NOTES (UNLESS OTHERWISE SPECIFIED): GENERAL

- 1) PCB IS 7-LAYER, .062" THICK.
 2) CONSTRUCTION IS SOLDER-MASK-OVER-BARE-COPPER (SMOBC).
 3) ACCEPTABILITY SHALL BE BASED ON PC-A-660, CLASS 2.
 4) THE FOLLOWING GERBER RS274X PHOTO TOOL FILES SHALL BE USED TO DEFINE ALL CRCUIT FEATURES:
- *.GTL TOP LAYER GERBER DATA
 *.G1 MID LAYER 1 GERBER DATA
- *GP1 NTERNAL PLANE LAYER 1 GERBER DATA
 *GP2 NTERNAL PLANE LAYER 2 GERBER DATA
 *GP3 NTERNAL PLANE LAYER 3 GERBER DATA
 *GP4 NTERNAL PLANE LAYER 4 GERBER DATA

- *.GBL BOTTOM LAYER GERBER DATA
 *.GTO TOP OVERLAY GERBER DATA
 *.GTS TOP SOLDER MASK GERBER DATA
 *.GTF TOP-SIDE SOLDER PASTE MASK
 *.GBO BOTTOM OVERLAY GERBER DATA
 *.GBS BOTTOM SOLDER MASK GERBER DATA
 *.GBP BOTTOM-SIDE SOLDER PASTE MASK
- THE PHOTO TOOL SHALL NOT BE COMPENSATED WITHOUT PRIOR ENGNEEPING 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 DMENSONS OF THE MASTER ARTWORK.

 7) THE CONDUCTIVE PATTERS SHALL BE POSITIONED SO THAT THE LOCATION OF ANY PAD OR LAND SHALL BE WITHN 0,005" DMARTER TO THE TRUE POSITION OF THE HOLE IT CRCUMSCRBES.

 8) ALL DRAL HOLE SIZES AND TOLERANCES APPLY AFTER PLATING.

 9) THE MINMUM ANNULAR RING SHALL BE 0,005".

 10) BOW AND TWIST SHALL NOT EXCEED 0,010" PER NCH.

 11) FOR PCB ROUTING DMENSIONS. XXX = +/-,005" XX = +/-,020"

MATERIAL

- 12) BASE MATERIAL IS FR4 EPOXY FBERGLASS
 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 MINIFASION PLATING TO PRESERVE SOLDERABUTY.

 16) COPPER THEVING ON LAYERS AS NEEDED

COATINGS

- MARKING

 19) THE LEGEND SHALL BE SCREEN-PRINTED USING PERMANENT YELLOW EPOXY NK.

 20) THE SCREEN PRINTING REGISTRATION ALLOWANCE IS 0.007". THERE SHALL BE NO NK ON ANY SOLDER PAD OR LAND.

 21) THE VENDOR CODE AND UL FLAMMABUTY RATING MAY BE ETCHED IN THE FOLL OR MARKED IN PERMANENT EPOXY NK (VENDOR'S OPTION). 17) THE SOLDERMASK SHALL BE RED LIQUID PHOTO-MAGEABLE PER PC-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.

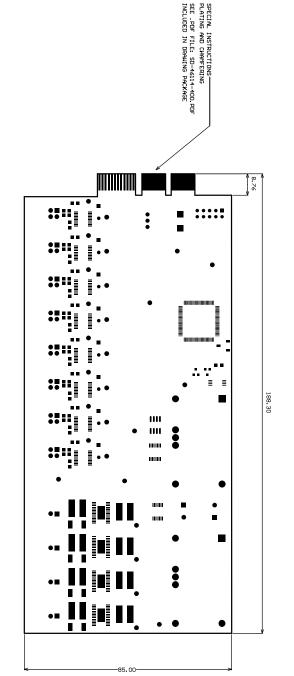
ELECTRICAL TESTING

22) ALL BOARDS SHALL BE ELECTRICALLY TESTED TO THE SUPPLED IPC-D-356A NET UST FOR CONTINUITY, OPBUS AND SHORTS.

Top Layer MidLayer1 (*.G1) InternalPlane2 (*.GP2) InternalPlane1 (*.GP1) Layer Stack Up Detail for: 175-00060, rev1, PCB, NRES RTD_TEC Board.PcbDoc (*.GTL) 1 oz 1 oz 1 oz 1/2 oz, COPPER THICKNESS oz Finished

Bottom Layer (*.GBL) InternalPlane4 (*.GP4) InternalPlane3 (*.GP3) 1 oz 1 oz 1/2 oz, oz Finished

THIS DRAWING EMBODES A PROPRETARY DESIGN OWNED BY LAS CUMBRES OBSERVATORY. IT IS SUBMITED FOR A SPECIFIC PLAPPOSE LINEER A CONFIDENTIAL FREATIVEMP, AND EXCEPT FOR PROPOSES SPRESSLY GRANIED IN WRITNG, ALL RIGHTS ARE RESERVED BY LAS CUMBRES OBSERVATORY.



PRIMARY PCB SPECIFICATIONS
(REFER TO COMPLETE SPEC LISTING AT LEFT FOR FURTHER DETAILS)
NUMBER OF LAYERS — 7
FINISHED THICKNESS — .062"
BASE MATERIAL — FR4 PLATING TYPE SOLDER MASK COLOR $1 \cdot 1 \cdot 1$ FR4 GOLD IMMERSION

