

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 LOBDELL 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

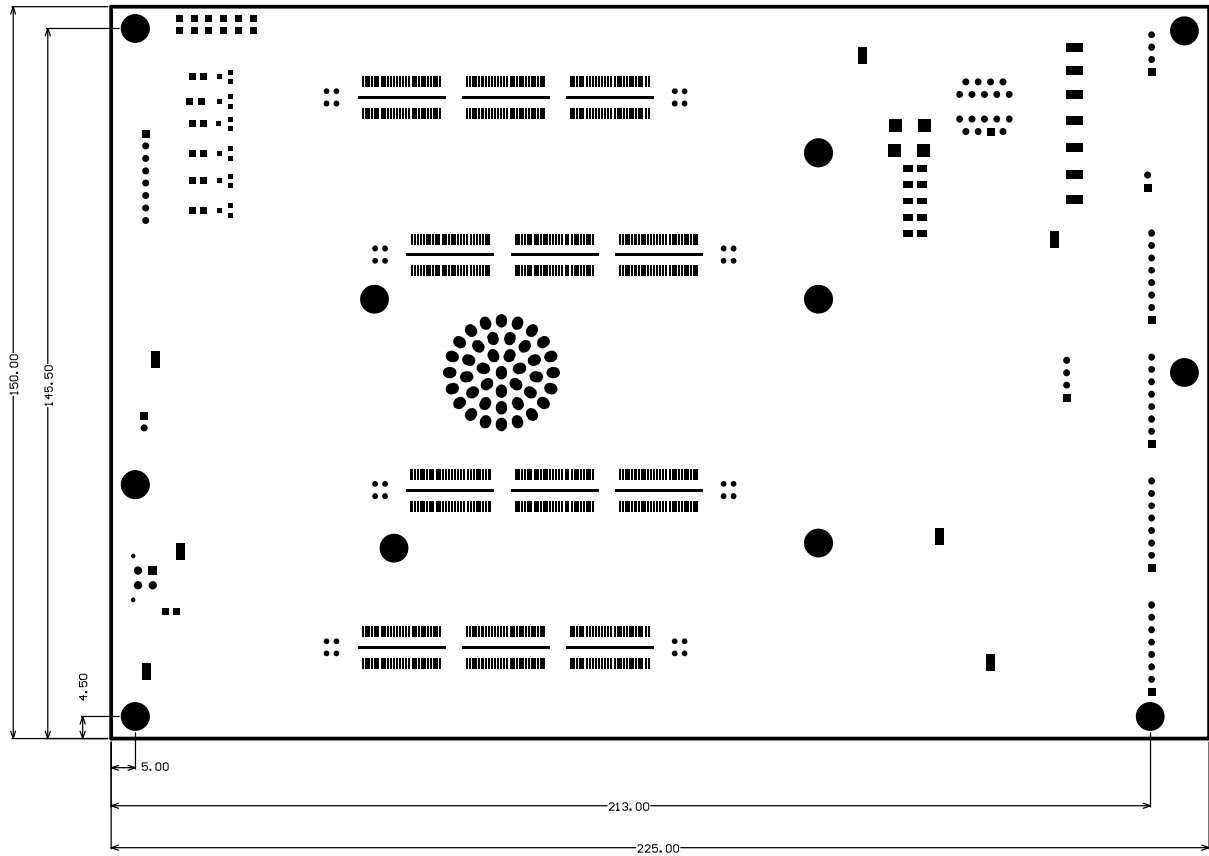
- 16) THE SOLDERMASK SHALL BE BLACK LIQUID PHOTO-IMAGEABLE PER IPC-SM-840, TYPE-B, CLASS 2.
- 17) THE SOLDERMASK REGISTRATION ALLOWANCE IS 0.003". THERE SHALL BE NO SOLDERMASK ON ANY SOLDER PAD OR LAND.

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
+12U (*.GP3)	1/2 oz
-12U (*.GP4)	1/2 oz
+5ANA (*.GP5)	1/2 oz
-5ANA (*.GP6)	1/2 oz
+28U (*.GP7)	1/2 oz
VCC (*.GP8)	1/2 oz
+24U (*.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 10/10/2012	DESIGNED Rich Lobdell	DRAWN Rich Lobdell	SCALE 1 : 1
CHECKED/DATE		APPROVED/DATE	
TITLE 175-00014, Sinistro Backplane			
REV C	DOC NO. — GPT	REV 3	SHEET 1 OF 20

NOTICE
THIS DRAWING EMBODIES A PROPRIETARY DESIGN OWNED BY LAS CUMBRES OBSERVATORY. IT IS SUBMITTED FOR A SPECIFIC PURPOSE UNDER A CONFIDENTIAL RELATIONSHIP, AND EXCEPT FOR PURPOSES EXPRESSLY GRANTED IN WRITING, ALL RIGHTS ARE RESERVED BY LAS CUMBRES OBSERVATORY.