Memory Circuits & Systems

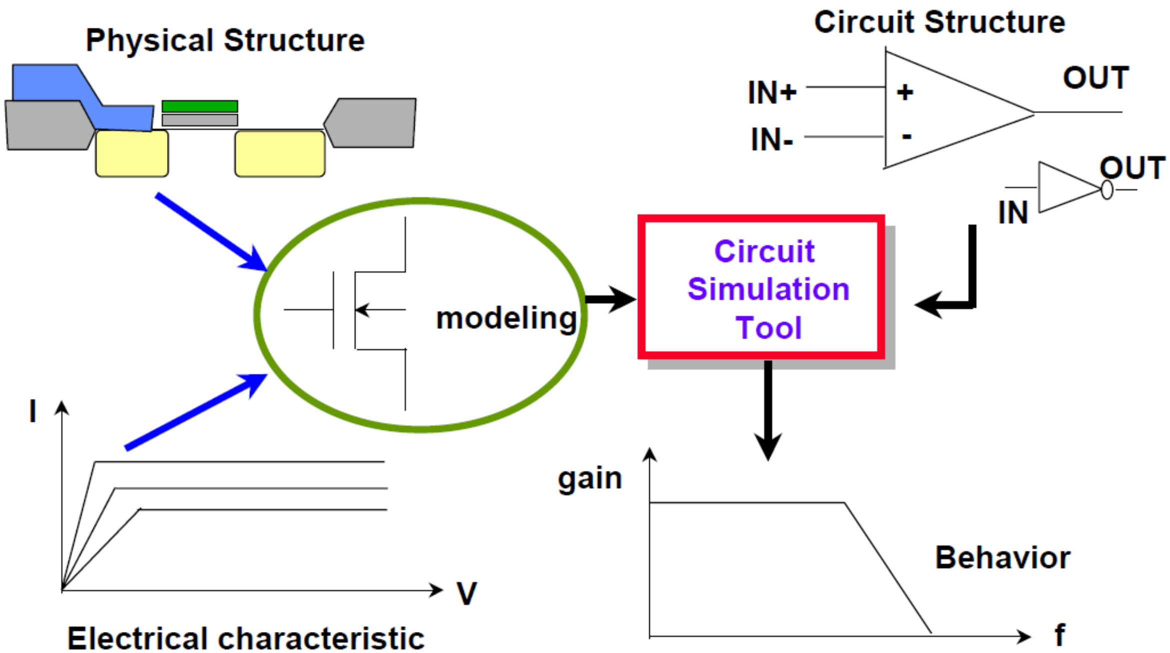
Design Tools
HSPICE

Professor Po-Tsang Huang

International College of Semiconductor Technology
National Yang Ming Chiao Tung University

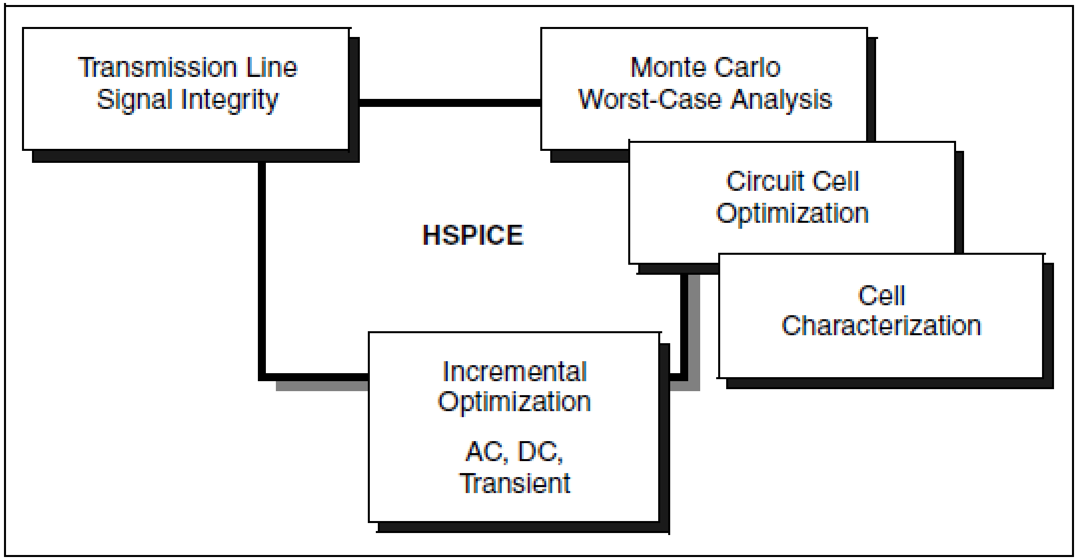
SPICE overview

- SPICE : Simulation Program with Integrated Circuit Emphasis
- Simulation : To predict the Circuit/System Characteristic after manufacturing
- Circuit simulation:



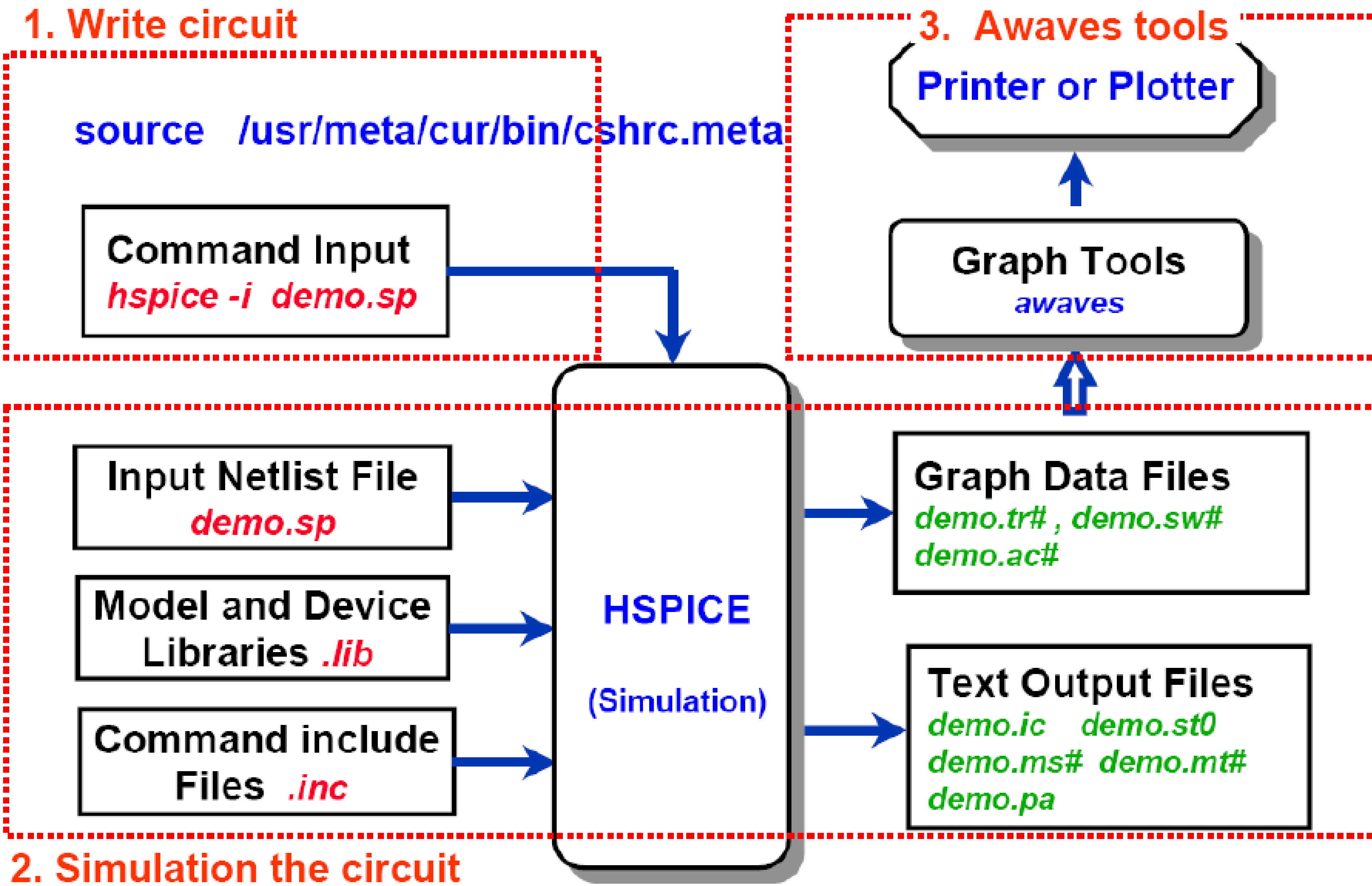
HSPICE Overview

- 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



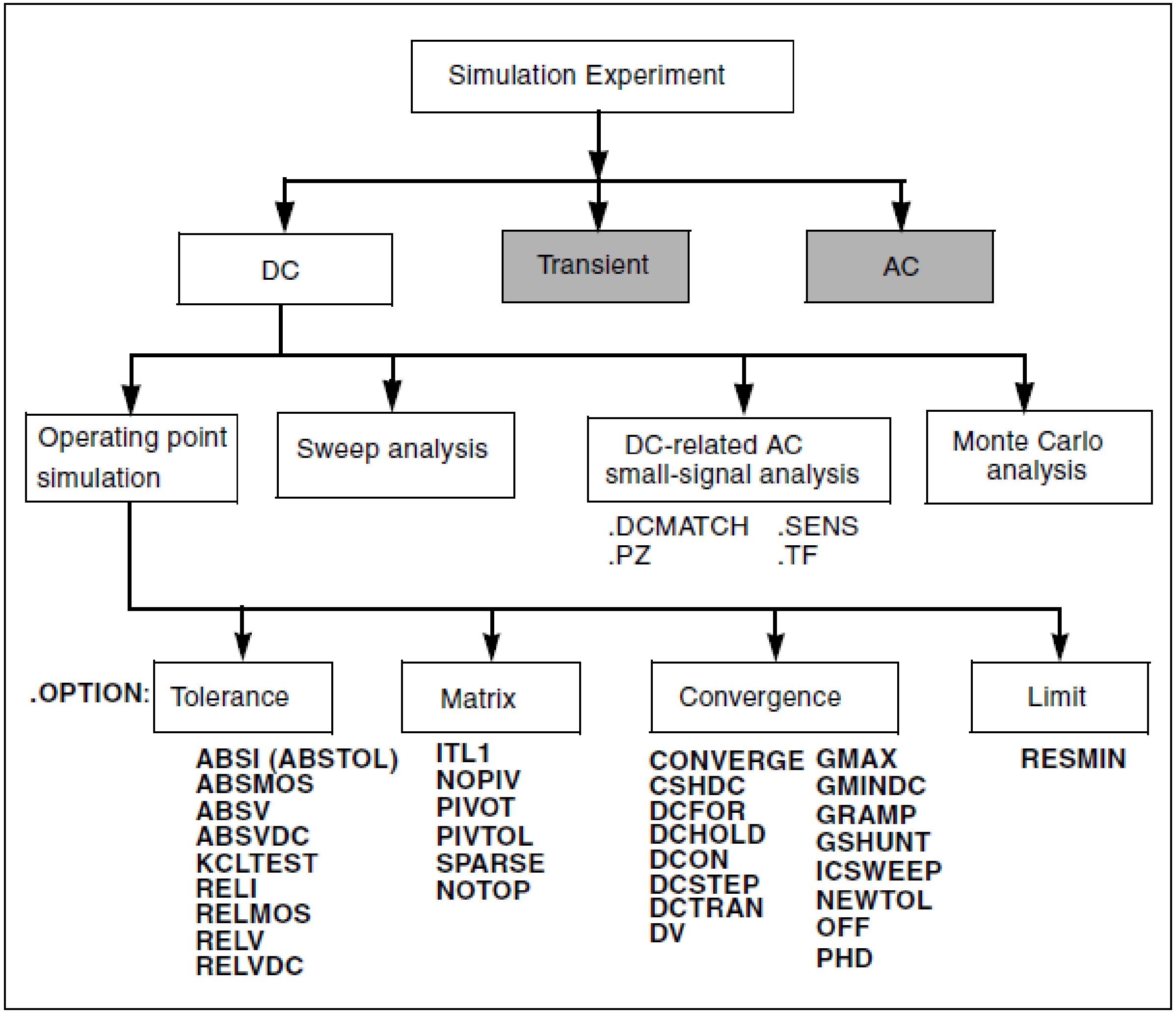
Sourced from : HSPICE® User Guide C-2009.03

HSPICE Data Flow



Simulation Modes

- Transient / DC Analysis / AC analysis



Netlist Structure for Spice

Title → Title Statement - Ignored during simulation

Models & Subckts → .model... or .LIB or .Subckt

Components →

```
c2  2  0   2pf
r1  1  0   1k
m1  1  2  3  4  mod  L=10u  W=30u
x3  2  3  INV
```

Sources →

```
v3  3  0 dc 0 ac 0 0 pulse 0 1 0 0.1 0.1 4 8
vin  in  0  sin(0 2 10k 0.5  0)
```

Controls →

```
.option nomod nopage
.tran 1 10
.print v(5) i(r1)
.plot v(3) v(in)
```

End file → .end

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
```

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

Library Input Statement

- .Include Statement

```
.Include '$installdir/parts/ad'
```

- .Lib Definition and Call Statement

```
.Lib TT  <-- Corner name
.Model nmos_tt nmos (level=49
Vt0=0.7+TNOM=27 .............)
.ENDL  TT
```

```
.Lib '~users/model/tsmc/logic06.mod' TT
```

- PROTECT <--Prevent the listing of included content

  .LIB '~users/model/tsmc/logic06.mod'' TT

  .UNPROTECT

Element Identifier & Scale Factors

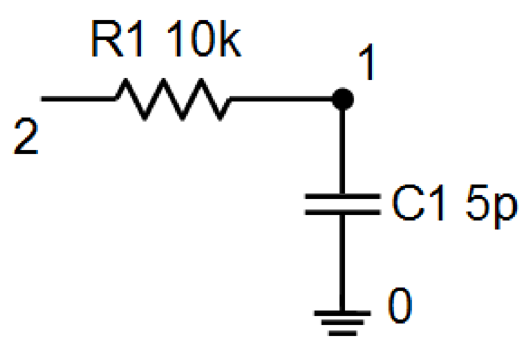
First Letter	Element
C	Capacitor
D	Diode
I	Current Source
L	Linear Inductor
M	MOS transistor
Q	Bipolar transistor
R	Resistor
V	Voltage Source
X	Subcircuit call

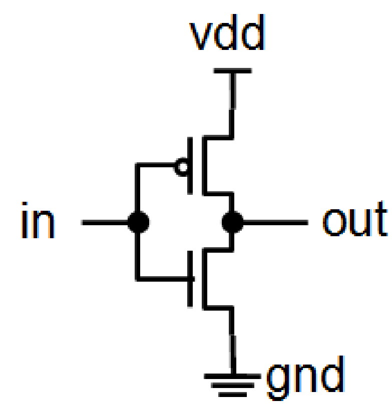
Prefix	Scale Factor Symbol	Factor
tera	T	1e+12
giga	G	1e+9
mega	Meg	1e+6
kilo	k	1e+3
milli	m	1e-3
micro	u	1e-6
nano	n	1e-9
pico	p	1e-12
femto	f	1e-15
atto	a	1e-18

Element Syntax

- Case insensitive!
- Passive components
  - Rxxx *node1 node2* value
  - C, L, etc.
  - ex, r1 1 2 10k

    c1 1 0 5p





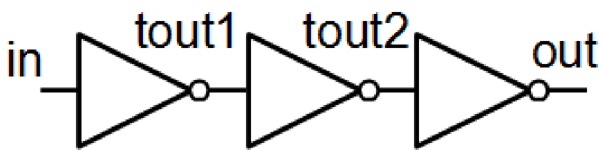
- Active component
  - Mxxx *Drain Gate Source Bulk Model* width length <multiple>
  - ex.

    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 …>*




```
.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
```

```
.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
```

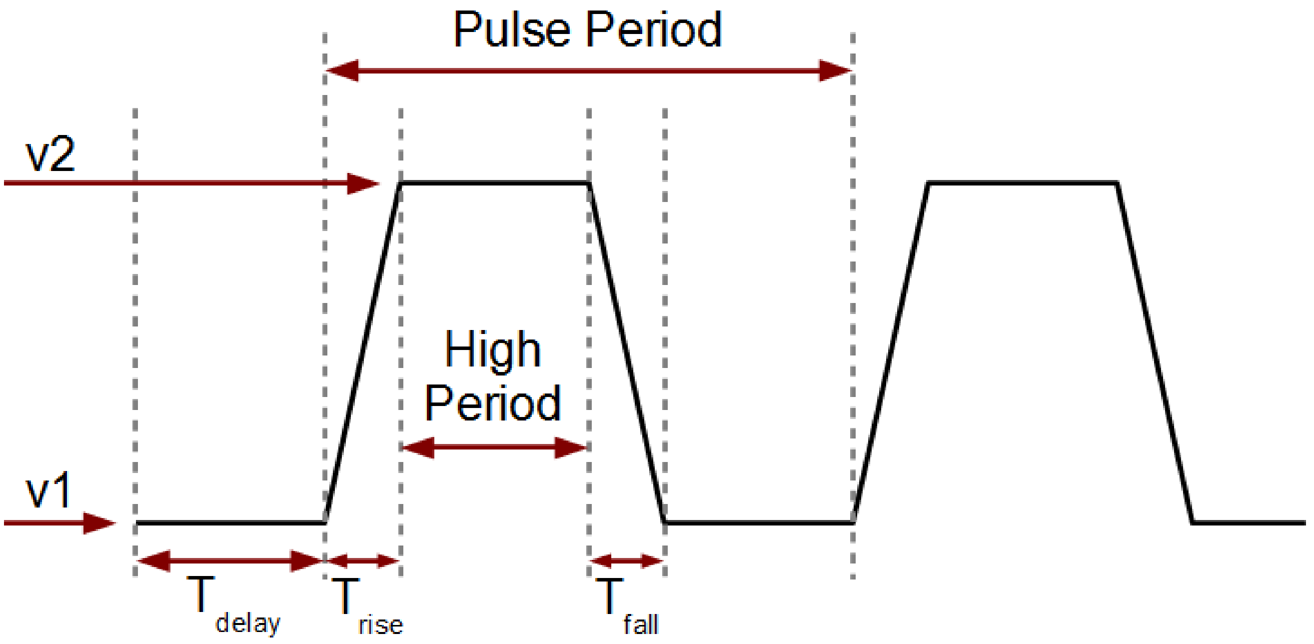
Independent Voltage and Current Sources

- 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

Pulse Voltage Source

- Vxxx node1 node2 PULSE(v1 v2 T_{delay} T_{rise} T_{fall} $P_{high\text{-}width}$ P_{period})

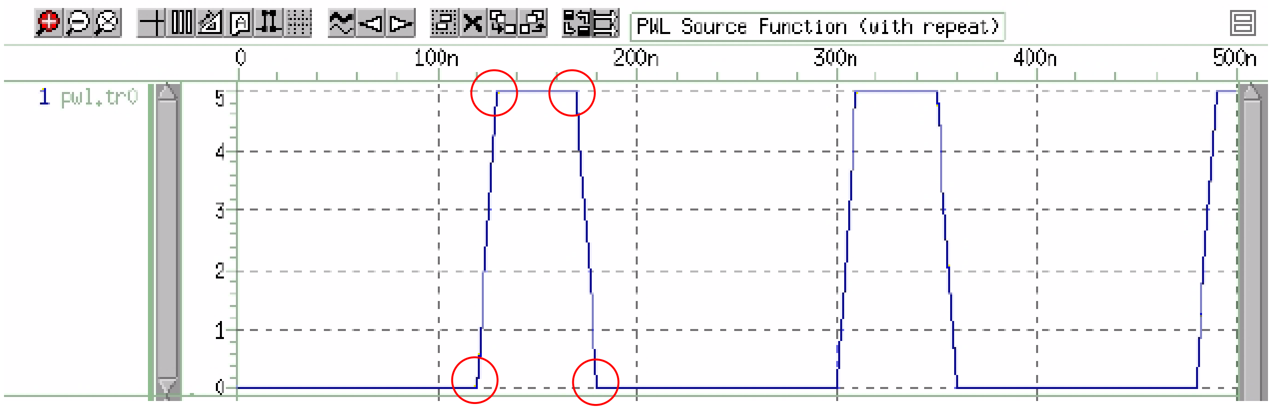
- Vin 1 0 PULSE(0v 5v 5ns 2ns 2ns 5ns 14ns)



Piecewise linear (PWL) Voltage Source

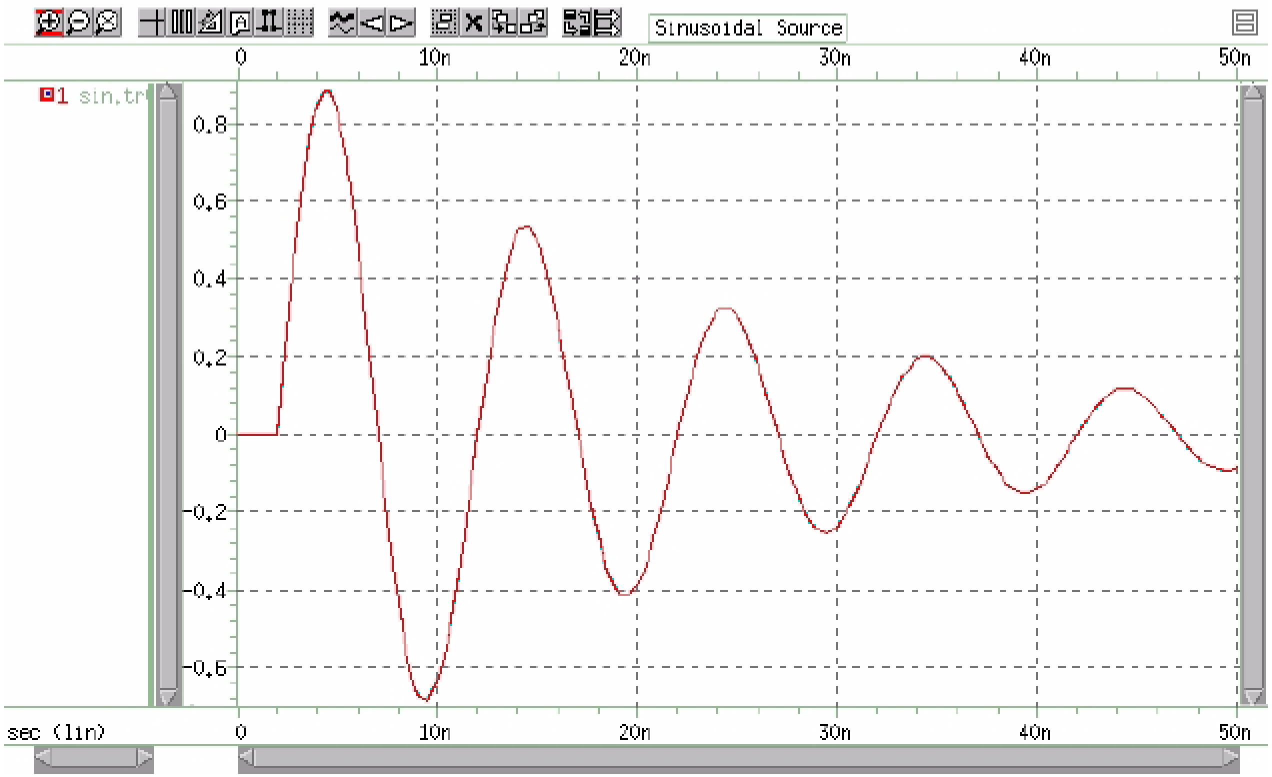
- Vxxx node1 node2 PWL(t1 v1, <t2 v2, t3 v3…> <R<=repeat>> <TD=delay>)

- 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)



Transient Analysis

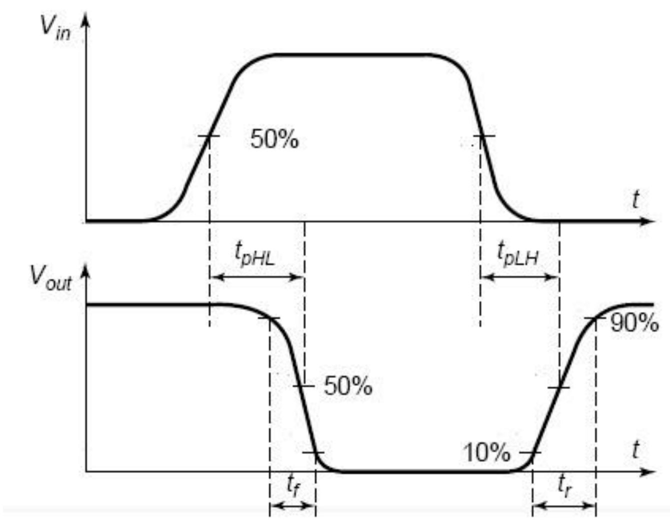
- .tran <timestep> <total simulation time>
- sweep <variable> <start value> <end value> <step>
- .tran 1p 5n
- .tran 1p 5n sweep temp 0 25 5

- .dc
- .ac
- HSPICE® User Guide: Simulation and Analysis

Measurements

- .meas(ure) *<mode> <name> <type description>*

- .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



Measurements

- .meas(ure) *<mode> <name> <type description>*
- Mode
  - DC / AC / TRAN
- Type
  - Trig, Targ / AVG / MAX / MIN / PP

- .measure dc vout find V(out1) when V(in)=0.9
- .measure tran Pavg avg I(out) from=5n to=50n

- Measured results are listed in .mt# file

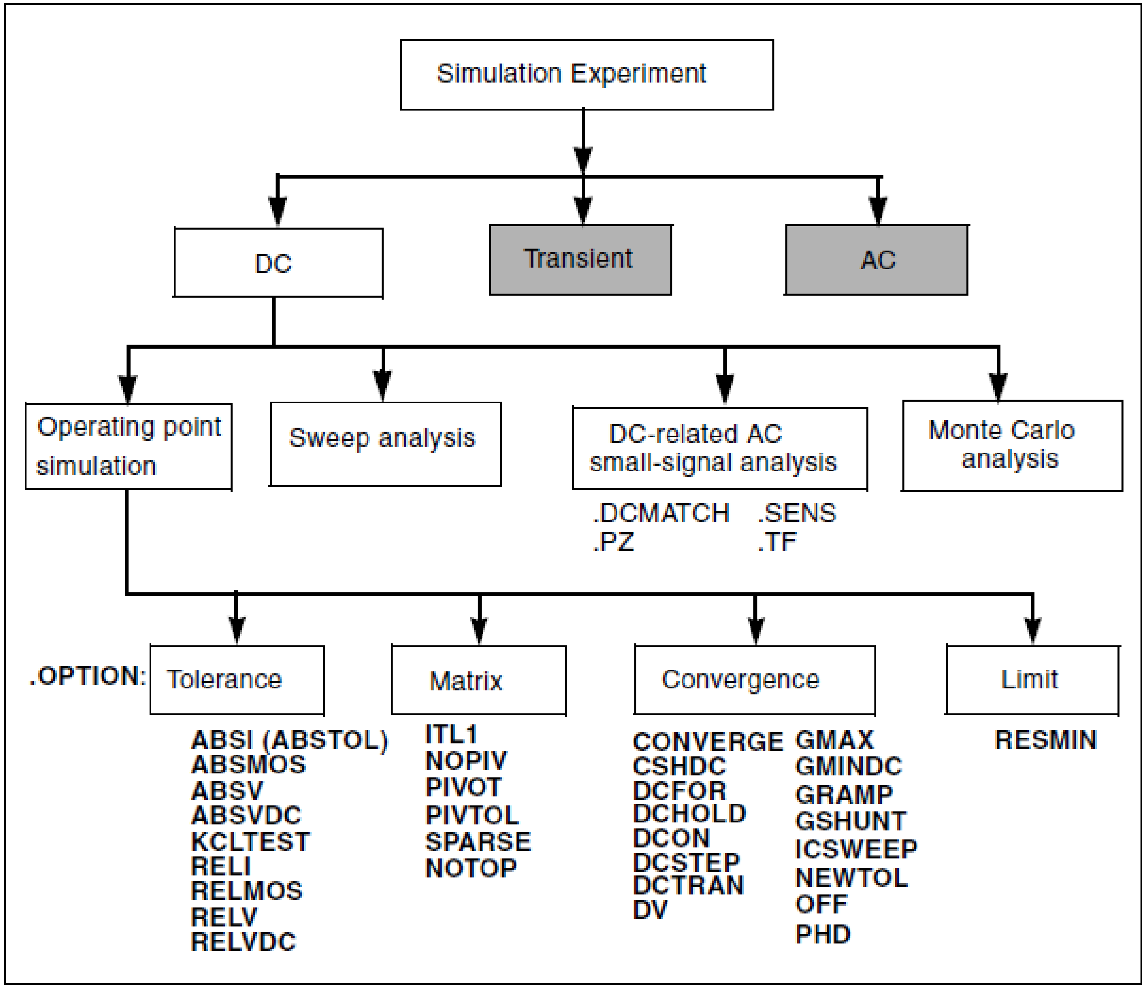
Multiple Analyses

- .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
```

Simulation Modes

- Transient / DC Analysis / AC analysis



DC Operating Point Analysis

- Initialization and analysis
  - First thing to set the DC operating point values for all nodes and sources : set capacitors OPEN & inductors SHORT
  - Using .IC or .NODESET to set the Initialized Calculation
- .OP statement print:
  - Node Voltages
  - Source Currents
  - Power Dissipation
  - Semiconductors Device Currents, Conductance, Capacitance

DC Sweep & DC Small Signal Analysis

- .DC Analysis : Syntax

```
.DC var1 start1 stop1 incr1 < var2  start2  stop2  incr2 > )
.DC var1 start1 stop1 incr1 <SWEEP var2 DEC/OCT/LIN/POI np start2 stop2 >)
```

- Examples

```
.DC VIN 0.25 5.0 0.25
.DC VDS 0 10 0.5 VGS 0 5 1
.DC TEMP -55 125 10
.DC TEMP POI 5 0 30 50 100 125
.DC xval 1K 10K 0.5K SWEEP TEMP LIN 5 25 125
.DC DATA=datanm SWEEP par1 DEC 10 1K 100K
.DC par1 DEC 10 1K 100K SWEEP DATA=datanm
```

AC Sweep & AC Small Signal Analysis

- .AC Analysis : Syntax

```
.AC  DEC/OCT/LIN/POI  np  fstart  fstop
.AC  DEC/OCT/LIN/POI  np  fstart  fstop  < SWEEP var start stop incr >
```

- Examples

```
.AC  DEC   10   1K   100MEG
.AC  LIN   100   1    100Hz
.AC  DEC  10    1      10K SWEEP Cload LIN 20
.AC  DEC  10    1      10K SWEEP Rx POI 2 5K 15K
.AC  DEC  10    1      10K SWEEP DATA=datanm
```

Transient Analysis

- .TRAN Analysis : Syntax

```
.TRAN tincr1 tstop1 <tincr2 tstop2 ...> <START=val>
.TRAN tincr1 tstop1 <tincr2 tstop2 ...> <START=val> UIC <SWEEP.. >
```

- Examples :

```
. TRAN  1NS  100NS
. TRAN  10NS  1US  UIC
. TRAN  10NS  1US  UIC  SWEEP TEMP -55 75 10 $ step=10
. TRAN  10NS  1US  SWEEP  load  POI  3  1pf  5pf  10pf
. TRAN  DATA=datanm
```

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
- Ex3:
  - .meas pwr avg POWER

Measurements(2/2)

- EX3:

```
.measure tpdr * rising prop delay
    + TRIG v(c) VAL='SUPPLY/2' FALL=1
    + TARG v(d) VAL='SUPPLY/2' RISE=1
.measure tpdf * falling prop delay
    + TRIG v(c) VAL='SUPPLY/2' RISE=1
    + TARG v(d) VAL='SUPPLY/2' FALL=1
.measure tpd param='(tpdr+tpdf)/2' * average prop delay
.measure trise * rise time
    + TRIG v(d) VAL='0.2*SUPPLY' RISE=1
    + TARG v(d) VAL='0.8*SUPPLY' RISE=1
.measure tfall * fall time
    + TRIG v(d) VAL='0.8*SUPPLY' FALL=1
    + TARG v(d) VAL='0.2*SUPPLY' FALL=1Measured
```

- results are listed in .mt# file

Output Files Summary (HSPICE)

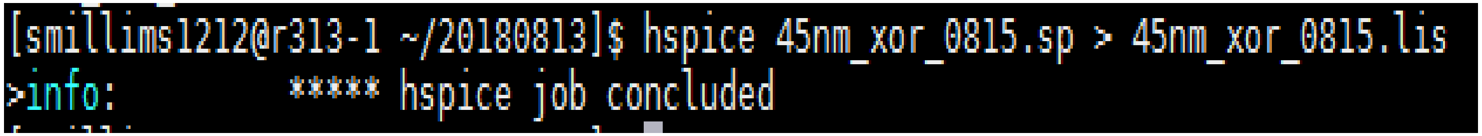
Output File Type	ExTensi
Output Lis	.lis #
DC Analysis Results	.sw #
DC Analysis Measurement Results	.ms #
AC Analysis Results	.ac #
AC Analysis Measurement Results	.ma #
Transient Analysis Results	.tr #
Transient Analysis Measurement Results	.mt #
Subcircuit Cross-Listing	.pa #
Operating Point Node Voltages (Initial Condition)	.ic

Output Statement

- Output Commands :
  - .PRINT : Print Numeric Analysis Results
  - .PLOT : Generates Low Resolution Plot in .lis file
  - .PROBE : Allows Save Output Variables Only into the Graph Date Files
  - .MEASURE : Print Numeric Results of Measured Specifications

Hspice Execution Command

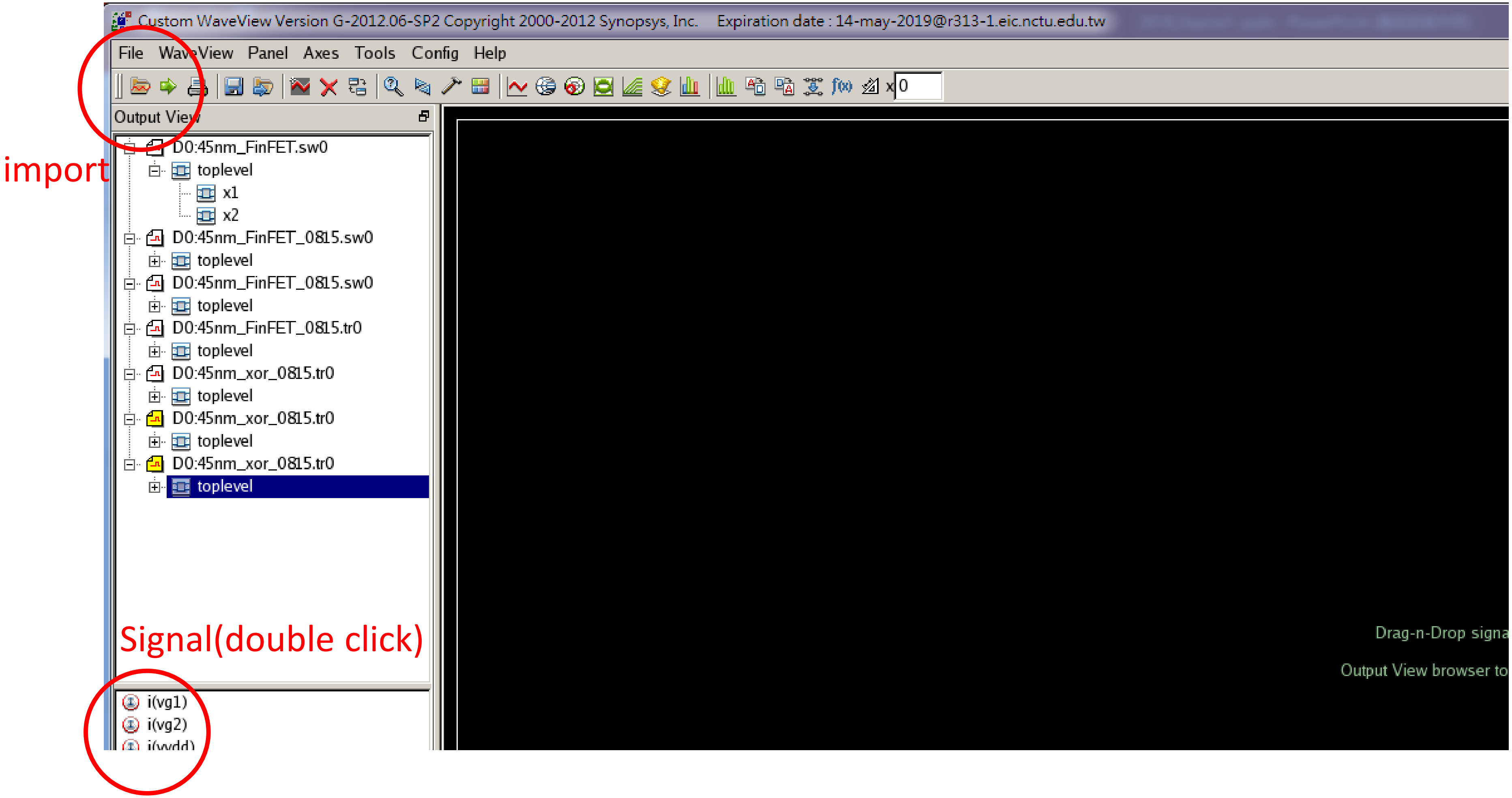
- HSPICE
  - %> hspice NETLIST.sp >NETLIST.lis
  - It will show Job Conclude!(or Job Abort!)
  - If abort, please see the result file(.lis) to find out the error



  - %> **hspice –i NETLIST.sp –o NETLIST.lis**

WaveView for Graphical Waveform

- %> **wv &**



Digital Vector File

- Vector Pattern Definition Section
  - Define Name、Magnitude 、I/O 、Time unit of stimulus
- Waveform Characteristic Section
  - Define Rise/Fall time 、Threshold voltage of signals
- Tabular Data Section
  - Define different logic level with corresponding time of stimulus

Vector Pattern

Syntax:

```
Radix     xxxx  xxxx
Vname     name[7:0]
IO        xxxx  xxxx
Tunit     ns
Period    xx
Trise     xx
Tfall     xx
Tdelay    xx
Vih   xx
Vil   xx

xxxx  xxxx
xxxx  xxxx
xxxx  xxxx
....
....
```

Example:

```
Radix     1111 1111
Vname     AAA[7:0]
IO        i i i i  i i i i
Tunit     ns
Period    10
Trise     0.5
Tfall     0.5
Tdelay    1
Vih       1.0
Vil       0.0

1010 1001  $ 0ns~ 10ns
0010 0111  $ 10ns~20ns
0110 0001  $ 20ns~30ns
....
....
```

Example:

```
Radix     4  4  1
Vname     A[3:0] B[3:0] C
IO        i    i   i
Tunit     ns
Period    10
Trise     0.5
Tfall     0.5
Tdelay    1
Vih       1.0
Vil       0.0

1111 1001  1 $ 0ns~ 10ns
0000 1111  1 $10ns~20ns
1111 0111  0 $ 20ns~30ns
....
....
```

.vec ‘file_name.vec’