

Title				SCHEMATIC, 486 CCD INTERFACE BOARD	
Size		Number		Revision	
B		780-00033		2	
Date:	7/6/2011		Sheet of		
File:	\\.\780-00033 rev 2, 486 CCD Interface Schematic				
Drawn By:					

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 -- MID LAYER 1 GERBER DATA  
\*G2 -- MID 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 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.  
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)

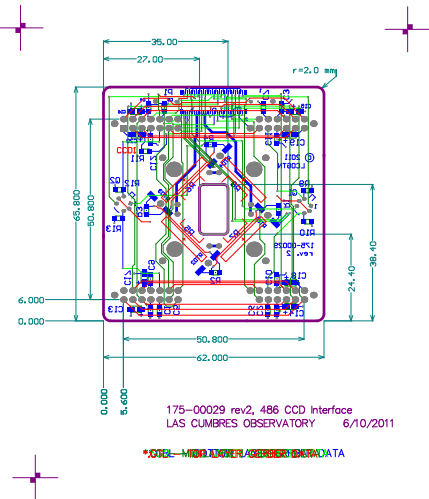
MARKING

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

20)

ELECTRICAL TESTING

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



Layer Stack Up Detail for: 175-00029 rev2, 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

NOTICE  
THIS DRAWING EMBODIES A PROPRIETARY DESIGN OWNED BY LAS CUMBRES OBSERVATORY. IT IS SUBMITTED FOR A SPECIFIC PURPOSE UNDER A CONFIDENTIAL RELATIONSHIP, AND EXCEPT FOR PURPOSES EXPRESSLY GRANTED IN WRITING, ALL RIGHTS ARE RESERVED BY LAS CUMBRES OBSERVATORY.

Las Cumbres Observatory Global Telescope Network		Las Cumbres Observatory, Inc. 6740 Cortona Dr. Goleta, CA 93117 www.lcog.net	
DATE 6/10/2011	DESIGNER Rich Loboll	APPROVED Rich Loboll	SCALE 1 : 1
PROJECT 175-00029, PCB, 486 CCD Interface			
REV C	REV NO. - GPT	REV 2	REV 1 OF X