50 Years of SPICE and Hundred-Billion Transistors per Chip Later

This talk offers a historical view of the advancement of algorithms and modeling techniques applied in the circuit simulator SPICE over the five decades of its existence. In its many commercial implementations SPICE is currently used for every single transistor-level circuit design.

First, the emergence of SPICE is described from the initial research of a graduate-student project to an industry standard today, more than five decades and one billion transistors per chip later. The two driving forces, progress in semiconductor technology and compute power, the former needing a design capability, and the latter enabling the simulator's development, are described on ahistorical background.

Second, the main reasons for the success of SPICE will be discussed highlighting its two key components, solution algorithms and semiconductor device models. While in the early days of SPICE most research was devoted to algorithms, semiconductor device modeling has been in the limelight for the last two decades. The relentless effort in device modeling was brought about by the acceleration of the pace in device scaling.

Third, new functionality, applications and performance over the last two decades will be addressed. New types of analyses, behavioral modeling languages, more accurate semiconductor device models in step with technology progress, solution-algorithm tuning along with parallelization, and the extension of SPICE to the design of quantum systems will be highlighted.

Based on the history, success and impact of SPICE, the presentation will conclude on the importance of the research in universities and research laboratories being placed in the public domain as a way to train new generations of integrated-circuit designers using open-source software to benefit the advancement of technology and science.