

NOTES (UNLESS OTHERWISE SPECIFIED):

1. FABRICATE PER IPC-A-600, CLASS 2.
2. MATERIAL:  
1 OZ. CU INTERNAL  
1 OZ. CU EXTERNAL, 2 OZ. AFTER PLATING  
BASE LAMINATE FR4
3. TOTAL THICKNESS OF PCB AFTER PLATING  
SHALL BE .062 +/- .005
4. SOLDERMASK BOTH SIDES USING PHOTO  
IMAGEABLE PROCESS. SOLDER MASK OVER  
BARE COPPER. SOLDER MASK SHALL BE  
BETWEEN FINE PITCH PADS. COLOR TRANSPARENT  
GREEN.
5. SILKSCREEN BOTH SIDES OF PCB USING  
NON-CONDUCTIVE EPOXY INK, COLOR  
WHITE. NO INK SHALL BE ON EXPOSED PADS.
6. GERBER ARTWORK PROVIDED. NO ALTERATIONS  
TO GERBER FILES WITHOUT PRIOR CONSENT  
FROM THE CALIFORNIA ASSOC. FOR RESEARCH  
IN ASTRONOMY.
7. ALL DIMENSIONS ARE IN INCHES. TOLERANCES  
ARE AS FOLLOWS:  
.XX: +/- .010  
.XXX: +/- .005

8. ALL HOLES TO BE +/- .003 IN. DIA. UNLESS  
OTHERWISE SPECIFIED. HOLE SIZES GIVEN ARE  
AFTER PLATING. PLATED THRU HOLES SHALL  
HAVE A MINIMUM OF .001 COPPER.
9. IT IS THE FABRICATORS RESPONSIBILITY TO  
SELECT THE BASE MATERIAL TO YIELD THE  
SPECIFIED IMPEDANCE CHARACTERISTICS TO  
WITHIN +/- 10% TOLERANCE.
10. ALL HOLES SHALL BE LOCATED WITHIN .003  
DIA. OF TRUE POSITION.
11. CONDUCTOR WIDTHS AND SPACING SHALL BE  
WITHIN +/- 20% OF PHOTO PLOT ORIGINALS.
12. WARP OR TWIST OF BOARD SHALL NOT  
EXCEED .010 INCH PER INCH
13. REMOVE ALL BURRS AND BREAK SHARP EDGES  
.015 MAX.
14. PLATED THRU HOLES AND EXPOSED PADS SHALL  
BE TIN/LEAD PLATED .0003 TO .0005 THK.

HOLE SCHEDULE		
HOLE DIA.	SYM.	QTY.
0.018	+	270
0.028	X	3
0.038	Y	204
0.046	T	12
0.063	X	89
0.071	X	10
0.124	□	2
0.125	◇	2
0.144	⊠	8

LAYER STRUCTURE (NOT TO SCALE)

1 OZ. CU FOIL, .0014	LAYER 1 TOP
PREPREG 7629, .0075	
1 OZ. CU, .0014	LAYER 2 MID1
0140 CORE, .0140	
*ETCH THIS SIDE, .0000	9. ALL TRACES ON LAYER 2 ARE 90 OHMS. NOM. TRACE WIDTH IS .011. BURIED MICROSTRIP.
PREPREG 7629, .0075	
1 OZ. CU, .0014	LAYER 3 GNDA
0140 CORE, .0140	
1 OZ. CU, .0014	LAYER 4 MID2
PREPREG 7629, .0075	
1 OZ. CU FOIL, .0014	LAYER 5 BOTTOM
	9. ALL TRACES ON LAYER 5 ARE 90 OHMS. NOM. TRACE WIDTH IS .011. MICROSTRIP.

