

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- 1) PCB IS 5-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

*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

- 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.
- 16) COPPER THEIVING ON LAYERS AS NEEDED

COATINGS

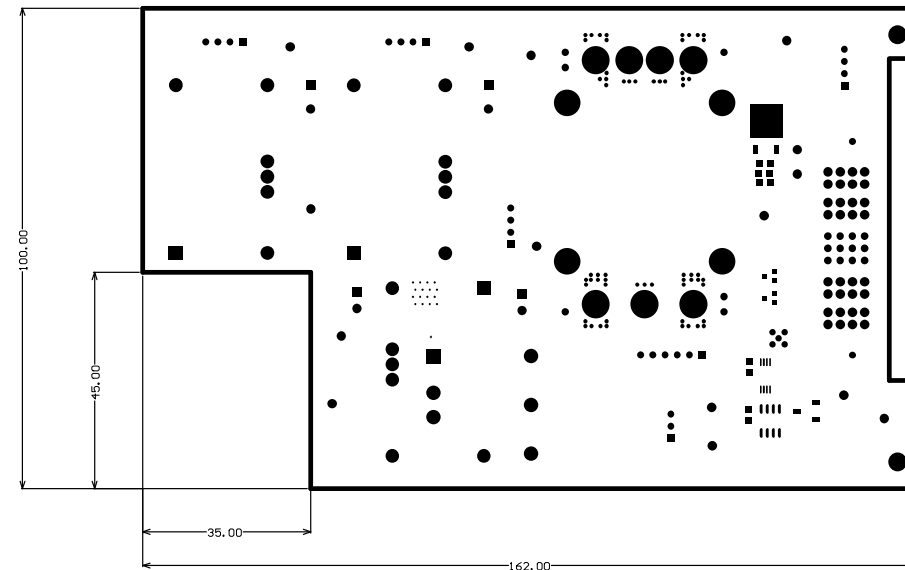
- 17) THE SOLDERMASK SHALL BE BLACK LIQUID PHOTO-IMAGEABLE PER IPC-SM-840, TYPE-B, CLASS 2.
- 18) THE SOLDERMASK REGISTRATION ALLOWANCE IS 0.003". THERE SHALL BE NO SOLDERMASK ON ANY SOLDER PAD OR LAND.

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 FOIL 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.



Layer Stack Up Detail for: 175-00035 rev2, Flex Pur Sply.PcbDoc

Layer Name	COPPER THICKNESS
Top Layer (*GTL)	3 oz
Internal Plane 1 (*GP1)	3 oz
Internal Plane 2 (*GP2)	3 oz
Internal Plane 3 (*GP3)	3 oz
Bottom Layer (*GBL)	3 oz

NOTE: COPPER THICKNESS: 3 oz ALL LAYERS

PRIMARY PCB SPECIFICATIONS

(REFER TO COMPLETE SPEC LISTING AT LEFT FOR FURTHER DETAILS)

NUMBER OF LAYERS	—	5
FINISHED THICKNESS	—	.062"
BASE MATERIAL	—	FR4
PLATING TYPE	—	GOLD IMMERSION
SOLDER MASK COLOR	—	BLACK

Las Cumbres Observatory
Global Telescope NetworkLas Cumbres Observatory, Inc.
6740 Cortona Dr.
Goleta, CA 93117
www.lcogt.net

DATE: 2/29/2012 DESIGNED: Rich Lobdill DRAWN: Rich Lobdill SCALE: 1 : 1

CHECKED DATE: APPROVED DATE:

FILE: 175-00035, FLEX POWER SUPPLY

REV: C DOC NO: - GPT REV: 2 SHEET: 1 of X