

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- 1) PCB IS 7--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
*GP4 -- INTERNAL PLANE LAYER 4 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

- 5) THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGINEERING APPROVAL.
PCB DESIGNER: RICH LOBELL 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 THEIVING ON LAYERS AS NEEDED
16a) NOTE EDGE PLATING CALLED OUT ON DRAWING 2 PLCS

COATINGS

- 17) THE SOLDERMASK SHALL BE BLACK LIQUID PHOTO--MAGEABLE PER IPC--SM--840, TYPE--B, CLASS 2.
18) THE SOLDERMASK REGISTRATION ALLOWANCE IS 0.003".

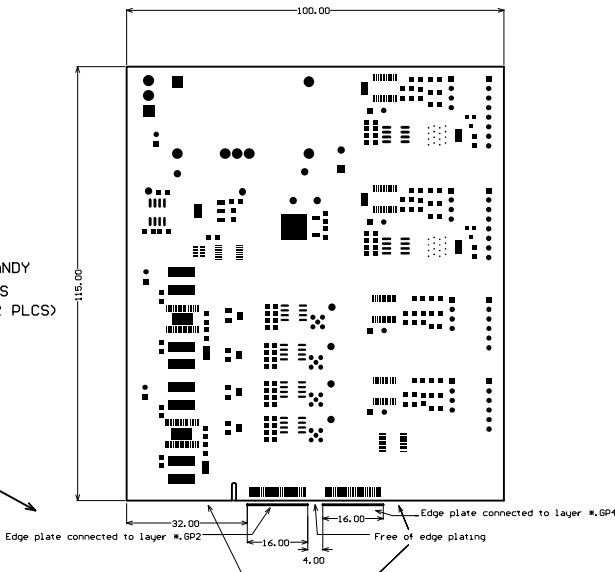
MARKING

- 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 (VENDOR'S OPTION).

ELECTRICAL TESTING

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

BRING TO ATTN OF ALEX CHANDY
NOTE: SPECIAL INSTRUCTIONS
FOR EDGE PLATING (2 PLCS)



Layer Stack Up Detail for: 175-00052, rev1, Temp Controller Dev Board.PcbDoc

Layer Name	COPPER THICKNESS
Top Layer (*.GTL)	1/2 oz, 1 oz Finished
Multi-Net(*.GP1)	1 oz
Mid-Layer (*.G1)	1 oz
AGND (*.GP2)	1 oz
Multi-Net (*.GP3)	1 oz
DGND (*.GP4)	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 -- 7
FINISHED THICKNESS -- .062"
BASE MATERIAL -- FR4
PLATING TYPE -- GOLD IMMERSION
SOLDER MASK COLOR -- BLACK

THIS DRAWING DERIVES A PROPRIETARY DESIGN OWNED BY LAS CUMBRES OBSERVATORY. IT IS SUBMITTED FOR A SPECIFIC PURPOSE UNDER A CONFIDENTIALITY AGREEMENT AND IS NOT TO BE REPRODUCED OR COPIED IN ANY MANNER WITHOUT THE WRITTEN PERMISSION OF LAS CUMBRES OBSERVATORY.

Las Cumbres Observatory Global Telescope Network		Las Cumbres Observatory, Inc. 6740 Cortona Dr. Goleta, CA 93117 www.lcogt.net	
DATE 9/17/2012	DESIGNER Rich Lobell	DRAWN Rich Lobell	SCALE 1 : 1
DESCRIPTION		PROJECT NAME	
TITLE 175-00052, Temp Control Dev Board			
REV C	REV BY	REV - GPT	REV 1
		SHEET 1 of X	