NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

1) PCB IS 12-LAYER, .062" THICK.
2) CONSTRUCTION IS SOLDER-MASK-OVER-BARE-COPPER (SMOBC).
3) ACCEPTABILITY SHALL BE BASED ON IPC-A-600, CLASS 2.
4) THE FOLLOWING GERBER RS274X PHOTO TOOL FILES SHALL BE USED TO DEFINE ALL CIRCUIT FEATURES:
   - *.GTL – TOP LAYER GERBER DATA
   - *.GP1 – INTERNAL PLANE LAYER 1 GERBER DATA
   - *.G1 – MID LAYER 1 GERBER DATA
   - *.GP2 – INTERNAL PLANE LAYER 2 GERBER DATA
   - *.GP3 – INTERNAL PLANE LAYER 3 GERBER DATA
   - *.GP4– INTERNAL PLANE LAYER 4 GERBER DATA
   - *.GP5– INTERNAL PLANE LAYER 5 GERBER DATA
   - *.GP6– INTERNAL PLANE LAYER 6 GERBER DATA
   - *.GP7– INTERNAL PLANE LAYER 7 GERBER DATA
   - *.GP8– INTERNAL PLANE LAYER 8 GERBER DATA
   - *.GP9– INTERNAL PLANE LAYER 9 GERBER DATA
   - *.GBL – BOTTOM LAYER GERBER DATA
   - *.GTO – TOP OVERLAY GERBER DATA
   - *GBO –- BOTTOM OVERLAY GERBER DATA
   - *.GTP – TOP-SIDE SOLDER PASTE MASK

   - *.GTS – TOP SOLDER MASK GERBER DATA
   - *.GBS – BOTTOM SOLDER MASK GERBER DATA
5) THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGINEERING APPROVAL.
   PCB DESIGNER: RICH LOBDILL PH (805) 880-1621 FAX (805) 961-1792.

FABRICATION TOLERANCES

6) END PRODUCT CONDUCTOR WIDTHS AND PAD DIAMETERS SHALL NOT VARY MORE THAN 0.002" FROM THE 1:1 DIMENSIONS OF THE MASTER ARTWORK.
7) THE CONDUCTIVE PATTERN SHALL BE POSITIONED SO THAT THE LOCATION OF ANY PAD OR LAND SHALL BE WITHIN 0.005" DIAMETER TO THE TRUE POSITION OF THE HOLE IT CIRCUMSCRIBES.

8.1) DRILL TOLERANCES +/- 0.003"
8.2) ALL DRILL HOLE SIZES AND TOLERANCES APPLY AFTER PLATING.
9) THE MINIMUM ANNULAR RING SHALL BE 0.005".
10) BOW AND TWIST SHALL NOT EXCEED 0.010" PER INCH.
11) FOR PCB ROUTING DIMENSIONS: .XXX = +/-.005" .XX = +/-.020"

MATERIAL

12) BASE MATERIAL IS FR4 EPOXY FIBERGLASS
13) SEE STACK-UP LEGEND FOR COPPER CLADDING CALL OUTS

PLATING

14) ALL HOLES AND CONDUCTIVE SURFACES SHALL BE PLATED WITH A MINIMUM OF 0.001" COPPER.
15) AFTER SOLDERMASK, ALL EXPOSED HOLES AND CONDUCTIVE SURFACES SHALL BE COATED WITH A GOLD IMMERSION PLATING TO PRESERVE SOLDERABILITY.

COATINGS

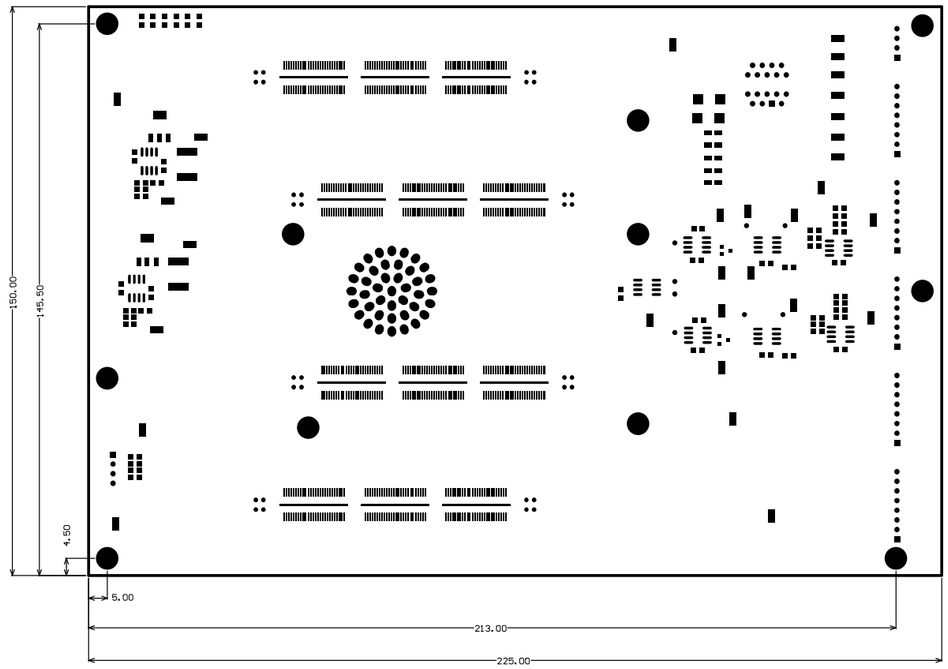
16) THE SOLDERMASK SHALL BE BLACK LIQUID PHOTO-IMAGEABLE PER IPC-SM-840, TYPE-B, CLASS 2.
17) THE SOLDERMASK REGISTRATION ALLOWANCE IS 0.003". THERE SHALL BE NO SOLDERMASK ON ANY SOLDER PAD OR LAND.

MARKING

18) THE LEGEND SHALL BE SCREEN-PRINTED USING PERMANENT YELLOW EPOXY INK.
19) THE SCREEN PRINTING REGISTRATION ALLOWANCE IS 0.007". THERE SHALL BE NO INK ON ANY SOLDER PAD OR LAND.
20) THE VENDOR CODE AND UL FLAMMABILITY RATING MAY BE ETCHED IN THE FOIL OR MARKED IN PERMANENT EPOXY INK (VENDOR'S OPTION).

ELECTRICAL TESTING

21) ALL BOARDS SHALL BE ELECTRICALLY TESTED TO THE SUPPLIED IPC-D-356A NET LIST FOR CONTINUITY, OPENS AND SHORTS.



Layer Stack Up Detail for: 175-00014_Rev1, PCB, Sinistro Backplane.PcbDoc

Layer Name	Copper cladding
Top Layer (*.GTL)	1/2 oz. (1 oz. Finished)
AGND (*.GP1)	1/2 oz
Mid-Layer 1 (*.G1)	1/2 oz
GND (*.GP2)	1/2 oz
+12V (*.GP3)	1/2 oz
-12V (*.GP4)	1/2 oz
+5ANA (*.GP5)	1/2 oz
-5ANA (*.GP6)	1/2 oz
+28V (*.GP7)	1/2 oz
VCC (*.GP8)	1/2 oz
+24V (*.GP9)	1/2 oz
Bottom Layer (*.GBL)	1/2 oz. (1 oz. Finished)

PRIMARY PCB SPECIFICATIONS

(REFER TO COMPLETE SPEC LISTING AT LEFT FOR FURTHER DETAILS)

NUMBER OF LAYERS	–	12
FINISHED THICKNESS	–	.062"
BASE MATERIAL	–	FR4
PLATING TYPE	–	GOLD IMMERSION
SOLDER MASK COLOR	–	BLACK

Las Cumbres Observatory
Global Telescope Network

Las Cumbres Observatory, Inc.
6740 Cortona Dr.
Goleta, CA 93117
www.lcogt.net

DATE	DESIGNED	DRAWN	SCALE
6/28/2011	Rich Lobdill	Rich Lobdill	1 : 1

CHECKED,DATE | APPROVED,DATE

TITLE: 175-00014, Sinistro Backplane

SIZE	DOC NO.	REV	SHEET
C	– GPT	2	1 OF 20