

High Density RF Loadboard Design

Martin Dresler and Oscar Solano
Verigy, Boeblingen, Germany

Heidi Barnes
Verigy, Cupertino, USA

Abstract

The current success of smartphones and the relentless trend to reduce cost and add capabilities every year are key drivers in the wireless semiconductor business. Combining various RF technologies into one device along with the desire for multi-site testing can easily increase the RF ATE test fixture port count to a range of 48 and beyond. While a few ATE vendors provide test equipment for this requirement, the question remains on how to manage the interfacing and the test fixture design for high density RF without losing performance. The current interfacing and layout paradigm will undergo a significant change. The RF connectors are moved further away from the DUT to allow easier assembly and debugging, but with higher risk of performance degradation due to loss, crosstalk and interference. This paper evaluates new hardware and layout techniques, discusses mechanical, performance and crosstalk requirements that need to be applied and describes the consequences this will have for loadboard design.

1. Introduction

The current success of smartphones and their trend toward reducing cost and increasing capability every year are key drivers in the wireless semiconductor business. While 2G transceivers require up to 7 RF pins, 3G transceivers may require 13 or more RF pins. Combining various RF technologies like Cellular, Bluetooth, GPS, FM and WLAN into one device will easily drive the RF multi-site port count to 64 and beyond in the next 3 to 4 years [4]. While a few ATE vendors provide test equipment for this requirement, the question remains on how best to manage the interfacing and the DUT (device under test) board layout.

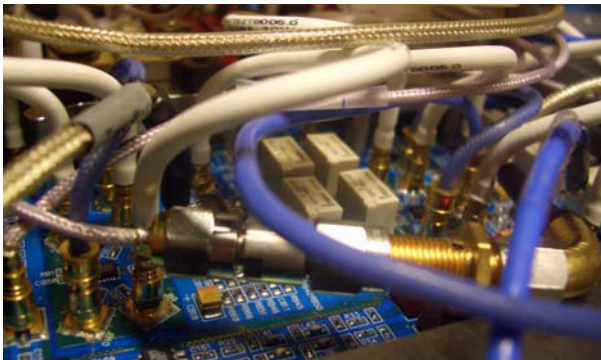


Figure 1: RF cable congestion of a 20+ RF pin project.

ATE vendors typically connect their tester RF hardware with blind mate connectors to a loadboard RF interface and from there with coax cables via SMP or SMA type of connectors to the loadboard. To avoid RF crosstalk and to achieve best performance, the connectors need to be placed as close as possible to the DUT [1]. This type of RF interfacing quickly consumes all of the useable component space surrounding the DUT and results in a struggle to achieve the same layout topology that is used on lower density devices.

This type of interfacing works fine for low RF pin count devices, but will it also succeed for 64+ in the near future? Figure 1 clearly illustrates the problem. Even for projects with less than 24 RF pins, RF interfacing and layout is challenging.

An alternative approach is to move the RF connectors further away from the DUT to allow easier loadboard assembly, debugging and tuning, but with higher risk of performance degradation due to loss, crosstalk and interference. Obviously, the current interfacing and layout paradigm of short microstrip RF traces with tuning components next to the DUT will undergo a significant change.

This paper evaluates new hardware and layout techniques to determine the key mechanical and electrical performance criteria that apply to this high density routing challenge and to identify the consequences this will have for loadboard design.

2. Mechanical Requirements

2.1. Connectors

One obvious way to deal with the high RF pin count challenge is to use smaller connectors. This has already happened on existing high port count systems. The trend of the last few years was to move from SMA to SMP and further to Mini-SMP connectors. Each step approximately reduces the size by 30% [6]. However, this will not be sufficient for a 48+ RF port

solution. The next step in size reduction can be achieved by ganging multiple connectors together, thus sharing the mechanical structure and eliminating wasted connector to connector spacing. As illustrated in Figure 3 (red circle), the Mini-Coax ganged connector has just the right size to still allow controlled via or fencing structures between neighbor pins while applying standard PCB fabrication processes.

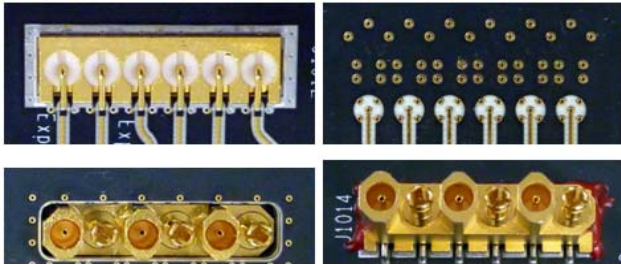


Figure 2: Reverse mount through a hole vs. normal surface mount ganged x6 Mini-Coax connector seen from board top and bottom [6].

reduce the blocking of the digital traces.

It is very important to ensure a repeatable and durable connection in production. The more single connectors and cables are used, the higher the risk of connector issues which would result in degraded electrical performance and even test escapes. Ganged connectors increase the mechanical robustness and reduce the chance of potential damage due to off-axis forces.

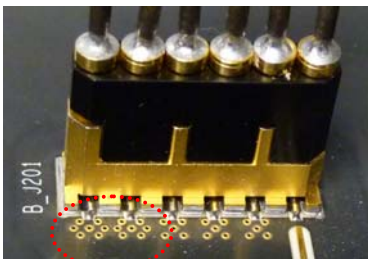


Figure 3: Ganged x6 Mini-Coax connector

Opening the connection requires a relatively high force. The maximum disengagement force is typically in the range of a single Mini-SMP full detent connector (max. 29 N). Engagement force typically correlates to a single smooth bore Mini-SMP (max. 11 N). In summary, the connector is easy to engage and still makes a solid connection without risk of unintentional or little by little release. Peeling off the whole connector during disengagement of the cable connections is far less likely than existing SMP connectors. The solder area is approximately 3 times larger, the ganged housing minimizes off-axis forces and the small diameter flexible cabling translates minimal force to the connector.

According to the specification, the mini-Coax ganged connector can handle more than 500 mating cycles similar to legacy SMA connectors. Some mini-SMP connectors are typically specified for not more than 100 mating cycles [6]. Overall, the ganged Mini-Coax connector is mechanically superior to the Mini-SMP connector.

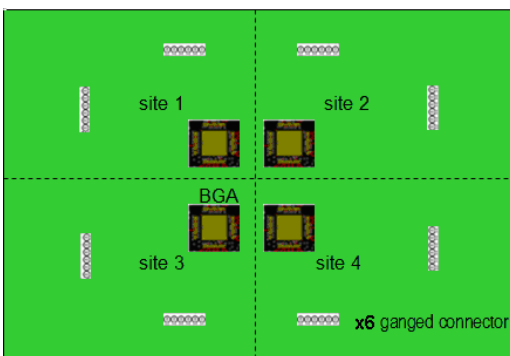


Figure 4: Quad site loadboard configuration with 48 RF ports and 8 x6 ganged RF connectors.

The next aspect regarding high density load boards is where to locate these mini-Coax ganged connectors. Looking at a quad site loadboard with 12 RF pins per site and assuming that most RF connections need tuning elements, it is quite obvious that it is impossible to locate all connectors and tuning components close to the DUT (Figure 4). On the contrary, a more structured configuration is necessary, which will result in longer traces. Due to the high density, it is also logical that at some point the RF traces and tuning elements will no longer fit all on the top layer of the loadboard and this best practice rule of thumb for existing ATE designs will no longer work.

These challenges, together with the question about connector performance and port to port isolation, will be discussed in the following chapters. The goal is to identify the optimum layout of the various blocks in the signal path to enable a robust layout topology that will work for high RF pin count devices.

2.2. Cables

Mechanical cable properties of interest are length, diameter, max bend radius, and bend radius and strength of the strain relief at the junction of the cable and connector.

Due to the amount of cables (64+) and the limited space on the loadboard, it is necessary to use a thin flexible cable. There are Mini-Coax RF cables available with a diameter of just 1.42mm. Other currently used RF cables, like Gore53, have 1.8mm and the legacy semi-rigid 3.5mm. The overall performance of the Mini-Coax cable is pretty much comparable to the legacy type of RF cables taking into account that the typical length is just around 15cm. The reduced diameter makes the cable less stiff and reduces the risk of damage at the junction to the connector. A strain relief will provide further protection. Overall, the Mini-Coax requires less space, increases routing flexibility and enables smooth handling during RF tuning and hardware debug.

2.3. RF Connector Interface

The challenge is to provide a robust 64+ RF port interface to the ATE test head, while still allowing manual access to the whole DUT application area of the loadboard.

Easy access to the application area on the loadboard is mandatory. This is not only true for the loadboard assembly phase, but especially for hardware debug or RF tuning. Typically, an RF loadboard requires an interconnection plate or ring, which contains the blind mate connectors building the interface between loadboard and tester RF resources. Since the application area on the loadboard is limited, the interconnection plate needs to be movable to allow easy access to the area under the plate. Practical movements for this purpose are allowing the plate to open and to swing individual elements upwards, depending on which area needs to be accessed. In some cases it is necessary to completely disengage the plate. The handling needs to be easy with no need of an additional tool like screwdriver.

Last but not least this interface must be quite robust to handle the large connector pressure. Blind mate connectors typically contain a spring so that minor misalignments won't cause a jam. As an example, if each connector has a spring preload of 8N and the board consists of 96 connectors, the total pressure can sum up to 768N.

3. RF Performance

High density RF loadboard design requires smaller connectors, smaller diameter RF cables, and longer (stripline) traces, since the application area and the access to the DUT is limited. This chapter will evaluate the relevant electrical performance for connectors, different coaxial cable lengths, for different mounting options, and for both microstrip and stripline PCB routing topologies.

3.1. Mini-Coax PCB Connector

The SMT mini-SMP has been the preferred connector so far for RF ATE applications with its small size and high frequency performance. Mini-Coax PCB connector should provide similar performance.

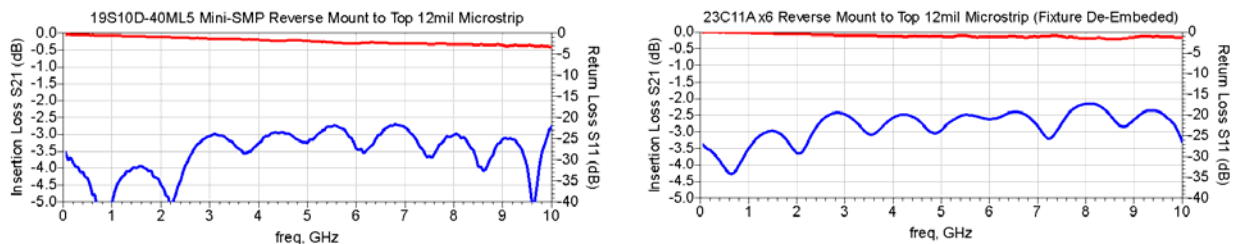


Figure 5: Performance comparison of surface mount Mini-SMP and ganged Mini-Coax connector (incl. via and microstrip line) to opposite side PCB microstrip performance using measurement based modeling techniques (left). Measured data from one of the better reverse mount transitions to 12mil microstrip (right).

Looking at the specification [6] for the new Mini-Coax connectors one sees that the connector by itself is expected to have DC to 20GHz performance with better than -20dB return loss to 6GHz. A first experiment with a larger signal pin SMT pad and added compensating ground relief does show very good results and gets very close to achieving the -20dB return loss goal to 6GHz. Investigations have shown that there is further room for improvement, especially when the simulated

and measured performance is compared. However, the demonstrated performance is already in the area that is achieved by typical production loadboards.

3.2. Cables

Parameters of interest are insertion loss, return loss, TDR impedance, and impact of cable bending.

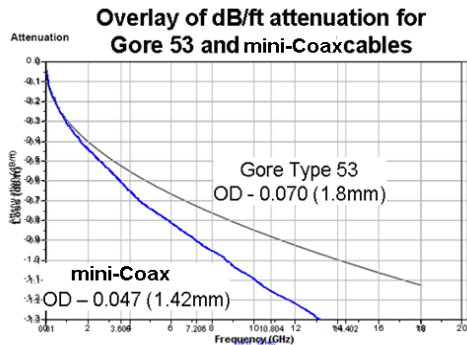


Figure 6: Measured insertion loss of (de-embedded) mini-coax cable [dB/ft] vs. legacy Gore53 specification [dB/ft].

Gore53 has been the preferred cable so far for RF ATE applications. According to the specification [6], the new Mini-Coax RF cable has an insertion loss of 3.6 dB/m (1.1 dB/ft) at 10GHz.

Comparing this value with the real measured loss data of Figure 6 provides excellent match. The corresponding loss with the popular Gore53 solution is 2.8dB/m (0.85 dB/ft) and the legacy much thicker semi-rigid cable is typically around 3dB/m. At 6GHz, the mini-coax has a loss of 2.6dB/m vs. Gore53 1.9dB/m. The mini-coax adds an additional loss of less than 0.1dB to 6GHz for a typical cable length of 20 cm, which is small compared to the losses on the PCB and connector. The comparison of the different cable assemblies must also include the connector losses. In this combination, Gore53 and Mini-Coax shows no significant difference in performance for the shorter cable lengths used on ATE applications.

Due to the amount of cables and the potential need to rebend the cables multiple times to allow access to loadboard components for tuning, the impact of cable bending on performance should be negligible. As a rule of thumb, performance degradation should be less than 0.05dB and phase variation less than 0.5%. Several 802.11a EVM measurements have been done to evaluate the effect of cable bending on real tests. However, the influence of bending is rather negligible and stays within the given limits.

3.3. Materials

One of the challenges with high density RF applications is the desire to minimize variations from site to site and yet there has been little interest in exploring the benefits of stripline routing. Microstrip with its combination of air and PCB material dielectric adds to the crosstalk challenges. The thicker plating on the outer layers along with the standard practice of using the Au plating as the mask for etching down to form the top layer pattern can lead to etching variations as well as undercutting reduction of trace width that is not visually observable. The addition of solder mask to protect against ESD is another source for variations to occur. On a low density application, where trace lengths are kept very short, these effects are not as significant, but on a high density application one questions if microstrip routing is still the right choice.

The other interesting aspect is to look at the insertion loss for different RF materials, trace widths and trace types (microstrip and stripline). Objective of this experiment is to prove that longer striplines can achieve similar performance as shorter microstrip.

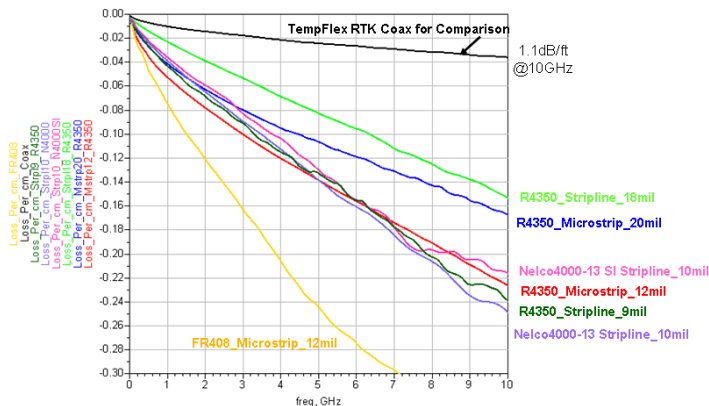


Figure 7: Insertion loss [dB/cm] through line experiments for different materials and conditions (fixture connections have been de-embedded). The different colors in Figure 7 describe the following configurations: Rogers4350 / stripline / 18mil (green), Rogers4350 / microstrip / 12mil (red), Rogers4350 / microstrip / 20mil (blue), Nelco4000SI / stripline / 9mil (pink), Rogers4350 / stripline / 10mil (dark green), Nelco4000 / stripline / 10mil, and FR408 / microstrip / 12mil.

The best configuration is represented by the green line (Rogers 4350, stripline, 18mil). At lower frequencies (2GHz) it has half the insertion loss compared to a Rogers 12mil microstrip line (red), which is currently the typical RF trace width used on RF loadboards. The performance gap is due to the smaller trace width and higher skin effect losses for the typical microstrip routing. A microstrip trace consists of an Au over Nickel composition and at high frequencies the signal currents will be highest in the outer plating layers making it critical to use a thicker low loss Au plating. According to Figure 7, a 5cm trace would have a loss of 0.48dB for the 18mil stripline configuration and 0.80dB for 12mil microstrip in R4350 at 6GHz. Utilizing low cost FR4 with 12mil traces generates a 1.4dB loss at 6GHz, which is roughly three times worse than the best case 18mil stripline routing.

3.4. Impact of Vias

The current practice of only routing RF on the outside layer microstrip can significantly increase the routing challenges for a high density loadboard. The ability to use inner stripline layers has the potential to improve isolation and increase routing flexibility.

The rule of thumb is that vias degrade performance, and so one finds creative connector mounting topologies like the reverse mount SMT SMP so that one can plug a cable in from the bottom while launching directly to microstrip on the topside. In this case the attempt to avoid the via transition has led to an exposed right angle bend of the signal pin that also is a source of radiation that cannot be fenced off with grounding vias like what is done with the standard SMT configuration.

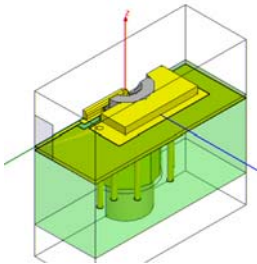


Figure 8: Specialized reverse mount SMT SMP to launch directly to opposite side microstrip, no via transition is required.

Applying this rule of thumb leads to a connector to PCB transition that is harder to optimize for return loss than if the connector is mounted in the normal SMT fashion and used with an optimized via transition to the opposite side. Figure 9 illustrates (using simulation data) that a normal surface mount SMP with a via to the opposite loadboard side can achieve better performance.

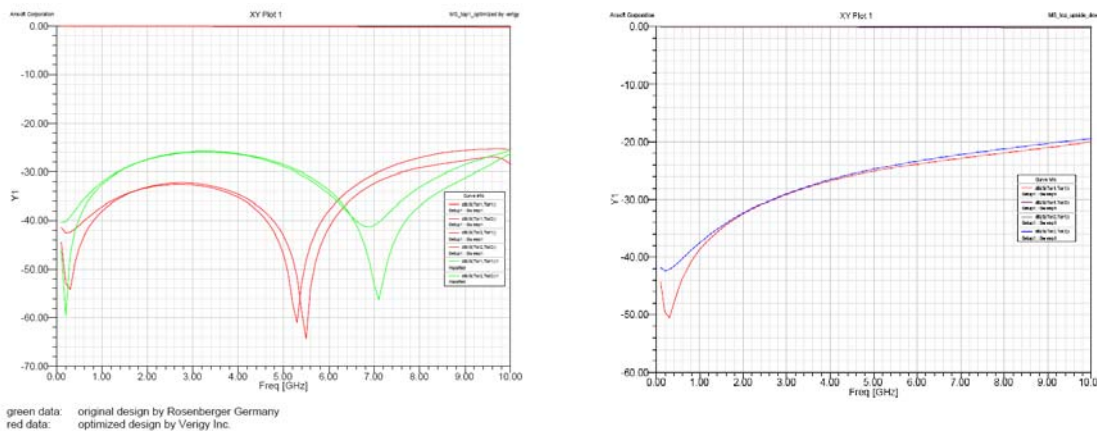


Figure 9: Left side: surface mount SMP with via to opposite side microstrip. Best case return loss is better than -30dB to 6GHz with Verigy optimized via transition.

Right side: reverse mount SMP to opposite side microstrip without via transition. Return loss is better than -20dB to 6GHz.

Impedance controlled via layout is a reality today. Different techniques have been evaluated in terms of performance and crosstalk [9]. It pushes the boundary towards more flexible layout approaches like the effective usage of striplines for RF.

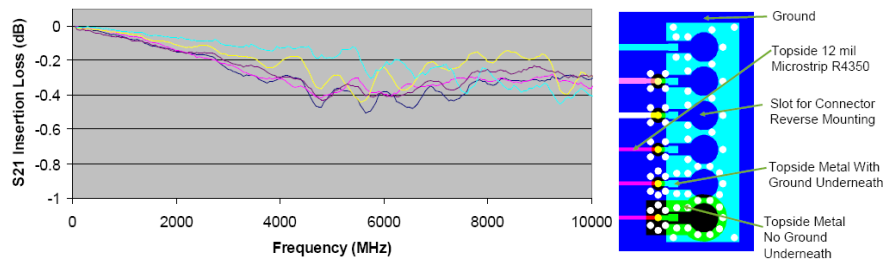


Figure 10: Mini-Coax connector S21 loss (6 layout variations according to the figure on the right) with PCB and coax transmission lines de-embedded from the measurement for comparison of just the connector performance.

Six experiments with slightly different layouts have been done for the standard surface mount topology. The best implementations achieve similar insertion loss of around -0.2dB at 6GHz compared to the reverse mount experiments (Figure 10).

This measurement revealed improvement areas. The performance of the experiments was not optimum, because the measured via impedance was approximately 46 Ohm due to the use of a 15mil drill size (vs. expected ~50 Ohm). Measurements with a 13mil drill size result in 49 Ohm impedance and provide information on the sensitivity to PCB fabrication tolerances and the ability to improve the correlation between simulations and measurements [5]. Additional optimization is planned to increase the reliability of achieving <-20dB return loss when transitioning to any layer of the PCB.

Data is available for the standard surface mount Mini-Coax connector with the 13mil drill via transition to top microstrip, but only for the full path measurement that includes the Mini-Coax cable, the connector, the via, 1.8cm of microstrip and the probe connection to a QFN pad. The return loss for this full path to a QFN DUT experiment resulted in -18dB @ 6GHz (red line in Figure 11, via to topside microstrip) and is getting closer to the simulated connector transition performance. The TDR plot shows how modifying certain features in the PCB footprint can help to reduce existing impedance discontinuities.

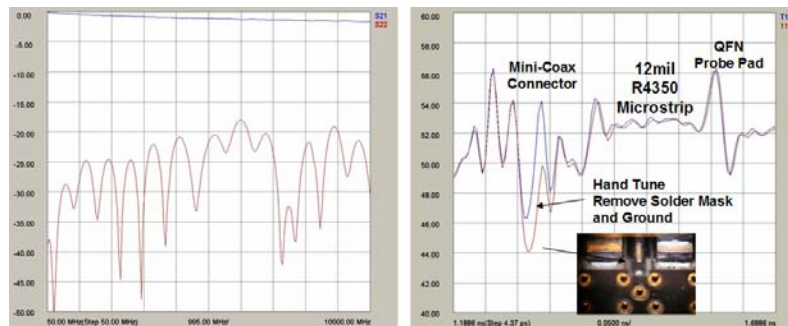


Figure 11: Mini-Coax connector return loss and TDR (PCB and coax not de-embedded).

The Mini-Coax surface mount connector experiments did show that via stubs must be avoided, because they act as a parasitic capacitance that skews the impedance and increases variations in the S21 insertion loss. For complex load boards with many layers, via stubs will be removed using a back-drill technology. A back-drilled via is a normal through-hole via in a PCB that connects two signal layers, but the dangling stub hanging down from the last signal layer to the bottom of the board has literally been drilled out. This removes the copper stub and significantly reduces the insertion loss ripple in the 5 to 10 GHz range.

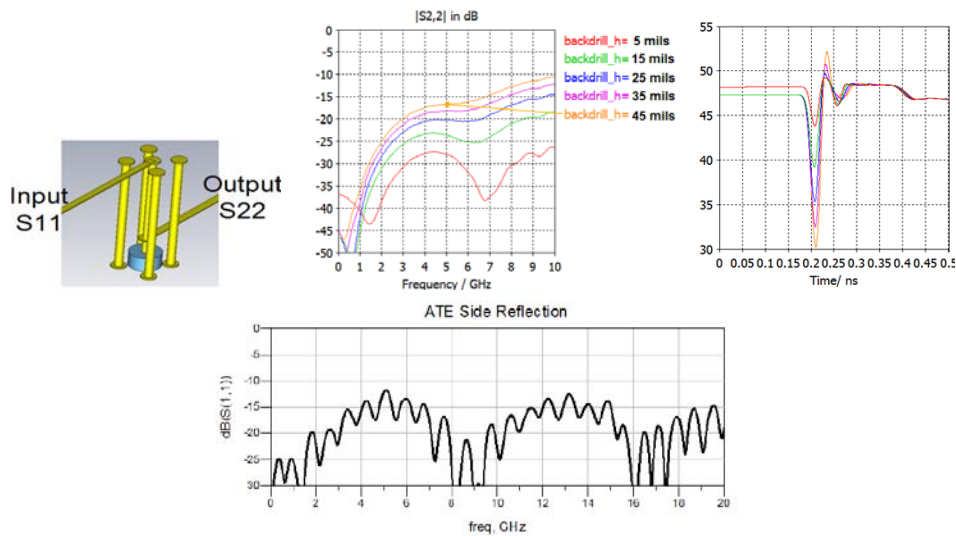


Figure 12: Simulated return loss with different back-drilled via stub lengths (center), the impact on impedance (right) and below the measured return loss of the Mini-Coax connector to inner stripline and the corresponding via stub capacitance at the end of the 46 ohm via section.

The impact of the vias seems to be rather small, as long as the impedance matching is done properly. Figure 12 center shows the return loss of a via depends on the length of the via stub left after the back drill. The return loss up to 10GHz is below -26dB if the via stub is 5mils or less. The measured data for a Mini-Coax connector transition to inner stripline does show this excess capacitance at the via transition and this indicates another opportunity to improve performance by tighter specification of the back-drill length.

Full path experiments have been performed on the high density applications board, from the PSRF interface ring through 15cm Mini-Coax cable with standard mount Mini-Coax connector, across 2cm of PCB routing with via transitions and probing at a QFN DUT pad. The results are shown in Figure 13. Excellent performance can be obtained when routing on microstrip and stripline with via transitions and across R4350 and Nelco materials [2].

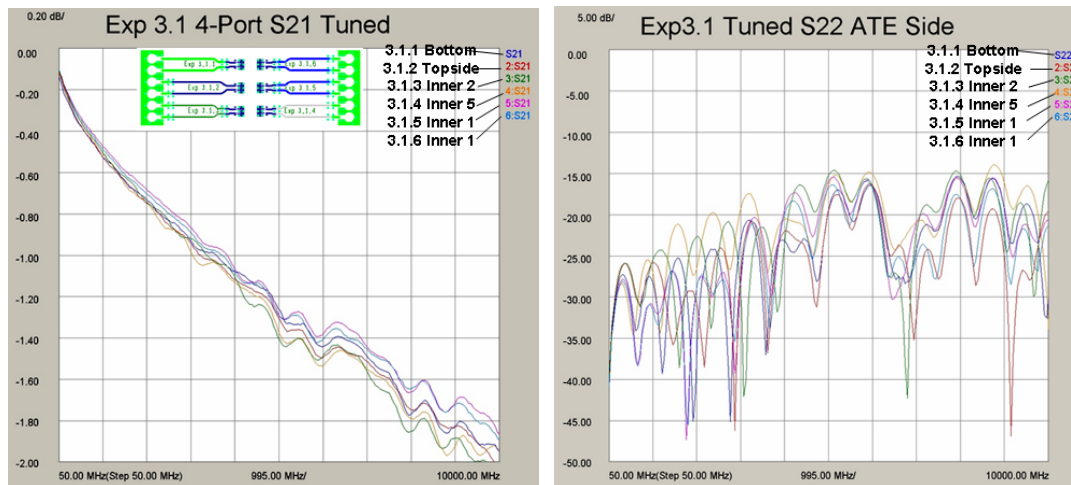


Figure 13: Measured full path losses going from the PSRF interface ring through the Mini-Coax 15cm cable with standard mount x6 ganged Mini-Coax connector, across 2cm of PCB routing with via transitions and probing at a QFN DUT pad. Experiments include stripline and microstrip routing for both R4350 and Nelco-4000-13 materials.

4. RF Isolation

4.1. Mini-Coax Connector

The first concern regarding the ganged Mini-Coax connector is isolation. It can be defined as electrical noise caused by mutual inductance and capacitance between lines due to their proximity.

According to specification, isolation is <-80dB up to 1GHz and <-60dB up to 4GHz. For the isolation measurement, we assume that there is no crosstalk at the RF cables (typ. >-100dB isolation) and very little crosstalk at the RF traces due to the relatively large pitch (less than -80dB isolation, calculation below).

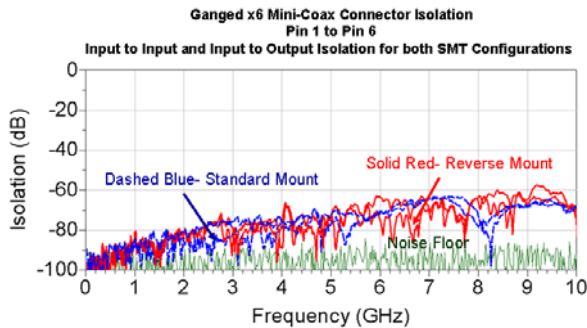


Figure 14: RF isolation between pin 1 and 6 of the ganged Mini-Coax connector.

Measurements (Figure 14) between connector pin 1 and pin 6 confirm the specification values: -81dB at 2GHz and -65dB at 6GHz. The crosstalk above 6GHz remains rather flat below -60dB.

Crosstalk will mainly occur due to the small conductor pitch of the ganged connector of Figure 2 (here: 3mm). Assuming the crosstalk depends mainly on mutual inductance, we can use the simple crosstalk estimation as it is shown in the equation below, where D and H represent the centerline separation and trace height [7]:

$$\text{Crosstalk [dB]} < 20 \log \left(1 / (1 + (D / H)^2) \right) \quad (1)$$

This estimation gives -80dB as the maximum crosstalk between pin 1 and pin 6 (D ~15mm, H ~0.15mm) and -52dB between two direct neighbor pins (D ~3mm). These benchmark figures help to classify the crosstalk measurement results in Figure 14 with other PCB type applications. This simple calculation shows that the measured crosstalk at 2 GHz would correspond to two traces with an offset of more than 15 mm. This is obviously a good isolation value. A layout guideline to avoid crosstalk is to separate signal traces by 5 times the trace width. Applying this to a trace width of 0.15mm results in a spacing of 5 x 0.3mm = 1.5mm and is much closer than the corresponding 3mm signal pitch used in the above calculation.

4.2. Isolation for Stripline and Microstrip Routing

High density RF applications can have additional crosstalk due to the proximity of the RF traces. A viable alternative is to move some of the signals immediately to a stripline. This has a couple of advantages. First of all, a stripline has better shielding against external electromagnetic interference (EMI), which can help when doing critical tests like noise figure measurements. Second and even more important is that, according to the formula above, doubling the distance will reduce the crosstalk by 12dB. Third, because a stripline is coupled to two ground planes in a symmetrical topology the near end cross talk (NEXT) is reduced and the far end crosstalk (FEXT) is cancelled [3]. This means more RF traces can be routed in one board layer with less crosstalk, which is important for high RF pin count devices. Of course, the via transitions to the stripline layer need to be laid out properly so as not to take away this advantage.

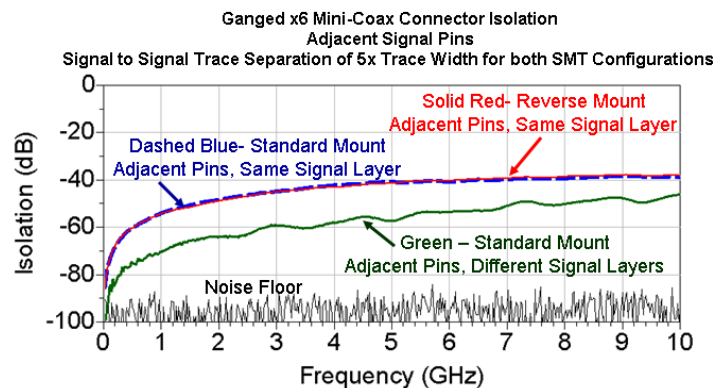


Figure 15: RF Isolation between adjacent pins for Reverse and Standard mounting configurations. Adjacent same signal layers are to top microstrip and the adjacent different signal layers are with inner stripline routing.

Figure 15 shows the far end crosstalk between adjacent signal paths going through the connector with a 3mm signal pin spacing and across 5.6 cm of PCB routing. At 2GHz the crosstalk for adjacent lines is -55dB for both the standard and reverse mount configuration. The green trace shows the benefit of routing adjacent pins on different signal layers with stripline routing. The black line shows the corresponding noise floor of the measuring network analyzer instrument. The reverse mount configuration does not include any via transitions, while the standard mount does include a via to get to the opposite side signal layer. The comparison of the blue and red traces above shows that the via transition (to get to the opposite side microstrip in the standard mount configuration) has little effect on the overall isolation when compared to the reverse mount configuration with no vias. The green trace with the stripline routing on different layers shows an improvement of 10 to 15 dB.

4.3. Ground Via Shielding Evaluation

The ganged Mini-Coax SMP connectors are now similar to the densities achieved with high speed digital connections using pogo block technology. Looking at lessons learned from pogo via transitions to a 250 mil thick loadboard shows the importance of using ground via stitching to fence off the fields [8, 9]. The spacing between the ground vias determines the isolation performance at higher frequencies and is typically 1/10th to 1/15th of the distance occupied by the signals wavelength in the PCB material. The wavelength at 6GHz is ~2.54cm (1 inch) in FR4 and with a 1/15th wavelength spacing the vias would be 1.7mm (68 mil) apart.

Adding a ground fill between signal traces should improve shielding, however if the ground fill has just a limited number of ground via connections than unwanted resonances can occur. The ultimate in RF shielding is to add rows of grounding vias to fence off the electromagnetic (EM) fields and increase isolation between the signal traces. Figure 16 shows the layout used for the stripline experiments.

The resulting data shown in Figures 17 and 18 is analyzed for both the full path that includes the fixture edge launch coaxial connectors and 5cm of the adjacent signal trace routing and for the case where the fixture effects have been de-embedded (calibrated out) to give an estimate for the isolation of a section of adjacent traces that are 2cm long. When comparing trace to trace isolation the larger FEXT crosstalk is used for comparison. The guard trace with only 2 ground vias does exhibit a resonance. This resonance is at multiples of the of the ~1.5GHz frequency that has a ¼ wavelength of ~2cm which corresponds with the 2cm spacing of the two ground vias in the guard trace. The resonances made it difficult to de-embed the test fixture.

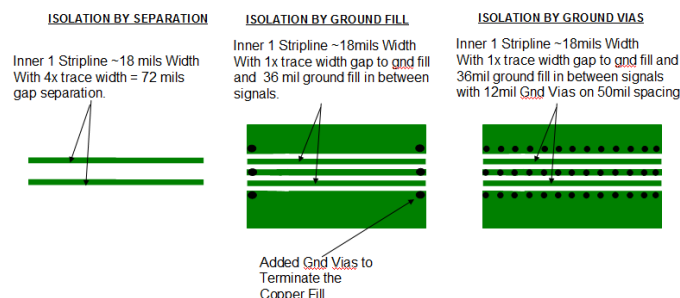


Figure 16: Three different PCB Layouts for measuring RF isolation between adjacent stripline signal traces: isolation by separation, isolation with guard trace, and isolation with ground via stitching.

However, even with just the full path data it is obvious that a guard trace provides little benefit if it does not have the 1/10th wavelength or smaller ground via spacing to avoid resonances and improve isolation in the frequency band of interest. The data after de-embedding the fixture connections clearly shows how adjacent stripline signals can almost cancel the FEXT coupling and that the addition of ground via stitching is less than a 10 dB improvement in isolation when routing stripline with 5 times the trace width separation. This data shows that simple signal trace separation for stripline routing is sufficient for isolating the signals.

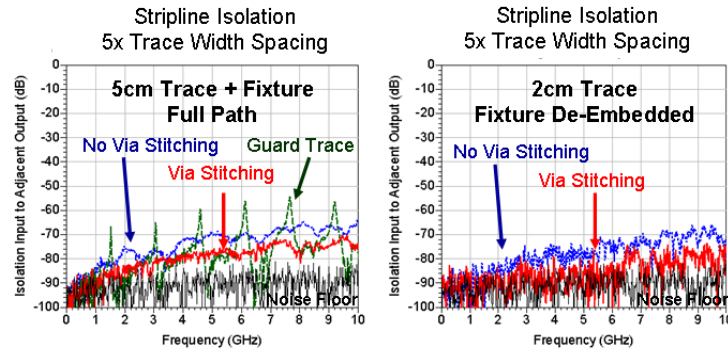


Figure 17: RF isolation for adjacent stripline routing with no shielding (blue), guard trace shielding (green), and ground via stitching (red); layout according to Figure 16. The noise floor of the measurement system is at the bottom in black.

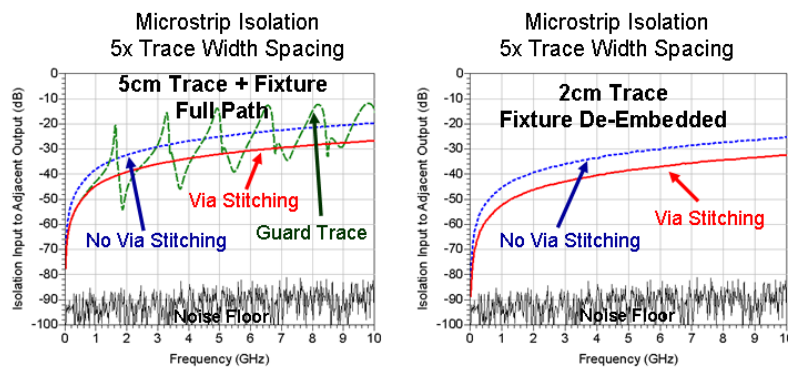


Figure 18: RF isolation for adjacent microstrip routing with no shielding (blue), guard trace shielding (green), and ground via stitching (red); layout according to Figure 16. The noise floor of the measurement system is at the bottom in black.

Overall, the RF isolation between the input of the aggressor signal and the output of the adjacent victim (FEXT) of the stripline implementation is ~40dB better than that of the similar microstrip implementation. The -70dB isolation for the stripline routing is already an excellent value and the addition of ground via stitching to fence off the EM fields is not needed. Ground via stitching does show a measurable improvement of ~8dB for the microstrip routing and is worth adding if the space is available.

The previous section analyzed isolation at the individual blocks of the signal path. It is also interesting to look at how crosstalk affects complex RF measurements like EVM (Error Vector magnitude). Experiments were done to look at crosstalk within the same layer as well as crosstalk over different types of GND layers. This will provide a rough idea, how much RF isolation is necessary to still allow solid test results. The objective is to demonstrate the performance improvements when RF is moved from microstrip to striplines.

The first step is to describe the crosstalk effects on an 802.11g EVM measurement with -20dBm signal power, when the interfering trace runs in parallel on the same layer. The interferer generates a frequency of variable power with 4MHz offset from the 2.4GHz carrier, but still within the channel. Any interferer frequency outside the main channel does not have an influence on the measurement. In a second step similar measurements have been done, but having the interferer trace on a separate layer. By doing this, the RF isolation improved roughly from -15dB (0.8mm gap) to -75dB.

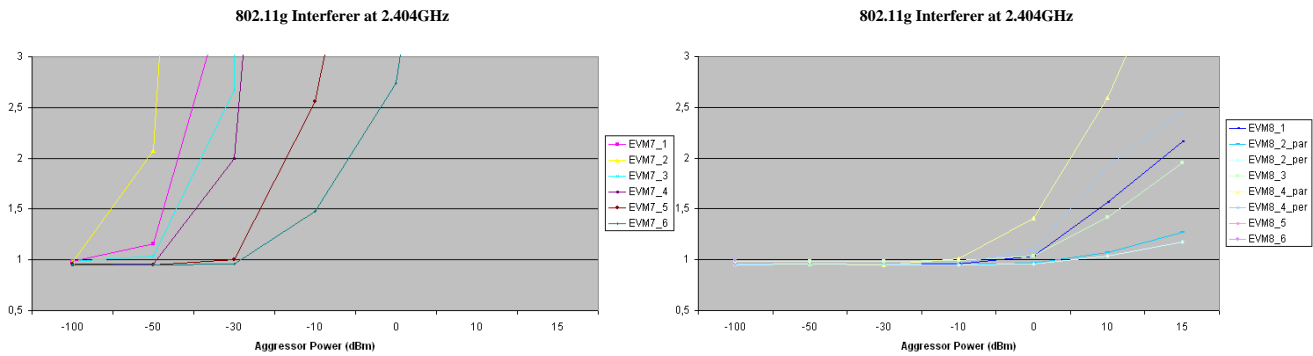


Figure 19: Interferer impact on 802.11g EVM at the same layer (left) and different layer (right) for different setups.

Figure 18 (left) illustrates the result for 6 different layouts, but all traces on the same layer. The yellow case is the worst; -50dBm of aggressor power already results in a significant impact on the EVM measurement result. Here, the separation between the 0.525mm wide microstrip traces is just 0.8mm and they are routed together for a total of 10cm. The dark purple color describes the same setup, but with 0.23mm wide stripline and a gap of 1mm. According to (1), the wider gap should improve isolation by ~4dB, but actually a significant impact is still seen at -30dBm of interferer power. This demonstrates again that striplines are more robust against crosstalk than microstrip. The brown trace shows the improvement when a guard trace topology is added and the best case green trace is achieved with ground via stitching between the aggressor and the signal.

Figure 18 (right) illustrates similar measurements, but having the interfering aggressor trace on a separate layer. By doing this, the RF isolation improved roughly from -15dB (0.8mm gap) to -75dB. The yellow line describes the worst case. The 0dBm aggressor has a significant impact on the measurement result. Signal and aggressor are running in parallel over a length of 10cm, but separated by 2 GND layers. The 0.52mm signal trace does a loop back with a gap spacing of only 1mm to the ATE tester for measurement and so there is the possibility that the outgoing signal could impact the adjacent return path signal. The next worst case experiment was EVM8_4_per. It has the same setup, but this time the aggressor crosses the signal in a perpendicular fashion. Here, measurement degradation starts roughly 10dB later and as expected is much less than when routed parallel to the victim signal trace.

Overall, crosstalk or RF leakage is a solvable challenge. The crosstalk at the transition into the socket and DUT has little room for improvement and will set the limit on how good the isolation can be between signals. RF traces can get very close on BGA packages and can only be separated further if smaller trace widths or via transitions to other layers are used. GND fencing can improve isolation by up to 10dB, but since it affects all layers on the loadboard it is not always the best solution. The simpler approach is to ensure enough of a gap between RF traces and minimize the length of parallel routing with minimal spacing. As rule of thumb, the gap should be at least 5 times the trace width pitch for trace routing. It results in -80dB crosstalk for a 2cm routing path of striplines. If this is not enough, doubling the offset will reduce the crosstalk by another -12dB. Using multiple stripline layers instead of microstrip will significantly increase the routing area for increasing the gap spacing between traces. Critical RF connections or clocks will need special care and should be routed on separate connectors, if -60dB isolation is not sufficient.

5. Summary

The increase of ATE RF ports to 48 and beyond to meet the relentless trend to add capabilities and reduce cost in the wireless communications market will dictate the need to re-evaluate the traditional methods for RF loadboard design. Combining RF technologies like Cellular, Bluetooth, GPS, FM and WLAN and also requiring multi-site testing will push the limits of existing loadboard technology. High port count ATE RF applications have the obvious issue of mechanically fitting 48+ ports as close to the DUT as possible and as shown above this can easily be addressed with Mini-Coax ganged connector concepts. Electrically the ganged connector approach can still maintain isolation to the -60dB level and beyond through the use of shielding vias and adequate spacing. The use of via transitions to provide flexibility in where and how the connector is surface mounted should not be something to avoid because of past “performance rules-of-thumb”. Properly designed connector via transitions to inner layers or opposite side microstrip have the potential of improving the impedance matching and shielding as compared with existing designs while increasing routing and placement flexibility.

The ability to route stripline has the potential to increase the layers available for routing. In addition, it limits the quantity of vias that are needed for isolation while providing similar path loss to that of microstrip. Improved techniques for

simulating matching networks using layout based simulations can facilitate spreading matching networks out further. It also helps to identify the best solution for layout trade-offs rather than relying completely on after the fact hand tuning. Overall the move towards higher densities for ATE RF testing looks quite reasonable with some improved engineering of available tester interface technologies to address the specific application challenges. Figure 19 summarizes the high-level key proof-of-concept points.

Key Requirements	Parameter		Proof Criteria
Mechanical Properties	Mini-Coax Connector	Size	30% smaller than mini-SMP
		Mating Cycles	better than mini-SMP
		Robustness	Same as full detent type with large solder area
	Cables	Diameter	20% less diameter than current Gore53
	Connector Ring	Access to Application Area	more flexible and convenient than current solution
RF Performance	Mini-Coax Connector	Insertion Loss	-0.2dB up to 6GHz or better
		Return Loss	-20dB up to 6GHz or better
	Via	Insertion Loss	-0.1dB up to 6GHz or better
		Return Loss	-20dB up to 6GHz or better
	Cables	Insertion Loss	less than -0.1dB delta to Gore53 up to 6GHz
		Bending Robustness	comparable with Gore53
	Materials	Insertion Loss	ability to reduce insertion loss by 50% for stripline to compensate long traces
RF Crosstalk	Mini-Coax Connector	Isolation	better than 60dB up to 6GHz
	Stripline vs. Microstrip	Isolation	>10dB better isolation for stripline
	Shielding	Isolation	30% better isolation for stripline
	Ground Via Stitching	Stripline	5x trace to trace spacing shows little difference with and without stitching
		Microstrip	Stitching does help even with 5x trace to trace spacing.

Figure 20: Summary of key concept requirements and their proof criteria.

6. References

- [1] C. Robertson, "Printed Circuit Board Designer's Reference", Prentice Hall 2004
- [2] M. Dresler, P. Hotz, "RF PCB Layout – How to Keep Pace with the Current Test Challenges", Semicon Europa 2007
- [3] W. Maichen, "When Digital Becomes Analog – Interfaces in High Speed Test", ITC 2008
- [4] K. Harvey, "Target 2012: ... we're closer than you think.", Semicon West 2008
- [5] T. Warwick, "Measurement Repeatability for RF Test Within the Load-board Constraints of High Density and Fine Pitch SOC Applications", ITC 2008
- [6] unknown, "RF Coaxial Connectors", [http:// www.rosenbergerna.com/coaxialconnectors.html](http://www.rosenbergerna.com/coaxialconnectors.html), Rosenberger
- [7] H. Johnson, M. Graham, "High-Speed Digital Design", Prentice Hall, 1993
- [8] Swanson, Hoefer, "Microwave Circuit Modeling Using Electromagnetic Field Simulation", Artech House, 2003
- [9] B. Szendrenyi, H. Barnes, et al, "Addressing the Broadband Crosstalk Challenges of Pogo Pin Type Interfaces for High-Density High-Speed Digital Applications", IMS 2007