DesignCon 2001

Nonlinear Full Wave Time Domain Solutions using FDTD_SPICE for High Speed Digital and RF

Neven Orhanovic
Raj Raghuram
Norio Matsui

Applied Simulation Technology
1641 North First Street, Ste 170
San Jose, CA-95112
PH: 408-436-9070
FAX: 408-436-9078
Email: **raghu@apsimtech.com**

2001 Wireless and Optical Broadband Conference

Abstract

With high speed digital designs extending well into the GHz range, quasi-static and TEM solutions are no longer valid. Digital designers like to see time domain waveforms as provided by SPICE. However, full wave effects are often accounted for only by frequency domain electromagnetic solvers. A simulator that combines FDTD and SPICE is used in this paper as a solution to this problem. The nonlinear circuit elements are simulated in SPICE and the structural elements in FDTD. The simulators are tightly integrated and talk to each other at each time-step. The approach is used to analyze difficult problems such as meanders, ground bounce and placement of decoupling capacitors in a PCB, and dispersion in microstrips.

Authors/Speakers

Neven Orhanovic

Current Activities

He is currently with Applied Simulation Technology working on time domain full wave methods.

Background

He received his BS degree in Electrical Engineering from the University of Zagreb, Croatia and his MS and Ph. D. degrees from Oregon State University, Corvallis. For a number of years, he has worked in the area of interconnect analysis developing numerical methods and simulation software.

Raj Raghuram

Current Activities

He is currently Director of Research and Development at Applied Simulation Technology working on circuit simulation and signal integrity issues.

Background

He obtained his Bachelor's degree from IIT Madras, India, and his MS and Ph. D. degrees from Stanford University. He has worked in the areas of Circuit Simulation and Signal Integrity at IIT Kanpur, Applied Simulation Technology, and National Semiconductor. He has authored over 30 papers and a book.

Norio Matsui

Current Activities

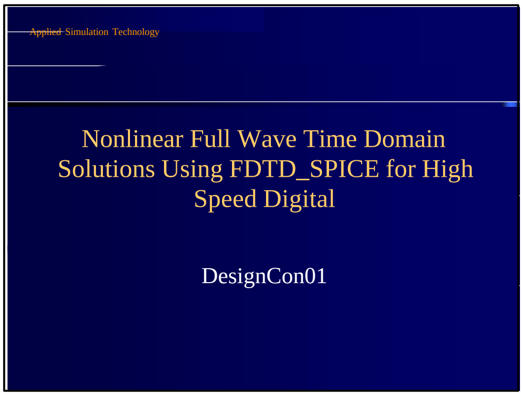
He is currently President of Applied Simulation Technology and is actively involved in Power Integrity, Signal Integrity, and EMI/EMC solutions.

Background

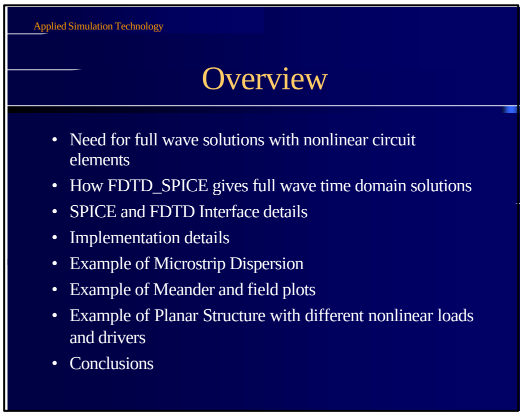
Norio Matsui holds a Ph. D. from Waseda University, Tokyo and was a researcher in NTT Labs for over 16 years. During this period, he developed noise simulation tools for Signal and Power Integrity as well as physical designs for high speed Tele-switching systems. Apart from authoring numerous papers, he also lectured at Chiba University.

Nonlinear Full Wave Time Domain Solutions Using FDTD_SPICE for High Speed Digital and RF

Slide #1



Slide #2



At today's high frequencies, a variety of new problems has cropped up in the analysis of structures such as PCBs, MCMs, packages, and ICs. This is true in both RF and high-speed digital applications, where frequencies are well over 1 GHz. For example, interconnects can no longer be analyzed by static models. Traces in PCBs are no longer simple transmission lines with TEM propagation. They exhibit significant dispersion. The loss in materials such as FR4 is very significant. So is the radiation from slots in PCBs. Skin effect is very important. Analysis of structures such as meanders and spirals is difficult even at lower frequencies, but almost impossible in the GHz range.

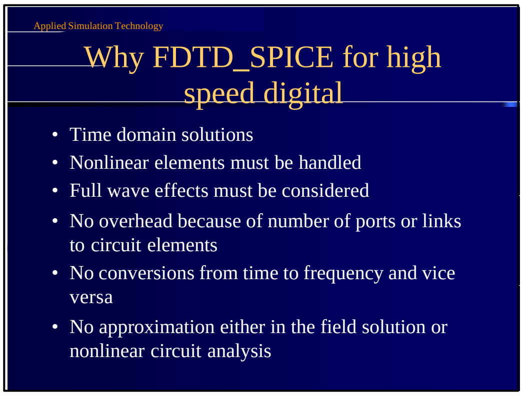
The problem is accentuated by the fact that many of these effects are best described in the frequency domain. However, nonlinear circuit elements are best analyzed in the time domain. Moreover, digital designers prefer to see waveforms in the time domain as provided by SPICE or other time domain simulators. Often digital and analog RF components are on the same board and interfere with each other. These problems are solved here by marrying a full wave time domain electromagnetic solver with a nonlinear circuit simulator. The full wave electromagnetic solver is based on FDTD (Finite Difference Time Domain) [1] and the industry standard SPICE is used as the circuit simulator. Both are based on time stepping algorithms and very convenient to interface to each other. While SPICE can handle the nonlinear circuit elements, FDTD can handle the structural elements without any approximations other than those related to meshing and time stepping.

At the end of the simulation both conventional SPICE waveforms as well 3D field distribution plots are available. The user can choose to view either depending on his background. An engineer more familiar with circuits can use the program very much as he would SPICE. At the same time, he is assured that the simulator takes care of all full wave effects.

Even where static solvers are accurate enough, FDTD is often superior to static or quasi-static solvers. Static solvers are usually either inductance or capacitance solvers and both need to be run to get a combined model. Both these solvers can take along time to run because they involve the solution of large matrices. Moreover, they produce frequency domain models, which need to be converted. FDTD automatically considers a distributed combined inductance-capacitance model valid in the time domain. As it uses and explicit solution method, there is no need to solve large matrices.

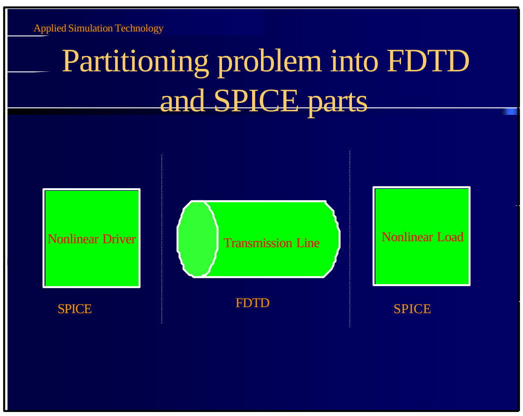
The combined program called FDTD_SPICE has been applied to a variety of problems. Application to a dispersive microstrip, a ground bounce and decoupling cap problem, a meander, and a crosstalk analysis problem are described here.

Slide #3



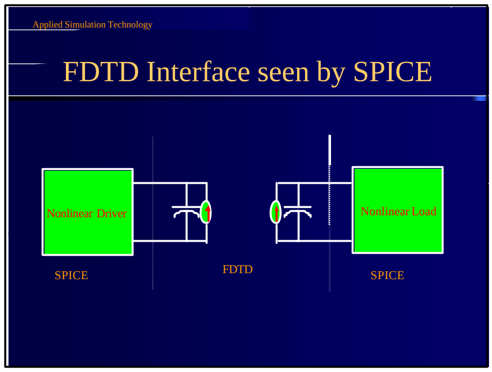
This approach has a number of advantages apart from being able to handle nonlinear elements and provide full wave time domain solutions. Standalone electromagnetic solvers, whether time or frequency domain, produce an n-port model for use in connection with a circuit simulator. A separate analysis has to be done for each port, considerably increasing simulation time when there are a number of ports. In addition, frequency domain solvers have to do a solution at a number of different frequencies to get an n-port model valid over a reasonable frequency range. As the FDTD_SPICE approach needs to do just one solution, irrespective of the number of number of ports or frequency range, simulation time is often more than an order of magnitude faster.

Slide #4



The basic idea behind the implementation is that FDTD and SPICE run simultaneously exchanging data at each time step. The circuit elements in the total problem are described in SPICE syntax and handled by SPICE while the structural elements (such as traces, reference planes, vias, etc.) are handled by FDTD. For example, a transmission line with a nonlinear driver at one end and a nonlinear load at the other may be split up between the two simulators as shown in this slide.

Slide #5

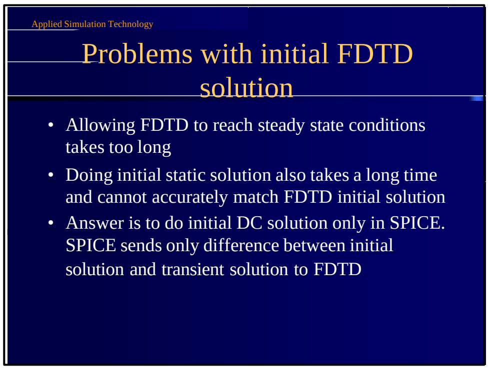


FDTD and SPICE exchange information at the ports specified. For the example in slide 4, a port can be defined between the nonlinear driver and input end of the transmission line and another between the output end of the transmission line and the nonlinear load. All the ports are two terminal ports, and the interface FDTD presents to SPICE is a current source in parallel with a capacitor [3]. The capacitor is constant while the current source is time varying. SPICE uses this circuit at its ports to analyze the total circuit at each time step. At the end of the analysis at that time step, SPICE passes the calculated port voltage to FDTD. FDTD converts this voltage into equivalent electric fields and “marches” the fields to the next time step. For the example in Slide 4, the circuit analyzed by SPICE at each time step is given.

As can be seen, the various parts of the SPICE circuit are essentially decoupled and convergence problems in SPICE are considerably minimized. The communication between SPICE and FDTD at each time step is done through pipes. This is more flexible and convenient than a function call. As a next step, if sockets are used, FDTD and SPICE can easily be made to run on separate machines or separate processors.

Nonlinear Full Wave Time Domain Solutions Using FDTD_SPICE for High Speed Digital and RF

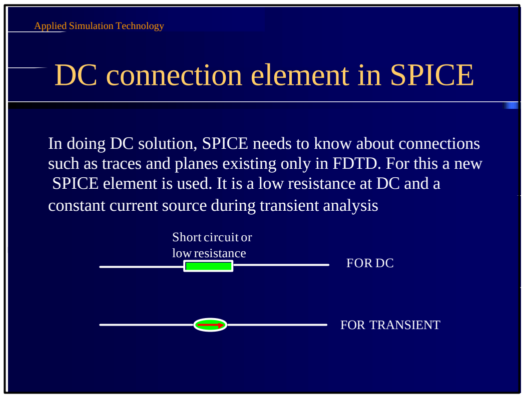
Slide #6



One aspect of the FDTD-SPICE integration not addressed in earlier work is the initial DC solution [1] [3]. Reference [2] partially addressed this issue. The DC solution in SPICE is relatively easy to do and always done prior to a transient solution. However, the initial DC solution in FDTD is difficult as FDTD is designed to calculate fields as they "march" in time. One possibility is to let the solution with the initial fields run for a long time until they stabilize. This, however, usually takes days to run and is not acceptable. Another possibility is to run a separate static solution prior to FDTD. This again may take a long time and needs another simulator. If the structure has magnetic materials and different dielectrics even the DC solution can take considerable time. The main advantage of FDTD is that it avoids the need to do expensive implicit solutions with large matrices. This is lost in having to do an initial static solution.

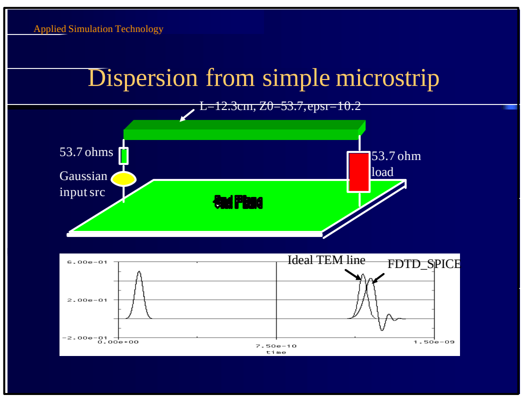
The way out of this problem is to make FDTD do only the transient part of the solution. One is usually only interested in the time varying fields and not in the initial static fields. The voltage SPICE passes to FDTD is always the difference between the transient solution and the initial DC solution [2].

Slide #7



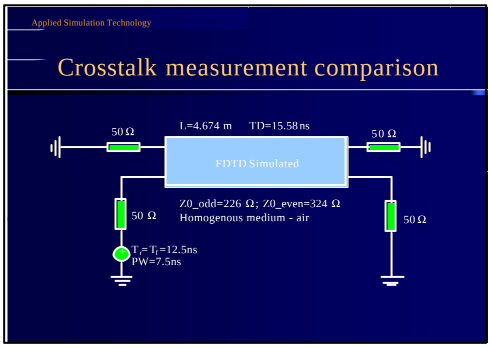
Another problem is that many parts of the circuit are linked by parts of the structure handled by FDTD. For example, in Slide #4, the driver and load in SPICE are connected through the transmission line in FDTD. For the initial DC solution in SPICE, the transmission line is just a short circuit or low resistance. This is handled by introducing a new element in SPICE. This element acts as a resistor for the DC solution. For the transient solution, the resistor is replaced by a current source of constant value equal to the DC current through the resistor. In order to apply this to a general geometry, we choose a reference node in each connected portion of the structure. From the reference node, we connect this new resistor element to each of the port terminals, which are in this connected portion. For example, in Slide #4, we would connect one new resistor element between the output of the driver and the input of the load. Another would be connected between the ground terminal of the source and that of the load.

Slide #8



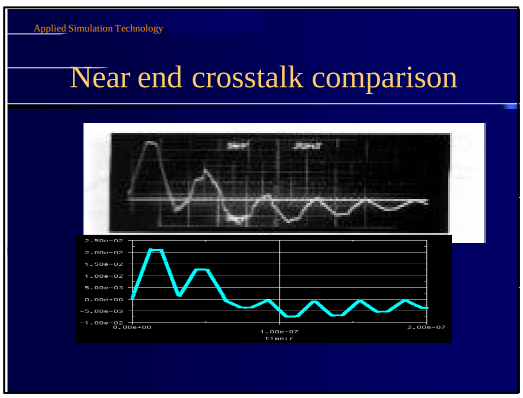
This slide shows how the behavior of even a simple microstrip cannot be predicted in the GHz range. The microstrip is matched at both ends and standard TEM analysis reproduces the input wave shape at the output with a suitable delay. However, the dominant mode has considerable dispersion and distorts the wave shape. The FDTD_SPICE solution shows the true distorted waveform [6]. This waveform shows an undershoot and is considerably different from the TEM solution. The input source is a Gaussian pulse with the width at the halfway points equal to 50 ps. The dispersion here could distort pulse shapes considerably.

Slide #9



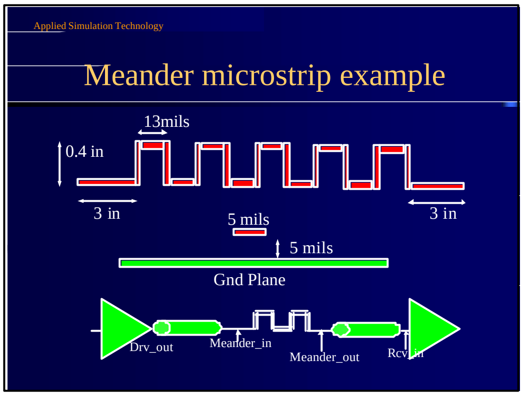
The approach here was validated against measurements for the structure shown here. It is a coupled line and the near end crosstalk from measurement was compared with that from measurement [7]

Slide #10



As this slide shows the comparison between FDTD_SPICE and measurement is excellent. This in spite of the fact that crosstalk is a second order phenomenon with much lower amplitudes.

Slide #11



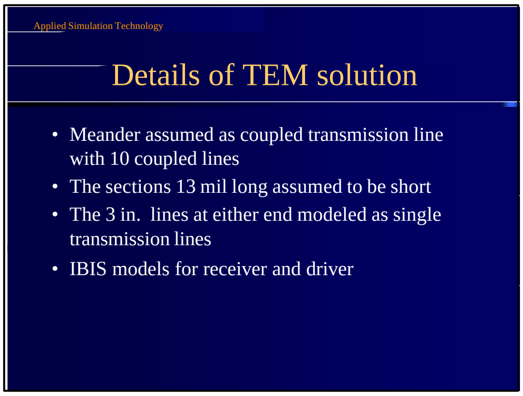
This slide shows another example analyzed by FDTD-SPICE. Here the meander alone is analyzed using FDTD. The transmission lines at the input and output ends of the meander as well as the driver and receiver are analyzed in SPICE. The driver and receiver are IBIS models of nonlinear circuits. The rise time here is relatively slow – 1ns. So the transmission lines are analyzed conveniently in SPICE.

Full wave effects are not very important here because of the slower rise time. However, the meander is still a very difficult structure to analyze, even when using static models. One approach is to model it as a coupled transmission line with 10 conductors assuming shorts at the ends. However, this ignores bends in addition to the shorts. The alternative is to model it using lumped Ls and Cs. However, one needs to run inductance and capacitance solvers to find these Ls and Cs, which can be time consuming. Moreover, the distributed nature of the structure is lost. Above all, important effects such as the high field concentrations at the edges of the meander traces are not modeled.

FDTD is able to model both the inductance and capacitance of the structure in a distributed sense properly accounting for effects such as field concentrations.

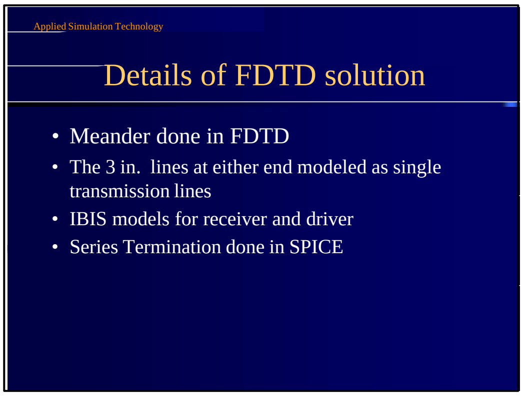
Nonlinear Full Wave Time Domain Solutions Using FDTD_SPICE for High Speed Digital and RF

Slide #12



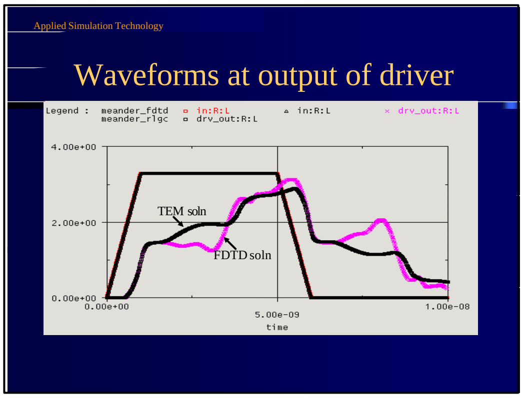
The TEM solution is just used for comparison. In this solution, the meander was modeled as a coupled transmission line with 10 conductors. The end sections of 13 mils were assumed as shorts.

Slide # 13



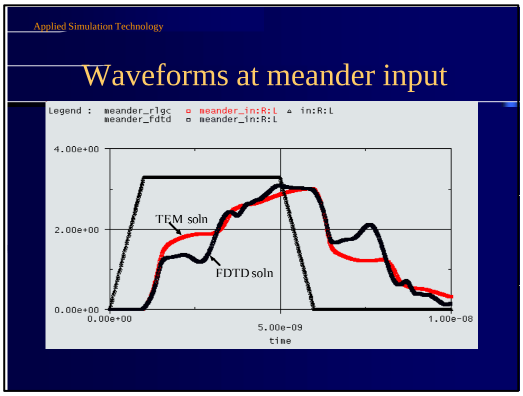
In the FDTD-SPICE solution, the meander alone is analyzed in FDTD. The other elements are analyzed in SPICE as in the TEM solution.

Slide #14



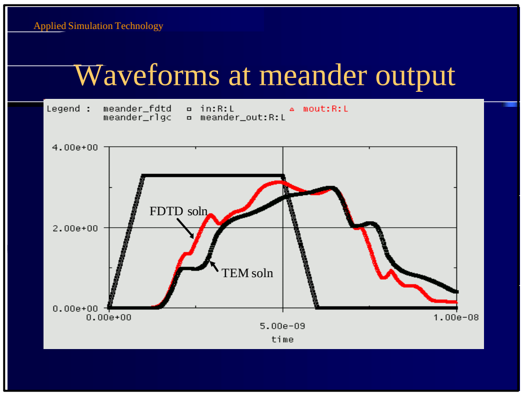
The TEM solution shows a reflection immediately from the meander, whereas the FDTD solution takes time to show the reflection. This is because the coupled transmission line is very short in length compared to the total length of meander. In addition, the impedance presented by the coupled line is different from that of the real meander.

Slide #15

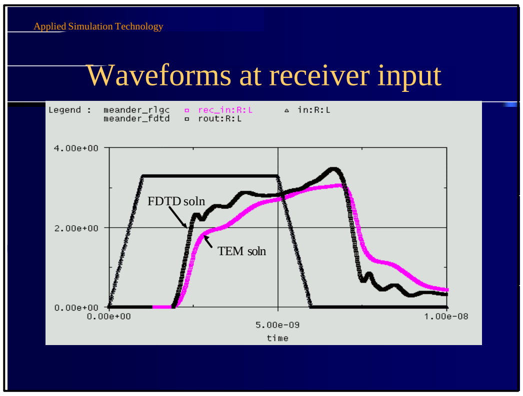


At the meander input again, the amplitude of the TEM solution is higher as it sees the meander reflection much sooner.

Slide #16

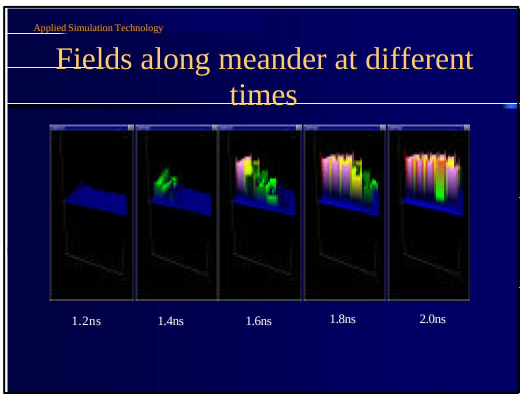


Slide #17



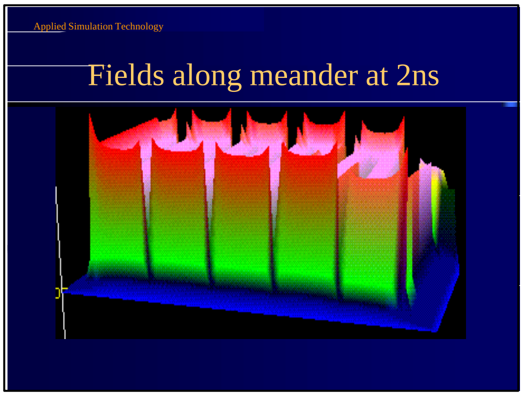
The waveforms at the meander output and receiver input agree better. Minor differences are due to different impedances of the coupled line and the real meander.

Slide #18



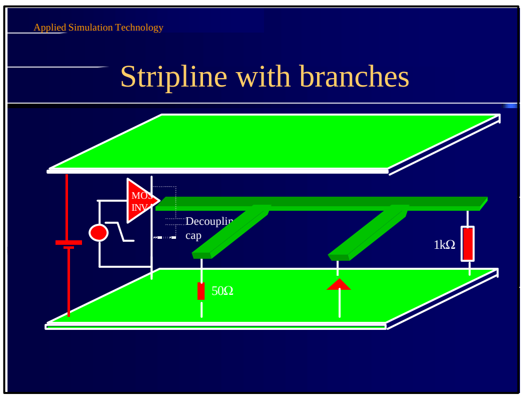
These field plots from FDTD (electric field in vertical direction) show how the fields progress along the meander. It also gives an indication of what fraction of the fields are directly going across the output instead of propagating through the meander.

Slide #19



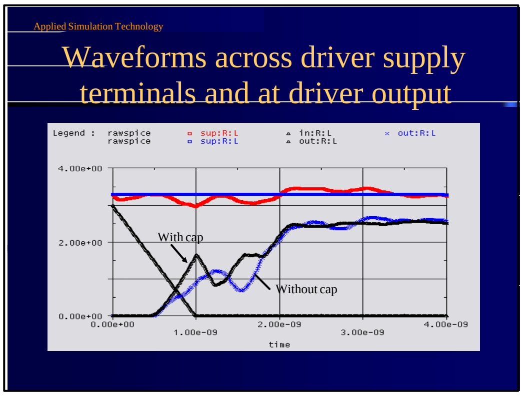
The electric field plot at 2 ns is shown magnified here. It shows the fields at the bends to be very high. It also shows the fields at the edges of the traces to be higher than at the center – something that is expected. This field distribution gives a lot of qualitative information. For example, it is clear that the high fields at the corners will lead to significant radiation. These fields are for a relatively slow rise time of 1 ns.

Slide #20



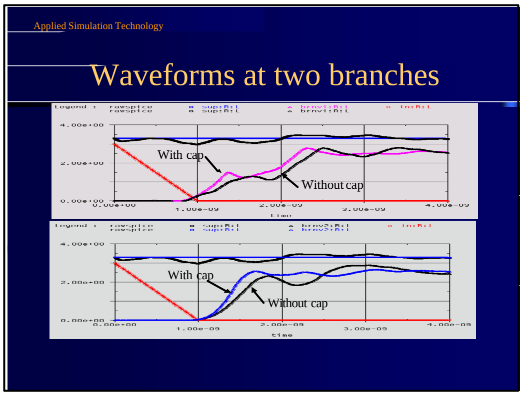
We use the example above to illustrate ground bounce and the use of a decoupling cap. In this geometry, the “ideal” power supply is located at one corner of the stripline. The driver sees a different supply voltage between the planes. We show that this voltage can bounce a lot and affect the switching characteristics of the driver. The driver is a simple CMOS inverter. This problem again is not easily solved even if full wave effects are not considered. The inductance of planes is hard to compute [4] [5], especially if they have holes, cutouts, etc. An integrated inductance-capacitance distributed model is very hard to obtain. Even if one obtains such a model, solving it in SPICE in the time domain is virtually impossible. The use of FDTD with SPICE takes care of all these effects automatically.

Slide #21



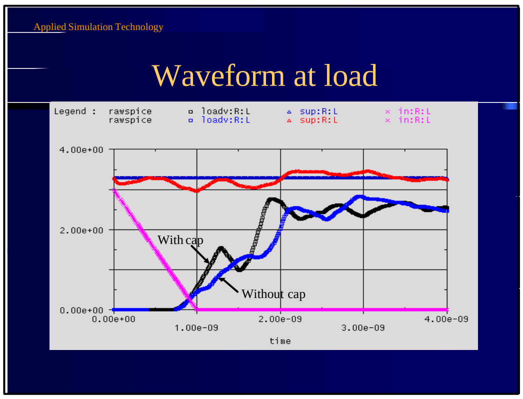
The waveform across driver supply terminals clearly shows a lot of ground bounce which goes away when a decoupling cap is introduced across the driver supply terminals. The driver output shows the effect of reflections from the stubs and takes a long time to rise to its final value. The waveform with the decoupling cap rises faster.

Slide #22



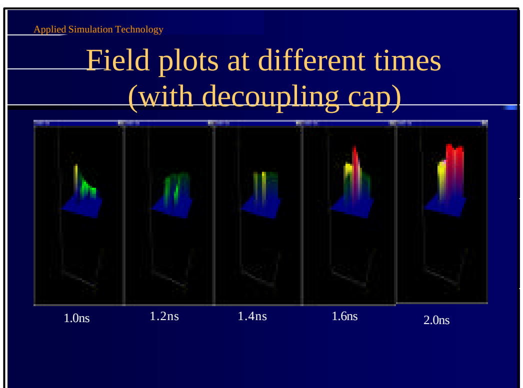
The waveforms with and without the driver-decoupling cap are shown at the loads connected to the two stubs or branches. As expected, the waveforms with the decoupling cap rise faster.

Slide #23



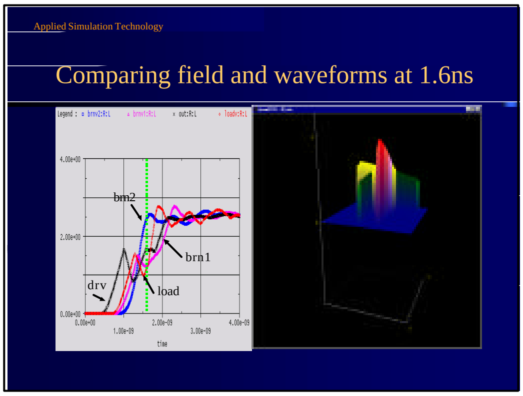
The waveform at the load end of the direct trace also shows the effect of various reflections and the influence of the decoupling cap.

Slide #24



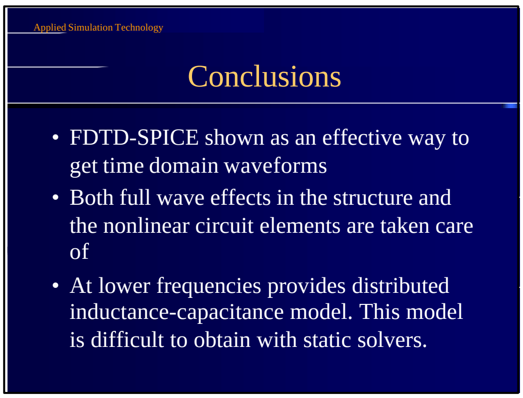
The field plots (electric field in the vertical direction) show how the waves progress along the trace and stubs as a function of time. Branch 2 and the direct trace have high impedance loads and the corresponding reflections are higher. These field plots complement the SPICE waveforms very well.

Slide #25



The field plot at 1.6 ns is compared with values of the various waveforms at 1.6 ns. The correspondence is easy to see with the voltage at the load of branch 2 having the largest value.

Slide #26



The various examples here bring out the power of FDTD-SPICE. SPICE can handle the nonlinear circuit elements and FDTD can handle the structure. Together they provide time domain waveforms and field plots without any approximations other than the normal ones related to meshing and time stepping. FDTD provides full wave solutions, which are needed in the GHz range where today's digital and RF circuits operate. Even where full wave solutions are not necessary, FDTD is considerably superior to using static solvers. Static solvers find inductance and capacitance and are often very slow because they have to use implicit techniques and need to solve large matrices. For example, when ground planes are present, inductance solvers have to finely mesh the ground plane and this makes the solution slow. More over, the distributed nature of the structure is lost in making inductance-capacitance models.

Problems such as meanders and ground bounce from power planes are solved relatively fast using FDTD-SPICE and almost impossible to solve in the time domain using any other technique.

REFERENCES

[1] A. Taflove, "Computational Electrodynamics: The FDTD Method", Artech House, Boston, 1995.

[2] A. Kobidze, A. Nishizawa, and S. Tanabe, "Ground Bouncing in PCB with Integrated Circuits", 2000 IEEE International Symposium on EMC, Washington D. C. , pp 349-352, 2000.

[3] V. A. Thomas, M. E. Jones, M. J. Piket-May, A. Taflove, and E. Harrigan, " The Use of SPICE Lumped Circuits as Sub-grid Models for FDTD Analysis", IEEE Microwave and Guided Letters, pp 141-143, vol. 4. No. 5, 1996.

[4] R. Raghuram and Norio Matsui, "EMC Analysis of Planar Structures Using Static Solvers", 1998 IEEE International Symposium on EMC, Denver, pp 561-566, 1998.

[5] R. Raghuram, Z. Yang, D. Divekar, and P. Wang, "Inductance Computation of Multiple Arbitrarily Shaped Planes", Proc. IEEE 3rd Topical Meeting on Electrical Performance of Electronic Packaging, pp 215-218, Monterey, 1994.

[6] J. I. Alonso, J. Borja, and F. Perez, "An Universal Model for Lossy and Dispersive Transmission Lines for Time Domain CAD of Circuits", IEEE MTT-S Digest, pp 991-994, 1991.

[7] C. R. Paul, Introduction to Electromagnetic Compatibility, p. 557, John Wiley, 1992.