Y2K – A SPICE Odyssey

Laurence W. Nagel
Omega Enterprises

Presented at the 2000 FSA Modeling Workshop
October 12, 2000

Abstract

The integrated circuit industry thrives on constant change and is not particularly known for tradition. It is curious, then, that the SPICE circuit simulation program, in one form or another, has been around the industry for almost thirty years. That means that many engineers entering this booming business today weren't even born when I released the first version of SPICE! In this talk, I will chart the journey of SPICE, starting as a teaching program at the University of California, Berkeley, and spreading into industry, launching a cottage industry of software houses writing and supporting "alphabet SPICES." I also will give credit to all of the early principals in this journey, and share some of my more amusing experiences during the journey. Nobody can say for sure, but I will offer my opinions on how this particular program has evolved in thirty years and yet stayed pretty much the same. I can think of no other computer program that can make that claim.

Y2K – A SPICE Odyssey

Laurence W. Nagel
Omega Enterprises

Presented at the 2000 FSA Modeling Workshop
October 12, 2000

I can think of no industry that has undergone as many rapid evolutionary changes as the integrated circuit has. Think of what has happened in only thirty years. Design rules shrunk from tens of mils to tenths of microns. Masks that used to be generated by cutting rubylith and photographic reduction now are generated by an electronically controlled beam of electrons. Chips that used to contain tens of transistors now contain tens of millions of transistors. And wafers that used to measure one inch in diameter now are eight or ten inches in diameter. Packages that used to have a dozen pins now have hundreds. The list goes on and on. And this in only three decades, less than half a human lifetime!

As the materials experts and processing specialists and semiconductor equipment makers have evolved the technology, the designers also have constantly evolved the methods we use to design integrated circuits and the software developers have generated generation after generation of new CAD tools to support the evolving design flows. Much of the tedium of designing an IC now is done by computers. Without the computers that are manufactured today, we would still be designing the circuits we designed 30 years ago. And, without the integrated circuits that we design today, we would still be stuck with the computers of 30 years ago. The computer industry and the IC industry have a very symbiotic relationship.

In this constant evolution of design methodology and CAD tools, most of the

CAD tools that were around 30 years ago, or even 20 years ago, have become extinct. There is one curious exception: SPICE, which is a circuit simulation program that predicts the electrical characteristics of an integrated circuit and was developed within the EECS Department at the University of California, Berkeley. In fact, the name SPICE is an acronym for "Simulation Program with Integrated Circuit Emphasis."

SPICE was announced to the world twenty-seven years ago in Waterloo, Canada at the Sixteenth Midwest Symposium on Circuit Theory on April 12, 1973. None other than Professor Donald O. Pederson of the University of California, Berkeley, presented the paper. I don't think anyone had a clue of the impact of that paper or the computer program it described would have on the integrated circuit industry.

While we all had modest expectations for SPICE, what happened was truly phenomenal. Within a few years, SPICE had achieved acceptance at almost all electrical engineering schools, and a cottage industry sprouted to supply SPICE derivatives to the rapidly expanding integrated circuit industry. To this day, almost thirty years later, students at most electrical engineering schools learn how to use SPICE or some derivative. The students today, and many people in this room who use SPICE, wern't even born when I released the first version of SPICE!

When people ask me why SPICE became so widely used, or what did I do to make SPICE so widely used, the honest answer is "I don't know." But I can speculate. The biggest reason, directly attributable to Don Pederson, is that SPICE was developed at a public university and was public domain from the beginning. Because it was developed primarily as a teaching tool to provide students insight into integrated circuit performance, many engineering colleges throughout the world rapidly embraced SPICE. When students entered the integrated circuit industry they would write to Berkeley to obtain a copy of SPICE to use on their job. And so the copies of SPICE proliferated and the usage of SPICE grew.

SPICE was simply the latest of circuit simulation programs to be developed at UC Berkeley and it followed in the steps of BIAS, SLIC, TIME, and CANCER, as well as numerous other lesser known programs. In fact, SPICE was largely a derivative of the CANCER program announced two years earlier in a paper presented by Professor Ronald A. Rohrer at the 1971 ISSCC, back when ISSCC was held in Philadelphia.

CANCER, an acronym for "Computer Analysis of Nonlinear Circuits, Excluding Radiation," was developed in an era when many circuit simulation programs were developed by large corporations with government contracts and were required to have a simulation capability that could evaluate the radiation hardness of a circuit. The name CANCER was a brash statement that this program never would simulate radiation and was not funded by the defense industry. CANCER was developed at Berkeley in the sixties, remember!

Actually, CANCER was a derivative of a program that was the class project of a series of courses taught by Ron Rohrer. At the time, I fully anticipated doing my doctoral research on some aspect of analog circuit design, since that was my first love in electrical engineering. My intent was to take the course to broaden myself a little bit. I had no idea!

The course was about circuit simulation, and Ron, always an innovative teacher, reasoned that we would learn a lot more by doing than by listening to Ron lecture. The rule was that as long as Don Pederson approved of the program, we all passed. If Don didn't like the program, we were in deep trouble.

My classmates in this unique learning experience were Bob Berry, Shi-Ping Fan, Frank Jenkins, Joe Pipkin, Steve Ratner, and Lynn Weber. I was put in charge of showing the program to Don Pederson and gaining his approval, I guess because I had done an undergraduate IC design project for Don and the class figured I was the best chance for a passing grade.

So, much of the secret of SPICE was in this class project led by Ron Rohrer. Bob Berry developed a sparse matrix solver that allowed us to simulate circuits orders of magnitude larger than previous programs could handle. The use of implicit integration algorithms provided a much more robust transient analysis capability. And the program had built-in models for semiconductor devices, so the user need only provide a set of model parameters as opposed to providing FORTRAN routines to model the devices. And we introduced adjoint solution methods to allow rapid sensitivity analysis and noise analysis.

When the course was over, Don Pederson gave his hearty approval, we all passed, and CANCER became my Master's project. The program was used heavily in undergraduate and graduate courses at Berkeley, and that gave me plenty of opportunity to improve the robustness of the algorithms for circuit simulation. Ron Rohrer and I spent many hours brainstorming and improving the program. Ron left Berkeley to work in industry about the time I finished my Master's project. Although CANCER was a success, there was still much to do, and I began my doctoral studies with Don Pederson as my Thesis advisor.

The name CANCER was not the most popular in the industry, mainly because of the medical implications, and my first job was to find a new name. I don't know how long I spent thinking up the name SPICE, and in retrospect it is amazing what graduate students spend time on. Anyway, I figured this whole project wouldn't go anywhere without a catchy name, and eventually SPICE was born. Some copies of SPICE were distributed in 1971, and that's why I consider next year to be the 30th anniversary of SPICE.

At this point, SPICE only had models for bipolar transistors because that's all we designed at Berkeley. That changed when Professor David A. Hodges left Bell Labs to join the faculty at UC Berkeley. He brought with him a paper on MOSFET circuit simulation, and the Shichman-Hodges model was put into SPICE in a weekend so that Dave could teach a course on MOSFET circuit design. Software development has become so sophisticated now that putting a model into a simulator takes months, but back then it only took a weekend.

The introduction of MOSFET models into SPICE introduced two phenomena that are still present in SPICE simulators today: DC convergence problems and too small timesteps in transient analysis. MOSFETs are difficult devices with outrageous conductance swings and tiny tiny capacitors. True, bipolar circuits sometimes have convergence problems, but MOSFETs really show up the problems.

The user interface never was particularly sophisticated, on purpose. The most important consideration was that the program had to be bulletproof. We ran an entire class of jobs in one batch run at night, and if one job crashed the program all the jobs behind it in the "input deck" had to be rerun. Turnaround times were rather long in those days, and the TA could spend the entire night running the SPICE jobs. I remember my friend Dick Dowell trying to explain to one student that TEMP=0 meant absolute zero and absolute disaster. The next night, the same job came back with TEMP=1E-10! Needless to say, we converted to Celsius!

Language is important too, especially in a place as diverse as a university. When transient analysis failed, SPICE used to issue an error message "TIMESTEP REDUCED TO ZILCH." That changed one night when an entire class with a project due the next day descended on me, one at a time, demanding to know the meaning of zilch. Amazing how graduate students spend their time.

The rigor of doctoral research enabled me to really pursue the algorithms of circuit simulation. It also allowed me the privilege of having Ellis Cohen as a roommate. Ellis is the unspoken hero of SPICE. Ellis was much more computer oriented, and understood data structures, memory management, and program architecture much better than I did.

At the time our computer resources consisted of a CDC 6400 mainframe that probably had the compute capability of an Intel 286. Our memory was limited to 100,000 octal words during the day, but at night we could have 140,000 octal! For those of you uncomfortable with octal, that's 256 Kbytes during the day and 384 Kbytes at night. So simulating a circuit of reasonable size was a little like fitting my size 11 foot into a baby's slipper. The best we could do was about 50 nodes and 25 bipolar transistors. Today, SPICE simulators chew up circuits a thousand times larger!

My doctoral research came to a conclusion, I ground out my Thesis in a painfully slow fashion, and I was off to Bell Laboratories to start my professional career. But the work on SPICE was hardly over. Many copies of the program were proliferating and many comments, bugs, suggestions, complaints, and huzzahs were arriving daily at Berkeley. That unsung hero, Ellis Cohen, took the SPICE I left and brought it to the level of professional software and the industry standard --- SPICE 2G6.

Of course, many versions of SPICE would follow. I wish I knew how many dissertations and Master's projects were spawned by SPICE. And, of course, we are all familiar with the cottage industry that emerged to provide alphabet SPICE for the industry. It's a several hundred million dollar industry now.

The program has evolved enormously over its thirty year lifespan. The most noticeable changes to SPICE were a graphical user interface, or GUI, and the addition of a schematic capture package. Neither of these improvements were universally appreciated by certain members of the user community, including the author of the program. More importantly, the algorithms in today’s versions of SPICE are much more robust than the algorithms of thirty years ago, and there are many new analysis capabilities and output post-processing features. The semiconductor device models in modern-day versions of SPICE are far superior to the models in the original SPICE. And of course, the staggering increase in computational power over the last thirty years has made SPICE capable of simulating much larger circuits. Still, in many ways today's SPICE is awfully reminiscent of the SPICE of old.

There are, I think, two reasons for this. First, SPICE addresses a basic need in the IC design flow: the need to simulate the electrical characteristics of a fairly small circuit at the transistor level with a high degree of accuracy. This need is pretty much the same today as it was thirty years ago. And the need is universal in all areas of integrated circuit design. And SPICE has and continues to address this need very well. So, if it isn't broke, nobody's going to want to fix it.

Second, the evolution of ever smaller devices has not, as yet, posed any fundamental need for evolutionary change in SPICE. The evolution, to date, has been in the semiconductor device models. Witness how many MOSFET models are available in most commercial versions of SPICE!

What has created a need for evolution in simulators is the wireless revolution. When SPICE was developed, those of us engineers who understood how a radio worked were few and far between. And the exploding markets of the time were memories, microprocessors, DSP's, as well as analog circuits like modems, codecs, and data conversion circuits. Now, RF Integrated Circuits are assuming a larger role in our industry and engineers and the tools are evolving. The need for robust steady-state simulation, the need for accurate simulation of intermodulation distortion and oscillator phase noise, the need to incorporate thermal effects in RF power amplifiers, have all caused a lot of activity in the development of RF simulators, which don't resemble the SPICE of old at all. So there is a lot of new work to be done, but none of it will reduce the need for SPICE.

I'd like to conclude with a story. After I graduated from Berkeley and began my career at Bell Labs, I continued my work on circuit simulators developing the ADVICE simulator. I also worked in modeling, characterization, and process and device simulation. I left the CAD business about sixteen years ago for my first love, integrated circuit design. But, as luck would have it, I was pulled back into the CAD business to participate in the development of yet another simulator, Celerity. Shortly thereafter, I left Bell Laboratories and started work at Anadigics managing a group that supported simulation tools for RF design. Then my big break came in 1998 when I founded my own consulting company, Omega Enterprises. I expected to do work primarily in the field of CAD, but to my delight found many opportunities in the field of my first love, analog circuit design. When I landed my first design project, I found myself in a position I'd never confronted before --- I needed a simulator! And so I found myself on the telephone to OrCAD, talking to a salesperson who probably wasn't even born when I was working on SPICE, and negotiating exactly how much I was going to pay for this simulator I had written almost thirty years ago. I couldn't resist, so I finally said "I know you don't know who I am, but I'm the person who wrote this program at Berkeley." He said "You are?" I said, "Look at the first reference in your PSPICE manual" Of course, it was my dissertation! He said, "You're L. W. Nagel?" I said, "Yep" So the salesperson delivers the *coup de gras* and says, "No kidding, gee, I'll throw in the Optimization Package for free!"