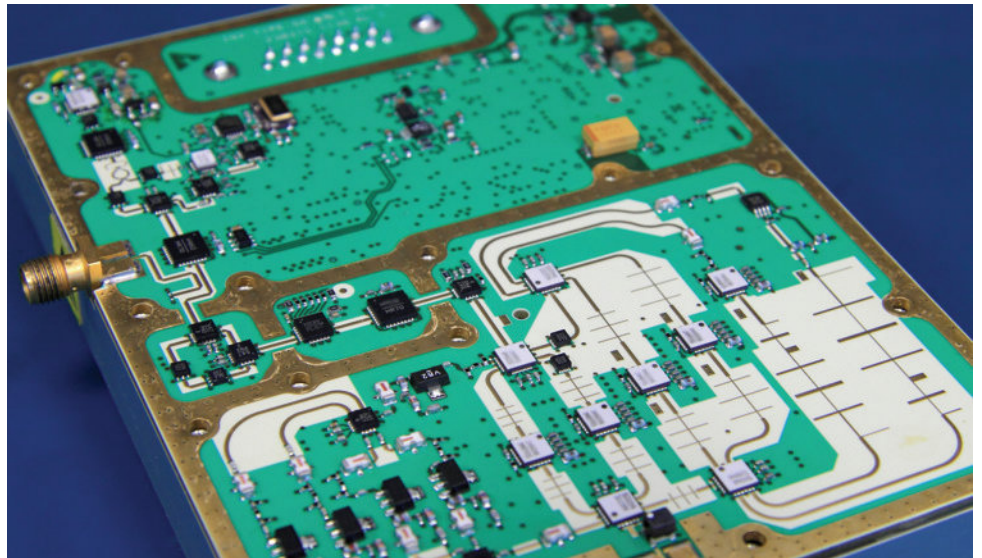


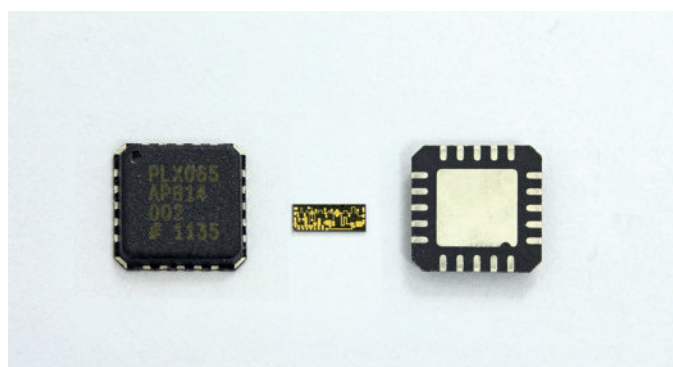
Technology Overview - Designing Laminate PCBs at Microwave Frequencies

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 high performance, low cost laminate PCB materials have made this possible. However, careful design and attention to detail is vital in order to avoid serious performance issues at microwave frequencies. This technology overview presents guidelines and useful tips for optimising the performance of SMT based laminate PCBs operating at microwave frequencies.

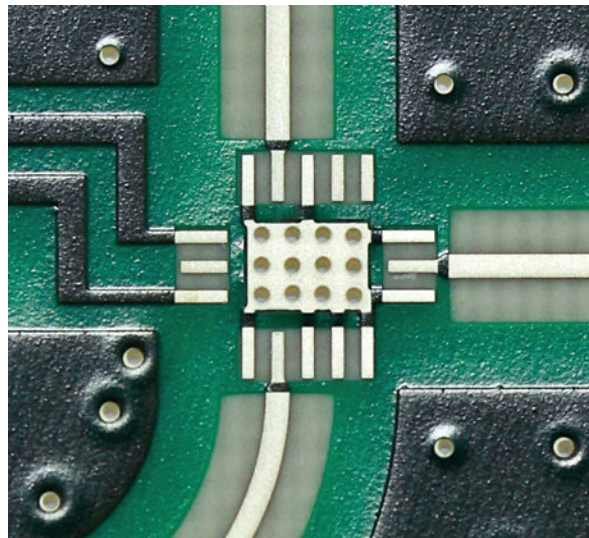


The mass market for consumer wireless products has resulted in huge strides in low cost packaging technology. The drive for miniaturisation has helped reduce package parasitics and this, together with careful co-design of ICs and packages, has allowed the upper operating frequency of SMT packaged ICs to be pushed to around 40GHz. The most popular package style for microwave ICs is the QFN (Quad Flat pack No-leads).

The photograph below 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.



Microwave amplifier IC, QFN packaged and bare die



PCB footprint for QFN microwave IC attachment

Grounding Inductance:

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 an improved thermal impedance. However, when the packaged part is mounted onto a PCB, the top 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:

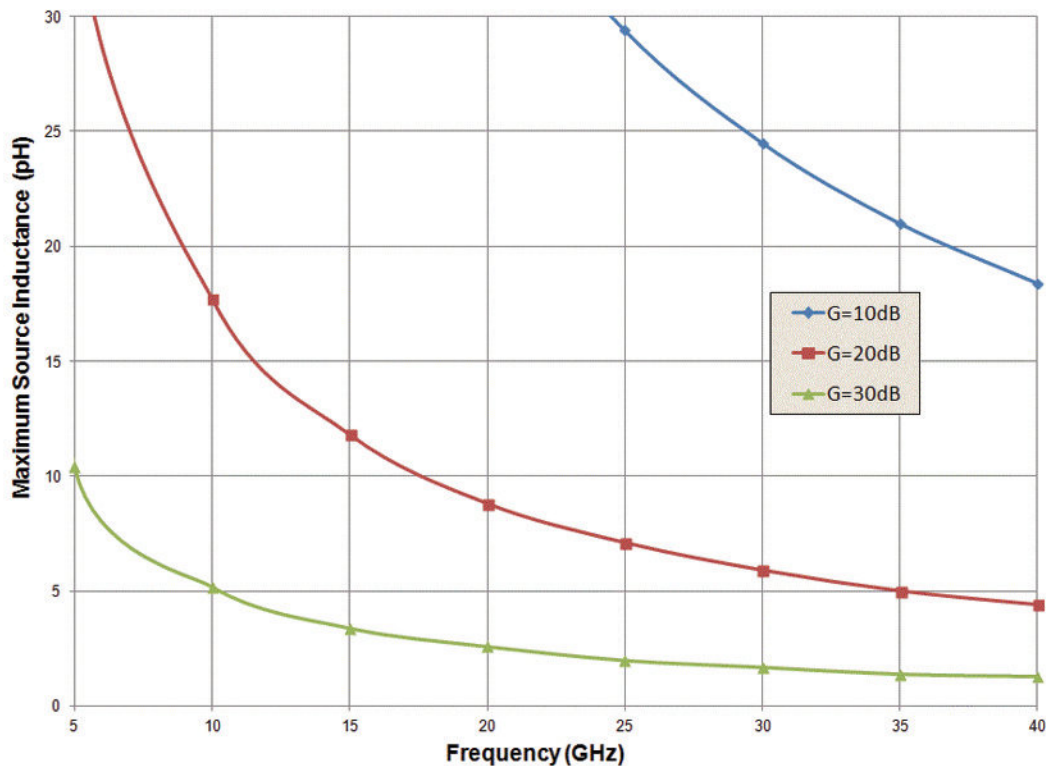
- Use an array of closely spaced vias to ground the package base
- Select a thin substrate height to reduce the inductance of the individual vias

The image above shows a photograph of a PCB design 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.

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 below shows 3 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 near ideal amplifiers (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.



Maximum grounding inductance versus frequency for different gains

The traces plotted above 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 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 $< 5\text{mm}$ 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 by following the simple approach:

- L_{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 $1.5 * L_{via} / N$

By way of example consider a 0.008" thick substrate (a popular choice for microwave PCBs using laminate PCB materials). The value of L_{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 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.

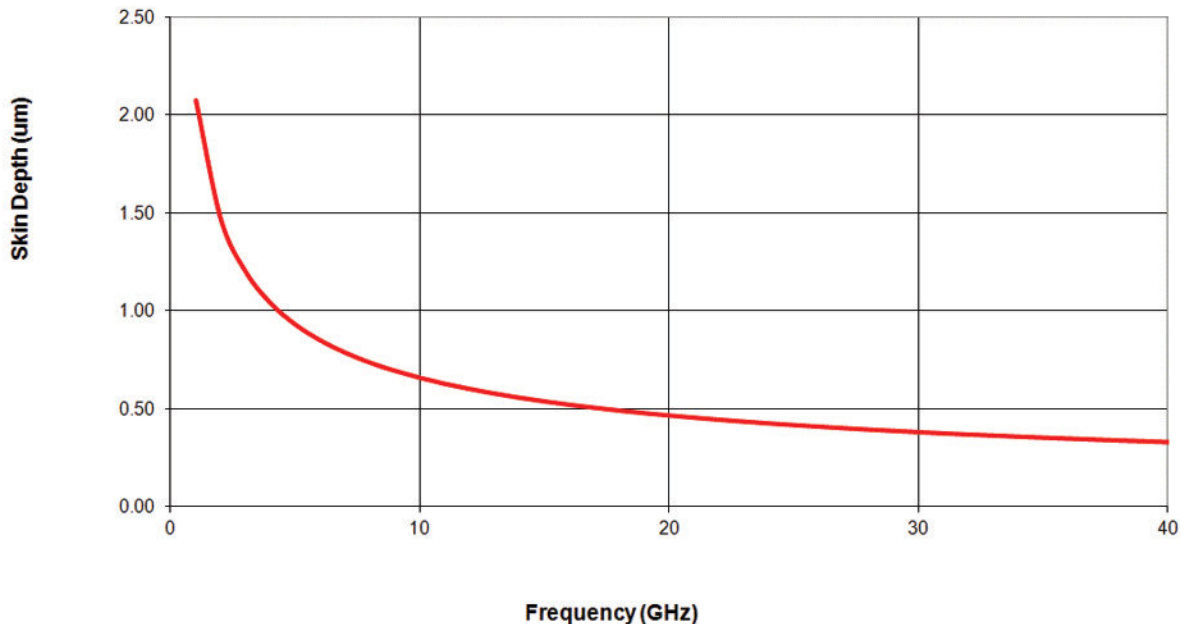
Substrate Selection:

Low cost laminate PCB materials with well controlled dielectric constant and low loss (low $\tan\delta$) are available from a number of manufacturers. These are well suited to volume production and typically have a dielectric constant (ϵ_r) of around 3.5. Microstrip or Grounded Coplanar Waveguide (GCPW) transmission lines are dispersive at high frequencies; radiation increases and other propagation modes (e.g. substrate modes and transverse resonance modes) start to have a significant effect. In order to ensure that 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 ϵ_r of 3.5 and a thickness of 0.008" could be used to around 79GHz.

Metallisation:

The laminate PCBs are offered with different "weights" of copper cladding. A weight of 0.5oz equates to 17.5 μm metal thickness, 1oz equates to 35 μm 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.



Skin depth versus frequency for copper



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 above shows the skin depth versus frequency for copper. Once the metal thickness is more than several 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 as 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.

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

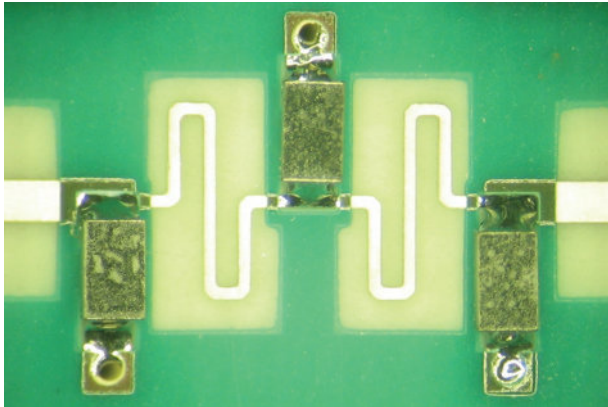
The tracks are generally covered so as to stop corrosion and/or oxidation of the copper. This generally takes the form of ENIG (Electroless Nickel Immersion Gold) for low frequency circuits where the loss of the Nickel barrier layer is not such an issue. However, at higher frequencies the presence of the Nickel can cause additional losses. An 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 effective dielectric constant.

Discrete Passives:

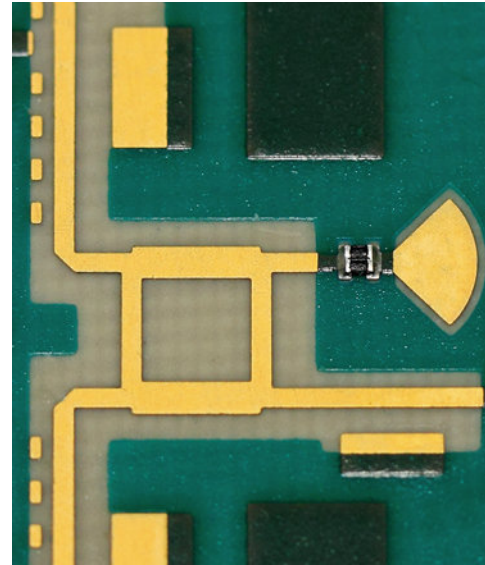
The drive to produce more compact electronic products has caused a continued reduction in the physical size of discrete resistors, capacitors and inductors (Rs, Cs and Ls). Component sizes of 0402 and 0201 are now in common use and even smaller sizes, such as 01005, are also available although less commonly 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 up 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.

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. 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.



Mixed lumped-distributed filter



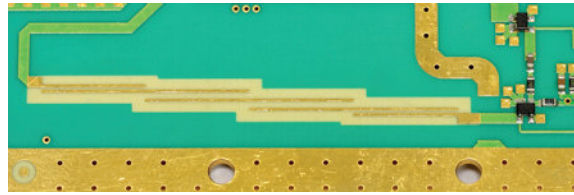
Branch-line coupler

Printed Components:

Although the metallisation 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 above left 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. 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.

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 is popular 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 above right 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.

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 below 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 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.



Coupled line filter

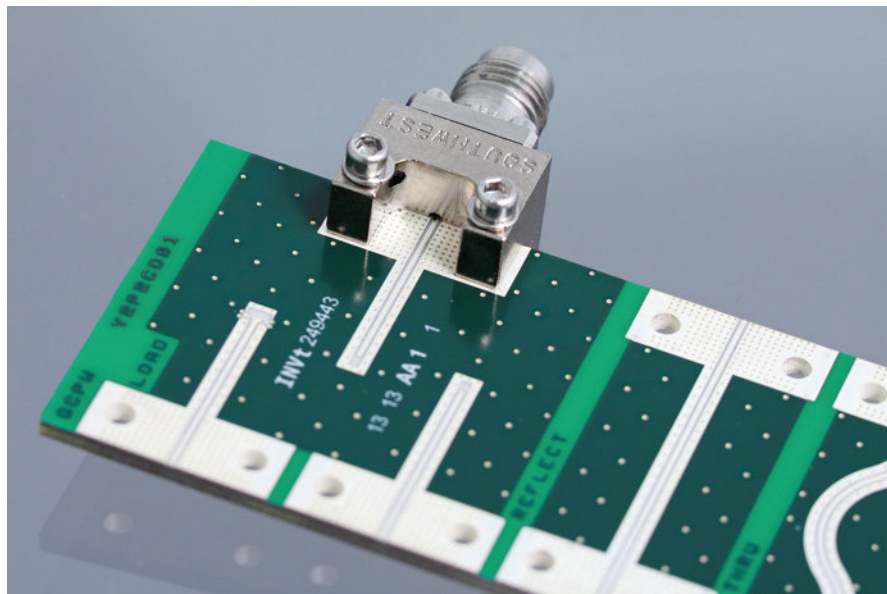
Co-axial Connectors:

Co-axial connectors are the most commonly used interface to microwave PCBs. Whilst 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 below 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.

Concluding Remarks:

Laminate PCB technology with SMT components is suitable for use to high microwave frequencies. 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.



Edge-mounted co-axial connector interfacing to a microwave PCB