Is SPICE Still An Educational Tool?

Laurence W. Nagel Omega Enterprises Consulting

Presented at MOS-AK Workshop Graz, Austria September 18, 2015

Simulation Program with Integrated Circuit Emphasis (SPICE)

IEEE MILESTONE IN ELECTRICAL ENGINEERING AND COMPUTING

SPICE (Simulation Program with Integrated Circuit Emphasis), 1969–1970

SPICE (Simulation Program with Integrated Circuit Emphasis) was created at UC Berkeley as a class project in 1969-1970. It evolved to become the worldwide standard integrated circuit simulator. SPICE has been used to train many students in the intricacies of circuit simulation. SPICE and its descendants have become essential tools employed by virtually all integrated circuit designers.

February 2011

333

Simulation Program with Integrated Circuit Emphasis (SPICE)

- SPICE was first released almost 45 years ago!!!
- The primary objective for developing SPICE was to provide an educational tool to help students gain insight into electronic circuits
- SPICE has evolved to handle many types of circuits and to accommodate very large circuits
- But have we lost touch with the original intent of SPICE as an educational tool?

SPICE As A Teaching Tool

- I had the privilege of teaching a sophomore class "Introduction to Electronic Circuits" this Spring Semester at Chabot College, a Community College in Hayward, CA
- The goal was to introduce students to circuits, starting with Kirchhoff's Laws and working up to amplifiers
- I also wanted the students to become excited about electronic circuits so they would be motivated them to continue
- I felt that no course in Electronic Circuits would be complete without learning how to use SPICE

SPICE and Device Models

- The educational purpose SPICE is to impart an understanding and intuition of how an electronic circuit operates
- The educational purpose of a device model in SPICE is to impart an understanding and intuition of how a device operates
- The device model also will impart an understanding of how changing parameters in a device will change the operation of the circuit







Copyright ©2014 Pearson Education, publishing as Prentice Hall



18 Sep 2015



Copyright ©2014 Pearson Education, publishing as Prentice Hall

Circuit Simulator Requirements for Education

- Should provide insight into how circuits work
- Should have simple, easy to understand models for semiconductor devices
- Should be easy to use, even for sophomores
- Has to be free
- Should be Open Source in the public domain

UC Berkeley Circuit Simulators (All Done Under the Watchful Eye of Donald O. Pederson)

- BIAS was developed by Bill Howard (and enhanced by Bill McCalla) to understand the DC behavior of bias circuits such as band-gap regulators
- SLIC was developed by Bill McCalla to understand the placement of poles and zeroes in integrated amplifiers
- SINC was developed by S. P. Fan to help students understand the transient behavior of digital circuits
- SPICE was developed by Larry Nagel to help students understand all kinds of circuits in all kinds of domains

UC Berkeley Circuit Simulators (All Done Under the Watchful Eye of Donald O. Pederson)

- Built-in models for the semiconductor devices in use at the time (BJTs and diodes)
 - My favorite quote of DOP: "a simulator is a breadboard with a beta knob on each transistor"
- Used in undergraduate electronics circuits courses (SPICE found its way into Gray and Meyer)
- Fairly bullet-proof because of heavy use
- But not universally beloved by the undergraduate student community

SPICE Circa 1974

- Input was a netlist describing the schematic in textual form (in punched card format)
- Output was line-printer plots of whatever circuit voltages and currents the student selected
- Not only DC analysis, AC small-signal analysis, transient analysis, but also noise analysis, sensitivity analysis, and distortion analysis
- Built-in models for BJTs, diodes, JFETs, and MOSFETs (thanks to Dave Hodges)

SPICE Netlist - Circa 1974

```
DIFFPAIR CKT - SIMPLE DIFFERENTIAL PAIR
VIN 1 0 SIN(0 0.1 5MEG 5NS) AC 1
VCC 8 0 12
VEE 9 0 -12
Q1 4 2 6 ONL
Q2 5 3 6 QNL
RS1 1 2 1K
RS2 3 0 1K
RC1 4 8 10K
RC2 5 8 10K
Q3 6 7 9 QNL
04 7 7 9 ONL
RBIAS 7 8 20K
.MODEL QNL NPN (BF=80 RB=100 CCS=2PF TF=0.3NS TR=6NS CJE=3PF
+
   CJC=2PF VA=50)
. END
```

SPICE Output - Circa 1974

	V5							
	~	Cr+3000.	0+3000*+	2	.330E+JO	100.6	02+30	1.0075
	00+3641-5							
	5-2156-00		 					
5-5	5.5247400		 					
	00+3+5C-4							
57.0	0.475.00							
80- K	C0-15511.1				• 			
	7.1716-32	 	! 					
10-30	100-11-1 100-11-1		 			· · ·		
10-10	7.2926-50							
10-30	0.1955.00				•			
10-16	00-3671-4-2		! 					
10-10	00-31-5 00-31-5 1-5			•••				
575	00-111-1			· • .				
10-10	10-11-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1	 	• • •					¦
10-31	00-2231-4					1		1
	00+2122-4			•••		•••		•••
10-10	5.2535-00 5.2535-00 5.2535-00			••	•••	•••		•••
10-10	5-111-15 5-111-00 5-111-00			•				
	CC+3+2+++							
10-20	5.44 34.00 5.74 34.00 5.63 35.00	••••	•••					
36-07 08-07	6.7275-00 5.275-00				. • . 			··.
10-11	1.7715-00				· • •			···
10-51	6-5022-07 6-1255-00 6-1255-00		••••		• • • •			
10-10-W	0043640°S			•				
	00-3-12-3			• ••				
10-30	4 - 55 - 5 - 60			•.				
555	07-31-5-5		 					
19-10	00-11-1-0							
10-10	00+3201+4 00+3201+4							
10-10	00-5415-4			 .				
10-10-10-10-10-10-10-10-10-10-10-10-10-1	00-1127.5			•				
10-30	5. 44 4.00							• • • •
111	6.2012 00 4.42 12 00				•			'
13-12	CD							

Why SPICE Was Successful

- It contained simple built-in models for BJTs, diodes, JFETs, and MOSFETs (thanks to Dave Hodges)
- It was fairly easy to use (some classmates may disagree)
- It was readily available and in the public domain
- At 8,000 lines of FORTRAN, students could make modifications to fit special needs
- Students could take SPICE with them to their new job

The Evolution of SPICE

- Began as a tool to aid in the understanding and design of analog circuits (uA 741)
- Invaluable tool for memory design
- As digital circuits entered the fray, became the tool of choice to characterize digital cell libraries (now the largest CPU consumer)
- As RF Integrated Circuits became feasible, new algorithms were added for sinusoidal steadystate analysis to assist RF design

The uA741 Op Amp



DRAMS



Domino Logic



18 Sep 2015

The RF Transceiver



18 Sep 2015

Moore's Law

"The complexity for minimum component costs has increased at a rate of roughly a factor of two per year ... Certainly over the short term this rate can be expected to continue, if not to increase. Over the longer term, the rate of increase is a bit more uncertain, although there is no reason to believe it will not remain nearly constant for at least 10 years. That means by 1975, the number of components per integrated circuit for minimum cost will be 65,000. I believe that such a large circuit can be built on a single wafer."

Gordon Moore, *Electronics Magazine*, 1965

Moore's Law



The Evolution of SPICE

As a part of the evolution, many things happened to SPICE

- The built-in models became numerous, complex, and overwhelming
- SPICE became oriented towards huge circuits (millions of lines of netlist)
- SPICE grew in size from about 8,000 lines of FORTRAN to 135,000 lines of C

SPICE and Device Models

- The educational purpose SPICE is to impart an understanding and intuition of how an electronic circuit operates
- The educational purpose of a device model in SPICE is to impart an understanding and intuition of how a device operates
- The device model also will impart an understanding of how changing parameters in a device will change the operation of the circuit

Educational Circuit Simulator Candidates

- PSPICE (Free Student Version)
- NGSpice (Open Source, Windows version)
- Berkeley SPICE
- LTSPICE
- Multisim (part of National Instruments Elvis)
- Cadence Tools (Virtuoso + SPECTRE)
- Tanner Tools (Now Part of Mentor Graphics)

Educational Circuit Simulator Criteria for Choice

- Is it easy to install and maintain on a PC running Windows?
- Does it have schematic capture?
- Is it fairly easy to use?
- Are the built-in models simple and accessible?
- Will it work on the exemplary circuits
- Is it standalone (no model libraries, no PDKs, no extra stuff to learn)

Educational Circuit Simulators

	Easy to Install	Schematic Capture	Easy to Use	Accessible Models	Standalone Program
PSPICE	Yes	Yes	Yes	No	No
NGSPICE	Yes	No	Yes	Yes	Yes
Multisim	Yes	Yes	Yes	No	Yes
LTSPICE	Yes	Yes	Yes	No	No
Cadence Tools	No	Yes	No	No	No
Tanner Tools	No	Yes	No	Yes	Yes
Berkeley SPICE	No	No	No	Yes	Yes

Which Educational Circuit Simulator?

I chose PSPICE despite some deficiencies

- Students had little problem downloading and installing the student version of PSPICE
- Learning Schematic Capture caused a lot of groans
- PSPICE did well on Exemplary Circuits 1, 2, and 3
- Things did not go so well on Exemplary Circuits 4 and 5, which required models for a BJT and a MOSFET
- I am still working on a simple way of specifying a model in PSPICE. Why does this have to be so hard?

PSPICE Schematic - Circa 2015



18 Sep 2015

PSPICE Output - Circa 2015



A Word About Modeling (For Sophomores)

- SPICE models are unbelievably complex, especially for students who just learning semiconductor devices
- The model parameters in most SPICE models make sense only to the author of the model
- The simulator needs to have an easy way to display device characteristics so the student can vary parameters and see the result on the characteristics
- We should never teach students (even sophomores) that they should obtain models from some library and use them without understanding them

A Challenge to the Modeling Community

- Develop models for the BJT, diode, JFET, and MOSFET that are simple and easy to use
- The models should contain no more than seven or eight parameters and the parameter names should mean something
- The models do not have to be exact nor fit every device; they just have to give a feel for how the device works
- Model documentation should be two pages or less for each model

Conclusions

- PSPICE can be a excellent aid in teaching sophomores how circuits work
- There is a learning curve, but most students are up to the task
- Problems arise when exploring circuits requiring device models, which PSPICE assumes come from a library
- The models in PSPICE (and other simulators) are anything but simple

Thank You!!!