

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- PCB IS 6-LAYER .062" THICK.
- CONSTRUCTION IS SOLDER-MASK-OVER-BARE-COPPER (SMOBC).
- ACCEPTABILITY SHALL BE BASED ON IPC-A-600, CLASS 2.
- 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
- \*GTO – TOP OVERLAY GERBER DATA
- \*GTS – TOP SOLDER MASK GERBER DATA
- \*GTP – TOP-SIDE SOLDER PASTE MASK
- \*GBO – BOTTOM OVERLAY GERBER DATA
- \*GBS – BOTTOM SOLDER MASK GERBER DATA
- \*GBP – BOTTOM-SIDE SOLDER PASTE MASK

- THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGINEERING APPROVAL.  
PCB DESIGNER: RICH LOBOLL PH (805) 880-1621 FAX (805) 961-1792.

FABRICATION TOLERANCES

- END PRODUCT CONDUCTOR WIDTHS AND PAD DIAMETERS SHALL NOT VARY MORE THAN 0.002" FROM THE 1:1 DIMENSIONS OF THE MASTER ARTWORK.
- 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 IF CIRCULARS/RECTS.
- ALL DRILL HOLE SIZES AND TOLERANCES APPLY AFTER PLATING.
- THE MINIMUM ANNULAR RING SHALL BE 0.005".
- BOW AND TWIST SHALL NOT EXCEED 0.010" PER INCH.
- FOR PCB ROUTING DIMENSIONS: XXX = +/--.005" XX = +/--.020"

MATERIAL

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

PLATING

- ALL HOLES AND CONDUCTIVE SURFACES SHALL BE PLATED WITH A MINIMUM OF 0.001" COPPER.
- AFTER SOLDERMASK, ALL EXPOSED HOLES AND CONDUCTIVE SURFACES SHALL BE COATED WITH A GOLD IMMERSION PLATING TO PRESERVE SOLDERABILITY.
- COPPER THICKING ON LAYERS AS NEEDED

COATINGS

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

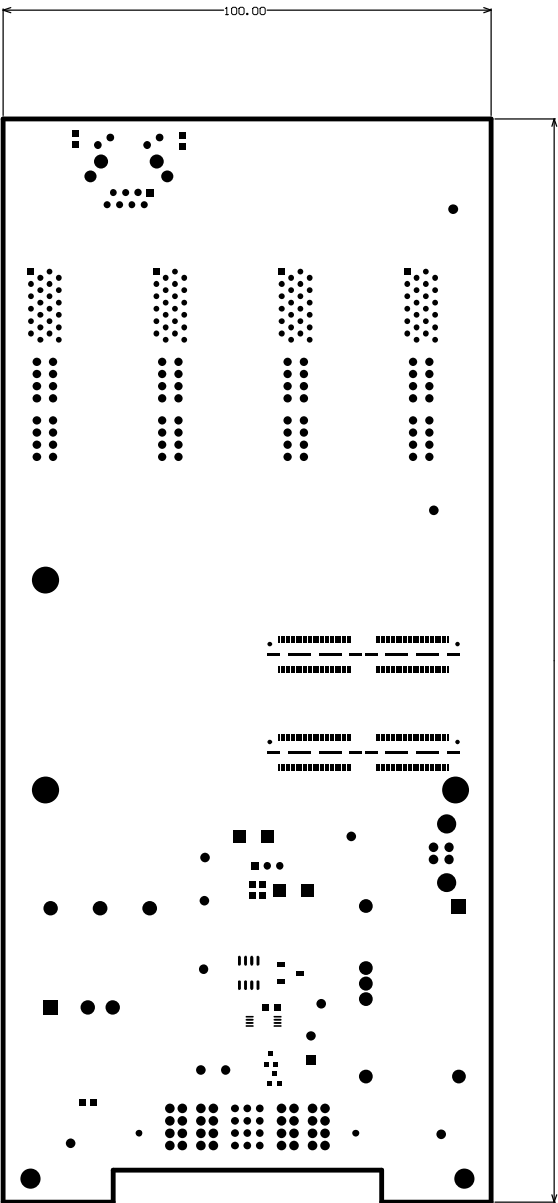
MARKING

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

ELECTRICAL TESTING

- 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, rev2, PCB, NRES Crate Slot MotherBoard.PcbDoc		
Layer Name	COPPER THICKNESS	
Top Layer (*.GTL)	1/2 oz,	1 oz Finished
MidLayer1 (*.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

<b>Las Cumbres Observatory</b> Global Telescope Network		<b>Las Cumbres Observatory, Inc.</b> 6740 Cortona Dr. Goleta, CA 93117 www.lcoginet	
DATE 09/26/2012	DESIGNED BY Rich Lobell	DRAWN BY Rich Lobell	SCALE 1 : 1
TITLE 175-00061, NRES, CRATE MOTHERBOARD			
REV C	REV NO.	REV - GPT	REV 2
		REV 1	REV OF X