

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- 1) PCB IS 6-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
 - *G1 - MID LAYER 1 GERBER DATA

- *GP1 - INTERNAL PLANE LAYER 1 GERBER DATA
- *GP2 - INTERNAL PLANE LAYER 2 GERBER DATA
- *GP3- INTERNAL PLANE LAYER 3 GERBER DATA

- *GBL - BOTTOM LAYER GERBER DATA
- *G10 - TOP OVERLAY GERBER DATA
- *G1S - TOP SOLDER MASK GERBER DATA
- *G1P - TOP-SIDE SOLDER PASTE MASK
- *G20 - BOTTOM OVERLAY GERBER DATA
- *G2S - BOTTOM SOLDER MASK GERBER DATA
- *G2P - BOTTOM-SIDE SOLDER PASTE MASK

- 5) THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGINEERING APPROVAL.
- PCB DESIGNER: RICH LOBBILL 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) 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.
- 16) COPPER THICKING ON LAYERS AS NEEDED

COATINGS

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

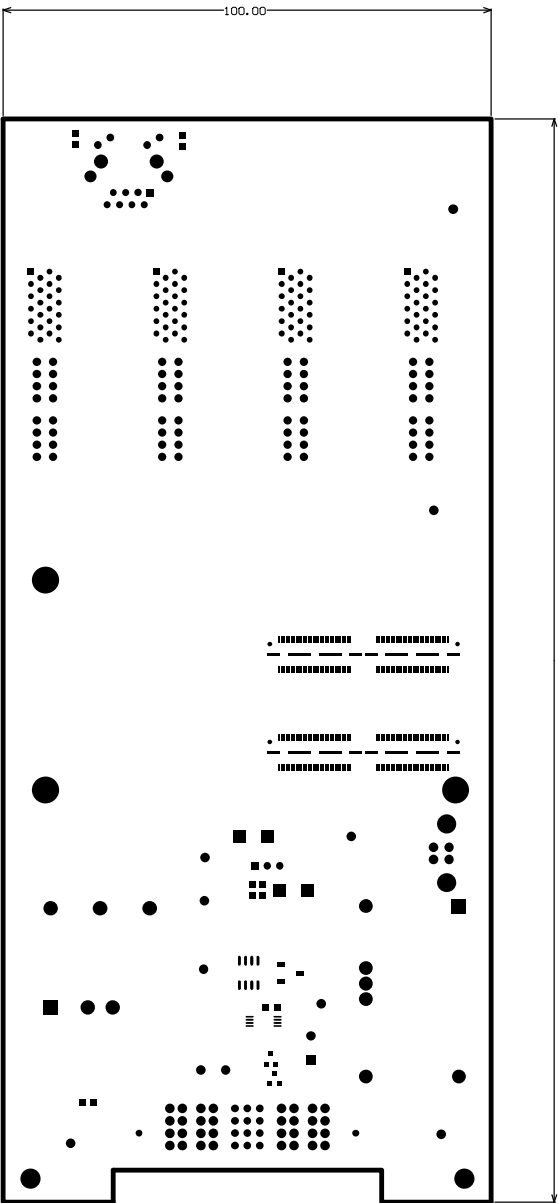
MARKING

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

ELECTRICAL TESTING

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

Layer Stack Up Detail for: 175-00061, rev1, PCB, NRES Cr ate Slot MotherBoard.PcbDoc		
Layer Name	COPPER THICKNESS	
Top Layer (*.GTL)	1/2 oz,	1 oz Finished
h1.d1ayer1 (*.G1)	1 oz	
InternalPlane1 (*.GP1)	1 oz	
InternalPlane2 (*.GP2)	1 oz	
InternalPlane3 (*.GP3)	1 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 - 6
- FINISHED THICKNESS - .062"
- BASE MATERIAL - FR4
- PLATING TYPE - GOLD IMMERSION
- SOLDER MASK COLOR - RED

DATE		09/26/2012		DESIGNER		Rich Lobell		DRAWN		Rich Lobell		SCALE		1 : 1	
C		175-00061, NRES, CRATE MOTHERBOARD		C		GPT		1		1		1		X	

Las Cumbres Observatory
Galileo Telescope Network

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