

## NOTES (UNLESS OTHERWISE SPECIFIED):

## GENERAL

- 1) PCB IS x-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  
 \*.GP5 - INTERNAL PLANE LAYER 5 GERBER DATA  
 \*.GBL - BOTTOM LAYER GERBER DATA  
 \*.G10 - TOP OVERLAY GERBER DATA  
 \*.G15 - TOP SOLDER MASK GERBER DATA  
 \*.G1P - 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 LOBOILL 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 ANNUAL RING SHALL BE 0.005".
- 10) BOW AND TWIST SHALL NOT EXCEED 0.010" PER INCH.
- 11) FOR PCB ROUTING DIMENSIONS: .JXX = +/-0.005" .JX = +/-0.020"

## 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.

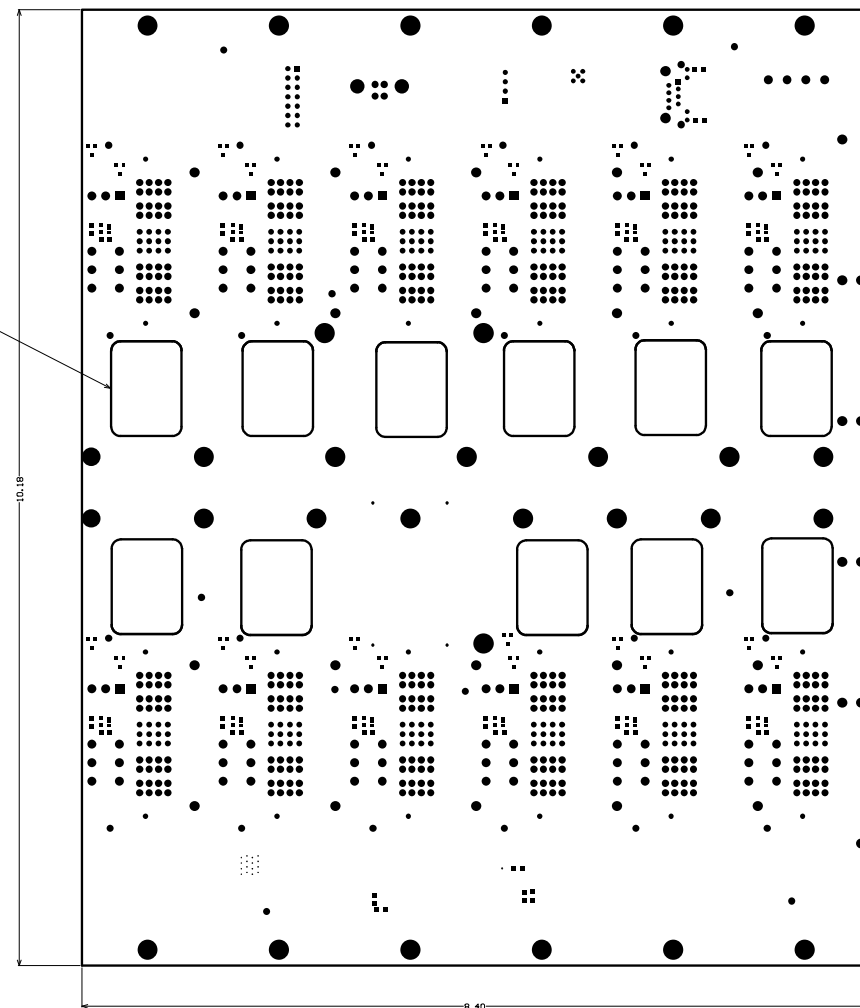
## MARKING

- 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: BOARD CUTOUTS (11 PLCS)



Layer Stack Up Detail for: 175-00024\_rev1, Backplane, Instrument Crate.PcbDoc

Layer Name	COPPER THICKNESS
Top Layer (*.GTL)	1/2 oz, 1 oz Finished
Mid-Layer 1 (*.G1)	1 oz
+24U (*.GP1)	2 oz
RTN (*.GP2)	2 oz
+3.3U (*.GP3)	1 oz
+12U (*.GP4)	1 oz
Signal GND (*.GP5)	1 oz
Bottom Layer (*.GBL)	1/2 oz, 1 oz Finished

NOTE: 2 OZ COPPER FOR CERTAIN LAYERS

## PRIMARY PCB SPECIFICATIONS

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

NUMBER OF LAYERS — 8  
 FINISHED THICKNESS — .062"  
 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 6/2/2011	DRAWN Rich Lobill	CHKD Rich Lobill	SCALE 1 : 1
DESIGNED BY 175-00024, BACKPLANE, INSTRUMENT CRATE		APPROVED BY	
REV C	DESCRIPTION - GPT	REV 1	PAGE 1 OF X