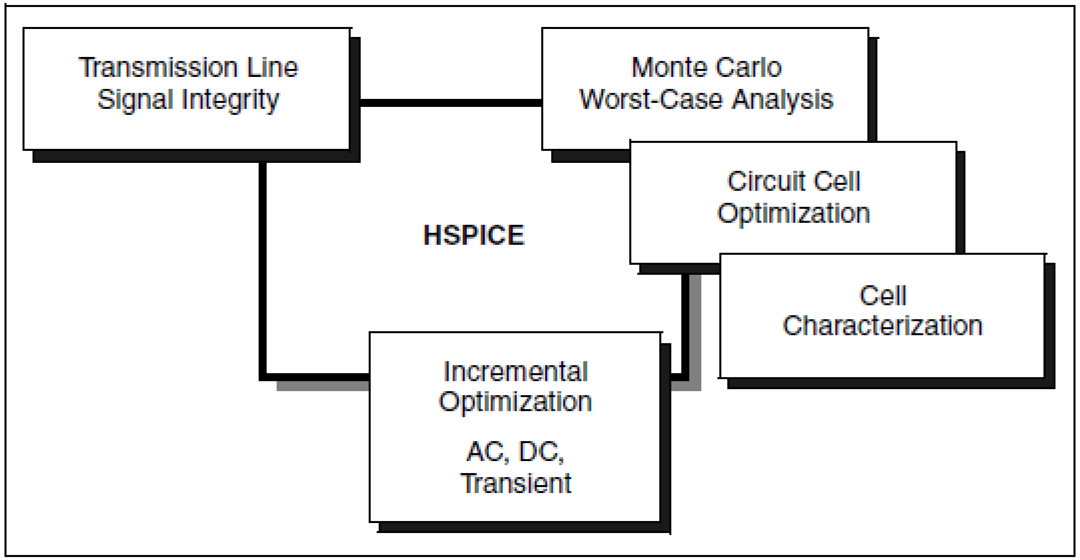
IEE5049-Spring 2012
Digital Integrated Circuit
HSPICE Tutorial

Tzu-Ting, Chiang
Department of Electronics Engineering
National Chiao Tung University
pebuch.ee96@g2.nctu.edu.tw

HSPICE Overview

- Synopsys HSPICE is an optimizing analog circuit simulator.
  - To simulate electrical circuits in steady-state, transient, and frequency domains.
  - Analysis of performance and yield, by using Monte Carlo, worst-case, parametric sweep, and data-table sweep analyses



Sourced from : HSPICE® User Guide C-2009.03

HSPICE Overview

- Important file types

Input File Type	File Name
Input netlist file	design.sp
Library input file	*library_name*

Output File Type	Extension
Output listing / log file	.lis
Transient analysis measurement results	.mt#
Transient analysis results (from .POST statement)	.tr#

Sourced from : HSPICE® User Guide C-2009.03

- >hspice -i <input_file_name.sp> -o <log_file_name.lis>
- >hspice -i adder.sp -o adder.lis

Netlist Structure

```
.TITLE Hspice tutorial

************************* hspice simulation options
.option post nomod brief measdgt=7 captab

************************* process and temperature options
.include ‘65nm_bulk.pm'
.temp 25
.global VDD GND

************************* parameters
.param supply=1v
.param wp=0.2u
.param wn=0.08u
.param slew=30p

************************* input / voltage sources
vs vdd gnd supply
vin tin gnd pulse(0 supply 0.5n slew slew 0.47n 1n)

************************* circuit
MM0 OUT IN VDD VDD PMOS W=wp L=0.065u M=1.0
MM1 OUT IN GND GND NMOS W=wn L=0.065u M=1.0

************************* hspice simulation modes
.tran 1p 5n *sweep width 0.045u 0.2u 0.001u

************************* measurements
.meas tran Tdr Trig v(in)    val='0.5*supply' rise=2
+                  Targ v(out) val='0.5*supply' fall=2
.meas tran Tdf Trig v(in)    val='0.5*supply' fall=2
+                  Targ v(out) val=‘0.5*supply’ rise=2

.end
```

- .TITLE
- .option
  - post
  - measdgt=7
  - captab
- .include / .lib
- .temp
- .global
- .param
- .end

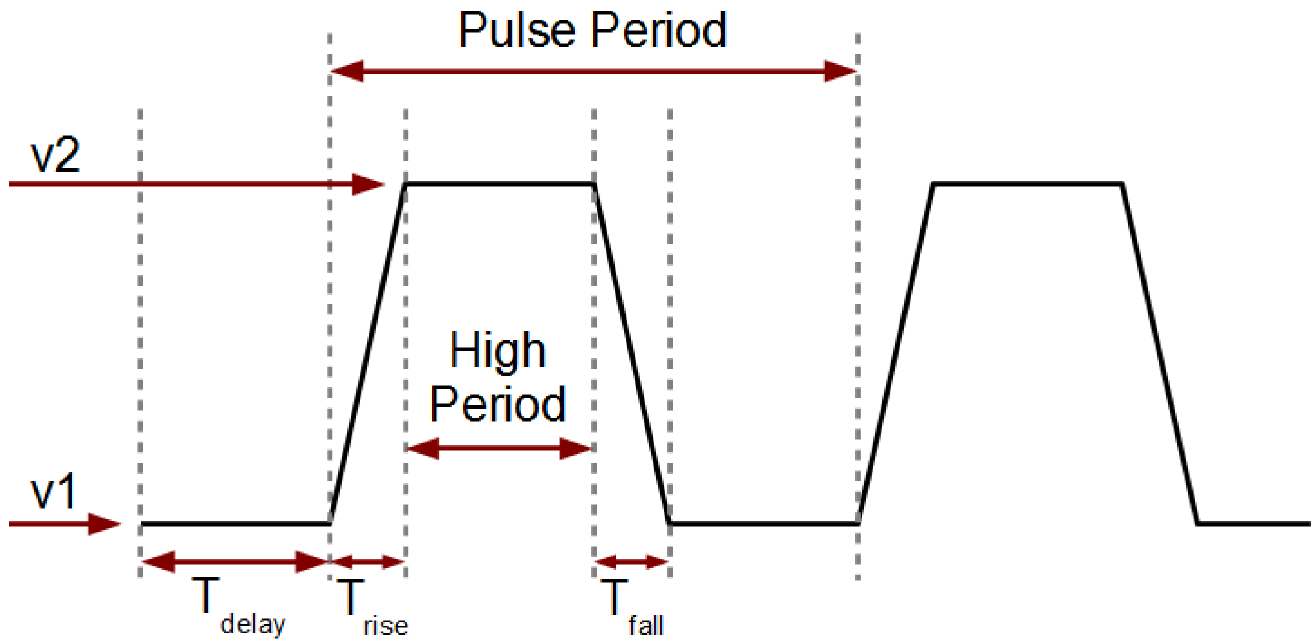
Input Voltage and Current Sources

- **Vxxx n+ n- [DC=] *dcval tranfun [AC=acmag acphase]***
  - v1 1 0 DC=5v
  - v2 2 0 5v
- **Ixxx n+ n- [DC=] *dcval tranfun [AC=acmag acphase]***
  - i3 3 0 5mA
- *tranfun*
  - PULSE($v1\ v2\ T_{delay}\ T_{rise}\ T_{fall}\ P_{high\text{-}width}\ P_{period}$)
  - PWL(t1 v1, <t2 v2, t3 v3…> <R<=repeat>> <TD=delay>)
  - SIN(V_{offset} V_{acmag} <Freq T_{delay} Dfactor>)
  - Exponential

Low Power Microsystems Design

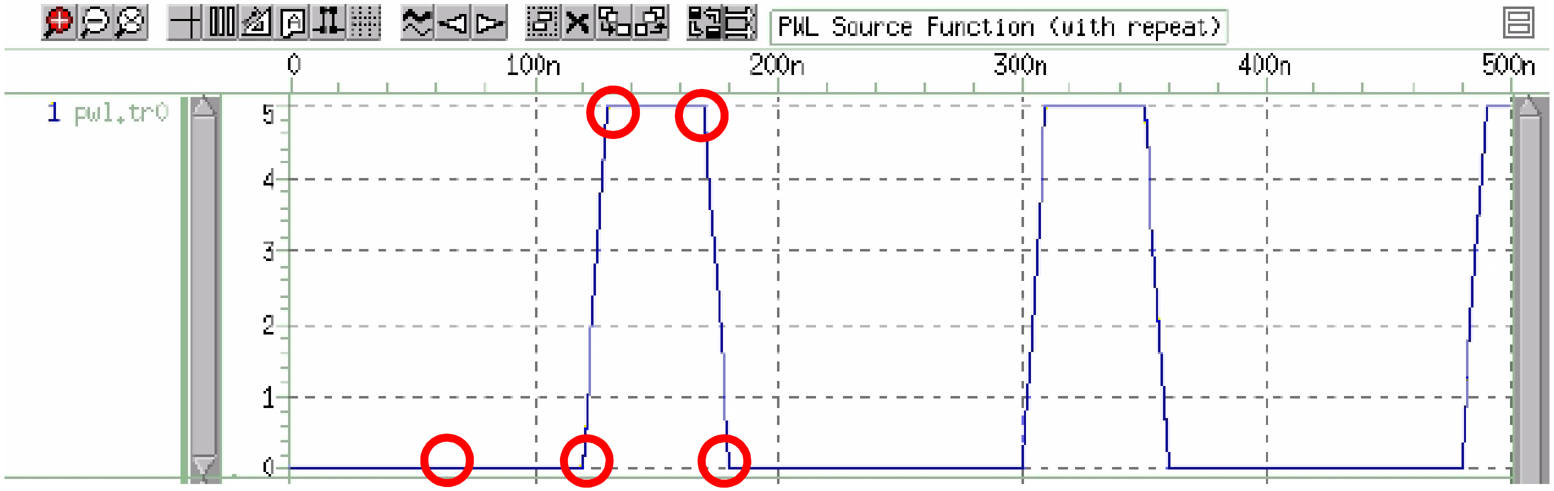
Pulse Voltage Source

- **Vxxx node1 node2 PULSE(v1 v2 T_{delay} T_{rise} T_{fall} $P_{high\text{-}period}$ P_{period})**
  - Vin 1 0 PULSE(0v 5v 5ns 2ns 2ns 5ns 14ns)



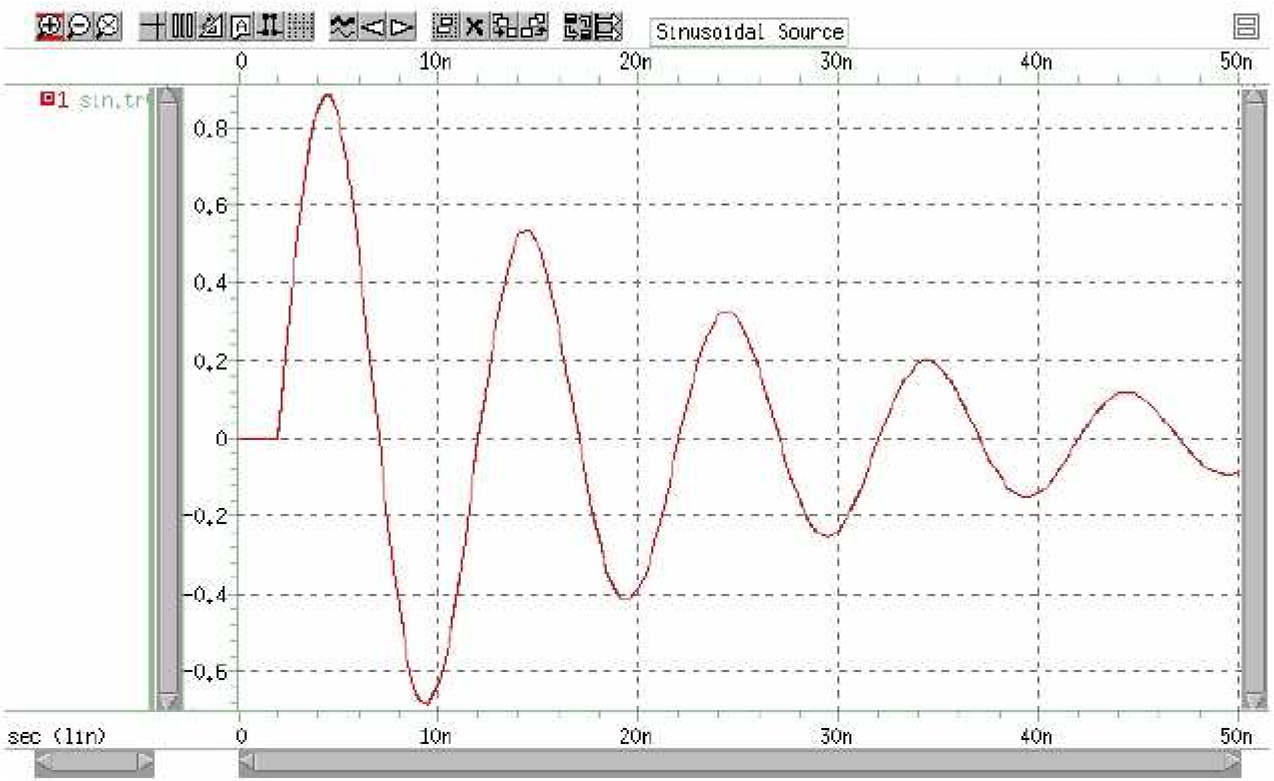
Piecewise linear (PWL) Voltage Source

- **Vxxx node1 node2 PWL(t1 v1, <t2 v2, t3 v3…> <R<=repeat>> <TD=delay>)**
  - v1 1 0 pwl(60n 0v, 120n 0v, 130n 5v, 170n 5v, 180n 0v, R 0)



Sinusoidal Voltage Source

- Vxxx node1 node2 SIN(V_{offset} V_{acmag} <Freq T_{delay} D_{factor}>)
  - VIN 3 0 SIN(0 1 100meg 2ns 5e7)



Element Identifier & Scale Factors

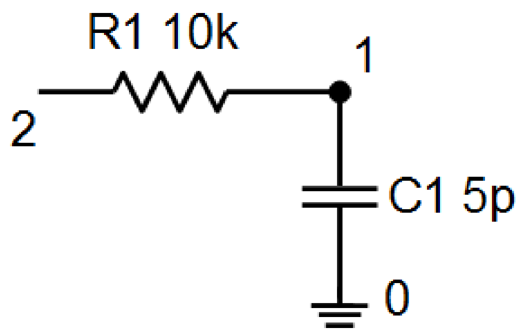
First Letter	Element
C	Capacitor
D	Diode
I	Current Source
L	Linear Inductor
M	MOS transistor
Q	Bipolar transistor
R	Resistor
V	Voltage Source
X	Subcircuit call

Prefix	Scale Factor Symbol	Factor
tera	T	1e+12
giga	G	1e+9
mega	Meg	1e+6
kilo	k	1e+3
milli	m	1e-3
micro	u	1e-6
nano	n	1e-9
pico	p	1e-12
femto	f	1e-15
atto	a	1e-18

Element Syntax

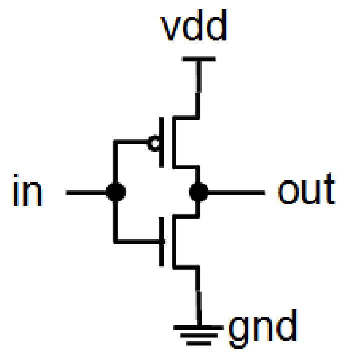
- Passive components
  - **Rxxx *node1 node2* value**
  - C, L, … etc. just the same syntax
    - EX: r1 1 2 10k

      c1 1 0 5p



- Active components
  - **Mxxx *Drain Gate Source Bulk Model* width length <multiple>**
    - EX: mtest1 out in vdd vdd pmos w=195n l=65n m=1

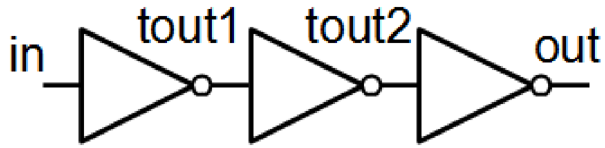
      mtest2 out in gnd gnd nmos w=65n l=65n m=1



Subcircuit

- ***.SUBCKT** **NAME Node1 <Node2 …> <param1 …>***
  circuit description
  .ENDS
- Subcircuit call
  - Xxxx *Node1 <Node2 …> NAME <param1 …>*

EX:



W/O subckt

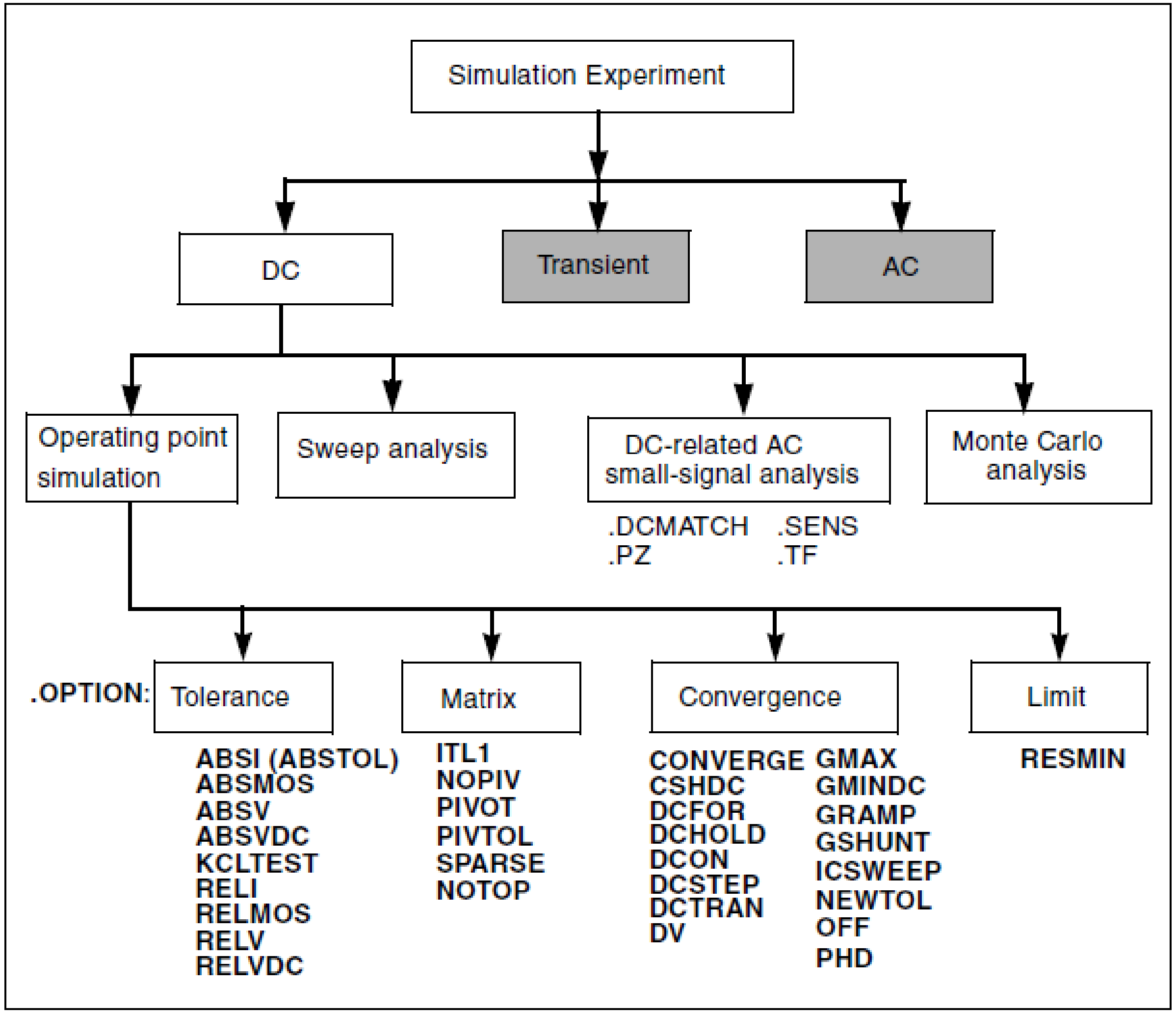
```
.global VDD GND

************************ circuit
MM0 tout1 IN VDD VDD PMOS W=195n L=65n
MM1 tout1 IN GND GND NMOS W=65n  L=65n

MM2 tout2 tout1 VDD VDD PMOS W=390n L=65n
MM3 tout2 tout1 GND GND NMOS W=130n  L=65n

MM4 out tout2 VDD VDD PMOS W=780n L=65n
MM5 out tout2 GND GND NMOS W=260n  L=65n

.end
```

W/ subckt

```
.SUBCKT INV IN OUT wp=195n wn=65n
MM0 OUT IN VDD VDD PMOS W=wp L=65n
MM1 OUT IN GND GND NMOS W=wn  L=65n
.ENDS

.global VDD GND

************************ circuit
X0 IN tout1 INV
X1 tout1 tout2 INV wp=390n wn=130n
X2 tout2 OUT INV wp=780n wn=260n

.end
```

Simulation Modes

- Transient / DC Analysis / AC analysis



Transient Analysis

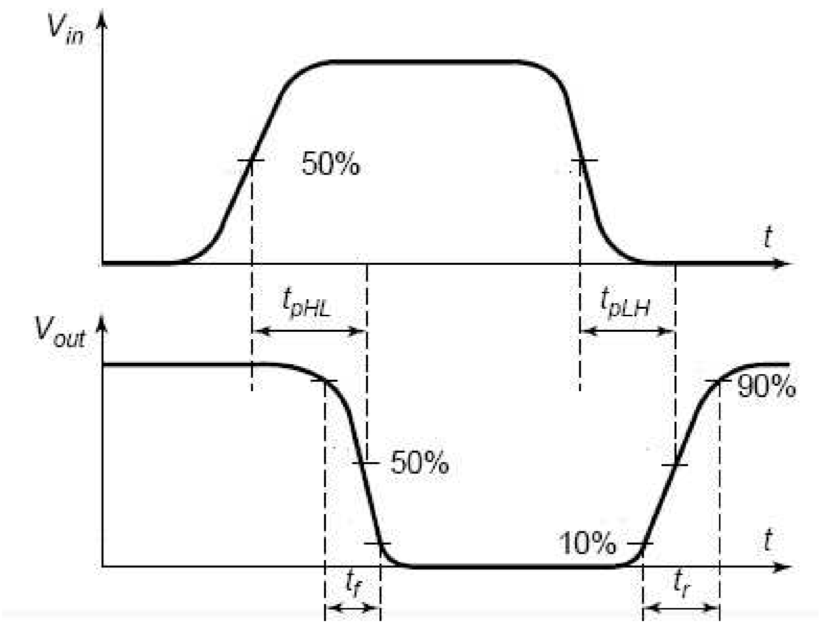
- .tran <timestep> <total simulation time>
  + sweep <variable> <start value> <end value> <step>
  - .tran 1p 5n
  - .tran 1p 5n sweep temp 0 25 5
- .dc
- .ac
- HSPICE® User Guide: Simulation and Analysis

Measurements(1/2)

- **.meas(ure) <*mode*> <*name*> <*type description*>**
  - Mode
    - DC / AC / TRAN
  - Type
    - Find, when/ AVG / MAX / MIN / Trig, Targ / PP
- EX1:
  - .meas dc vout find V(out1) when V(in)=0.9
- Ex2:
  - .meas tran Pavg avg I(out) from=5n to=50n

Measurements(2/2)

- EX3:
  - .meas tran TpHL Trig v(in) val=0.5V rise=2
    + Targ v(out) val=0.5V fall=2
  - .meas tran TpLH Trig v(in) val='0.5*supply' fall=2
    + Targ v(out) val=‘0.5*supply’ rise=2
- Measured results are listed in .mt# file

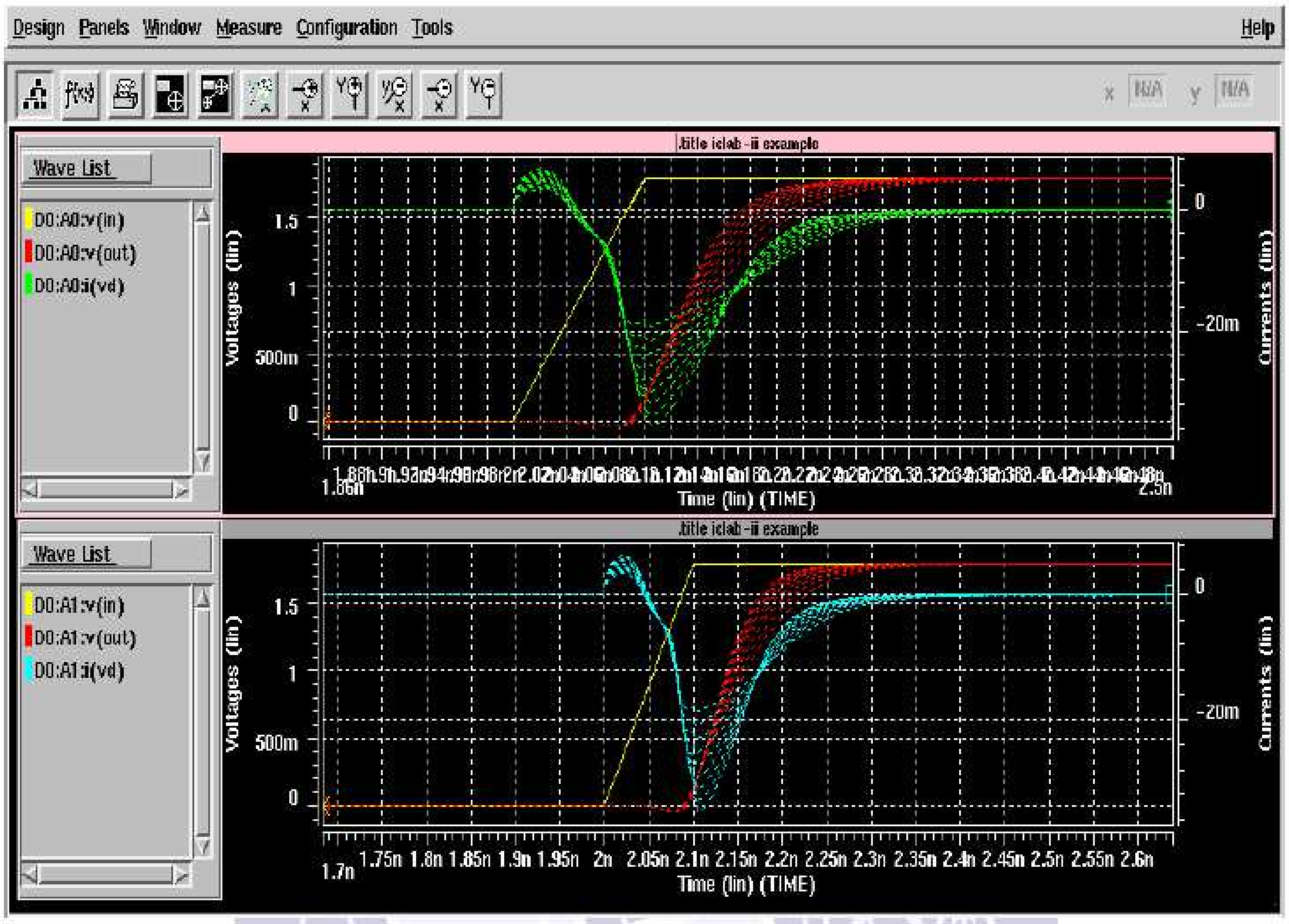


Multiple Analyses

- .alter
  - Simulation all over again using new parameters
  - .mt0 .mt1 .mt2 etc.
  - .tr0 .tr1 .tr2 etc.
- .sweep
  - Simulation using new parameters but with the same initial point
  - All in the same .mt0/.tr0

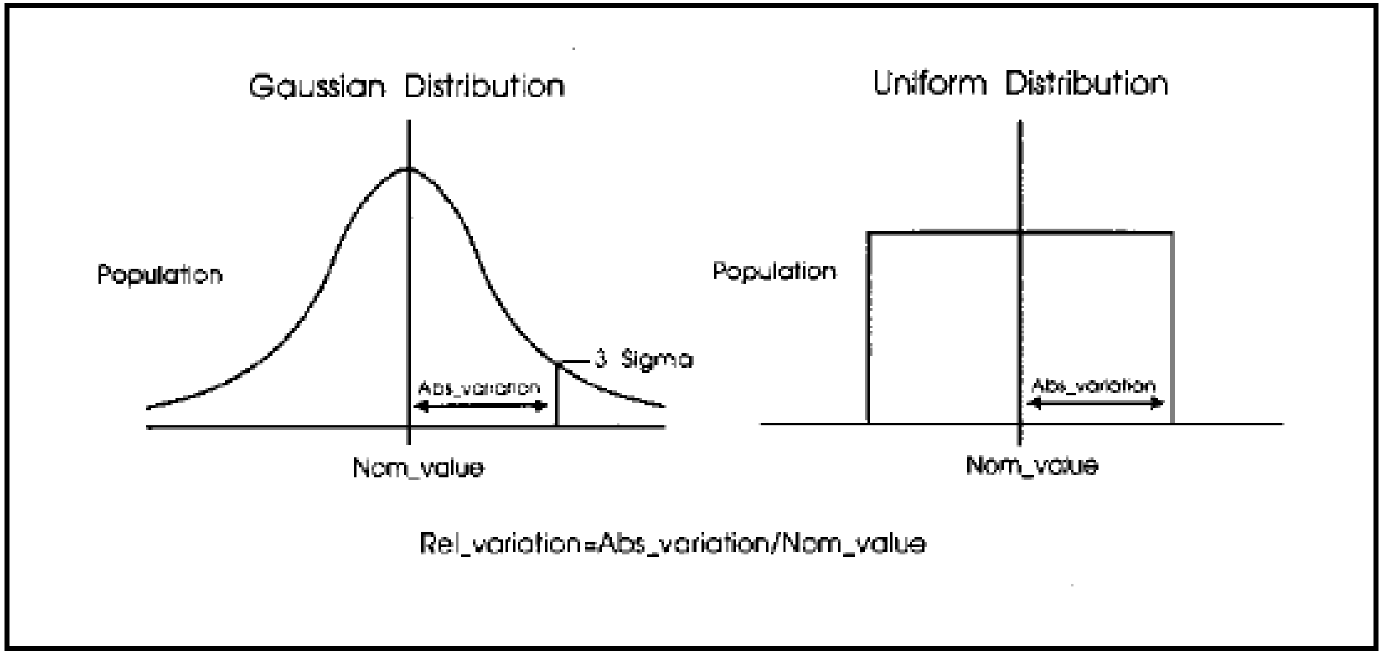
```
************************* process and temperature options
.temp 25

************************* parameters
.param supply=1v
.param wp=0.2u
.param wn=0.08u
.param slew=30p

************************* hspice simulation modes
.tran 1p 5n  sweep temp 0 25 5

************************* measurements
.meas tran Tdr Trig v(in)    val='0.5*supply' rise=2
+                Targ v(out) val='0.5*supply' fall=2
.meas tran Tdf Trig v(in)    val='0.5*supply' fall=2
+                Targ v(out) val='0.5*supply' rise=2

.alter
.param supply=0.5v temp=25
.alter
.param supply=0.5v temp=50

.end
```

Avanwaves – Avanwaves Window



Monte Carlo Simulation

- Monte Carlo analysis uses a **random number** generator to create the following types of functions:
  - Gaussian Parameter Distribution
  - Uniform Parameter Distribution
  - Random Limit Parameter Distribution



Monte Carlo Simulation

- **.param xx=AGAUSS(nominal_val, abs_variation, sigma)**
  - **AGAUSS :** Gaussian distribution function using absolute variation
  - **nominal_val :** Nominal value for Monte Carlo analysis and default value for all other analyses
  - **abs_variation :** Vary the nominal_val by ±abs_variation
  - **sigma :** The abs_variation is specified at the sigma level. For example, if sigma=3, then the standard deviation is abs_variation divided by 3.
- **EX:**
  - vth0 = 0.423
  - vth0 = 0.423+agauss(0,30m,1)
  - .param wp=200n+agauss(0,10n,1)

[HSPICE® User Guide: Simulation and Analysis]

[Star-Hspice Manual 2001]

Technology Corner

- Typical (TT)
- Fast PMOS Fast NMOS (FF)
- Slow PMOS Slow NMOS (SS)
- Fast PMOS Slow NMOS (FPSN)
- Slow PMOS Fast NMOS (SPFN)
  - .lib ’90nm.lib’ TT
  - .lib `90nm.lib` FF
- Best Case
  - -25 ℃ / 1.1V / FF corner
- Worst Case
  - 125℃ / 0.9V / SS corner