PRACTICAL RF PRINTED CIRCUIT BOARD DESIGN

Geoff Smithson.

Overview

The electrical characteristics of the printed circuit board (PCB) used to physically mount and connect the circuit components in a high frequency product will have a significant impact on the performance of that product. The potential magnitude of the effect of the PCB design increases with frequency as the parasitic elements tend to a similar magnitude to the typical lumped components used. This leads to the requirement for increased inclusion of electrical models representing PCB structures into the circuit simulation. In practical radio products the requirements on the PCB design can sometimes be in conflict with 'best manufacturing' practise. In order to achieve the most optimum solution in all respects the radio designer, the PCB designer and the manufacturing engineer must all have some appreciation of each others requirements.

PCB Materials

A variety of different materials are used to fabricate PCBs. These materials can also be assembled in a variety of different ways potentially using multiple laminates, different materials and different plated through via structures. A range of finishes can be used making use of materials such as Gold, Nickel, Tin and Lead. Typical board materials used for radio circuits are FR4, Rogers R04003 and Rogers RT/Duroid. All of these come in a variety of grades and forms and have different electrical characteristics and cost.

Hybrid structures combining two of these materials are possible, allowing a combination of properties to be used for different circuit areas. Some examples of hybrid structures that have been used in commercial products include a duroid and FR4 construction used for microwave satellite receiver low noise down converter blocks produced by various companies in the 1990s as well as a Rodgers and FR4 hybrid construction used by Plextek in a formula one telemetry system. (A photograph of the Rodgers R04003 and FR4 hybrid is shown at the end of this paper).

A summary of the main electrical characteristics of three commonly used board materials for radio products are given in the table below. The cost increases with increased dielectric tolerance and reduced loss.

	Typical Dielectric constant	**Dielectric constant tolerance**	**Loss tangent**
Board Material:			
FR4	4	+/-5% → +/-25%	0.01
Rogers R04003	3.38	+/- 0.05	0.0027
Rogers RT/Duroid	2.2	+/- 0.03	0.0009

Table 1. Summary of the principle properties of different radio PCB materials.

Geoff Smithson is an RF Engineer with Plextek Ltd, London Road, Great Chesterford, Essex.

Tel: +44 (0)1799 533200. Fax: +44 (0)1799 533201. Email: gs@plextek.co.uk

PCB Fabrication Process

The most common processes used in the fabrication of PCBs are drilling, plating, bonding and etching. Some new developments are underway where lasers are utilised to distinguish conducts. The principle advantage of this development being the ability to use smaller dimensioned features including track widths down to 0.05 mm.

In all but the lowest cost radio products multilayer PCBs are used. These comprise of a number of laminates of board material individually etched and bonded. The principle advantages of this being the ability to use more than two conductor layers reducing the required board space. The disadvantage being increased cost.

A typical PCB structure used in volume radio products, such as GSM mobiles, is a four layer or six layer FR4 multilayer construction. In addition to different structures for the laminates, a variety of different structures can be used for the plated through holes (known as 'via' holes). The most basic structure being simply to have all holes going through all layers of the board. For a multilayer board these can then be used to link tracks on any layer to any other or all other layers. The advantage of this structure being the reduced processing time as the vias are all drilled in one process. The disadvantage being that all vias take up space on all layers. More complex structures are possible using blind or buried vias. The term 'blind via' refers to a via between a number of layers including, an outer layer, but not through all layers. The term 'buried via' refers to a via between a number of layers, not including an outer layer and not through all layers.

The type of via structure used will have a mayor effect on the processes required to fabricate the PCB and consequently the cost of the finished board. Examples of these three different via types are shown in the PCB cross-section below.

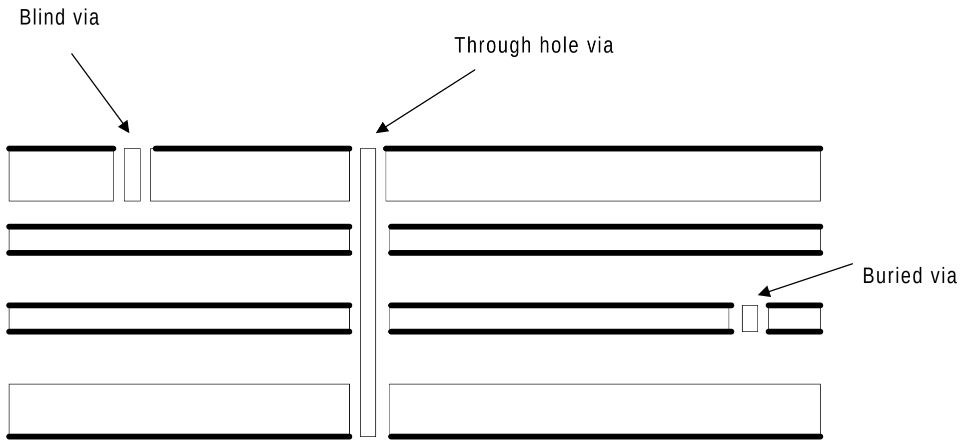

Fig 1. PCB cross-section showing different plated through hole types.

PCB Fabrication Limitations

In order to design a product using a PCB it is important to understand the practical limitations of the fabrication process. The manufacturing processes used to etch the tracks, drill the holes and assemble the laminates have limitations. Exceeding these limitations will significantly increase the cost of the PCB as manufacturing yields reduce.

The typical minimum limit on track thickness for a high volume, complex product is around 0.2 mm. A similar minimum limit applies to the minimum gap between two different conducts. (The larger the minimum track and gap width used, the lower the cost of the PCB). Some PCB manufacturers offer etched track widths down to 0.125 mm. These do however come at a premium and should only be specified if absolutely necessary and after detailed discussions with the chosen vendor. The typical tolerance on the width of etched tracks is +/- 0.025mm.

The minimum via hole size used will have a significant effect on the cost of the board as it limits the number of PCBs that can be drilled as a stack. Not withstanding the radio performance implications of small diameter via holes, typical minimum sizes of 0.25 mm are commonly used. A reduced cost will be achieved if the minimum hole diameter of 0.5 mm can be used.

A summary of typical process limitations is given in the table below for FR4 PCBs. Three arbitrary cost brackets have been chosen to highlight the range of tolerances. Actual available tolerances and the cost implications of different processes will vary with PCB vendor.

	Low cost radio product PCB.	**Medium cost radio product PCB.**	**High cost radio product PCB.**
Minimum track and gap	0.25 mm	0.2 mm	0.125 mm
Copper etch tolerance	+/- 0.05 mm	+/- 0.025 mm	+/- 0.01 mm
Smallest hole diameter	0.7 mm	0.5 mm	0.2 mm
Number of layers	2	4	6
Via structure	Through hole	Through hole and blind	Through hole, blind and buried
Board finish	Hot air solder levelled	Gold flash	Gold plated

Table 2. Summary of typical PCB process limitation for a range of cost options.

PCB Parasitic Elements

The term 'PCB parasitic' is used to refer to a physical attribute of the PCB that has an impact on the performance of the circuit. For example in a high frequency radio circuit it is simple to see how a long thin track will be inductive and a large pad over a ground plan will be capacitive. Consider, for instance, a surface mount 0805 size capacitor on a 0.25 mm thick double sided FR4 PCB above a ground plane. The equivalent high frequency circuit for the whole of the above structure can be described by the following circuit.

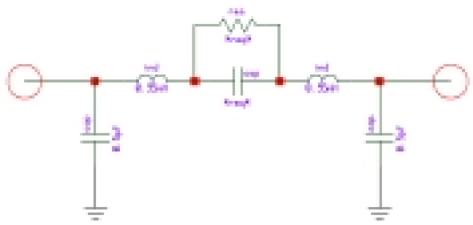

Fig 2. Equivalent circuit for an SMT capacitor mounted on an FR4 PCB with ground plane.

As can be seen in the diagram above, the simple capacitor now has a significant number of other elements attached to it in order to model both the device 'imperfections' and effects of the PCB. These include shunt capacitors to model the pad capacitance and two series inductors to model the device pad metalisation and PCB track inductance.

In addition to the parasitic elements described above, used to model the series capacitor, the impedance of the connections between circuit components and the ground plane must also be modelled in real circuits. A plated through via hole will have a significant inductance and good circuit simulation packages will include models to allow there addition. The typical inductance of a 1.6 mm long, 0.2 mm diameter via hole can be as much as 0.75 nH. This may seem very small, but as can be seen in the circuit example below, it can be significant.

Component Parasitic Elements

In a similar way to the PCB, components also have non ideal characteristics. While the use of surface mount devices helps to reduce the effect of this on radio circuit performance (principally due to the small construction and reduced lead length) it is significant.

The term component 'Q' or Quality factor is used as a figure of merit for a reactive device. The Q value is the ratio of the energy stored by the device to energy dissipated. Typical values for a wire-wound inductor would be of the order of 50 and for a ceramic COG dielectric capacitor of the order of 200 (the Q of a component is frequency dependent).

Ground Planes

A number of different ground plane strategies can be adopted for a radio product PCB. There is no unique answer to the best strategy for a given type of product. Some people are great advocates of split ground planes for analogue, digital, radio or audio circuits. Experience on a number of radio products at Plextek has shown that a single low impedance ground plane for all parts of the circuit is usually a good starting point. Often attempts to split the ground planes causes more problems than it solves. Careful consideration of the flow of currents throughout the product is essential to minimise digital interference with audio and radio circuits. Given the proliferation in the use of DSP and microprocessors in radio products this is a very significant issue.

Circuit Example

The potential significance of the parasitic PCB and component elements detailed above can be demonstrated with an example. Consider a radio designer who is charged with designing a low pass filter for use on the output of a power amplifier, used in the output stage of a transmitter, to reduce the out of band harmonic emissions.

Requirement:	Transmitter output power:	+30 dBm
	Third harmonic level (max) at Tx output:	-40 dBc
	Fifth harmonic level (max) at Tx output:	-50 dBc
	Required max out of band emissions:	-30 dBm
	Carrier frequency:	1 GHz
	Termination impedance:	50 Ω

Therefore, at least 20 dB of attenuation is required at the third harmonic frequency (3 GHz) and at least 10 dB at the fifth harmonic frequency (5 GHz).

Examination of standard curves shows that a 5th order Butterworth will give significantly greater attenuation than required (> 30 dB @ 3rd harmonic and > 50 dB @ 5th harmonic).

The circuit shown below is a 5th order Butterworth filter. The simulated ideal ‘parasitic free’ performance for this filter is also shown.

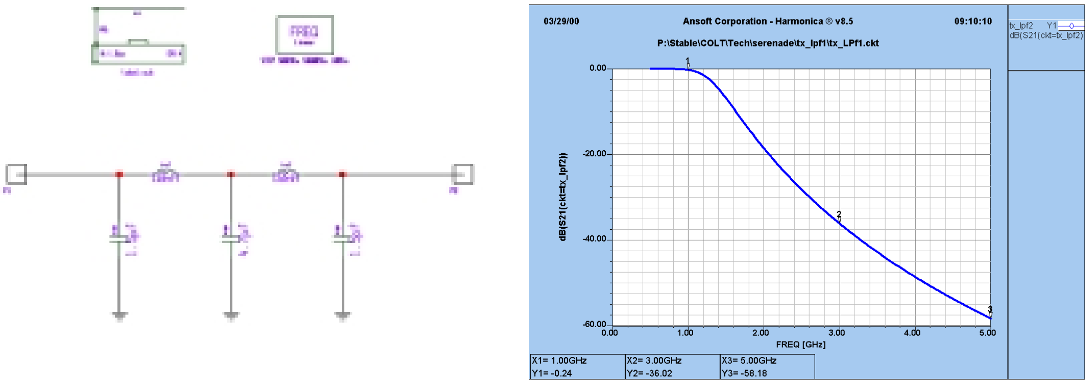

Fig 3. Ideal lumped 5th order Butterworth low pass filter with simulated response.

The revised circuit for the same 5th order Butterworth filter, including PCB and component parasitic elements is shown below along with the revised simulated performance. It can clearly be seen that the actual performance will be very different from the ideal case.

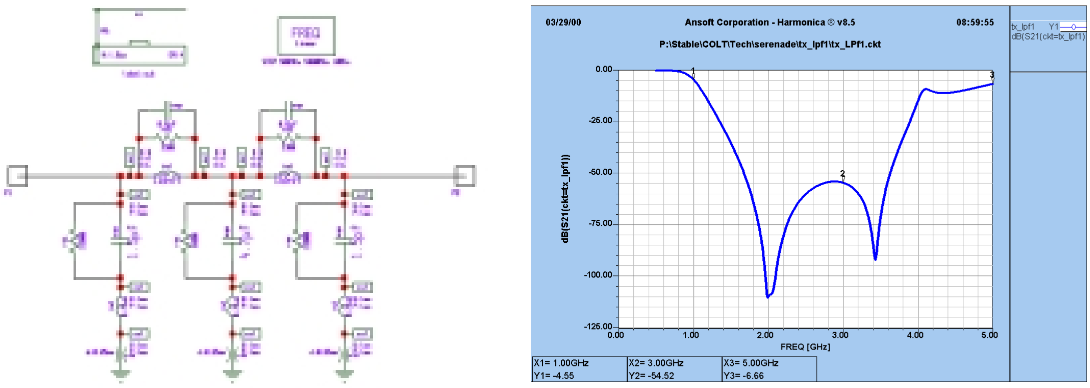

Fig 4. ‘Real’ lumped 5th order Butterworth low pass filter including PCB and component parasitics.

As can be seen from the two graphs, the 3rd harmonic rejection has improved by nearly 20 dB and the 5th harmonic rejection has been reduced by over 50 dB. Additionally the pass band loss has increased from 0.25 dB to 4.5 dB.

Summary

This paper has considered the attributes of the three base PCB materials commonly used for radio circuits. In addition to a choice of materials, the PCB designer also has the option to assembly the board from a number of potentially different material laminates of potentially different thickness. This can be very helpful in allowing a single board to accommodate dissimilar circuitry, for example baseband and radio or high frequency radio and analogue. As with the choice of plated through hole structure used, the designer must carefully consider the cost implications of hybrid and multilayer constructions.

PCB fabrication process limitations and PCB parasitic effects are discussed. For an optimum commercial radio product it is important for the PCB layout engineer and the circuit design engineer each understand both of these issues. Finally an example is presented to show the significant effect of lumped component and PCB parasitics on the performance of a radio circuit.

Two examples of commercial RF products designed by Plextek are shown below.

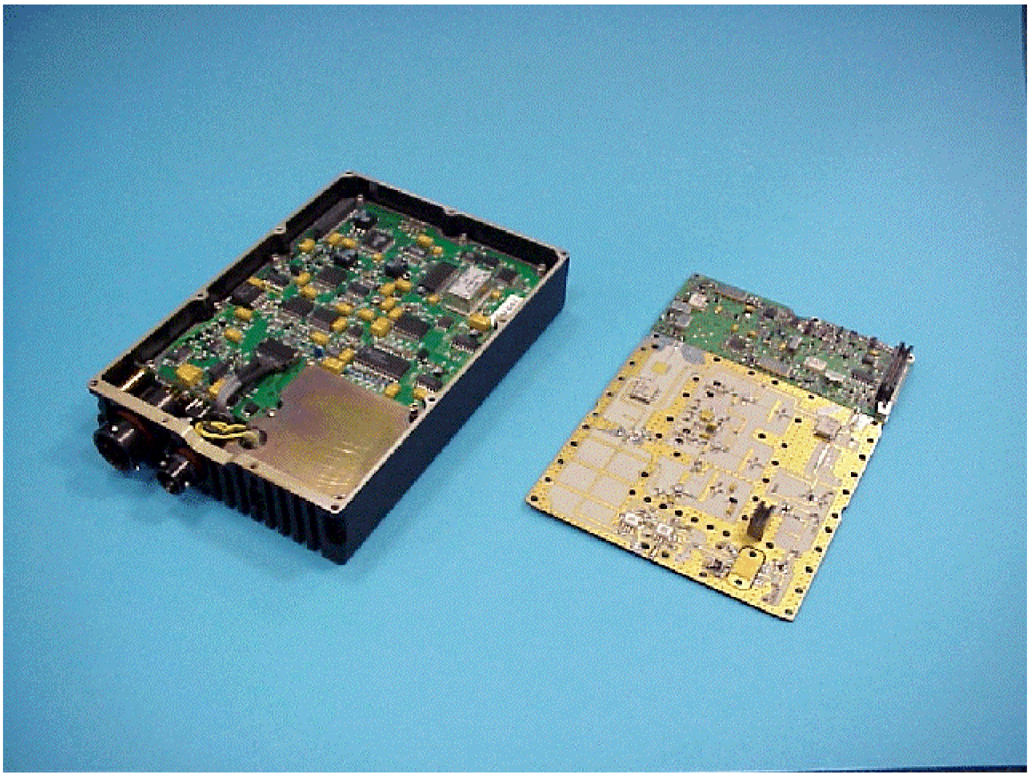

Photo 1. Formula 1 telemetry system using a hybrid Rogers R04003 / FR4 PCB.

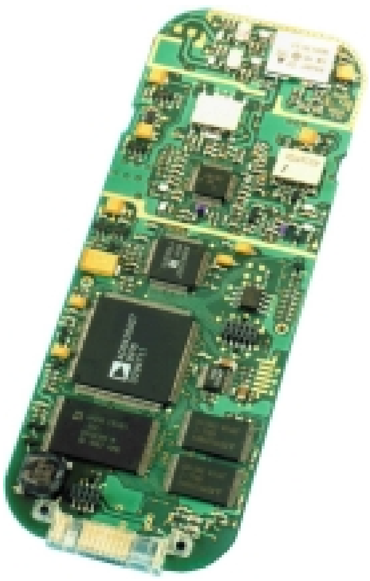

Photo 2. A six layer PCB with through and blind vias used in a GSM phone.