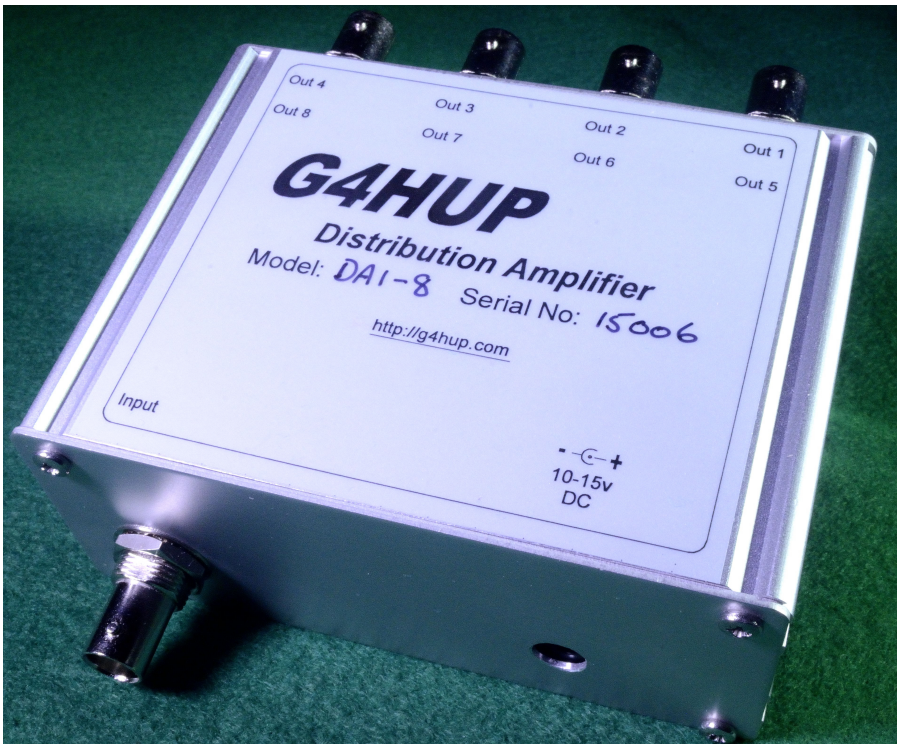
G4HUP

8 Channel
Distribution Amplifier

Technical Manual



Document Author	Dave Powis, G4HUP
Date	24 Mar 2015
Version	Issue 1
Document Ref	HUP-05-024

http://g4hup.com Tel +44 (0)1473 737717 g4hup@btinternet.com

Contents

Unit Specifications

Model Ref High Level Distribution Amplifier DA1-8HL

Serial No

Input Frequency[1]	10	MHz
Input Level	+10	dBm (Max i/p +5dBm)
Output Frequency	10	MHz
Output Level[2]	+10	dBm per channel (0dB Atten)
Gain Variation between channels	<4%	At 10MHz
Phase variation between channels	<±3°	At 10MHz
Inter-channel Isolation	>25	dB typ. at 10MHz
Max output	+10	dBm (recommended)
Supply Voltage	10 – 15	V
Supply Current	500	mA @ 13.4vdc

Notes:
1 The High Level DA is broadband in nature, but measurements quoted here have been conducted at 10MHz.
2 Output level quoted is without any attenuators in the circuit.
3 Power level measurements quoted here have been taken with HP435 Power Meter.

Scope of Document

This document is intended to provide all necessary information to guide users in the construction and installation of all variants of the G4HUP General Purpose Distribution Amplifier Model DA1-8 in normal operation.

Ready built units are supplied complete and tested, with a full test report.

Assembly kit units are supplied DC tested.

This document is relevant for DA1-8 units constructed on GPDAV2.0 PCB's.

Reference data can be found on the DA1-4 pages of the G4HUP web-site, including any identified issues or problems – http://g4hup.com/DA/DA1-4.html.

DA1-8 Distribution Amplifier Outline

The 8 way Distribution Amplifier, DA1-8HL, is formed from two DA1-4 PCB's in a single housing. One board is designated the A board, and the other is the B board. There are minor variations in the construction of each board, which are detailed below.

The two PCB's are mounted one above the other in a similar housing to the DA1-4 series, but with 45.5mm height instead of 30mm. The single input socket appears on the front panel of the unit, along with the power socket, while on the rear side there are two banks of 4 BNC connectors – lower (outputs 1 to 4) and upper (outputs 5 to 8).

Constructors and users are also recommended to check the DA1-4 Technical Manual for detail in addition to these notes. Only variations on that information will be noted here.

At V1 of this document, only the 10MHz version of the DA1-8HL has been tested.

Overall Description

The DA1-8HL uses the Aux Output connector position on both the A and B PCBs to pass the signal from Board A to Board B. The input connector is provided on the A board only, but after the LPF the signal is diverted into the Aux Output/Variable Attenuator circuit block. Here it is split into two, with one path linking back into the A board splitter and amplifier channels, via a bridge at P1 and 0R resistor R9.

Board B is mounted immediately above Board A, and two wire links between the two boards are used to connect the Aux Output BNC socket positions on each board together – no BNC connectors are used. The signal on board B is taken through 0R resistors at R2, R5 and R3 to feed into the Board B splitter and amplifier channels.

The 8v regulators and their associated capacitors are provided on both PCB's, but the input power socket and the protection diode D1 are only provided on PCB A. Another wire link is used from the cathode end of D1 to link vertically up to the pad for the D1 cathode on Board B.

Only these three links are required between the two PCB's.
The BNC sockets used in the DA1-8HL are NOT compatible with those used in current production DA1-4 variants – hence upgrading and existing DA1-4 to a DA1-8 version is not recommended. Low profile BNC sockets MUST be used in the 8 way version, due to enclosure size limitations.

Output Phase Equalisation

As noted in the DA1-4 Manual, phase equalization is very good between the four outputs of one PCB. Measurements have shown that in spite of the linking between the two boards, the phase difference is still very small – of the order of 1.5° (see Fig 11), which is within the phase variations that could be expected on a single bank of outputs. However, because of this potential for each set of four outputs to have a difference between them, it is recommended that any applications which require close phase tolerance between signals should use feeds from the same bank of BNC sockets.

Physical Description

The DA1-8 is housed in an extruded aluminium case, measuring 109 x 80 x 45.5 mm (4.375 x 3.2 x 1.75 inches approx)

External connections are provided for:

- Input signal - BNC
- Output signals – 8 x BNC
- 2.1mm DC power connector

Construction Notes

Check the package contents (if you have a Full Kit version) and make sure all parts are present, and that you can identify them. Build each board as a single item, following the changes below – they are not quite identical in the component placements.

Refer to the DA1-4HL manual for construction sequence and process, with the following changes:

Board A – built as recommended, plus:

- Provide 16R resistors in positions R2, R4 and R5.
- Provide 0R resistor in position shown for R9, below – Fig 1.
- Bridge P1 as shown in Fig Y below

Board B – build as recommended except:

- Do not provide C2, LPF
- Do not provide D1
- Install 0R resistors at R2, R5 and R3 (as per Fig 2, below)

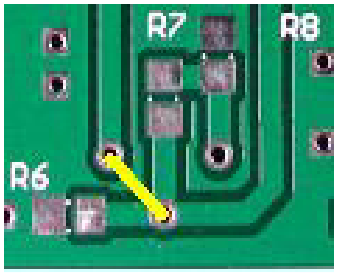
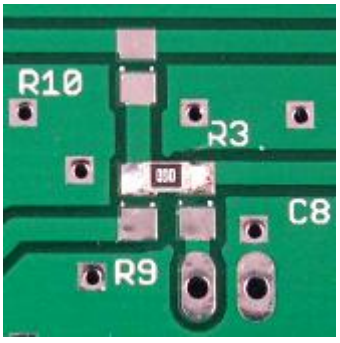
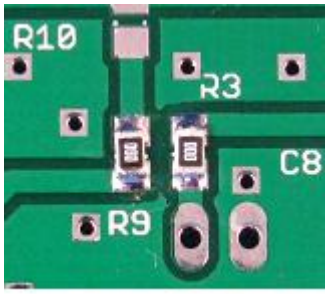


Fig 1 – Bridging out the Variable Attenuator stage





A – variable attenuator bypassed B – variable attenuator active

Fig 2 – R3/R9 signal routing options

Case Preparation

The PCB's slide into slots in the extruded case. Board A locates in the lowest slots, while Board B is 5 slots higher. The two boards are securely held by the BNC bushes on the panel. Front and rear panels must be drilled to accommodate the BNC connectors and power connector.

You can use the supplied guides either as 'stick-on' overlays, or as information sources to do you own measuring and marking.

It is recommended that you download and print the separate Drilling Overlay to ensure accuracy

Fig 10 is a drilling guide for the front panel. Note that the aluminium panels are **not** symmetrical! Take care to get the panel the right way up before you mark out the drilling positions!

Fig 11 is the rear panel drilling guide – the same warning applies here!

Slide PCB A into the case and lay the rear panel over it. Secure the 4 BNC bushes with washers and nuts. Fasten the panel to the box outer using the self-tapping screws provided. Now slide Board B into the case, 5 slots higher up, so that it's BNC connectors pass through the upper set of holes. As before, secure the BNC bushes with washers and nuts.

Check that the front panel drillings fit the input BNC connector and give good access to the DC power connector.

Remove the rear panel and two PCB's as a single unit. Using suitable tinned copper wire, link the signal and ground connections of the Aux Output BNC connector positions between the two boards, as shown in Fig 3.

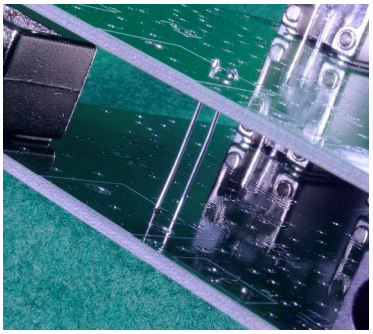


Fig 3 10MHz connection between the two PCB's

Similarly, solder a length of wire from the cathode of D1 on Board A directly to the point above it where the cathode of D1 would be connected on Board B – this will connect the power supply to the Board B regulator circuit. See Fig 4.

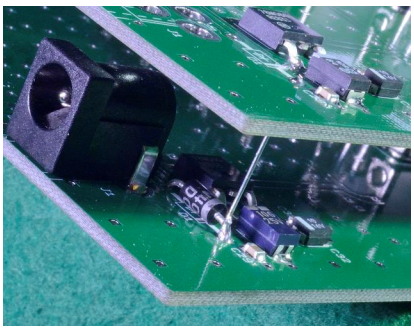


Fig 4 Power connection between boards

At this stage the full 8 way circuit can be tested – remember to terminate all unused outputs in 50R!

If any attenuators are required in the individual output circuits to equalize levels, they can be installed at this stage.

Once testing is complete, re-install the assembly into the case, and attach the front panel.

The four rubber feet can be attached to the underside of the case.

Finally, remove the backing from the adhesive label and install it on the top of the case -make sure you get it the right way round!

Errata and Addenda

This section contains information about components that have been changed or added compared with the original PCB design.

See http://g4hup.com/VDA/DAerrata.html for full details, versions impacted and resolution guidance, including pictorial support.

Component Locations

Figs 8 and 9 respectively show the locations of components on the top side and lower side of the PCB for individual DA1-4 PCB's

Maintenance

Construction Practices

Kits are not supplied with any solder, but it is recommended that a small diameter, good quality cored flux solder is used, to ensure minimum flux residues on the PCB after assembly. The PCB will accept lead-free solder, and components used are generally ROHS compliant, and should therefore also accept lead-free solder if you prefer.

It is recommended that lead based solder is used for maximum reliability of soldered joints.

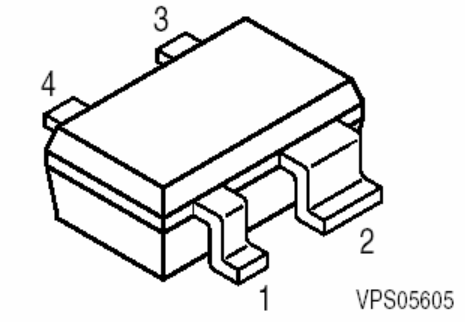
Change History

Date	Iss No	Comment	Author
15 Mar 2015	0.A	First Draft version	G4HUP
24 Mar 2015	1.0	First issued version	G4HUP

End of text – Diagrams follow

Component List DA1-4HL – Board A

ID	Value	Spec	Comment
PCB	GPDA	V2.1	
Box			109x80x45.5mm
X1 - X5		BNC	Low profile Right Angle
J1	2.1mm		DC Power PCB mtg
C1, 9-12, 14, 16, 18, 20, 21-29, 31	100n	0805/50v	
C6, 8	330p	0805/50v	
C7	560p	0805/50v	
C13, 15, 17, 19	10n	0805/50v	
C30	10u	25V	
C32	10u	16v	
R2, 4, 5, 10, 13 - 16	16R	0805/ 0.125W	
R3, C3	0R	0805/ 0.125W	Use 0R in place of C3
R11, 12	33R	0805/ 0.125W	
R17 - 20	68R	1206/ 0.2W	
R21 - 32	SOT	0805/ 0.125W	4 x 0R supplied
L2,3	1uH	1008	
D1	1N4001		DC Protection
IC2 - 5	BGA616		
IC6	78M08		
Rubber Feet	4		
Adhesive Decal	1		



BGA 616 connections

Input – pin 1
Output pin 3
Ground pins 2,4

Pinout data courtesy of Infineon BGA616 data sheet.

Component List DA1-4HL – Board B

ID	Value	Spec	Comment
PCB	GPDA	V2.1	
X2 - X5		BNC	Low profile Right Angle
C9-12, 14, 16, 18, 20, 21-29, 31	100n	0805/50v	
C13, 15, 17, 19	10n	0805/50v	
C30	10u	25V	
C32	10u	16v	
R10, 13 - 16	16R	0805/ 0.125W	
R2, 3, 5	0R	0805/ 0.125W	
R11, 12	33R	0805/ 0.125W	
R17 - 20	68R	1206/ 0.2W	
R21 - 32	SOT	0805/ 0.125W	4 x 0R supplied
IC2 - 5	BGA616		
IC6	78M08		

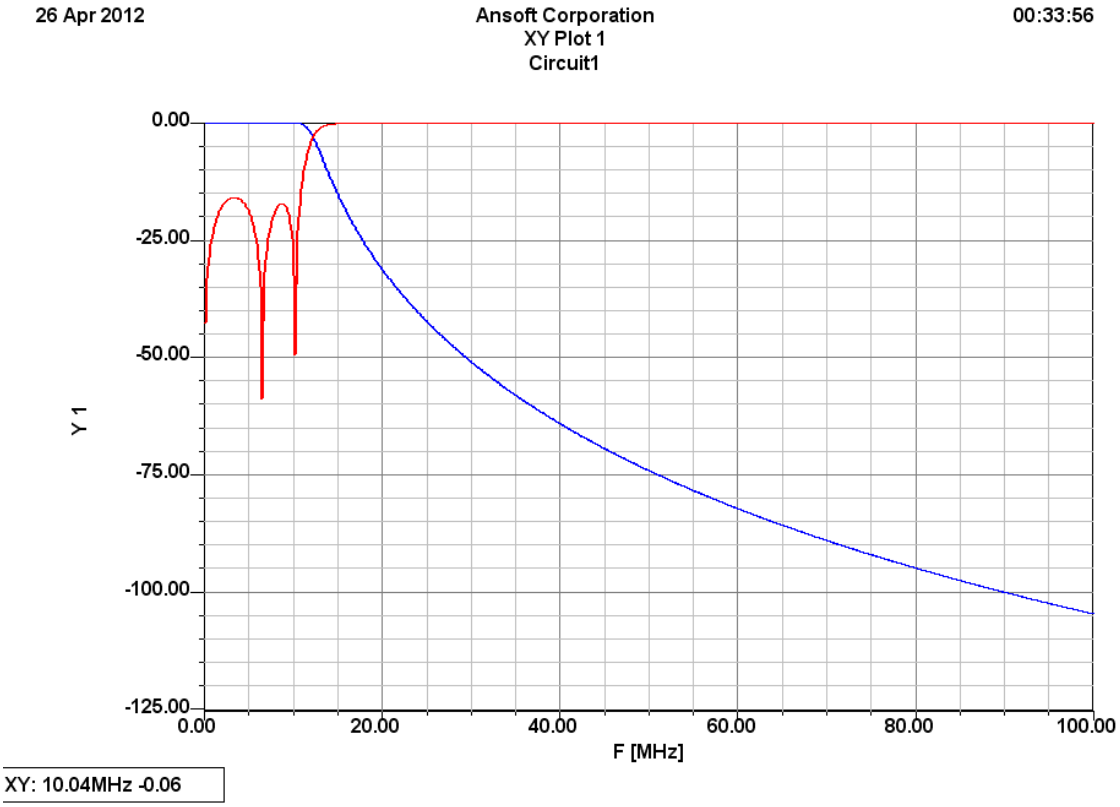


Fig 5 – 10MHz Low Pass Filter response

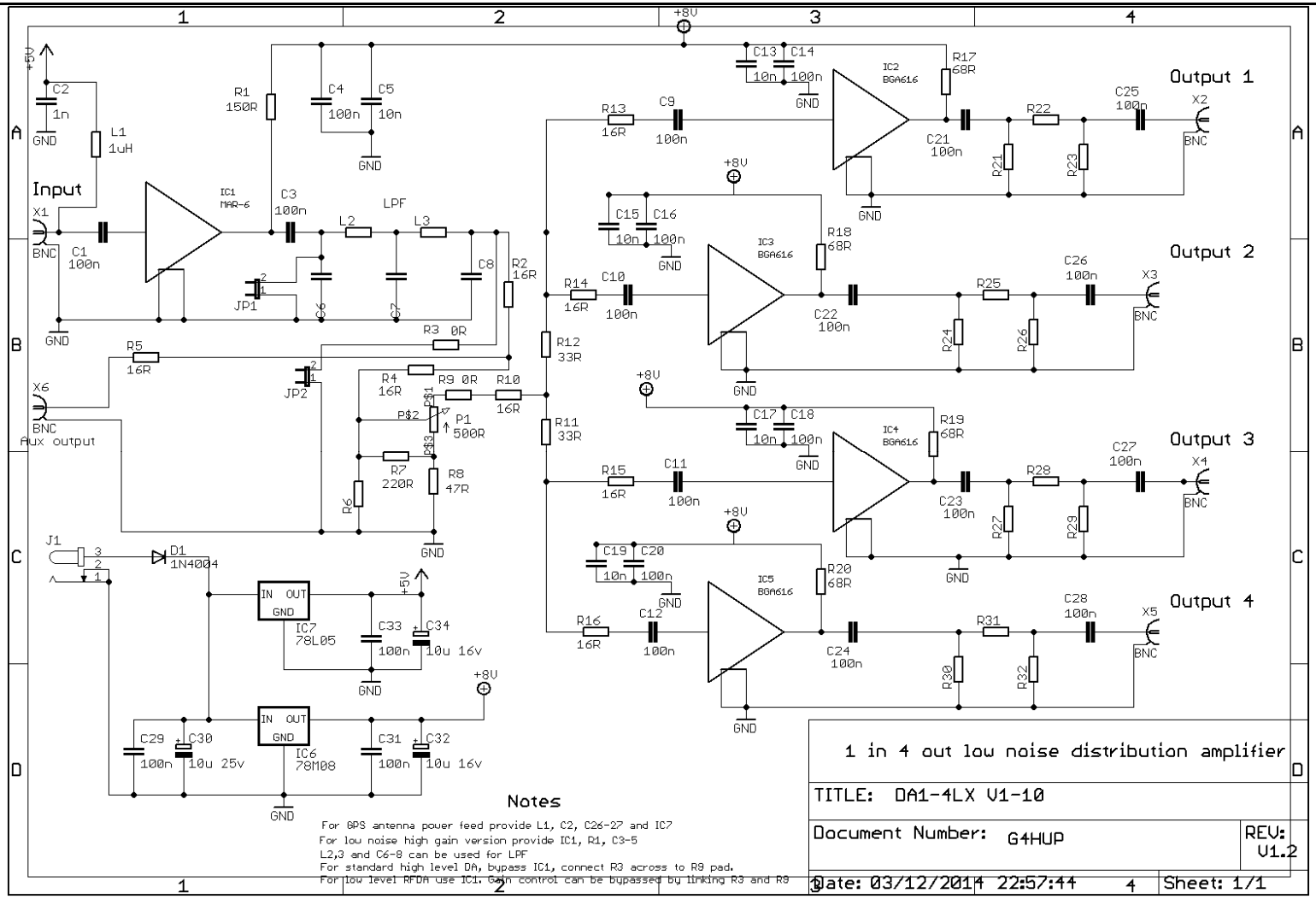


Fig 6 – General Purpose Distribution Amplifier circuit

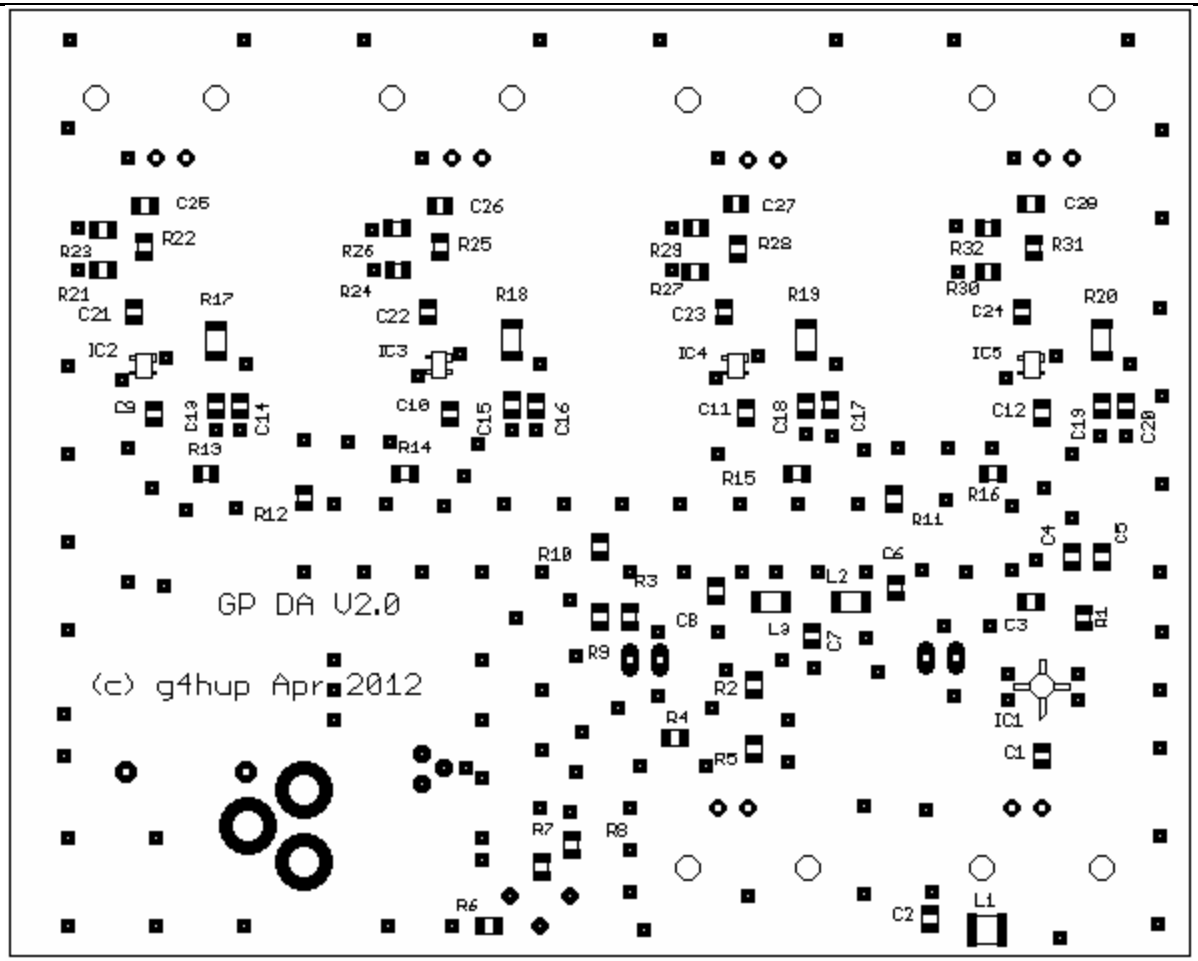


Fig 7 – PCB Component side Locations

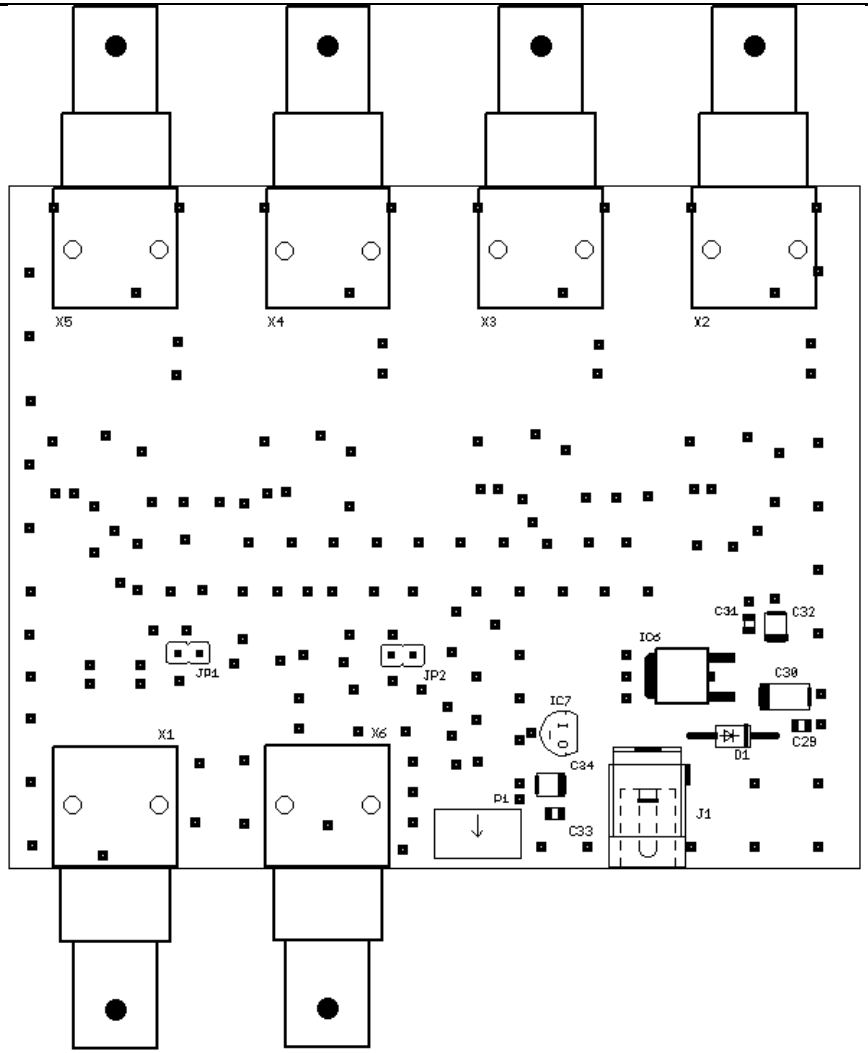
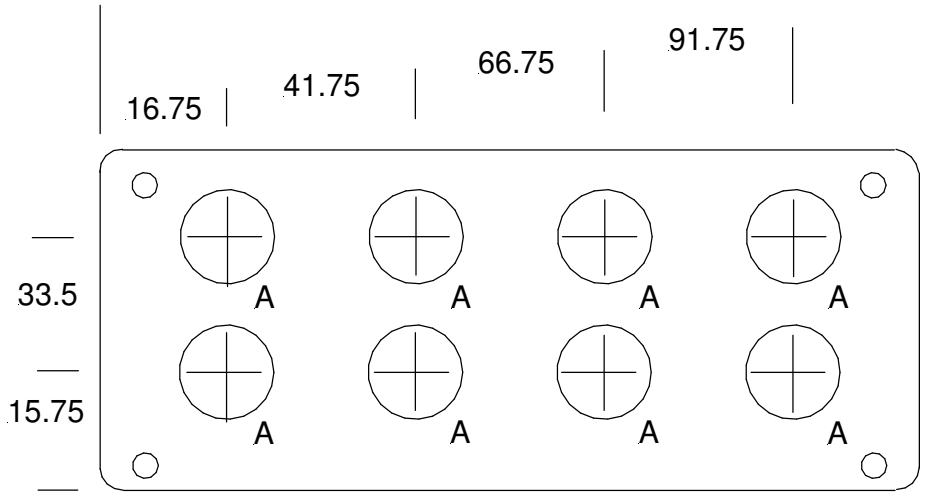


Fig 8 – Top side PCB overlay



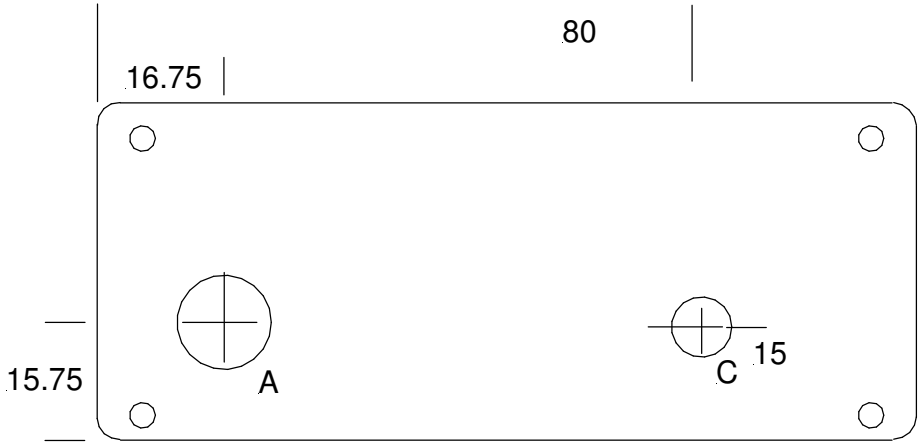
Rear panel

CAUTION!! Blank panel is not symmetrical
– ensure that screw holes are offset as shown

– view shown is as seen from outside case

All dimensions in mm
A – 12.5mm dia; B – 5mm dia; C – 8mm dia

Fig 9 - Rear panel drilling guide



Front panel

CAUTION!! Blank panel is not symmetrical
- ensure that screw holes are offset as shown
- view shown is as seen from outside case

Fig 10 – Front panel drilling guide

Download full size drill overlay from http://g4hup.com/DA/DA1_8.htm

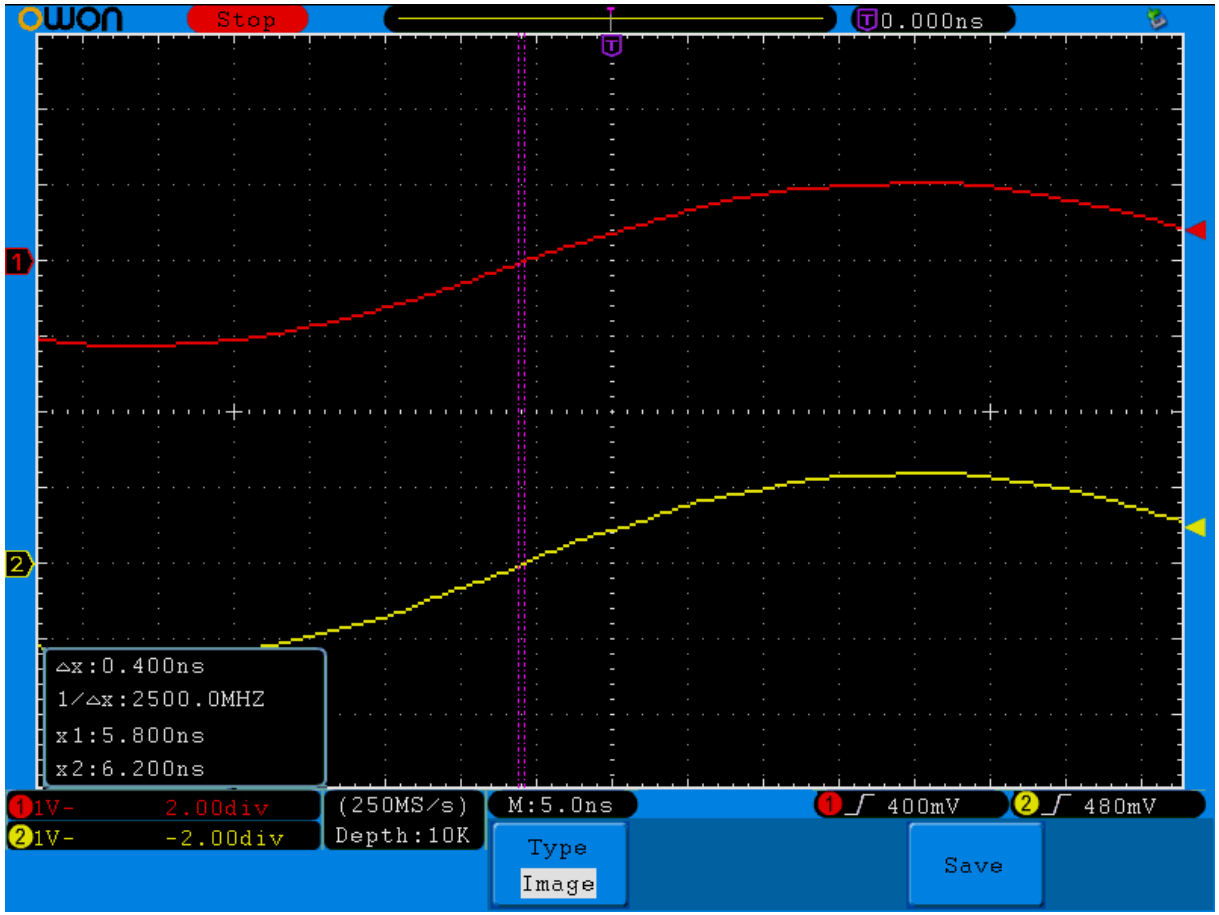


Fig 11 – Phase variation between output channels in different banks at 10MHz

– typ 1.5° (0.4ns)

Appendix 1 – Pi Attenuator Resistor Values

The following table gives data for construction of resistive Pi type attenuators. It is taken from the RSGB published Microwave Handbook, Volume 2, page 10.6. In the table R1 refers to the shunt resistors to ground (2 off) while R2 refers to the series reistor.

Many of the values are not readily available within the standard E24 series – so if you decide not to use precision resistors, but to round the values to suit availability, then you should round both R1 and R2 values in the same direction – ie up or down, to preserve the correct ratio. However, this will result in the impedance being slightly away from the intended 50R.

dB	R1	R2
1	870	5.8
2	436	11.6
3	292	17.6
4	221	23.9
5	178	30.4
6	150	37.4
7	131	44.8
8	116	53
9	105	62
10	96	71
12	84	93
14	75	120
16	69	154
18	64	196
20	61	248
25	56	443
30	53	790

Table A1 – Pi Attenuator resistors