

## NOTES (UNLESS OTHERWISE SPECIFIED):

## GENERAL

- 1) PCB IS 5-LAYER, .055" 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  
\*G2 - MID LAYER 2 GERBER DATA  
\*GP1 - INTERNAL PLANE LAYER 1 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 LOBOLL 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 EXCEPT 6mil VIA HOLES
- 9) BREAKOUT OK ON 6mil VIAS (6mil VIAs OK'd by Alex Chandy per telecon 11/8/2011)
- 10) BOW AND TWIST SHALL NOT EXCEED 0.010" PER INCH.
- 11) FOR PCB ROUTING DIMENSIONS: .JXX = +/- .005" .JX = +/- .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

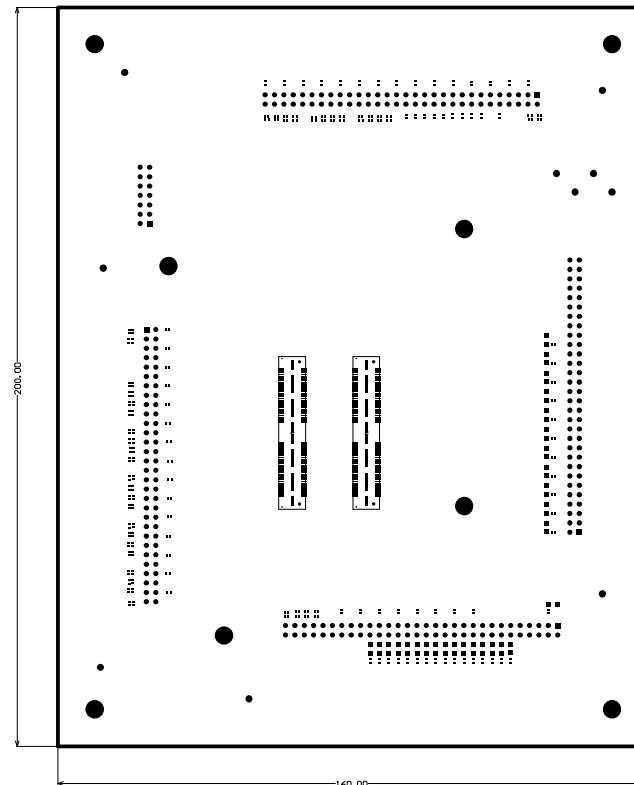
- 17) THE SOLDERMASK SHALL BE BLACK LIQUID PHOTO-IMAGEABLE PER IPC-SM-840, TYPE-B, CLASS 2.
- 18) THE SOLDERMASK REGISTRATION ALLOWANCE IS 0.003". THERE SHALL BE NO SOLDERMASK ON ANY SOLDER PAD OR LAND.

## 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: 175-00036, rev1, Super Sniffer.PcbDoc

Layer Name	COPPER THICKNESS
Top Layer (*.GTL)	1/2 oz, 1 oz Finished
Mid-Layer 1 (*.G1)	1/2 oz
Internal Plane 1 (*.GP1)	1/2 oz
Mid-Layer 2 (*.G2)	1/2 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	5
FINISHED THICKNESS	.055"
BASE MATERIAL	FR4
PLATING TYPE	GOLD IMMERSION
SOLDER MASK COLOR	BLACK

<b>Las Cumbres Observatory</b> Global Telescope Network		Las Cumbres Observatory, Inc. 6740 Cortona Dr. Goleta, CA 93117 www.lcogt.net	
DATE 11/11/2011	DRAWN Rich Loboll	DATE 11/11/2011	SCALE 1 : 1
DESIGNED BY 175-00036, SUPER SNIFFER BOARD		APPROVED BY	
REV C	REV 1	REV 1	OF X