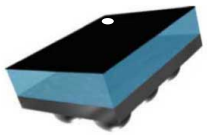
MLPF-WB-01D3

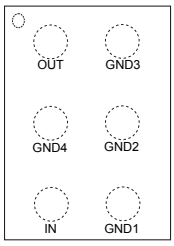
Datasheet

2.4 GHz low pass filter matched to STM32WBxx in UFQFPN and VFQFPN packages



Chip scale package on glass 6 bumps

Pin-out top diagram (top view - bumps down)



Product status link
MLPF-WB-01D3

Features

- Integrated impedance matching to to STM32WBxx in UFQFPN and VFQFPN packages
- 50 Ω nominal impedance on antenna side
- Deep rejection harmonics filter
- Low insertion loss
- Small footprint
- Low profile ≤ 630 µm after reflow
- High RF performances
- RF BOM and area reduction
- ECOPACK2 compliant component

Applications

- Bluetooth 5
- OpenThread
- Zigbee®
- IEEE 802.15.4
- Optimized for STM32WBxx in UFQFPN and VFQFPN packages

Description

The MLPF-WB-01D3 integrates an impedance matching network and harmonics filter. The matching impedance network has been tailored to maximize the RF performances of STM32WBxx in UFQFPN and VFQFPN packages. This device uses STMicroelectronics IPD technology on non-conductive glass substrate which optimizes RF performances.

1 Characteristics

Table 1. Absolute ratings (T_{amb} = 25 °C)

Symbol	Parameter	Value	Unit
P_{IN}	Input power RF_{IN}	10	dBm
V_{ESD}	ESD ratings human body model (JESD22-A114-C), all I/O one at a time while others connected to GND	2000	V
T_{OP}	Maximum operating temperature	-40 to +105	°C

Table 2. Impedances (T_{amb} = 25 °C)

Symbol	Parameter	Value			Unit
		Min.	Typ.	Max.	
Z_{IN}	STM32WBxx single-ended impedance	-	Matched to STM32WBxx in UFQFPN and VFQFPN packages	-	Ω
Z_{OUT}	Antenna impedance	-	50	-	Ω

Table 3. Electrical characteristics and RF performances (T_{amb} = 25 °C)

Symbol	Parameter	Test conditions	Value			Unit
			Min.	Typ.	Max.	
f	Frequency range		2400		2500	MHz
IL	Insertion loss $\lvert S_{21}\rvert$			0.9	1.1	dB
RL_{IN}	Input return loss $\lvert S_{11}\rvert$		19	21		dB
RL_{OUT}	Output return loss $\lvert S_{22}\rvert$		20	21		dB
Att	Harmonic rejection levels $\lvert S_{21}\rvert$	Attenuation at 2fo (4800 – 5000) MHz	34	39		dB
		Attenuation at 3fo (7200 – 7500) MHz	60	63		dB
		Attenuation at 4fo (9600 – 10000) MHz	53	57		dB
		Attenuation at 5fo (12000 – 12500) MHz	44	45		dB

1.1 RF performances

Figure 1. Transmission (dB)

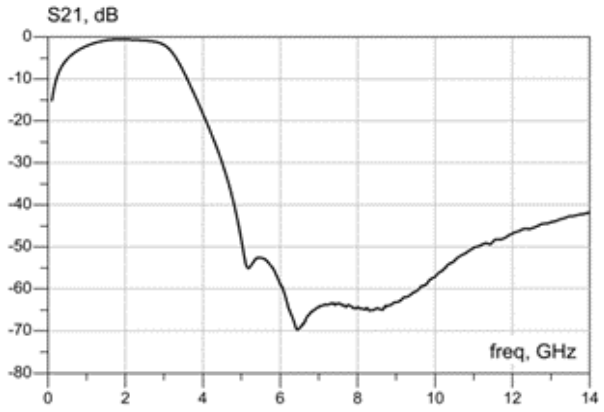


Figure 2. Insertion loss (dB)

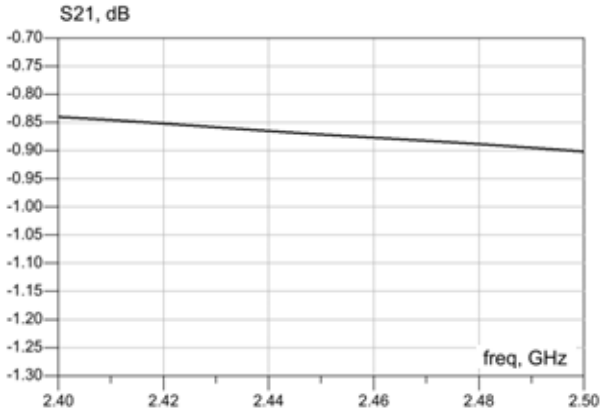


Figure 3. Input return loss (dB)

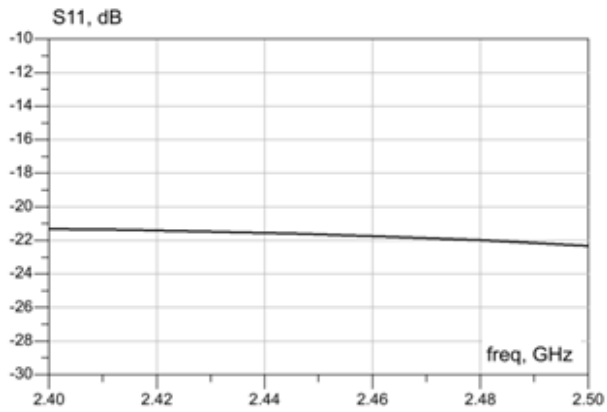


Figure 4. Output return loss (dB)

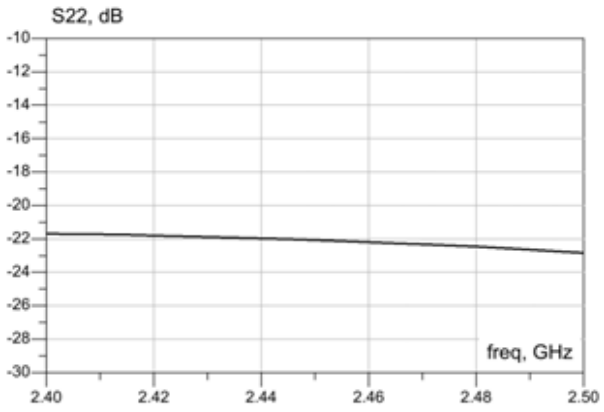


Figure 5. Attenuation 2f0 (dB)

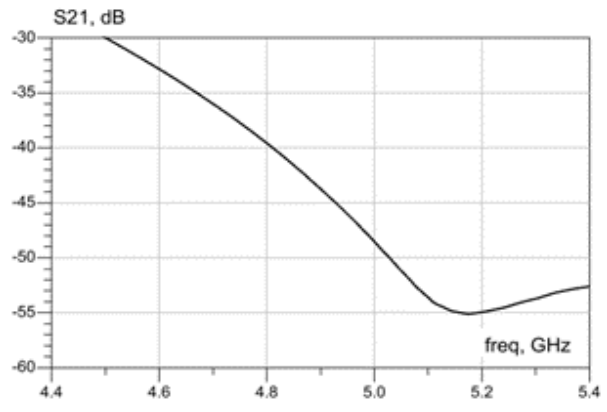


Figure 6. Attenuation 3f0 (dB)

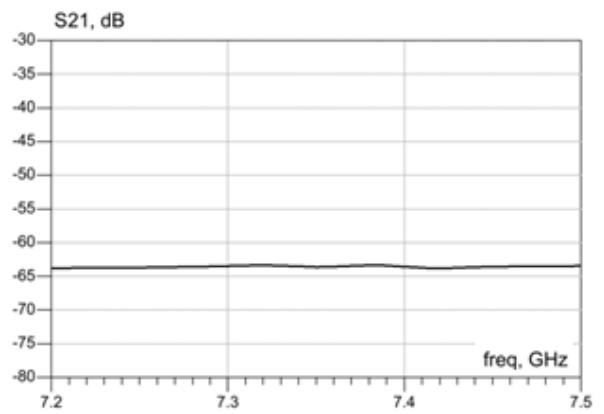


Figure 7. **Attenuation 4f0 (dB)**

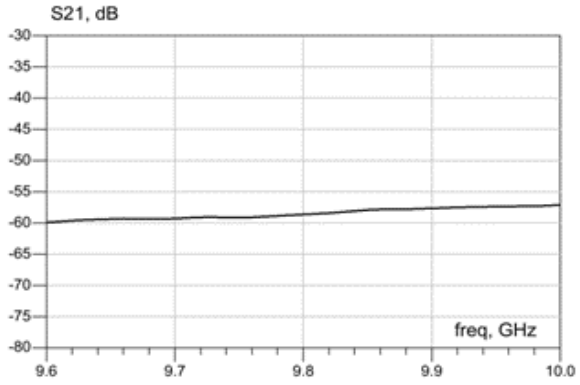
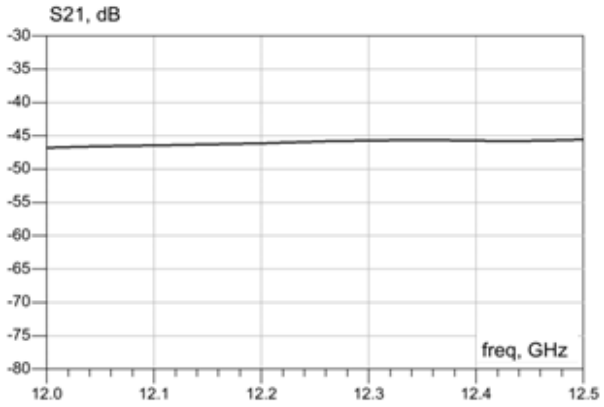


Figure 8. **Attenuation 5f0 (dB)**



2 Package information

In order to meet environmental requirements, ST offers these devices in different grades of ECOPACK packages, depending on their level of environmental compliance. ECOPACK specifications, grade definitions and product status are available at: www.st.com. ECOPACK is an ST trademark.

2.1 CSPG package information

Figure 9. CSPG package outline (bottom view - bumps up)

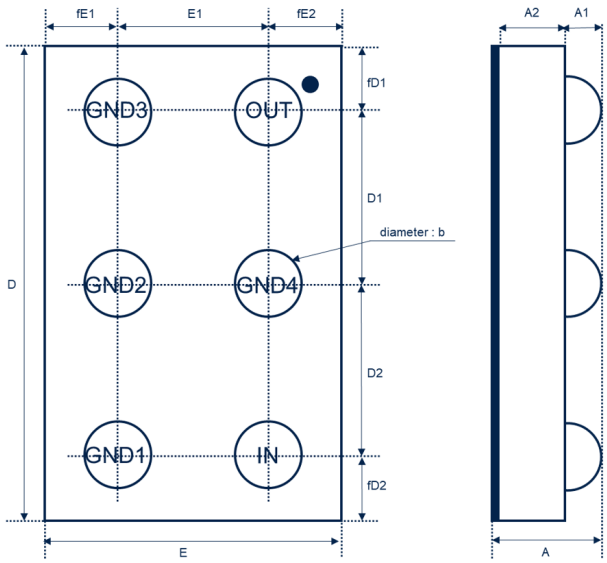


Table 4. CSPG 6 bumps mechanical data

Ref.	Dimensions		
	Millimeters		
	Min.	Typ.	Max.
A	0.580	0.630	0.680
A1	0.180	0.205	0.230
A2	0.380	0.400	0.420
b	0.230	0.255	0.280
D	1.550	1.600	1.650
D1		0.577	
D2		0.577	
E	0.950	1.000	1.050
E1		0.500	
fD1		0.223	
fD2		0.223	
fE1		0.250	
fE2		0.250	

Figure 10. **Marking**

Figure 11. **Top view**

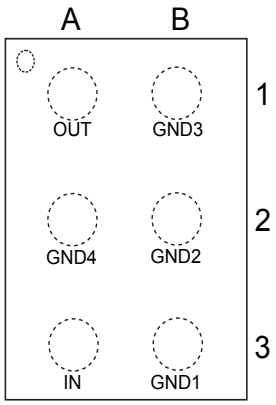


Table 5. **Pad description top view (pads down)**

Pad ref	Pad name	Description
A1	OUT	Antenna
A2	GND4	Ground
A3	IN	STM32WBxx out
B1	GND3	Ground
B2	GND2	Ground
B3	GND1	Ground

Figure 12. **Tape and reel outline**

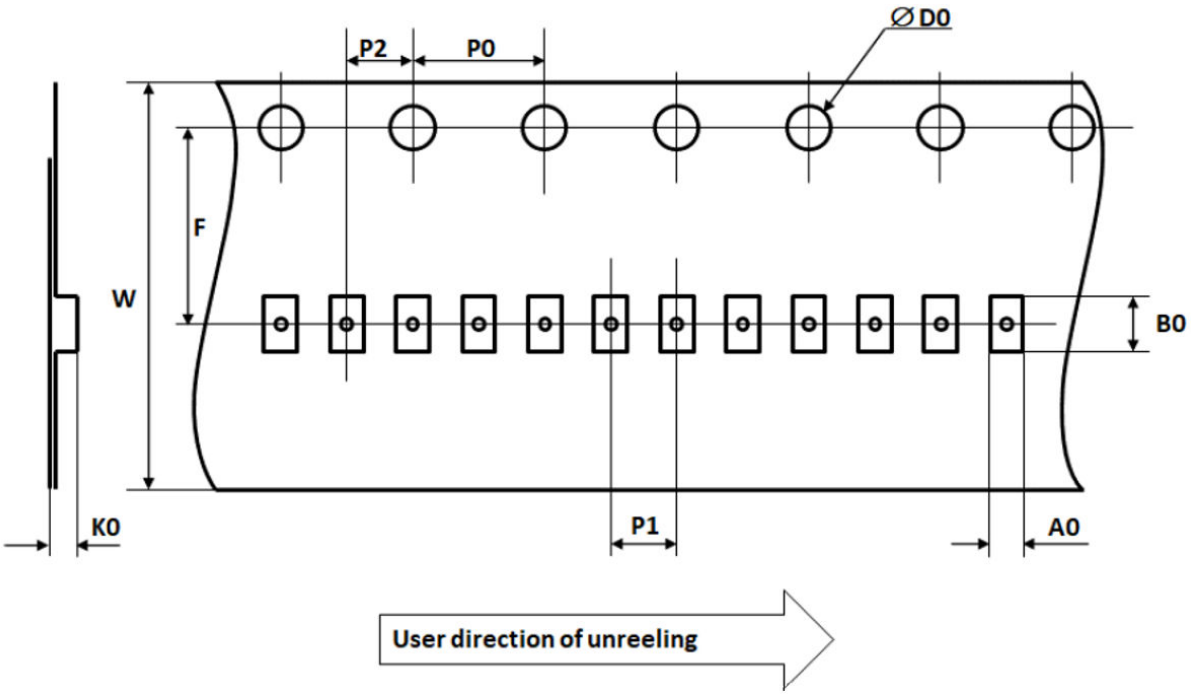


Table 6. Tape and reel mechanical data

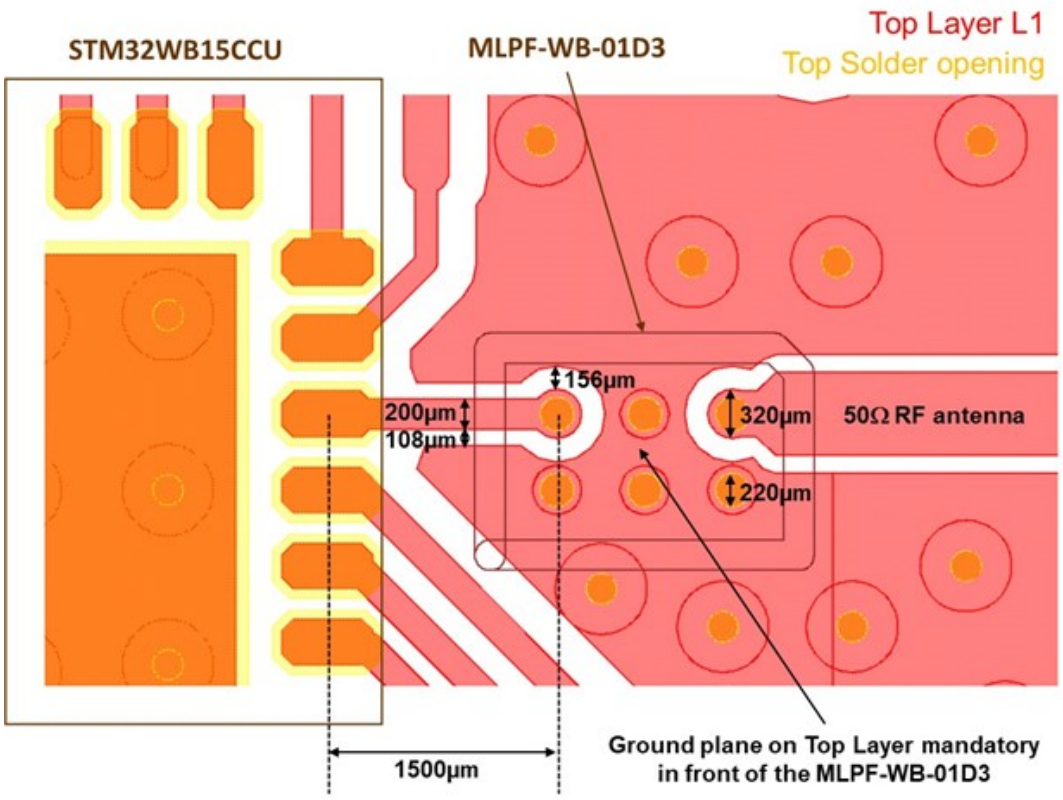
Ref	Dimensions		
	Millimeters		
	Min	Typ	Max
A0	1.06	1.09	1.12
B0	1.66	1.69	1.72
D0	1.40	1.50	1.60
F	3.45	3.50	3.55
K0	0.69	0.72	0.75
P0	3.90	4.00	4.10
P1	1.95	2.00	2.05
P2	1.95	2.00	2.05
W	7.90	8.00	8.30

3 Recommendation on PCB assembly

3.1 Land pattern

Layout example using STM32WB15CCU.

Figure 13. PCB land pattern recommendations



The transmission line between MLPF and antenna is dimensioned to 50 ohms characteristic impedance.

The transmission line between STM32 is dimensioned to 67 ohms characteristic impedance.

These transmission line characteristic impedances have to be followed as close as possible.

Moreover, lines physical dimensions will have to be tuned according to specific PCB stack-up, if different from the one presented in datasheet, to keep expected characteristic impedance values.

The ground plane vias density needs to be maximized near the MLPF-WB-01D3 area to ensure optimal RF performances.

Figure 14. PCB stack-up recommendations

#	Name	Material	Type	Weight	Thickness	Dk
	Top Solder	Solder Resist	Solder Mask		0.03mm	3.5
1	Top Layer		Signal	1oz	0.037mm	
	Dielectric 1	FR-4	Core		1.55mm	3.95
2	Bottom Layer		Signal	1oz	0.037mm	
	Bottom Solder	Solder Resist	Solder Mask		0.03mm	3.5
	Bottom Overlay		Overlay			

3.2 Stencil opening design

Figure 15. Footprint - 3 mils stencil -non solder mask defined

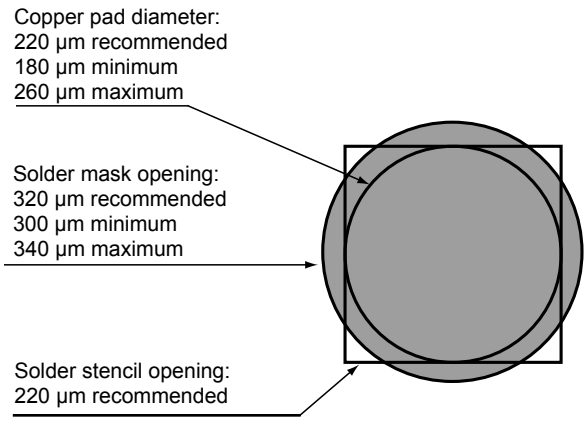
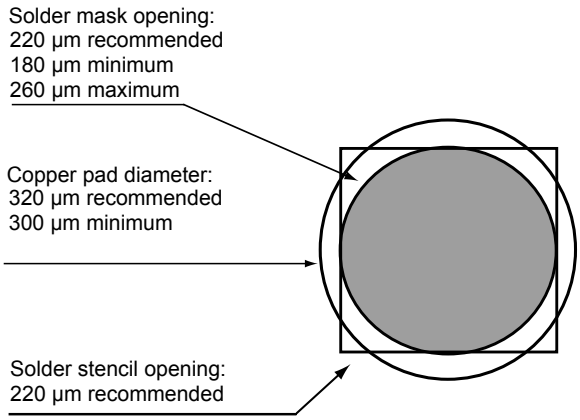


Figure 16. Footprint - 3 mils stencil - solder mask defined



3.3 Solder paste

1. Halide-free flux qualification ROL0 according to ANSI/J-STD-004.
2. “No clean” solder paste is recommended.
3. Offers a high tack force to resist component movement during high speed.
4. Use solder paste with fine particles: powder particle size 20-38 µm.

3.4 Placement

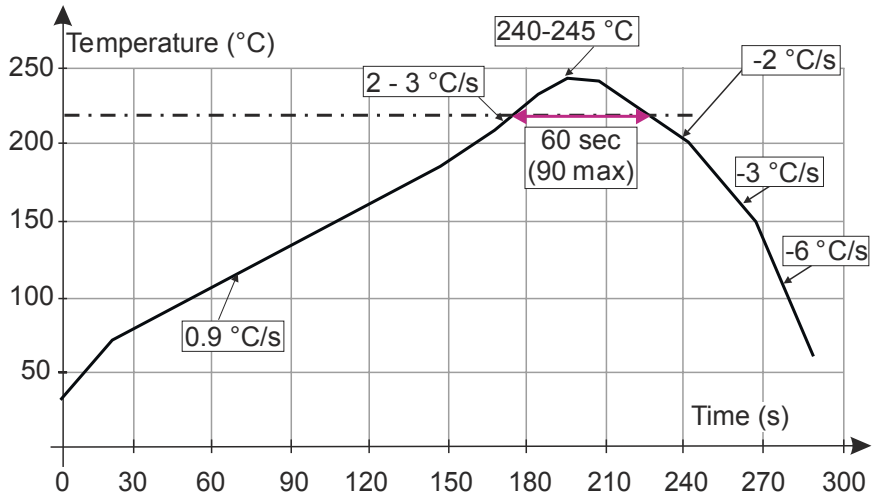
1. Manual positioning is not recommended.
2. It is recommended to use the lead recognition capabilities of the placement system, not the outline centering
3. Standard tolerance of ±0.05 mm is recommended.
4. 1.0 N placement force is recommended. Too much placement force can lead to squeezed out solder paste and cause solder joints to short. Too low placement force can lead to insufficient contact between package and solder paste that could cause open solder joints or badly centered packages.
5. To improve the package placement accuracy, a bottom side optical control should be performed with a high resolution tool.
6. For assembly, a perfect supporting of the PCB (all the more on flexible PCB) is recommended during solder paste printing, pick and place and reflow soldering by using optimized tools.

3.5 PCB design preference

1. To control the solder paste amount, the closed via is recommended instead of open vias.
2. The position of tracks and open vias in the solder area should be well balanced. A symmetrical layout is recommended, to avoid any tilt phenomena caused by asymmetrical solder paste due to solder flow away.

3.6 Reflow profile

Figure 17. **ST ECOPACK recommended soldering reflow profile for PCB mounting**



Note: Minimize air convection currents in the reflow oven to avoid component movement.

Note: More information is available in the application note:

- *AN2348 Flip-Chip: "Package description and recommendations for use"*

4 Ordering information

Table 7. **Ordering information**

Order code	Marking	Package	Weight	Base qty.	Delivery mode
MLPF-WB-01D3	TW	CSPG	1.82 mg	5000	Tape and reel

Revision history

Table 8. **Document revision history**

Date	Revision	Changes
17-Feb-2023	1	Initial release.