SPICE In The Twenty-First Century

Laurence Nagel

Omega Enterprises Consulting

Presented at MOS-AK 2013 Workshop

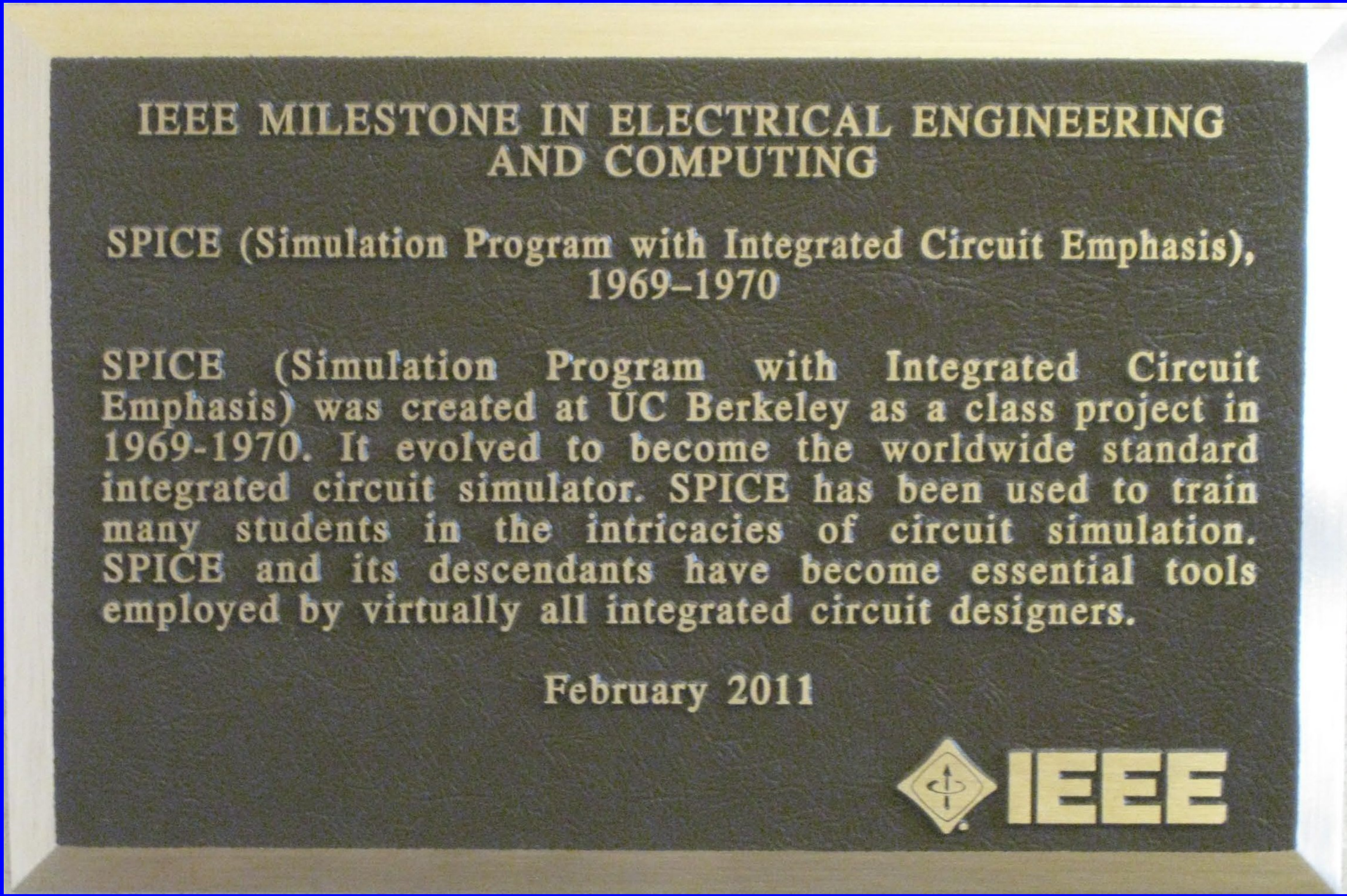
Bucharest, Romania

September 20, 2013

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
- How will the evolution of SPICE continue in the next 40 years???

Simulation Program with Integrated Circuit Emphasis (SPICE)



The SPICE Era

1973 - SPICE1

1975 - SPICE2

1981 - HSPICE

1984 - PSPICE

1984 - Eldo

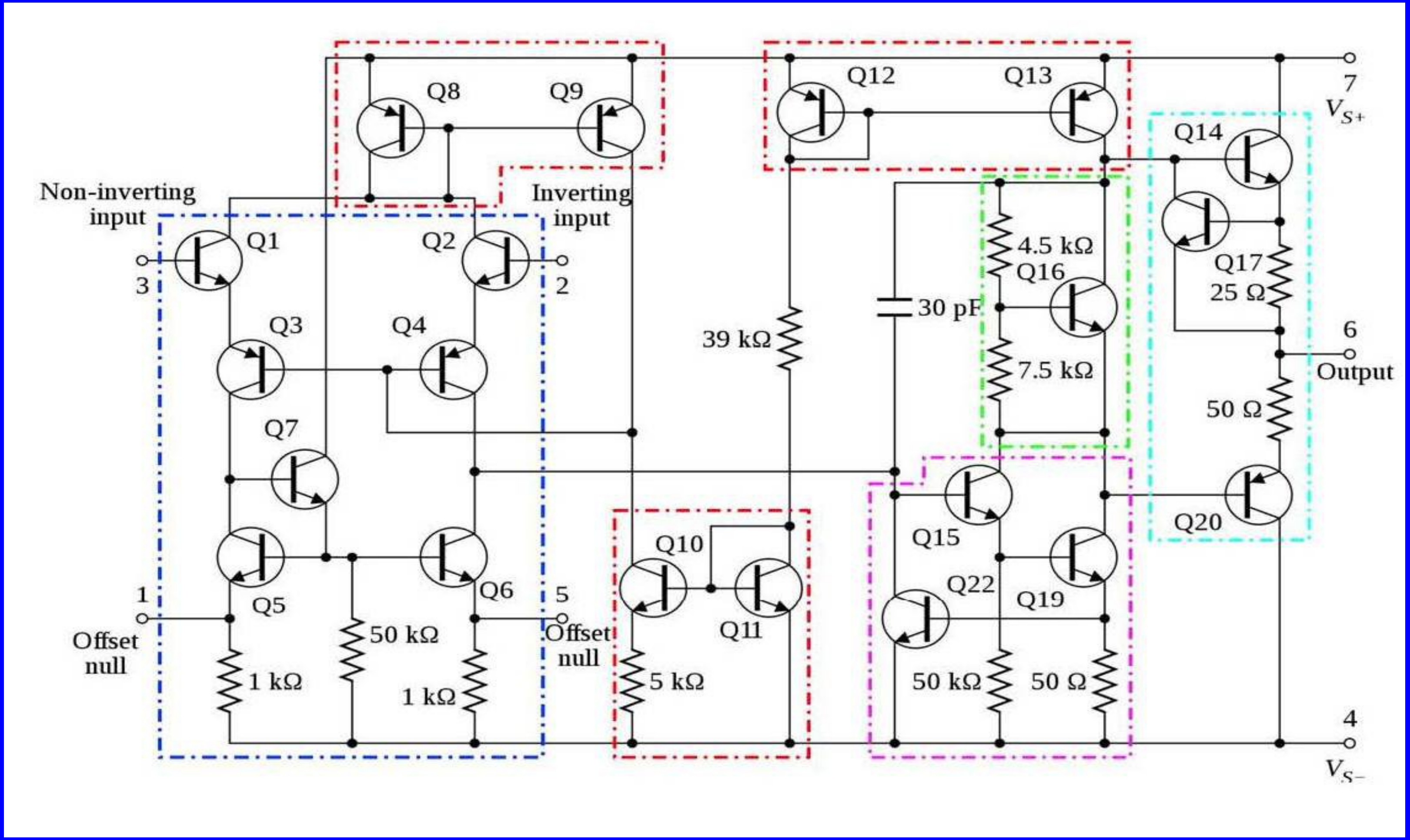
1986 - SPECTRE

1989 - SPICE3

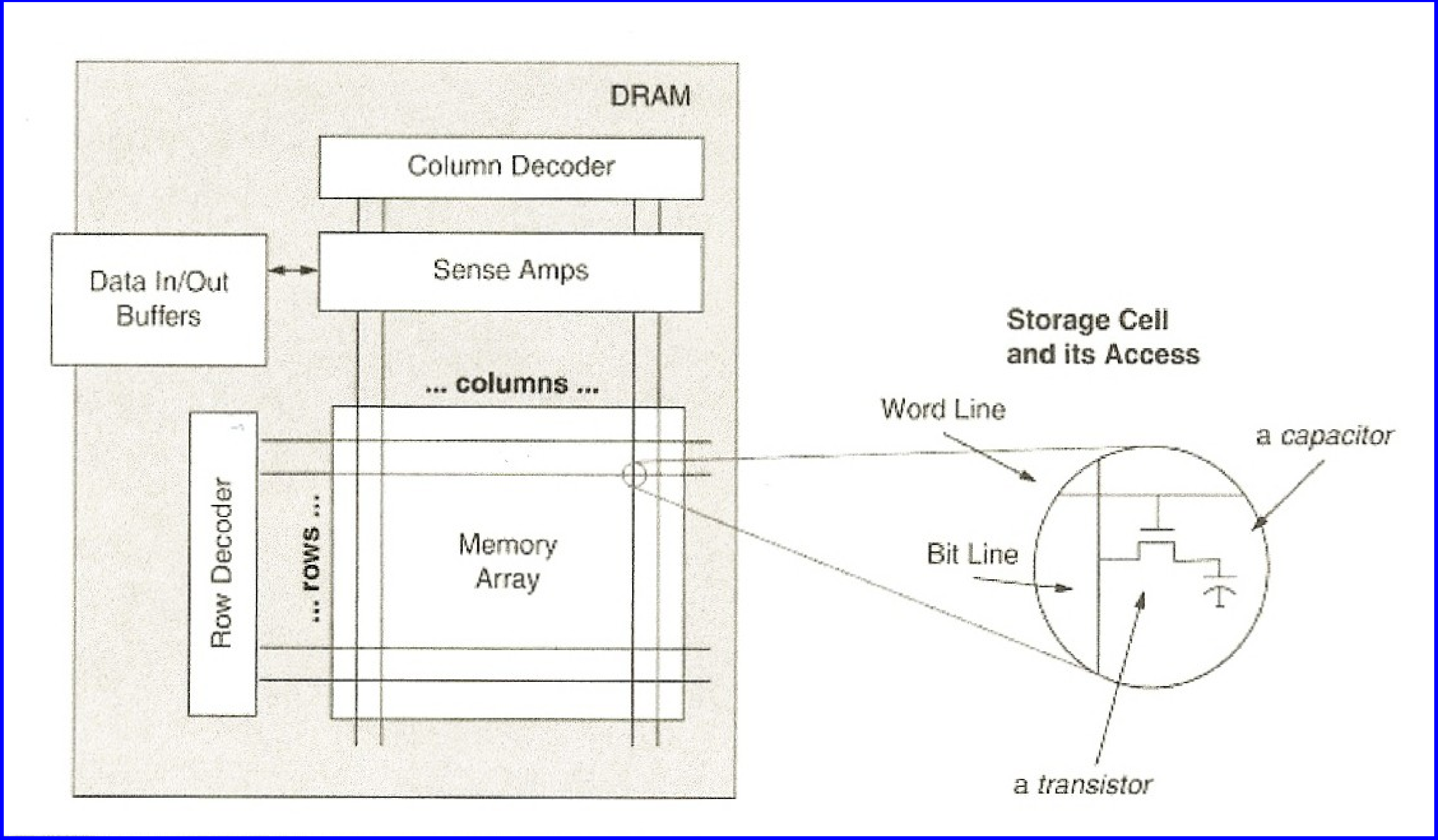
The Evolution of SPICE

- 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

The uA741 Op Amp

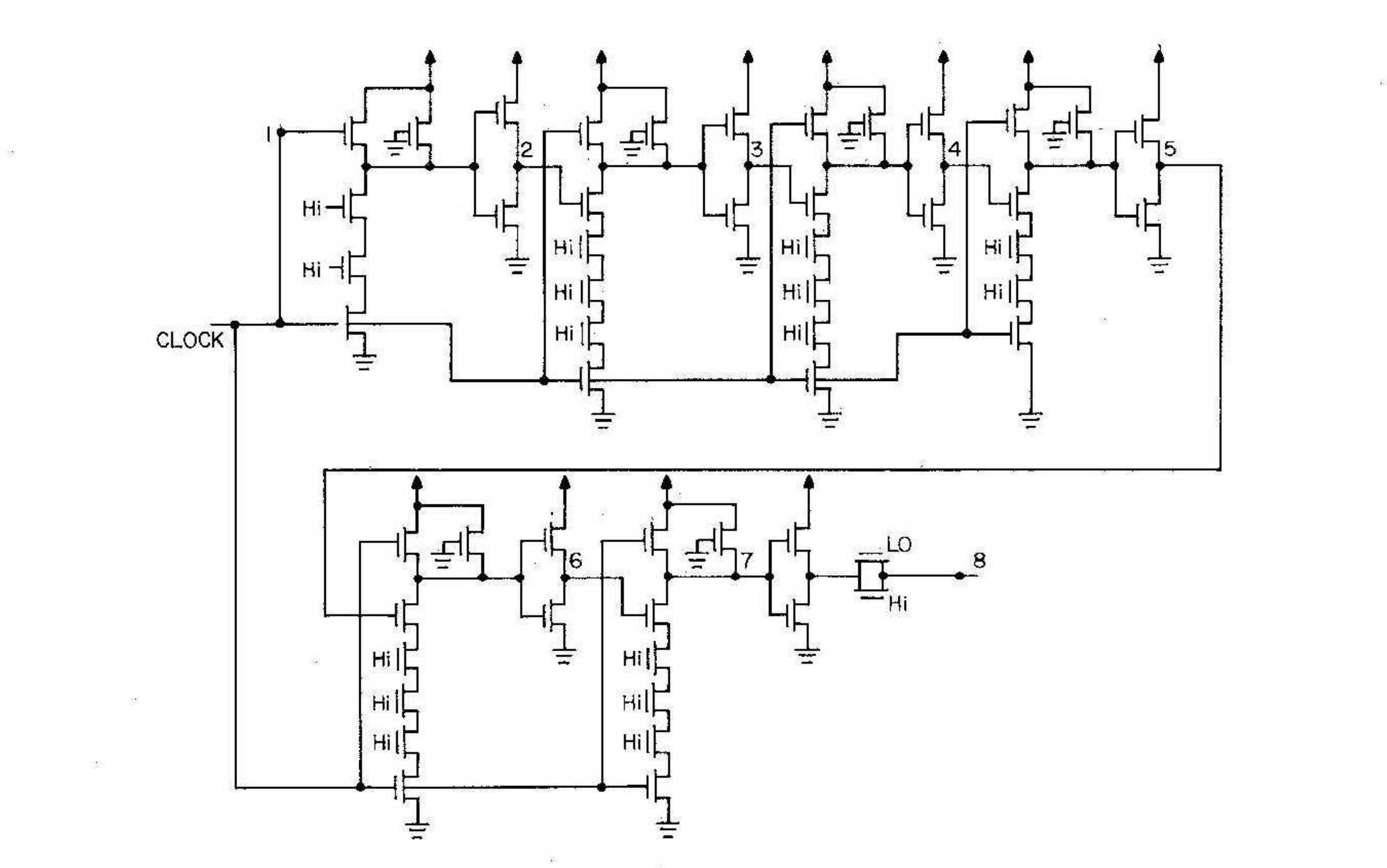


DRAMS

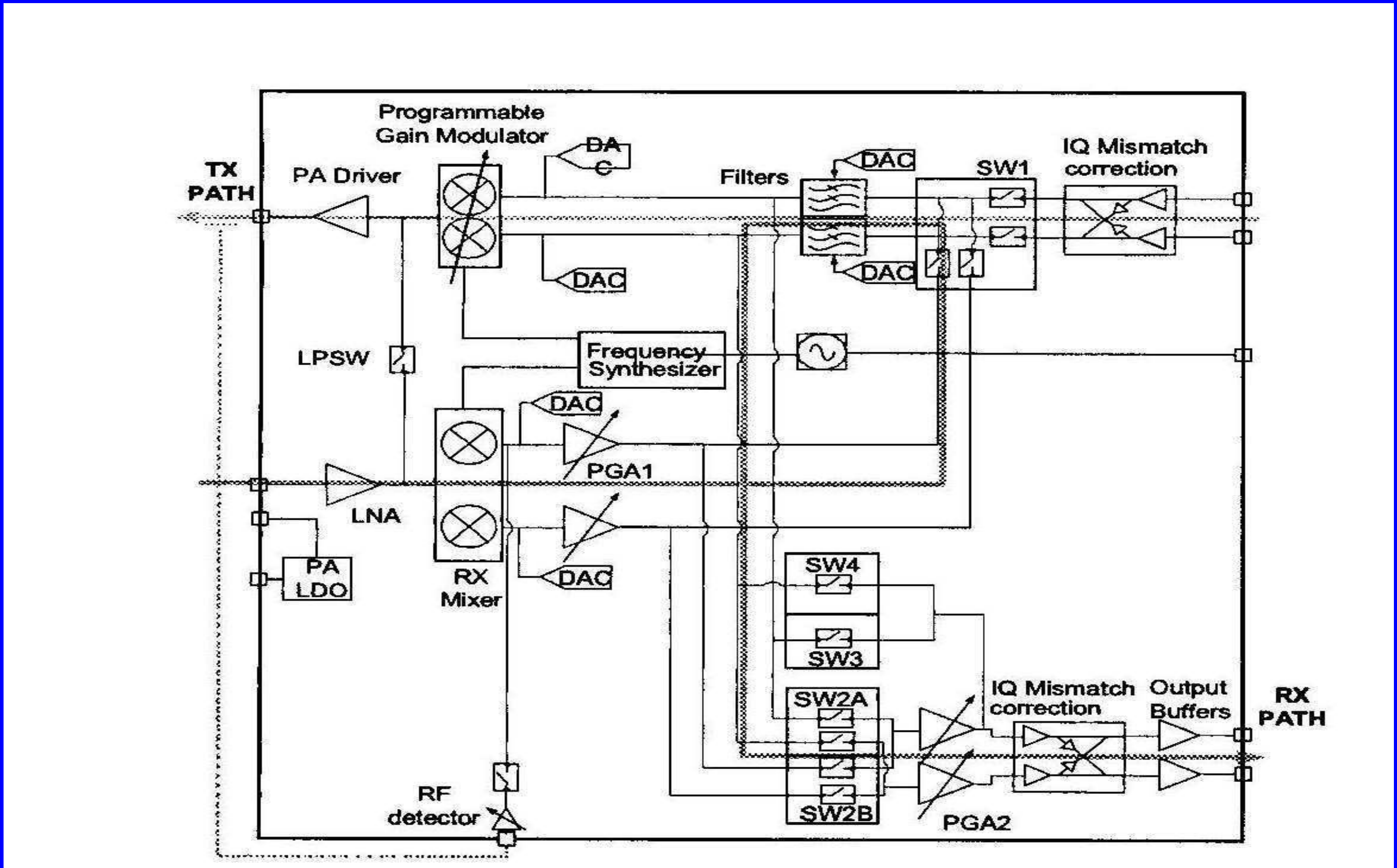


DRAM
Column Decoder
Data In/Out
Buffers
Sense Amps
... columns ...
Row Decoder
... rows ...
Memory
Array
Storage Cell
and its Access
Word Line
a capacitor
Bit Line
a transistor

Domino Logic



The RF Transceiver



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

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)

How Will SPICE Evolve in the Future?

1. SPICE will remain an Open Source tool
2. SPICE will take advantage of new hardware (GPU's, Cloud Computing, even Smart Phones)
3. SPICE will include an advanced version of ADMS to accommodate model development
4. SPICE will accept Verilog-A as input
5. SPICE will include RF Analysis
6. SPICE will include Variational Analysis
7. SPICE will include Thermal Analysis

SPICE Will Remain an Open Source Tool

Preaching to the Choir!!!

- SPICE has lasted as long as it has in part because it is Open Source
- Being Open Source makes SPICE a far better educational tool
- Being Open Source makes SPICE a far better research tool

SPICE Will Take Advantage of New Hardware

- Parallel processing
- Special purpose hardware such as GPUs
- Cloud computing
- Personal computing (tablets and smart phones which run Android or iOS)

SPICE Will Take Advantage of New Hardware

- Exploit the “obvious parallelism” in model evaluation using Graphics Processing Units
- Exploit the “obvious latency” in large circuits
- Large systems need multiple copies of SPICE simulating different portions of the circuit simultaneously
- This is an important application of parallel computing that requires effective interprocess communication

XYCE – Parallel SPICE from Sandia National Laboratories

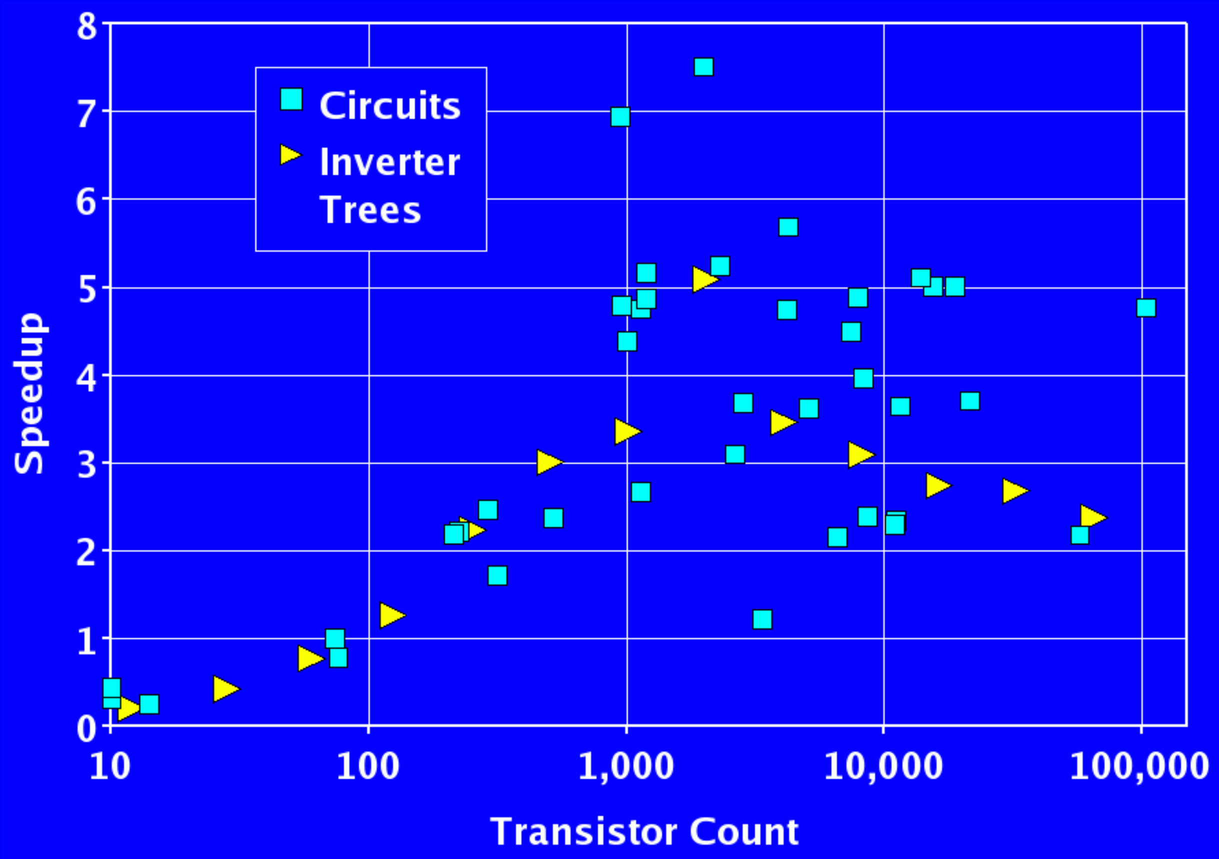
- “SPICE Compatible” circuit simulator that exploits parallel processing hardware
- Developed at Sandia National Labs
- 14 Years in the making
- Will be released as an Open Source tool in the very near future

http://xyce.sandia.gov

Graphics Processing Units (GPUs)

- Originally designed for accelerating graphics for really cool games
- Really a specialized processing tailored to single-instruction multiple-data (SIMD)
- Available for any PC for about $300
- Rick Poore of Agilent first recognized that GPU's would be wonderful for circuit simulation

GPU-Accelerated Circuit Simulation

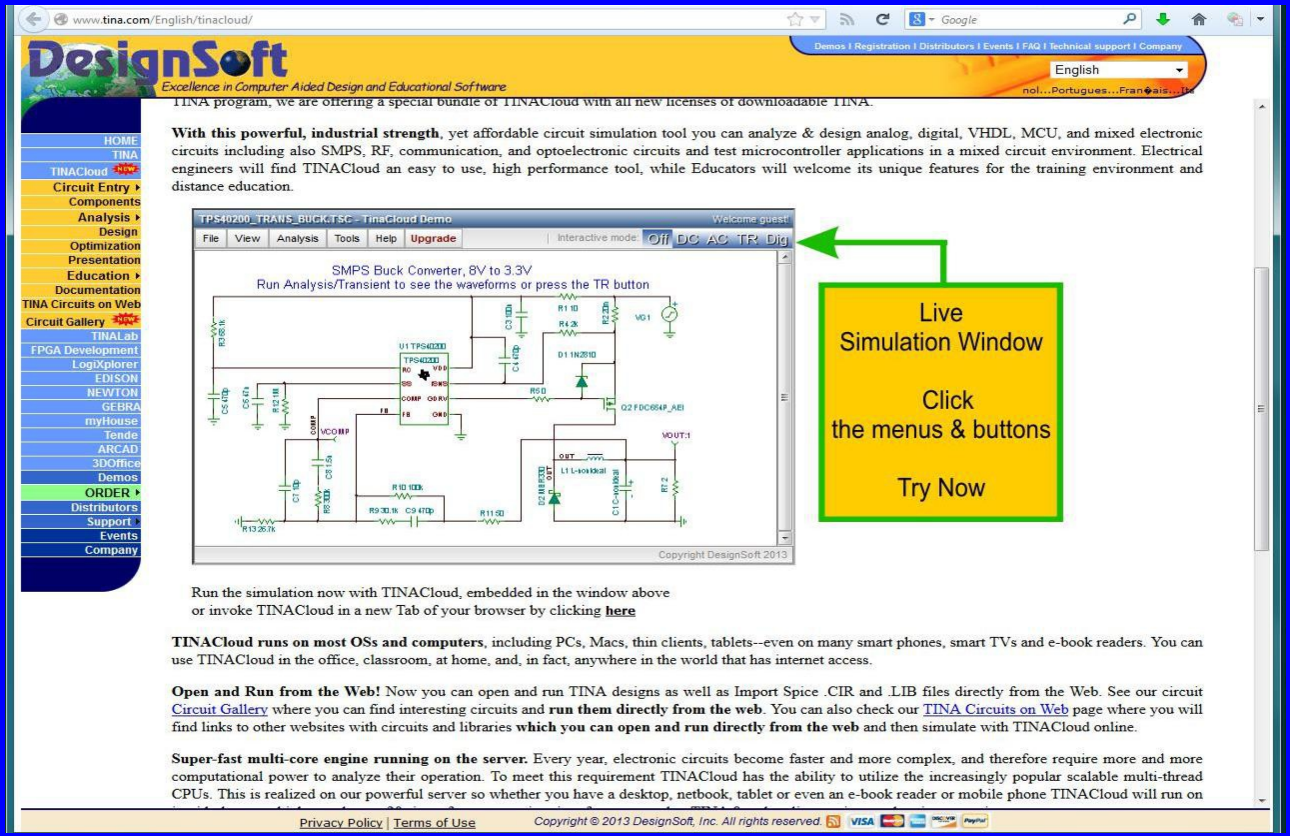


R. E. Poore, "GPU-Accelerated Time-Domain Circuit Simulation," IEEE CICC, June 2009.

SPICE on GPUs

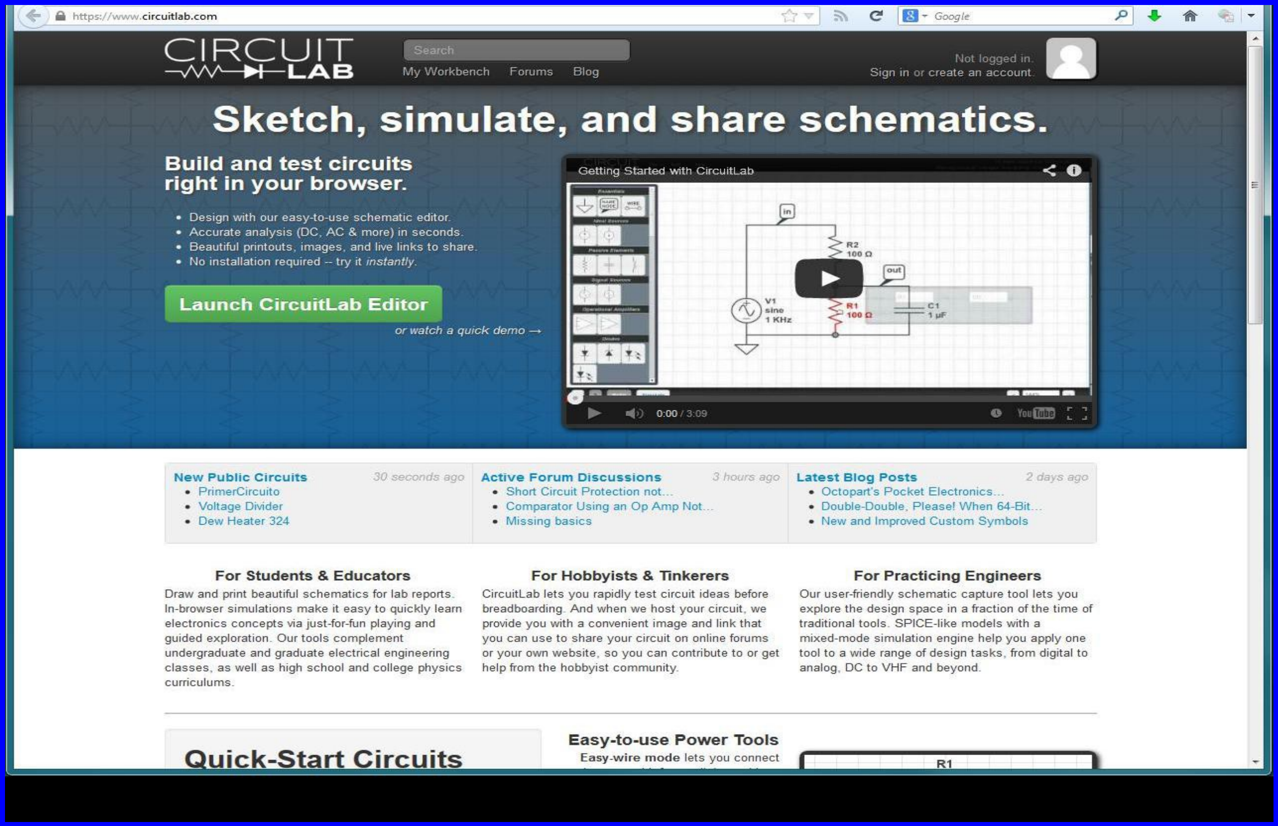
- Rick Poore and I gave a seminar at nVidia on Circuit Simulation in May, 2011
- That lead to nVidia gifting some hardware to University of Rome
- Francesco Lannutti then worked at nVidia in the summer of 2012
- Francesco made some progress, but there is still much to do

SPICE on Cloud Computing from DesignSoft



http://www.tina.com/English/tinacloud

SPICE on Cloud Computing from CircuitLab



http://www.circuitlab.com

SPICE on a Smart Phone

For fun, Oregon State graduate, Igor Vytyaz, created an Android app EveryCircuit for circuit simulation, but it developed into a full-time job.

SPICE Will Include an Advanced Version of ADMS

- The replacement for CMOS has not yet been invented
- During the search, hundreds of “new” device models will have to be added to SPICE to try out new technology ideas
- ADMS was created to accommodate this need

But

The Present Problem with ADMS

Holger Vogt
2013-07-20

Link

Dear Simon,

unfortunately adding BSIMCMG or BSIM6 is not like adding BSIM4. The new model code is only published in Verilog A language. Integrating the code into ngspice requires using a Verilog-to-C compiler like adms. The current adms however is not capable translating the new Berkeley models (see https://sourceforge.net/mailarchive/message.php?msg_id=30884036).

So we need sombody contributing to advance the adms code before we can integrate the new models!

Holger

We desperately need a Verilog-A ADMS!!!

SPICE Will Accept Verilog-A as Input

- In the future, electronic circuits will be built with radically different elements
- SPICE will need to accommodate new elements with greater ease than is now possible
- SPICE needs to understand a circuit description language such as Verilog-A or VHDL-A
- This will give circuit designers and model developers the same flexibility that software engineers have in a programming language

SPICE Will Accept Verilog-A as Input

- Verilog-A compatibility will aid compact model development by allowing full power of Verliog-A language
- Verilog-A compatibility will allow noncritical portions of the circuit to be described at the behavioral level
- Partitioning the circuit into behavioral (functional) blocks will aid parallelization

SPICE Will Include RF Analysis

- By the end of the 1980's, at around the 1μm technology node, CMOS transistor f_T had extended well into the GHz region
- As transistors became faster, it became possible to integrate RF circuits and the wireless explosion was on
- This necessitated an entirely new line of algorithms and simulators

RF Simulation Programs

- 1988 - Microwave Design System (MDS)
- 1991 - Libra
- 1994 - ADS
- 1996 - SPECTRE RF
- 1998 - Eldo RF
- 2003 - Qucs
- 2004 - HSPICE RF

SPICE Will Include RF Analysis

- Each simulator had different algorithms that worked on some RF circuits but not others
- The user interface and netlist description varied from program
- With the exception of Qucs, none of the programs were Open Source
- RF Simulators are only slowly working their way into educational institutions

SPICE Will Include Variational Analysis

- Variational analysis in the past has been added to SPICE almost as an afterthought
- It has been assumed that variations in circuit components have been small
  - 3σ on the order of 10% or 20%
- For the last forty years, worst case corner analysis has been used instead of variational analysis

SPICE Will Include Variational Analysis

- Present FinFET Technology shows tremendous variations in transistors
  - Line edge roughness (LER)
  - Random dopant (RDD)
  - Metal gate granularity (MGG)
- The net result is a variation of six orders of magnitude in off current (the simplest device parameter!)

SPICE Will Include Variational Analysis

- As an educational tool, students now need to learn variational analysis from the very start
  - Variational analysis is not an afterthought!
- New technologies now have to be specified with random variations
- SPICE needs to have Variational Analysis integrated into the basic framework

SPICE Will Include Thermal Analysis

- Devices are being placed closer and closer to each other
- Devices are being placed on “substrates” that are not good thermal conductors
- MOSFET models already include thermal effects, but only “self heating”
- Thermal coupling between devices is or will be significant in analog IC design

SPICE Thermal Analysis

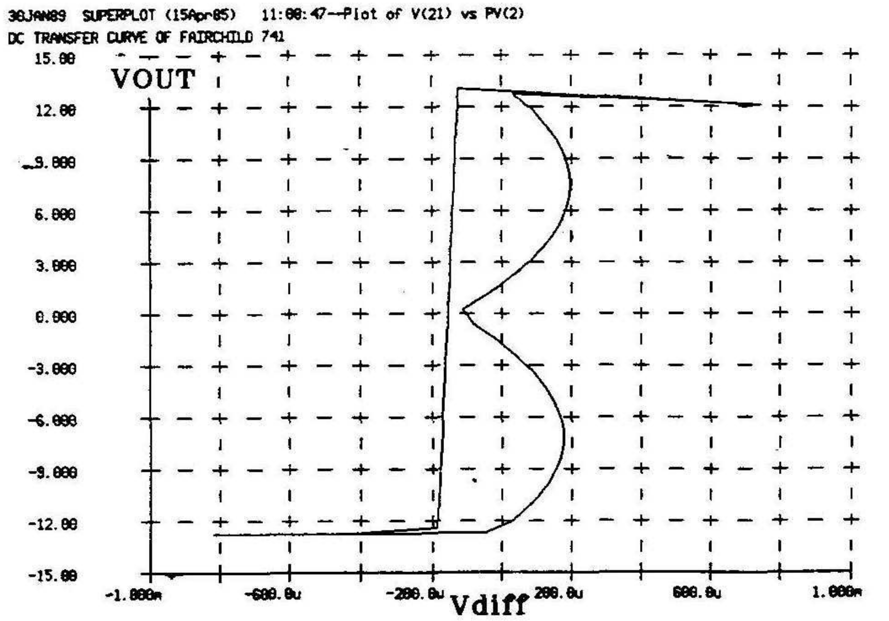


Figure 6: LM 741 Output Voltage vs. Differential Input Voltage as computed by SPICE 2G.6 and Extended SPICE.

R. Vogelsong and C. Brzezinski, "Extending SPICE for Electro-Thermal Simulation," IEEE CICC, June 1989.

How Will SPICE Evolve in the Future?

1. SPICE will remain an Open Source tool
2. SPICE will take advantage of new hardware (GPU's, Cloud Computing, even Smart Phones)
3. SPICE will include an advanced version of ADMS to accommodate model development
4. SPICE will accept Verilog-A as input
5. SPICE will include RF Analysis
6. SPICE will include Variational Analysis
7. SPICE will include Thermal Analysis

Thank You!!!