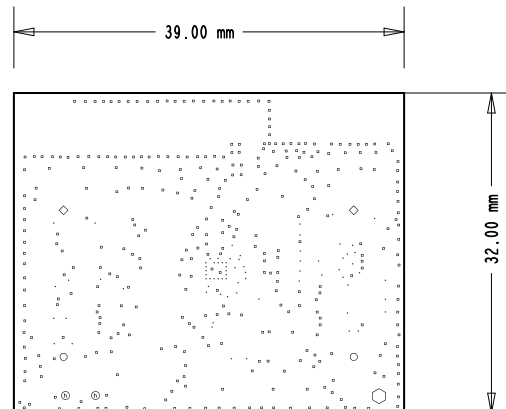
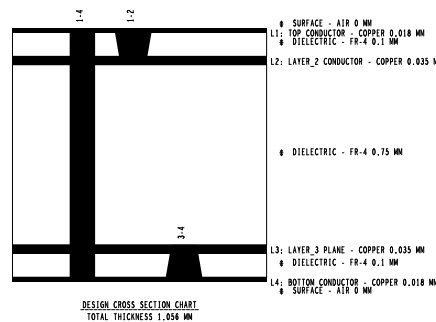






[illegible]

DRILL CHART: TOP to BOTTOM			
ALL UNITS ARE IN MILLIMETERS			
FIGURE	SIZE	PLATED	QT
•	0.2	PLATED	33
	1.5	PLATED	1
	0.7	NON-PLATED	2
	0.9	NON-PLATED	2
	0.9	NON-PLATED	2

DRILL CHART: LAYER_3 to BOTTOM			
ALL UNITS ARE IN MILLIMETERS			
FIGURE	SIZE	PLATED	QTY
-	0.1	PLATED	16

DRILL CHART: TOP to LAYER_2			
ALL UNITS ARE IN MILLIMETERS			
FIGURE	SIZE	PLATED	QTY
-	0.1	PLATED	58

	PREFERRED FABRICATION REQUIREMENTS	
REFER TO THE PANEL DRAWING IF SUPPLIED OTHERWISE USE DETAILS BELOW		
PRINTED CIRCUIT BOARDS THAT REQUIRE PANEALISATION		
01	ANY PCB THAT DOES NOT HAVE A 5mm CLEARANCE FROM PCB EDGE TO COPPER/ COMPONENTS ALONG THE LONGEST PARALLEL EDGES	
02	ANY ODD SHAPE PCB <X ROOM	
PANEL SIZE, WASTE EDGE (BORDER) AND WEAVING		
01	ASSEMBLY PANEL TO BE A 3x4 UPT ARRAY	
02	PANEL BORDER TO BE 10mm ON ALL SIDES FULLY CROSS HATCHED IN COPPER ON BOTH SIDES	
03	BOARD EDGE TO BOARD EDGE INTERNAL WEAVING TO BE 10mm	
04	MAXIMUM PANEL SIZE NOT TO EXCEED 380mm x 440mm	
TOOLING HOLES		
01	ADD 3 TOOLING HOLES 2.5mm +/- 0.05mm DIA IN PANEL BORDER 5mm FROM PANEL EDGE	
FIDUCIALS		
01	ADD 3 FIDUCIALS ON BOTH SIDES (1mm DIA / 2mm DIA CLEARANCE) 5mm FROM PANEL EDGE	
BREAKOUTS (FOR REFERENCE SEE ROUTING DETAIL BELOW)		
01	ADD BREAKOUTS IN SAFE AREAS AWAY FROM TRACKS, TOOLING HOLES, SMD PADS, VIAS, TEST PADS, GROUND PLANES, VITAL SILKSCREEN, THROUGH HOLES, OVERCHARGING COMPONENTS ETC.	
02	USE LOCATIONS SHOWN BY "M" IF PRESENT	
03	AT LEAST TWO BREAKOUTS MUST BE ADDED ALONG THE EDGE OF A CIRCUIT IF GREATER THAN 75MM IN LENGTH	
04	AT LEAST ONE BREAKOUT MUST BE ADDED ALONG THE EDGE OF A CIRCUIT IF LESS THAN 75MM IN LENGTH	
05	THE SPACING BETWEEN BREAKOUTS SHOULD BE BETWEEN 6MM TO 30MM	
06	BREAKOUTS TO BE POSITIONED 12mm FROM CIRCUIT CORNER TO ALLOW CUTOFF ACCESS	
07	STEPPED GERBERS MUST BE SENT TO PCB DESIGNER FOR VERIFICATION BEFORE MANUFACTURE COMMENCES	
08	IF ANY OF THIS SECTION CANNOT BE ADVISED TO THEM PLEASE CONSULT THE PCB DESIGNER	
PREFERRED ROUTING / V-SCORING REQUIREMENTS		
ROUTING		REQUIRED
01	USE A 2.4 MM (+/- 0.1mm) ROUT	
<p>The diagram illustrates a routing setup on a circuit board. It shows a central horizontal track labeled 'CIRCUIT BOARD' with a width of 1.00 MM. To its left is a semi-circular 'BREAK-OFF STRIP' with a width of 1.5MM. The distance from the center of the break-off strip to the start of the main track is 3.00 MM. On the right side of the main track, there are two circular features labeled '(RATBITTE) 0.50MM DRILLED HOLES (3 OFF)'. A dimension line indicates a 'FULL RADIUS (2 POS'NS)' for the curved section of the track leading to these holes.</p>		
V-SCORING		NOT REQUIRED
WHEN V-SCORING IS APPLIED TO THE BOARD AS A METHOD OF REMOVING THE BREAK-OFF STRIPS THE FOLLOWING RULES ARE TO BE APPLIED		
01	SOLDER MASK TO EDGE CLEARANCE (0.5mm (20thau))	
02	COPPER TO EDGE CLEARANCE (ON ALL LAYERS) = 1.00mm (40thau)	
03	SCORE ANGLE = 30 deg	
04	REMAINING WEB AFTER SCORING IS ONE THIRD OF THE PCB NOMINAL THICKNESS	
<p>This diagram shows a cross-section of a PCB being scored at a 30-degree angle. Below the score line, it specifies 'WEB = (1/3 Tpcb)', indicating that one-third of the total thickness remains after scoring.</p>		

NOTES		
01	BOARD TO BE MANUFACTURED TO IPC-A-600 CLASS 2	
02	BOARD IS TO MEET UL94V0 APPROVAL	
03	ALL BOARDS MUST BE 100% TESTED FOR ELECTRICAL CONTINUITY AND ISOLATION	
04	NO REPAIRS MAY BE PERFORMED ON ANY IMPEDANCE CRITICAL TRACKS (WHERE APPLICABLE)	
05	ALIGNMENT OF ALL LAYERS TO BE SUCH THAT NO BREAKOUT OCCURS	
06	SUPPLIER'S IDENTIFICATION MARK, FLAMMABILITY RATING AND DATACODE MUST BE APPLIED TO THE BOARD - BOTTOM SIDE	
07	VARIATION IN TRACK WIDTH AND GAP TO MEET IMPEDANCE REQUIREMENTS ARE NOT COMPROMISED SO LONG AS THERE ARE MINIMAL, AND THE OVERALL BOARD THICKNESS IS NOT COMPROMISED	
08	MATERIAL-BMS COMPLIANT HIGH Tg FR4 (V147, T1678, 370HR, B1755V/1650V OR EQUIVALENT)	
09	BOB AND TIE TO BE NO GREATER THAN 0.15% ACROSS DIAGONALS	
10	NON-FUNCTIONAL PADS MAY BE REMOVED FROM INTERNAL LAYERS (WHERE APPLICABLE)	
11	ALL STUBS MAY BE REMOVED	
12	SILKSCREEN SHOULD NOT BE ALTERED WITHOUT THE PCB DESIGNERS APPROVAL	
13	CHECK BUILD ORDER AGAINST PLOTS	
14	ALL DIMENSIONS IN mm (UNLESS STATED)	
15	FINISHED BOARD THICKNESS	1.56mm +/-10%
16	SURFACE FINISH	ELECTROLESS NICKEL/IMMERSION GOLD
17	SOLDER RESIST	GREEN PHOTO-IMAGABLE
18	SILKSCREEN COLOUR	WHITE
19	MINIMUM TRACK WIDTH	0.09mm
20	MINIMUM GAP	0.09mm
21	MINIMUM P.T.H PAD SIZE	0.09mm
22	MINIMUM PITCH OF SURFACE MOUNT PADS	17mm
23	NO TOP SIDE SURFACE MOUNT PADS	171
24	NO BOTTOM SIDE SURFACE MOUNT PADS	171
ADDITIONAL NOTES		

[illegible]

DRILL DETAIL NOTES	
ALL PLATED HOLES ARE FINISHED SIZES WITH $\pm 0.075\text{mm}$ TOLERANCE	
ALL NON-PLATED HOLES ARE FINISHED SIZES WITH $\pm 0.050\text{mm}$ TOLERANCE	
ALL VIAS ARE DRILLED SIZES WITH $\pm 0/-$ DRILL TOLERANCE	
TOTAL PLATED HOLE QTY	414
TOTAL NON-PLATED HOLE QTY	6
TOTAL PLUGGED VIAS	339



COMPANY CONFIDENTIAL

TOLERANCES UNLESS OTHERWISE STATED

0	PLACE DECIMALS	+/- 1
1	PLACE DECIMALS	+/- 0.5
2	PLACE DECIMALS	+/- 0.1

DRAWN BY KL	DATE 16/10/2015
CHK'D BY ??	DATE ??/??/????



© COPYRIGHT 2014

TITLE	MANUFACTURE DETAIL
	??-??-????

DRAWING NUMBER
224-15-E

