

An Interactive Website as a Tool for CAD of Power Circuits

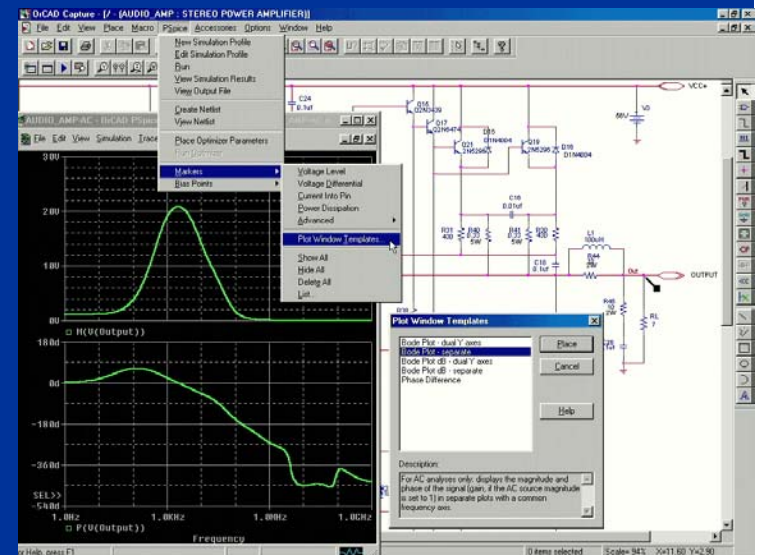
Bartłomiej Świercz
Łukasz Starzak
Mariusz Zubert
Andrzej Napieralski

Technical University of Łódź, Poland
Department of Microelectronics and Computer Science



Commercial vs. Free CAD Software

- Commercial
 - High license cost
 - Models and algorithms: often not the best
 - Not education-oriented
 - Free releases have limitations
- Open Source-like Projects
 - Free
 - Customizable
 - New models & algorithms
 - Support for education
 - Disadvantages also exist

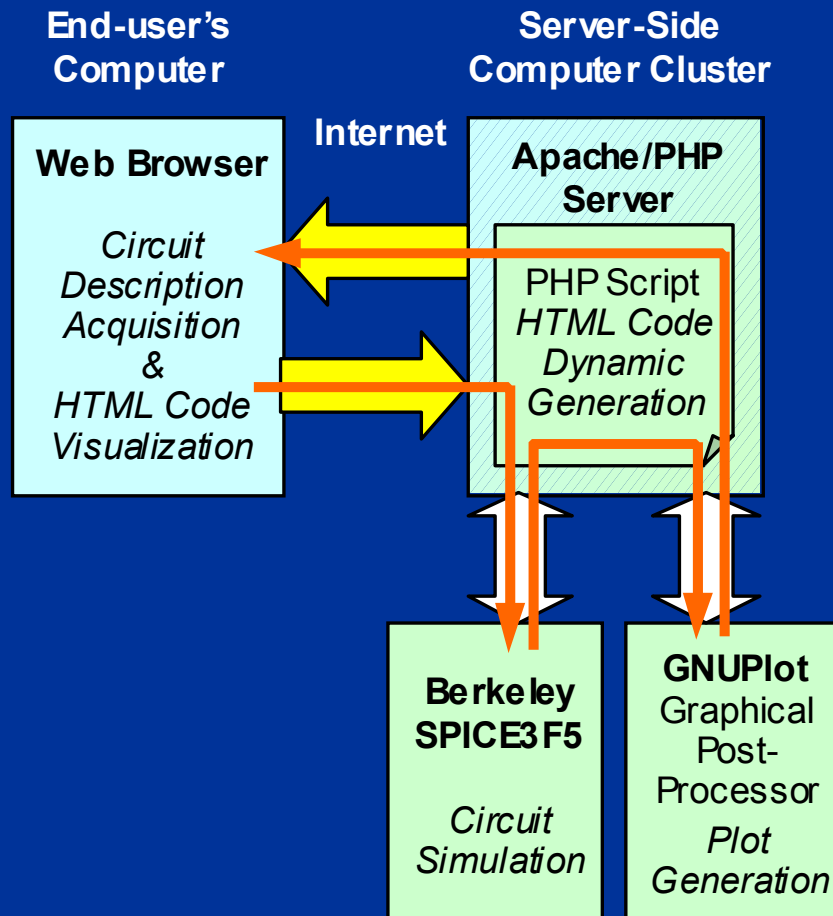


Internet & WWW: New Perspectives

- Making software available
 - Developers: easier software spreading
 - Users: easier software usage (no installations, updates etc.)
 - Support for education and distance learning
- Application development
 - Community of involved users
 - New releases are immediately available
 - Higher software quality: tested by many users



Package Structure

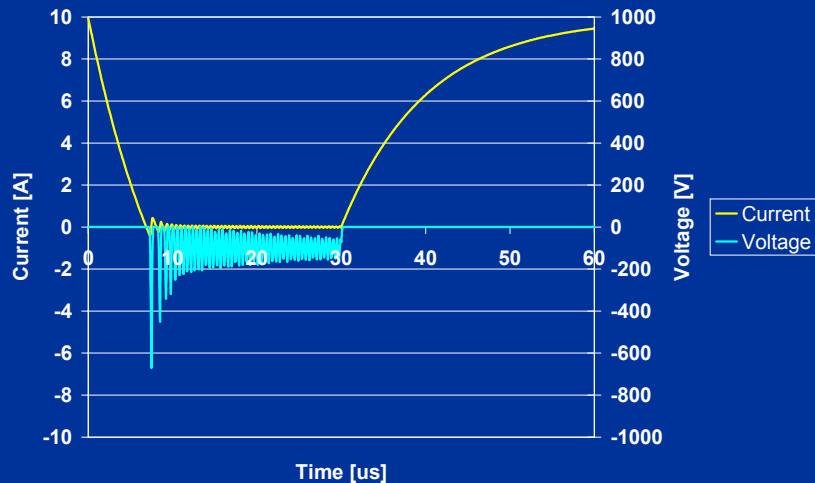
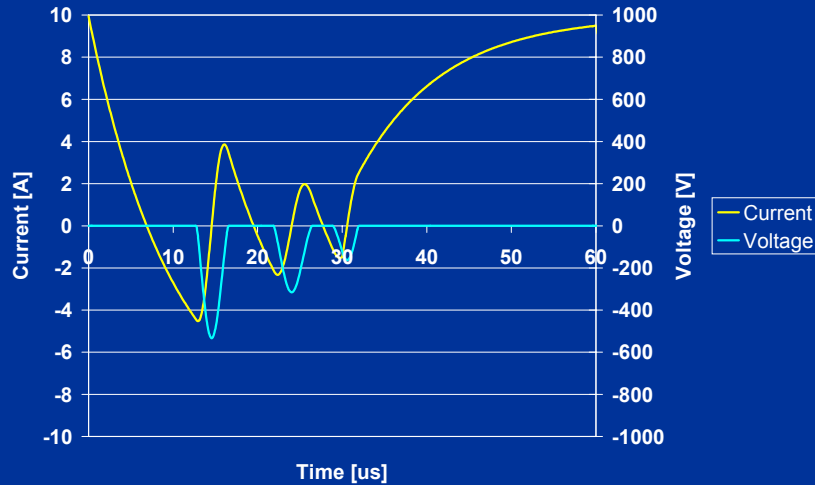


- Simulation and data processing on the server
- The end user only needs a web browser
- Only Open Source (e.g. GNU License) or other free software has been used during development

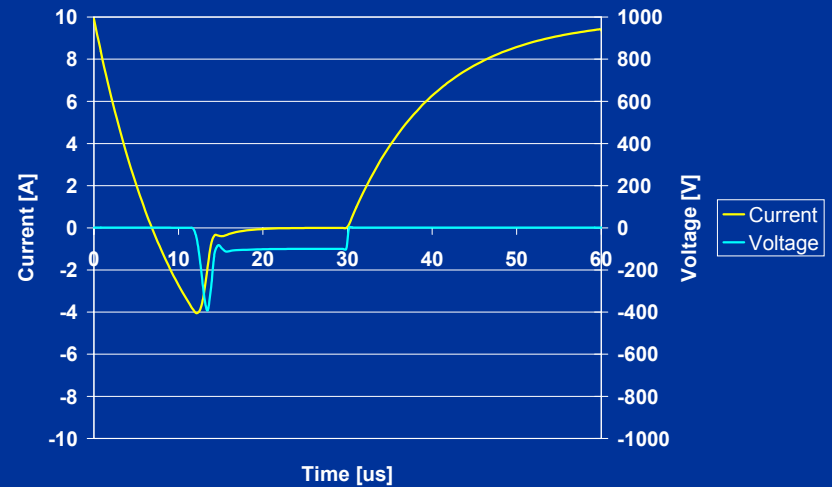


Power Semiconductor Device Model Comparison

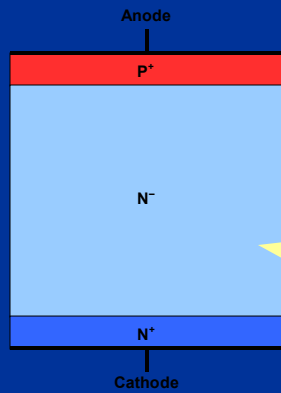
Behavioral SPICE models



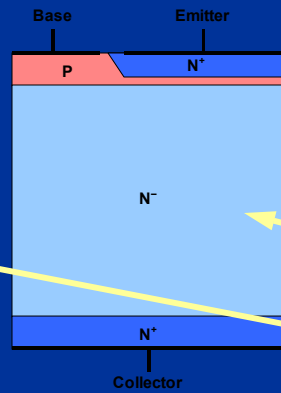
Distributed model simulation



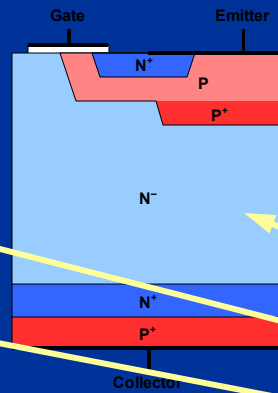
The Wide Base Layer



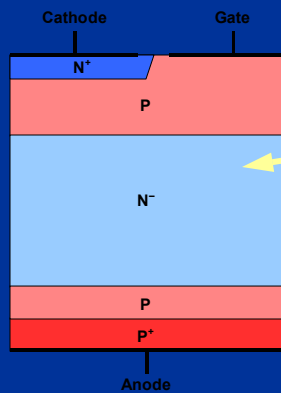
PIN Diode



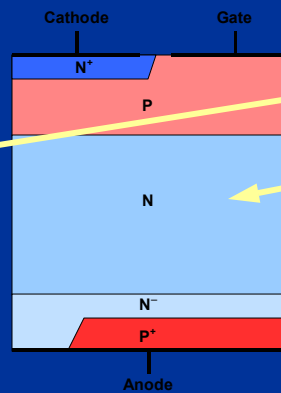
Power BJT



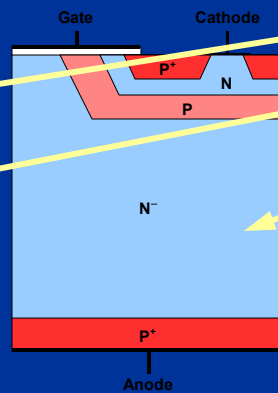
IGBT



SCR



GTO



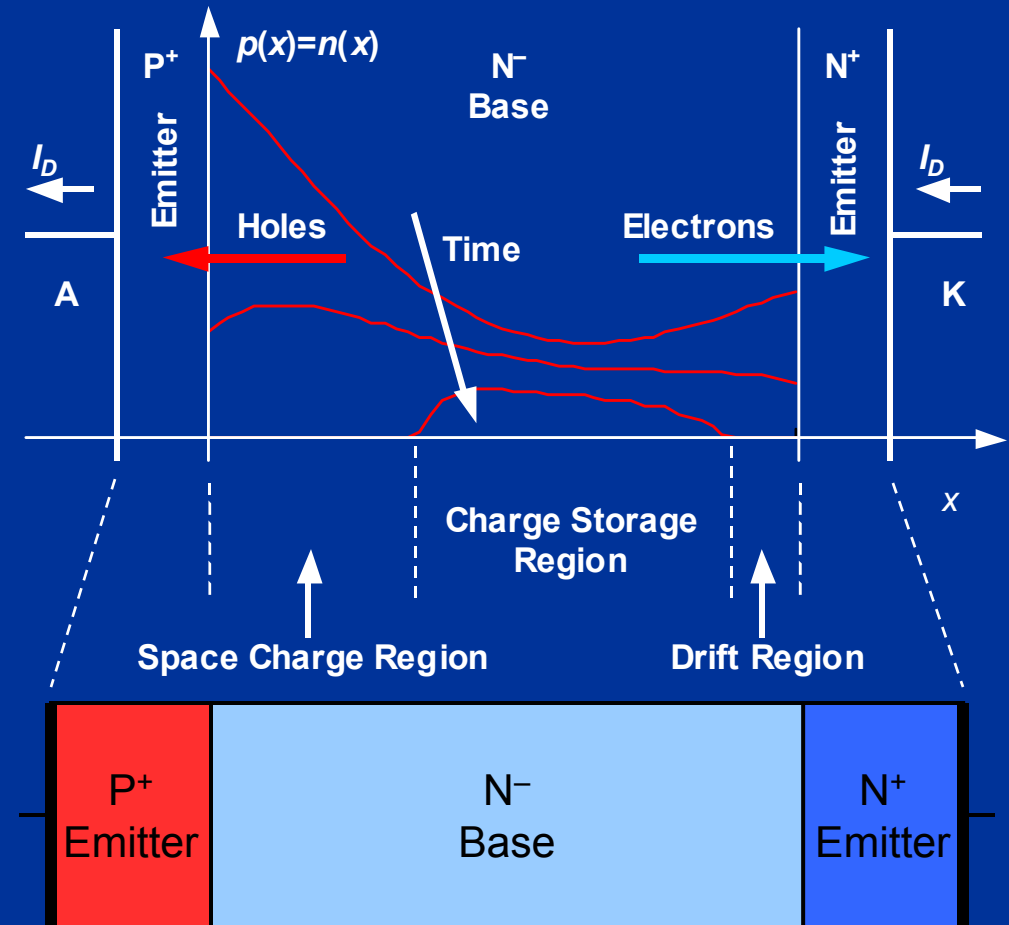
MCT

Storage of Charge Carriers

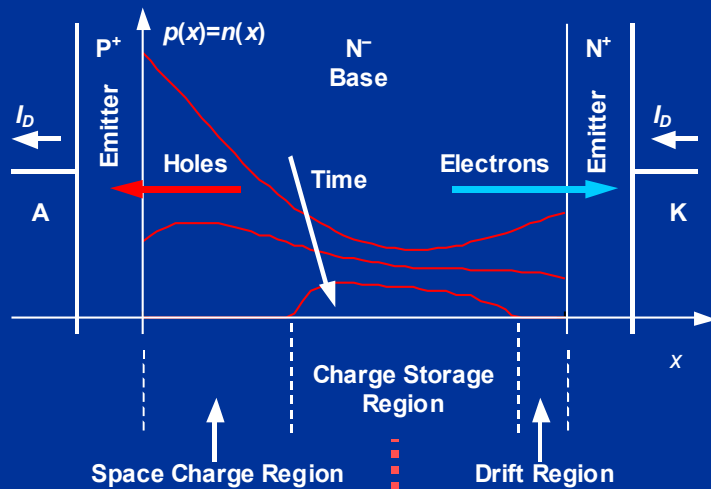


Developing Better Power Device Models

- Proper modeling of dynamic behavior is of great importance
- Stored charge sweeping & space charge development
- The PIN diode is a good starting point



PIN Diode Model



- One-dimension distributed model
 - Modular approach
 - Numerical solution of the diffusion equation
 - Voltage drop is returned
- Easy to include in the Berkeley SPICE code (as compared to Saber and VHDL-AMS)
- Correct results as compared to 2D simulations

$$\frac{dE(x)}{dx} = \frac{\rho_{ch}(x)}{\epsilon_{Si}} = \frac{q}{\epsilon_{Si}} (p_m(x) + N_D)$$

$$\frac{\partial^2 p(x,t)}{\partial x^2} - \frac{1}{D} \left(\frac{p(x,t)}{\tau} + \frac{\partial p(x,t)}{\partial t} \right) = 0$$



Straightforward Operation, Fast & Accurate Simulation

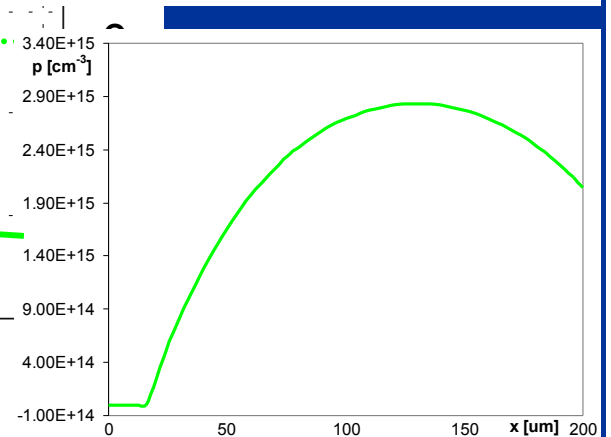
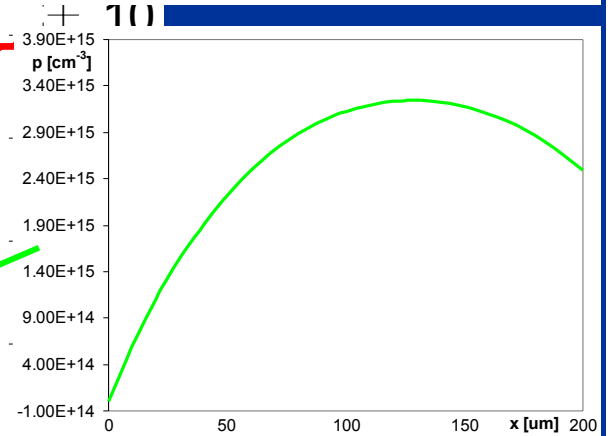
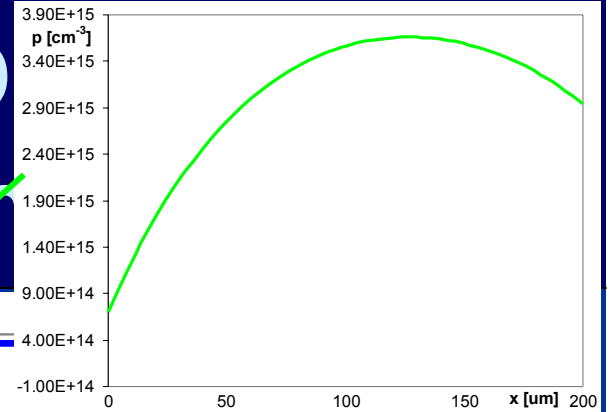
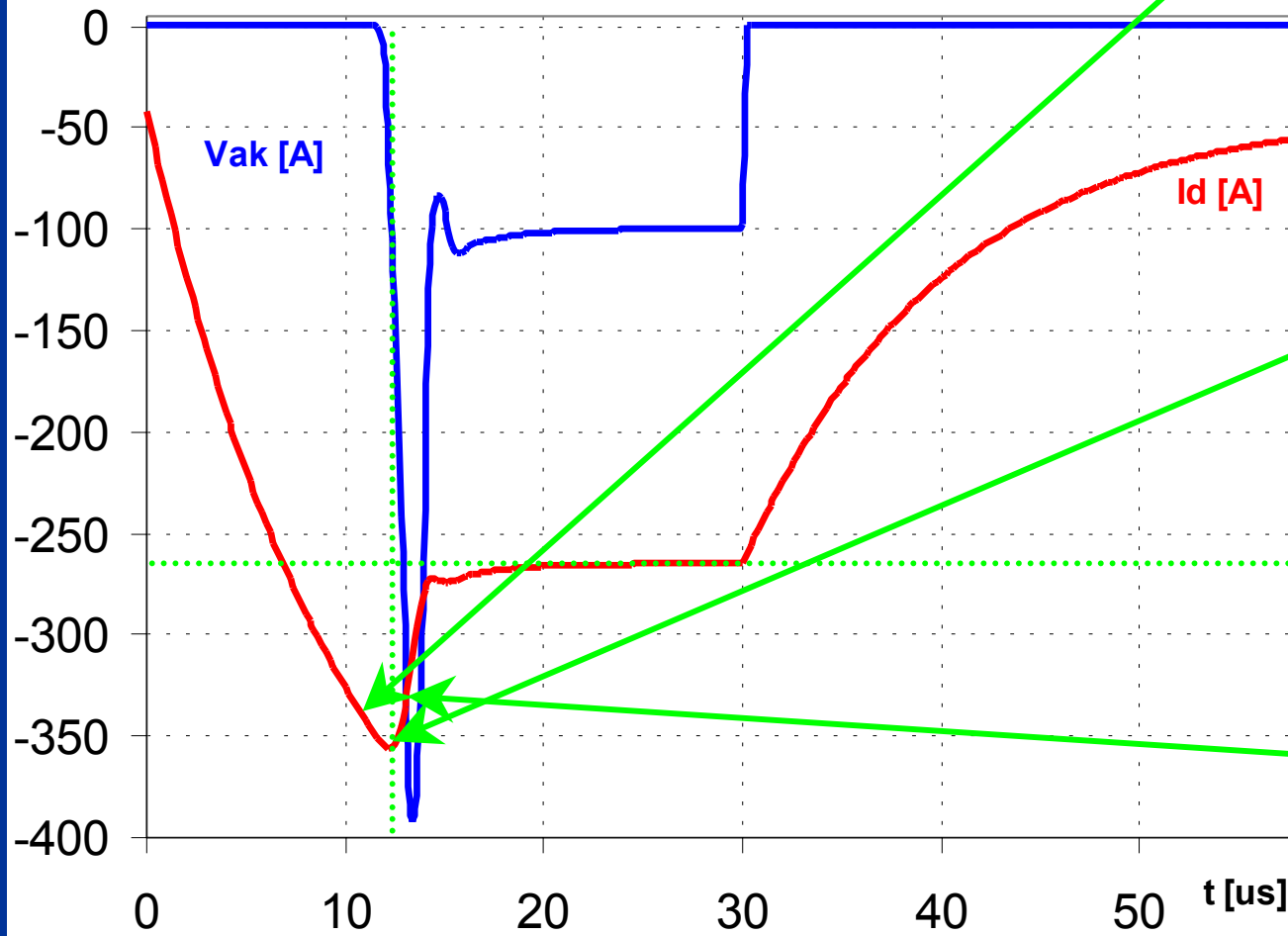
The screenshot displays the DMCS-SPICE website interface, which is used for circuit simulation. The interface is divided into several sections:

- SPICE Netlist Entry Point:** This section contains a text area for entering the SPICE netlist. The netlist includes parameters like `METHOD=Gear`, `ITL4=200`, and various component definitions for a diode model. It also includes analysis settings for a transient simulation.
- Circuit To Be Simulated:** A schematic diagram of a circuit is shown, featuring a square-wave voltage source, an inductor, a resistor, and a diode.
- Result Visualization:** A plot showing the simulation results. The Y-axis is labeled `VAK [V], -ID [A], VS [V]` and the X-axis is `Time [s]`. The plot displays three traces: `V(14)` (red), `I(VS)` (blue), and `V(11)` (green). The traces show a transient response over time.

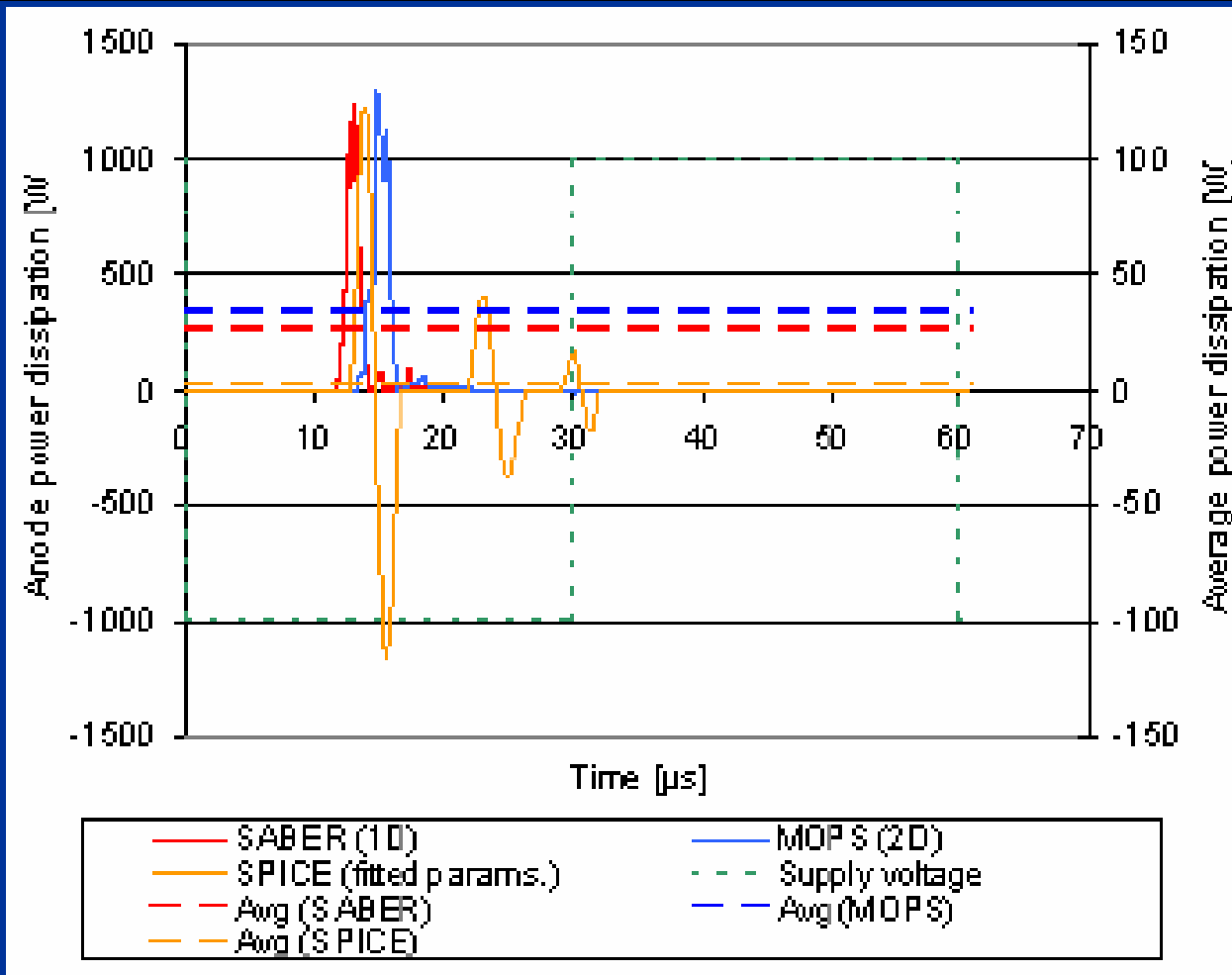
Yellow arrows indicate the flow of the simulation process: from the netlist entry point to the circuit diagram, and from the circuit diagram to the result visualization.



Straightforward Op Fast & Accurate Sim



Straightforward Operation, Fast & Accurate Simulation



Web page statistics

Oct 2002–Feb 2003

Domain	Postfix	Counts	
Poland	.pl	623	34.01%
...
Commercial USA	.com	254	13.86%
USA education	.edu	68	3.71%
Canada	.ca	47	2.57%
UK	.uk	36	1.97%
Australia	.au	27	1.47%
France	.fr	19	1.04%
Belgium	.be	16	0.87%
USA military	.mil	10	0.55%
Japan	.jp	10	0.55%
...

**Total visiting
1907 times**

<http://spice.dmcs.p.lodz.pl>
<http://www.dmcs.p.lodz.pl/dmcs-spice>

