

1 2 3 4 5 6 7 8 9 10

A
B
C
D
E
F
G
H

A
B
C
D
E
F
G
H

PCB Layer Stackup:

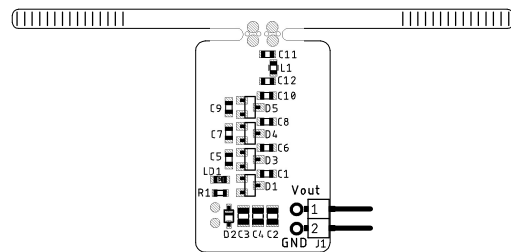
#	File	Layer	Properties
1	GTO	Top SilkScreen	white
2	GTS	Top SolderMask	green
3	GTL	Top Copper	35um
4	-	FR4 Dielectric	-
5	GBL	Bottom Copper	35um
		PCB thickness	1.0mm total
6	GBS	Bottom SolderMask	green
7	GB0	Bottom SilkScreen	white
-	TXT	Drill File (Excellon Format)	
-	GKO	Keepout/Dimension File	

0.086 / 10%

Additional Information:

PCB size: (1) 66.30x32.13
 PCB finish: (2) HASL
 Drill file format: (3) EXCELLON
 Gerber file format: (4) GERBER_RS274X
 Minimum clearance: (5) 6mil
 Minimum track width: (6) 6mil
 Electrical layer count: (7) 2
 Minimum drill diameter: (8) 0.3mm
 (mentioned drill diameters are with finish)
 PCB edge conditioning: (9) milling / V-cut allowed where possible

0.076 / 8%



Sym	N°	Mils	MM	Qty	Plated
+	1	12	0.30	25	YES
x	2	24	0.60	4	NOT
□	3	39	1.00	6	YES

*.INF1 - Information Layer file *.INF2 - Drill size/type chart			
Project:	mWessenger_v3		
Variant:			A
Author:	VJ	Revision:	
Modified:	20-Oct-18 20:58	Scale:	1:1

1 2 3 4 5 6 7 8 9 10