Internal Electromagnetic Compatibility of RF Devices

Antonije R. Đorđević
School of Electrical Engineering, University of Belgrade

I Introduction

Design and manufacturing of radio-frequency (RF) devices are delicate due to various parasitic effects that become pronounced at high frequencies. For example, the gain and efficiency of active elements (transistors and vacuum tubes) are reduced due to parasitic reactances and the finite velocity of movement of the charge carriers. Resistors, coils, and capacitors have pronounced parasitic effects, so that they behave like non-ideal elements. The electromagnetic-field propagation effects become pronounced. Finally, parasitic coupling becomes significant.

This coupling is considered as the electromagnetic compatibility (EMC) problem. We can distinguish between the external compatibility (coupling between the device and its environment) and the internal compatibility (coupling between various parts within the device). RF devices usually contain active elements (amplifiers). The parasitic coupling can distort the transfer characteristic of the device and cause an instability of the active elements, which can make these elements start oscillating. Only the internal compatibility is treated in this paper.

A simplified view of an RF device is shown in Figure 1. To reduce the coupling with the environment, the device is usually well shielded by an almost contiguous metallic box. (A rectangular box is shown in Figure 1.) The RF signals are usually fed into the device and out of it using coaxial connectors, as shown in Figure 1. Other transmission lines (e.g., power supply) are usually carefully filtered at the locations where they penetrate the shield, but they are not shown. Active and passive components are usually mounted on a printed-circuit board (PCB), i.e., the motherboard or plug-in boards.

This paper primarily deals with RF devices. However, the results and conclusions are also applicable to digital devices (e.g., computers). Signal spectra of modern fast computers extend well into the microwave region, so that the RF and digital devices have similar EMC problems.

The internal EMC is enabled by providing separate shields for sensitive parts of the device (e.g., the input stage), laying out the components and interconnects to separate the input and output stages, carrying signals by coaxial cables at sensitive levels as well as very high levels, providing a good filtering of the power supply, etc. The purpose of this paper is to point out to some sources of the parasitic coupling that may be overlooked and which can cause severe problems, in particular at GHz frequencies.

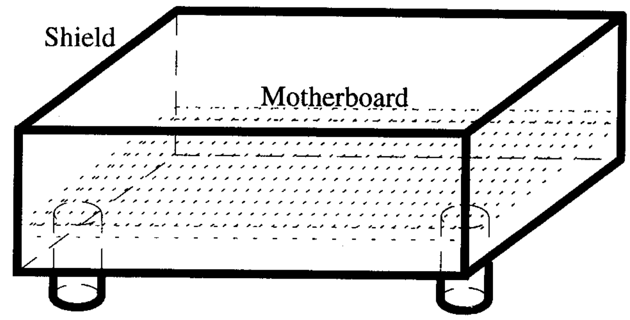


Figure 1. Simplified view of an RF device.

Multilayered PCBs are used as modern technology solutions, in particular in digital devices, as they can provide a high packaging density of signal interconnects. However, in RF devices, classical, double-sided boards still dominate. RF interconnects within a device are primarily made as transmission lines. These lines guide the electromagnetic energy, and reduce the signal cross-talk and distortion. For the classical boards, microstrip lines and coplanar waveguides dominate in practice. For the multilayered boards, striplines are most important. Section II deals with printed-circuit lines and related EMC problems.

Power-distribution conductors, used to feed active elements, can cause problems due to inadequate filtering and parasitic resonances, as described in Section III. Various active and passive components, integrated circuits, and plug-in boards are mounted on the surface of the motherboards. Their interconnecting pins and internal conducting paths can cause parasitic coupling, treated in Section IV. The metallic shield of the RF device behaves like a waveguide resonant cavity, which can jeopardize the device performance, as described in Section V. Sections II-V contain some practical data that show levels of the parasitic coupling and give recommendations for reducing it.

The parasitic coupling is usually classified as the conductive coupling (by means of conducting paths) and the radiation coupling (by means of stray electromagnetic fields). At high frequencies, it is often hard to distinguish between these two mechanisms. For example, a printed line can pick up a signal at one end due to a radiation coupling, and then carry the signal to the other end by means of a guided wave. The transmission along the line is enabled by the conductors, which may be described as the conductive coupling, although the wave propagation can hardly be reduced only to the conductive effects.

In the analysis of internal EMC, it is convenient to distinguish between broadband and resonant coupling mechanisms. The broadband coupling occurs, for example, between two transmission lines or between two mounted components, elevated above the motherboard. This coupling exists in a broad frequency range, although it is usually more pronounced at high frequencies. Examples are given in Sections II and IV. The resonant coupling is caused by self resonances of various structures within the devices, such as reactively terminated transmission lines, slots in the PCB, slots between the PCB and the metallic box of the device, box resonances, improperly grounded coaxial lines that extend within the box, wires, etc. The resonant coupling is pronounced near one or several discrete frequencies. Examples are given in Sections II, III, and V.

II Printed-Circuit Transmission Lines

High-frequency interconnects between various parts of the device must be designed so to securely guide the electromagnetic energy. At the present state of the art, these interconnects are predominantly made using printed-circuit lines, which are a compromise between the quality and cost. There exists a variety of printed lines. In the RF techniques, the microstrip lines (MSL), coplanar waveguides (CPW), and striplines (SL) dominate. Cross sections of these lines are shown in Figure 2. The striplines can be made only on multilayered boards.

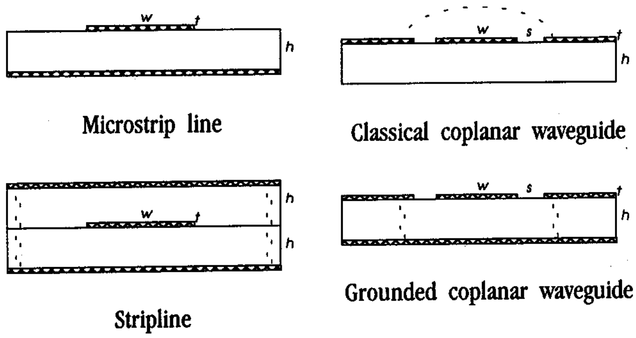


Figure 2. Cross sections of some printed transmission lines.

A MSL consists of a hot conductor (strip), which is printed on one face of the board. The other face is covered by a metallic foil, which plays the role of the ground. In the basic version, a CPW consists of a hot strip conductor and two wide grounded strips on the same face of the substrate. These two grounded strips must be densely interconnected by wire jumpers, to keep the strips at the same potential. A technologically better solution is the grounded CPW. It consists of the basic CPW with an added metallic foil on the other face of the PCB, like a MSL. The three grounded conductors are densely interconnected by a series of vias. A SL consists of a hot strip sandwiched between two ground planes, which must be well interconnected. In contrast to the MSL and CPW, the SL has a practically homogeneous dielectric. Note that all three structures are transmission lines, and that the CPW is improperly called a waveguide. The dominant wave on the SL is the TEM mode, while the dominant waves on the MSL and CPW are quasi-TEM modes.

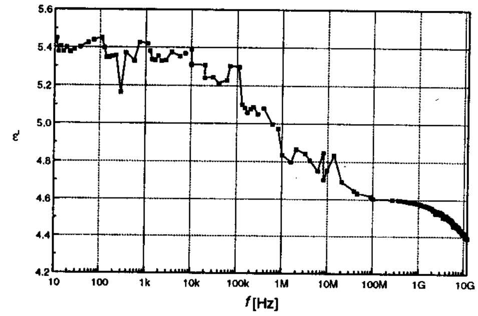
In the RF and digital techniques, FR-4 is the predominantly used substrate for PCBs. It consists of a glass cloth and epoxy resin. The basic advantage over other substrates is a relatively low cost and good mechanical properties, while high dielectric losses are the basic disadvantage. Because of the losses, the relative permittivity of FR-4 has large variations over frequency, as shown in Figure 3 [1]. The material properties also vary from manufacturer to manufacturer, and even from lot to lot.

For a given substrate, the first task in the design of a printed line is the evaluation of the cross-sectional dimensions to obtain a certain characteristic impedance. In the RF techniques, the standard impedances are 50 W for the majority of professional equipment and 75 W for TV equipment. As an example, Table 1 shows typical variations of the substrate parameters and printing tolerances, along with the corresponding influence on the characteristic impedance of a MSL, whose nominal impedance is 75 W. The nominal data for the line are: substrate thickness $h = 59\,\mathrm{mil}$, metallization thickness $t = 2.8\,\mathrm{mil}$ (copper, $2\,\mathrm{oz/sq.in}$), relative permittivity $\varepsilon_r = 4.3$, loss tangent $\tan\delta = 0.018$ (at 1000 MHz), copper conductivity $\sigma = 56\,\mathrm{MS/m}$ and strip width $w = 51\,\mathrm{mil}$. The attenuation coefficient of this line at 1000 MHz is $\alpha = 2.6\,\mathrm{dB/m}$ [2], about 1/5 of which comes from the conductor losses, and 4/5 from the dielectric losses.

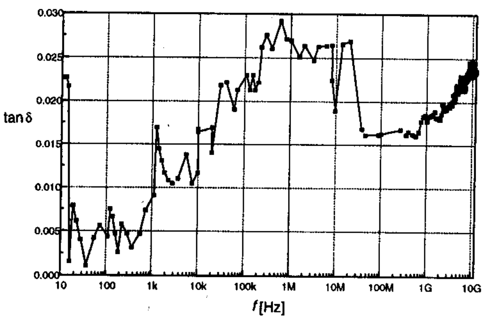
Table 1. Characteristic impedance of microstrip line for typical variations of dimensions and relative permittivity. The first row is for the nominal data, and the last two rows show the worst cases.

w [mil]	t [mil]	h [mil]	ε_r	Z_c [Ω]
51	2.8	59	4.3	75.0
52	2.8	59	4.3	74.4
50	2.8	59	4.3	75.7
51	*3.3*	59	4.3	74.8
51	*2.3*	59	4.3	75.2
51	2.8	*64*	4.3	77.9
51	2.8	*54*	4.3	71.9
51	2.8	59	*4.55*	73.2
51	2.8	59	*4.05*	77.0
50	*2.3*	*64*	*4.05*	80.8
52	*3.3*	*54*	*4.55*	69.3

The CPW has the slot width (s) as the design parameter. For the same substrate as for the MSL and for the characteristic impedance of 75 W, Table 2 shows the strip width (w) as a function of the slot width [2]. Reducing the slot width decreases the electromagnetic coupling with the environment, decreases the sensitivity to the substrate thickness and permittivity, but increases the sensitivity to the printing tolerances. If the slot is wide, the CPW resembles the MSL.



(a)



(b)

Figure 3. Dielectric parameters of FR-4 versus frequency.

Table 2. Strip width (w) of a grounded coplanar waveguide versus the slot width (s).

s [mil]	15	20	40	80
w [mil]	18	23.5	38	47

We also give two examples of the design of a SL on the FR-4 substrate [2]. For the substrate thickness $h = 12$ mil and metallization thickness $t = 0.7$ mil, the strip width is $w = 3.4$ mil for the 75 W characteristic impedance. The attenuation coefficient of this line at 1000 MHz is $\alpha = 5$ dB/m, about 2/3 of which comes from the conductor losses, and 1/3 from the dielectric losses. For a thicker substrate, $h = 30$ mil, the strip width is $w = 10$ mil, and the attenuation coefficient is $\alpha = 3.2$ dB/m (about 1/2 comes from the conductor losses). Similarly as with the CPW, the ground conductors of a SL must be densely interconnected, usually at a distance not exceeding 1/10 of the wavelength in the dielectric at the highest operating frequency.

At the motherboard, there usually exist several printed lines, which can be arbitrarily positioned. These lines are not fully shielded, so that there exists parasitic coupling. The coupling level depends on the particular kind of the lines, cross-sectional dimensions, substrate properties, line lengths, and frequency. Under similar conditions, the coupling is highest between MSLs, and lowest between SLs. The numerical analysis of this coupling can be performed using various techniques. If the lines are arbitrarily positioned, a three-dimensional (3D) tool is required [3,4]. If the lines are parallel, one can start with the two-dimensional (2D) analysis to evaluate the matrix parameters of the coupled lines [2], followed by the circuit-theory approach to evaluate the response [5]. Such a combined analysis is significantly simper and faster than the 3D approach. It yields accurate results if the length of the coupled lines is at least twice the distance between the lines.

As an example, we consider two parallel coupled lines, whose length is L. Such a structure is the simplest example of a multiconductor transmission line, and it can be considered as a four-port network (Figure 4). Suppose that one port is driven by a generator, while the remaining ports are matched. The main signal flow is from the generator, along the driven line, to the other port of that line. This signal is delayed (due to the finite velocity of wave propagation), attenuated (due to the conductor and dielectric losses), and distorted. The distortion principally comes due to the losses (including the dielectric dispersion), influence of the other, coupled line (which enables two modes to propagate), and due to the hybrid electromagnetic field structure. Although the distortion can be significant, it is smaller than if the signal is transmitted using arbitrarily shaped conductors that do not form a transmission line. At the ports of the coupled line, cross-talk signals appear. The near-end cross talk (at the generator end) comes from a mismatch between the generator network and the multiconductor line, and from the reflection at the far end. With MSLs and CPWs, the far-end cross-talk is stronger than the near-end cross-talk, and it is predominantly caused by different propagation velocities of the two modes. With SLs, the near-end cross-talk is usually stronger than the far-end cross-talk.

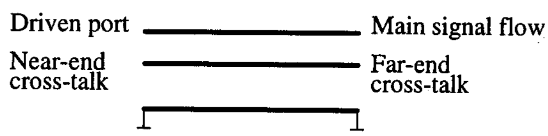


Figure 4. Two coupled lines as a four-port network.

Figure 5 shows the frequency dependence of the cross-talk for two coupled MSLs, whose length is 100 mm and separation distance 500 mil [2,5]. The other data are the same as for the single MSL given earlier. The near-end cross-talk is, approximately, a periodic function of frequency or the line length, because of the interference of the direct coupling at the near end and the wave reflected from the far end. In the frequency range considered, the far-end cross-talk steadily increases. Table 3 shows the maximal cross-talk level in the frequency range 0-1200 MHz for several separations and line lengths.

Figure 6 shows the frequency variations of the cross-talk for two coupled grounded CPWs, whose slot width is 40 mil, length 100 mm and separation 500 mil [2,5]. The other data are the same as for the single CPW given earlier. At lower frequencies, the cross-talk is qualitatively similar to that for two coupled MSLs (Figure 5), except that the level for the CPWs is lower. However, at higher frequencies, the cross-talk increases due to a resonance (at about 3 GHz) of the ground conductors, which are interconnected at a spacing of 25 mm. The cross-talk at high frequencies can be reduced by increasing the density of these interconnections, which adds to the manufacturing cost. Table 4 shows the maximal cross-talk in the band 0-1200 MHz for several separations and lengths.

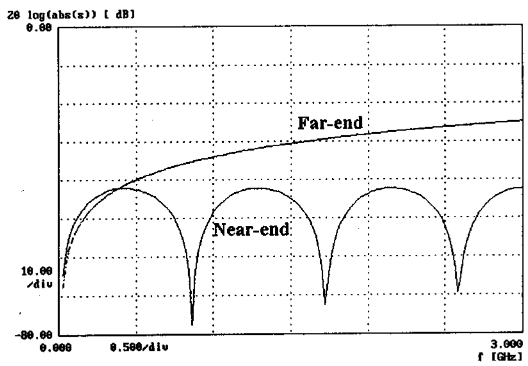


Figure 5. Cross-talk between two coupled microstrip lines when the separation is 500 mil and length 100 mm, versus frequency.

Table 3. Cross-talk [dB] at 1200 MHz between two coupled microstrip lines versus the separation (d) and length (L).

d [mil]	L=25 mm	L=50 mm	L=100 mm
100	-21.1	-21.2	-16.3
200	-30.6	-27.3	-21.5
500	-43.0	-38.3	-32.4
1000	-52.1	-47.3	-41.4
2000	-60.5	-55.6	-49.8
5000	-70.2	-65.3	-59.5

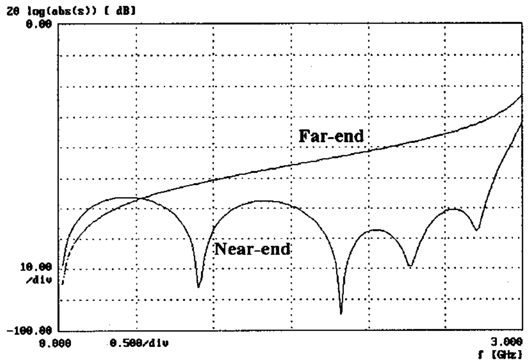


Figure 6. Cross-talk between two coupled grounded coplanar waveguides when the slot width is 40 mil, separation 500 mil, and length 100 mm, versus frequency. Short-circuiting vias are at a spacing of 25 mm.

Figure 7 shows the frequency dependence of the cross-talk for two coupled SLs, when the substrate thickness is 12 mil, line length 100 mm, and separation 20 mil [2,5]. The other data are the same as for a single SL, as given earlier. It is assumed that the two ground planes are interconnected densely enough to prevent any resonance effects. The near-end cross-talk is higher than the far-end cross-talk. Figure 8 shows the maximal cross-talk level versus the separation between the striplines for two substrate thicknesses (12 mil and 30 mil). For the thicker substrate, the total thickness of the PCB is practically the same as for the examples of MSLs and CPWs given before. The coupling between the SLs in this case is weaker than for the other lines. The modal velocities for coupled SLs are practically equal, so that the corresponding dispersion is much smaller than for the open lines. Hence, SLs are often used for the transfer of fast digital signal in multilayered PCBs. The disadvantage of SLs (excluding the higher manufacturing cost) is higher losses due to a small strip width and because the complete field is localized in the dielectric. The dielectric losses can be significantly reduced using high-quality but more expensive substrates. For the 30 mil substrate, the total losses can thus be reduced by a factor of about 2. For the 12 mil substrate, the reduction is for about 1/3, at 1000 MHz.

Table 4. Cross-talk [dB] at 1200 MHz between two coupled grounded coplanar waveguides versus the separation (d) and length (L) when the slot width is 40 mil.

d [mil]	L=1"	L=2"	L=4"
100	-31.7	-30.4	-26.9
200	-44.5	-42.4	-36.5
500	-58.4	-54.6	-47.4
1000	-68.7	-63.0	-57.2
2000	-75.4	-69.5	-63.6
5000	-84.2	-78.3	-72.4

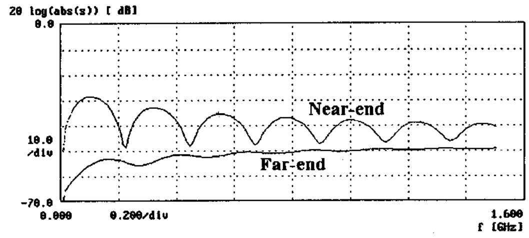


Figure 7. Cross-talk between two coupled striplines when the substrate thickness is 12 mil, separation 20 mil, and length 100 mm, versus frequency.

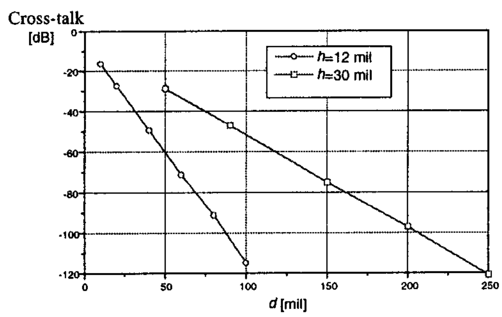


Figure 8. Maximal cross-talk level between two coupled striplines versus the separation (*d*) for two substrate thicknesses (*h*).

The coupling between MSLs and CPWs can be reduced if these lines are covered by a metallic plate, so that the structure approaches SLs. Hence, the isolation between transmission lines that are located on the bottom face of the PCB in Figure 1 is better than between lines on the top face as the PCB is closer to the floor than to the ceiling of the metallic box. The coupling between the lines increases if the lines are located close to an edge of the PCB.

All the examples presented above show cases of broadband coupling. Resonances of structures that are located near the lines can contribute to the parasitic coupling. The resonant coupling can be caused, for example, by slots in the PCB that are made for mounting various components, which can resonate at certain frequencies. The resonant coupling can be caused by transmission lines that are poorly matched at their terminals. One example was the increased cross-talk due to the CPW ground planes. Another example is given in Figure 9. This figure shows the cross talk between the same two MSLs as in Figure 5 when there is the third line in the middle, which is open-circuited at both ends [2,5]. The resonant coupling can be caused by waveguide resonances of the metallic box enclosing the device (see Section V), slots between the PCB and the box, slots in the box, coaxial lines and wires suspended above the motherboard, etc. In the design, it is necessary to identify such potentially dangerous structures and prevent the resonances to provide a stability of the device performance.

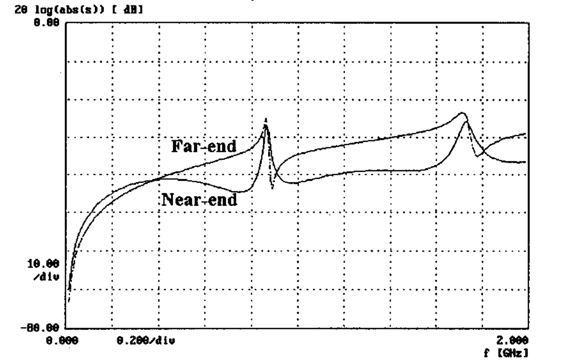


Figure 9. Cross-talk between two coupled microstrip lines with the third open-circuited line in the middle, when the separation is 500 mil and the length is 100 mm, versus frequency.

III Coupling by Power-Distribution Lines

The power used to feed the active components is distributed over the PCB from the power supply using a system of conductors. Usually, the PCB has a metallization over most of its surface, which is used as the ground. Hence, a good practice is to make the power-distribution conductors in the form of transmission lines. In multilayered PCBs, the power is usually distributed using metallic foils buried in the PCB, which are simultaneously used as the high-frequency signal ground. In any case, the power-distribution conductor is connected at one end to the power supply. At this location, there is a low-pass filter. As observed from the conductor, a very high or a very low impedance is seen, depending on the filter topology. Hence, for the high-frequency signal, the power-distribution conductor appears as either open-circuited or short-circuited to ground. A similar situation occurs at the active components, where additional filtering is usually provided. At high frequencies, the power-distribution conductor behaves like a resonator, of a relatively high quality factor. Hence, it can contribute to the instability of the active elements. This conductor can pick up the RF signal directly from the active component (e.g., the output stage of an amplifier) or at any other location due to the electromagnetic coupling with high signal level conductors. At the resonances, this RF signal spreads all over the conductor. It can be coupled to the input stage of the amplifier through the power-supply filter that has a small attenuation or by an electromagnetic coupling.

To illustrate the problem, Figure 10 shows the power-distribution line with low-pass filters at both ends. Figure 11 shows the transfer function between the ports of this system [3]. The resonances of the line cause a poor isolation between the ports. In this analysis, the coils and capacitors were treated as ideal elements. In practice, these elements have parasitic effects that reduce the performance of the filters, so that the isolation between the ports is even much worse than shown in Figure 11. Similar effects occur for the electromagnetic-field coupling with the power-distribution line.

Figure 10. Schematic of power-distribution line and low-pass filters at its terminals.

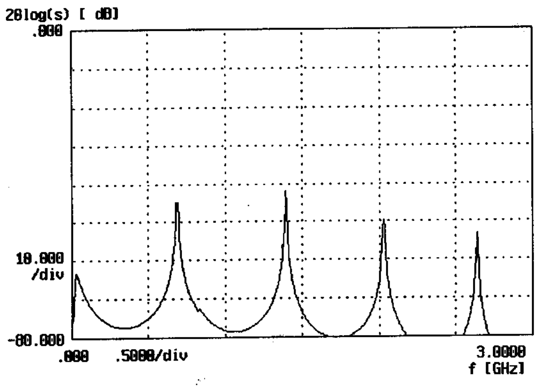


Figure 11. Coupling by power-distribution line of Figure 10.

The simplest way of improving the isolation is to suppress the resonances of the power-distribution conductor. This can be achieved, for example, using several resistors connected between the conductor and the ground. In series with each resistor, a capacitor should be placed to avoid the d.c. dissipation in the resistors. This leads to the structure shown in Figure 12, for which the transfer function is shown in Figure 13 [3]. A similar improvement is also achieved for the case of the electromagnetic coupling.

A particular problem is the cross-over region of the RF line and the power-distribution line. If the lines are located on the same face of the PCB, the power-distribution line can skip over the RF line using a wire jumper. If the lines are on opposite faces of the PCB, the best policy is to keep intact the ground of the RF line. A jumper can be used for the power-distribution line to skip this ground. If, however, the cross-over is to be completely manufactured as a printed structure, both lines should be grounded coplanar waveguides, as shown in Figure 14. In the cross-over region, the grounds on both faces should be well interconnected. In any case, a parasitic coupling occurs in this zone between the two lines. The resonances of the power-distribution line can cause a coupling to distant parts of the circuit. There is also a reflection on the RF line, particularly pronounced near the resonances of the power-distribution line, as shown in Figure 15 [2,5]. These problems can be solved by suppressing the resonances of the power-distribution line and by a compensation of the discontinuity on the RF line.

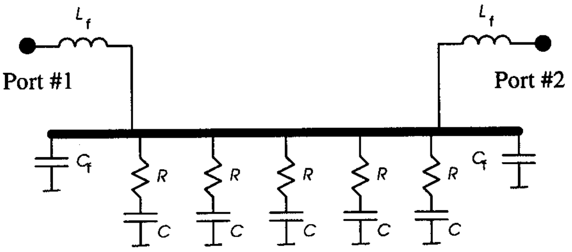


Figure 12. Schematic of power-distribution line of Figure 10 with the resonance-suppressing elements.

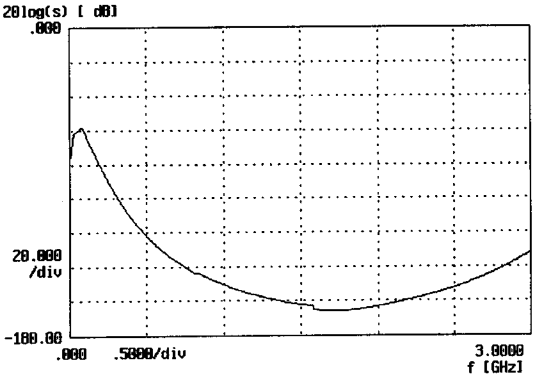


Figure 13. Coupling by power-distribution line of Figure 12.

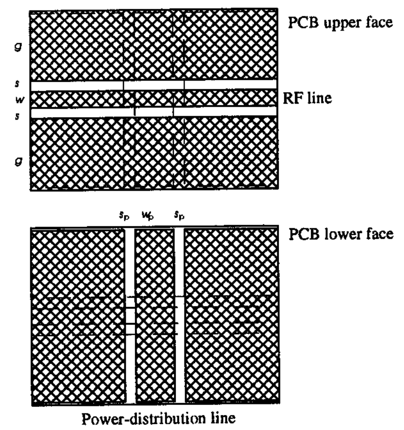


Figure 14. Cross-over of RF and power-distribution lines.

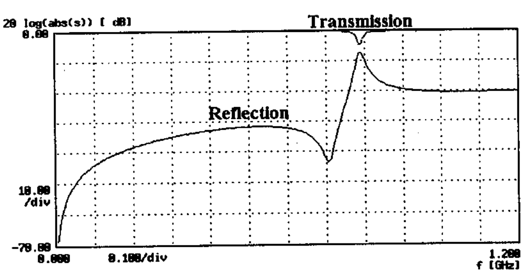


Figure 15. Reflection and transmission coefficients on RF line of Figure 14 when the power-distribution line has resonances.

IV Coupling Among Mounted Components

Various discrete and integrated components, as well as plug-in boards are mounted high above the motherboard. These components can be strong sources of the electromagnetic fields, as well as receivers of this field. Most components are small with respect to the operating wavelength. Some of them behave, approximately, as small loops (magnetic dipoles), others like electric dipoles. However, the behavior of most components can be described only by a combination of the two dipoles. The coupling between two components decays relatively quickly with increasing the separation, but it can be influenced by the surrounding parts and, in particular, by various resonances. As an example, Figure 16 shows the model of a plug-in board placed close to a MSL [4], both located above the ground plane. The height of the conducting path on the board is 15 mm, and the width 12 mm. The length of the MSL is 50 mm, and the other data are the same as in Section II. Figure 17 shows the coupling between the board and the line versus frequency and the separation.

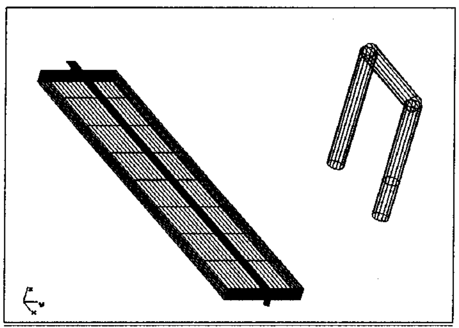


Figure 16. Plug-in board near transmission line.

The parasitic coupling can be lowered by reducing the dimensions of the loop that couples with the electromagnetic field. For example, the plug-in board can be made in MSL or CPW techniques so that it has a grounded backplane foil. This foil can be connected to the motherboard ground by one or more contacts. Qualitatively similar results as in Figure 17 are obtained for the coupling between any other mounted component and a nearby transmission line, as well as for the coupling between two mounted components. The coupling level depends on the mutual position and distance, as well as on the electric and magnetic moments. These moments are approximately proportional to the height of the loop above the ground plane. Hence, to reduce the coupling, the components should be mounted as low as possible, and their pins should be very short. A similar problem is the coupling between the input and output of a mounted multiport component (e.g., an integrated amplifier). The component grounding must be carefully made so that near each signal pin there is at least one grounding pin.

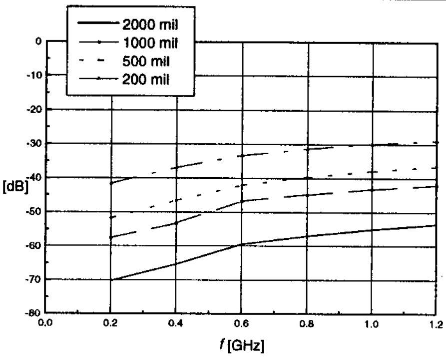


Figure 17. Coupling between plug-in board and line of Figure 16 versus frequency and separation.

V Box Resonances

To reduce the electromagnetic coupling between the RF device and its environment, i.e., to fulfill the external EMC requirements, the device is usually well shielded by a metallic box, which can have only small openings. All transmission lines that pass through the shield must be shielded lines (usually coaxial lines) or well filtered. The metallic box, however, behaves electromagnetically like a resonant cavity. At the box resonant frequencies, the internal EMC of the device is easily ruined. We shall consider the box to have a rectangular shape. As an example, we take its dimensions to be a=300 mm, b=200 mm, and d=80 mm. The resonant frequencies of the box can be evaluated as

$$f_{rmnp} = \frac{c}{2}\sqrt{(\frac{m}{a})^2+(\frac{n}{b})^2+(\frac{p}{d})^2} \ ,$$

where c is the speed of light in a vacuum, while m, n, and p are integers (0,1,2,...), where only one of them can be zero. For this example, the first 10 resonant frequencies are given in Table 5. In the case considered, $a>b>d$, so that the lowest resonance is in the TM_{110} mode. Even a loose coupling between an RF part and the cavity can excite a strong resonant field. This field, in turn, can excite a strong signal at any other part of the circuit, which can be far away. This can create a feedback that can cause an instability of the amplifier chain.

As an example, consider two plug-in boards located in a metallic box, as shown in Figure 18 [4]. We assume the same box dimensions as before, while the plug-in boards are approximated by loops whose height and width are 20 mm.

Table 5. Resonant frequencies of a box whose dimensions are a=300 mm, b=200 mm, and d=80 mm.

Mode #	m	n	p	f_r [GHz]
1	1	1	0	0.901
2	2	1	0	1.250
3	1	2	0	1.581
4	3	1	0	1.677
5	2	2	0	1.803
6	1	0	1	1.941
7	0	1	1	2.019
8	1	1	1	2.601
9	3	2	0	2.652
10	2	0	1	2.656

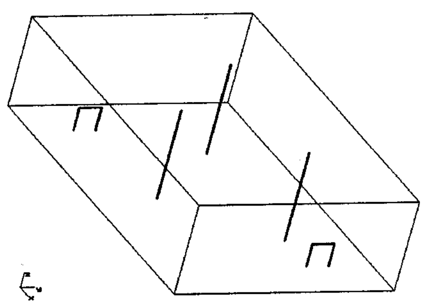


Figure 18. Two plug-in boards in metallic box and posts for controlling resonances.

Let us assume, first, that the box is removed and the plug-in boards are located above a ground plane. The broadband coupling between the plug-in boards is shown in Figure 19 by the solid line. The coupling is relatively weak and it steadily increases with frequency. Assume, now, that the plug-in boards are in the metallic box, as shown in Figure 18, but there are no posts that influence the resonances. The coupling between the plug-in boards is shown by the dashed line in Figure 19, which clearly shows the influence of the box resonances. Near these resonances, the coupling becomes very strong, even close to 0 dB (which would correspond to a galvanic contact between the loops). The box resonant frequencies can be increased by dividing the box into smaller units or by introducing appropriate reactive loadings. The objective is to shift the resonances above the operating band of the RF device and above the frequency where the active elements may become unstable. As an example, Figure 18 shows three metallic posts that interconnect the floor and the ceiling of the box. The effect of these posts is shown by the dotted line in Figure 19. A technically better solution is to dampen the box resonances by using resistive posts or by resistively loading the metallic posts. For example, if the resistance of each post is 300 W, the coupling between the plug-in boards is shown by the dash-dot line in Figure 19. The resonances can be dampened in other ways, like placing microwave absorbers on three non-parallel box walls. If the PCB divides the box into two parts, as shown in Figure 1, both resonant cavities must be dampened.

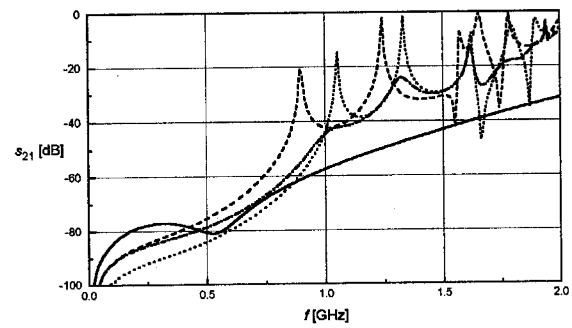


Figure 19. Coupling between plug-in boards of Figure 18: fff boards above ground plane; f f f boards in box; ñ ñ ñ boards in box with metallic posts; f ñ f boards in box with resistive posts.

VI Conclusions

This paper shows some sensitive spots that can cause parasitic coupling and thus jeopardize the internal EMC of RF devices. Consequences of the parasitic coupling can be, for example, distorting the transfer function of the device and instabilities of the active elements. Measures are also presented for curing the problems.

Instead of the standard classification of the parasitic coupling paths to the conduction and radiation coupling, this paper stresses two coupling mechanisms: the broadband and the resonant coupling. The broadband coupling occurs with printed transmission lines and components that are mounted high above the motherboard. The resonant coupling occurs because of self resonances of various structures within the device, like reactively loaded lines, slots, metallic enclosures, wires, etc.

Although the parasitic coupling can never be fully avoided, a careful design may reduce the coupling level to acceptable limits. The techniques for suppressing the broadband coupling paths are based on increasing the separation between the lines, taking a thinner substrate for the PCB, taking coplanar waveguides with narrower slots, using striplines instead of open lines (MSL and CPW), shielding of components that are erected high above the motherboard, using plug-in boards with a ground plane, making a layout of the PCB where the signal gradually increases from one end towards the other one, etc. The first step in suppressing the resonant coupling is to identify the parasitic resonators. They can be formed by lumped elements with their parasitics, combinations of lumped elements and transmission lines, reactively loaded lines, accidentally created transmission lines and cavities, etc. The next step is to select the suppressing technique. First, the resonator can be removed (e.g., by a galvanic contact all along a slot). Second, the lowest resonant frequency of a resonator can be shifted upwards above the limits of the operating range of the device and above the highest frequency where there exist possibilities of instabilities. This can be achieved by an appropriate reactive loading or splitting the resonator (e.g., by posts in the resonant box, dividing the box into several smaller parts, and short-circuiting the slot resonator in the middle). This technique should be applied with care, as it may have a counter effect of lowering the resonant frequencies. Third, it is possible to lower the quality factor of the resonators by an appropriate loading (e.g., by introducing resistors or microwave absorbing materials).

References

[1] Biljić, R.M., ***Characterization of dielectric substrates of microwave planar transmission lines***, D.Sc. Thesis, manuscript, School of Electrical Engineering, University of Belgrade, 1998.

[2] Djordjević, A.R., Baždar, M.B., Harrington, R.F., and Sarkar, T.K., ***LINPAR for Windows: Matrix Parameters for Multiconductor Transmission Lines (software and user's manual)***, Artech House, Boston, 1996.

[3] Djordjević, A.R., Baždar, M.B., Sarkar, T.K., and Harrington, R.F., ***AWAS for Windows: Analysis of Wire Antennas and Scatterers (software and user's manual)***, Artech House, Boston, 1995.

[4] Kolundžija, B.M., Ognjanović, J.S., Sarkar T.K., and Harrington, R.F., ***WIPL: Electromagnetic Modeling of Composite Wire and Plate Structures (software and user's manual)***, Artech House, Boston, 1995.

[5] Djordjević, A.R., Baždar, M.B., Vitošević, G.M., Sarkar, T.K., and Harrington, R.F., ***MATPAR: Scattering Parameters of Microwave Networks with Multiconductor Transmission Lines (software and user's manual)***, Artech House, Boston, 1989.

Summary: Some critical issues are considered that can jeopardize the internal electromagnetic compatibility of radio-frequency (RF) devices. Two coupling mechanisms are stressed: the broadband coupling (e.g., coupled printed transmission lines, components that are mounted high above the motherboard, etc.) and the resonant coupling (due to self resonance of reactively terminated transmission lines, slots in PCBs, metallic enclosures, etc.). Practical data are given for the coupling between various structures, including transmission lines and plug-in boards, that can be used to assess the parasitic coupling during the design stage of the device. Techniques are described for reducing the coupling and, in particular, for suppressing the spurious resonances.