

Microcontrollers

ApNote

AP2421

: Additional file
APXXXX01.EXE available

EMC by Simulation

Reduction of Electromagnetic Radiation by Detection and
Optimization of Critical Networks on a PCB during the Layout-process

Oliver Winkler / Siemens HL MC AT and ZT PP 5

1 Introduction 3

2 Electromagnetic Compliance 4

2.1 Signal Integrity, Emission and Susceptibility 4

2.2 The earlier - the better 4

2.3 The Power of EMC-Simulation 5

3 Theory of Electromagnetic Simulation 7

3.1 The reason of radiation 7

3.2 Near Field and Far Field 8

3.3 Differential and Common Mode Radiation 8

3.4 Modelling for simulation 9

4 IBIS - Input/Output-Buffer-Information-Specification 10

4.1 IBIS characteristics 10

4.2 Usage of IBIS 11

5 Simulating the Electromagnetic Scanning of PCB 12

5.1 Simulation input 12

5.2 Results of the Simulation 12

5.3 Simulation of a PCB 13

5.4 Measures 16

6 Usage of EMC-Simulation-tools 17

6.1 Measure to reduce emission 17

6.2 Improvement 17

6.3 Simulation Time 19

7 Conclusion 20

8 Attachment..... 21

AP2421 ApNote - Revision History		
Actual Revision : Rel. 01		Previous Revision : none (Original Version)
Page of actual Rev.	Page of prev.Rel.	Subjects (changes since last release)

1 Introduction

Unwanted electromagnetic radiation from electronic devices has become a serious problem worldwide raising the spectre of legal liability to say nothing of threat human life. All electronic devices emit electromagnetic radiation when operating. Since today's devices are more sensitive to electromagnetic fields, their normal operation can be disrupted by emissions from adjacent equipment.

To get under control this unwanted electromagnetic effects the attitude towards how to make devices electromagnetic compliant has to be changed, first. It has to be changed from costly "damage" control to EMC measures that have already been integrated in the PCB-design process [1]. To realize this there are different methods, of course. In the following it is shown some experience made by EMC-simulation tools.

2 Electromagnetic Compliance

2.1 Signal Integrity, Emission and Susceptibility

EMC consists of three major disciplines (also see Figure 1): Signal integrity, emission and susceptibility. Signal integrity contains all problems of crosstalk, reflection, delay and all its effects on rise and fall times. Signal integrity effects on the stable functionality of a electronic device. If any

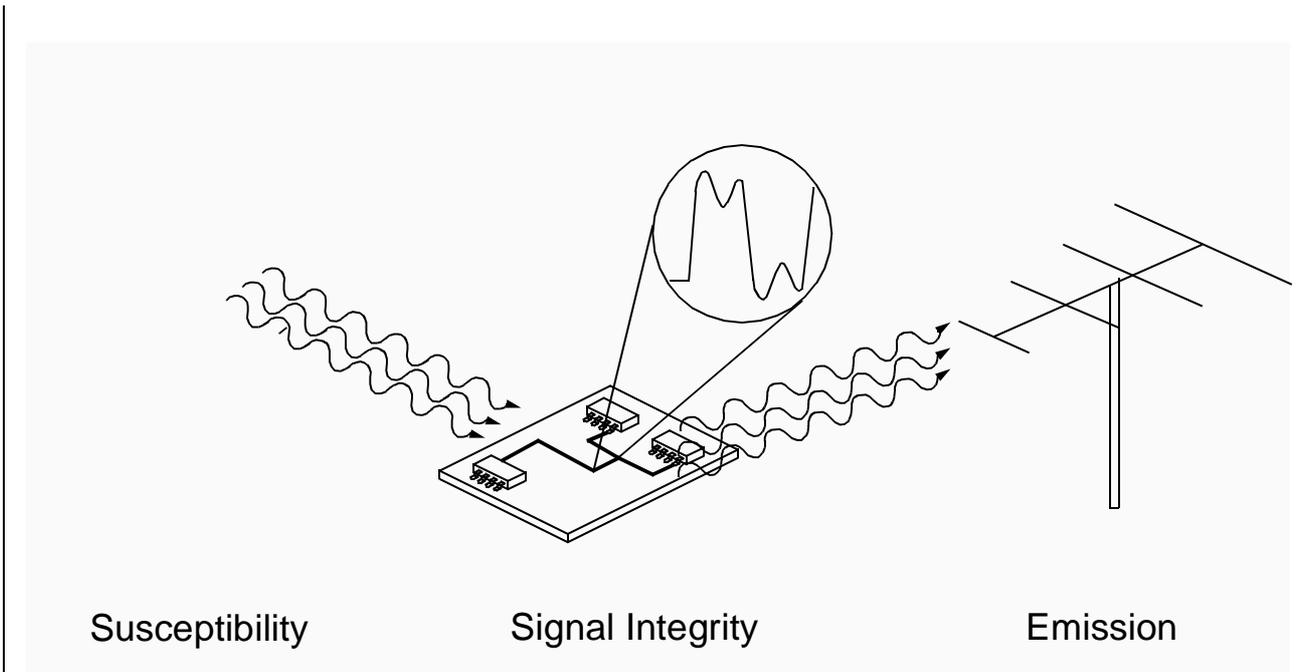


Fig. 1: Three Disciplines of EMC

of these effects mentioned above occur, the device would not work correctly. Signal integrity is relevant for high speed boards working at a clock frequency starting at 50 . . . 100 MHz. Emission means all kinds of electromagnetic fields produced by an electric device. Emission is a problem mostly caused by digital devices. At least by electromagnetic susceptibility a device is influenced by electromagnetic fields of another device. But it is mostly the analog part of a device which is influenced by susceptibility.

2.2 The earlier — the better

Electromagnetic compliance (EMC) must not simply be added to a device in a post-PCB-design process. This is due to the fact that the most post-design measures do not fight the reasons but the results of electromagnetic incompliance. As Figure 2 shows the later measures to improve EMC are made the smaller the choice of measures becomes and — what is most important — the more expensive the measures and thus the product itself becomes. In other words, EMC has to be designed with the PCB or even the whole device.

Since plenty of EMC-simulation-software was developed in recent years, now, some became tools that help a designer of an electronic device to make his product electromagnetic compliant. By

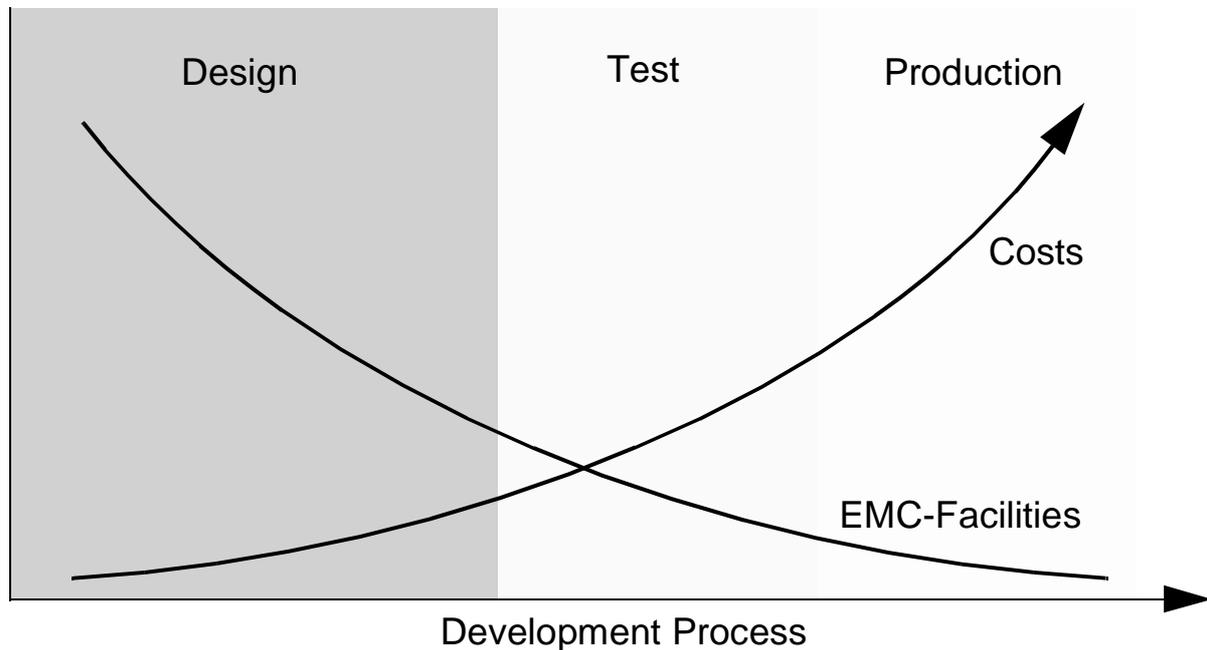


Fig. 2: EMC-Facilities and their Costs as a Function of the Development Process

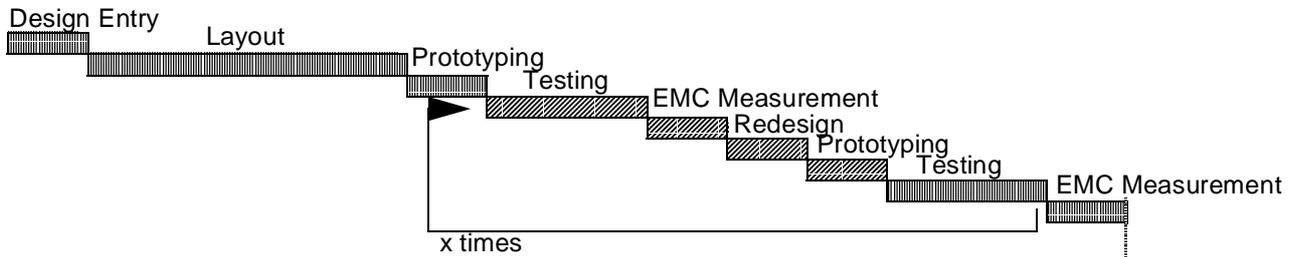
simulation of critical parts of a PCB-design the designer gets spontaneous feedback of his attempt to increase EMC by routing a net in a certain way. So, the optimization of a PCB-design can be done completely in the layout phase of a PCB-design — without building up a physical prototype but a virtual prototype. Verification and optimization within a virtual prototype shortens iteration loops and reduces time for measurements in a test-chamber (Figure 3).

To summarize, an EMC-simulation-tool can definitely not tell if a device will or will not pass the test standard given by law. An EMC-simulation-tool will help to design a device as good as possible; it will help to improve the quality (EMC) of the device; and it will help to save time and costs.

2.3 The Power of EMC-Simulation

To be honest with EMC-Simulation tools, there are many things which can not be realized by simulation. But the further development continues more and more features become implemented. Figure 4 shows an overview of the simulation tools' productive usage for each discipline. It is easy to see that tools for signal integrity are already integrated in the design process. There are two kinds of tools for the computation of emission. Tools for calculating emission of a PCB are in the second half of 1996 being implemented successfully in several design environments. Although there are already tools for calculating emission of whole devices (i.e. PCB, enclosure and cables) available it will last at least until May/June 1997 to guarantee their efficient use. Of course, the use of a tool for PCB emission will be helpful and almost a soft requirement to reach this goal. And to have a future look, tool which allow the simulation of electromagnetic susceptibility will be available and useable in 1998.

Without EMC Simulation:



With EMC Simulation:

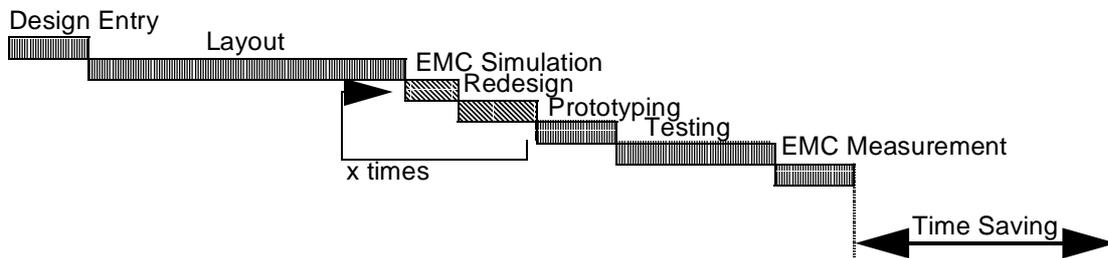


Fig. 3: PCB Development Process with and without EMC-Simulation

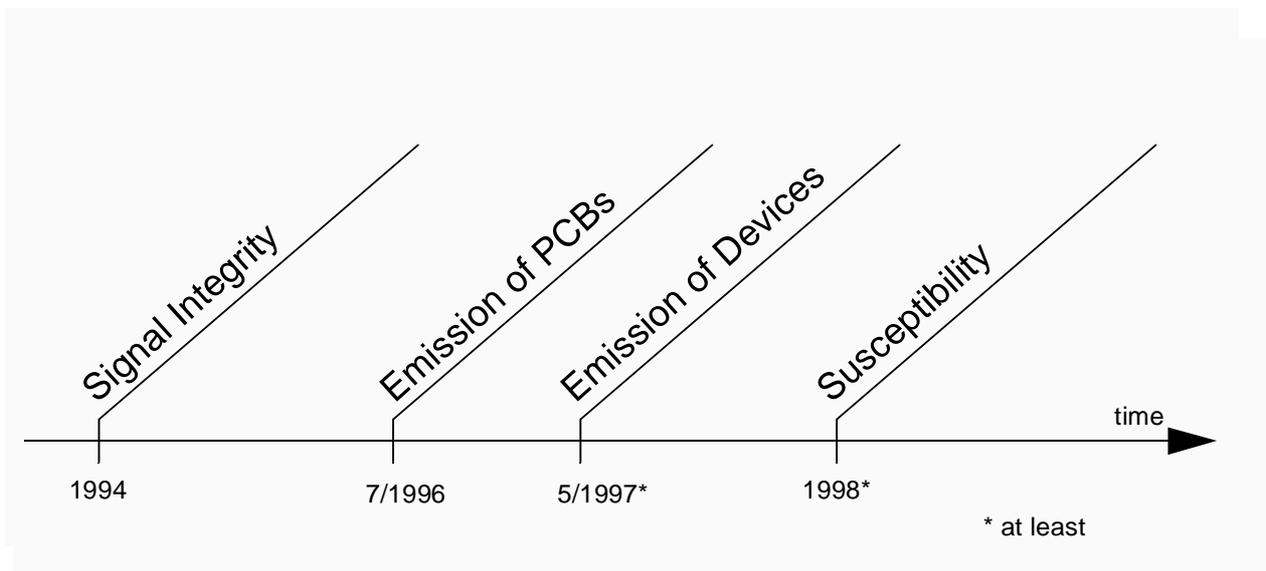


Fig. 4: Roadmap for integration of EMC-simulation tools into product design process

3 Theory of Electromagnetic Simulation

3.1 The reason of radiation

Having a closer look at a certain net on a PCB this net can be divided into several trace segments which are connected by several nodes. Figure 5 shows that each net can be modelled by a source,

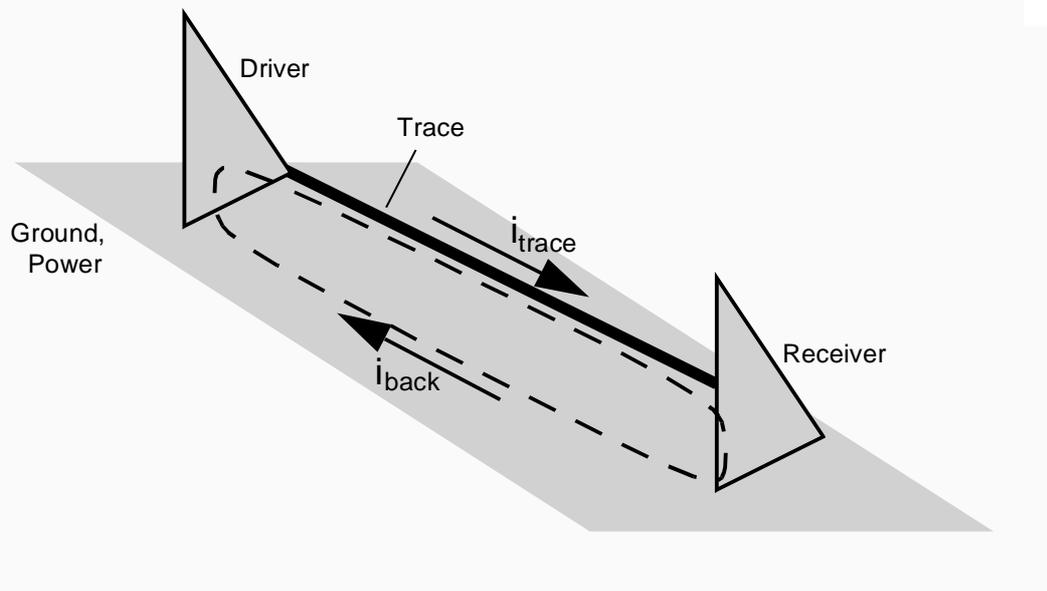


Fig. 5: Loop antenna structures

some traces and a sink. Remember that in most cases the traces are located right over a ground plane. Current now flows from the source to the sink. Since both source and sink are connected with the ground plane there are currents within the ground plane backwards, from the sink to the source. It is well-known that these currents flow right beyond the traces. So it is easy to see that the current behaves like current in a loop antenna structure. And so these loop antenna structures have the same effects like a magnetic loop antenna: They cause magnetic radiation.

3.2 Near Field and Far Field

Figure 6 shows the difference between the near field and the far field. The far field gives information

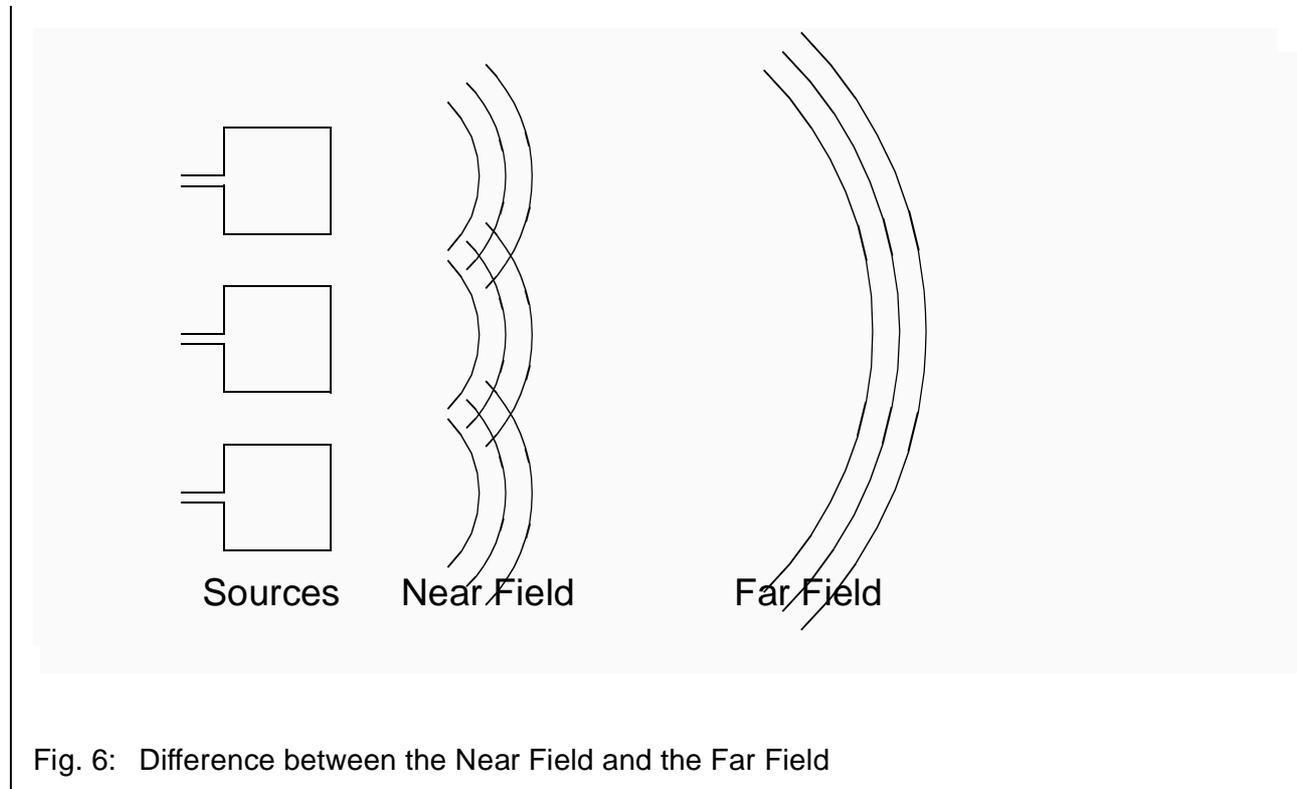


Fig. 6: Difference between the Near Field and the Far Field

about the total energy radiated by all antenna structures and regards all sources as one point source. Actually most measurements done in a test chamber are measurements of the far field. Although the final measurement is of the type far field measurement, during the layout phase it is necessary to get some knowledge about the near field. This is due to the major goal to detect and optimise critical nets on a PCB. The near field contains information about the several sources which created the near field. By inspecting the near field of a PCB an area right on the PCB can be determined containing the source of the radiation. By having this detailed information about the intensity and the location of radiation's source it is obvious which net has to be optimised in order to bring radiation down.

3.3 Differential and Common Mode Radiation

There is also a distinction whether the radiation is differential mode or common mode. Differential mode radiation comes from the nets i.e. from the currents in the nets (so-called differential mode currents) on the PCB themselves. Common mode radiation results from parasite effects like induced currents (so-called common mode currents) in mass areas, power areas and cables. Although the common mode currents are not as high as the differential mode currents most of a device's radiation is common mode radiation. Especially for far field analysis common mode radiation has to be taken into account.

3.4 Modelling for simulation

As already explained all nets going to be simulated are divided into several trace segments. Each trace has got at least one source and one sink. Whereas the trace segments' geometric (length, width and height) and material data (mostly copper) can be derived from the layout tool the information about current sources and sinks have to come from an internal library containing driver and receiver models. These library models consist of voltage-current-curves (v-i-curves) which characterize the pulldown or pullup behaviour of the driver/receiver. One way to obtain these models and their curves is by converting an IBIS-model (Input/Output-Buffer-Information-Specification) provided by the manufacturer of the component.

4 IBIS — Input/Output-Buffer-Information-Specification

Since component models e.g. spice models may show any information about technology details many component manufacturers refused to publish any simulation models. But simulation becomes more and more important, so a neutral standard has been created, called IBIS (Input/Output-Buffer-Information-Specification).

4.1 IBIS characteristics

The IBIS standard is neither fixed on any simulator type nor it is limited on any component manufacturer nor it is fixed on any hardware platform. Figure 7 shows how one IBIS-ASCII-file replaces plenty of simulator specific libraries. The lower part of Figure 9 shows that the IBIS model

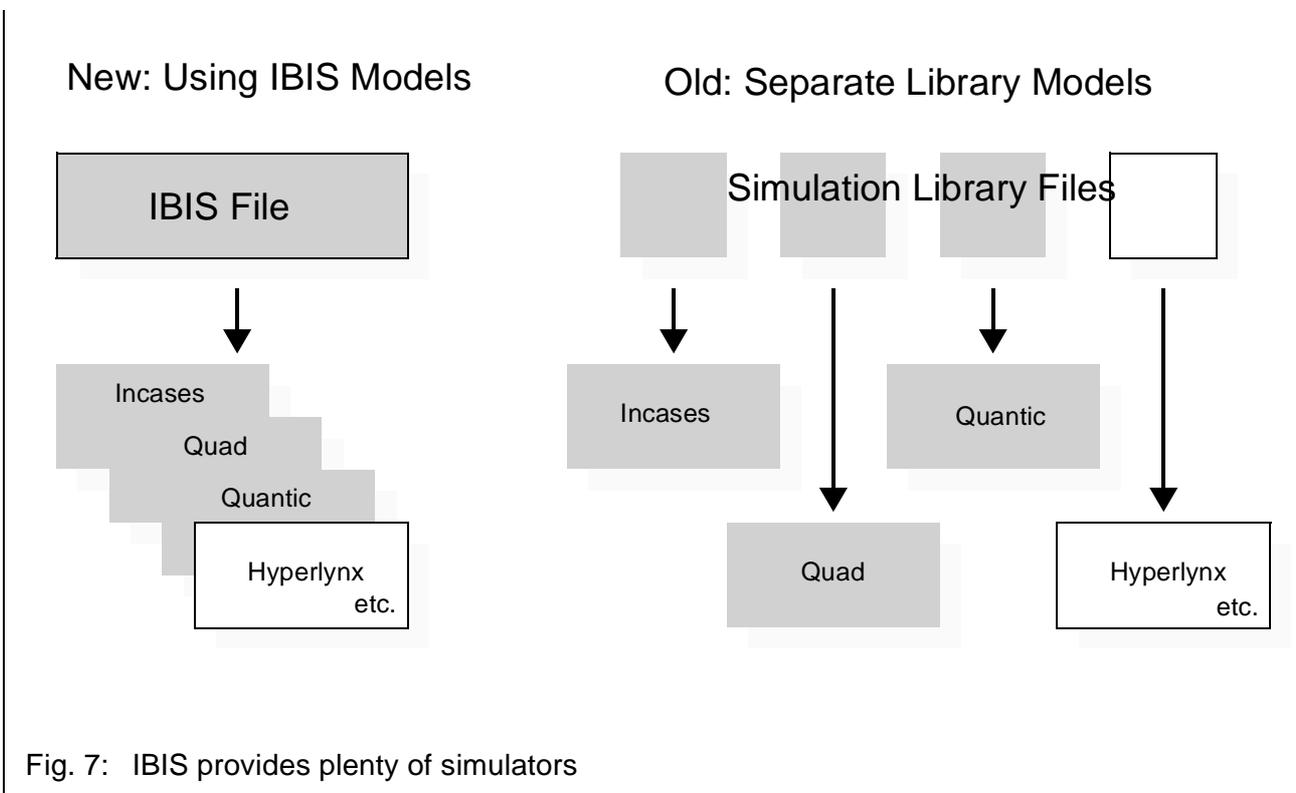


Fig. 7: IBIS provides plenty of simulators

has to be provided by the components' manufacturer. It can also be seen, that the IBIS file is accessed by the simulator via a converter provided by the software manufacturer.

The IBIS model is of the type behavioural and it contains model name, model type (e.g. input, output, i/o, tri-state, open-sink, open-source) and capacities. Further, there are data like v-i-curves for pullup and pulldown, curves for clamping diodes, package information and pin assignments. So, it is only a description of the components input and the output buffers, no information at all about its inside.

For more information contact <http://www.eia.org/eig/ibis/ibis.htm>

4.2 Useage of IBIS

An IBIS model can be used for signal integrity analysis like crosstalk, delay and reflection and even for EMC-simulation of PCB. However, effects like Vss-Bouncing and Vcc-Bouncing and even simultaneous switching noise can not be simulated by the IBIS model (Figure 8). These are effects

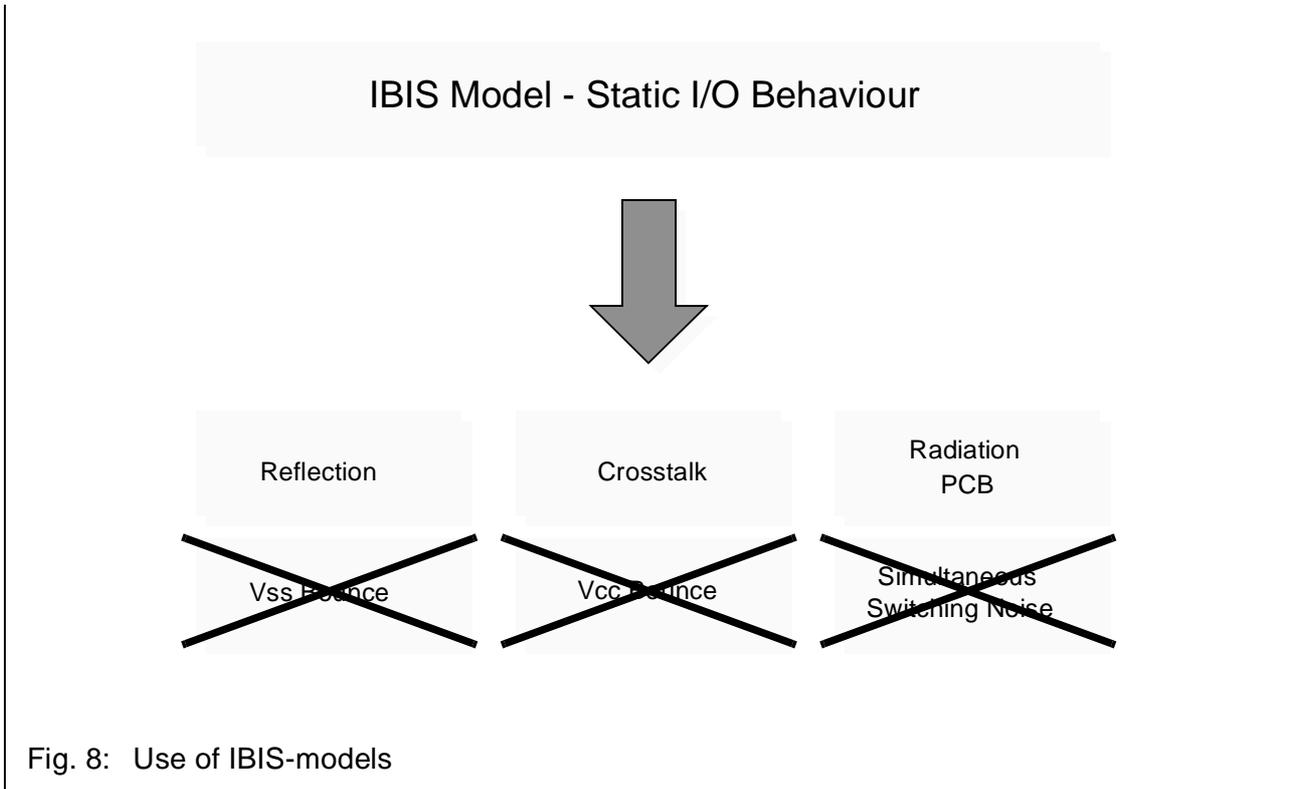


Fig. 8: Use of IBIS-models

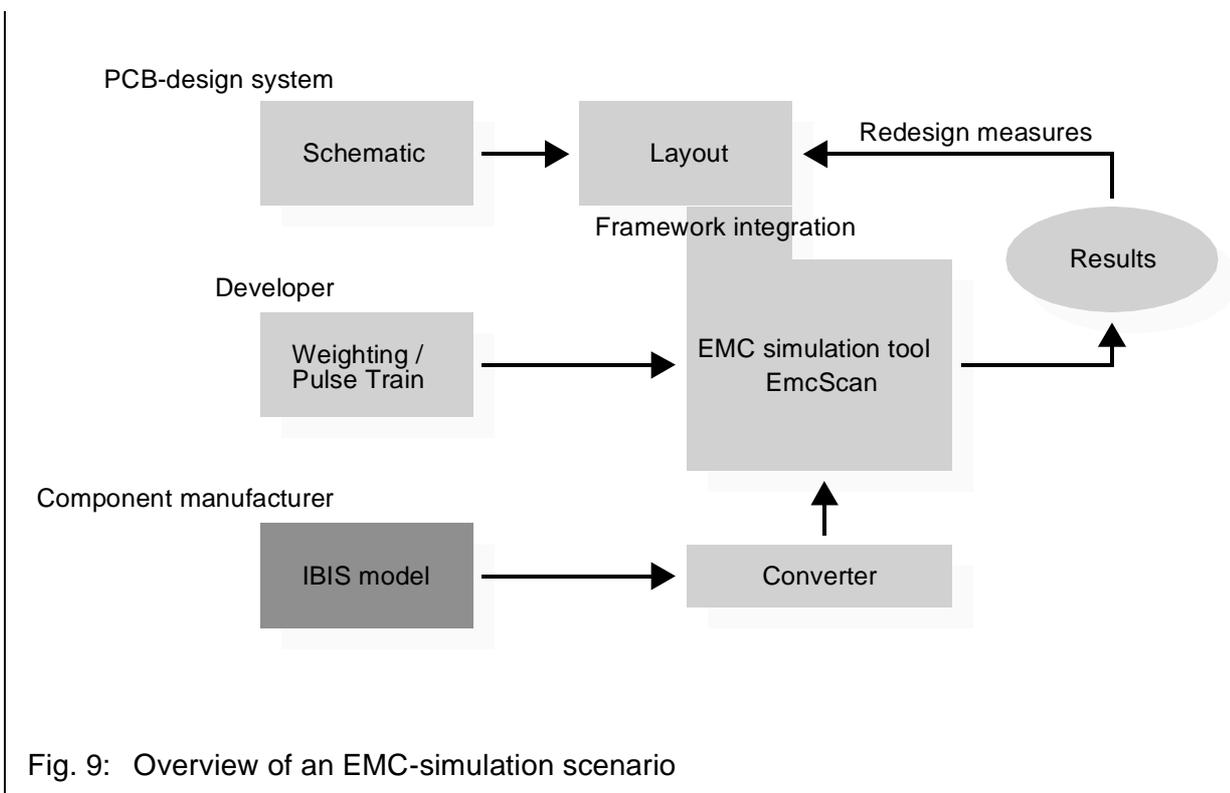
of the component itself that are not regarded in a simulation process, yet. But the more exact a simulation should become the more exact a simulation model has to be. Therefore, some additional information should be added to the IBIS-standard which would bring simulation and at least design and development of products one step further.

5 Simulating the Electromagnetic Scanning of PCB

The following describes both the design flow and the experience made with the EmcScan Simulator from Quantic Laboratories.

5.1 Simulation input

Figure 9 shows the whole scenario of an EMC-simulation process. After the layout has been generated fully or partially its data will be passed to the simulator. They contain geometric data of the nets, data about the components and also data about the board's layers and its materials. The extraction can be done within the layout tool's framework.



Another important input is the weighting of the nets. This weighting symbolizing the traffic on the nets defines a certain percentage of pulses for each net. It is obviously just a worst case estimation but including the library driver models (generated out of IBIS-models) exact computation becomes possible.

5.2 Results of the Simulation

The simulation computes several results. They can be accessed in the following manners:

- Amplitude of the magnetic field vs. frequency
- Amplitude of currents on the traces vs. frequency
- Colour map of the PCB showing the magnetic field in certain areas for a selected frequency
- Colour map of the PCB showing the current-density on the traces for a selected frequency
- Quality factor to compare several designs (near future development)

5.3 Simulation of a PCB

The simulation consists of the four steps shown in Figure 10. In the first step, the transmissionline-simulation, all selected nets are divided in several trace segments. For each trace segment current waveforms and voltage waveforms (Figure 11) are calculated regarding both crosstalk and

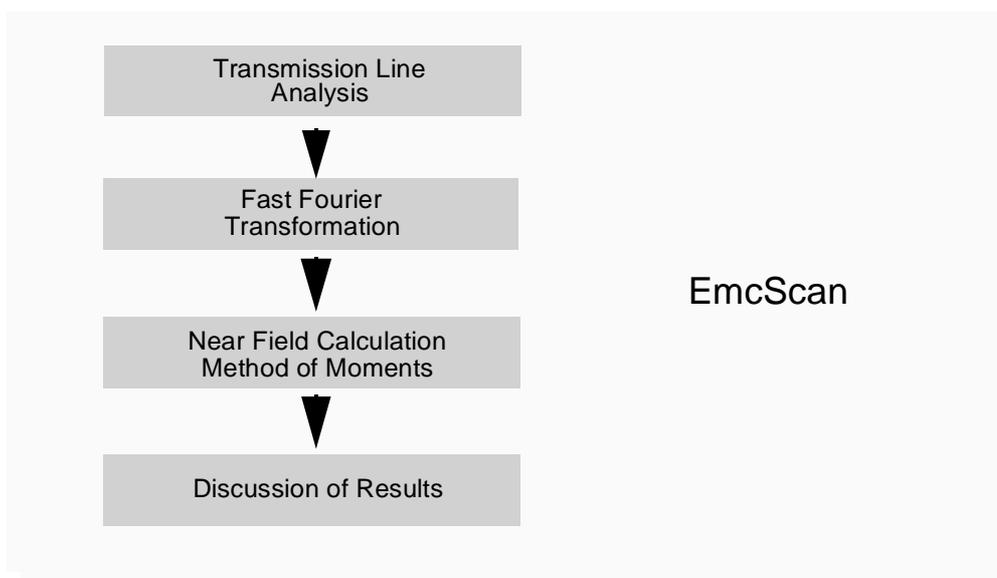


Fig. 10: Flowchart of the EmcScan simulation

reflection. Based on these results the Fast-Fourier-Transformation (FFT) processes the spectral analysis of each trace segment's current. Figure 12 shows the FFT result for one trace segment: the amplitude of current as a function of discrete frequency steps. Using these results and the theory of loop antenna structures mentioned above the radiated magnetic field is computed by the method of moments in step three. Again, each trace segment is taken into account. Figure 13 shows the result of the field calculation 5 mm above the PCB. Due to the method of modelling loop antenna structures producing magnetic radiation, the field shown in Figure 13 is of the type magnetic field. Kept in mind to do a worst case simulation this result is a summary of the highest field amplitudes. It gives an overview what frequencies may cause EMC-problems. Once such a potential problem has marked out, the locations of the radiating nets can be found by exploring a colour map (Figure 14) of the selected frequency. Red areas show locations of high radiation. A real hardware scanning of the PCB would produce a similar colour plot. However, the big advantage of a software scanning is that it takes place within the PCB-design process. Since the currents on the traces are responsible for the radiation the critical nets for this certain frequency can be fixed easily by exploring a second map (Figure 15) showing the traces coloured by the current density.

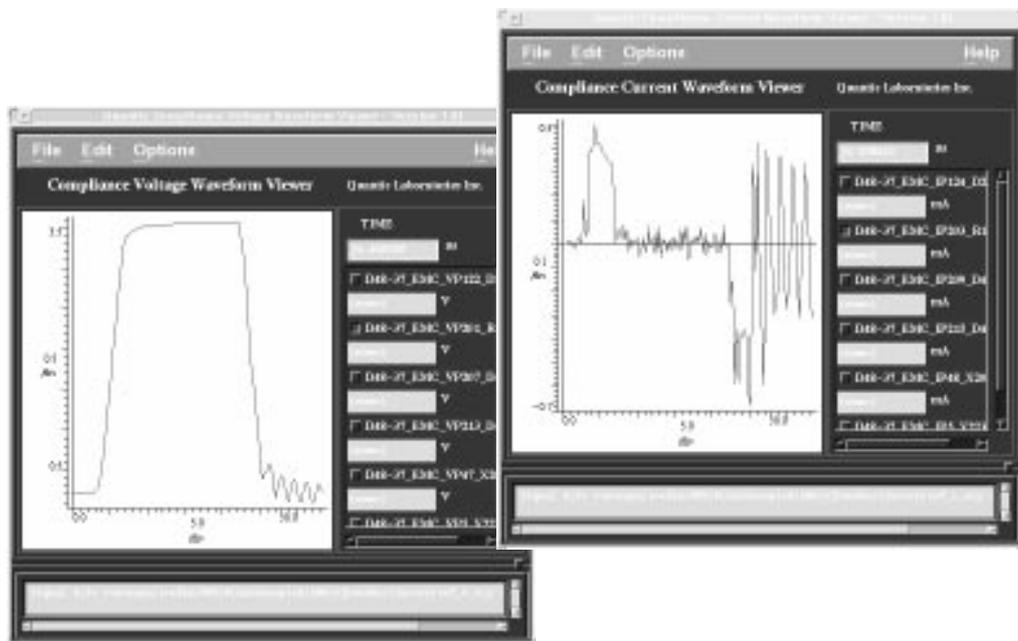


Fig. 11: Voltage (left) and current (right) waveforms

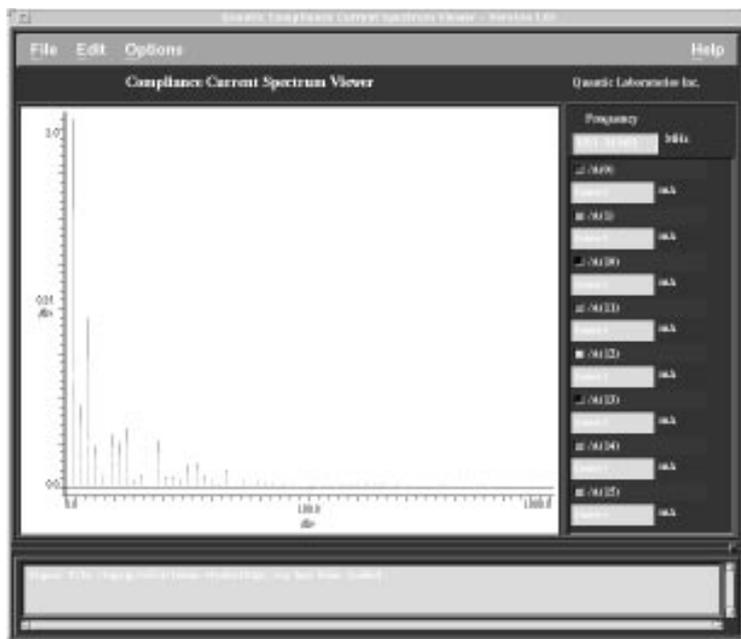


Fig. 12: Result of the Fast-Fourier-Transformation FFT

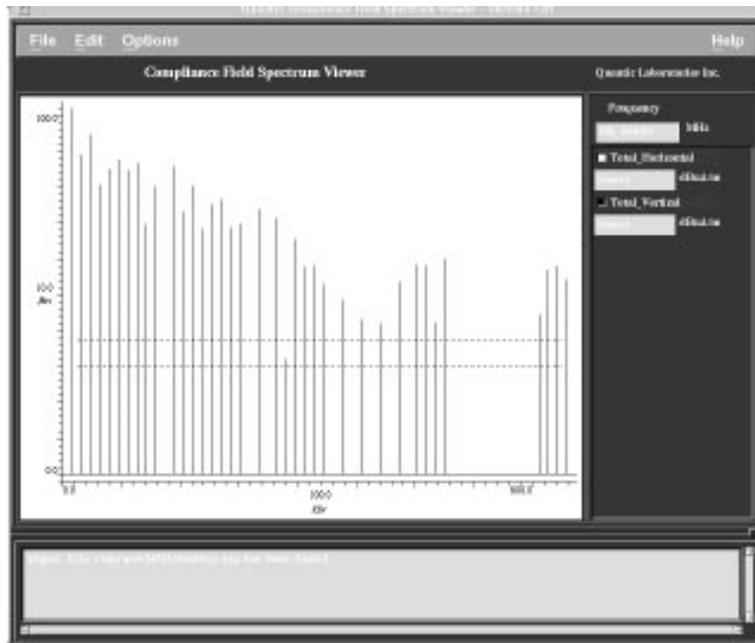


Fig. 13: Totally radiated Near Field of a PCB

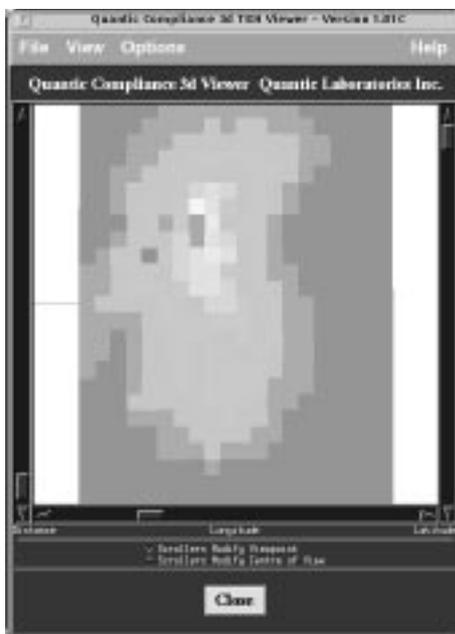


Fig. 14: Colour Map at a certain frequency. Red dots show areas with high magnetic radiation

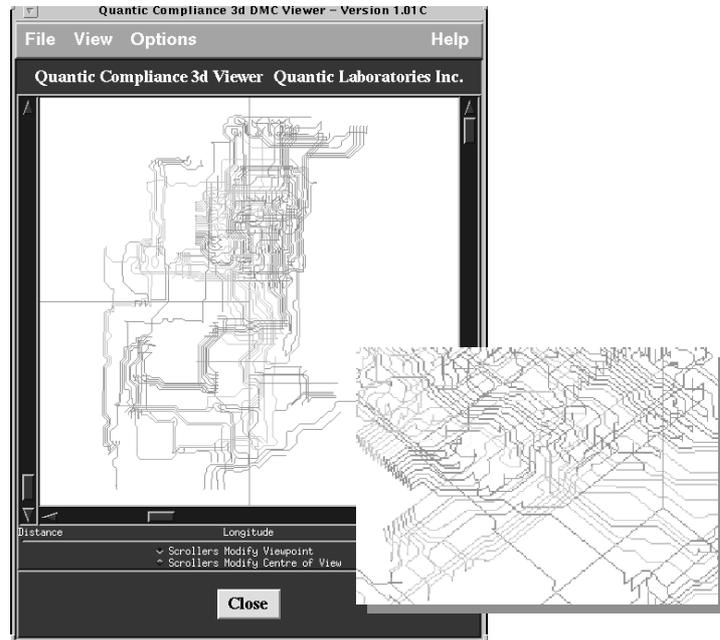


Fig. 15: Currents on the traces. Red traces show high current density

5.4 Measures

As already described in the previous, radiation directly depends on the current intensity. Thus, to reduce radiation, it is necessary to reduce current density in critical traces or nets.

To achieve this following measures can be taken:

- adding components (e.g. a serial resistor)
- rerouting the trace / net
- changing the layer
- changing the placement of components on board.

Since the simulation takes place after or — what would be much better — during the layout phase the above mentioned measures are easy to implement.

6 Usage of EMC-Simulation-tools

All results of the previous chapter were taken from an industrial product design process. Figure 13 shows a high magnetic emission at a frequency of 16 MHz. The so-called “hot spot” in the upper left of Figure 14 shows the source of the emission. In detail, the trace producing the radiation can be pointed out in Figure 15.

6.1 Measure to reduce emission

By extracting the critical network the reason of the high current density and the high radiation in the trace segment between (1) and (2) becomes evident. In Figure 16 the driver of the net is located at

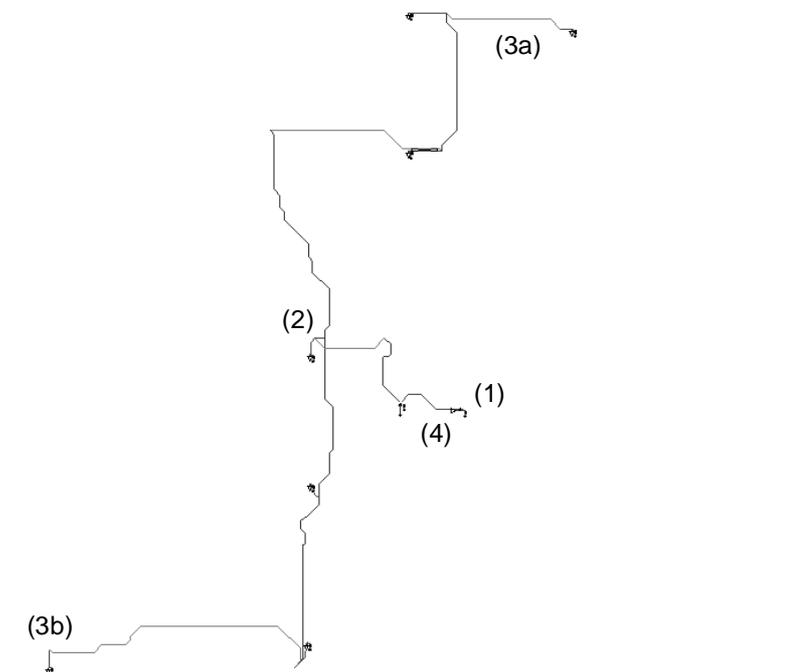


Fig. 16: Critical Network that has to be optimised

(1). In (2) the net splits to the receiver pins (3a) and (3b) what causes a high current flow and therefore a high radiation of the trace segment. A serial resistor (4) placed as near as possible to the driver pin (1) reduces the current in the trace segment. Of course, a resistor may cause signal integrity problems in the within the net. By trying out several types of resistors the best value can be found. Figure 17 shows the voltage waveforms with a resistor of 0 Ω (no resistor), 80 Ω (best solution) and 200 Ω.

6.2 Improvement

Figure 18 shows how a 80 Ω resistor reduces the radiation of the net. On the left side there is the amplitude of radiation vs. frequency of the net without any resistor. On the right side there is the amplitude of radiation vs. frequency of the net with an 80 Ω resistor. Radiation is about 10 dBμA lower. There is also an obvious decrease of radiation in a higher frequency range up to 25 dBμA.

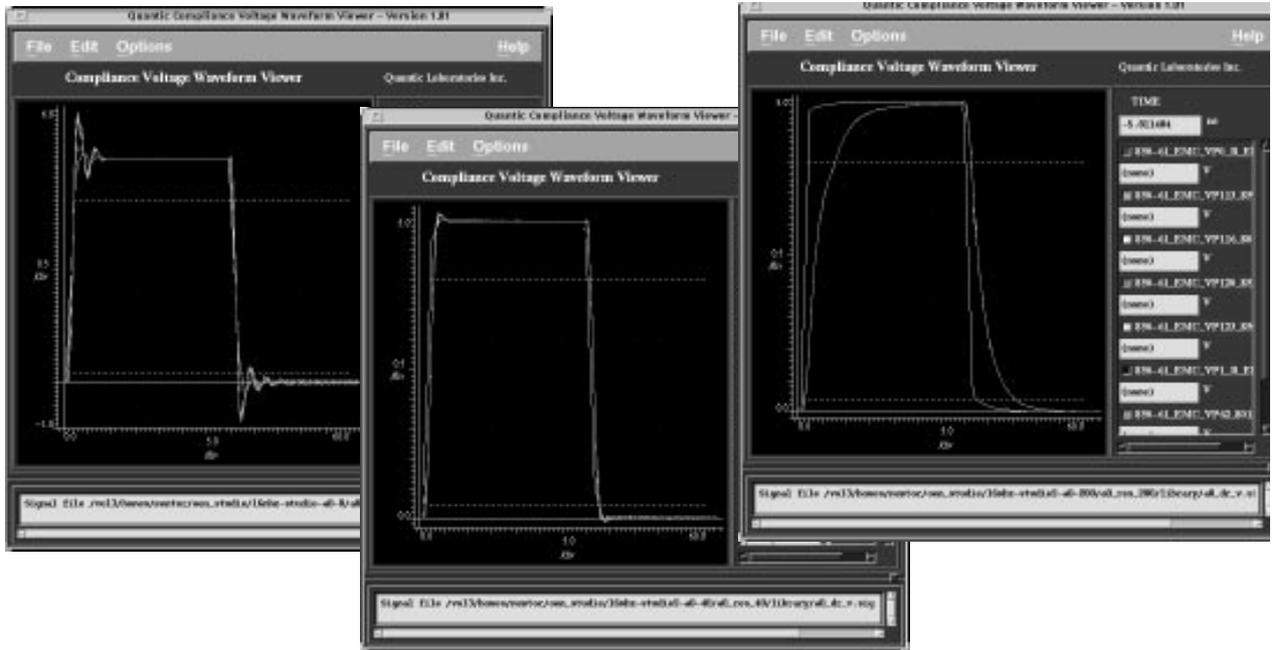


Fig. 17: Voltage Waveforms for Several Resistor Values — 0 Ω, 80 Ω, 200 Ω

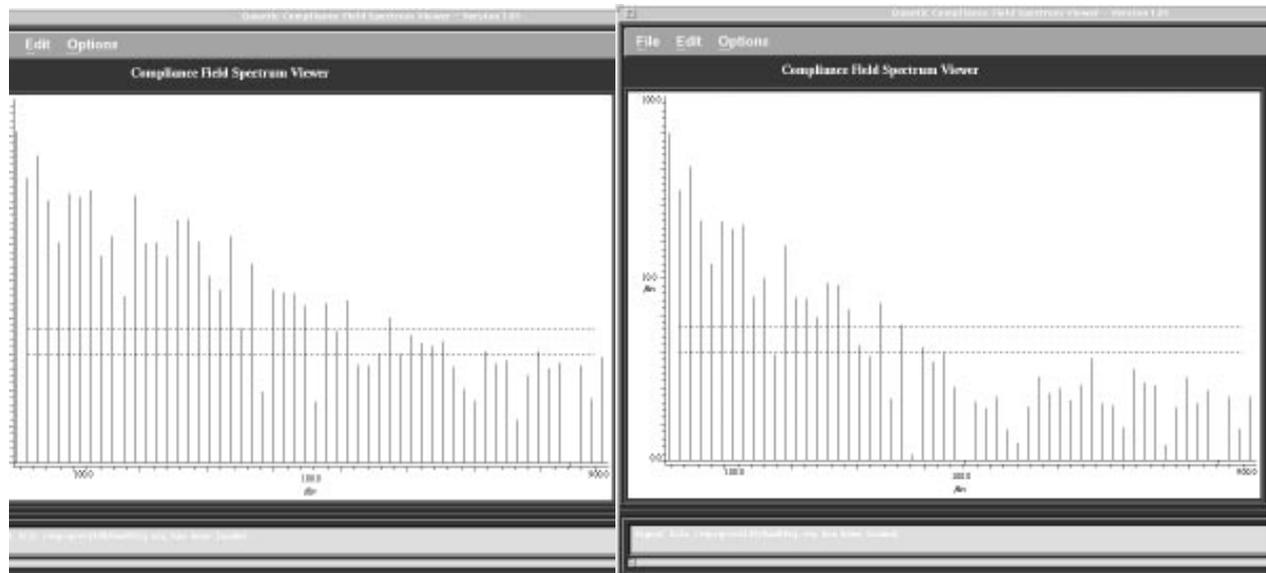


Fig. 18: Emission of the Original (left, 0 Ω) and Redesigned (right, 80 Ω) Trace

Further, a look at the far field of the trace would show a more evident difference between the original and the redesigned trace, since higher frequencies become more critical while they turn from the near to the far field.

6.3 Simulation Time

The time a simulation-run lasts depends on the system that is used. On e.g. a Sun Ultra 2 a simulation of 16 nets takes about one hour of time. Actually, it can be said that the simulation of a whole complex PCB can be done in an overnight run. The simulation of one net, during the optimising process, can be done within five minutes.

7 Conclusion

It has to be pointed out that simulation will never replace a final measurement since all laws and standards require measurement for verification. However, by integrating simulation in the development process and by creating and optimising not a physical but a virtual prototype positive effects on the whole product can be achieved.

In a short term the number of measurements decreases. Simultaneously the PCB-designers' EMC-know-how increases.

In a mid-term simulation results in a improvement of quality and image, it shortens the development process, reduces the number of measurements and required physical prototypes. It also gives a feedback if a measure taken has been successful. So the product can be designed in a most efficient way.

In a few words, simulation saves time and money. It is quite an investment but it is more an innovation.

8 Attachment

8.1 List of EMC-Software-Manufacturers

The following list names several manufacturers of EMC-Simulation software to compute emission of a electronic device. Not all of them have the full functionality described in the previous.

- HyperLynx, Inc, Redmond, USA
<http://www.hyperlynx.com>
- Incases Engineering GmbH, Paderborn, Germany
<http://www.pad.incases.com>
- Quad Design Technology, Camarillo, USA
<http://www.quaddesign.com>
- Quantic Laboratories, Inc., Winnipeg, Canada
<http://www.quantec-emc.com>
<http://www.megasolution.de>

8.2 References

- [1]** John Lavery, PhD, Electromagnetic Services Inc.
Quantic Lab., EmcScan Manual
- [2]** Tim Williams: EMC for Product Designers
Newnes: Oxford, Boston, Johannesburg, 1996
- [3]** Anton Kohling: CE-Konformitätskennzeichnung
Publicis MCD Verlag / VDE Verlag GmbH

8.3 Assistance

Siemens AG Otto-Hahn-Ring 6
ZT PP 5 81739 Munich
Germany

Peter Kaiser Phone: +49 (0)89 / 6 36 - 4 78 89
E-mail: Peter.M.Kaiser@mchp.siemens.de

Markus Klamet Phone: +49 (0)89 / 6 36 - 4 32 70
E-mail: Markus.Klamet@mchp.siemens.de