The Life of SPICE

Laurence W. Nagel

Omega Enterprises Consulting

Presented to Consultants Network of Silicon Valley
June 9, 2009

IC Technology Changes in the Last 40 Years

- Design rules in mils (1 mil = 25.4 micron)
- Masks from rubylith
- Chips with tens of transistors
- Wafer sizes of one inch
- Packages with a dozen pins

- Design rules in nanometers
- Masks from e-beam
- Chips with tens of millions of transistors
- Wafer sizes of ten or twelve inches
- Packages with hundreds of pins

SPICE (Simulation Program with Integrated Circuit Emphasis)

- First Released in 1971 and announced in 1973 at the Sixteenth Midwest Symposium on Circuit Theory
- Rapidly adopted by universities and industry in the early 1970's
- SPICE 2G6 became the de facto industry standard in the late '70s
- Most of the CAD tools from the '70s are extinct, so why is SPICE still around?

A Perspective on Computing in the '70s

- 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 on Intel 286
- The maximum available memory was 100,000 octal 60 bit words daytime and 140,000 octal at night

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 …
- I was appointed liaison to Don Pederson

So Who Were Those Guys?

- Ron Rohrer, Professor
- Bob Berry
- Shi-Ping Fan
- Frank Jenkins
- Larry Nagel (de facto group leader)
- Jesse Pipkin
- Steve Ratner
- Lynn Weber

Ron Rohrer's Class Project

- Larry Nagel wrote the netlist parser and the analysis core
- Bob Berry wrote a sparse matrix LU decomposition package
- Lynn Weber developed a noise analysis feature that utilized adjoint network techniques
- Everybody spent an insane number of hours getting the program to work
- Don Pederson heartily approved of the program, and we all passed the course!

CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation)

- We named the program CANCER (credit to my ex wife) and it became my Master's project with Ron Rohrer as my advisor
- CANCER was the first simulator to utilize sparse matrix techniques, so it could run circuits with hundreds of nodes
- Used Adjoint Circuit techniques to implement Sensitivity Analysis and Noise Analysis
- About 6000 lines of FORTRAN code

CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation)

- DC operating point analysis, small-signal AC analysis and transient analysis in one package
- Built-in models for diodes and bipolar transistors
- Modified Newton-Raphson iteration with heuristics that worked well with bipolar circuits
- Implicit integration techniques to reduce problems with the widely spread time constants of an IC

SPICE (Simulation Program with Integrated Circuit Emphasis)

- CANCER was never released, but was renamed SPICE and released into the public domain in 1971
- The Shichman-Hodges MOSFET model was added to assist Dave Hodges in teaching a MOSFET design course
  - The tiny capacitances and high impedances brought about two unique features in SPICE: Timestep reduced to zero and No Convergence in DC Analysis

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
- Heavy use of SPICE by students led to many improvements in robustness
- At the time, could handle fairly large circuits
- Written in fairly portable FORTRAN

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 SPICE
- 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 was released around 1978 and became the industry standard version of SPICE

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 SPICE
- 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

Why is SPICE Still Around?

- SPICE provides the capability to accurately simulate the DC, AC, and transient characteristics of a fairly large circuit at the device level
- SPICE is in the public domain
- It is taught at almost all universities
- It clearly is the industry standard

The Real Reason SPICE is Still Around

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