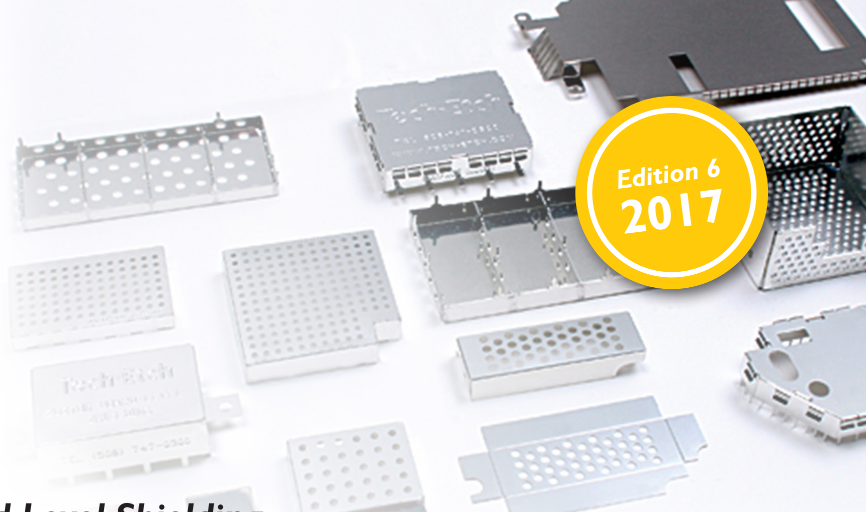

An introduction to Board Level Shielding

Shielded enclosures for PCB

Electronic circuits can be shielded for various reasons. Some circuits are shielded for thermal stability like used in reference circuitry. Another, more obvious reason, is for protection against humidity or water. Some circuits require shielding to avoid being interfered by other electronics or to reduce emission.

This newsletter deals with shielding at printed circuit-board (PCB) level. This shielding method will be referred to as Board Level Shielding (BLS).

Faraday cage...or not?

The Faraday Cage is often referred to as the ultimate way of shielding electronics. However, an absolute way to shield electronics from DC to daylight, as the Faraday Cage is often supposed to do, can't be achieved. Circuitry has to be cooled which means vent holes. In addition: electronics need to be supplied and require communication lines and that means power and signal lines. Servicing might require the shield to be opened occasionally. Therefore seams and slots are unavoidable. Shielding boxes, regardless of size and shape, therefore add attenuation to radio frequency (RF) energy rather than 'total' shielding. The basic questions that have to be addressed is: *"How much attenuation at which frequency (band) is required?"*.

Board level shielding considerations

Many factors have influence on the selection of shielding on PCB level as figure 1 shows.

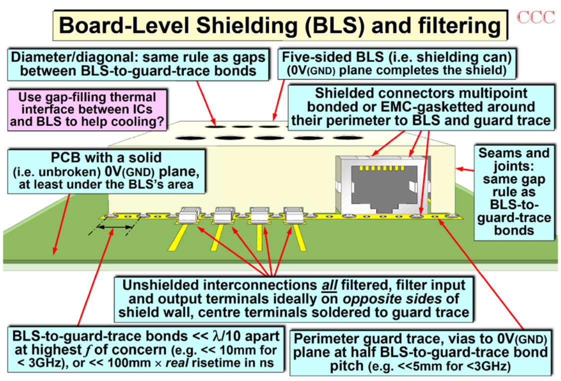

Figure 1: Board-Level Shielding (BLS) considerations. Figure reproduced with approval of Cherry Clough Consultants Ltd, www.cherryclough.com

Frequency and dimensions

Lots of fixation spaces in EMC are governed by frequency. The maximum anticipated frequency for which the shield shall provide attenuation is, as a rule-of-thumb (refer to figure 1):

$$d \ll \frac{c}{10 \cdot f}$$

d	f	c
m	Hz	$\frac{m}{s}$

Where $c = 3 \cdot 10^8$ [m/s] and 'f' is the highest frequency of concern.

An important remark to the spacing has to be made. A spacing of $\frac{c}{10 \cdot f}$ as figure 1 shows will be small enough not to cause resonances. In practise, a $\frac{c}{20 \cdot f}$ will provide an attenuation of *about* 10 dB. It is therefore recommended to use a spacing of $d \ll \frac{c}{20 \cdot f}$ in cases where attenuation of the BLS is very important.

The sixth side of the board level shield

Board level shields are open at the bottom when soldered to the PCB. This means that, if no precautions are taken, electromagnetic fields will emit from the PCB-side. The largest physical dimension of the BLS will determine the lowest frequency that will be radiated. This dimension will be the diagonal of the BLS. The ground plane can be used to close the shield as figure 2 shows. The ground plane will therefore be the sixth side of the BLS.

About the author

"Since 2000, Mathieu Melenhorst has been a consultant for various organisations in the EMC branche. Together with Dr. Ir. Mark van Helvoort, Melenhorst co-wrote the book "EMC van Installaties – Op weg naar elektromagnetische compatibiliteit", issued by Vakmedianet, The Netherlands (ISBN 978-90-125-8552-1).

HF Technology - Technical Newsletter

This newsbulletin is issued by HF Technology - Krommenie, The Netherlands - www.hftechnology.nl - +31 (0)75 - 628 37 17

Cavity resonances

Electromagnetic waves in the far field propagate in air that has a characteristic impedance of about 120π[Ω]. If these waves hit a metallic surface that has a low impedance, mismatch will occur. As a consequence, reflections happen. These reflected waves might hit other metallic surfaces and when confined in a box, standing waves are created. These standing waves happen at specific frequencies that are determined by the physical dimensions of the box. A BLS can be such a box where these standing waves or cavity resonances can occur. The fill factor of the cavity determines the quality factor of the cavity hence its likeliness to resonate. Absorbing material can help decreasing the quality factor thus decrease the likelihood of resonance.

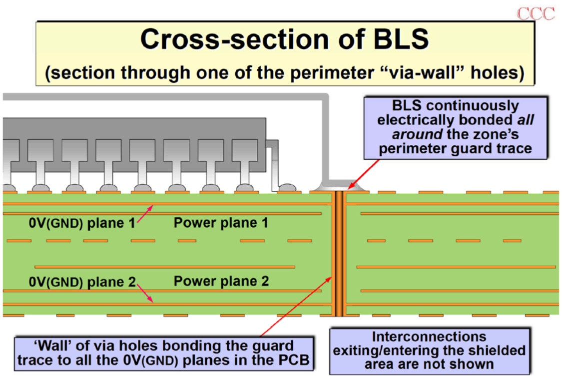

Figure 2:
Cross-sectional view of the BLS Figure reproduced with approval of Cherry Clough Consultants Ltd, www.cherryclough.com

Interface lines

Electronics are connected to the world 'outside' of their confinement (shielded box). It is important to remember that all traces that connect the shielded enclosure to the world outside will behave like accidental antennas: they weren't designed to act as an antenna but they do behave like one. Noise will be carried from the inside to the outside world and vice versa. Possible mitigation methods to reduce the transferred noise[1] that will be explained in this newsletter include **filtering** of the signal lines and/ or proper **trace routing**.

Signal filtering

Figure 3 shows the basic principle of common and differential-mode signal filtering. The majority of EMC emission problems are caused by common-mode currents. These are filtered by capacitors 'C1', 'C2' and 'C3'. Capacitors 'C1' and 'C2' are differential-mode filter components.

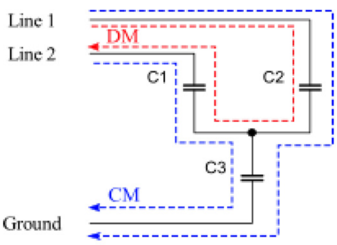

Figure 3:
Filtering: Common-mode and differential-mode

[1] also known as *re-radiated* noise.

Figure 1 shows lines that are filtered in *differential mode*. Figure 4 shows an example of the effect: signal and noise are clearly separated and filtering does not affect the signal contents.

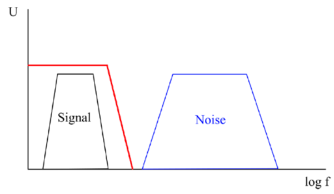

Figure 4: Filtering out-of-band noise

Filtering fails when noise (partially) occupies the signal frequency band. Figure 5 shows that parts of the signal will be transferred as well as parts of the noise yet a small band can be regarded as interference of the signal as it contains both.

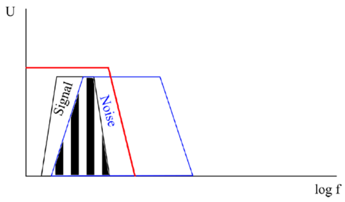

Figure 5: Transferred signal and noise

Figure 5 also shows that part of the noise is also within the passband of the filter.

PCB Trace routing

Figure 2 shows traces that run inside the shielded part of the PCB. It is advisable routing traces away from the BLS in close vicinity of a ground plane as scenario 1 shows in figure 6 shows.

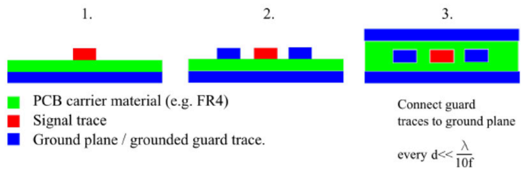

Figure 6: Trace routing

Scenarios 2 and 3 provide extra shielding of the trace if required. Signal traces shall be kept away from board edges, splits or gaps to avoid accidental antennas.

The author would like to thank mr. K.Armstrong of Cherry Clough Consultants for the many fruitful discussions.

TECHNOLOGY
www.hftechnology.nl
075 - 628 37 17
Postbus 343 - 1560 AH Krommenie