

# EFFICIENT PARAMETER EXTRACTION OF HIGH-SPEED IC-INTERCONNECTS BASED ON 3D FIELD SIMULATION USING FIT

T. Wittig\*, T. Weiland\*, F. Hirtenfelder\*\*, W. Eurskens\*\*\*

\*Darmstadt University of Technology, Germany; \*\*Computer Simulation Technology GmbH, Darmstadt, Germany;

\*\*\*Infineon Technologies AG, Munich, Germany

(e-mail: wittig@temf.tu-darmstadt.de)

**Abstract:** With increasing frequency, parasitic effects in interconnecting structures of integrated circuits have a considerable impact on the device's performance. An important approach to analyze these effects already in the early design stage is the generation of equivalent circuit models, that can be included in the general circuit simulation process. This paper describes a parameter extraction based on a 3D time domain field simulation using the Finite Integration Technique (FIT). From the field solution, broadband S-parameters are calculated and used for the computation of SPICE equivalent circuit parameters. The advantages of this method are extensively demonstrated by means of a 24 pin IC package.

## 1. Introduction

Continuously increasing frequencies in all areas of microelectronics enlarge the parasitic effects of interconnect networks and limit the performance of integrated circuits and systems [1]. For instance, the serial inductances as well as mutual capacitances and inductances of bond wires and pins have serious influence. In particular, they change the signals shape and lead to crosstalk problems.

Clearly, these effects are not in the scope of SPICE [2] simulations of the ICs logical performance. For complex shaped structures like packages and interconnects, the only reliable approach to predict parasitic effects is a full 3D field simulation. The Finite Integration Technique (FIT) [3,4] has been proven in many applications to be an accurate and efficient simulation tool, even for very large field problems up to spatial dimensions of many wavelengths.

In a time domain calculation, the pulse shape of the digital signal can be used directly as excitation for the field simulation. The resulting output time signals might again be taken as input for circuit simulations of the IC network instead of the ideal digital pulse.

In a different approach, a single computation in the time domain offers a convenient and feasible way to calculate the structure's frequency characteristics over a broad band of frequencies. The calculated input and output signals are used to obtain the S-parameters of the interconnect. The S-parameters are the basis for the parameters extraction of an equivalent circuit.

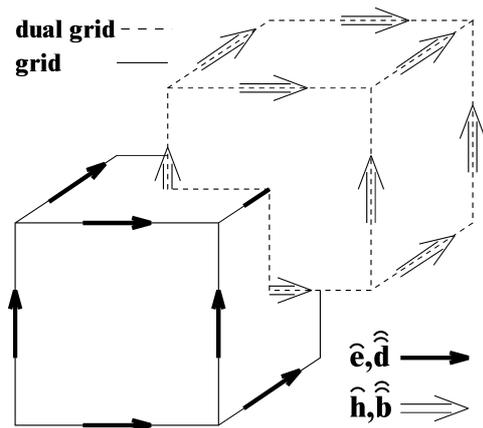
The complete behavior of the interconnect structure in a certain frequency range is describable by an equivalent circuit model, which can easily be considered in the general SPICE simulation of the IC's logical performance in an early design stage. This vitally supports the permanent demand for shortening the design cycles and minimizes expensive prototyping.

## 2. Numerical methods

### 2.1 Finite Integration Technique (FIT)

FIT generates exact algebraic analogues to Maxwell's equations, that guarantee physical properties of computed fields and lead to a unique solution.

Maxwell's equations and the related material equations are transformed from the continuous domain into a discrete space by allocating electric voltages on the edges of a grid  $G$  and magnetic voltages on the edges of a dual grid  $\tilde{G}$ . The allocation of the voltage and flux components on the grid can be seen in Fig. 1.



**Figure 1:** Allocation of the voltage and flux components in the mesh.

The discrete equivalent of Maxwell's equations, the so-called Maxwell's Grid Equations are shown in Eqs. (1)-(4). This description is still an exact representation and does not contain any approximation errors.

$$\mathbf{C}\hat{\mathbf{e}} = -\frac{d}{dt}\hat{\mathbf{b}} \quad \tilde{\mathbf{C}}\hat{\mathbf{h}} = \frac{d}{dt}\hat{\mathbf{d}} + \hat{\mathbf{j}} \quad (1,2)$$

$$\hat{\mathbf{S}}\hat{\mathbf{b}} = \mathbf{0} \quad \tilde{\mathbf{S}}\hat{\mathbf{d}} = \mathbf{q} \quad (3,4)$$

In these equations  $\hat{\mathbf{e}}$  and  $\hat{\mathbf{h}}$  denote the electric voltages between grid points and the magnetic voltages between dual grid points, respectively. The symbols  $\hat{\mathbf{d}}$ ,  $\hat{\mathbf{b}}$  and  $\hat{\mathbf{j}}$  are fluxes over grid or dual grid faces. Due to the consistent transformation analytical properties of the fields are maintained resulting in corresponding discrete topological operators on the staggered grid duplet. The topology matrices  $\mathbf{C}$ ,  $\tilde{\mathbf{C}}$ ,  $\mathbf{S}$  and  $\tilde{\mathbf{S}}$  correspond to the curl- and the div-operators. The tilde indicates that the operator belongs to the dual grid. The discrete analogue of the coupling between voltages and fluxes is represented by the material matrices  $\mathbf{M}_\epsilon$ ,  $\mathbf{M}_{\mu^{-1}}$  and  $\mathbf{M}_\kappa$ .

$$\hat{\mathbf{d}} = \mathbf{M}_\epsilon \hat{\mathbf{e}} \quad (5)$$

$$\hat{\mathbf{h}} = \mathbf{M}_{\mu^{-1}} \hat{\mathbf{b}} \quad (6)$$

$$\hat{\mathbf{j}} = \mathbf{M}_\kappa \hat{\mathbf{e}} + \hat{\mathbf{j}}_A \quad (7)$$

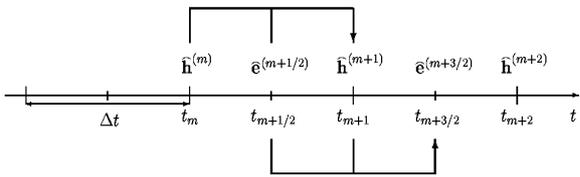
These matrices have diagonal form and contain the unavoidable approximations of any numerical procedure. The boundary of the calculation domain is assumed to be ideal electric, ideal magnetic or ‘open’. In the case of an open boundary, a perfectly matched layer (PML) [5] is additionally wrapped around the structure to absorb incident waves at the boundary.

## 2.2 Time Domain Simulation

The FIT can be applied to both the frequency and the time domain, where especially the latter is very efficient for many rf-applications. The discretization of the time derivative can be formulated as an explicit algorithm, the so-called leap-frog algorithm. The calculation of each further time step only requires one matrix-vector multiplication. Additionally, the method is very memory efficient. For the lossless case, the scheme is described in Eqs. (8) and (9) and supplementary illustrated in Fig 2. FIT in time domain is therewith a generalization of the FDTD [6] method.

$$\hat{\mathbf{h}}^{(m+1)} = \hat{\mathbf{h}}^{(m)} - \Delta t \mathbf{M}_{\mu^{-1}} \mathbf{C} \hat{\mathbf{e}}^{(m+1/2)} \quad (8)$$

$$\hat{\mathbf{e}}^{(m+3/2)} = \hat{\mathbf{e}}^{(m+1/2)} + \Delta t \mathbf{M}_\epsilon^{-1} \left( \tilde{\mathbf{C}} \hat{\mathbf{h}}^{(m+1)} - \hat{\mathbf{j}}^{(m+1)} \right) \quad (9)$$



**Figure 2:** The leap-frog scheme.

The maximum stable time step inside an equidistant grid is restricted by the Courant criterion to

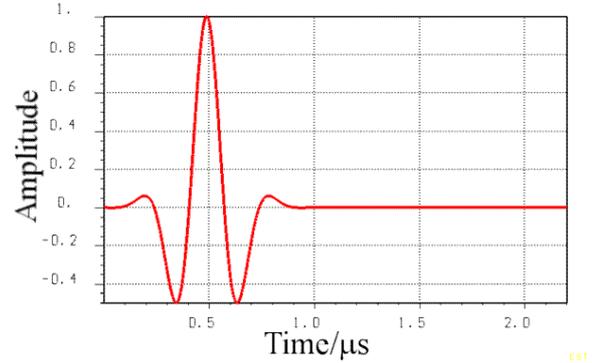
$$\Delta t \leq \frac{1}{c \sqrt{\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} + \frac{1}{\Delta z^2}}} \quad (10)$$

The excitation signal for the time domain calculation can be fed in the structure on any desired path of edges in the normal grid. This path is referred to as discrete port. More complex waveguide structures can be excited with a so-called waveguide port.

## 2.3 S-Parameter Calculation

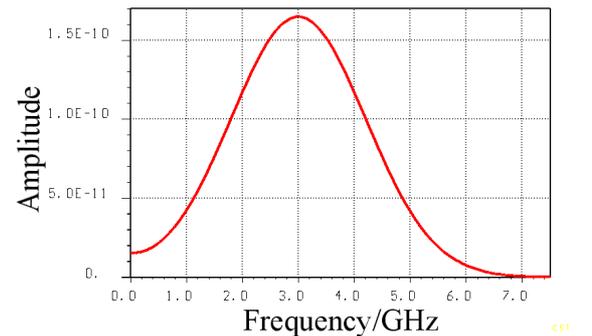
The FIT enables the computation of field components at all grid edges. However, in many practical cases of microwave device design the port behavior, usually expressed as S-parameters, is of sole importance. For the computation of S-parameters one or several discrete or waveguide ports are defined in the computational domain or at its boundary, respectively. In the case of a waveguide port, first a 2D mode pattern is calculated in the port plane, which later is used for excitation. The discrete port represents a discrete impedance combined with a feeding current source. To obtain a full S-parameter matrix, all ports are excited successively, while the output signals at all further ports are recorded. For all subsequent discussions only the case of discrete ports will be considered. At the excited port the input and output signals a and b are calculated from the current, voltage and wave impedance of the port, which are known from the electric and magnetic field components. At all other ports only the time dependent output signals, resulting from the field calculation, are taken into account.

To calculate the S-parameters in a wide range of frequencies a broadband input signal is chosen. In Fig. 3 a Gauss modulated sine function is shown as a typical example.



**Figure 3:** Time dependent broadband input signal.

All input and output signals resulting from the time domain simulation are transformed into the frequency domain by application of a Discrete Fourier Transform (DFT) algorithm. Fig. 4 shows the frequency representation of the time signal already shown in Fig. 3 above.



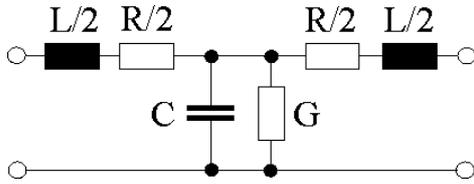
**Figure 4:** FFT of Gauss shaped input signal.

Finally, the S-parameters are obtained as the ratio of these frequency domain representations of the direct incident and inverse wave components.

## 2.4 Circuit Parameter Extraction

Many conventional circuit parameter extraction algorithms are based on separate static simulations of the electric and the magnetic field of a device to receive equivalent values for the inductivity  $L$  and capacity  $C$ . Obviously, this approach does not contain any dynamic behavior of the structure. Therefore, its validity is restricted to relatively low frequencies.

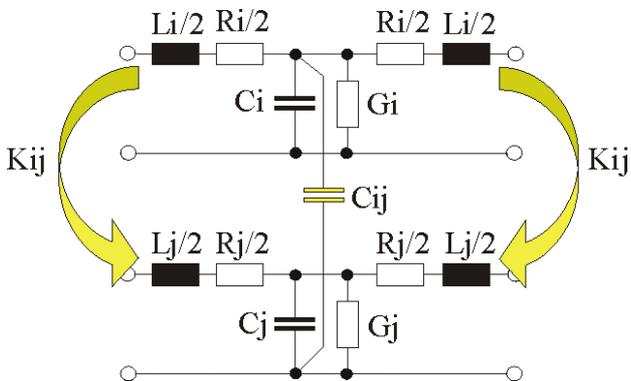
In this paper we use an equivalent circuit model based on a fundamental T-structured transmission line model as shown in Fig. 5.



**Figure 5:** Fundamental transmission line model.

The left and right ends of this network coincide with two discrete ports in the 3D structure, that have to be defined in advance. The  $S_{11}$ - and  $S_{21}$ -parameters of the respective ports, obtained by the time domain simulation, are used to compute the values of  $L$ ,  $C$ ,  $R$  and  $G$  by solution of linear systems. This way, the best fit between zero and a predefined extraction frequency is achieved both for the phase and the absolute value. For this frequency range the model approximates the full dynamic performance of the interconnect.

Additionally, if several transmission lines are considered in the parameter extraction, crosstalk between the line models has to be taken into account. The network is therefore extended by a coupling capacity between the individual transmission lines and a mutual coupling between the inductances as shown in Fig. 6.



**Figure 6:** Expanded transmission line model considering a coupling capacity and inductive coupling.

Due to the first order character of the network model, reliable parameter fits are obtained up to a spatial extension of about  $\lambda/10$ . For many practical applications this range is sufficient.

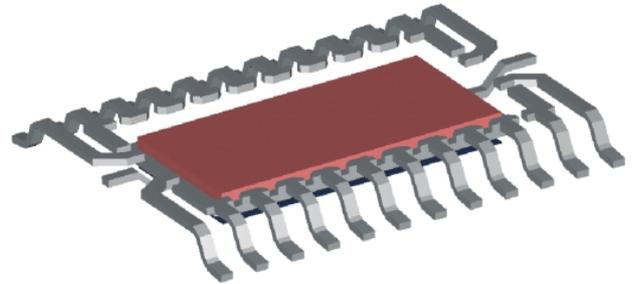
In case of larger structures, higher order transmission

line models can be used for approximation. Unfortunately the calculation of ‘true’ higher order models increases the optimization effort drastically. But already the separation of the first order model into a number of serial similar transmission line models improves the approximation quality in both frequency and time domain noticeably.

## 3. Time Domain Field Simulations

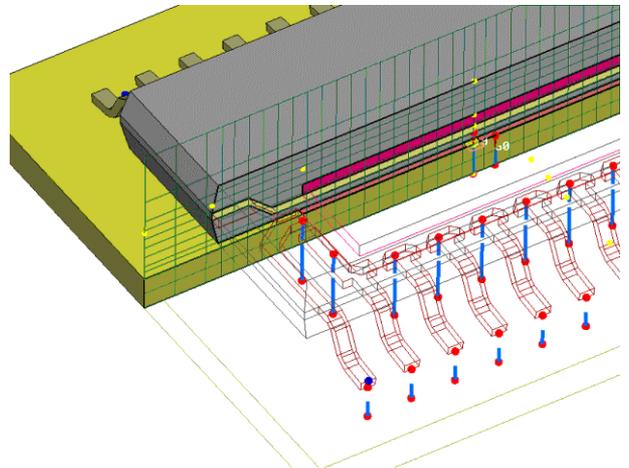
### 3.1 The IC Model

The field simulation and parameter extraction process as described so far are validated by an extensive investigation of a 24 pin high-speed IC package. The P-TSSOP24 IC package used by Infineon Technologies AG was chosen as an representative example. Its leadframe is shown in Fig. 7. The model consists of the leadframe, the silicon chip and the dielectric matter of the covering molding material.



**Figure 7:** Leadframe of high speed IC package with attached piece of silicon.

The operating frequency range of interest is 1-2 GHz according to applications of mobile communication units (GSM 900 and GSM 1800 MHz). All following computations are performed with the simulation tool CST MicroWave Studio 3.0 (MWS) [7], which is based on FIT. The structure shown in Fig. 7 is discretized by use of a Cartesian mesh with varying density to achieve the best possible spatial approximation (Fig. 8).



**Figure 8:** Position of the discrete ports at the beginning and end of each pin, as well as the mesh used for all computations.

Hereby the largest grid cell size is chosen to be smaller than 1/15 of the shortest excited wavelength inside the material to minimize the dispersion error. The whole model consists of 43x46x12 cells which results to a total number of about 24000 mesh cells.

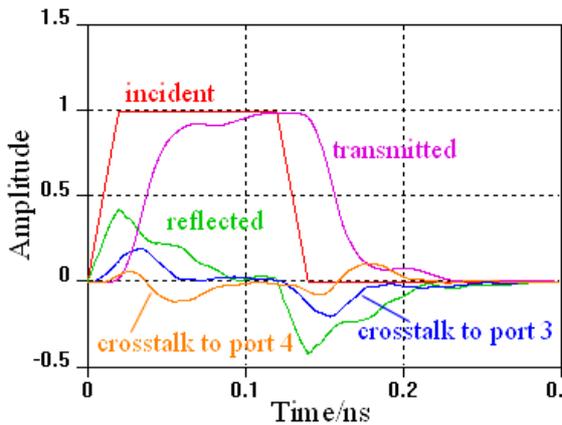
Every pin is provided with one discrete port at each end. As a result the structure has 48 ports in total. The impedance of each port is set, to 50 Ohm.

The metallic parts are assumed to be perfect electric conductive (PEC), while the other materials are defined by their permittivity and conductivity.

### 3.2 Digital Simulation

Different time signals were applied to the model. In Fig. 9, the results of a simulation with a digital pulse signal are presented.

The first port is excited by a trapezoidal input signal with a hold time of 100 ps and rise and fall times of 20 ps. The total signal length amounts to 500 ps. Fig. 9 shows the effectively transmitted signal as well as the reflection. As expected, the field effects occur most distinct at the rising and falling flanks. Additionally, it is obvious that the crosstalk to the ports 3 and 4 of the adjacent pin is pronounced and its influence is not negligible.



**Figure 9:** Pulse excitation signal and resulting output time signals.

The time signals resulting from the field simulation can be used directly as input for a circuit simulation tool like SPICE. The digital simulation therefore yields the most flexible results of the structure's digital response.

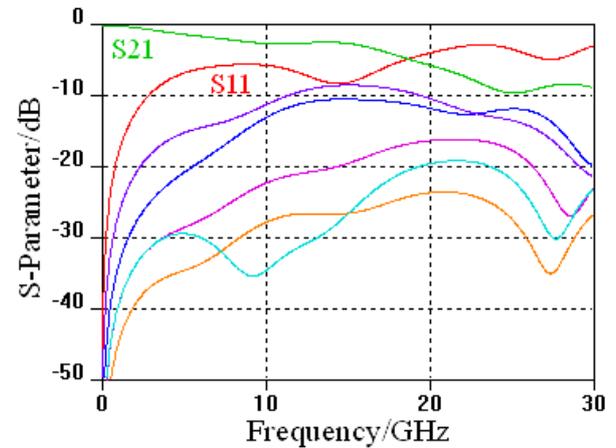
Unfortunately, this approach is restricted to relatively short pulses, due to the fact that the maximal time step is fully predefined by the discretization according to Eqn (10). In this example the resulting maximal stable time step is 0.25 ps, which requires the calculation of 2000 time steps. The total computation time on a 350 MHz PC took 350 s. For low clock speed systems the input signal is much longer, but the time step cannot be changed in order to obtain stable results. Therefore, the computation time increases dramatically.

### 3.3 Broadband Simulation

To overcome this disadvantage for low speed or analog systems the structure was excited by a broadband pulse covering also low frequencies as shown in Fig. 3. The S-

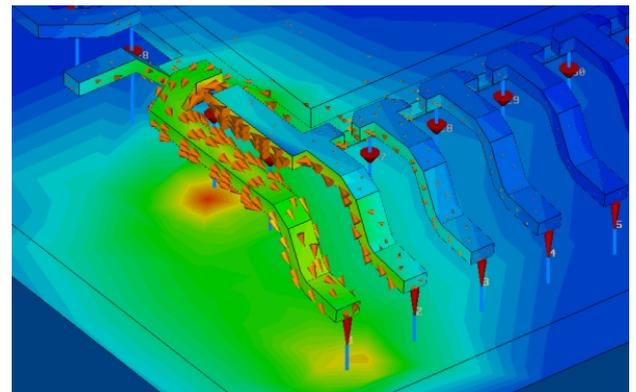
parameters are calculated from the simulated time signals as a more general representation of the structure's behavior.

All 24 outermost ports were excited successively to receive a full S-parameter matrix. Each time signal contained 781 time steps and required 82 s for computation. With an additional calculation time of 120 s for the mesh generation, the overall computation time was of about 35 min. Fig. 10 shows some important S-parameters in a range of 0 to 30 GHz with port 1 being excited. The both upper graphs display the transmission and reflection of port 1 in dB, the other graphs represent the crosstalk to neighboring pins.



**Figure 10:** Computed S-parameter with excitation of port 1.

For visualization purposes, the fields can be monitored at discrete frequencies. Fig. 11 gives the current density which is calculated by  $H \times n$  at a frequency of 5 GHz as an overlay of contour- and arrowplot.



**Figure 11:** Contour and arrowplot of the current density at 5 GHz.

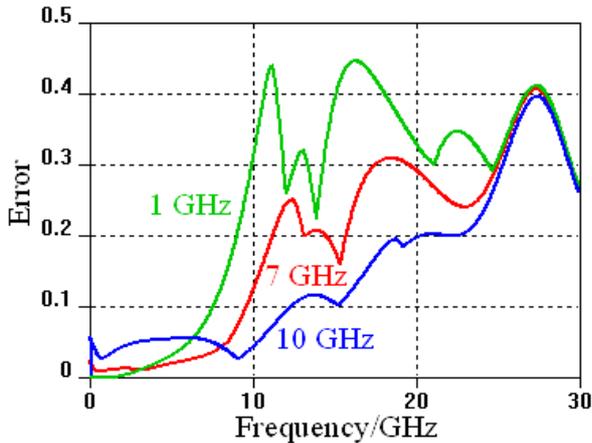
It is obvious that the current flows mainly in the excited first port but also the crosstalk current induced in the neighboring pins can be clearly noticed.

## 4. Equivalent Circuit Extraction

Since the S-parameters cannot usually be used as input for further circuit simulations of the IC, an equivalent circuit model is extracted and investigated in both frequency and time domain.

#### 4.1 Frequency Domain Evaluation

The main parameter controlling the quality of the equivalent circuit is the extraction frequency. Fig. 12 shows the maximal error between the S-parameters resulting from the field solution and the ones calculated from the circuit data for three models derived at different extraction frequencies.



**Figure 12:** Maximal error between field and circuit S-parameters for different extraction frequencies.

All error curves in Fig. 12 show the main characteristic of the transmission line model. The error always has its minimum at low frequencies (roughly between DC and the extraction frequency) which is the most important bandwidth for many simulation tasks. The model is therefore valid up to a certain frequency corresponding to spatial dimension of the pin of approximately 1/10 of the wavelength.

Obviously the lowest extraction frequency of 1 GHz yields an equivalent circuit which is very accurate in the extraction point and below. The maximal error in the range of DC up to 2 GHz is far below 1 %. However, above 2 GHz the error of the model is increasing strongly.

If the extraction frequency is selected to be at higher values, the match between DC and the extraction point is less precise, but there is a good agreement up to higher frequencies. The case of 10 GHz extraction frequency displays an average error of 5 % between 0 and 10 GHz, while up to 15 GHz the maximal error is around 10 %.

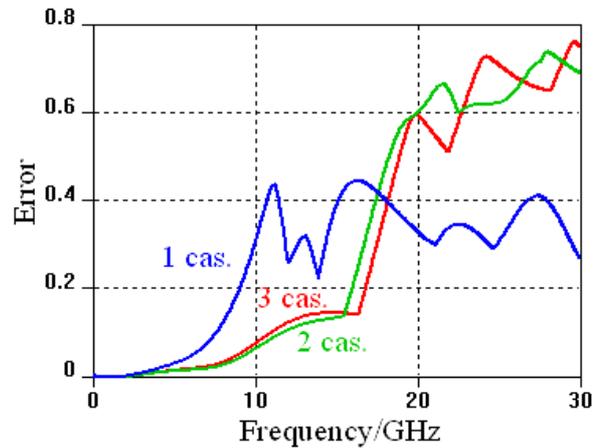
For higher extraction frequencies the assumptions of a transmission line are not valid anymore, the resulting models become unusable.

#### 4.2 Cascading

As mentioned above, a further improvement of the network model is obtained by separation of the equivalent circuit into similar subnetworks that are connected in cascade. Fig. 13 shows the maximal error for different numbers of cascades for the 1 GHz model already used in Fig. 12.

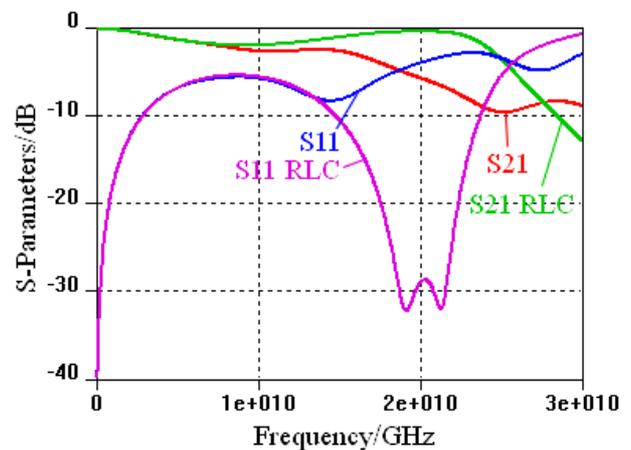
The resulting equivalent circuit preserves the very low error for frequencies below 2 GHz and at the same time it extends the range of validity (error < 10 %) from

about 7 GHz to 12 GHz. The error in the range above 20 GHz is further increased compared to the model without cascading, but this region is not covered by the transmission line model anyway. The circuit models of 2 and more cascades do not display significant differences in the error curves in this case.



**Figure 13:** Maximal error between field and circuit S-parameters for an extraction frequency of 1 GHz and different numbers of cascades.

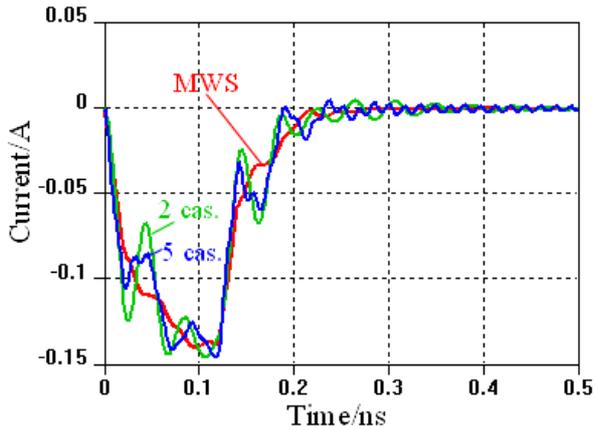
In Fig. 14 the original S-parameters gained from the field simulation are compared with the ones resulting from a SPICE simulation of the extracted model. The related curves show a very good agreement up to about 15 GHz.



**Figure 14:** Comparison of field and circuit S-parameters for an extraction frequency of 1 GHz, separated into two cascades.

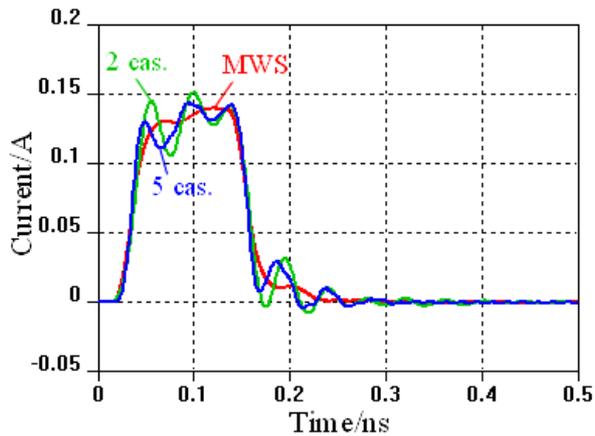
#### 4.3 Time Domain Evaluation

For most simulation tasks in digital systems, the time domain behavior of the equivalent circuit is even more important than the frequency response. The model derived at a frequency of 1 GHz is therefore investigated with different numbers of cascades in time domain. The circuit was excited at port 1 by a voltage pulse source as in Fig. 9. The currents at the ports 1 to 4 are compared to the currents monitored in the discrete ports of the MWS 3D field simulation.

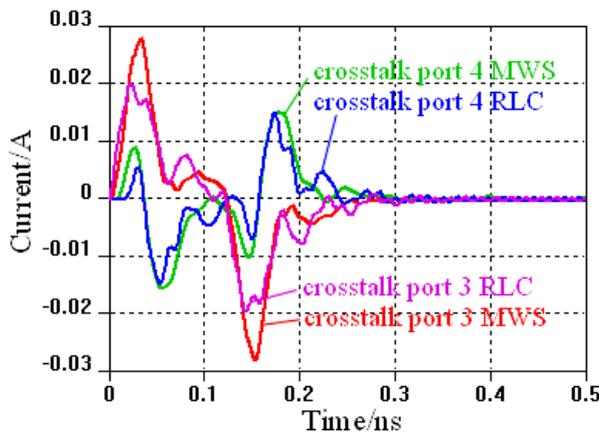


**Figure 15:** Comparison of the currents in the exciting voltage source at port 1.

In Fig. 15 the current of the exciting voltage source is compared. The graphs exhibit generally good agreement, whereby the 2 cascade model displays more overshooting than the circuit with 5 cascades.



**Figure 16:** Comparison of the currents at the end of the pin (port 2).



**Figure 17:** Comparison of the crosstalk currents in the adjacent pin with the 5 cascades model.

Fig. 16 shows the transmitted current at port 2 which is the end of the excited pin. Again, the extracted model is a very good approximation of the original signal. Especially the time delay is modeled very accurately.

Finally, the crosstalk currents in the adjacent pin are compared in Fig. 17 only for the 5 cascades model. The simulation data is also well fitted with only small errors.

## 5. Conclusions

In this paper, the inclusion of a full 3D field simulation into a conventional circuit simulation tool like SPICE was investigated by different approaches. The Finite Integration Theory in time domain was shown to be a powerful method for fast and accurate field solutions of the simulated IC interconnect. For high clock speeds, transient simulations using true digital waveforms as excitation signal represent a very flexible and fast approach, which is, however, inefficient for analog or slower digital systems. In the latter case, the extraction of a transmission line based circuit model yields a very good approximation of the parasitic behavior of the package. These models are valid, with a reasonable error, up to frequencies for which the structure's dimensions are below 1/10 of the corresponding wavelength. The generated circuit model was shown to provide a very satisfactory approximation of the electromagnetic behavior not only in frequency but also in time domain.

## 6. References

- [1] E. Chiprout, "Interconnect and Substrate Modeling and Analysis: An Overview", IEEE Journal of Solid-State Circuits, Vol. 33, No. 9, 1988, pp. 1445-1452.
- [2] L.W. Nagel, "SPICE", a computer program to simulate semiconductor circuits", Univ. California, Berkeley, Techn. Rep. ERL-M500, 1975.
- [3] T. Weiland, "A Numerical Method for the Solution of the Eigenwave Problem of Longitudinally Homogeneous Waveguides", Electronics and Communication (AEÜ) Vol.31 (1977), p. 308.
- [4] T. Weiland, "Time Domain Electromagnetic Field Computation with Finite Difference Methods" Numerical Modeling, Electronic Networks, Devices and Fields, Vol. 9, pp. 295-319 (1996).
- [5] J.P. Berenger, "A Perfectly Matched Layer for the Absorption of Electromagnetic Waves", Journal of Computational Physics, Vol. 114, 1994, pp. 185-200.
- [6] K.S. Yee, "Numerical Solution of Initial Boundary Value Problems Involving Maxwell's Equations in Isotropic Media", IEEE, Antennas and Propagation 14, 1966, pp.302-307.
- [7] "CST MicroWave Studio User Manual, Version 2.0", CST GmbH, Darmstadt, Germany.