

VLSI Design

Tsung-Chu Huang

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

2011/10/17

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

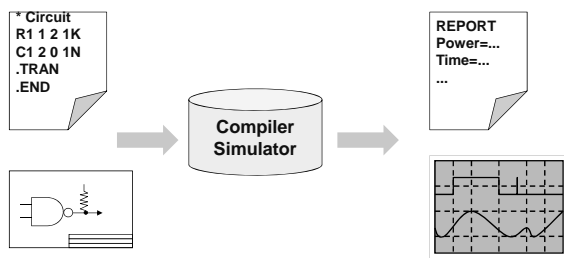
1. Simulation Program with IC Emphasis
2. Originated by Prof. D. O. Pederson, UCB, '70s.
3. SPICE I, II, III → Public Domain in the Internet.
4. HSPICE: MetaSoft Inc., CA. → Avant! Inc., CA.
5. PSPICE: MicroSim Inc., CA. → OrCAD Inc.
→ Cadence Inc.
6. IS-SPICE: IntuSoft, Inc. Interactivity Emphasized.

A Story

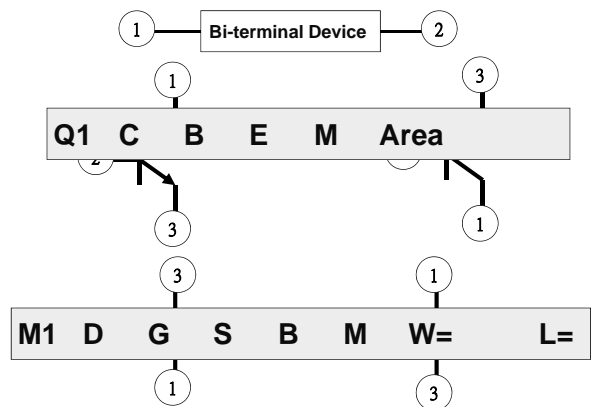
Braton: Espresso PLA-Based Synthesizer

Pederson: SPICE Addressing to Espresso?

Basic Flow



Node Ordering



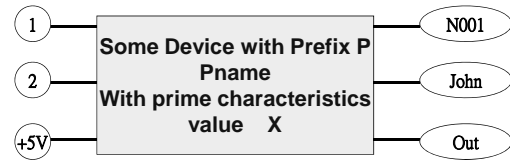
Circuitry Description



Pname 1 2 +5V N001 John Out

Implied Model

Basic Devices with Prime Characteristics

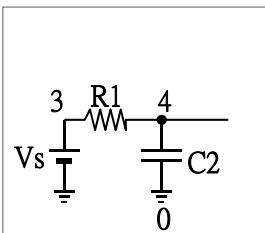


Pname 1 2 +5V N001 John Out X

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-dx1'
+LMAX = 2.01E-05 WMIN = '1.183E-06-dxw' WMAX = '1.9983E-5-dxw'
+LEVEL = 49 VERSION = 3.1
+VTH0 = 'dvtm+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 UAI = 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 UCI = -5.716306E-11 LUCI = 5.31648E-17
+WUCI = -2.396233E-17 PUCI = -2.142338E-24 KILL = 0
+WPRT = 24.11496 LPRT = -1.086812E-05 WPRT = -0.000500091
+PPRT = 1.361775E-11
+CU = cjn CJSW = cjswn CJSWG = cjswn
+MU = 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 CQSO = cgon
+XTI = 3 N = 1 CAPMOD = 0
+NQSMOD = 0 XPART = 1 CF = 0
+CALCACM = 1 SFVFLAG = 0 VFFLAG = 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

$$3.14 \times 10^{-7} = 3.14E-7 = .314\mu$$

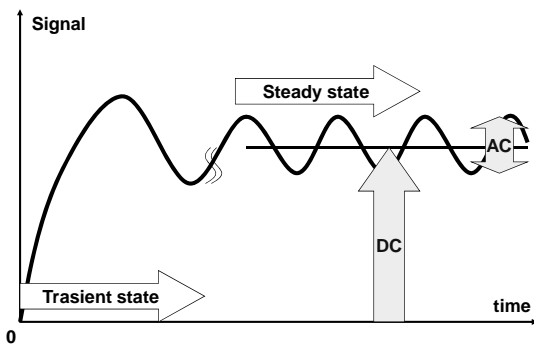
Basics: Physical Units

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
DC Analysis	V, I, etc	.OP .TF .PARAMETER	.SENS .TEMP
AC Analysis	Frequency (Hz)	.FOUR .PZ	.Noise .DISTO

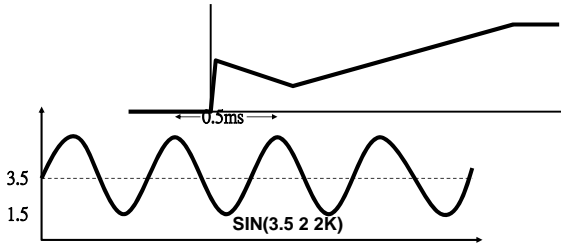
Basic SPICE Source

```

* 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)

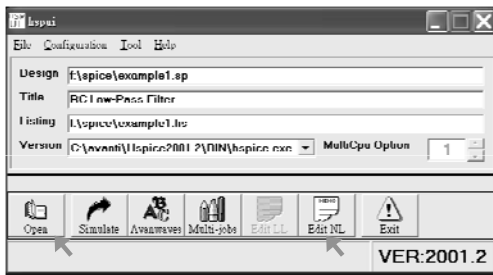


Homewrok #2

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.)

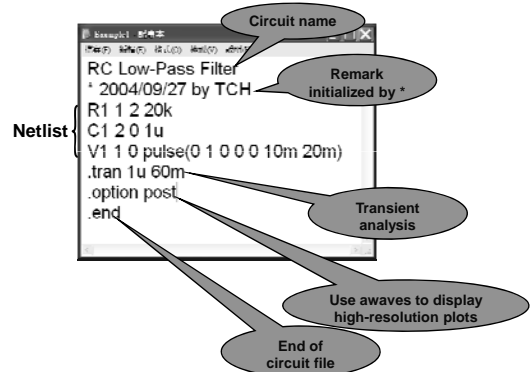
Quick Tutorial using HSPICE



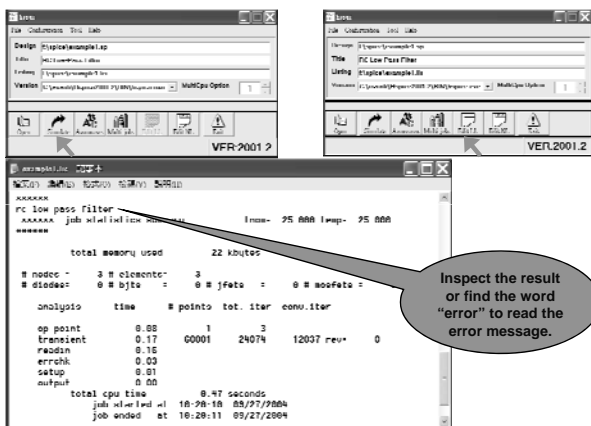
Open a SPICE File

Edit a SPICE File

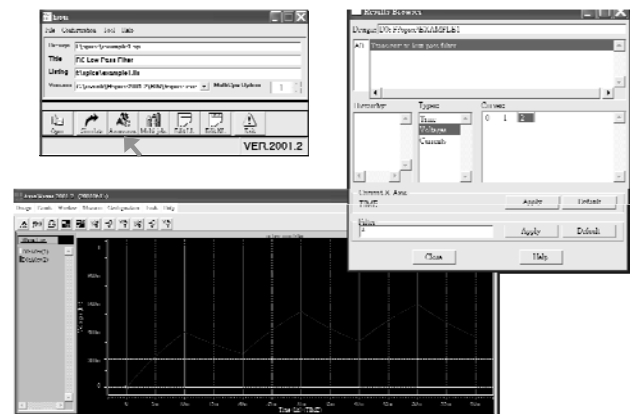
Basic Structure of a SPICE File



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#)	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=
	AC (.ma#)		(abscissa)	From =	To =		
	TRAN (.m#)		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
```