ALCIOM
5, Parvis Robert Schuman
92370 CHAVILLE - FRANCE

Tel/Fax : 01 47 09 30 51
contact@alciom.com

www.alciom.com

Projet :

White paper

DOCUMENT :

High speed and RF PCB routing : Best practises and recommandations

REFERENCE :		
AL/RL/1130/007		
DATE :	VERSION :	AUTHOR :
29/07/2011	**1A**	**Robert Lacoste / ALCIOM**

ABSTRACT:

This document provides some experience-based advices on PCB routing techniques, applicable either to RF designs, high speed logic design, or EMC-constrained designs.

Based on an article published by the author in Circuit Cellar magazine, August 2011

DOCUMENT HISTORY

DATE	VERSION	AUTHOR	COMMENT
29/07/11	1A	R.Lacoste / ALCIOM	Initial version

CONTENT

1 Introduction

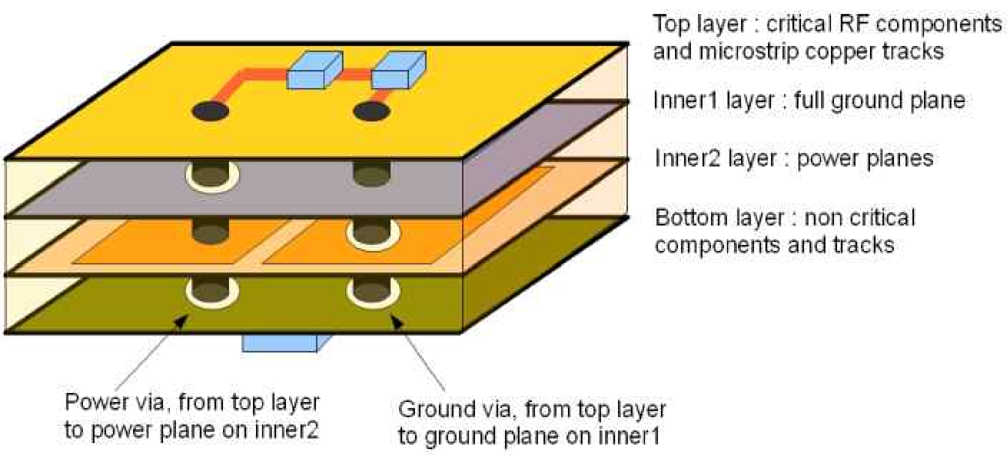
The basic rules for solid-rock RF PCB design are quite easy to understand. Moreover the same rules applies for high speed logic designs and EMC-constrained designs : high speed means fast signal edges which means high frequencies everywhere. This document summarizes examples of bad and good PCB routing practices.

2 Ground

One of the most usual problem in PCB design, and also system design, is the lack of a good ground structure. On any project, select any grounded node and imagine that it is no longer actually connected to ground but to a voltage source of a couple of volts. It wouldn't work anymore, isn't it ? Such a situation is unfortunately very common on poorly designed PCB's: When high frequency signals are involved then any conductor behave as a transmission line, the voltage between its ends is no longer null if its impedance, at the working frequency, is not very low.

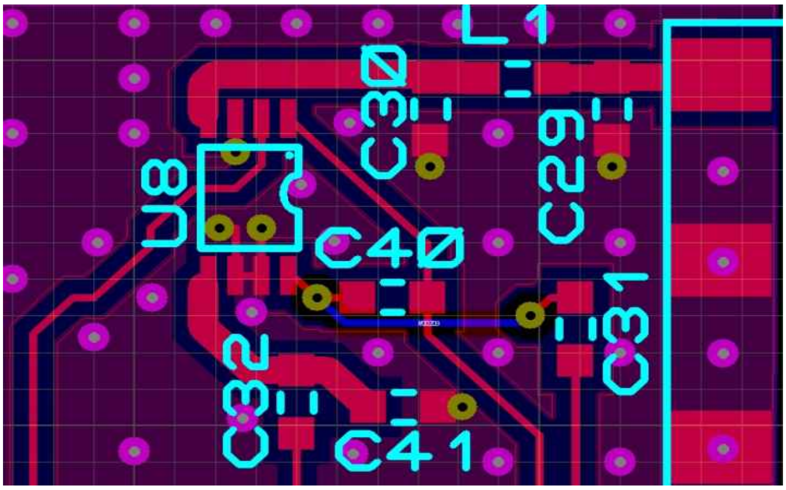
So, as a general rule, as soon as ground level is not well defined thinks are getting difficult.

There is only one solution : **do your best to have a solid-rock common ground.** And on the PCB there is only one easy solution : accept to spend a couple of extra dollars and use at least a **four layer PCB**. As illustrated below, a four-layer PCB allows to devote one of the inner layers to a full ground plane. This is the best possible ground : a full, plain, sheet of copper, as large as the PCB, insuring minimal impedance between any couple of grounded points. **Never break this ground plane by routing any small track on this ground layer** : there should be only a ground plane on that layerr. As soon as some copper is removed on this ground plane then this introduces parasitic impedance’s on neighbor tracks. On such a 4-layer design, usually the side of the PCB closest to the ground plane is then used to place all RF components using microstrip techniques,. The opposite side is used for less critical components, and lastly the second internal layer is used for power supplies, using power planes as large as possible to minimize they impedance as well. This has the extra advantage to implement zero cost filtering capacitors as the power planes and the ground plane are facing each other.



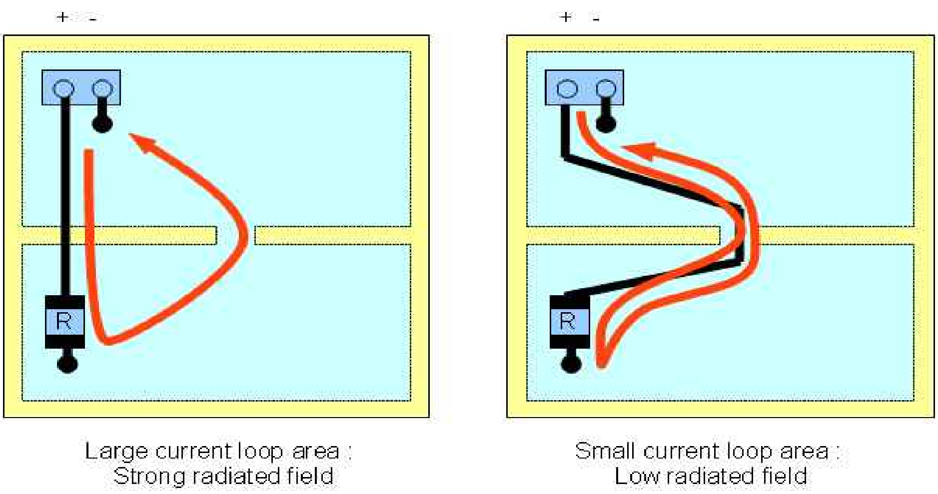
Using a four-layer PCB is the easiest solution for a good grounding structure. Usually the first inner layer is then devoted to a full plain ground plane. The top layer, just above the ground plane, is used for all critical RF components and microstrip tracks. The second inner layer is used for power supplies, and lastly the bottom layer can be used for less critical components and tracks.

For cost reduction purpose a double-sided PCB may be required. This is possible of course, but it is far more difficult. When there is a need to route tracks on both sides of the PCB in the same area then a good ground plane is no longer insured. The only solution is then to implement as large as possible ground planes on both sides, interconnected by plenty of via's. This is a non trivial task, and usually the best solution is to try to avoid any bottom layer tracks below the most critical RF sections.



If you definitively need to stay with a double-side PCB then your life will be more complex, as the ground plane will be shared between top and bottom layers. You need to insure that there is at least a full ground plane under the most critical sections, using top side routing as much as possible and only a couple of tracks on the bottom side. Plenty of interconnecting vias are needed to interconnect the bop and bottom grounds. Bottom side tracks, in blue on the figure, should never cross the wider RF tracks on the opposite side.

Split ground planes are sometimes recommanded, for example with a ground plane for the logic sections and a ground plane for the analog components, interconnected at a single point. The idea is to minimize noise through the ground planes. Unfortunately it is very difficult to properly implement such a concept. In particular, it is then mandatory to route all tracks going from one region to the other exclusively above this interconnecting point. If not then this gives a very good antenna which will either transmit or receive spurious signals, as shown below. In 99% of the cases a full unique ground is more reliable and gives better results than split grounds, as long as the placement of the components is adequate, may be except for some audio projects or in case of strong ESD risks.

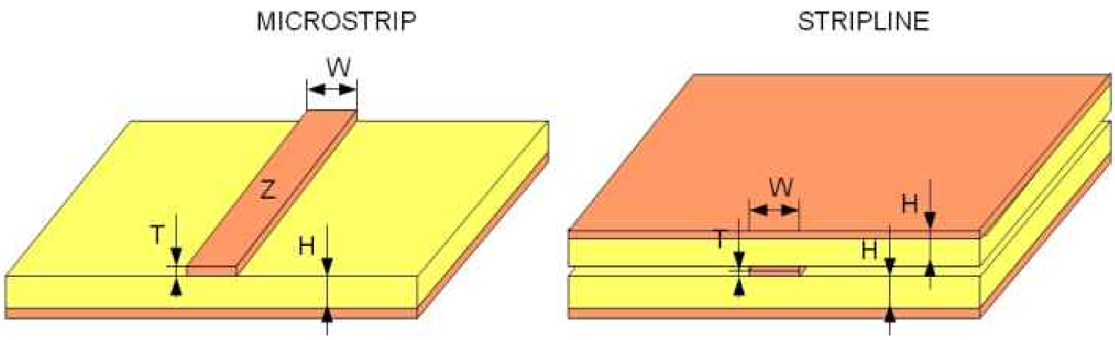


Split ground plane is usually a bad idea, except if there is a very specific need like strong ESD risks. If not, any track going from one ground area to the other should cross the boundary just under the interconnecting point (right). If not (left) you will have a current loop meaning EMC problems.

3 Track impedance and reflections

Any high frequency or high speed signal must be routed while taking care of impedance matching. If not then signal strength losses will appear, and more importantly nasty reflection phenomenons (see « TDR basics », Circuit Cellar 225).

In order to have an impedance matching the receiver impedance must be the complex conjugate of the source impedance, which means they must be identical if the impedance's are purely resistive, and the characteristic impedance of the interconnecting line must be well selected. In the vast majority of RF designs all sub-systems have a 50-ohm impedance, therefore the PCB tracks must have a 50-ohm characteristic impedance. This translates to a precise PCB track width, depending on PCB technology. The two most common solutions is either microstrip technique, meaning routing the RF track just above a full ground plane, or stripline, where the RF track is sandwiched between two ground planes. Figure below gives the most common track widths in both techniques when using a standard FR4 laminate.



$Z=50\ ohm, T=35\mu m$ $FR4\ substrate\ (\epsilon_r=4.6)$	
H	*W*
1.6mm	2.95mm
0.8mm	1.45mm
0.5mm	0.9mm

$Z=50\ ohm, T=35\mu m$ $FR4\ substrate\ (\epsilon_r=4.6)$	
H	*W*
0.8mm	0.65mm
0.5mm	0.4mm
0.35mm	0.26mm

A microstrip is a copper track just above a plain ground plane, whereas a stripline is sandwiched between two ground planes. The tables gives you the track width for a 50-ohm characteristic impedance for classical FR4 PCB thicknesses, calculated with AppCad.

These widths are usually not very critical, meaning that a 5% error is usually neglectible, but the good order of magnitude must be respected. Lastly track width is only a concern when the track length is longer than a tenth of the wavelength at the working frequency. For example at 2.4Ghz the wavelength in the air is c/f, with c equal to the speed of the light in free space which is 3.10^8m/s and f=$2.4.10^9$ Hz, which gives 125mm. A tenth is 12.5mm. However here the signal is not in the air but on a PCB, where the signal speed will be lower than the speed of the light. It is reduced by the square root of the relative dielectric constant of the material which is 4.3 for FR4; Therefore the final critical length is 12.5mm/sqrt(ε_r), giving 6mm. At 2.4GHz at least any track longer than 6mm must have the correct width.

4 Schematic and components selection

The PCB design starts with the schematic, and more exactly with the selection of the components. Of course surface mount devices (SMD) are prefered, as smaller components and shorter wires means best RF performances. But which package size is optimal, for example for all the passive resistors and capacitors ? One useful criteria is to look at the track width as calculated for a 50-ohm impedance. Intuitively the best RF performances are usually acheived if the width of the component is close to track width : this will reduce impedance matching problems between the track and the component pad.

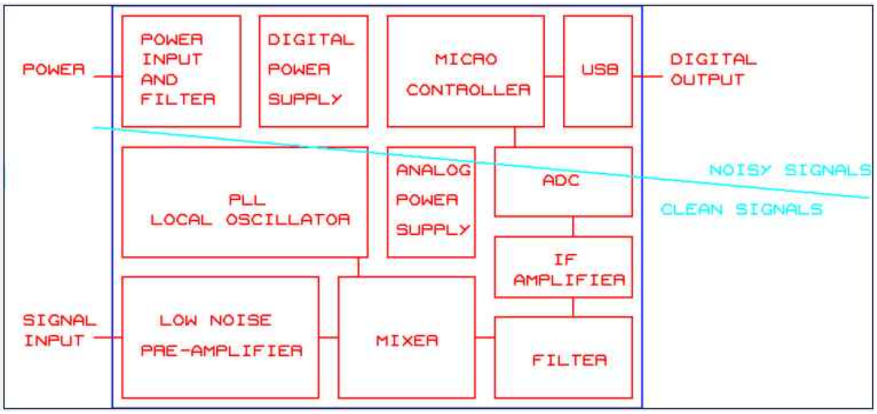
Impedance mismatches will be reduced if you select components with a package roughly the same size than the calculated track width. Here a 0603 package will be the better choice for this 0.9mm wide microstrip track..

Test points must be planned at the schematic phase too, as well as extra components like test connectors, which could not be populated is not needed for production batches, or decoupling capacitors which allows to open the circuit during test phase.

The power supplies needs also specific care : implement separated digital and analog power supplies, even using only a small SMT ferrite bead to isolate them. If there are successive high frequency amplifying stages then take care to decouple the power supply of these successive circuits, if not you will very probably get an oscillation somewhere.

5 The placement phase

High frequency problems are drastically minimized when the tracks are short and straightforward. Therefore the placement of the components on the PCB is really the most critical phase of PCB design. Experience shows that as much time should be spent on placement than on routing. Start by segmenting the different areas of the PCB : digital sections ? Critical RF sections ? Power supplies ? Avoid any loop-back risks which means for example don't place an RF output connector close to an input.



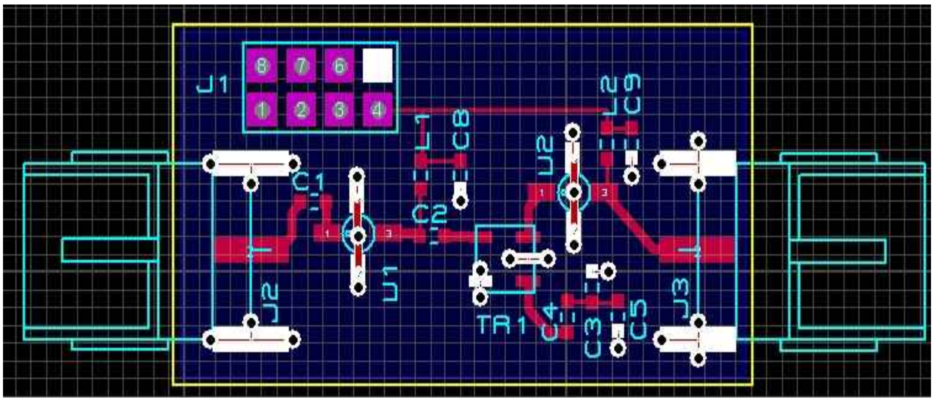
The first step in placement is to define a clear placement strategy, based on the constraints of your project. Separating "dirty" areas and sensitive parts is key, as well as avoiding loop-backs between low level inputs and high power outputs.

When the placement strategy is defined, start by placing the most critical components first, trying to simplify as much as possible the routing of the RF or high speed signals. Don't forget to place their associated decoupling and peripheral components as close as possible, this is where looking at the schematic helps. Always try to rotate components, or to swap pins where possible, in order to limit the number of signal crossings. Each crossing will mean vias, which mean potential impedance matching problems.

When all critical parts are placed you can then place the less critical sections : digital control, power supplies, etc. If using a four-layer PCB then one of the good solutions is to put all critical components on the top side, and all decoupling and power supply stuff on the bottom. This insures that the decoupling is as close as possible to the destination circuits, and simplifies the routing. You can also put all components on the same side but the PCB dimensions will be significantly larger.

6 Ground connections and parasitic components

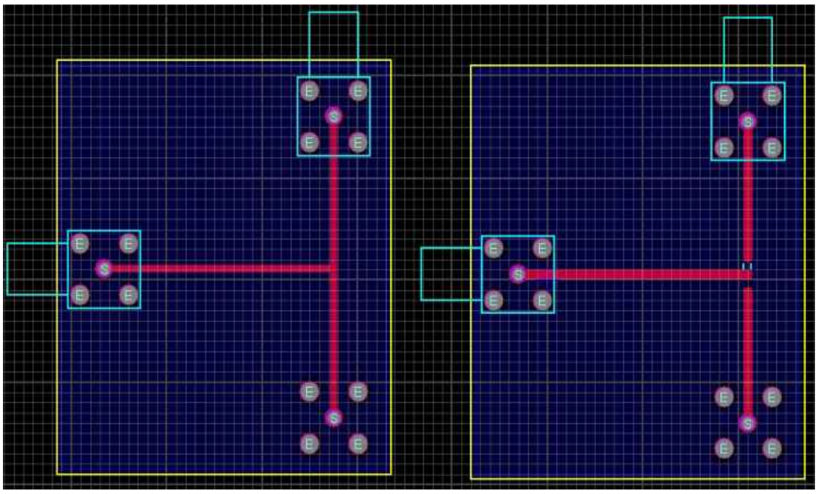
Now all components are placed on the PCB, and the routing phase will start. Never click on the auto-route button (at least not at that stage) ! The PCB designer knows which tracks are critical, the tool doesn't, at least except if you have spent a lot of time to explain it what to do. Starts by connecting all grounded pads to the ground layer. For each grounded pad, add a ground via as close as possible to the pad, going directly the the inner ground plane for a 4-layer design or to a grounded copper area on the other side for double-sided design. Never use tracks to route a ground : remember that the inductance of a 10mm long track is around 1nH. This may not seems much, but the impedance of a 1nH inductance is 12 ohm at 2GHz, not a neglectful value as compared to 50 ohm. When using RF IC's with several grounded pins, avoid to use only one ground via, but a separate via for each pin as well as some ground vias under the chip itself. This will give the lowest impedance to ground. In the same spirit never use thermal relief when a component grounded pad is inside a ground plane : these small tracks going between the pad and the ground plane facilitates the soldering but are marvelous small inductors in RF designs and should be avoided.



*Start by interconnecting each of the to the underlying ground plane with dedicated ground vias as close as possible to the component.
A via cost nothing, and good grounding is priceless...*

Then route all critical RF or high speed tracks, using as much as possible the calculated track width for good impedance matching, except if the track is very small (shorter than a tenth of the wavelength as discussed). Usually route then the power supplies, using power planes where possible, and finish up the board with all the non critical tracks.

Another common error is the routing of “tee” structures on the RF signals, for example when a PCB which could be populated in two versions, one where the RF signal goes to connector A and one where the RF signal goes to connector B. If routing is done without any specific care then the RF signal will be split into two tracks, one going to a connector and one which is open-ended. However an open-ended wire is not at all neglectful in RF. Transmission line theory shows that such an open circuit is in fact equivalent to a short circuit to ground at a precise frequency, for which the line length is a quarter of the wavelength, making it a resonator. For example a 40mm long open wire on an FR4 substrate will resonate at about 2.7GHz. So open-ended lines or tee structures should be avoided

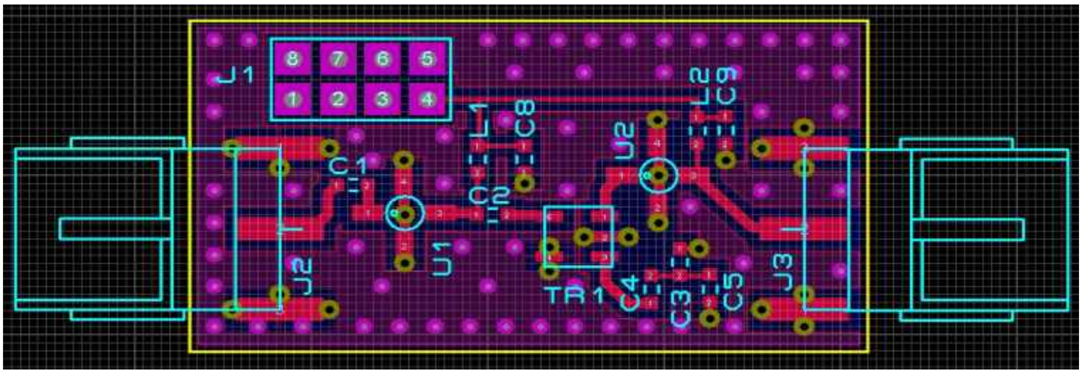


In this example the same PCB can be populated with a connector either in top or bottom position. This layout showed on the left is catastrophic as the unused open-ended track will act as a band-stop filter at its resonant frequency. On the right this improved design solve the issue, with a small bypass capacitor or 0-ohm resistor implanted in either top or bottom position

7 Ancillary ground planes

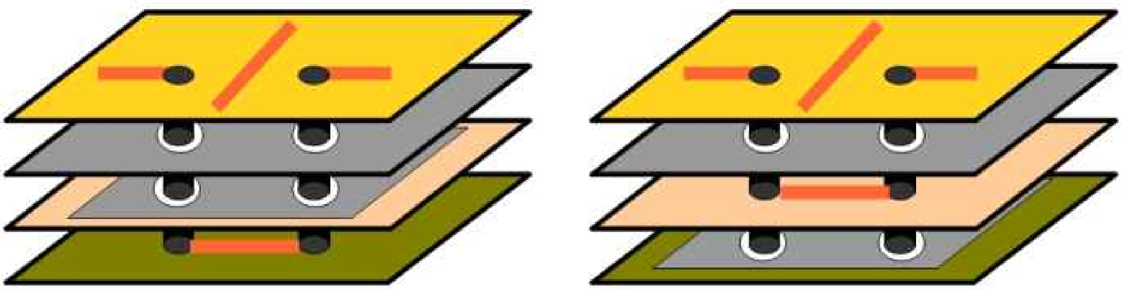
Let's assume the PCB is four-layer, with a full ground plane on one of the inner layer. Should copper pour be also used on the two external layers for additional ground planes ? The answer is usually yes, as a grounded copper area between two tracks will helps to decouple the signals. However it must be handled with care. Firstly the clearance between the grounded area and the RF tracks must not be too small : If not, the proximity of the copper will change the impedance of the track. You will in fact move from a microstrip structure to a so-called coplanar waveguide(CPW). You could, but then you would need to recalculate the track width for a 50-ohm impedance. However if you keep a clearance at least equal to the track width you shouldn't have any problem.

The second issue is the interconnection of the different ground planes : it is not enough to interconnect them in a couple of points, ground interconnecting vias should be spread all around the ground planes for a proper equipotential structure. As a rule of thumb the distance between two ground vias must not exceed a tenth of a wavelength at the highest working frequency. If the highest working frequency is 2.4GHz then a via every 6mm or so is needed. It is also fundamental to put ground vias all around the PCB edges. These vias, if closely spaced, will limit the RF leakage through the PCB laminate, especially if you take care to stop the power planes before this via line.



Additional ground plane can be added on top and bottom layers, filling all free space. However the clearance must stay large enough not to perturbate the RF tracks, and it is mandatory to use regularly spaced ground vias to insure a proper grounding structure. Densely spaced ground vias at the board outline avoid some RF leakage.

What could be done if two RF tracks need to cross each other ? If there is no other solution then vias must be used : either moving one of the RF tracks to the opposite side, still using microstrip above the power plane, and back, or using strip line. In this last solution the idea is to temporary use the power plane as a RF plane, and to route one of the two RF signals between two ground planes. With proper track widths and with plenty of ground vias around to limit the RF leakage through the laminate satisfactory results could be acheived. The same technique can be used when using shield cans : when a RF track needs to exit a SMT-soldered shield can it could be router either through the bottom layer or through an inner layer.



These figures shows you two alternatives when you absolutely need to have two RF tracks crossing each other : Either through the bottom layer (left), or through a stripline on the inner2 layer (right). In both case plenty of vias should be used to avoid RF leakages

8 References

A Practical Guide to High-Speed Printed-Circuit-Board Layout

John Ardizzoni, Analog Devices

http://www.analog.com/library/analogdialogue/archives/39-09/layout.html

High speed board layout guidelines
Altera
http://www.altera.com/literature/an/an224.pdf

RF/Microwave PC Board design and layout
Rick Hartley, L-3 Avionics systems
http://www.jlab.org/accel/eecad/pdf/050rfdesign.pdf