2.1.: MODELING TUTORIALS

This chapter is an introduction to the modeling of electronic components. We will cover the main aspects of modeling and discuss some important modeling terminology, and lead over to the simple question, why not just use existing library models instead of cumbersome modeling efforts.

What are Models?

Models—as implemented in simulators—are generally a set of firmly implemented model equations for which the user can access and define the parameter values externally.

For example, in the case of an ohmic resistor, the equation is:

$$v = R * i \qquad (1)$$

The parameter involved is designated R. The parameter is determined by the slope of the characteristic resistance curve.

The definition of the model equations and the method for extracting the parameters is called modeling. A model following the form of above initially only applies to DC. For frequencies above a few Megahertz, this resistance model is too simple and must be expanded. Equation 1 therefore either becomes more complex, or one lets the model stand as is, but supplements it for a subcircuit made up of lead inductances and parasitic capacitors.

Who Defines the Models?

Although many different models have been proposed over the years, some of them have proven to match, e.g., a certain transistor type of many different manufacturers. Thus, those models have become so-called industry standard models. After having been implemented in the simulator tools, they have received wide acceptance and usage. Examples are the MOS2, MOS3, BSIM3v3, the Gummel-Poon, the Statz and the Curtice models. The Gummel-Poon model became very popular just after having been implemented in Berkeley SPICE in the early 1970s. Other models are the result of industry consortiums, like the bipolar VBIC model.

Often, the modeling engineer can select from among different models for a certain type of transistor. Using SPICE, choices for MOS transistors are MOS model level2, level3 or BSIM3. Many modern simulators, yet not SPICE, allow the definition and implementation of user-defined models. Using ADS, the user can create symbolically defined devices (SDDs). Currently, standardization attempts are underway to develop a modeling language known as HDL-A (Hardware Description Language for Analog Applications) that can be recognized by all simulators.

As stated above, the model equations are permanently installed in the simulator code. The parameters of these equations are called the model parameters. Their values are adjusted so that the measured data and the simulated data fit best together.

Available Model Types

There are three different hierarchies of models for electronic design. In the order of complexity, we can distinguish between:

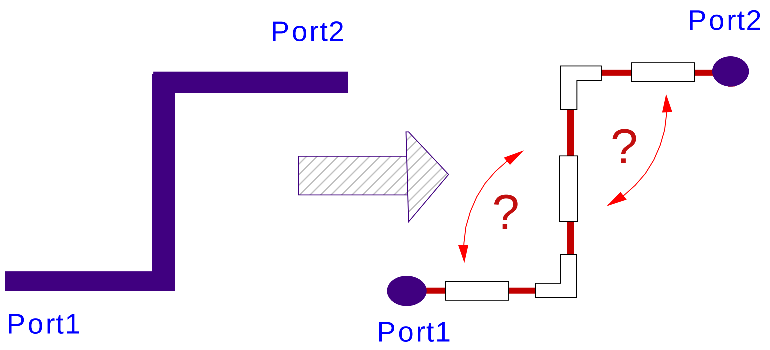
- Models for discrete linear (R, L, C) and nonlinear (transistors) components (lumped circuit elements)
- Models for specific nonlinear high-frequency effects (for example, transmission lines including skin effect and dispersion effect etc.)
- Models for complete printed circuit boards (PCBs) (solving the Maxwell equations including the mounted components)

These three model types can be found in the following types of simulators:

Application	Simulator type Example of a simulator
discrete components	Circuit Simulator Spice, Saber, Eldo, Pspice etc.
improved HF components	RF Circuit Simulator ADS, Spectre
PCBs including all links	electromagnetic Simulator (EM) Momentum

Strictly speaking, the entire assembly, including interactions between individual components, would always need to be simulated. Yet, simulation would only be possible with EM simulators, and the application of Maxwell's equations is generally regarded as overkill. For this reason, simulators with simplified models have been developed. The first of these was SPICE. The models included in this simulator are adequate up to about several 100MHz.

Above this range, effects—such as frequency dispersion—arise that require more complex models. This is the domain of simulators like those that are part of HP's Advanced Design System (ADS). Above 10 GHz, cross-coupling between the components on the circuit board becomes increasingly important. An example, a 50 Ohm strip line with a specific layout in double-L form, is shown in Figure 1. Figure 2 depicts the measurement and simulation results from both ADS and Momentum.



Layout

the problem of cross-talk when using a circuit simulator with discrete lumped components

Figure 1. Cross-talk problem when using a circuit simulator with discrete lumped components

Although ADS (as an important enhancement to SPICE) includes models for specific strip line layouts such as special models for the corners of the depicted L structure and models for the dispersion and skin effect, it assumes no cross-talk or interaction between the individual components of the layout or even interaction with the component itself.

The Electromagnetic (EM) Simulator, on the other hand, is able to precisely cover all these effects. Figure 2 shows that the measured resonance is also predicted by the circuit simulator, but at another frequency. The EM Simulator is able to accurately predict both the shape of the curve and the resonance frequency, by taking into account all the cross-coupling between the double-L strip line structure.

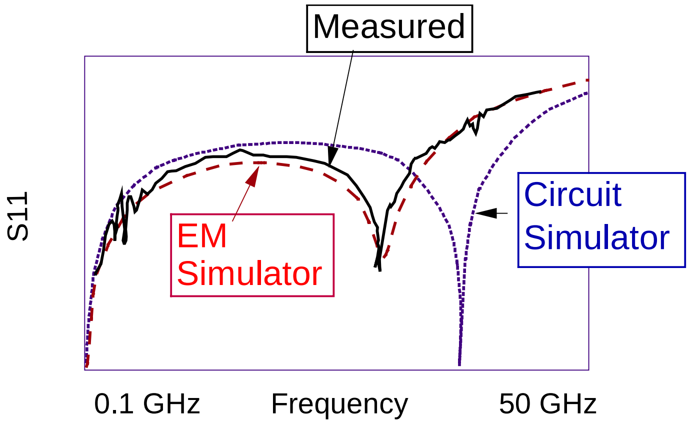


Figure 2. Simulation results compared to the measurement result

As Figure 3 illustrates, EM Simulators require enormous amounts of computing power. This can only be justified for critical and specific simulation problems, for example, details of the whole PCB (printed circuit board). Using the example of a Y branch, the large number of segments into which this component would need to be divided to obtain good simulation results becomes apparent. This implies much longer simulation times for EM Simulators as compared to circuit simulators.

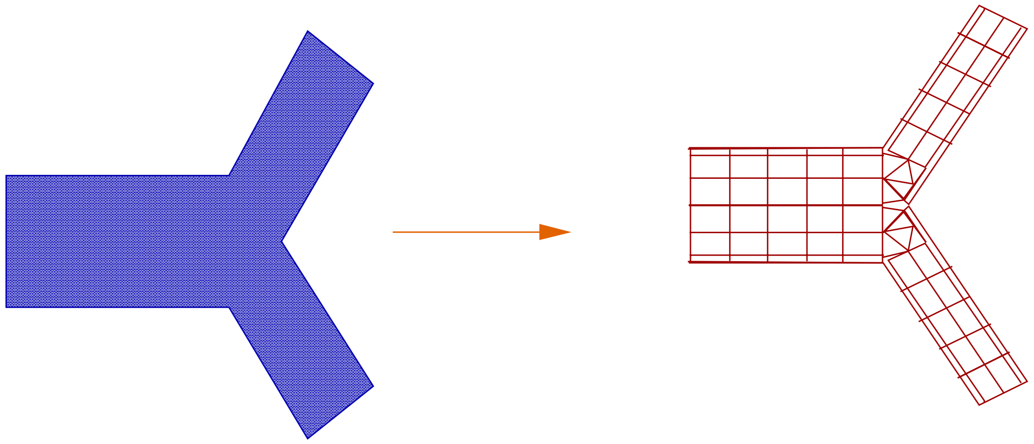


Fig. 3: Segmentation required for EM simulation of a Y branch.

An interesting combination of EM and circuit simulators can be obtained if the S-parameters produced by an EM Simulator can be used as a black-box circuit description in the circuit simulator. This yields an acceptable compromise between accuracy and simulation speed. Momentum in ADS can provide this type of solution.

Model Types for Active Components

SPICE-like simulators include permanently installed ideal resistors, capacitors, and inductors, ideal lines as well as diodes, bipolar transistors, MOS and GaAs transistors. You simply define the model parameter values and then run the simulation.

The models used for nonlinear components can be broken down into three classes:

physical models		
	diode	
	bipolar transistor	Gummel-Poon and VBIC models
	MOS transistor	MOS2 model
empirical models		
	MOS transistor	MOS3, BSIM3 models
	GaAs transistor	Statz and Curtice models
tabular models		
	diodes, transistors etc.	Root model

The equations for physical models are based on fundamental physical considerations. In the case of a diode, the model is based on the diffusion equation for the PN junction. Thus, the diodes current formula is:

$$i = IS * \exp(v / (N * vt))$$

The modeling parameters are:

- IS, the saturation current (a basic parameter of the wafer production process)
- N, the recombination modeling factor
- vt, the thermal voltage

Aside from its clean curve fit, a good physical model is characterized by parameter values (Io and N) that have a physical meaning.
The advantage to using physical models is that sufficiently precise modeling can be expected even in those regions where no curve has been performed. This is because, by definition, a physical model reflects the total physical behavior of the component in all operating ranges.

In the case of an empirical or analytical model, there is no direct relationship between the model equations and the basic physical laws. As an example, let us look at the Curtice model for GaAs transistors. Here, the current i_{DS} is described by:

$$i_{DS}(v_{GS}, v_{GS}) = (A0 + A1 v_{GS} + A2\, v_{GS}^2 + A3\, v_{GS}^3)\tanh(GAMMA * v_{DS})$$

In this equation, v_{GS} represents the effective Gate Source voltage, including the pinch-off effect. The parameters Ai are so-called fitting parameters and describe a third-order relationship between the Drain current i_{DS} and the Gate voltage v_{GS} at a fixed Drain voltage v_{DS}. On the other hand, the dependence of the Drain current on the Drain voltage v_{DS} is described by a tanh function and the additional fitting parameter GAMMA.

Because of their formulation, empirical models should only be used for simulations of those measurement regions from which their parameters have been extracted. In certain situations, this may be a significant limitation in comparison to physical models.

Both physical and empirical models are made up of sets of predefined equations. Only the modeling parameters are accessible to the user, and they are the only things the modeling engineer can adjust.
The advantage to using physical and empirical models is that they can be relatively easily transferred from one simulator to another. The only prerequisite is that the model equations must exist in exactly the same manner on the other simulator. The other simulator will then, within the limits of calculation errors and simulator-specific convergence conditions, produce the same result from the parameter set as the first simulator.

Modeling for tabular models is based directly on the measured values without using model equations. The components of such models are hyper-elements such as transcapacitances. In the case of a 3-pin component with three pin voltages vi , these elements have the general form:

```
C = f(v1, v2, v3)
```

In the case of a GaAs transistor, the following formula is used for the tabular model:

```
iD = iD_DC(v GS ,v DS )+j 2*PI *f *CD(v GS ,v DS )*H(f)
```

where:

iD_DC(v GS ,v DS)	models the Drain current for DC (but only in tabular form)
CD(v G S,v DS)	represents a nonlinear transcapacitance at the Drain
H(f)	covers the dispersion between DC and AC behavior. (a specific GaAs transistor effect)

Finally, iD is available in a tabular form. Intermediate values are interpolated with spline functions. As the equation indicates, it is sufficient to acquire S-parameters at as many DC operating points as possible and then to derive the model from them.

The advantage of a tabular model like the Root Model is that it is able to provide a curve fit even when physical or empirical models fail due to their fixed, predefined equations. However, tabular models are generally linked to specific simulators, and the transfer of modeling results to another simulator is very complex. This is because the other simulator must be able to handle the specific format of the table being transferred. Another interesting point is that tabular models may be limited to accurately model the frequency harmonic's interferences only up to a certain order that is related to the mathematical order of the spline functions. Also, a luped schematic model can be used outside the measurement limits (e.g. S-parameter measurements within a limited frequency range, extrapolating the model performance to DC or above the upper frequency limit), and a lumped schematic model may also be easier to interprete for statistical variation analysis than a table model.

Within the group of transistor models, the MOS models can be considered separately since they are scalable, see fig.4. This means, based on a set of different transistor geometry measurements, a common set of parameters is derived which is valid afterwards for all geometries of that specific MOS process. As a consequence, MOS models like MOS2, MOS3, BSIM3v3, and MOS9 have a complex formulation and thus are also rather complex to be modeled. The SPICE models of the other types of transistors are, however, not scalable.

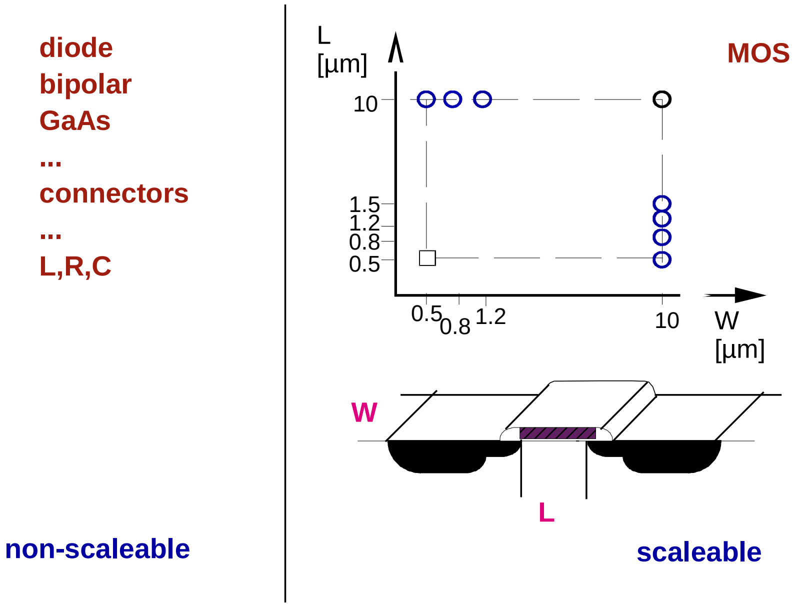
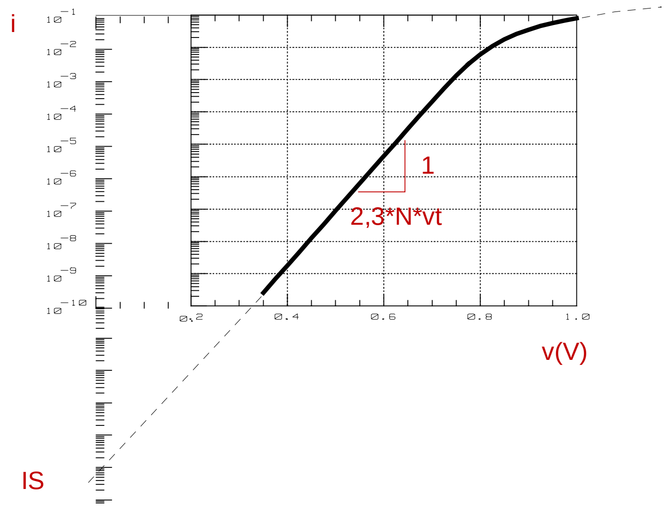


Fig. 4: scaleable and non-scaleable SPICE models

What is Modeling?

Modeling is finding the parameter values for the ususally fixed simulator model equations at which the simulation produces a curve that corresponds extremely well with the curve representing the measured data. When the two curves are identical, or nearly identical, it is referred to as a good fit. A good model allows you to make clear predictions about the behavior of the component in general applications. Figure 5 shows a good fit between the measured data (the solid line) and the simulation results (the dashed line).



```
                     Stimulus for measurement and simulation
                    /
                   v
                ------
                N * vt
i  =  IS  *  e   \    \
 \      \         \    Temperature voltage, a known parameter
  \      \         Model parameter
   \      Model parameter
    Simulation result, to match the measurement result.
```

Fig. 5: A good fit between the measured data and the simulation result (dashed)

The above equation is part of the standard diode model of the simulator. It is therefore the parameters IS and N that have to be determined. In this example, IS represents the y-intersect of the half-logarithmically plotted measurement curve, and N is calculated from the slope of the curve.

Types of Measurements Required for Modeling

It is generally true that modeling at the operating point of a component is no longer adequate for today's applications. Since almost all signals being processed today are digital, the total output characteristic is intersected with the clock rate of the digital signal. Therefore, a complete modeling from DC to the operating frequency for analog and up to five times that frequency for digital signals has to be applied.

With passive components, i.e. capacitors, resistors, inductors and similar components, discrepancies between SPICE-type simulations and the actual component only arise above approximately 100MHz. In this case, an S-parameter measurement for the region > 50MHz is performed. For modeling, an equivalent schematic—whose core is the ideal component—is developed. Based on the trace of the S-parameter curves, additional components are then successively added to that circuit. Their values are then determined from the measurements, and the simulation curves are brought into alignment with the measured ones by either manual parameter tuning or automatic optimization.

The following methodology generally applies to active components:

- Modeling the parasitic components (feed resistances, stray capacitances etc.) from S-parameter measurements
- Measuring and fitting the input characteristic
- Adjusting the transfer curve (ß for bipolar, gm for MOS transistors)
- Fitting the output characteristic. For MOS, fitting the dynamic output resistance
- Modeling the space charge capacitances (CV measurements)
- Adjusting the S-Parameter curves for the small signal behavior in the GHz range
- Checking the device performance in the time domain (which is most often omitted)

This means that regardless of the transistor technology involved, the following modeling tasks must be performed:

DC:

`iin   = f(vin)` Exception: MOS `iin = 0`

`iout  = f(vin,vout)` S_{11} is used instead !!

CV (1MHz):

$$C_s = \frac{C_j}{\left(1 - \frac{v}{V_j}\right)^{M_j}}$$

S-parameters (>50MHz):

```
CD    = Ttransit * gm                 d(iout)
                           with  gm = --------
                                      d(vin)
```

Fig. 6 illustrates these tasks.

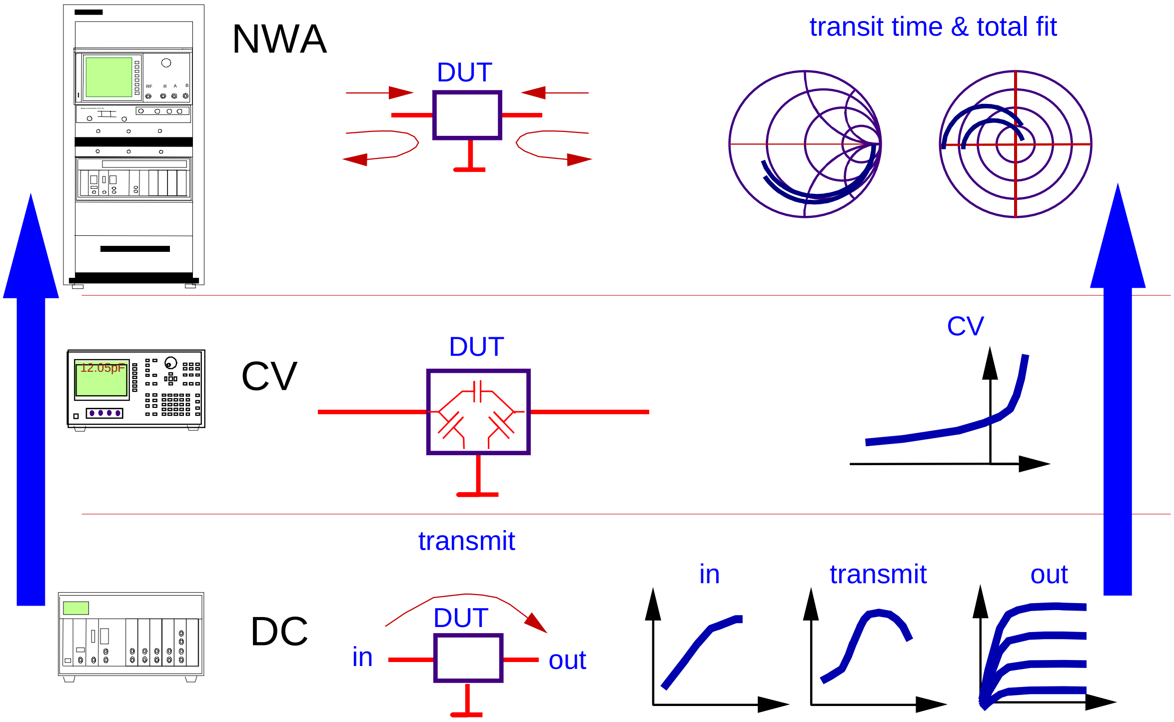


Fig. 6.: Overview of the modeling measurements required for the different modeling steps

Why use Modeling Instead of Component Libraries?

Component libraries, the core of simulation tools, contain ready-to-use sets of parameters for previously modeled electronic components. Today, with the trend towards higher frequencies and smaller geometries, components are operating at their limits. This underlines the importance of identifying critical components and modeling them very carefully. The properties and tolerances of critical operating points are the keys for stable production with high yield. As an example, a bipolar transistor is often characterized by its current amplification ß = iC / iB which is reflected mainly by the Gummel-Poon parameter, BF. Component libraries, even if their quality is good (which is not always the case), contain model parameters that represent a typical component. They do not contain information about statistical spread.
Figure 7 illustrates how widely spread the trace of ß is and how the parameter value BF varies related to the manufacturing of the transistor. It shows the spread of current amplification ß for bipolar transistors on a wafer. The dotted line is a simulation result using library parameter values.

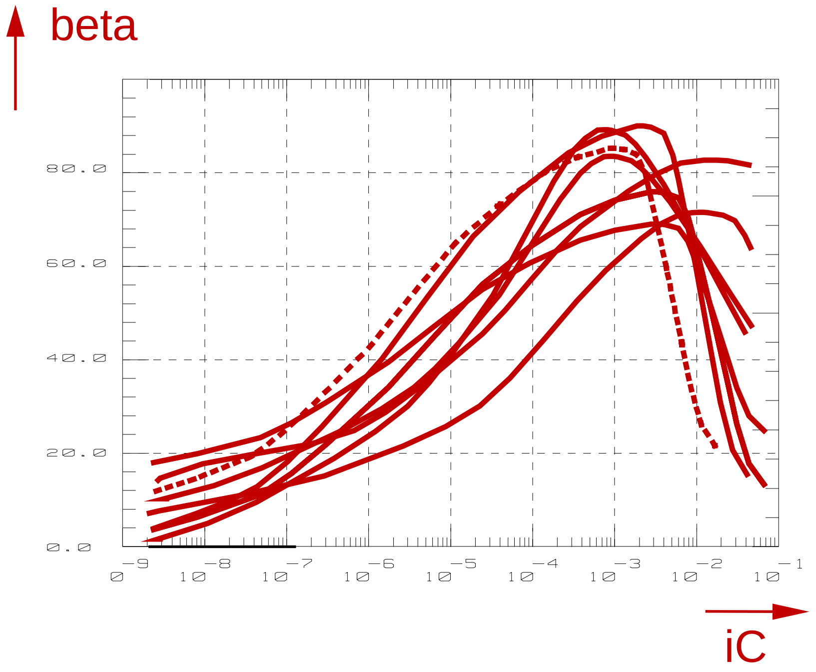


Fig. 7: Spread of current amplification ß for bipolar transistors on a wafer.
Dotted line: simulation result with library parameter values

If this transistor were the critical component of the whole design, and if we would simply use its parameters from a component library, we would not be able to obtain reliable simulation results. As a consequence, producing the circuit might lead to major reliability problems.

Another example is the design of VCOs (voltage-controlled oscillators) in the GHz range. If it were determined that the selection of the resonance capacitor represented a critical factor for production results, modeling would be the ideal answer to the problem. Modeling would give valuable information about the limit frequency of this component, until it changes its behavior from being an ideal capacitor to become affected by the parallel resistor and the non-negligible inductor, where its resonance frequency lies and how it behaves above that frequency. In our example, this would lead to more detailed specifications for the manufacturer of the capacitor and, as a consequence, would receive a better production yield, the ability to analyze the reserves of the circuit design, and the ability to apply design centering.

Of course, no library models are available for components that employ the latest leap in technology. For companies with access to latest technologies, modeling becomes the key factor for achieving successful market share for their products. The way to stay ahead of the competition is to use the proprietary models of the proprietary lead technologies in the advanced product design.

Figure 8 provides another perspective. The example represents a TDR (time-domain reflectometry) measurement of an IC housing, a component that, in the past, was often considered non-critical. However, with today's technologies, where the chip inside the housing operates in the GHz range, its performance can be drastically affected by the package.

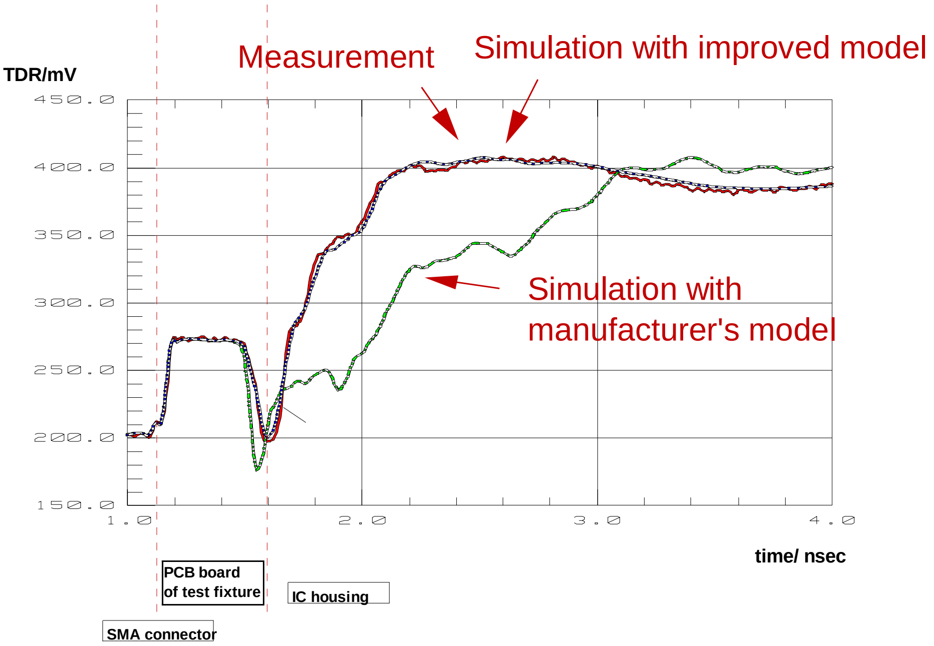


Fig. 8: TDR measurement results from a single pin of an IC housing. Also shown are simulation results using the manufacturer's set of parameters and the improved result due to a more detailed and accurate user-defined model.

see: H.Katzier, R.Reischl, P.Pagnin: SPICE Models for High-Pincount Board Connectors, IEEE Trans.on Components, Packaging and Manufacturing Technology, part B, Vol. 19, No.1, Feb.1996, pp. 3-6

Looking at figure 8 and the measurement result, we see the TDR test pulse reaching the test fixture PCB board with a higher characteristic impedance than the one of the cable before. Then, the signal reaches the IC housing. Right after the backdrop at the beginning of the package, we see the open-end reflection of the open pin unside the package.
Next, checking the simulation result of the manufacturer's model and its parameters, we only achieve a pretty poor fit. After having developed a more accurate and more detailed model for the package and having determined its model parameters carefully, we can improve the modeling fit quite drastically.

Figure 8 also shows simulation results using the manufacturer's set of parameters and the improved result using a more detailed and accurate user-defined model. With accurate packaging and connector modeling, it can be determined whether a more expensive component must be bought, or if some slight modifications to the existing, less expensive components are sufficient to meet the application specifications. This can save a considerable amount of money in production.

Some additional comments about component libraries

Close examination often reveals that many parameters are standard or default values, and therefore have not been specified for the component in question. The parameter values with no physical meaning (for example, NF >> 1 or VJ >> 1 for the Gummel-Poon model also have not been specified for the component). In this case, it is questionable if the simulation result in the time domain, a range which is not commonly used for modeling, would be reasonably accurate.

The parameters for reverse operation are often either zero or infinite. The model is switched off for this particular operating mode. This can drastically affect time-domain simulation, because this is where switching from on to off, and from forward to reverse operation will occur.

Direct Extraction Vs. Model Parameter Optimization

Depending on the complexity involved, a model for an electronic component is described by anywhere from 10 to more than 100 parameters. When applying direct parameter extraction, one measurement or one measurement region is defined for each parameter in which it is dominant. While this is a very time- and labor-intensive process, it has the advantage of making the parameter values unique and reproducible. If, however, only an optimizer is used to fit the curves, the parameter values after optimization are dependent on the starting values before the optimization. This is because the optimization problem is mostly over-defined (more parameters than there are measurement regions), or the optimizer locates a local minimum of the error function. This is often called the addition of optimizer noise to the real model parameter values. This is a specific problem when many individual components of the same production lot have been modeled, and a statistical analysis of the parameter interdependencies is executed (factor or principal component analysis). In this case, such a statistical analysis might lead to parameter interdependencies that have no physical background. A common compromise is to extract as many parameters as possible directly, and to employ the optimizer only for the fine-tuning, as is illustrated in figure 9.

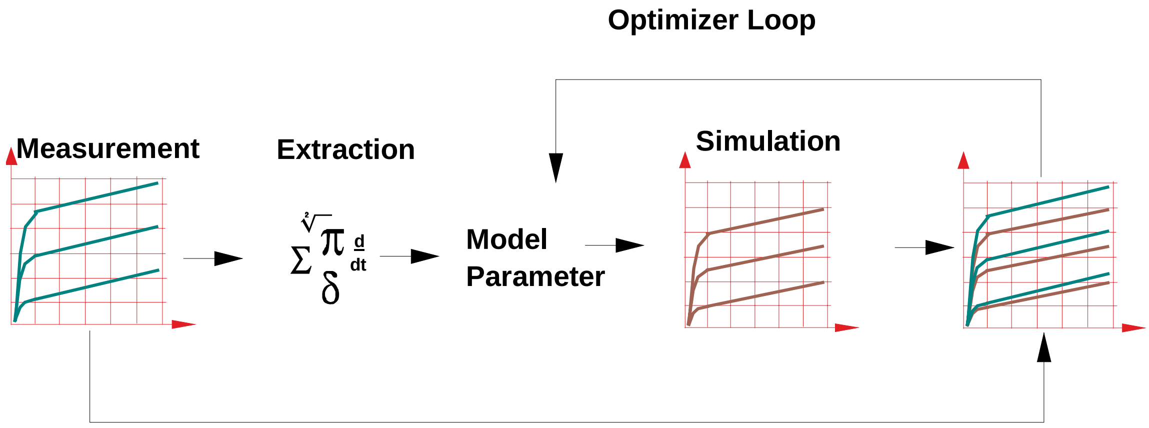


Fig. 9: 'Direct extraction' of parameters using the optimizer only for fine-tuning.

Last but not least

Once the appropriate model has been created, remember to perform the following tasks:

- Define the valid region for the developed model (frequency limits, operating point restrictions, max.amplitude, etc.)
- Document your work
- File the model parameters in the component library

Once all of these tasks have been completed, it can be assumed that the model willaccurately predict the behavior of the component in all operating ranges.

Publications:

F.Sischka, 'Device Modeling and Measurement for RF Systems', VLSI conference proceedings 1999, Lisbon, Kluwer Academics.

F.Sischka, 'RF Measurements And Modeling, With Special Emphasis on Test Structures', Tutorial Short Course at the ICMTS 2000, Monterey, CA.