

NOTES (UNLESS OTHERWISE SPECIFIED):

GENERAL

- 1) PCB IS 6-LAYER .062" THICK.
  - 2) NO SOLDERMASK
  - 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 - MD LAYER 1 GERBER DATA
    - \*G2 - MD LAYER 2 GERBER DATA
    - \*GP1- INTERNAL PLANE LAYER 1 GERBER DATA
    - \*GP2 - INTERNAL PLANE LAYER 2 GERBER DATA
    - \*GBL - BOTTOM LAYER GERBER DATA
    - \*GBP - BOTTOM-SIDE SOLDER PASTE MASK
  - 5) THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGINEERING APPROVAL.
- PCB DESIGNER: RICH LOBBILL 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 LAMINATE: POLYIMIDE
- 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) ALL EXPOSED HOLES AND CONDUCTIVE SURFACES SHALL BE COATED WITH A GOLD IMMERSION PLATING TO PRESERVE SOLDERABILITY.

COATINGS

- 16)
- 17)

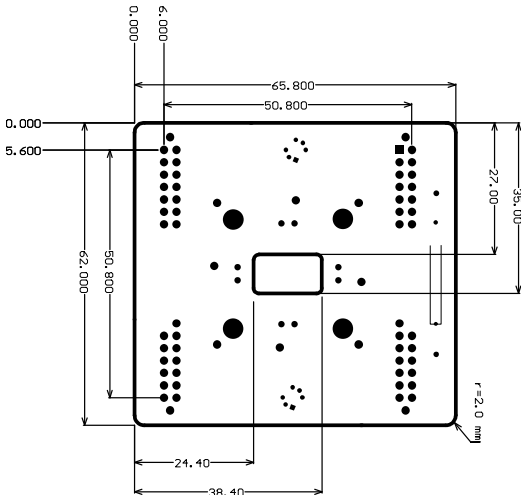
MARKING

- 18) NO NK MARKING IS TO BE USED
- 19) ETCHED MARKING IS ACCEPTABLE
- 20)

ELECTRICAL TESTING

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

Layer Stack Up Detail for: 175-00029 rev3, 486 CCD Interface,PcbDoc		
Layer Name	Copper Cladding	
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	
Internal Plane 2 (*.GP2)	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   | 6              |
| FINISHED THICKNESS | .062"          |
| BASE MATERIAL      | POLYIMIDE      |
| PLATING TYPE       | GOLD IMMERSION |
| SOLDER MASK COLOR  | NONE           |

Las Cumbres Observatory  
Global Telescope Network

DATE

7/16/2013

DESIGNED

Rich Lobell

DRAWN

Rich Lobell

SCALE

1 : 1

Las Cumbres Observatory, Inc.  
6740 Cortona Dr.  
Goleta, CA 93117  
www.lcoginet

CREATED

TITLE 175-00029, PCB, 486 CCD Interface

REV

C

DATE

REV

REV

3

REV

1 of 1

REV

X