Is It Time for SPICE4?

Laurence W. Nagel
Omega Enterprises
Randolph, NJ

One Perspective on SPICE

Q: If you could go back over your career and change something within your field, what would you change?

Pease: "I'd shoot the guys who were going to invent SPICE."

Robert A. Pease, ad insert from EDN "Movers and Shakers" 2003 supplement.

The Early Origins of SPICE

- SPICE began as an innovative class project under the direction of Ron Rohrer in the academic year 1969-1970
- The class topic was circuit synthesis but became a class on circuit simulation
- We learned by doing --- we wrote a simulator!
- The final judge of success was Don Pederson: if Don approved, we passed. Otherwise ...

So Who Were Those Guys?

- Ron Rohrer (Professor)
- Bob Berry
- Shi-Ping Fan
- Frank Jenkins
- Larry Nagel (ad-hoc project leader)
- Jesse Pipkin
- Steve Ratner
- Lynn Weber

A Perspective on Computing in 1970

- The computer at UC Berkeley at that time was a CDC 6400
- The input to the computer was punched cards
- The output of the computer was from the line printer
- The MIPS rate was comparable to an Intel 286
- The maximum available memory was 100,000 octal 60 bit words daytime and 140,000 octal at night

CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation)

- The simulation program developed in Ron Rohrer's class was named CANCER by my wife and became my Master's project with Ron Rohrer
- CANCER contained DC operating point analysis, small-signal AC analysis, transient analysis, noise analysis, and sensitivity analysis
- CANCER utilized sparse matrix techniques
- CANCER had built-in device models
- CANCER was about 6,000 lines of FORTRAN

SPICE (Simulation Program with Integrated Circuit Emphasis)

- CANCER was never released
- However, CANCER was renamed SPICE by me and SPICE was released into the public domain in 1971
- SPICE was immediately adopted by several universities and incorporated into circuit design courses
- As students graduated, they took SPICE with them into industry

Why SPICE Was Successful

- Public Domain
- DC, AC, Transient, Noise, and Sensitivity Analyses in the same program
- Built-in models for diodes, bipolar transistors, MOSFETs, and JFETs
- At the time, could handle fairly large circuits
- Written in fairly portable FORTRAN
- Student oriented and fairly bulletproof

SPICE Limitations

- According to student feedback, not very user friendly!
- Limited error checking
- DC Nonconvergence
- No Transient Timestep Control
- No dynamic memory allocation
- After all, this was a class project!

SPICE2

- Once SPICE was released, I began the development of SPICE2 as a part of my doctoral research with Don Pederson
- This work allowed me to study the algorithms and techniques of circuit simulation in depth
- This work involved a total rewrite of SPICE

SPICE2

- First released into the public domain in 1975
- Contained all features of SPICE1
- Data structures totally revamped to incorporate dynamic memory allocation
- Thorough upgrade of DC convergence and transient numerical integration algorithms
- About 8,000 lines of FORTRAN

More About SPICE2

- After I left UC Berkeley to work at Bell Labs, Ellis Cohen took command
- Ellis spent endless hours improving and debugging SPICE2
- Ellis is largely responsible for SPICE 2G6, which became the industry standard version of SPICE, and was released around 1978

University Use of SPICE2

- SPICE2 replaced SPICE at many universities and was adopted by many more universities
- At this point, SPICE simulations were an integral part of circuit design courses and even included in Gray & Meyer
- SPICE2 was used as a platform for research that spawned hundreds of research projects

Industrial Use of SPICE2

- Many industrial research centers adopted SPICE2 and developed proprietary versions of the program, including Bell Labs (ADVICE), Texas Instruments (TISPICE), Motorola (MCSPICE)
- Shawn and Kim Hailey formed Meta Software and modified a copy of SPICE 2E into the most successful version of a commercial SPICE known as HSPICE
- Numerous other “alphabet SPICEs” followed

Why SPICE2 was Successful

- Public domain
- Totally compatible with SPICE1
- Dynamic memory allocation
- Vastly improved DC convergence and transient timestep control
- The addition of many useful features such as subcircuits, transmission lines, etc.

SPICE2 Aftermath

- After SPICE2G6, work on SPICE at Berkeley waned considerably
- During that time, many universities did research on circuit simulation, often using SPICE as a platform, but no new versions of SPICE emerged
- In many ways, SPICE was considered a solved problem

SPICE3

- Not until 1983 (about five years after SPICE 2G6) did Tom Quarles do a Master's project at UC Berkeley where he converted SPICE2G6 into a RATFOR version that he named SPICE3
- During this work, several limitations of SPICE2 were observed, including the difficulty of adding new built-in models

SPICE3

- Tom Quarles continued his research and developed the next version of SPICE
- In 1989, SPICE3 was released into the public domain
- This later version of SPICE3 was coded in the C language and utilized the more sophisticated data structures of C
- SPICE3 contains about 135,000 lines of C code
- The latest version 3F5 was released in 1993

University Use of SPICE3

- Adopted by many universities who welcomed SPICE3 both as a more robust circuit simulator and as a computer program utilizing a modern language and its more sophisticated data structures
- Prompted many new research projects in circuit simulation, particularly more computer-science oriented projects

Commercial Use of SPICE3

- Microsim adapted a version of SPICE3 for the most popular of all SPICE programs --- PSPICE
- Many other companies utilized SPICE3 as a platform for additional “alphabet SPICE” programs

Why SPICE3 Was Successful

- Public Domain
- Easy to add device models, which has become the defining point of circuit simulators
- Modern data structures and the C language made new enhancements easier for researchers who didn't understand FORTRAN

The Aftermath of SPICE3

- Several circuit simulations challenges have arisen since SPICE3
  - RF Design requires sinusoidal steady-state simulation such as ADS, Libra, and HP MDS
  - Very large integrated circuits require “fast” SPICE simulators such as HSIM and TimeMill
  - With submicron transistors, even leakage current needs to be simulated

Industry's Response in the Aftermath of SPICE3

- Several new versions of SPICE have been developed
  - SPECTRE by Cadence does a fairly good job on RF design problems as well as standard SPICE applications
  - Eldo is a very good replacement for SPICE
  - ADS is fair at RF design problems
- None of these programs are public domain!

The Need for SPICE4

- The latest public domain version of SPICE is over 10 years old!
- There is no public domain circuit simulator that is useful in RF design problems (steady-state time-domain analysis)
- There is no public domain “fast” SPICE simulator capable of dealing with full chip simulation

Some Concluding Comments

- SPICE has been successful because it has been in the public domain, has gained widespread acceptance, and has spawned countless additional research projects in circuit simulation
- Attacking a new research project without an up-to-date platform like SPICE used to be is a burdensome project
- We need a new SPICE platform for circuit simulation research!
- Let's hope nobody tries to shoot the next generation of SPICE developers!

The Real Reason SPICE Is Still Around

- Two Visionaries in the CAD Industry
  - Ronald A. Rohrer
  - Donald O. Pederson