Development of Complex Curricula for Molecular Bionics and Infobionics Programs within a consortial* framework**

Consortium leader

PETER PAZMANY CATHOLIC UNIVERSITY

Consortium members

SEMMELWEIS UNIVERSITY, DIALOG CAMPUS PUBLISHER

The Project has been realised with the support of the European Union and has been co-financed by the European Social Fund ***

**Molekuláris bionika és Infobionika Szakok tananyagának komplex fejlesztése konzorciumi keretben

***A projekt az Európai Unió támogatásával, az Európai Szociális Alap társfinanszírozásával valósul meg.

VLSI Design Methodologies

(VLSI tervezési módszerek)

Simulation techniques

(Szimulációs technikák)

PÉTER FÖLDESY

The topics are covered in this chapter:

- Analysis methods
- Symbolic analysis
- DC, AC transfer curves and behavior
- Transient simulation
- Noise, sensitivity, monte-carlo, corner analysis
- Evaluation of results, possible calculations on them

Section I

Why to simulate, how, and with which tools.

Why analyze circuits?

- Determine the influences of element parameters on circuit behavior
- Extraction of dominant circuit behavior
- Error and tolerance analysis

Circuit modeling

- Support of model generation for analog circuit blocks (on different hierarchical levels)
- Allow for overall circuit simulation by use of behavioral and macro-models

Why analyze circuits?

- Circuit sizing
  - Support manual or computer-aided circuit synthesis, remember for circuit templates
  - Derivation of symbolic (generic) sizing, usage for optimization
- Circuit optimization
  - The analysis of parameter dependence is the key for constraint driven design and automatic optimization

How to analyze operation?

- Paper and pencil
  - the most important way of estimating operation, design new architectures
- *Symbolic analysis*
  - A kind of automatic paper and pencil analysis
  - Based on simplified empirical models, than symbolic mathematics helps to simplify dependences (e.g. Maple, Matlab, Mathematica)
  - Important feature is the numerical simplification, that helps to see the important functionality

How to analyze operation? *Electronic simulators*

- These simulators that use mathematical models to emulate the behavior of electronic device or circuits built from devices
- The models are typically analytical (or mixed behavioral) and filled up with actual parameters of the used technology
- The most famous is the *SPICE* (Simulation Program with Integrated Circuit Emphasis) and its variants (HSPICE, PSPICE, Spectre, etc.)
- The SPICE combined first many analysis functionalities, like operating point solutions, transient analysis, and various small-signal analyses

Electronic circuit simulators.

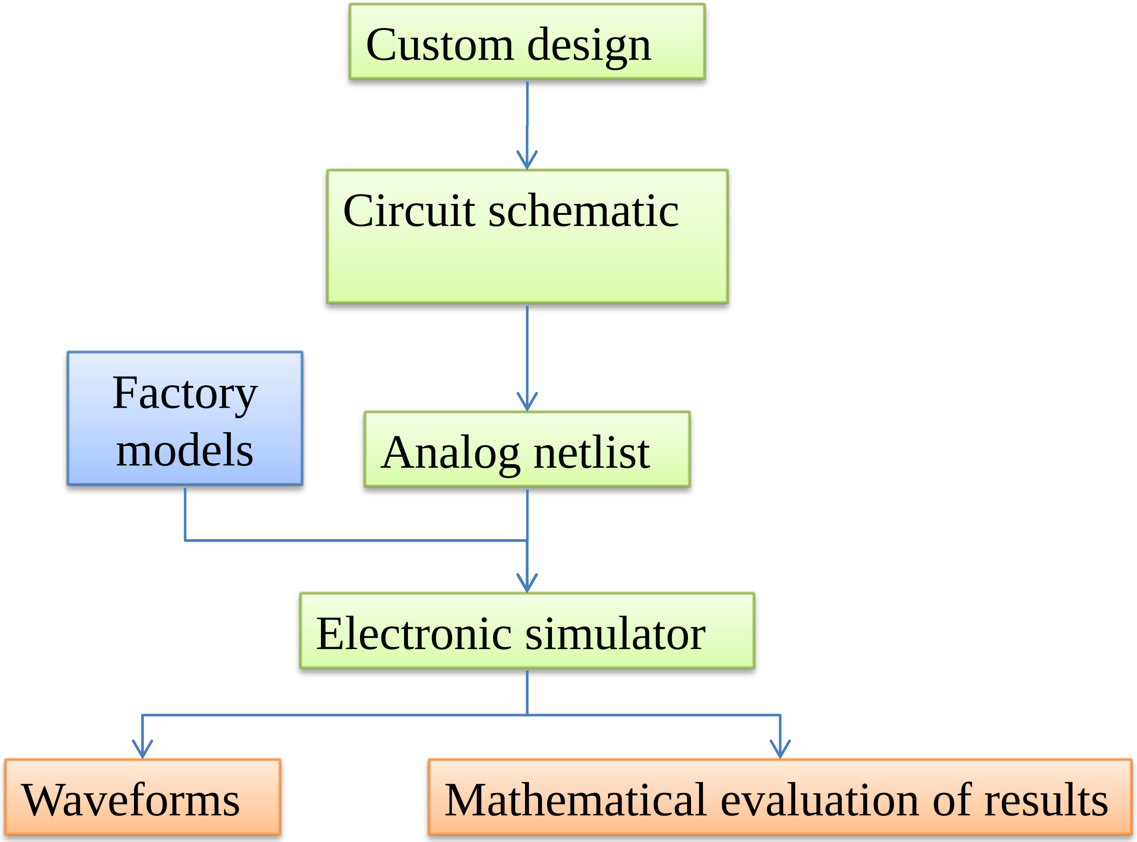
The most frequently used models are

- for transistors are the BSIM3, BSIM4, BSIMSOI, mostly physical models
- for realistic resistors, capacitors, inductors compound models of ideal elements of the same and/or diodes
- user defined controlled current and voltage sources, or models in Verilog-A or VHDL-AMS

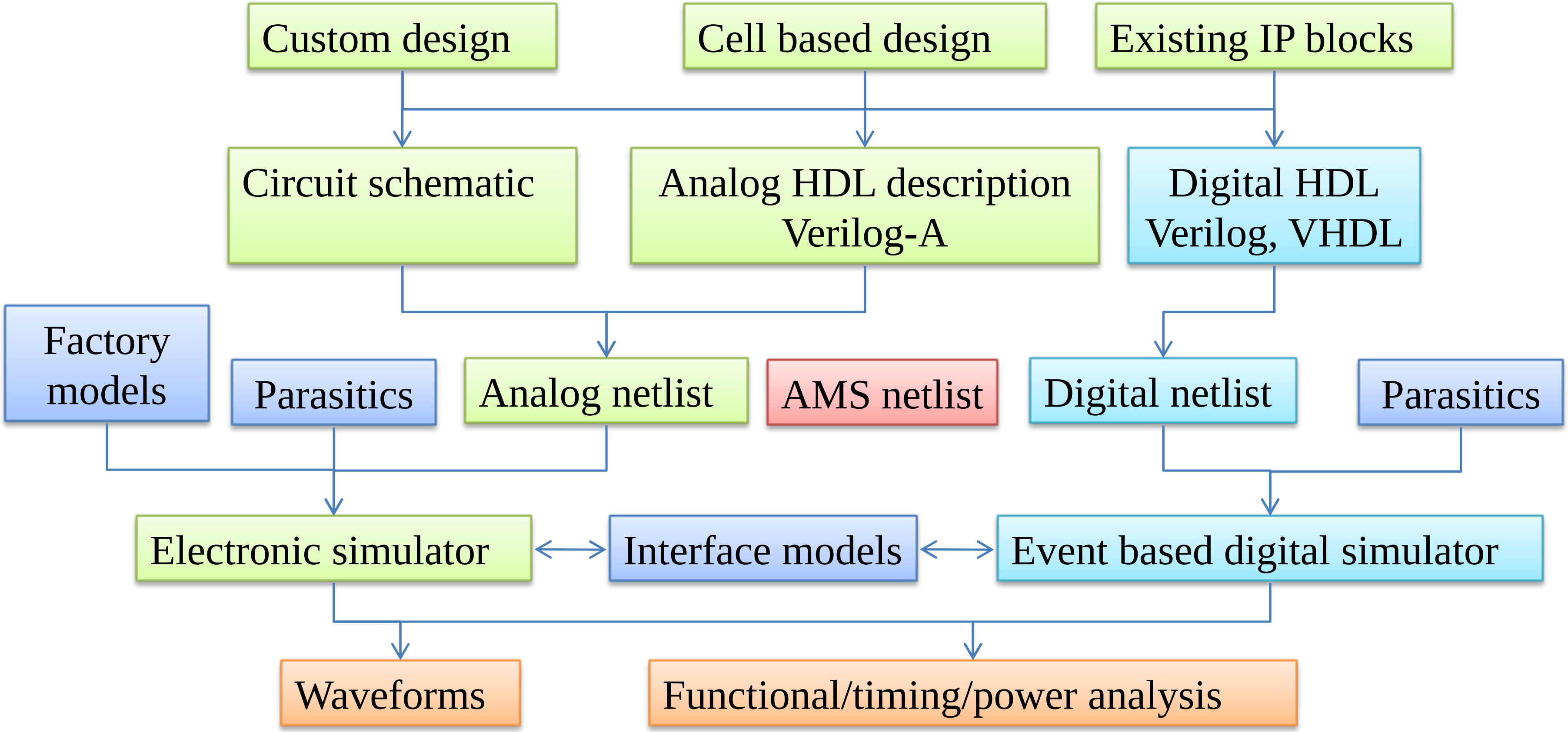
Electronic circuit simulators

- During operation, first a netlist is created containing the circuit elements (transistors, resistors, capacitors, etc.) and their connections.
- After translation to equations, nonlinear differential algebraic equation systems, the simulator solves it using implicit integration methods.
- Finally the evaluation comes with waveforms or with scripts and equations, statistics.

Simplified electronic circuit simulation flow



Mixed-signal electronic circuit simulation flow



Section II

Advanced simulation methods and tools

Advanced versions of simulators: *fast-SPICE*

- In many cases, the circuit is far too large for direct analog simulation (it works up to a few thousand transistors only)
- The solution is to drop precision and gain speed. The simulators are capable to trade precision versus speed are called fast-SPICE simulators (e.g. NanoSIM – Synopsys, UltraSim – Cadence, FineSim Pro – Magma)

Advanced versions of simulators: fast-SPICE

- Such simulators are used for transistor level verification of large custom, analog/mixed-signal, RF, memory and SoC designs. The key of them is hierarchical simulation with adaptive partitioning algorithms, and balance tradeoff separately.
- The runtime for a DSP, 12-16 bit ADC is about a few hours. With classic non accelerated simulators, only the initial condition is found under this time!
- Worth to mention the hardware accelerators or emulators (e.g. FPGA based logic equivalents).

Advanced versions of simulators: *Mixed signal simulators*

- The mixed signal simulators are combination of analog oriented nonlinear equation solver SPICE simulator and an event based digital simulator
- There are mixed HDLs, like Verilog-AMS or VHDL-AMS (remember, the verilog-A is only for analog behavior).
- The key is the interface elements in between. Which are simple linear, piece-wise linear, or non-linear elements with timing information up to complex transistor circuits.

Advanced versions of simulators: Mixed signal simulators: Interface elements

- The most simple is simple step with given rise, fall, high, low levels (No control on input delay at the input ports of analog module. No control on transition time at the input ports of analog module).
- More general interface models should be calibrated or validated against the transistor level implementation (if they are simplified) to make them accurate enough for the simulations to make sense.

Advanced versions of simulators: Mixed signal simulators

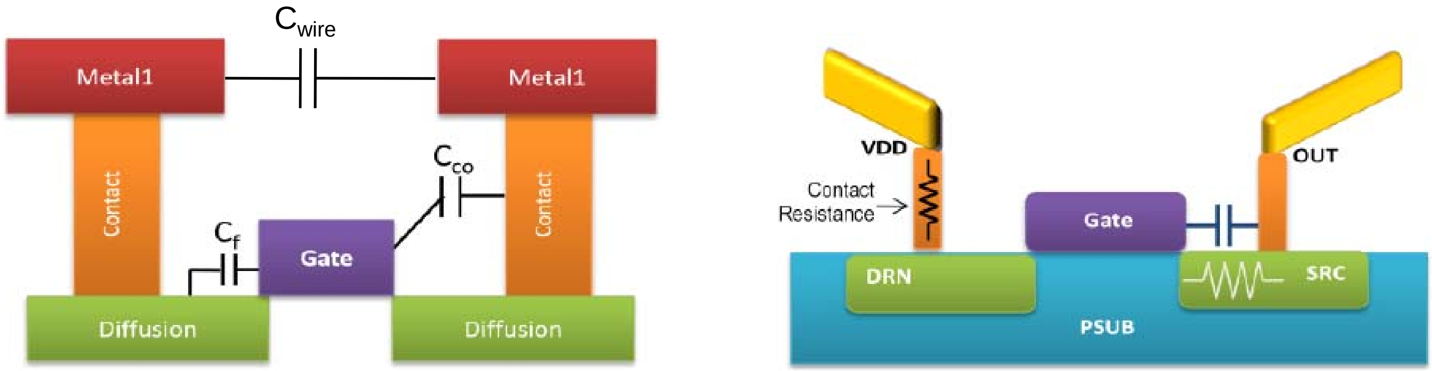
- In an ultimate form, when many blocks are represented with different views (digital, behavioral analog, transistor level), a system is represented as a soup of different models specified and tuned for different verification situations.
- An example for it the Cadence’s hierarchy editor.
- As the simulation time elapses, the models may also change. E.g. in a PLL, the initial lock-in cycles are simulated in analog domain, while when its output clock signal is stable, the digital form is more adequate.

Advanced versions of simulators: Mixed signal simulators

- The question of modeling rises with the possibility of quick change of representation.
- Behavioral modeling has great potential for faster and better mixed-signal system verification.
- Replacing smaller components with small amounts of switching activity may not give faster performance, a component with significant switching activity is replaced by its behavioral model, we have a great improvement in simulation speed.

Parasitic simulation

- Simulation case, when the parasitic elements (resistance, capacitance and inductance) of interconnections are also involved in the functional simulation.

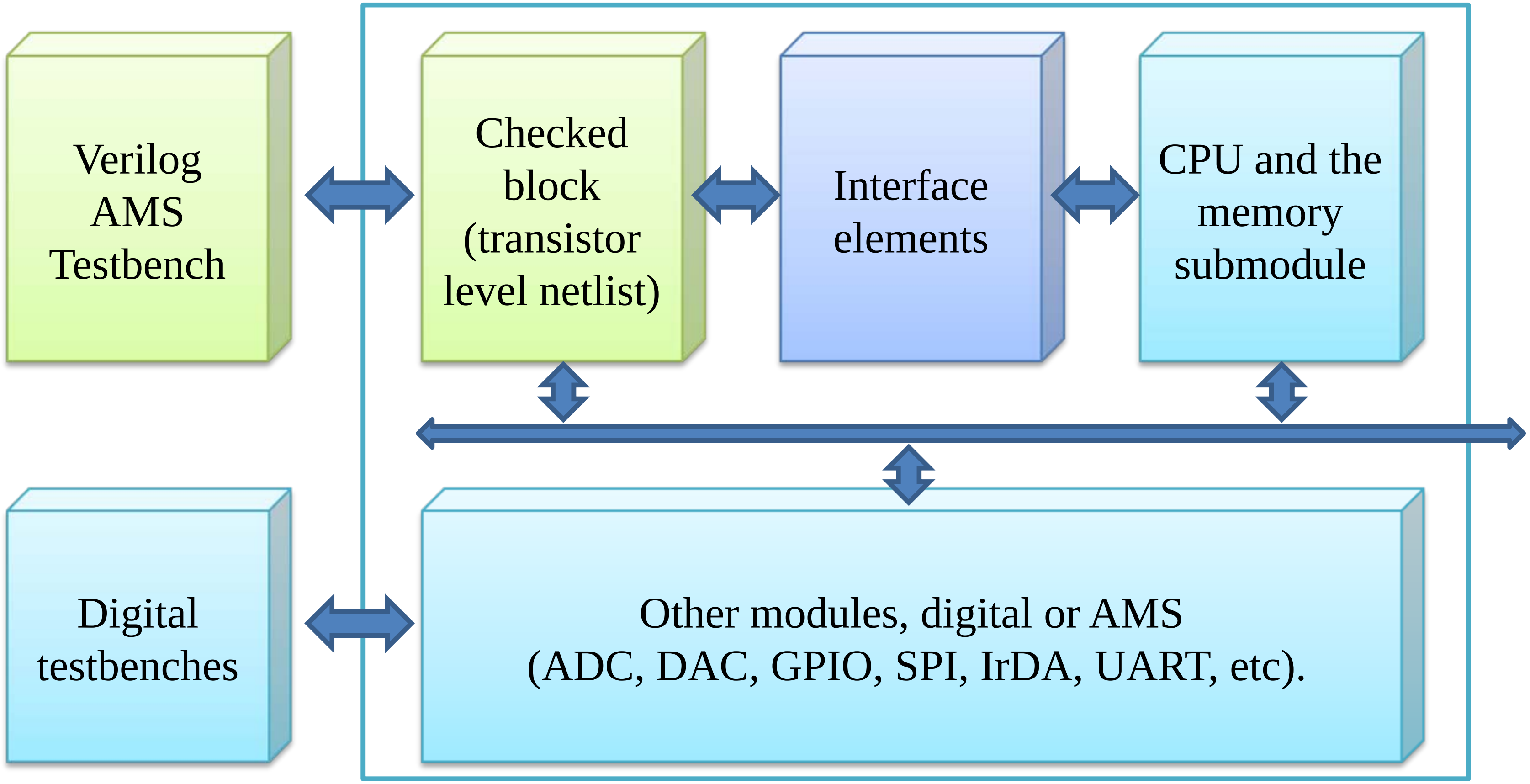


Typical situations where parasitic affect operation.

Importance of parasitic simulation

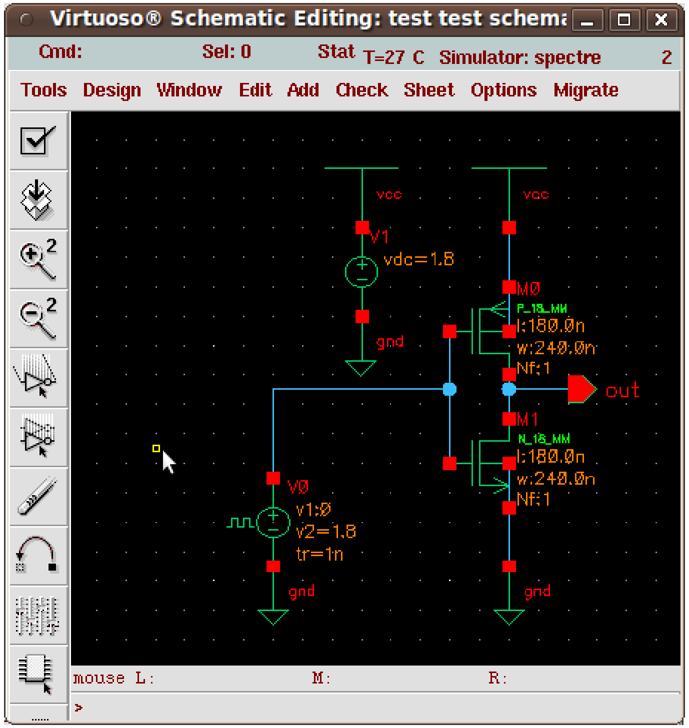
- In analog and digital designs the parasitic effects slow down the operation and could cause fatal timing and functional failure.
- Interconnect and device parasitic effects are estimated to account for over 60 percent of the delay at 28-nm. Design margins (timing, noise, and power) are decreasing, making accuracy more critical for parasitic extraction and simulation.
- Post-layout simulation (parasitic simulation) runtimes are increasing 2x to 4x with every new process generation as chip transistor counts double and new parasitic effects come into play.

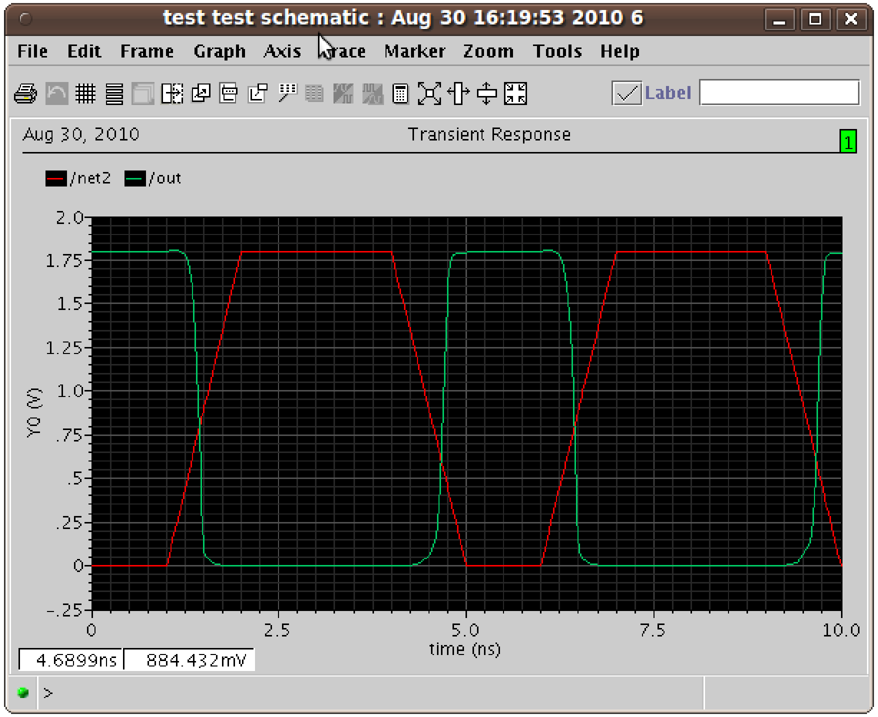
Mixed signal description of a SoC



Typical components of a simulator environment

- Schematic editor, netlist, simulator itself, waveform window





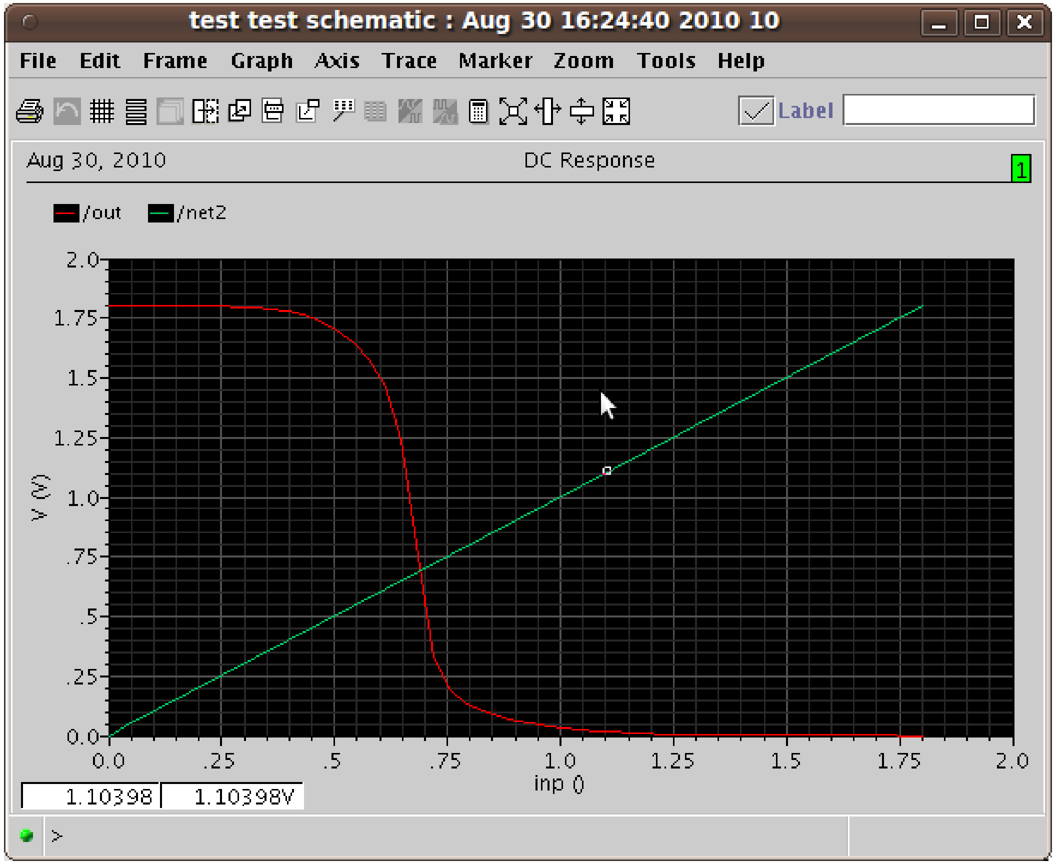
Section III

Typical tasks and functions that an electrical simulator shall know

Common analysis functions

- DC or static (nonlinear operation point calculation)
  - The operating point, power consumption, amplifier dynamic range, etc. can be easily characterized under static conditions
  - Operating point calculation, used for initial condition calculation for further analysis's
  - Transfer curve, a series of operation point calculation by sweeping a few parameters, like transistor size or input
  - Temperature, power supply dependency sweep
  - Sensitivity as small signal partial differentia calculation

- Transfer curve, a series of operation point calculation by sweeping the input of the inverter



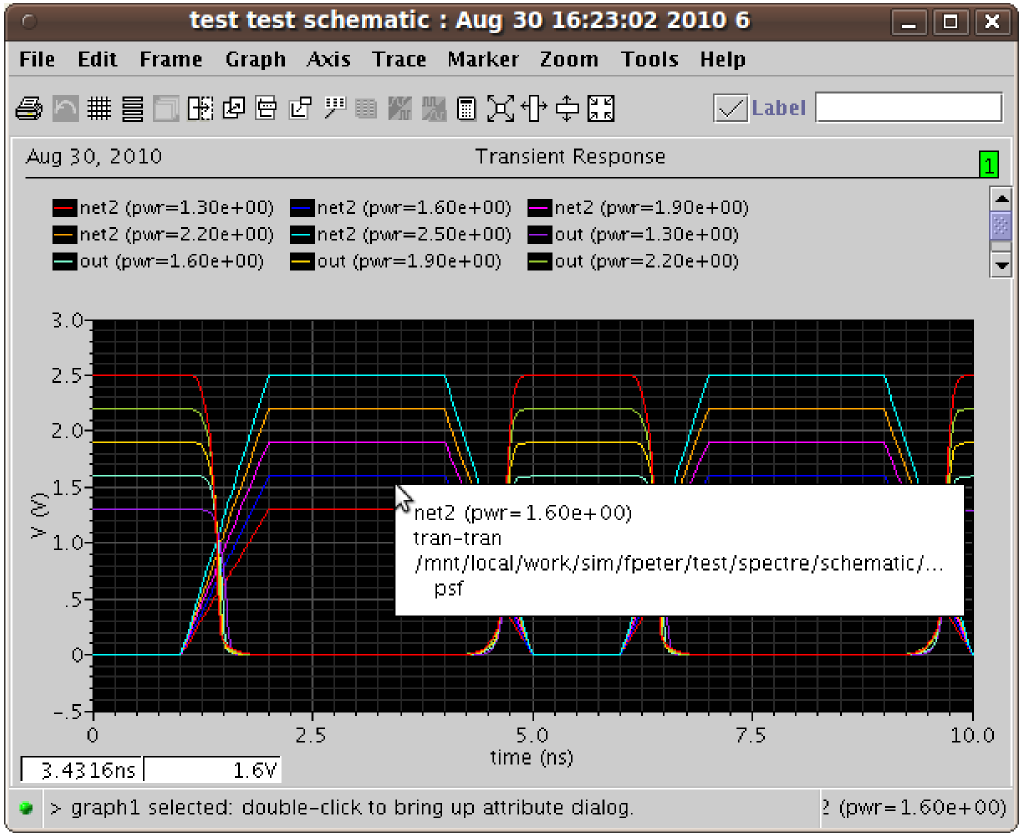
Common analysis functions

- AC (linear small-signal frequency domain analysis)
  - Small signal behavior around a DC operating point
  - Pole/zero analysis for identify instability
  - Frequency sweep and small signal response
  - Phase response for identify instability

Common analysis functions

- Transient (time-domain large-signal behavior)
  - Combined with DC sweeps, many parameter dependency can be analyzed how they affect actual operation like temperature, power supply changes
  - Observation of nonlinear behavior
  - Distortion analysis (harmonic distrortion)
  - Extensions for Static State (periodic) operating point
- Monte-carlo
  - Manufacturing imperfection effect of component variations on performance

- Transient combined with DC sweeps of the power supply of the example inverter



Common analysis functions

- Corner analysis
  - Not really a new analysis function, but a permuted parametric sweep of commonly accepted parameters.
  - These are the operating conditions (temperature, power supply) divided typically to slow, typical, and fast corners.
  - The manufacturing process variations are also included
  - The goal of the corner analysis is to find the worst-case scenarios as the weak points of our circuit and so help to make it robust against imperfections.

Common analysis functions

- Noise analysis
  - The intent is to predict the noise of the circuit in the form of a noise spectrum (interpreted as an AC type analysis with operation frequency sweep).
  - Most simulators can yield a noise power for devices that results from integrating this noise spectrum over a bandwidth. Easy to point to the most noisy element.
  - The most important noise sources:
    - White (shoot) noise: result of discrete charges
    - Johnson noise: high energy electron jumps
    - Flicker noise (1/f)

Evaluating results

- Difficult to evaluate waveforms without tools
- Digital operation related “measurements” on the analysis listed above – remember for digital flow!
  - Rise time, fall time, usually defined as 10%, 90% cross
  - Overshoot
  - Delay
  - Output impedance
  - Cross-talk
  - Power consumption, peak, average

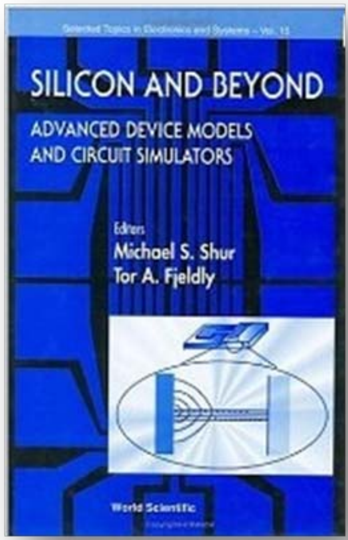
Evaluating results

- The question of “operational” in analog design has no a yes/no answer. But a composition quantities like:
  - Gain of an amplifier
  - Frequency response (bandwidth)
  - Dispersion of the results in e.g. a monte carlo analysis
  - Distortion (RMS harmonics)
  - dB (intensity: 10*log(A/B), power: 20*log(A/B))
  - RMS (root-mean-square) noise
  - Average power consumption

Conclusions

- The circuit analysis is an important part of the circuit design.
- It helps to select and parameterize architecture
- Gives insight view to circuits without expensive and time consuming try and correct cycles
- The analysis with automated measurement and evaluation functions helps to model blocks for digital and analog/mixed-signal flow for implementation and quick simulation

Recommended literature



Silicon and Beyond: Advanced Device Models and Circuit Simulators
Michael Shur (Editor), Tor A. Fjeldly (Editor)
Publisher: World Scientific Publishing Company; 1st edition (July 15, 2000)

Comprehension questions:

I. Why to simulate circuits instead of manufacture and try-out?

II. What are the ways to simulate the behavior of a circuit?

III. What is the SPICE simulator?