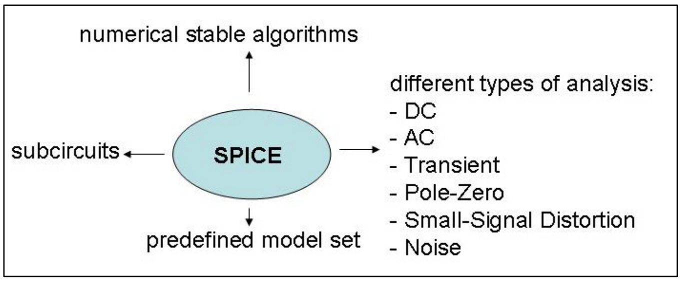
Tutorial: Simulation of Electronic Circuits Using the MSL SPICE3 Package

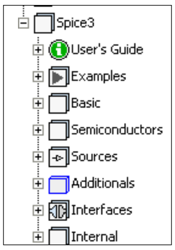
The general-purpose, open source SPICE (**S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis) simulator of the University of Berkeley, California, especially its models inspired to develop a corresponding Modelica library. At Fraunhofer IIS EAS Dresden important semiconductor models were directly extracted from the SPICE simulator to form the actual Modelica.Electrical.Spice3 library. The tutorial introduces into the usage of that library, which is done by three steps. After visiting the tutorial one should be able to work with the Modelica.Electrical.Spice3 library as well as one should know possible problems to be solved in future.

STEP ONE: The Berkeley SPICE3 Simulator

The SPICE3 simulator is presented, especially its history, some simulation algorithms, and why its performance is such high. Using an example **netlist** the simulation flow is explained. In detail all models are presented with special emphasis of semiconductor devices as well as the various kinds of analysis. Subcircuits and some options to control both models and analysis are presented.



STEP TWO: The Actual Modelica.Electrical.Spice3 Library



The basic idea is presented on structuring this library. All models available are presented within a simple example circuit including principles of modelling. In more detail semiconductor models are explained as well as aspects of their application, especially model cards. The diode model is explained down to its SPICE sources. Using some circuits of STEP ONE the results are compared to SPICE results. A special point is the actual handling of initial values. An example gives the idea of handling subcircuits. Finally, further circuits including some numerical critical circuits are presented to give a feeling for actual possibilities and limits.

STEP THREE: The Future Spice3 Library

The actual state of the future development is presented. Further models which will be added to the standard library, first of all the MOSFET level 2 transistor, are demonstrated. Otherwise, a netlist translator is under development which is able to transform given SPICE netlists to Modelica models. Its usage is shown. An overview is given on future development of the Spice3 Library as well of open problems.

Authors: Kristin Majetta, Sandra Böhme, Christoph Clauß, Fraunhofer IIS EAS Dresden
This work was supported by the European ITEA2 projects EUROSYSLIB, and MODELISAR.

Fraunhofer IIS

ITEA 2 INFORMATION TECHNOLOGY FOR EUROPEAN ADVANCEMENT