How SPICE Transformed Integrated Circuit Design

Laurence W. Nagel

Omega Enterprises Consulting

Presented at ISAT/DARPA Workshop

Naturally Expressed Rapid Design (NERD)

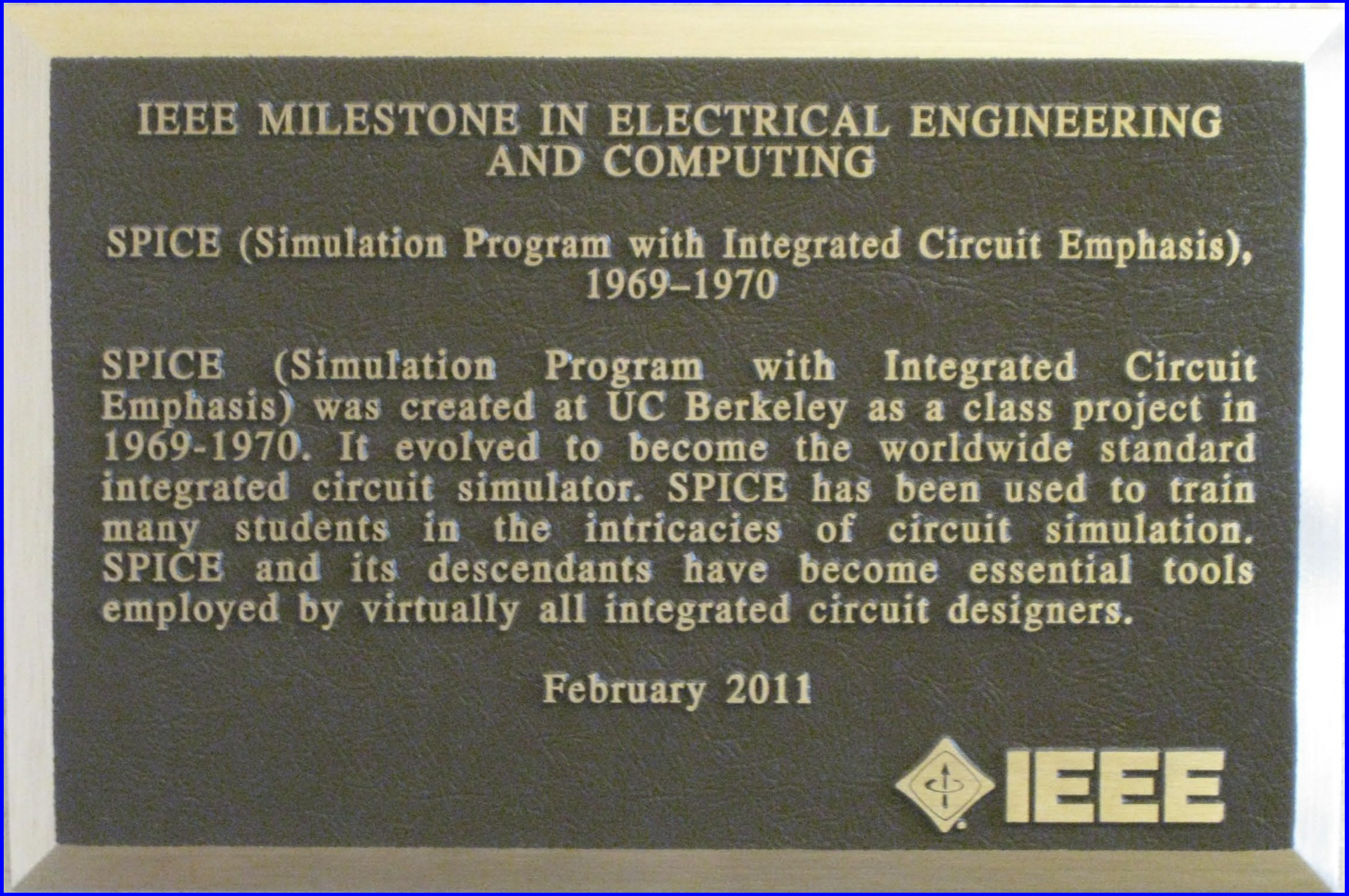
Warrenton, VA

August 14-15, 2014

Simulation Program with Integrated Circuit Emphasis (SPICE)

- SPICE was first released over 40 years ago!!!
- Virtually every EE student has to learn SPICE to learn how to design integrated circuits (and to graduate)
- SPICE is still around because it has evolved to remain a vital and useful tool in the design process
- What has driven SPICE evolution in the last 40 years?

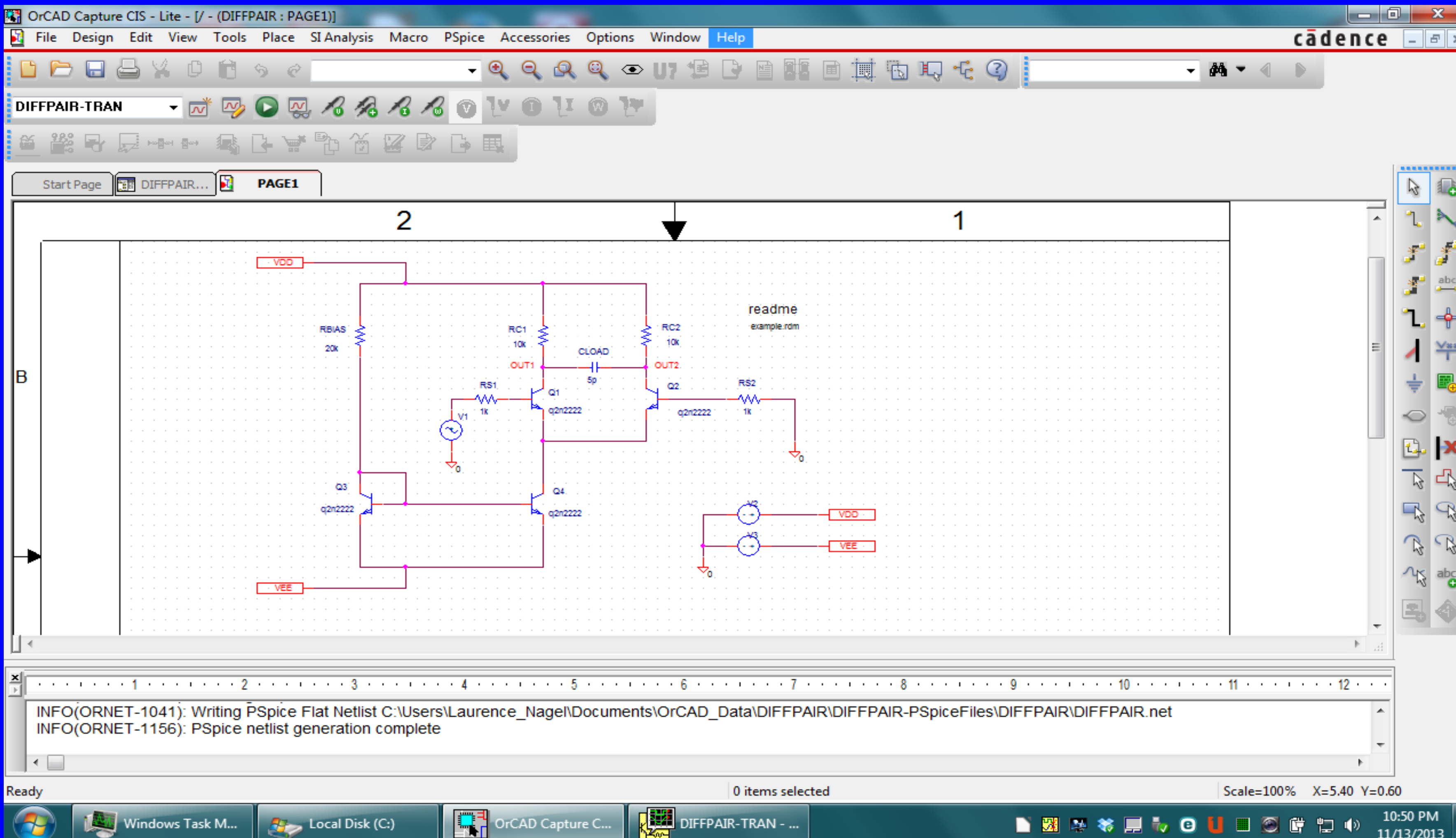
Simulation Program with Integrated Circuit Emphasis (SPICE)



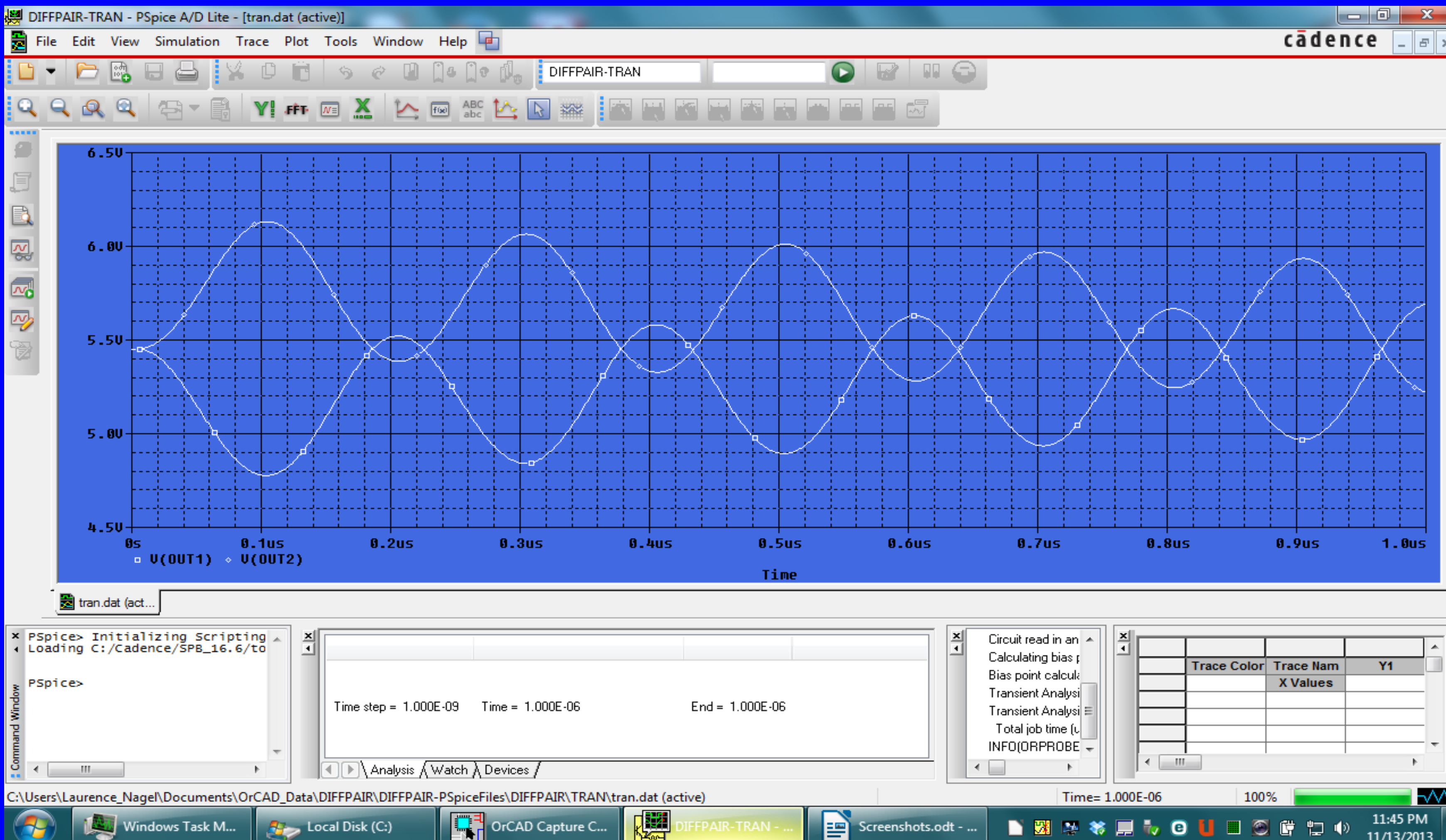
Simulation Program with Integrated Circuit Emphasis (SPICE)

- SPICE is a computer tool that allows an engineer to simulate a circuit
  - Predict how a circuit will work without building and testing the circuit
- The input is a circuit schematic, or a netlist describing the schematic in textual form
- The output is whatever circuit voltages and currents the engineer wants to know
- SPICE works for dc, ac and transient time-domain analysis

PSPICE Schematic



PSPICE Output Waveform



Advantages of Using SPICE

- Allows the student to learn how circuits work without having to build them
- Allows the engineer to verify that a circuit works properly without having to build it
- Allows the engineer to estimate circuit operation over process, voltage, and temperature (PVT)
- Allows the engineer to evaluate the sensitivity of the circuit to component variations
- Allows the engineer to evaluate design alternatives prior to building anything

Disadvantages of Using SPICE

- SPICE doesn't always “work”
- Student/engineer spends too much time playing with computers and not enough time thinking about circuits
- Student/engineer puts too much trust in SPICE and not enough trust in his or her thought process

“but the circuit must work --- SPICE said it would!”

(disillusioned undergraduate student)

Pre-SPICE Milestones

- 1906 - Lee De Forest Invents the Audion
- 1947 - Point Contact Transistor Invented
- 1959 - Planar Integrated Circuit Process Invented
- 1960 - MOS Transistor Invented
- 1963 - Complementary MOS Invented
- 1966 - ECAP Simulation Program Published
- 1966 - Bill Howard Writes BIAS
- 1971 - BIAS-3, SLIC, and CANCER First Published
- 1971 - SPICE First Released

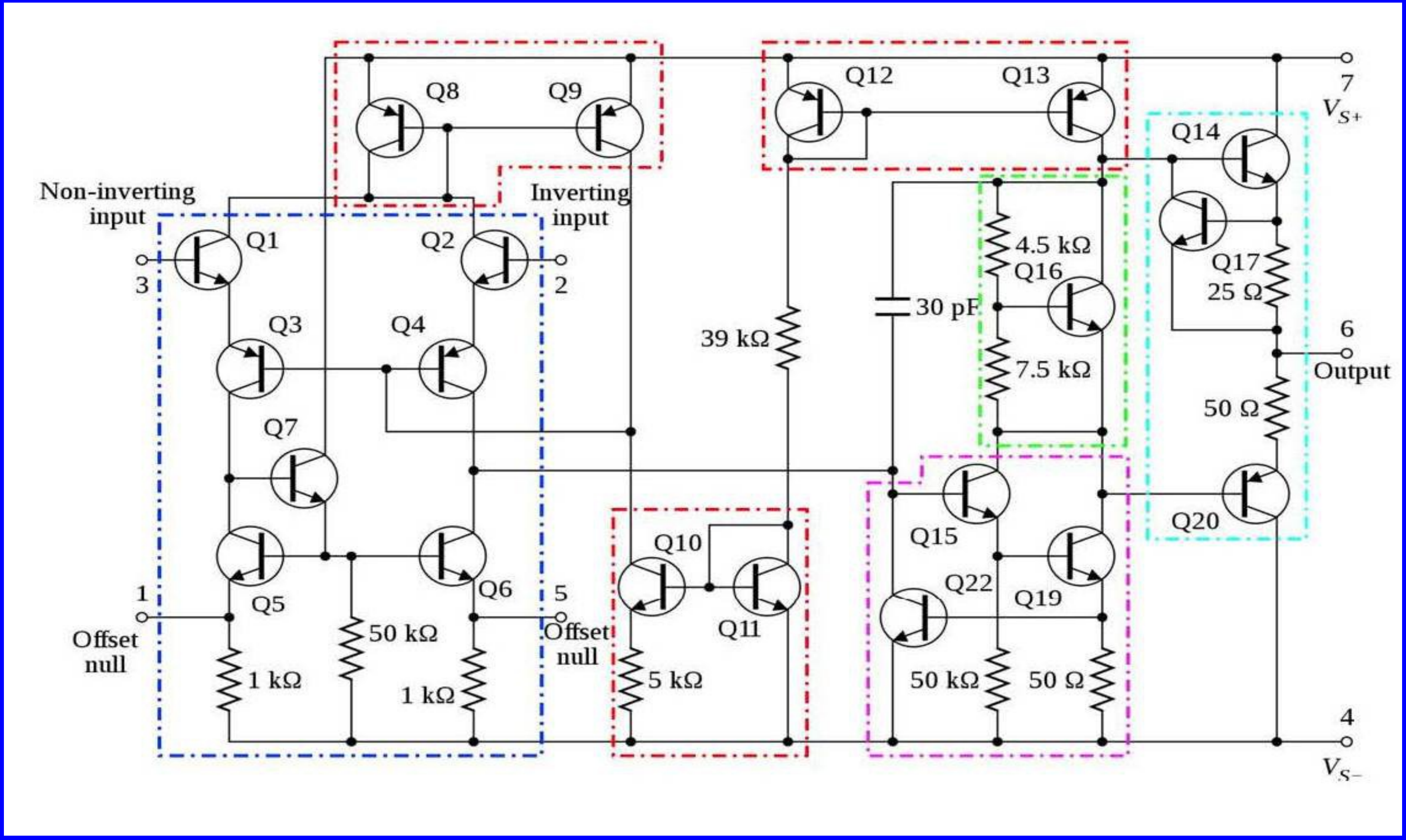
A Perspective on Computing in 1970

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

A Perspective on IC Design in 1970

- The uA 741 Op Amp was a classic design that illustrated the need to perform many different types of analysis
  - DC Operating Point Analysis (with sensitivity)
  - AC Small Signal Analysis
  - Transient Analysis
  - Noise Analysis
  - Distortion Analysis

The uA741 Op Amp (1968)



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

CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation)

- The simulation program developed in Ron Rohrer’s classes was named CANCER and became my Master’s project with Ron Rohrer
- DC operating point analysis, small-signal AC analysis and transient analysis in one package
- Built-in models for diodes and bipolar transistors
- CANCER was the first simulator to utilize sparse matrix techniques

CANCER (Computer Analysis of Nonlinear Circuits, Excluding Radiation)

- 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
- Use of Adjoint Circuit techniques to implement Sensitivity Analysis, Noise Analysis, and Distortion Analysis using Volterra Series
- About 6,000 lines of FORTRAN code

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
- SPICE was used in undergraduate EE courses at UC Berkeley as a teaching tool
- SPICE also was used by graduate students in their IC design research projects

Why Was SPICE 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

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 then passed the reigns on to Andrei Vladimirescu, who also made substantial improvements
- SPICE 2G6 was released in 1981 and became the industry standard

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

SPICE3

- In 1983 Tom Quarles did a Master's project at UC Berkeley where he converted SPICE2G6 into a RATFOR version that he named SPICE3
- In 1989, SPICE3 was released into the public domain
- This later version of SPICE3 then was coded into the C language to utilize 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

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 steady-state analysis to assist RF design

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 Reasons SPICE is Still Around

- Two Visionaries in the IC Industry
  - Ronald A. Rohrer
  - Donald O. Pederson
- A tremendous amount of effort on the part of a huge team of graduate students

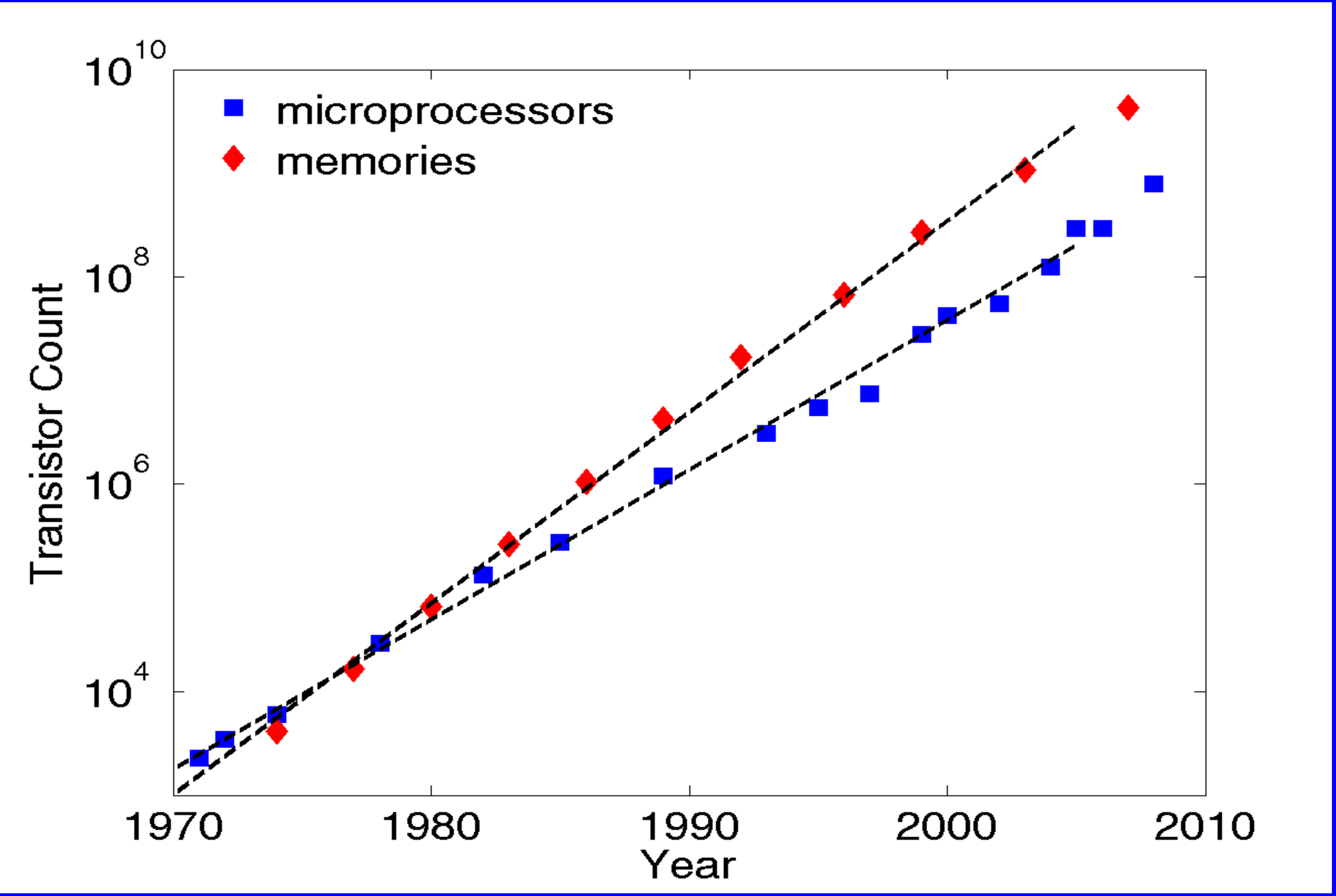
Thanks!

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



microprocessors
memories
Transistor Count
10^{10}
10^{8}
10^{6}
10^{4}
1970
1980
1990
2000
2010
Year

SPICE Version of Moore's Law

SPICE CPU = Timepoints

* (Newton Iterations / Timepoint)

* (CPU / Newton Iteration / Transistor)

* (Transistors)

SPICE Version of Moore's Law

- Timepoints increase by at least √2 every two years
- Newton Iterations / Timepoint is constant
- CPU / Newton Iteration / Transistor is the CPU required to evaluate a device model and is relatively constant.
- Transistors increase by at least √2 every two years

This is at least an N^2 Process!!!

SPICE Version of Moore's Law

- Fortunately, computer CPUs get faster $\sqrt{2}$ every two years

  Still ...

- SPICE CPU consumption doubles every four years!!!