

# A Guide to the Design of Laminate PCBs at Microwave Frequencies

**Liam Devlin and Graham Pearson**

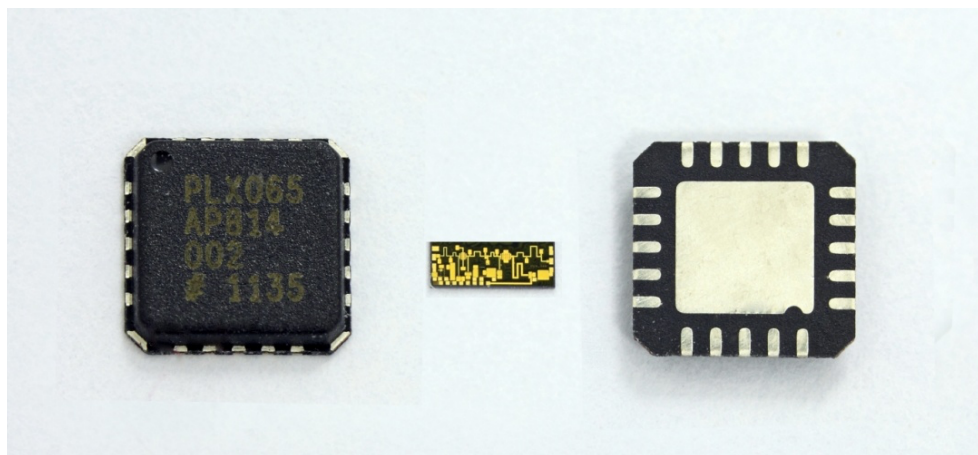
Plextek RF Integration, London Road, Great Chesterford, Essex, CB10 1NY, UK;  
(liam.devlin@plextekRFI.com)

## Abstract

Laminate PCB technology is now routinely used for the realisation of low cost subsystems operating at frequencies up to around 40GHz. The wide availability of both microwave components in SMT compatible packages and low cost, high performance laminate PCB materials have made this possible. Although the use of microwave laminate PCBs is now common, careful design and attention to detail is still vital in order to avoid serious performance issues. This paper presents guidelines and useful tips for optimising the performance of SMT based laminate PCBs operating at microwave frequencies.

## Using SMT Packaged Microwave Frequency ICs

The mass market for consumer wireless products has resulted in huge strides in low cost packaging technology. The drive for miniaturisation has helped reduce package parasitics and this, together with careful co-design of ICs and packages, has allowed the upper operating frequency of SMT packaged ICs to be pushed to around 40GHz. The most popular package style for microwave ICs is the QFN (Quad Flat pack No-leads). The photograph in Figure 1 shows a bare die microwave amplifier MMIC and both faces of the QFN package for which it was designed. The exposed paddle on the underside of the package is normally the ground connection and is connected to the backside of the die. It is clear that the use of the bare die still offers the ultimate in size reduction but the use of an SMT packaged component means that assembly and handling is comparatively straight forward.



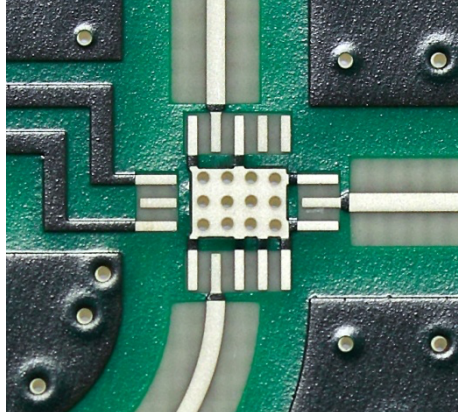
**Figure 1: Microwave amplifier IC, QFN package and bare die**

When using microwave ICs, it is vital to have adequately low grounding inductance. The ground inductance acts as series inductive feedback around the IC. If it becomes too high it can seriously degrade performance and can even cause instability. With most commercially available QFN packaged microwave ICs, the exposed paddle on which the die is mounted is solid metal. This gives both a low grounding inductance and a low thermal impedance. However, when the packaged part is mounted onto a PCB, the top substrate layer of the PCB is normally the microwave substrate and the backside of this layer is the microwave ground. The package base is connected to the microwave ground using through substrate vias, which increase both the grounding inductance and the thermal impedance. The PCB grounding inductance tends to dominate the overall grounding inductance of the IC.

There are two main approaches to reducing grounding inductance:

- Use an array of closely spaced vias to ground the package base
- Select a thin substrate height to reduce the inductance of the individual vias

Figure 2 is a photograph of a PCB designed for attachment of a QFN packaged microwave IC. The array of closely spaced vias in the package base area can clearly be seen. It is important that the pitch between the vias is electrically small otherwise coupling between vias can give rise to resonances. The vias in the photograph are plated vias. These can be filled to minimise potential yield issues during assembly. However, this approach increases the PCB cost. It is possible to avoid filling the vias and to maintain high yields but this requires careful optimisation of the PCB layout and assembly processes.

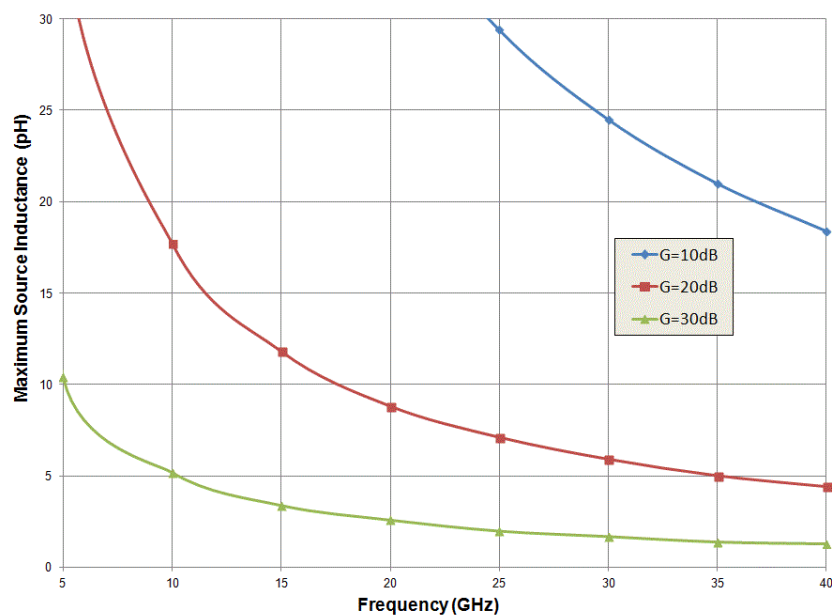


**Figure 2: PCB footprint for QFN microwave IC attachment**

When trying to determine whether the planned grounding approach for a microwave IC is adequate, two questions arise:

- What level of grounding inductance can be tolerated?
- How do I estimate the grounding inductance of the PCB?

The grounding inductance that can be tolerated decreases with increasing gain and increasing frequency. The graph in Figure 3 shows three curves of maximum grounding inductance versus frequency. The curves are for gain levels of 10dB, 20dB and 30dB. They were produced by analysing the effect of grounding inductance on a near ideal amplifier (100dB return losses and 200dB isolation). Simulations were undertaken to determine the level of grounding inductance that would cause the near perfect return loss to degrade to 20dB.



**Figure 3: Maximum grounding inductance versus frequency for different gains**

The traces plotted in Figure 3 indicate the recommended maximum grounding inductance. Catastrophic degradation may not occur for grounding inductances immediately above these levels; this is the total grounding inductance that can be tolerated with relatively modest performance degradation. It can be seen that increasing frequency and increasing gain both reduce the level of grounding inductance that can be tolerated. With passive ICs or frequency converting ICs the grounding inductance for the 10dB trace can be used.

The array of PCB vias connecting the exposed paddle of the package to the PCB ground usually accounts for the majority of the grounding inductance. It is possible to estimate the grounding inductance of the array of vias and guidelines on doing this are provided below. Whilst this is clearly an approximation, at frequencies where the QFN exposed paddle is a fraction of a wavelength ( $<\lambda/2$ , which is 5mm at 30GHz) and the inductance is low, the approximation seems to work reasonably well. For frequencies above this, or if improved accuracy is desired, full 3D EM simulation can be used.

A first order approximation to the level of grounding inductance that can be expected can be obtained using the following simple approach:

- $L_{\text{via}}$  (inductance of a single PCB via)  $\approx 0.4\text{nH}$  per mm of substrate thickness
- For an array of  $N$  closely spaced vias the effective inductance is approximately  $1.5 \cdot L_{\text{via}}/N$

By way of example consider a 0.008" thick substrate (a popular choice for microwave PCBs using laminate PCB materials). The value of  $L_{\text{via}}$  would be around 80pH. If an array of 16 vias were used in the ground paddle, the total inductance would be approximately 7.5pH. From the curves of maximum inductance above it can be seen that this would be adequate for a 20dB gain amplifier operating at frequencies up to 24GHz. As previously mentioned this approach offers guidelines rather than exact figures but it should provide a useful indication of whether grounding inductance is low enough to avoid any potential problems.

A more detailed description of SMT packaging at microwave frequencies can be found in [1].

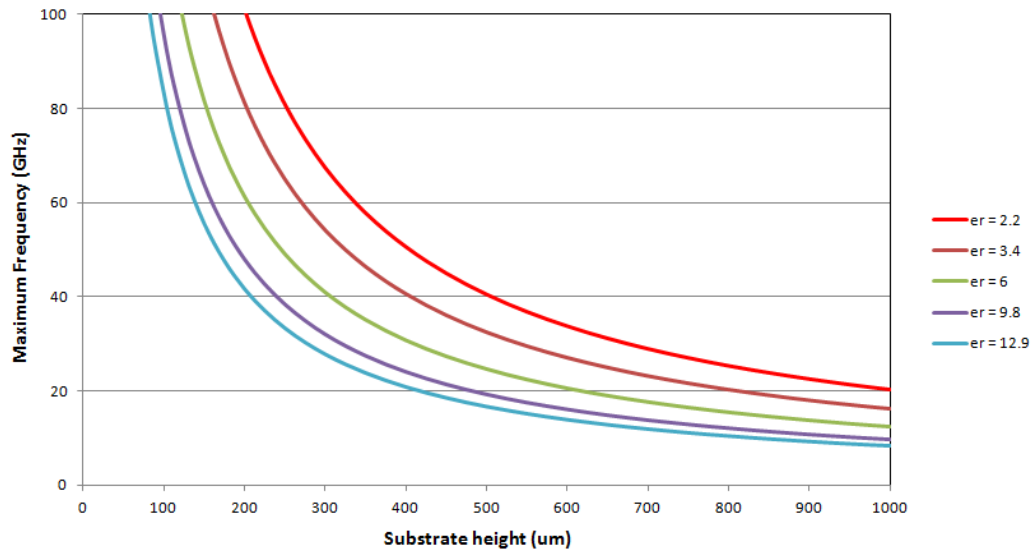
## Laminate PCB Materials for use at Microwave Frequencies

Low cost laminate PCB materials with well controlled dielectric constant and low loss (low  $\tan\delta$ ) are available from a number of manufacturers. These materials are well suited to volume production and typically have a dielectric constant ( $\epsilon_r$ ) of around 3.5. Microstrip and Grounded Coplanar Waveguide (GCPW) transmission lines are dispersive at high frequencies, which means that radiation increases and other propagation modes (e.g. substrate modes and transverse resonance modes) can start to have a significant effect. In order to ensure that the dispersive effects are modest, the maximum substrate thickness should be less than one tenth of a wavelength ( $\lambda/10$ ) in the substrate material at the highest frequency of interest.

If the same material is considered, with a substrate thickness of 0.02" (0.508mm), then the maximum recommended frequency, using the guideline above, is 31.5GHz. However, even for ICs operating at frequencies lower than this, IC vendors often recommend a thinner substrate height, with 0.008" (~0.2mm) being common. The reasons for this are that this substrate height is readily available and the inductance of through substrate vias will be lower, so easing the problems of IC ground inductance discussed above. In terms of avoiding the effects of dispersion, a substrate with an  $\epsilon_r$  of 3.5 and a thickness of 0.008" could be used to around 79GHz.

Figure 4 plots the recommended upper operating frequency against substrate thickness for a range of commonly used dielectric constants.

Laminate PCBs are offered with different "weights" of copper cladding. A weight of 0.5oz equates to 18 $\mu\text{m}$  metal thickness, 1oz equates to 35 $\mu\text{m}$  and so on. It is obviously important to select a metal thickness that is adequate to handle the required DC current. The DC current carrying capability is also affected by the choice of substrate material and the ambient operating temperature.

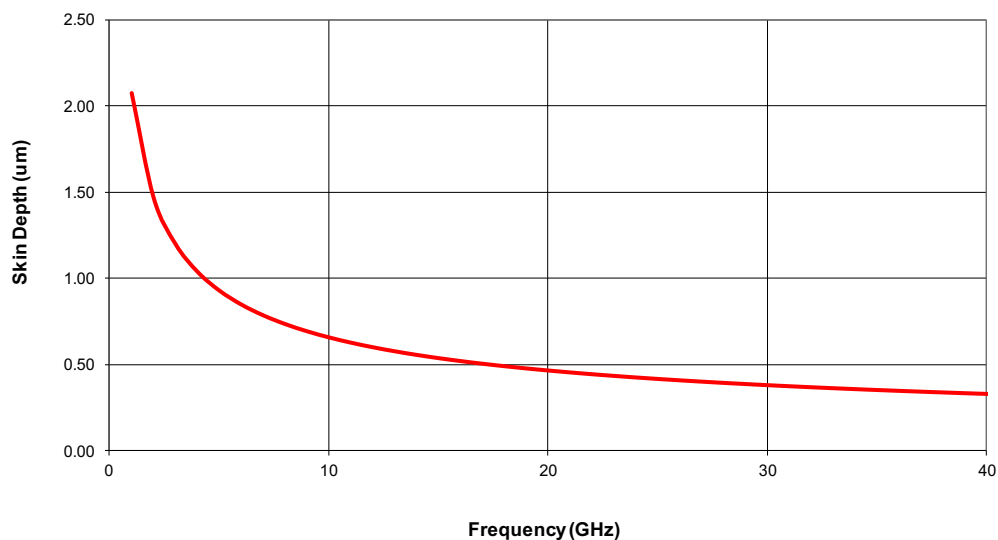


**Figure 4: Recommended upper operating frequency of different substrates**

Metallisation thickness also affects RF insertion loss. At low frequencies increasing the metal thickness will reduce the loss. However at higher RF and microwave frequencies this is not always the case because of skin effect. This is the tendency for RF current flow to be concentrated closer to the surface of the conductor. Skin depth is defined as the thickness of metal through which 63% of the RF current flows. The graph in Figure 5 shows the skin depth versus frequency for copper. Once the metal thickness is more than a few skin depths, there is little to be gained in reducing loss by increasing metal thickness. However, there are a number of other advantages to be had from using a thinner metallisation:

- Track definition will be better
- Material cost will be lower (less copper)
- Processing (etching) time will be reduced and so processing costs lower

The first of these points will allow better control of transmission line impedances and pad parasitics; it is an important consideration if structures such coupled line filters (discussed in more detail below) are to be included. The other two points are only really significant for PCBs intended for volume production.



**Figure 5: Skin depth versus frequency for copper**

In the case of PCBs for very high power amplifiers both the metallisation and the substrate material must be selected carefully. An inappropriate choice can result in vaporisation of tracks due to excessive RF power flow.

The copper tracks are generally covered so as to stop corrosion and/or oxidation. The covering generally takes the form of ENIG (Electroless Nickel Immersion Gold) for low frequency circuits where the loss due to the Nickel barrier layer is not such an issue. However, at higher frequencies the presence of the Nickel can cause significant losses. One alternative is a silver finish but this may eventually tarnish which could be an issue. PCBs usually include a solder resist layer to define the apertures for the solder paste for SMT components. Solder resist is often omitted from the RF traces to avoid it affecting the loss and dielectric constant.

Surface roughness is another issue of potential concern. The term refers to the small scale fluctuations from perfect flatness of the PCB material at the dielectric-metal interface. It has the effect of increasing the insertion loss of routing tracks and printed components such as filters. Its significance increases with increasing frequency. For electrically small interconnects it has little impact but for electrically long structures its effects can be significant and it must be taken into account. Figure 6 shows the simulated insertion loss of a 50mm length of 50Ω transmission line on a 0.008" thick laminate substrate. The red trace ignores surface roughness whilst the blue trace includes the effect of surface roughness of 1.7μm (this is the RMS surface roughness). The additional loss reaches 0.5dB at 30GHz.

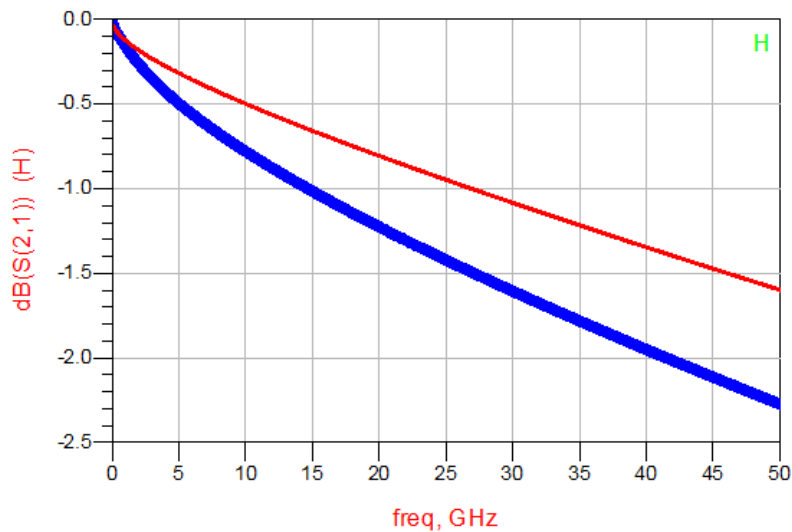


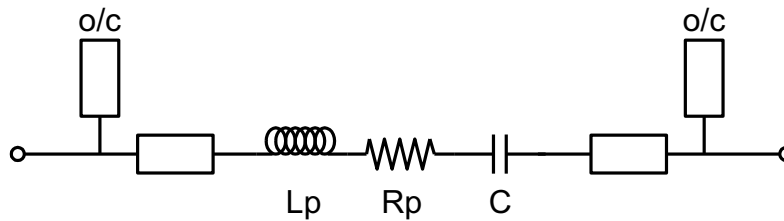
Figure 6: The effects of 1.7μm surface roughness on the loss of a 50mm long line

## Discrete SMT Passive Components

The drive to produce more compact electronic products has caused a continued reduction in the physical size of discrete resistors, capacitors and inductors (Rs, Cs and Ls). Component sizes of 0402 and 0201 are now in common use and even smaller sizes, such as 01005, are also available although less commonly used. One of the consequences of the reduced size is reduced component parasitics, which facilitates their use at higher frequencies.

The main parasitic component of a resistor or capacitor is an effective series inductance. For 0402 components it is around 0.4nH and for 0201 components around 0.25nH. As long as this inductance is accounted for in the design process, discrete Rs and Cs can be used for biasing, matching and filtering at frequencies in to the microwave range. In the case of DC blocks, for example, the series inductive element of the discrete capacitor can be absorbed into a low pass filter with the shunt Cs realised as printed open-circuit stubs. This allows the realisation of low-cost, compact DC blocks at high microwave frequencies.

The PCB pads required for attachment of the discrete parts also have a parasitic effect and should be included to accurately simulate microwave frequency performance. Figure 7 shows the equivalent circuit for an SMT capacitor mounted on a PCB. As well as the parasitic series inductance ( $L_p$ ) it includes an equivalent series resistance ( $R_p$ ) to model the capacitor losses. In many instances the effect of  $R_p$  will be modest and can be neglected. However, when used at low impedance points (for example in PA matching networks) its impact can be significant.

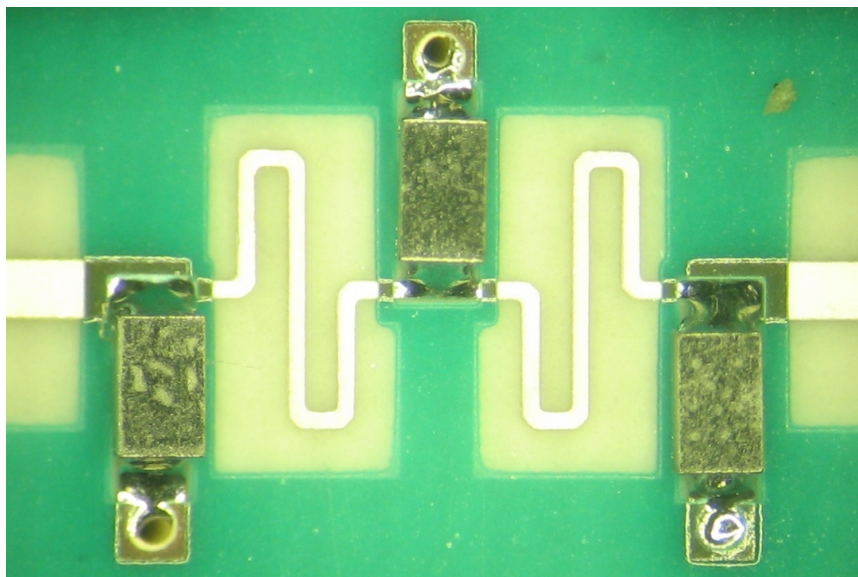


**Figure 7: Equivalent circuit of an SMT capacitor mounted on a PCB**

With chip inductors the main parasitic, in addition to the resistive losses of the inductor, is a parallel capacitance. The effect of this is to cause the inductor to become open-circuit resonant at a particular frequency (the self resonant frequency). This limits the usefulness of discrete inductors at microwave frequencies. It is possible to make use of the open circuit resonance to provide a high impedance, narrow-band bias choke or to provide a rejection notch in a filter. However at frequencies above the self resonance the inductor looks more capacitive than inductive and discrete SMT inductors are not often used at microwave frequencies. One exception is conical chokes which are used as broadband bias chokes.

## Printed Components

Although the metallisation pattern on laminate PCBs is not printed (it is defined photo-lithographically) this term is used to describe components formed using the metallisation pattern. Shunt open circuit stubs can be used as shunt capacitors, high impedance (narrow) transmission lines can be used in place of series inductors and shunt short-circuit stubs can be used as shunt inductors. This approach allows the realisation of filters and matching structures. The photograph in Figure 8 shows a low pass filter realised using a combination of discrete capacitors for the shunt elements and high impedance transmission lines for the series inductive elements. Openings in the resist layer are evident around the series RF tracks. The inductive parasitics of the shunt capacitors and associated grounding vias mean that the filter rejection of this structure will degrade at higher frequencies. However, this is easily improved by the addition of open circuit stubs.

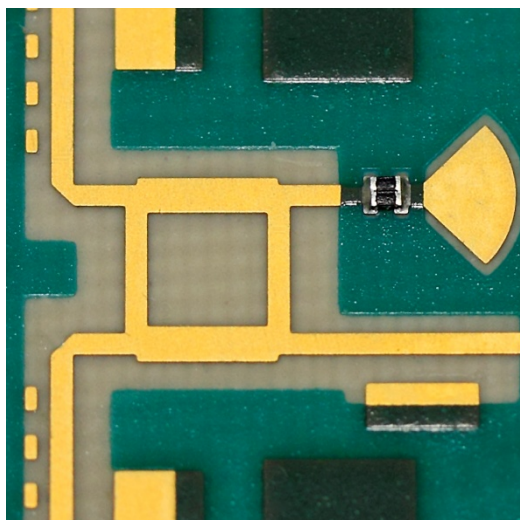


**Figure 8: Photograph of a mixed lumped-distributed filter**

Various forms of power splitters and combiners can be realised with printed structures, the ubiquitous Wilkinson divider being one of the most common. For broadband quadrature splitters the Lange coupler offers good performance but the required track widths and spacings mean that these are not practical using laminate PCB processing (they also require cross-overs to link fingers, which add a degree of complexity). Branch-line couplers have less demanding feature requirements. They also have lower loss than Lange couplers but cover a more modest bandwidth. The photograph in Figure 9 shows a branch-line coupler operating at around 15GHz. The resist coating has been omitted from the RF tracking. A

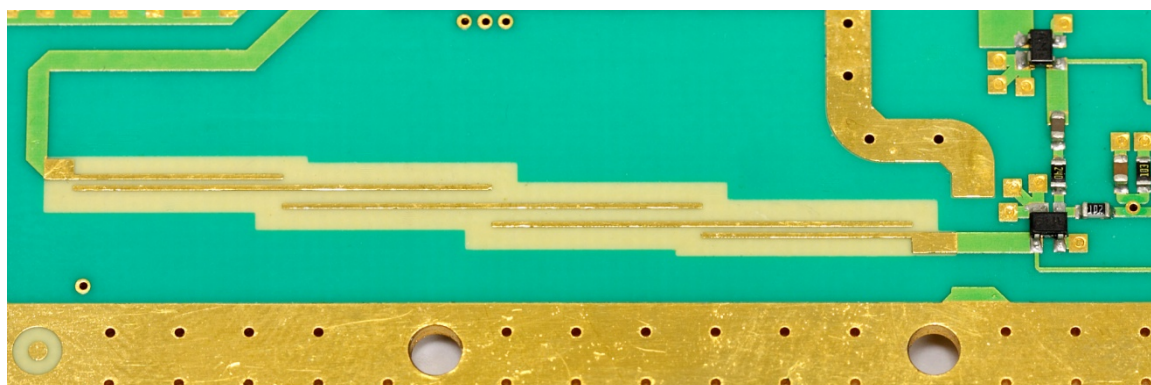


radial stub has been used to provide an RF ground for the termination resistor at the isolated port. This avoids the additional inductance that would be present if through substrate vias were used to provide the RF ground.



**Figure 9: Photograph of a branch-line coupler**

Although it is not practical to realise Lange couplers using laminate PCB technology, other edge coupled structures, such as directional power samplers and coupled line filters, can be realised. The photograph in Figure 10 shows a coupled line filter realised on a laminate PCB. Other coupled line filters, such as interdigital, comb-line and hairpin, can also be realised. However, it is important to understand the line width (and spacing) tolerances that will be achievable in production and to design filters with adequate bandwidth to ensure low pass-band insertion loss with unit to unit production. This means that high rejection close to the pass band is not really practical and other technologies must be considered if this is required.



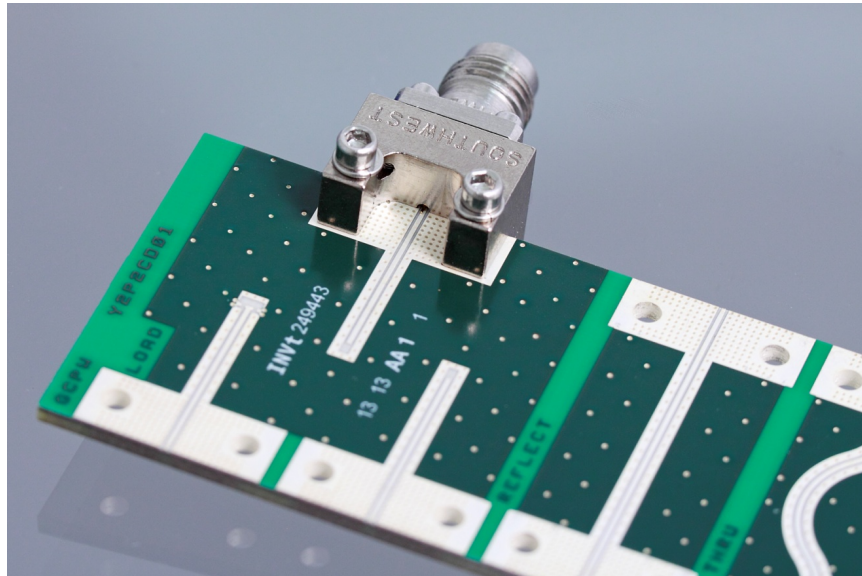
**Figure 10: Photograph of a coupled line filter**

## Co-axial Connectors

Co-axial connectors are the most commonly used interface to microwave PCBs. Although connectors claiming performance to 50GHz and above are now readily available, care and attention to detail is required to ensure acceptable performance to such high frequencies. The real key is ensuring good ground plane continuity. The connection between the outer of the co-axial connector and the PCB ground must be continuous, in close proximity to the co-axial connector and free from any structures that could cause parasitic resonances. This is true whether an edge mounted connector is used or the PCB is assembled into a housing.

A number of manufacturers now offer high performance edge-mounted co-axial connectors suitable for use to 40GHz and beyond. These normally have a clamping mechanism to ensure a good connection between the co-axial outer and the PCB ground. The photograph in Figure 11 shows an example of such

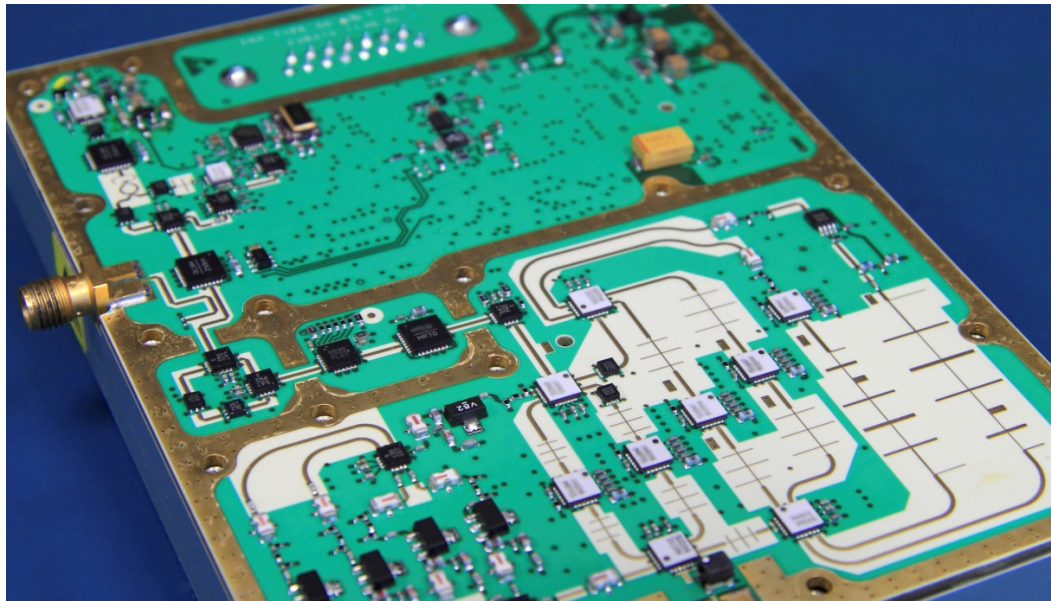
an edge mounted connector interfacing to an appropriately designed PCB. The high density of PCB grounds at the connector to PCB connection point is clearly visible. In this instance the connector is mated to the open standard of a set of PCB TRL calibration standards used to allow calibration to the reference planes of a PCB mounted IC.



**Figure 11: Photograph of an edge-mounted co-axial connector interfacing to a microwave PCB**

## **Concluding Remarks**

Laminate PCB technology with SMT components is suitable for use to high microwave frequencies; an example is shown in Figure 12. Volume production costs can be low and there are numerous PCB manufacturing facilities and assembly houses capable of producing such assemblies. However, there are still many pitfalls for the inexperienced and good design practises and attention to detail are essential if optimum performance is to be achieved.



**Figure 12: Photograph of a laminate microwave PCB**

## **References**

- [1] Liam Devlin, "A Guide to SMT Packaging of Microwave ICs", proceedings of the RF and Microwave Society (ARMMS) Conference, November 19<sup>th</sup> and 20<sup>th</sup>, 2012