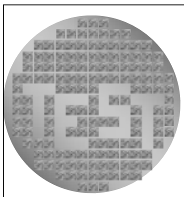


VLSI Design

Tsung-Chu Huang

Department of Electronic Engineering
National Changhua University of Education
Email: tch@cc.ncue.edu.tw

2016/03/08

Circuit Simulation Tools

1. Switch Level:
   - Verilog, VHDL, SILOS, Susie, etc.
2. Transistor Level (Linear & Exponential) Simulation
   - PowerMill, TimeMill, Star-Sim
3. Time-Domain
   - Star-HSPICE, SBTSPICE, Spectre (RF)
4. Frequency Domain
   - ADS, Harmonica, Eldo (RF)
5. Other Approaches
   - Matlab

Introduction to SPICE

1. Simulation Program with IC Emphasis
2. Originated by Prof. D. O. Pederson, UCB, ’70s.
3. SPICE I, II, III → Public Domain in the Internet.
4. HSPICE: MetaSoft Inc., CA. → Avant! Inc., CA.
5. PSPICE: MicroSim Inc., CA. → OrCAD Inc.
   → Cadence Inc.
6. IS-SPICE: IntuSoft, Inc. Interactivity Emphasized.

Espresso with SPICE

A story from some USC professor in 1990.

Espresso is a heuristic program for compressing multi-valued logic circuits.

Would you like a cup of Espresso?

It’s too bitter! Give me some SPICE!

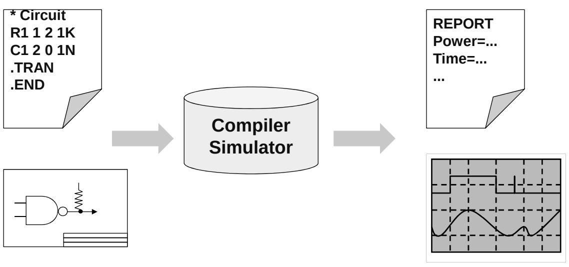
SPICE is a simulation program with IC emphasis.

Donald O. Pederson, UCB

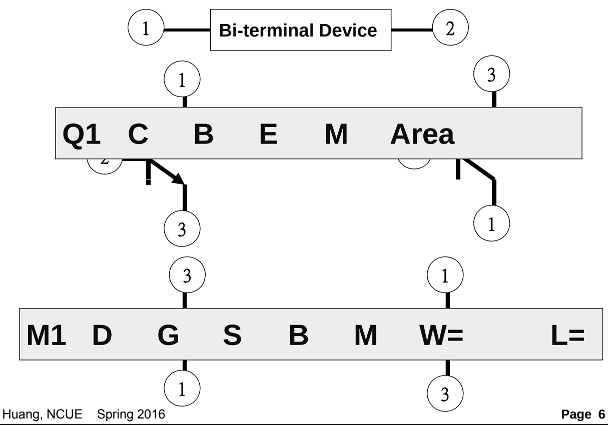
Robert K. Brayton, UCB

“A minimized logic circuit needs an RC-level simulation for verifying the signal integrity,” implied by Pederson.

Basic Flow



Node Ordering

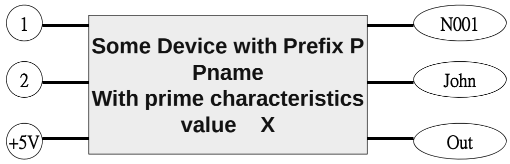


Circuitry Description

Pname 1 2 +5V N001 John Out

Implied Model

Basic Devices with Prime Characteristics

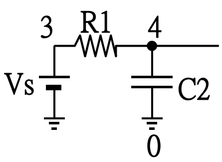


Pname 1 2 +5V N001 John Out X

e.g., L, R, C, V, I, S, E, F, G, H

Example

A Low-Pass 1st-Order RC Circuit



Vs	**3**	**0**	**9V**
R1	**3**	**4**	**56**
C2	**4**	**0**	**3P**

SPICE Modeling

Pname N1 N2 Mname X Options

.MODEL Mname MType (Parameter1=...,
+Parameter2=..., Parameter3=..., ... etc.)

Example of SPICE Model

a part of some .35um library

```
.MODEL nch.1              NMOS   (                       LMIN    = '1.207E-06-dxl'
+LMAX    = 2.01E-05        WMIN    = '1.183E-06-dxw' WMAX    = '1.9983E-5-dxw'
+LEVEL   = 49              VERSION = 3.1
+VTH0    = 'dvtn+0.536027‘ . . . . . . . . .
+AT      = 2219.528        LAT     = -0.02686487     WAT     = -0.002395568
+PAT     = 2.596705E-08    UTE     = -2.065168       LUTE    = -4.132515E-07
+WUTE    = 4.668071E-07    PUTE    = 3.33133E-13     UA1     = 1.422613E-10
+LUA1    = -1.44336E-15    WUA1    = 9.693937E-16    PUA1    = 1.598886E-22
+UB1     = -2.172033E-18   LUB1    = 1.404262E-24    WUB1    = 5.546184E-26
+PUB1    = 6.856764E-31    UC1     = -5.716306E-11   LUC1    = 5.31648E-17
+WUC1    = -2.396283E-17   PUC1    = -2.142338E-24   KT1L    = 0
+PRT     = 24.11496        LPRT    = -1.086812E-05   WPRT    = -0.000500091
+PPRT    = 1.361775E-11
+CJ      = cjn            CJSW     = cjswn          CJSWG    = cjswgn
+MJ      = 0.3297355      MJSW     = 0.1303453      MJSWG    = 0.1303453
+PB      = 0.6944804      PBSW     = 0.6944804      PBSWG    = 0.6944804
+CTA     = 9.42521E-4     CTP      = 4.91382E-04    PTA      = 1.485657E-03
+PTP     = 1.485657E-03   CGDO     = cgon           CGSO     = cgon
+XTI     = 3              N        = 1              CAPMOD   = 0
+NQSMOD  = 0              XPART    = 1              CF       = 0
+CALCACM = 1              SFVTFLAG = 0              VFBFLAG  = 1
+NLEV    = 3              AF       = 0.85           KF       = 1.0E-24
+JS      = 6.27496E-05    JSW      = 5.79847E-10          )
```


Basics: Orders

10^{12}	T	Tera-	T
10^{9}	G	Giga-	G
10^{6}	M	Mega-	MEG
10^{3}	K	Kilo-	K
10^{0}			
10^{-3}	m	Mini-	M, m
10^{-6}	μ	Micro-	U
10^{-9}	n	Nano-	N
10^{-12}	p	Pico-	P
10^{-15}	f	Femto-	F

$$3.14\times10^{-7}=3.14E-7=.314u$$

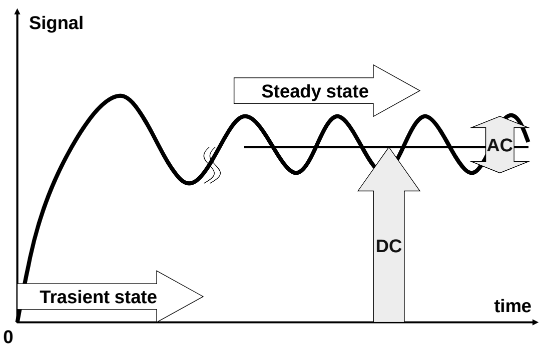
Basics: Physical Units

Physical Var.	Prefix, Symbol	Units
Time	Time, t	Second, s
Voltage	V	Volt, V
Current	I	Amp, A
Frequency	f	Hz
Resistance	R	Ohm, Ω
Conductance	G	Mho
Capacitance	C	F
Inductance	L	Henry

Basics: More Prefix

Physical Var.	Prefix, Symbol	Device	Prefix, Symbol
Time	Time, t	Diode	D
Voltage	V	VCVS	E
Current	I	CCCS	F
Frequency	f	VCCS	G
Resistance	R	CCVS	H
Conductance	G	MOS	M
Capacitance	C	BJT	Q
Inductance	L	Switch	S
		Sub-circuit	X

Basic Analyses



Basic Analyses

3 Major Analyses	Basic Command	X-Axis (Variable)	Basic Application
Transient Analy.	.TRAN Ts Tf	TIME (s)	Delay Measurement
DC Analysis	.DC Vs B E Δ	V, I, etc	Transfer Function
AC Analysis	.AC Dec R B E	Frequency (Hz)	Bode Plot

Extension Analyses

3 Major Analyses	X-Axis (Variable)	Extension	Statistical
Transient Analy.	TIME (s)		.Monte Carlo
		.OP	
DC Analysis	V, I, etc	.TF	.SENS
		.PARAMETER	.TEMP
AC Analysis	Frequency (Hz)	.FOUR	.Noise
		.PZ	.DISTO

Basic SPICE Source

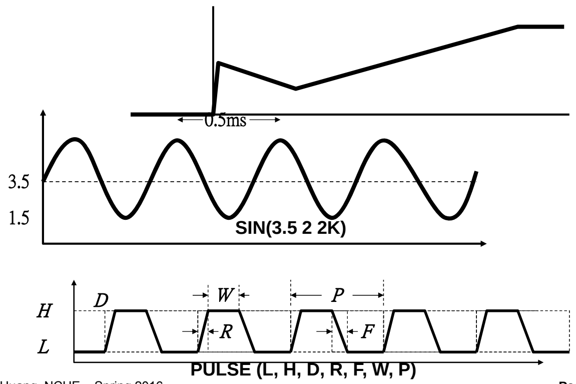
```
* Circuit Name
*Netlist
M2  1   2   3   4   MN
.Model  MN   NMOS ...
R3   0   IN     33K
Cb   Out    0    2.3nF
....
```

* *Remark prefixed with a star sign*

- TRAN 1ps 2us
- END

Source Function

PieceWise Line, PWL(0 0 1u 1 1 0.5 3.5 2)



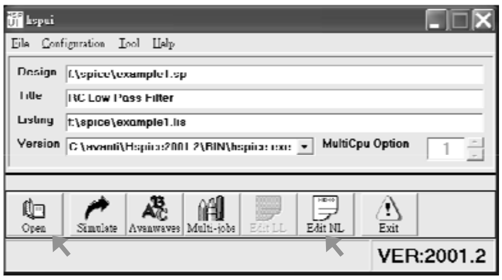
PULSE (L, H, D, R, F, W, P)

Homewrok #2

1. **Familiarize any SPICE Tool. Manually or by a C- or Perl- Script, makeup or update SPICE programs to select CMOS inverters with Gate-Inversion Voltages at 1.5, 2.0, 2.5, 3.0, 3.5 Volts under a 5V technology. Customize the rest given conditions.**

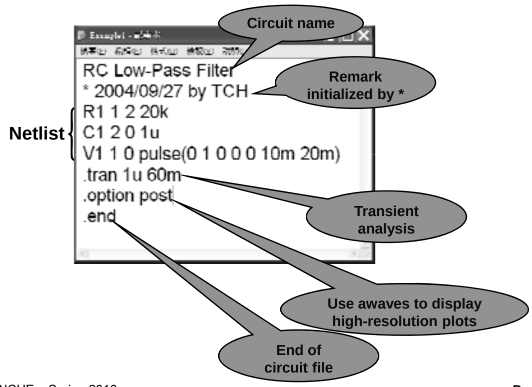
Due to 10/20 (Thu.)

Quick Tutorial using HSPICE

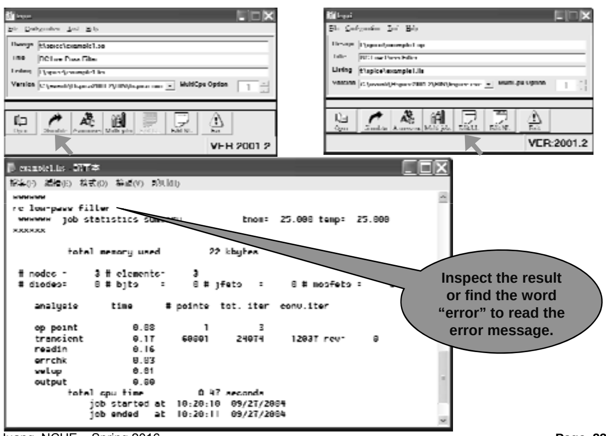


Open a SPICE File **Edit a SPICE File**

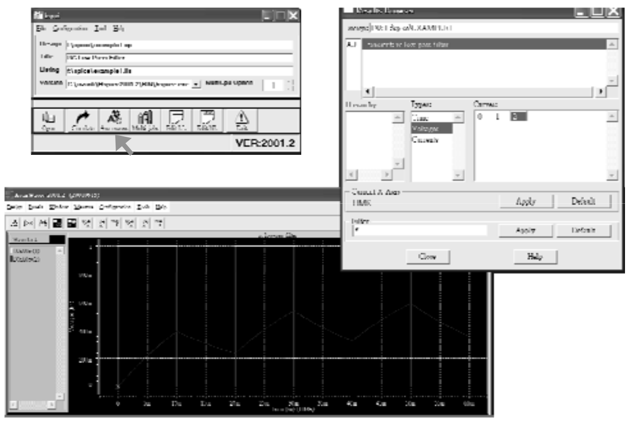
Basic Structure of a SPICE File



Invoke Simulation



Invoke AvanWaves



Some Useful Commands

- **ALTER**
  initialize all simulations except altering the following changes until another .alter or .end
- **MEASURE**
  measure energy, power or integral values in a specified range
- **PRINT variables**
- **IC V(node3)=2.5**
  initial condition

Measurement in HSPICE

Com.	Type	Name	Measured-Function		Range/Position		(Optimize)
			Function	**Variable**	**Start point**	**End Point**	
.MEAS	DC (.ms#) AC (.ma#) TRAN (.mt#)	*Symbol*	MIN MAX AVG RMS	V(node) I(element) Vdb(node) Idb(element) POWER	Trig variable1 VAL=val RISE/FALL /CROSS=#	Targ variable2 VAL=val RISE/FALL /CROSS=#	GOAL= MINVAL= WEIGHT=
			(abscissa)		From =	To =	
			Find	V(node) I(element) Vdb(node) Idb(element) POWER	At When		
			(abscissa)				

EXAMPLES

```
.MEAS TRAN Ix FIND Iout AT 3.7ns

.MEAS DC Vy FIND Vout AT 1.4ms

.MEAS TRAN Px MAX POWER FROM = 2ns
+                        TO = 3ns

.MEAS TRAN Tdelay
+ TRIG Vin  RISE = 100 VAL = 0.5*VDD
+ TARG Vout FALL = 100 VAL = 0.5*VDD
```


MatLab Toolbox for HSPICE

- **Installation**
  1. **Courtesy of Prof. Perrot**
  2. **http://www-mtl.mit.edu/~perrot**
- **SPICE Data Interface**
  1. **MEAS or PRINT to .list file and scanf by C programs**
  2. **Import MEAS table .mt# to EXCEL or Matlab**
  3. **Load Signals to Matlab using HSPICE toolbox**
- **Electronic Design Automation**
  1. **Iteratively approaching with a heuristics**
  2. **System Call:**
     `system("hspice –input ckt.sp >> ckt.lis");`