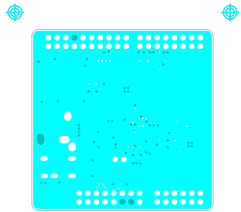
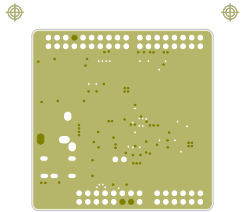


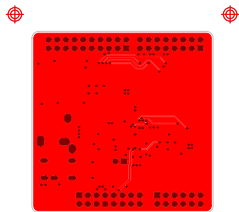
	PROJECT
	ARD-AUDIO-DA7212
DRAWING NUMBER 190-03-B <u>C1</u>	
LAYER	<u>TOP SIDE TRACK</u>



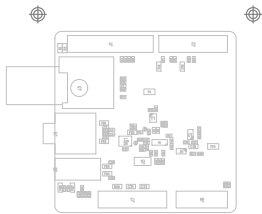
	PROJECT
	ARD-AUDIO-DA7212
	DRAWING NUMBER 190-03-B C2
LAYER	INNER LAYER 2



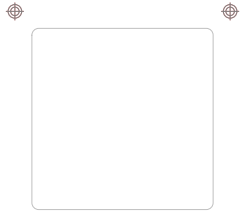
	PROJECT
	ARD-AUDIO-DA7212
	DRAWING NUMBER 190-03-B_C3
LAYER	INNER LAYER 3



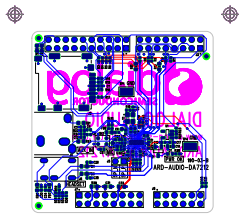
	PROJECT
	ARD-AUDIO-DA7212
DRAWING NUMBER 190-03-B C4	
LAYER	BOTTOM SIDE TRACK



LAYER		PROJECT
		ARD-AUDIO-DA7212
	DRAWING NUMBER	190-03-B C1
	TOP SIDE TRACK DRAWING	

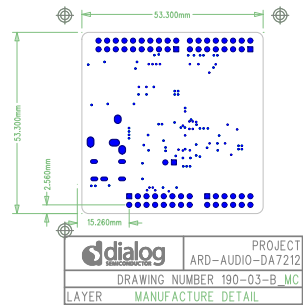


	PROJECT
	ARD-AUDIO-DA7212
	DRAWING NUMBER 190-03-B
LAYER	



	PROJECT
	ARD-AUDIO-DA7212
	DRAWING NUMBER 190-03-B TS
LAYER	TOP SILKSCREEN Y DRAWING

IF IN DOUBT ASK!



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	175
TOTAL NON-PLATED HOLE QTY	0



COMPANY CONFIDENTIAL

TOLERANCES UNLESS OTHERWISE STATED 0 PLACE DECIMALS +/- 1 1 PLACE DECIMALS +/- 0.5 2 PLACE DECIMALS +/- 0.1	DRAWN BY BL	DATE 05/12/13	DIALOG SEMICONDUCTOR UK LTD 2 MULTREES WALK EDINBURGH EH1 3DQ	© COPYRIGHT 2013	
	CHK'D BY WF	DATE 05/12/13		TITLE MANUFACTURE DETAIL	DRAWING NUMBER 190-03-B

PREFERRED PANELISATION REQUIREMENTS	
REFER TO THE PANEL DRAWING IF SUPPLIED OTHERWISE USE DETAILS BELOW	
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 E.g. ROUND
PANEL SIZE, WASTE EDGE (BORDER) AND WEBBING	
01	PREFERRED PANEL SIZE TO BE 300mm X 230mm (MAXIMUM 450mm SQUARE)
02	PANEL BORDER TO BE 10mm ON ALL SIDES, FULLY CROSS HATCHED IN COPPER ON BOTH SIDES
03	BOARD EDGE TO BOARD EDGE INTERNAL WEBBING TO BE 10mm
TOOLING HOLES	
01	ADD 3 TOOLING HOLES 2.5mm $\pm$ 0.05 DIA. TO PANEL BORDER 5mm FROM BORDER EDGE
FIDUCIALS	
01	ADD 3 FIDUCIALS ON BOTH SIDES (1mm DIA./2mm DIA. CLEARANCE) 5mm FROM PANEL EDGE
RATBITES (FOR REFERENCE SEE ROUTING DETAIL BELOW)	
01	ADD RATBITES IN SAFE AREAS AWAY FROM TRACKS, TOOLING HOLES, SM PADS, VIAS, TEST PADS, GROUND PLANES, VITAL SILKSCREEN THROUGH HOLES, OVERHANGING COMPONENTS ETC.
02	USE LOCATIONS SHOWN BY '+' IF PRESENT
03	THERE MUST BE SUFFICIENT RATBITES TO ENSURE PCB STABILITY
04	THE SPACING BETWEEN RATBITES SHOULD BE BETWEEN 40mm TO 50mm
PREFERRED ROUTING / V-SCORING REQUIREMENTS	
ROUTING	REQUIRED
01	USE A 2.40mm ($\pm$ 0.1mm) ROUT
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	GOLDEN MASK TO EDGE CLEARANCE = 0.50mm (20thou)
02	COPPER TO EDGE CLEARANCE (ON ALL LAYERS) = 1.00mm (40thou)
03	SCORE ANGLE = 30°
04	REMAINING WEB AFTER SCORING IS AS ONE THIRD OF THE PCB NOMINAL THICKNESS

[illegible]

LAYER/ LAYER TYPE	BOARD STACK	COPPER WEIGHT
01 MIXED		305g/m
02 GROUND		152g/m
03 GROUND		152g/m
04 MIXED		305g/m