

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- 1) PCB IS 4-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
*GP2 - INTERNAL PLANE LAYER 2 GERBER DATA

*GBL - BOTTOM LAYER GERBER DATA
*GTO - TOP OVERLAY GERBER DATA
*GTS - TOP SOLDER MASK GERBER DATA
*GTP - TOP-SIDE SOLDER PASTE MASK

*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) 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.

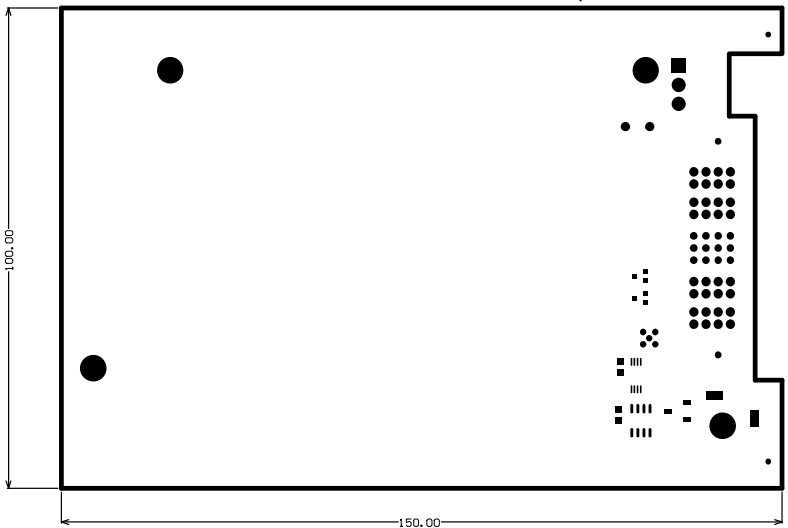
MARKING

- 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.

NOTE: D-shaped board cutouts (9 pics)



Layer Stack Up Detail for: 175-00033, rev1, Crate Slot Power Interface.PcbDoc

Layer Name	COPPER THICKNESS
Top Layer (*GTL)	1/2 oz, 1 oz Finished
Internal Plane 1 (*GP1)	2 oz
Internal Plane 2 (*GP2)	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	-	4
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 9/1/2011	DESIGNED Rich Lobdill	DRAWN Rich Lobdill	SCALE 1 : 1
CHECKED/DATE		APPROVED/DATE	
FILE 175-00033, Crate Slot Power Interface			
REV C	DOC NO. - GPT	REV 1	SHEET 1 of X