhesive based mechanical supports as they fall off over time.	
15	Ensure adequate pull back for internal planes especially for high voltages and in cases where the board must slide into metal tracks. Avoid routing high voltage traces under metal packaged parts to avoid potential shorts	
16	Ensure the design minimizes the use of "through-hole" parts to avoid higher stuffing costs and difficulty in repair or modification	