SPICE (07)

國科會晶片系統設計製作中心

Jul. 2001

許文俊 Email: wjhsu@cic.edu.tw
林俊賓 Email: cplin@cic.edu.tw

Course Objectives

- Know Basic elements for circuit simulation
- Learn the basic usage of standalone spice simulators
- Know the concept of device models
- Learn the usage of waveform tools
- Advanced features of spice simulator

Further Reading

- **HSPICE 使用手冊**

  PDF格式檔：/usr/meta/cur/docs/hspice.pdf

- **HSPICE版本新增功能說明**

  PDF格式檔：/usr/meta/cur/docs/hspice00.4.RN.pdf

  PS格式檔 : /usr/meta/cur/docs/hspice00.4.RN.ps

- **HSPICE - awaves使用手冊**

  PDF格式檔：/usr/meta/cur/docs/avanwaves.pdf

  PS格式檔 : /usr/meta/cur/docs/ avanwaves.ps

- **SBTSPICE使用手冊**

  PDF格式檔：/usr/sbt/man/manu.pdf

  PS格式檔：/usr/sbt/man/manu.ps

- **SBTPLOT 使用手冊**

  PDF格式檔：/usr/sbt/man/plot.pdf

  PS格式檔：/usr/sbt/man/plot.ps

- **問題諮詢**

  http://www.cic.edu.tw

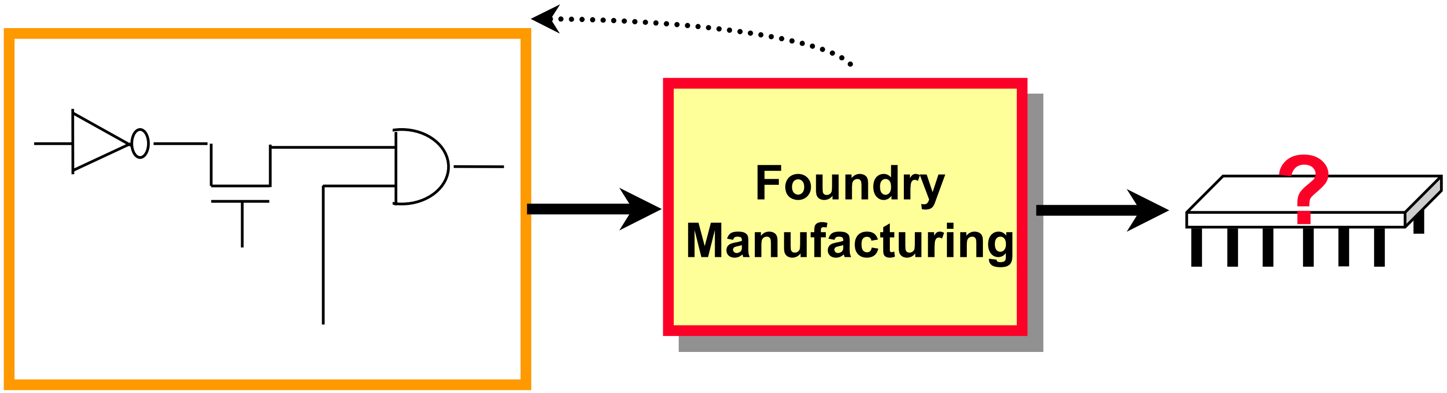
Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Circuit Design Background

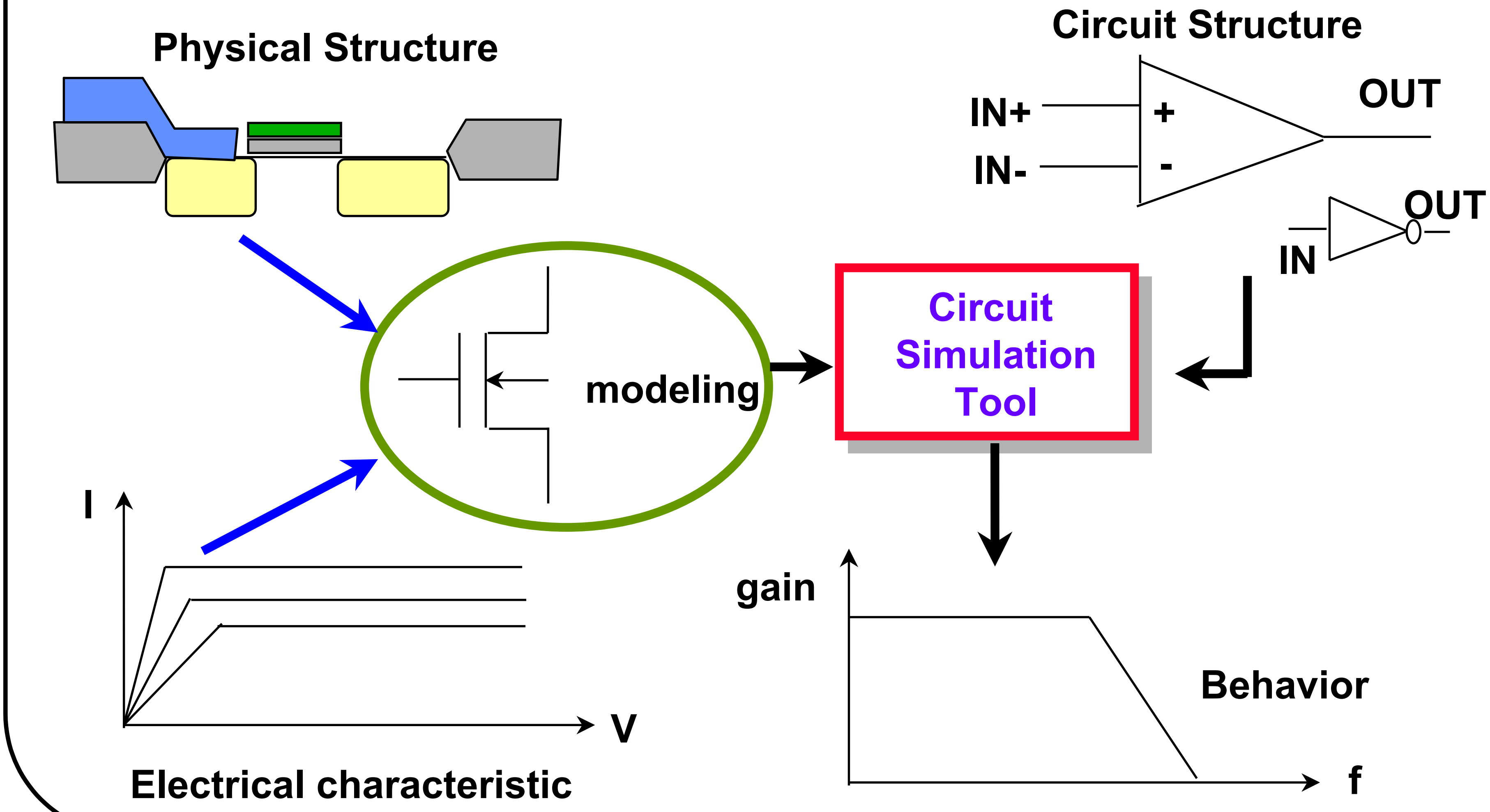
Circuit/System Design :

A procedure to construct a physical structure which is based on a set of basic component, and the constructed structure will provide a desired function at specified time/ time interval under a given working condition.



To predict the Circuit/System Characteristic after manufacture

(2). Circuit Simulation Background



(3). SPICE Background

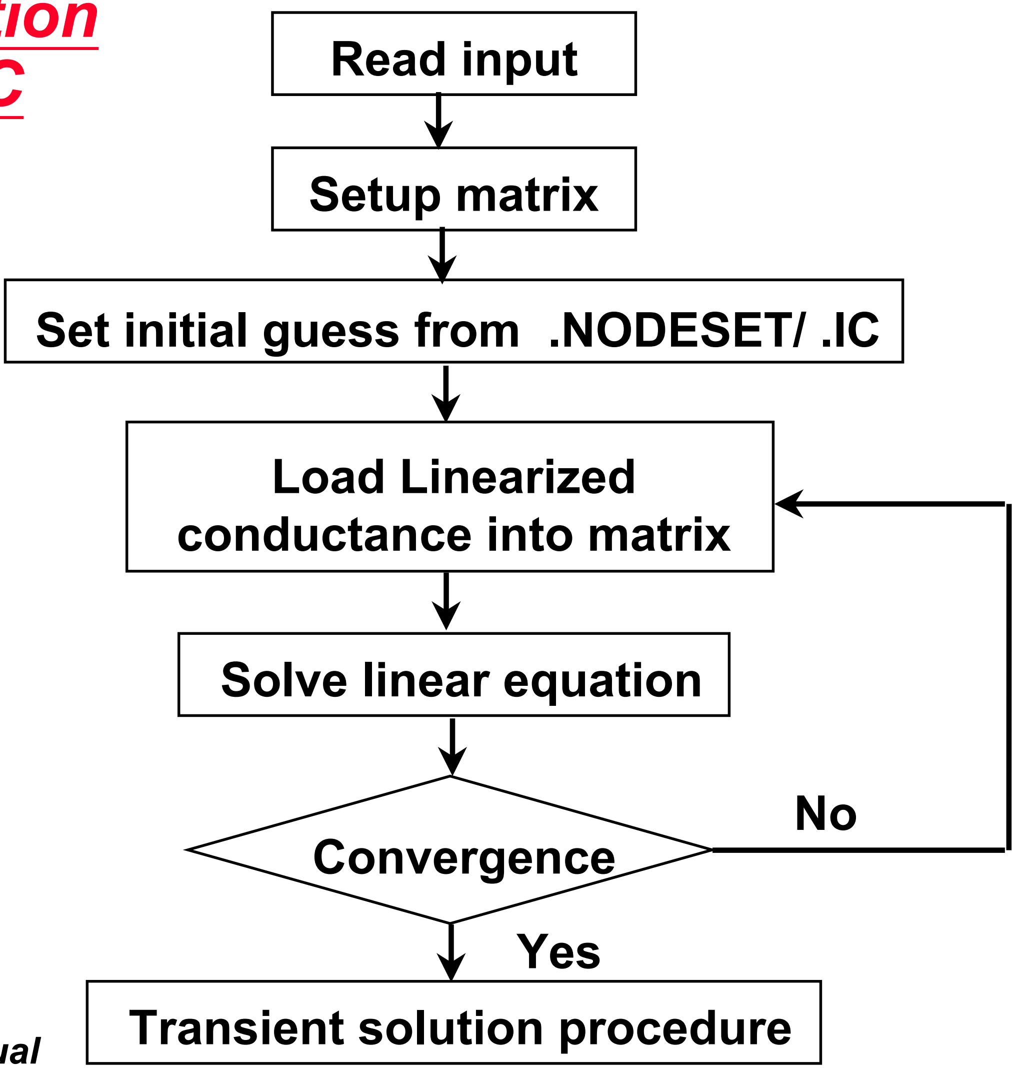
- SPICE : Simulation Program with Integrated Circuit Emphasis
  - Developed by University of California/Berkeley (UCB)
  - Successor to Earlier Effort *CANCER*
  - Widely Adopted, Become *De Facto* Standard
- Numerical Approach to Circuit Simulation
  - Circuit Node/Connections Define a Matrix
- Must Rely on Sub-Models for Behavior of Various Circuit Elements
  - Simple (e.g. Resistor)
  - Complex (e.g. MOSFET)

Source : IEEE 1997 CICC Educational Sessions, E3.3

(4). SPICE Introduction

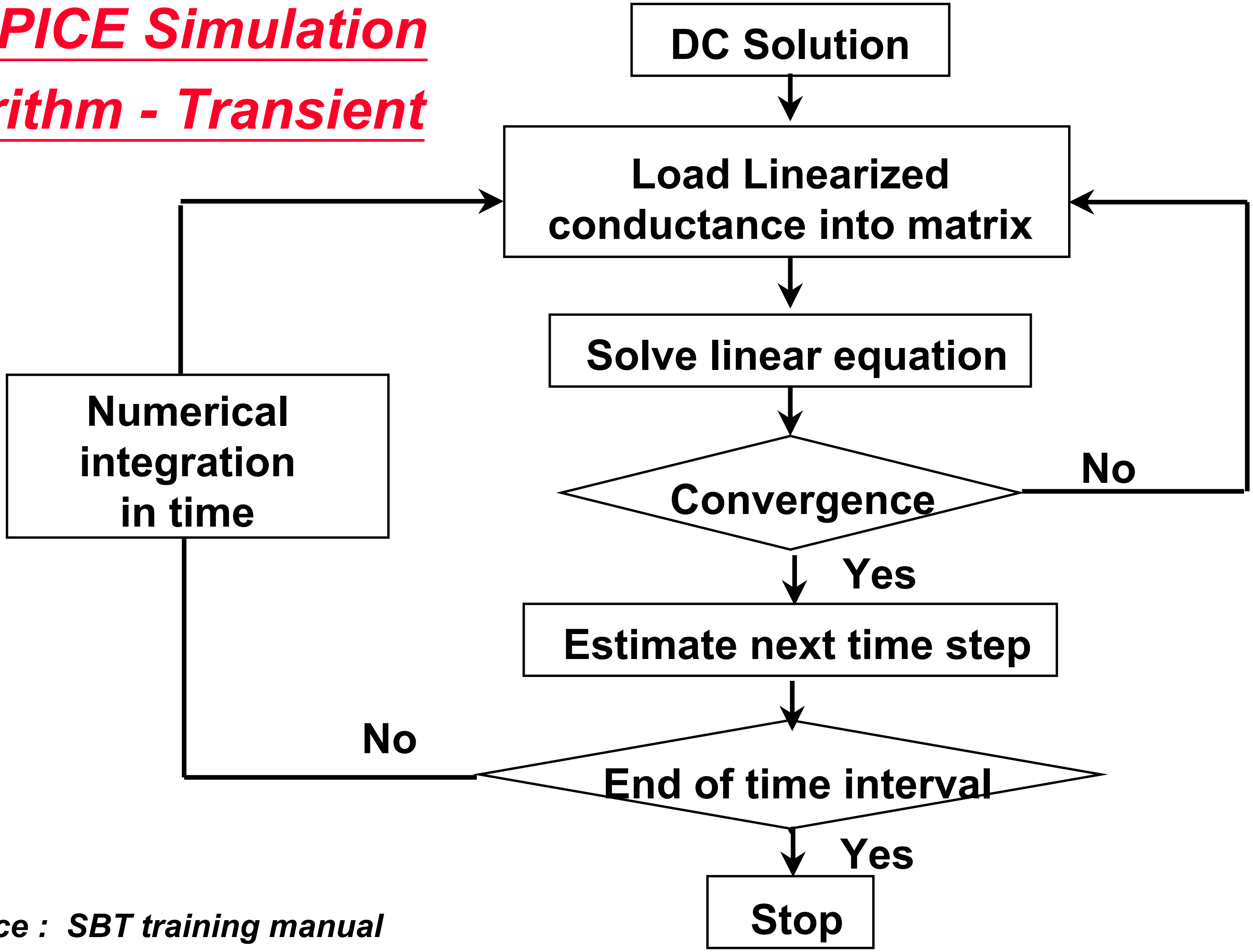
- SPICE generally is a Circuit Analysis tool for Simulation of Electrical Circuits in Steady-State, Transient, and Frequency Domains.
- There are lots of SPICE tools available over the market, SBTSPICE, HSPICE, Spectre, TSPICE, Pspice, Smartspice,ISpice ...
- Most of the SPICE tools are originated from Berkeley's SPICE program, therefore support common original SPICE syntax
- Basic algorithm scheme of SPICE tools are similar, however the control of time step, equation solver and convergence control might be different.

(5). SPICE Simulation Algorithm - DC


Read input
Setup matrix
Set initial guess from .NODESET/ .IC
Load Linearized conductance into matrix
Solve linear equation
Convergence
No
Yes
Transient solution procedure

Source : SBT training manual

(6). SPICE Simulation Algorithm - Transient



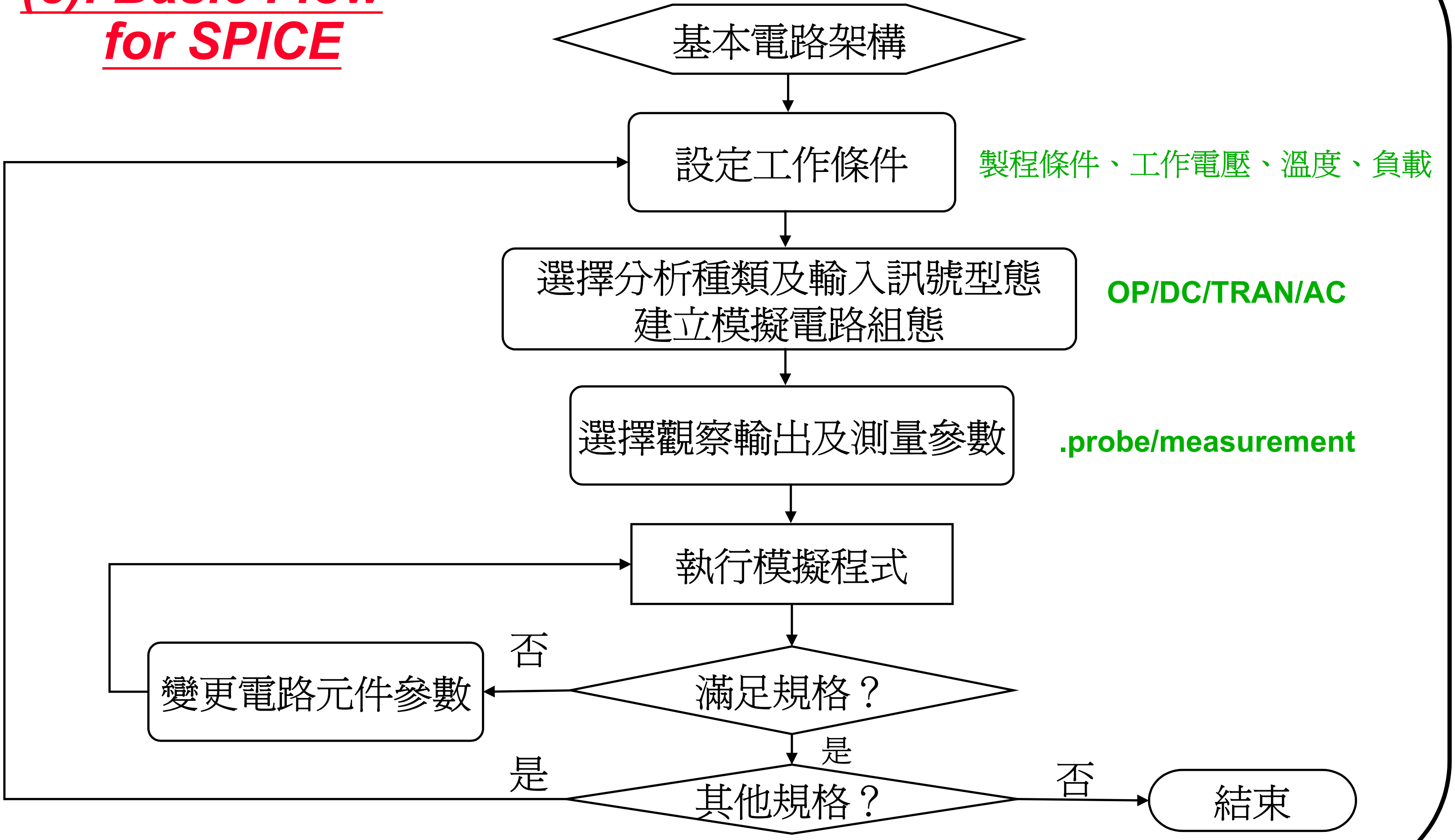
Source : SBT training manual

(7). Basics for Using SPICE Tools

SPICE 之外所需的基本概念

- 了解元件的基本特性
- 熟悉所設計電路的功能
- 了解需要驗證的電路規格及對應的模擬種類及電路組態
- 了解電路的輸入信號特性
- 了解電路各項規格的相依性及優先程度
- 了解電路元件參數與架構對各項電路特性的相關性，以利模擬結果的改進

(8). Basic Flow for SPICE



基本電路架構

設定工作條件

製程條件、工作電壓、溫度、負載

選擇分析種類及輸入訊號型態
建立模擬電路組態

OP/DC/TRAN/AC

選擇觀察輸出及測量參數

.probe/measurement

執行模擬程式

滿足規格？

否

變更電路元件參數

是

其他規格？

是

否

結束

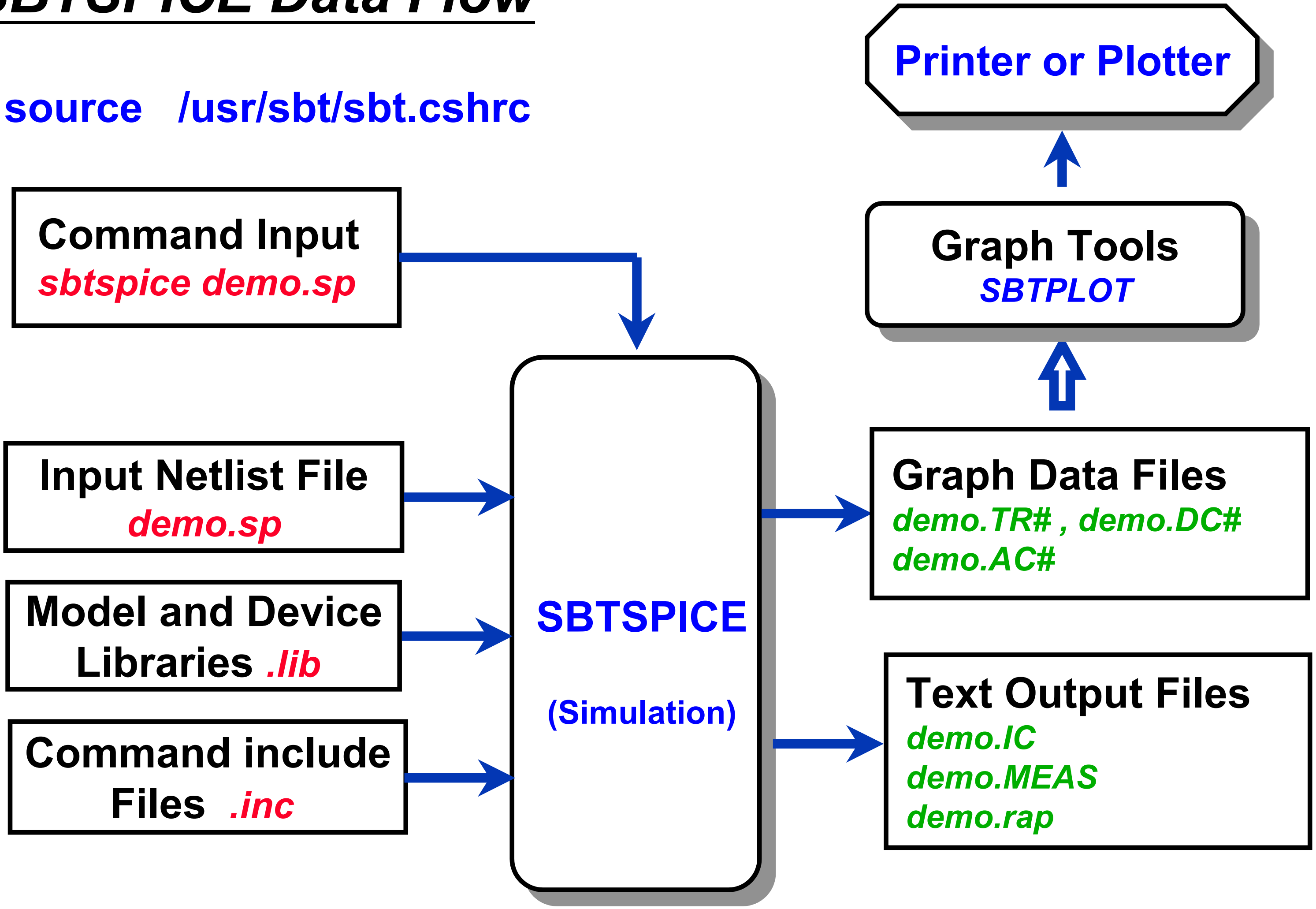
Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

2. Simulation Input and Controls

(1). SBTSPICE Data Flow

source /usr/sbt/sbt.cshrc



Printer or Plotter

Graph Tools
SBTPLOT

Command Input
sbtspice demo.sp

Input Netlist File
demo.sp

Model and Device Libraries *.lib*

Command include Files *.inc*

SBTSPICE

(Simulation)

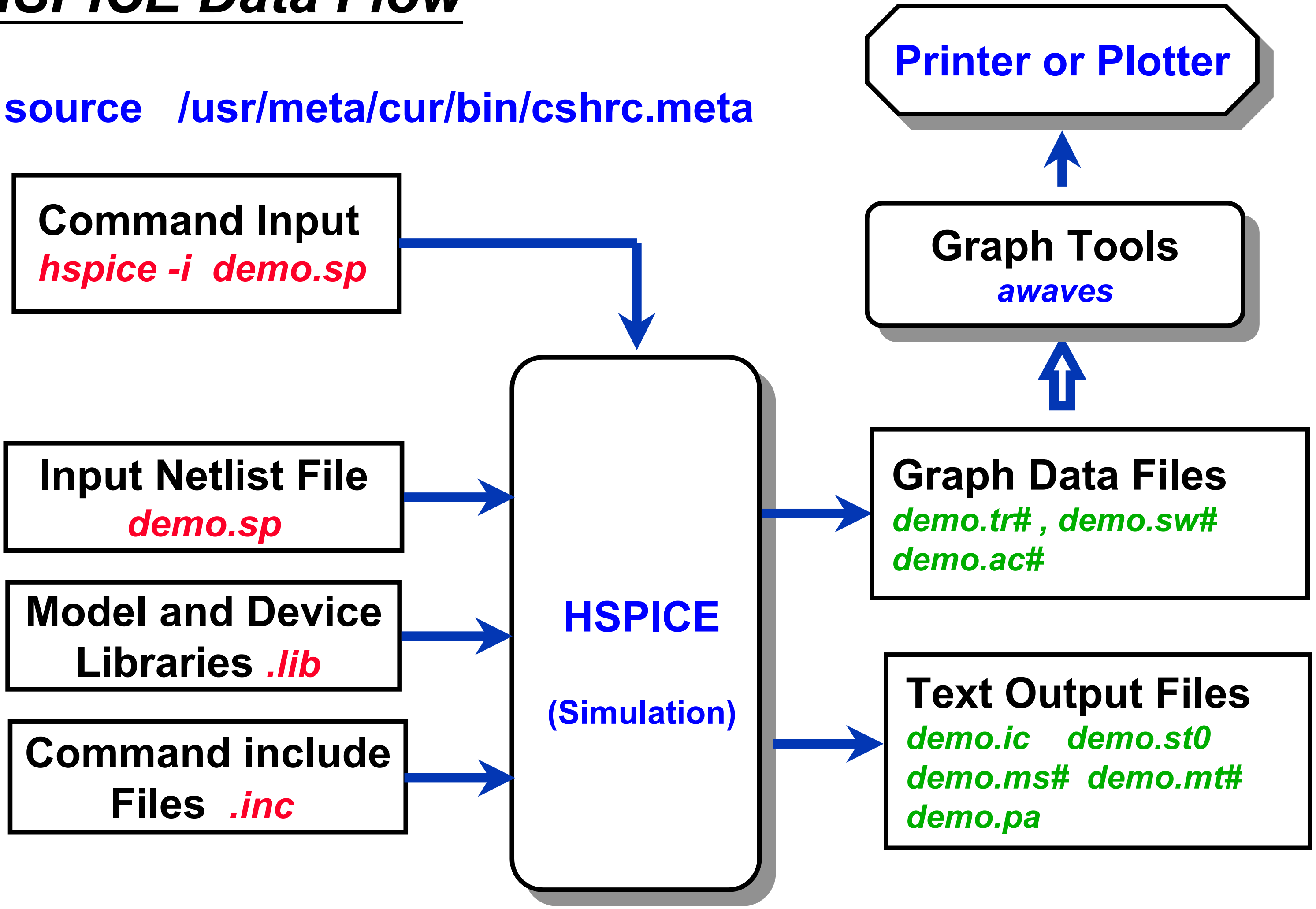
Graph Data Files
demo.TR# , demo.DC#
demo.AC#

Text Output Files
demo.IC
demo.MEAS
demo.rap

2. Simulation Input and Controls

(1). HSPICE Data Flow

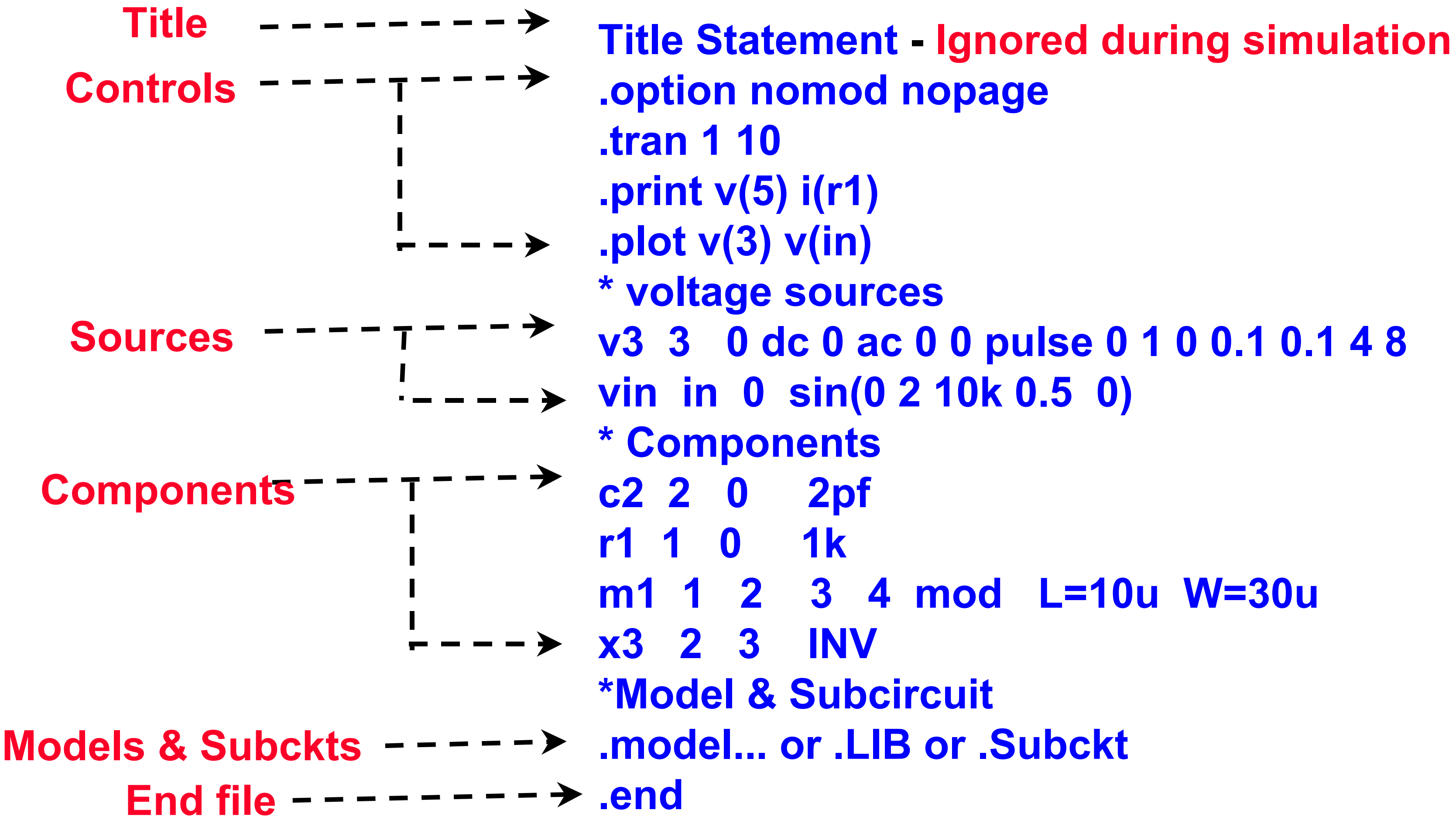
source /usr/meta/cur/bin/cshrc.meta



(2). Netlist Statements and Elements

TITLE	First line is Input Netlist File Title
*** or $**	Commands to Describe Circuit
.OPTIONS	Set Conditions for Simulation
Analysis(AC,DC,TRAN..) & .TEMP	Statements to Set Sweep Variables
.PRINT/.PLOT/.PROBE/.GRAPH	Set Print, Plot, and Graph Variables
.IC or **.NODESET**	Sets Initial State
.VEC \`digital_vector_file\`	Sets Input Stimuli File
Sources (I or V)	Sets Input Stimuli
Schematic Netlist	Circuit Description
+	In first Column ,+, is Continuation Char.
.SUBCKT/.ENDS	Sets/Ends Subcircuit Description
.MEASURE (Optimization Optional)	Provides Scope-like Measurement Capability
.LIB or .INCLUDE	Call Library or General Include Files
.MODEL Library	Element Model Descriptions
.DATA or **.PARAM**	Specify parameters or Parametric Variations
.ALTER	Sequence for In-line Case Analysis
.DELETE LIB	Remove Previous Library Selection
.END	Required Statement to Terminate Simulation

(3). Netlist Structure (SPICE Preferred)



Title

Controls

Sources

Components

Models & Subckts

End file

```
Title Statement - Ignored during simulation
.option nomod nopage
.tran 1 10
.print v(5) i(r1)
.plot v(3) v(in)
* voltage 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)
* Components
c2  2   0     2pf
r1  1   0     1k
m1  1   2     3   4  mod   L=10u  W=30u
x3   2   3    INV
*Model & Subcircuit
.model... or .LIB or .Subckt
.end
```

(4). Element and Node Naming Conventions

● **Node and Element Identification:**

- **Either Names or Numbers (e.g. data1, n3, 11, ...)**
- **0 (zero) is Always Ground**
- ★ **Trailing Alphabetic Character are ignored in Node Number, (e.g. 5A=5B=5)**
- **Ground may be 0, GND, !GND**
- **All nodes are assumed to be local**
- **Node Names can be may Across all Subcircuits by a .GLOBAL Statement (e.g. .GLOBAL VDD VSS)**

(4). Element and Node Naming Conventions(Cont.)

- **Instance and Element Names:**

C	Capacitor
D	Diode
E,F,G,H	Dependent Current and Voltage Controlled Sources
I	Current
J	JFET or MESFET
K	Mutual Inductor
L	Inductor
M	MOSFET
Q	BJT
R	Resistor
O,T,U	Transmission Line
V	Voltage Source
X	Subcircuit Call

- **Path Names of Subcircuits Nodes:** e.g. @x1.x2.mn[vth],@x1.x2.mn[id]
  V(X1.bit1), I(X1.X4.n3)

(5). Units and Scale Factors

- **Units:**

R	Ohm	(e.g. R1 n1 n2 1K)
C	Farad	(e.g. C2 n3 n4 1e-12)
L	Henry	(e.g. L3 n5 n6 1e-9)

- **Scale Factors :**

F	1e-15
P	1e-12
N	1e-19
U	1e-6
M	1e-3

K	1e3
Meg	1e6
G	1e9
T	1e12
DB	$20\log_{10}$

Examples:

1pF
1nH
10Meg Hz
vdb(v3)

Warning: in SBTSPICE 1.e-15F , will be interpreted as 1e-15 fento Farad

- **Technology Scaling :** All Length and Widths are in Meters

Using .options scale=1e-6 ➡ L=2 W=100

(6). Input Control Statements : .ALTER

● .ALTER Statement : Description

- Rerun a Simulation Several Times with Different

  Circuit Topology
  Models
  Elements Statement
  Parameter Values
  Options
  Analysis Variables, etc.

- 1st Run : Reads Input Netlist File up to the first .ALTER

- Subsequent : Input Netlists to next .ALTER, etc.

(6). Input Control Statements : .ALTER (Cont.)

- **.ALTER Statement : Example**

```
*file2: alter2.sp   alter examples        $ Title Statement
.lib 'mos.lib' normal
.param wval=50u Vdd=5V
r4   4   3    100
.
.
.alter
.del lib 'mos.lib' normal                 $ remove normal model lib
. lib 'mos.lib' fast                      $ get fast model lib
.alter
.temp -50 0 50                            $ run with different temperature
r4   4   3    1K                          $ change resistor value
c3   3   0    10p                         $add the new element
.param wval=100u Vdd=5.5V                 $ change parameters
.end
```

(6). Input Control Statements : .ALTER (Cont.)

- .ALTER Statement : Limitations
  - CAN Include:
    - ➡ Element Statement (Include Source Elements)
    - ➡ .DATA, .LIB, .INCLUDE, .MODEL Statements
    - ➡ .IC, .NODESET Statement
    - ➡ .OP, .PARAM, .TEMP, .TF, .TRAN, .AC, .DC Statements
  - CANNOT Include:
    - ➡ .PRINT, .PLOT, .GRAPH, or any I/O Statements

(7). Input Control Statements: .DATA

- **.DATA Statement: Inline or Multiline .DATA Example**

Inline .DATA Example

```
.TRAN 1n 100n SWEEP DATA=devinf
.AC DEC 10 hhz 100khz SWEEP DATA=devinf
.DC TEMP -55 125 10 SWEEP DATA=devinf
*
.DATA devinf  Width   Length   Vth   Cap
+              10u     100u     2v    5p
+              50u     600u    10v   10p
+             100u     200u     5v   20p
......
.ENDDATA
```

Multiline .DATA Example

```
.PARAM Vds=0  Vbs=0  L=1.0u
.DC DATA=vdot
.DATA vdot
Vbs   Vds     L
 0    0.1   1.0u
 0    0.1   1.5u
-1    0.1   1.0u
 0    0.5   1.0u
............
.ENDDATA
```

(8). Input Control Statements: .TEMP

● .TEMP Statement: Description

- When TNOM is not Specified, it will Default to 25 °C for HSPICE

  When TNOM is not Specified, it will Default to 27 °C for SBTSPICE

- Example 1:

```
.TEMP   30   $ Ckt simulated at 30 °C
```

- Example 2:

```
.OPTION  TEMP =  30   $ Ckt simulated at 30 °C
```

- Example 3:

```
.TEMP   100
D1  n1  n2  DMOD   DTEMP=30     $ D1  simulated at 130 °C
D2  n3  n4  DMOD                $ D2  simulated at 100 °C
R1  n5  n6      1K
```

HSPICE : DTEMP
SBTSPICE : TEMP

(8). Input Control Statements: .OPTION

● .OPTION Statement : Description

- ■ .Option Controls for
  - *Listing Formats*
  - *Simulation Convergence*
  - *Simulation Speed*
  - *Model Resolution*
  - *Algorithm*
  - *Accuracy*

- ■ .Option Syntax and Example

```
.OPTION opt1 <opt2> .... <opt=x>
```

```
.OPTION LVLTIM=2 POST PROBE SCALE=1
```

(8). Input Control Statements: .OPTION(Cont.)

● .OPTION Keywords Summary :

- ■ General Control Options
  - *Input, Output*
  - *CPU*
  - *Interfaces*
  - *Analysis*
  - *Error*
  - *Version*
- ■ DC Operating Point and DC Sweep Analysis
  - *Accuracy*
  - *Matrix*
  - *Input,Output*
  - *Convergence*
  - *Pole/Zero*
- ■ Model Analysis
  - *General*
  - *MOSFETs*
  - *Inductors*
  - *BJTs*
  - *Diodes*
- ■ Transient and AC Small Signal Analysis
  - *Accuracy*
  - *Speed*
  - *Timestep*
  - *Algorithm*
  - *Input, Output*

(9). Library Input Statement

- **.INCLUDE Statement** Copy the content of file into netlist

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

- **.LIB Definition and Call Statement** File reference and Corner selection

```
.LIB TT    ← Corner name
.MODEL nmos_tt nmos (level=49  Vt0=0.7
+TNOM=27 ..........................)
.ENDL  TT
```

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

← Corner name

- **.PROTECT** ← Prevent the listing of included contents

```
.PROTECT
.LIB "~users/model/tsmc/logic06.mod" TT
.UNPROTECT
```

(10). Hierarchical Circuits, Parameters, and Models

● .SUBCKT Statement : Description

■ .SUBCKT Syntax

.SUBCKT subname n1 <n2 n3...> <param=val...>

n1 ... Node Number for External Reference; Cannot be Ground node (0)
Any Element Nodes Appearing in Subckt but not Included in this list are Strictly LOCAL, with these Exceptions :

(1) Ground Node (0)

(2) Nodes Assigned using .GLOBAL Statement

(3) Nodes Assigned using BULK=node in MOSFET or BJT Models

param Used ONLY in Subcircuit, Overridden by Assignment in Subckt Call or by values set in .PARAM Statement

■ Subcircuit Calls (X Element Syntax)

```
.Xyyyy n1 <n2 n3...>  subname <param=val...>  <M=val>
.XNOR3   1    2   3    4     NOR   WN=3u   LN=0.5u  M=2
```

(10). Hierarchical Circuits, Parameters, and Models (Cont.)

- **.SUBCKT Statement : Examples**

```
.GLOBAL VDD
VDDA   VDD   0  VALUE
.PARAM VALUE=5V
:
:
.TRAN 1n 100n
*
.SUBCKT  INV   IN  OUT   WN=2u WP=8u
M1   OUT  IN  VDD  VDD  P  L=0.5u  W=WP
M2   OUT  IN   0     0     N  L=0.5u  W=WN
R1   OUT    4     1K
R2       4    5    10K
.ENDS    INV
*
X1   1      2     INV    WN=5u   WP=20u
X2   2      3     INV    WN=10u  WP=40u
*
.PRINT TRAN  V(2)  V(X1.4)  I(X2.M1)
```

(11). Example Circuit

```
              Invter gain
              .lib ’ls35_4_1.l'  tt
              .option   acct post
              .param  vref=1.0 Wmask=25u LMask=0.8u vcc=5
              .subckt inv out inp  d
              mn1 out inp  0 0  nch w=Wmask l=Lmask
              mp1 out inp  d d  pch w=Wmask l=Lmask
              .ends inv
subckt call → x1 out   inp  vdd  inv
              vdd vdd 0 dc vcc
              vin inp 0 dc 0 pulse(0 vcc  0 1ns 1ns 2ns 5ns)
              .dc vin   0  vcc 0.01 sweep data=d1
              .tran 0.1ns 10ns sweep data=d1
              .meas tran tpd trig v(inp) val=2 rise=1
              +                    targ v(out) val=3 fall=1
              .probe v(inp) v(out)
```

```
.data d1
Lmask Wmask
0.6u 250u
2.0u 420u
.enddata
.end
```

Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

Source / Stimuli ： 提供電路驅動來源

Source types

1. 固定值獨立電源

   提供固定偏壓或固定驅動電流

2. 時變/頻變 獨立電源

   提供變動的電壓或電流輸入，一般供輸入信號用

3. 時變/頻變 壓控/源控 相依電源

   提供可控制的電壓或電流源，一般供建立模型用

   壓控電壓源(VCVS)
   壓控電流源(VCCS)
   流控電壓源(CCVS)
   流控電流源(CCCS)

(1). Independent Source Elements: AC, DC Sources

● Source Element Statement :

- Syntax :

```
Vxxx n+ n-  < <DC=>dcval> <tranfun> <AC=acmag, <acphase>>
Iyyy n+ n-  < <DC=>dcval> <tranfun> <AC=acmag, <acphase> <M=val>
```

- Examples of DC & AC Sources :

```
V1   1   0   DC=5V
V2   2   0   5V
I3   3   0   5mA
```

```
V4   4    0   AC=10V, 90
V5   5    0   AC  1.0   180
```

*AC or Freq. Response Provide Impulse Response

- Examples of Mixed Sources :

```
V6   6    0   5V    AC=1V, 90
V7   7    0   0.5V  AC  1.0 SIN (0 1 1Meg)
```

(2). Independent Source Functions : Transient Sources

● Transient Sources Statement :

■ Types of Independent Source Functions :

- ★ Pulse (PULSE Function)
- ★ Sinusoidal (SIN Function)
- Exponential (EXP Function)
- ★ Piecewise Linear (PWL Function)
- Single-Frequency FM (SFFM Function)
- Single-Frequency AM (AM Function)

(2). Indep. Source Functions : Transient Sources(Cont.)

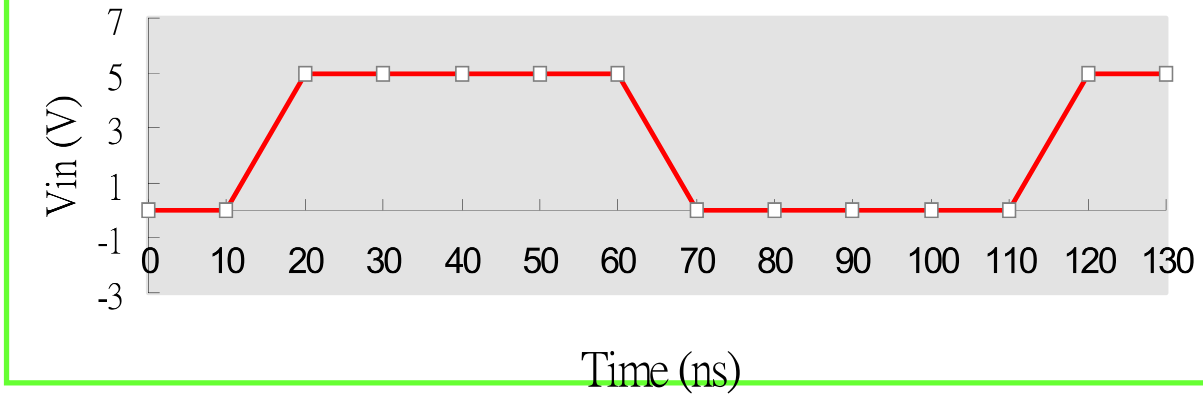
● Pulse Source Function : PULSE

■ **Syntax :**

PULSE (V1 V2 < Tdelay Trise Tfall Pwidth Period >)

■ **Example :**

Vin 1 0 PULSE (0V 5V 10ns 10ns 10ns 40ns 100ns)



(2). Indep. Source Functions : Transient Sources(Cont.)

● Sinusoidal Source Function : SIN

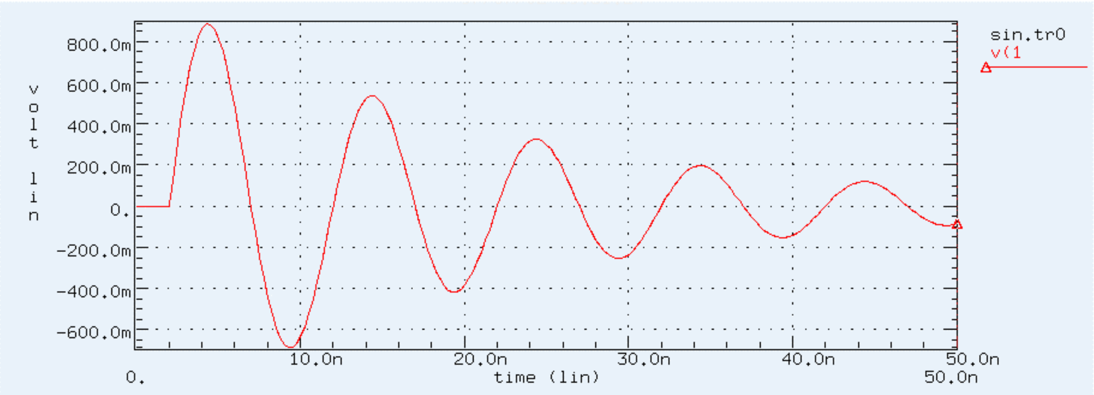
- **Syntax :**

SIN (Voffset Vacmag < Freq Tdelay Dfactor >)

$$\text{Voffset} + \text{Vacmag}^{*}\ e^{-(t-TD)\ ^{*}\text{Dfactor}}\ ^{*}\ \sin(2\pi\ \text{Freq}(t-TD))$$

- **Example :**

Vin 3 0 SIN (0V 1V 100Meg 2ns 5e7)



(2). Indep. Source Functions : Transient Sources(Cont.)

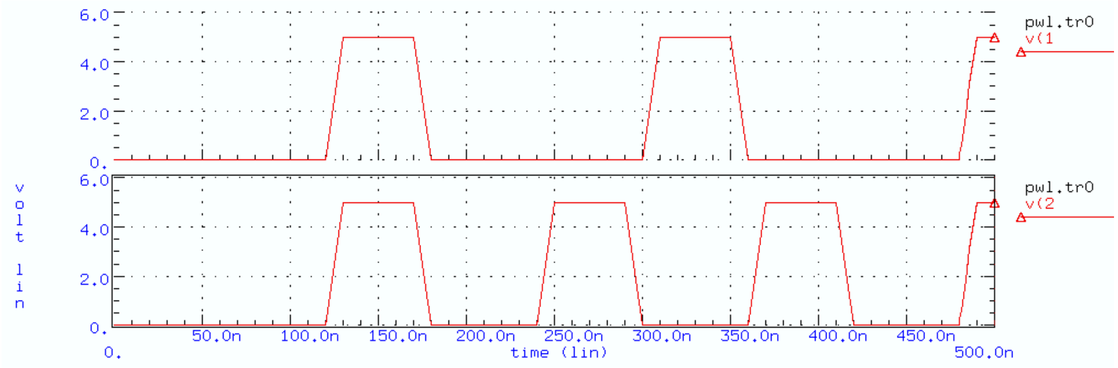
● Piecewise Linear Source Function : PWL or PL

- **Syntax :**

```
PWL  ( <t1 v1  t2 v2 ......> <R<=repeat>> <Tdelay=delay> )
$ R=repeat_from_what_time TD=time_delay_before_PWL_start
```

- **Example :**

```
V1  1  0  PWL  60n 0v, 120n 0v, 130n 5v, 170n 5v, 180n 0v, R 0
V2  2  0  PL  0v 60n, 0v 120n, 5v 130n, 5v 170n , 0v 180n , R 60n
```



(2). Indep. Source Functions : Transient Sources(Cont.)

● Specifying a Digital Vector File : .VEC

■ Syntax :

```
.VEC  ` digital_vector_file`
$ The digital vector file consists of three parts:
•  Vector Pattern Definition
•  Waveform Characteristics
•  Tabular Data
```

■ Digital Vector File Example :

```
; Vector Pattern
Radix  1 2 3 44
vname  v1 va[[1:0]] vb[3:1] vc[8:1]
io  i i i oo
tunit  ns
```

(2). Indep. Source Functions : Transient Sources(Cont.)

- Specifying a Digital Vector File : .VEC
  - Digital Vector File Example (Cont.) :

```
; Waveform Characteristics
slope  1.2
trise 0.2  1 3 7 FF
tfall  0.5  1 3 7 FF
tdealy 0.5 1 3 7 FF
vih 3.3  1 3 7 FF
vil  0.0  0 0 0 00
; Tabular Data
period 10
1 0 2 45
1 3 7 FF
```

(3). Voltage and Current Controlled Elements

- **Dependent Sources (Controlled Elements) :**
  - **Four Typical Linear Controlled Sources :**

Voltage Controlled Voltage Sources (VCVS) --- E Elements

Voltage Controlled Current Sources (VCCS) --- G Elements

Current Controlled Voltage Sources (CCVS) --- H Elements

Current Controlled Current Sources (CCCS) --- F Elements

E(name)	*N+*	*N-*	*NC+*	*NC-*	*(Voltage Gain Value)*
Eopamp	*3*	*4*	*1*	*2*	*1e6*
Ebuf	*2*	*0*	*1*	*0*	*1.0*

  - **Voltage Controlled Resistor (VCR) and Capacitor (VCCAP)**
  - ***Polynomial Controlled Sources POLY(1) ,POLY(2), POLY(3)***

Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Analysis Types & Orders

- Types & Order of Execution :
  - DC Operating Point : *First Calculated for ALL Analysis Types*

    .OP .IC .NODESET

  - DC Sweep & DC Small Signal Analysis :

    .DC .TF .PZ .SENS

  - AC Sweep & Small Signal Analysis :

    .AC .NOISE .DISTO .SAMPLE .NET

  - Transient Analysis:

    .TRAN .FOUR (UIC)

- Other Advanced Modifiers :
  - Temperature Analysis, Optimization

(2). Analysis Types : 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
- If UIC Included in .TRAN ==> Transient Analysis Started Directly by Using Node Voltages Specified in .IC Statement
- .NODESET Often Used to Correct Convergence Problems in .DC Analysis
- .IC force DC solutions, however .NODESET set the initial guess

.OP Statement :

- .OP Print out :(1). Node Voltages; (2). Source Currents; (3). Power Dissipation; (4). Semiconductors Device Currents, Conductance, Capacitance

(3). Analysis Types : DC Sweep & DC Small Signal Analysis

● DC Analysis Statements :

- ■ .DC : Sweep for Power Supply, Temp., Param., & Transfer Curves
- ■ .OP : Specify Time(s) at which Operating Point is to be Calculated
- ■ .TF : Calculate DC Small-Signal Transfer Function (.OP is not Required)
- ■ .PZ : Performs Pole/Zero Analysis (.OP is not Required)

● .DC Statement Sweep :

- ■ Any Source Value
- ■ Temperature Value
- ■ DC Circuit Optimization
- ■ Any Parameter Value
- ■ DC Model Characterization

Sweep over model parameter is not allowed
Monte Carlo sweep is not supported in SBTSPICE

(3). Analysis Types : DC Sweep & DC Small Signal Analysis (Cont.)

- .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
```

(4). Analysis Types : AC Sweep & Small Signal Analysis

● AC Analysis Statements :

- .AC : Calculate Frequency-Domain Response
- .NOISE : Noise Analysis

● .AC Statement Sweep :

- Frequency
- Temperature
- Optimization
- Element
- .param Parameter

(4). Analysis Types : AC Sweep &Small Signal Analysis (Cont.)

- .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  1pf  10pf
.AC  DEC  10  1  10K  SWEEP  Rx  POI  2  5K  15K
.AC  DEC   10  1  10K  SWEEP  DATA=datanm
```

(4). Analysis Types : AC Sweep &Small Signal Analysis (Cont.)

● Other AC Analysis Statements:

■ .NOISE Statement : Only one noise analysis per simulation

```
.NOISE  v(5)  VIN  10          $  output-variable, noise-input reference, interval
```

V(5) <- node output at which the noise output is summed
VIN <- noise input reference node
10 <- interval at which noise analysis summary is to be printed

(5). Analysis Types : Transient Analysis

- **Transient Analysis Statements :**
  - .TRAN : Calculate Time-Domain Response
  - .FOUR : Fourier Analysis
  - .FFT : Fast Fourier Transform
- **.TRAN Statement Sweep :**
  - Temperature
  - Optimization
  - .Param Parameter

(5). Analysis Types : Transient Analysis (Cont.)

- **.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
```

(5). Analysis Types : Transient Analysis (Cont.)

● Other Transient Analysis Statements:

■ .FOUR Statement :

```
.FOUR  100K  V(5)  V(7,8)          $  fundamental-freq , output-variable1,2,.....
```

Note1: As a part of Transient Analysis

Note2: Determines DC and first Nine AC Harmonics & Reports THD (%)

■ .FFT Statement :

```
.FFT v(1,2) np=1024 start=0.3m stop=0.5m freq=5K window=Kaiser alfa=2.5
```

Note1: Window Types : RECT, BLACK, HAMM, GAUSS, KAISER, HINN....

Note2: Determines DC and first Ten AC Harmonics & Reports THD (%)

Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Output Files Summary:

Output File Type	Extension
Output Listing	on screen
DC Analysis Results	.DC#
DC Analysis Measurement Results	.MEAS#
AC Analysis Results	.AC#
AC Analysis Measurement Results	.MEAS#
Transient Analysis Results	.TR#
Transient Analysis Measurement Results	.MEAS#
Subcircuit Cross-Listing	.PA#
Operating Point Node Voltages (Initial Condition)	.IC

(1). 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

(2). Output Statements:

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

- Output Variables:
  - DC and Transient Analysis : Displays Individual Voltage, Current, & Power
  - AC Analysis : Display Real & Imag. Components of Voltage & Current.....
  - Element Template Analysis : Display Element-Specific Voltage, Current.....
  - .MEASURE : Display User-Defined Variables Defined in .MEAS Statement

(3). Output Variable Examples: DC, Transient, AC, Template

DC & Transient Analysis :

- Nodal Voltage Output : V(1), V(3,4), V(X3.5)
- Current Output (Voltage Source) : I(VIN), I(X1.VSRC)
- Current Output (Element Branches) : I2(R1), I1(M1), I4(X1.M3)

AC Analysis :

- AC : V(2), VI(3), VM(5,7), VDB(OUT), IP(9), IP4(M4)

R : Real

I : Imaginary

M : Magnitude

P : Phase

DB : Decibels

Element Template :

- @x1.mn1[vth]
- @x1.mn1[gds]
- @x1.mn1[gm],@x1.mn1[gbs],@x1.mn1[cgd]

(4). Regional Analysis of Power for Transient Analysis

.option rap = x <Rap_Tstart=Tstart><Rap_Tstop=Tstop>

0 < x < 1 , The nodes with average power consumption greater than (1-x)*(total power consumption) will be listed

x = 1 will dump all power information of nodes

Tstart is the start time for power report, default is 0
Tstop is the stop time for power report, default is simulation stop time

All RAP output is stored in file .rap

(5). Output Variable Examples: Parametric Statements

- **Algebraic Expressions for Output Statements:**
  - .PRINT DC V(IN) V(OUT) PAR('V(OUT)/V(IN)')
  - .PROBE AC Gain=PAR('VDB(5)-VDB(2)') Phase=PAR('VP(5)-VP(2)')
- **Other Algebraic Expressions :**
  - Parameterization : .PARAM WN=5u LN=10u VDD=5.0V
  - Algebra : .PARAM X='Y+5'
  - Functions : .PARAM Gain(IN, OUT)='V(OUT)/V(IN)'
  - Algebra in Element : R1 1 0 r='ABS(V(1)/I(M1))+10'
- **Built-In Functions :**

sin(x) cos(x) tan(x) asin(x) acos(x) atan(x) sinh(x) tanh(x) abs(x)
sqrt(x) log(x) log10(x) exp(x) db(x) min(x,y) max(x,y) power(x,y)...

(6). Displaying Simulation Results: .PRINT & .PLOT

- Syntax :

```
.PRINT anatype ov1 <ov2 ov2...>
Note : .PLOT with same Syntax as .PRINT, Except Adding <pol1, phi1>
        to set plot limit
```

- Examples :

```
.PRINT  TRAN  V(4)  V(X3.3)  P(M1)  P(VIN)  POWER  PAR('V(OUT)/V(IN)')
.PRINT  AC  VM(4,2)  VP(6)  VDB(3)
.PRINT  AC  INOISE  ONOISE  VM(OUT)  HD3
.PRINT  DISTO  HD3  HD3(R)  SIM2
.PLOT  DC  V(2)  I(VSRC)  V(37,29)  I1(M7)  BETA=PAR('I1(Q1)/I2(Q1)')
.PLOT  AC  ZIN  YOUT(P)  S11(DB)  S12(M)  Z11(R)
.PLOT  TRAN  V(5,3) (2,5)  V(8) I(VIN)
```

(7). Displaying Simulation Results: .PROBE & .GRAPH

● .PROBE Statement :

.PROBE Syntax : .PROBE anatype ov1 <ov2 ov2...>

Note 1 : .PROBE Statement Saves Output Variables into the Interface & Graph Data Files

Note 2 : Set .OPTION PROBE to Save Output Variables Only, Otherwise HSPICE Usually Save All Voltages & Supply Currents in Addition to Output Variables

(8). Output Variable Examples: .MEASURE Statement

General Descriptions :

- .MEASURE Statement Prints User-Defined Electrical Specifications of a Circuit and is Used Extensively in Optimization
- .MEASURE Statement Provides Oscilloscope-Like Measurement Capability for either AC , DC, or Transient Analysis
- Using .OPTION AUTOSTOP to Save Simulation Time when TRIG-TARG or FIND-WHEN Measure Functions are Calculated

Fundamental Measurement Modes :

- Rise, Fall, and Delay (TRIG-TARG)
- AVG, RMS, MIN, MAX, & Peak-to-Peak (FROM-TO)
- FIND-WHEN

(9). MEASURE Statement : Rise, Fall, and Delay

- Syntax :

```
.MEASURE  DC|AC|TRAN  result_var  TRIG ... TARG ... <Optimization Option>
```

- ***result_var*** *: Name Given the Measured Value in HSPICE Output*
- ***TRIG ...*** *: TRIG trig_var VAL=trig_value <TD=time_delay> <CROSS=n>*
  +   <RISE=r_n> <FALL=f_n|LAST>
- ***TRIG ...*** *: TRIG AT=value*
- ***TARG ...*** *: TARG targ_var VAL=targ_value <TD=time_delay>*
  +   <CROSS=n|LAST> <RISE=r_n|LAST> <FALL=f_n|LAST>
- ***<Optimization Option>*** *: <GOAL=val> <MINVAL=val> <WEIGHT=val>*

- Example:

```
.meas  TRAN  tprop  trig  v(in)  val=2.5  rise=1   targ  v(out)  val=2.5  fall=1
```

(10). MEASURE Statement : AVG, RMS, MIN, MAX, & P-P

● Syntax :

```
.MEASURE  DC|AC|TRAN  result  FUNC  out_var  <FROM=val1>  <TO=val2>
+                                   <Optimization Option>
```

- *result_var* : *Name Given the Measured Value in HSPICE Output*
- *FUNC* : *AVG ----- Average*   *MAX ----- Maximun*   *PP ---- Peak-to-Peak*
  MIN ------ Minimum   *RMS ----- Root Mean Square*
- *out_var* : *Name of the Output Variable to be Measured*
- *<Optimization Option>*: *<GOAL=val> <MINVAL=val> <WEIGHT=val>*

● Example:

```
.meas  TRAN  minval       MIN   v(1,2)    from=25ns   to=50ns
.meas  TRAN  tot_power    AVG   power     from=25ns   to=50ns
.meas  TRAN  rms_power    RMS   power
```

(11). MEASURE Statement : Find & When Function

● Syntax :

```
.measure  DC|AC|TRAN  result  WHEN  ... <Optimization Option>
.measure  DC|AC|TRAN  result  FIND out_var1 WHEN ...<Optimization Option>
.measure  DC|AC|TRAN  result_var  FIND out_var1 AT=val <Optimization Option>
```

- **result** : Name Given the Measured Value in HSPICE Output
- **WHEN ...** : WHEN out_var2=val|out_var3 <TD=time_delay>
  + <CROSS=n|LAST> <RISE=r_n|LAST> <FALL=f_n|LAST>
- **<Optimization Option>** : <GOAL=val> <MINVAL=val> <WEIGHT=val>

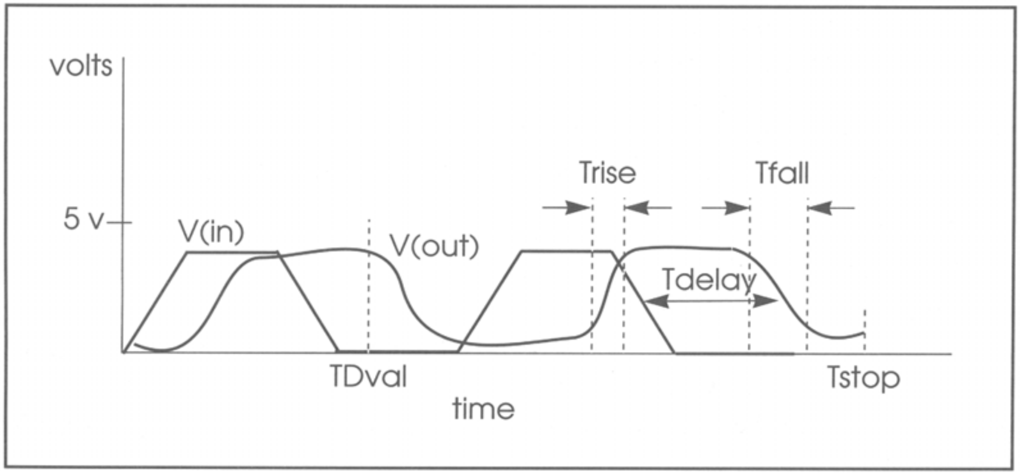
● Example:

```
.meas  TRAN  fifth    WHEN v(osc_out)=2.5V rise=5
.meas  TRAN  result  FIND   v(out)  WHEN v(in)=2.5V rise=1
.meas  TRAN  vmin   FIND   v(out)  AT=30ns
```

(12). MEASURE Statement : Application Examples

Rise, Fall, and Delay Calculations :

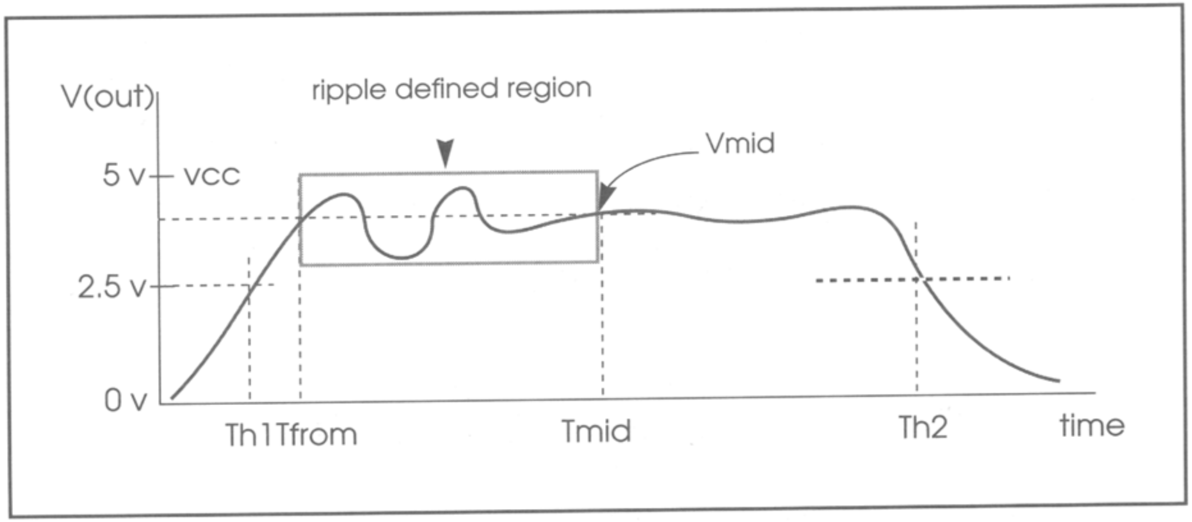
```
.meas TRAN Vmax MAX    v(out)  FROM=TDval             TO=Tstop
.meas TRAN Vmin MIN    v(out)  FROM =TDval            TO =Tstop
.meas TRAN Trise TRIG  v(out)  VAL='Vmin+0.1*Vmax'  TD=Tdval    RISE=1
+                TARG  v(out)  VAl='0.9*Vmax'                   RISE=1
.meas TRAN Tfall TRIG  v(out)  VAL='0.9*Vmax'       TD=Tdval    FALL=2
+                TARG  v(out)  VAl='Vmin+0.1*Vmax'              FALL=2
.meas TRAN Tdelay TRIG v(in)   VAL=2.5              TD=Tdval    FALL=1
+                 TARG v(out)  VAL=2.5                          FALL=2
```



(12). MEASURE Statement : Application Examples(Cont.)

● Ripple Calculation :

```
.meas TRAN  Th1     WHEN    v(out)='0.5*v(Vdd)'   CROSS=1
.meas TRAN  Th2     WHEN    v(out)='0.5*v(Vdd)'   CROSS=2
.meas TRAN  Tmid    PARAM='(Th1+Th2)/2'
.meas TRAN  Vmid    FIND    v(out)                AT='tmid'
.meas TRAN  Tfrom   WHEN    v(out)='Vmid'         RISE=1
.meas TRAN  Ripple   PP     v(out)                FROM='tfrom'    TO='tmid'
```



(12). MEASURE Statement : Application Examples(Cont.)

- **Unity-gain Freq, Phase margin, & DC gain(db/M):**

```
.meas  AC  unitfreq       WHEN  vdb(out)=0    FALL=1
.meas  AC  phase          FIND  vp(out)       WHEN vdb(out)=0
.meas  AC  'gain(db)'     MAX   vdb(out)
.meas  AC  'gain(mag)'    MAX   vm(out)
```

- **Bandwidth & Quality Factor (Q):**

```
.meas  AC  gainmax        MAX   vdb(out)
.meas  AC  fmax           WHEN  vdb(out)=‘gainmax’
.meas  AC  band           TRIG  vdb(out)   VAL=‘gainmax-3.0’    RISE=1
+                         TARG  vdb(out)   VAL=‘gainmax-3.0’    FALL=1
.meas  AC  Q_factor       PARAM=‘fmax/band’
```

Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Types of Elements:

- **Passive Devices :**
  - ■ ***R ---- Resistor***
  - ■ ***C ---- Capacitor***
  - ■ ***L ---- Inductor***
  - ■ ***K ---- Mutual Inductor***
- **Active Devices :**
  - ■ ***D ---- Diode***
  - ■ ***Q ---- BJT***
  - ■ ***J ---- JFET and MESFET***
  - ★ ■ ***M ---- MOSFET***
- **Other Devices :**
  - ■ ***Subcircuit (X)***
  - ■ ***Behavioral (E,G,H,F,B)***
  - ■ ***Transmission Lines (T,U,O)***

(2). Passive Devices : R, C, L, and K Elements

● Passive Devices Parameters :

	Resistor	Capacitor	Inductor	Mutual Inductor
Netlist	Rxxx, n1,n2, mname, rval	Cxxx, n1,n2, mname, cval	Lxxx, n1,n2, mname, lval	Kxxx, Lyyy, Lzzz, kval
Temperature	DTEMP, TC1, TC2	DTEMP, TC1, TC2	DTEMP, TC1, TC2	
Geometric	L, M, W, SCALE	L, M, W, SCALE	M, SCALE	
Parasitics	C		R	
Initialization		IC(v)	IC(i)	

● Examples :

```
R1    12    17    1K    TC1=1.3e-3  TC2=-3.1e-7
C2      7     8     0.6pf    IC=5V
LSHUNT  23   51   10UH 0.01  1    IC=15.7mA
K4  Laa Lbb  0.9999
```

(3). Active Device : BJT Element

- **BJT Element Parameters :**

TYPE	Parameters
Netlist	Qxxx, nb, nc, ne, ns, mname
Temperature	DTEMP
Geometric	AREA, AREAB, AREAC, M
Initialization	IC(VBE, VCE), OFF

- **BJT Syntax Examples :**

```
Q100  NC  NB  NE  QPNP  AREA=1.5  AREAB=2.5  AREAC=3.0  IC= 0.6, 5.0
```

- **BJT Model Syntax :**

```
.MODEL mname  NPN (PNP)  <param=val> .........
```

- **BJT Models in SBTSPICE:** *Gummel-Poon Model*

(4). Active Device : MOSFET Introduction

● MOSFET Model Overview :

- ■ MOSFET Defined by : (1). MOSFET Model & Element Parameters
  (2). Two Submodel : CAPOP & ACM
- ■ CAPOP : Specifies MOSFET Gate Capacitance
- ■ ACM : Modeling of MOSFET Bulk_Source & Bulk_Drain Diodes

● MOSFET Model Levels :

- ■ Available : All the public domain spice model
- ■ Level = 4 or 13 : BSIM1
- ■ Modified BSIM1
- ■ Level = 5 or 39 : BSIM2
- ■ Level = 49 : BSIM3.3
- ■ Level = 8 : SBT MOS8

(5). MOSFET Introduction : Element Statement

- MOSFET Element Syntax :

```
Mxxx nd  ng  ns  <nb>  mname <L=val> <W=val> <AD=val> <AS=val>
+                            <PD=val> <PS=val> <NRD=val>
+                            <NRS=val>
+                            <OFF> <IC=vds,vgs,vbs> <M=val>
+                            <TEMP=val> <GEO=val> <DELVTO=val>
```

- MOSFET Element Statement Examples:

```
M1   24   2   0   20   MODN    L=5u     W=100u     M=4
M2    1   2   3    4   MOD1      5u        100u
M3    4   5   6    8   N   L=2u W=10u AS=100P AD=100p PS=40u PD=40u
```

```
.OPTIONS   SCALE=1e-6
M1   24   2   0   20   MODN    L=5    W=100     M=4
```

(6). MOSFET Introduction : Model Statement

- **MOSFET Model Syntax :**

```
.MODEL mname  NMOS  <LEVEL=val> <name1=val1> <name2=val2>..........
.MODEL mname  PMOS  <LEVEL=val> <name1=val1> <name2=val2>..........
```

- **MOSFET Model Statement Examples:**

```
.MODEL  MODP  PMOS  LEVEL=2  VTO=-0.7  GAMMA=1.0......
.MODEL  NCH   NMOS  LEVEL=39  TOX=2e-2  UO=600..........
```

- **Corner_LIB of Models:**

```
.LIB TT or (FF|SS|FS|SF)
 .param  toxn=0.0141 toxp=0.0148......
.lib ‘~/simulation/model/cmos.l’  MOS
.ENDL TT or (FF|SS|FS|SF)
```

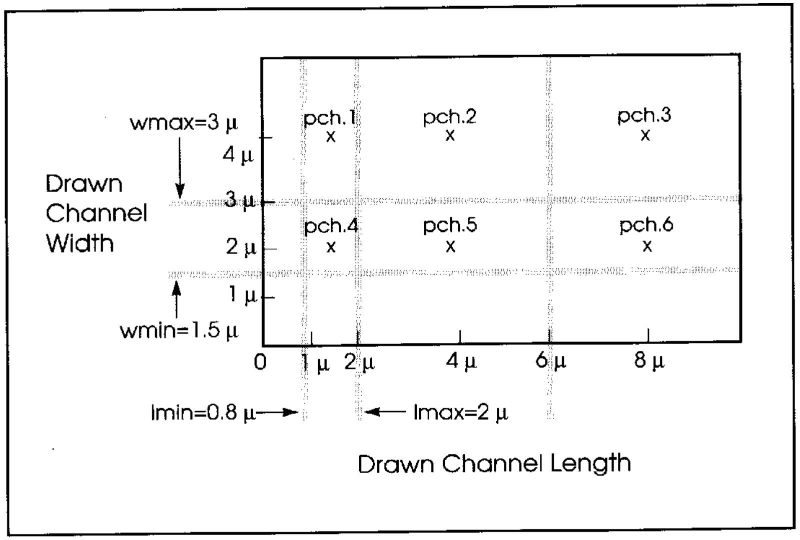
```
.LIB MOS
 .MODEL NMOD NMOS (LEVEL=49
+  TOXM=toxn  LD=3.4e-8 , ......)
.ENDL MOS
```

(7). MOSFET Introduction : Automatic Model Selection

● Automatic Model Selection :

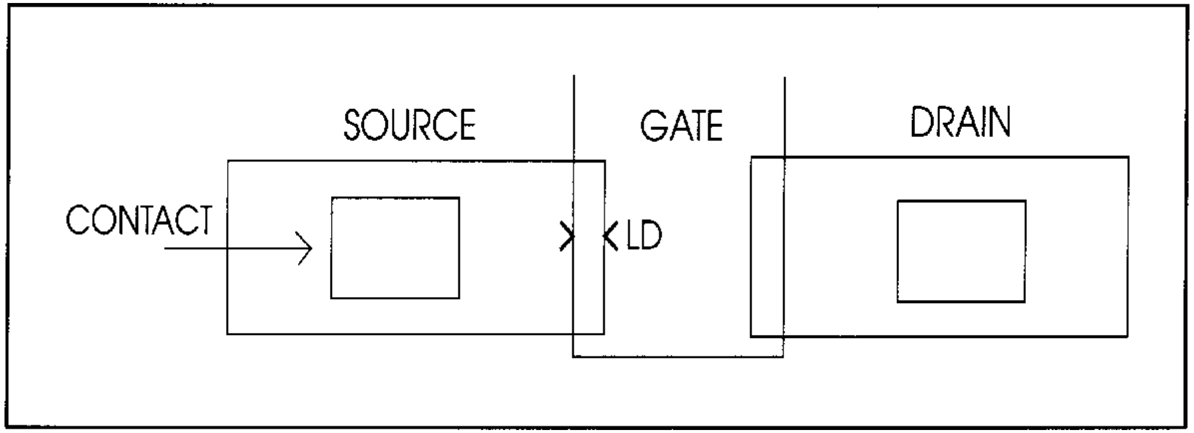
- HSPICE can Automatically Find the Proper Model for Each Transistor Size by Using Parameters, LMIN,LMAX,WMIN, & WMAX in MOSFET Models

```
.MODEL pch.4  PMOS  WMIN=1.5u WMAX=3u LMIN=0.8u LMAX=2.0u
.MODEL pch.5  PMOS  WMIN=1.5u WMAX=3u LMIN=2.0u LMAX=6.0u
M1  1  2  3  4  pch  W=2u  L=4u  $ Automatically Select pch.5 Model
```



(8). MOSFET Introduction : MOSFET Diode Model

- **MOSFET Diode Model : ACM**
  - **Area calculation Method (ACM) Parameter Allows for the Precise Control of Modeling Bulk-Source & Bulk_Drain Diodes within MOSFET Models**
- **ACM=0 MOSFET Diode: (Conventional MOSFET Structure)**

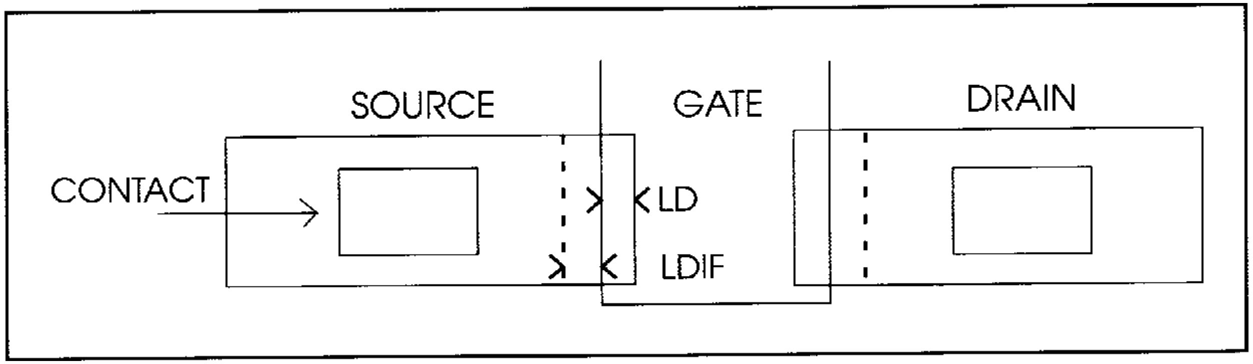


- **ACM=0 : PN Bulk Junction of MOSFET are Modeled in the SPICE-style.**
- **ACM=0 : Not Permit Specifications of HDIF & LDIF.**

$$\text{ADeff} = \text{M} \bullet \text{AD} \bullet \text{WMLT}^2 \bullet \text{SCALE}^2$$

(8). MOSFET Introduction : MOSFET Diode Model (Cont.)

● ACM=1 MOSFET Diode: (Not Popular)



- ACM=1 : ASPEC-style Diode Model.
- ACM=1 : Parameter Function of Element Width.
- ACM=1 : AS, AD, PS, & PD are not Used.
- ACM=1 : JS and CJ Differ from the SPICE_style Diode (ACM=0)

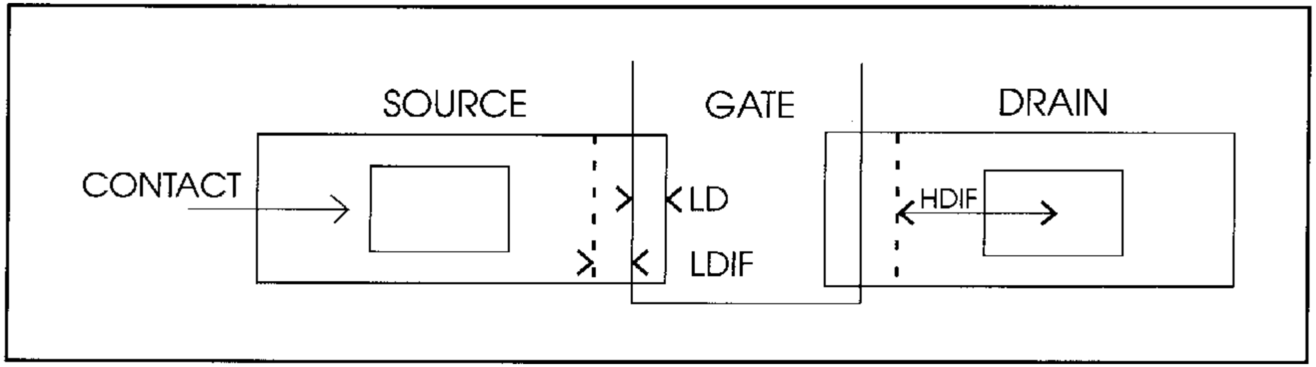
  Ignore AS,AD, PS,PD parameters

ADeff = Weff • WMLT

Pdeff = Weff

(8). MOSFET Introduction : MOSFET Diode Model (Cont.)

● ACM=2 MOSFET Diode: (MOSFET LDD Structure)



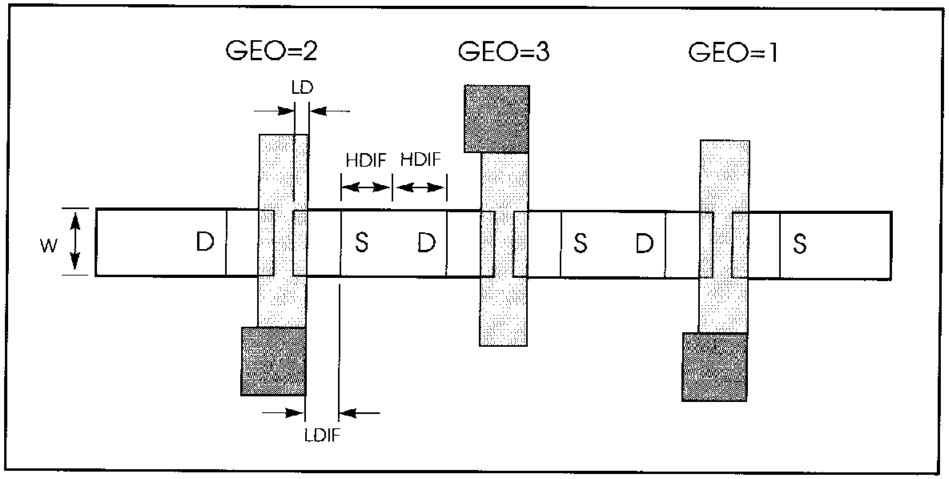
- ■ ACM=2 : HSPICE_Style Diode Model, Combination of ACM=0 & 1.
- ■ ACM=2 : Supports both Lightly & Heavily Doped Diffusions by Settling LD, LDIF, and HDIF Parameters.
- ■ ACM=2 : Effective Areas and Peripheries can be Calculations by LDIF & HDIF (i.e. AS, AD, PS, & PD can be Omitted in MOS Element Statement)

$$\boxed{ADeff = 2 \cdot HDIF \cdot Weff}$$

$$\boxed{PDeff = 4 \cdot HDIF + 2 \cdot Weff}$$

(8). MOSFET Introduction : MOSFET Diode Model (Cont.)

● ACM=3 MOSFET Diode : (Stacked MOSFET Diode Model)



■ ACM=3 : Extension of ACM=2 Model that Deals with Stacked Devices.

■ ACM=3 : AS, AD, PS, & PD Calculations Depend on the Layout of the Device, which is Determined by the Value of Elememt Parameter GEO.

■ ACM=3 : GEO=0 (Default) Indicates Drain & source are not Shared by other Devices

(9). MOSFET Introduction : Gate Capacitance Models

MOSFET Gate Capacitance Models:

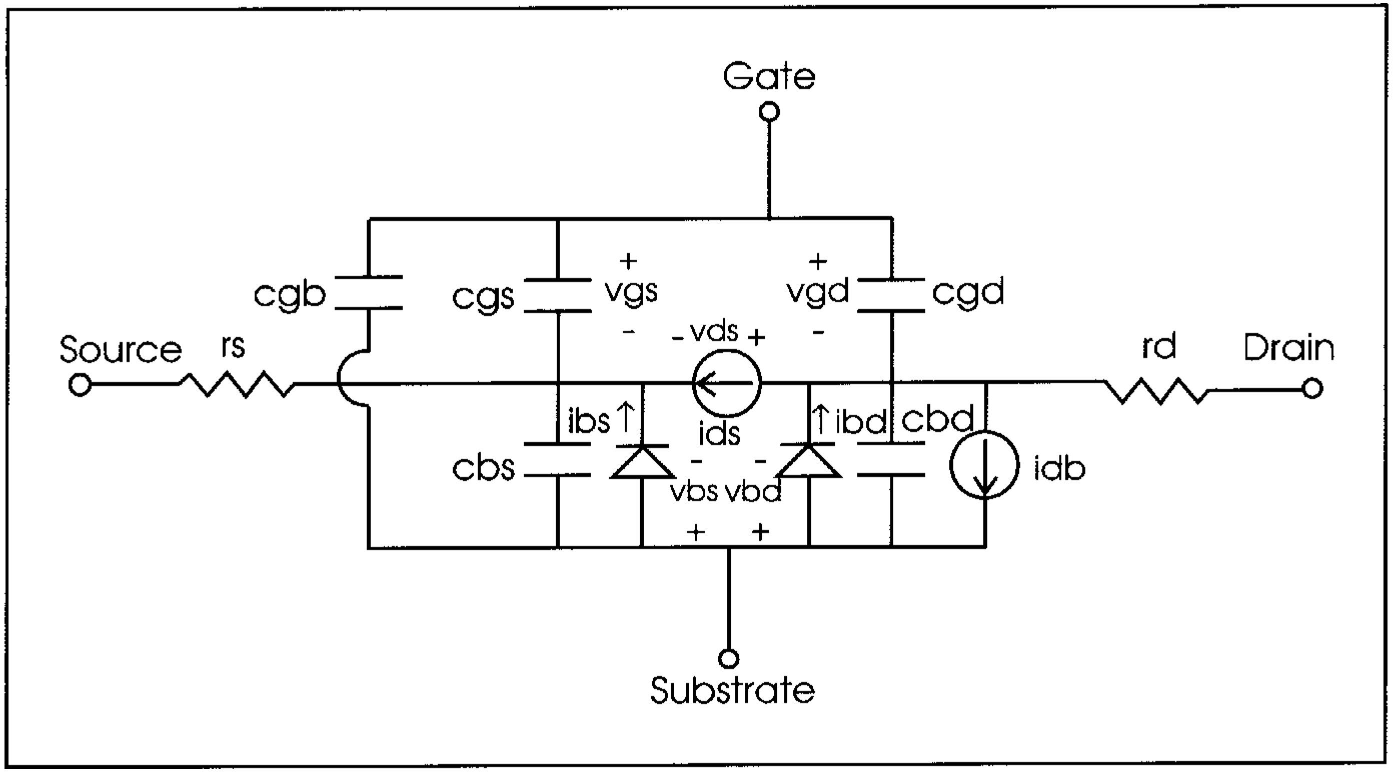
- Capacitance Model Parameters can be Used with all MOSFET Model Statement.
- Model Charge Storage Using Fixed and Nonlinear Gate Capacitance and Junction Capacitance.
- Fixed Gate Capacitance : Gate-to-Drain, Gate-to-Source, and Gate-to-Bulk Overlap Capacitances are Represented by CGSO, CGDO, & CGBO.
- Nonlinear Gate Capacitance : Voltage-Dependent MOS Gate Capacitance Depends on the Value of Model Parameter CAPOP.

MOSFET Gate Capacitance Selection :

- Available CAPOP Values = 0, 1, 2(General Default), 4

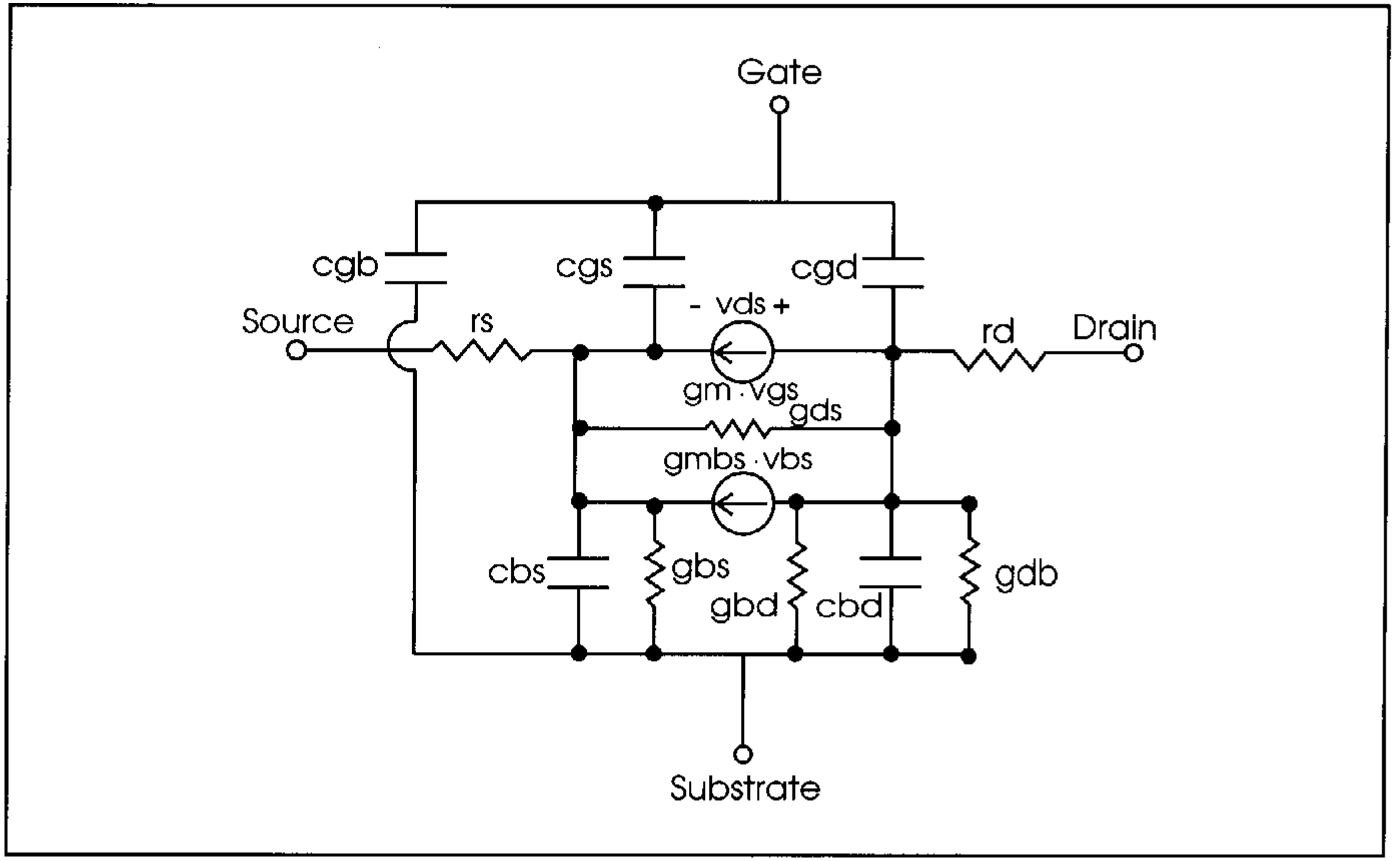
(10). MOSFET Introduction : Equivalent Circuits

- **MOSFET Equivalent Circuit for Transient Analysis:**



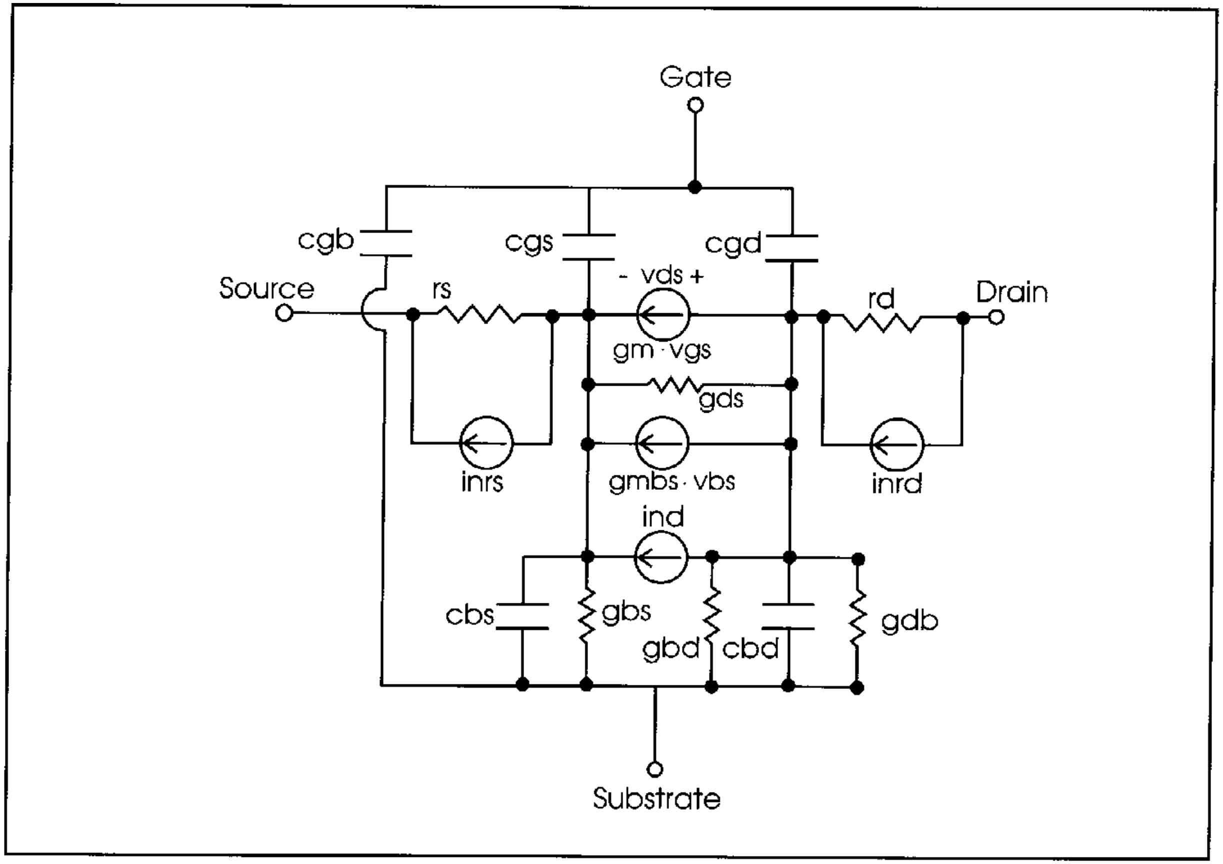
(10). MOSFET Introduction : Equivalent Circuits (Cont.)

- **MOSFET Equivalent Circuit for AC Analysis:**



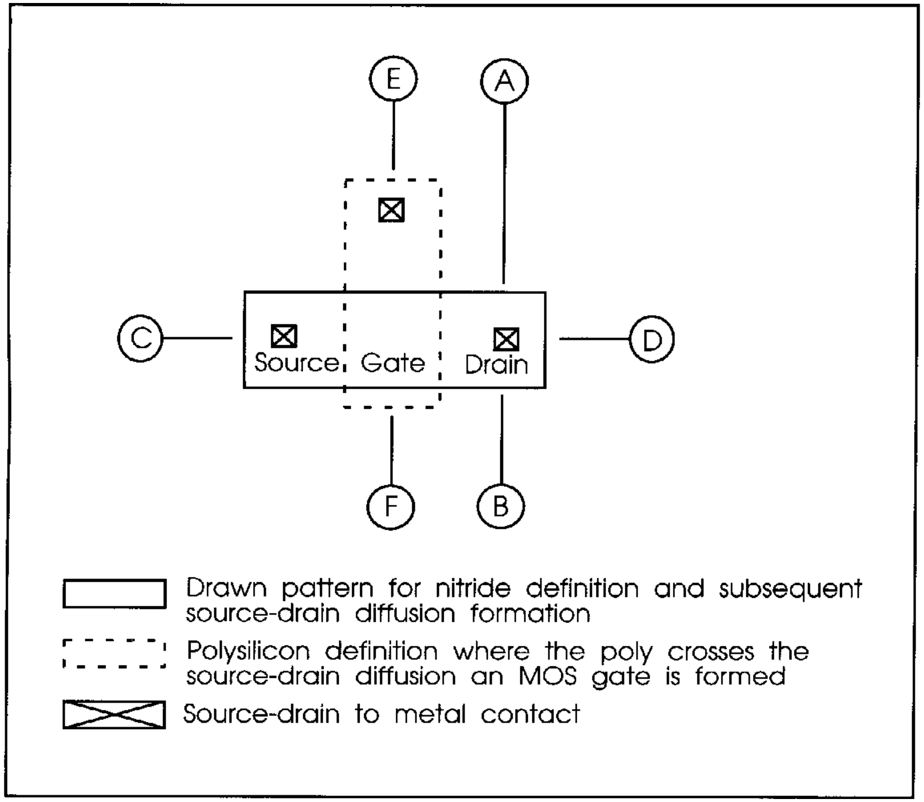
(10). MOSFET Introduction : Equivalent Circuits (Cont.)

- MOSFET Equivalent Circuit for AC Noise Analysis:



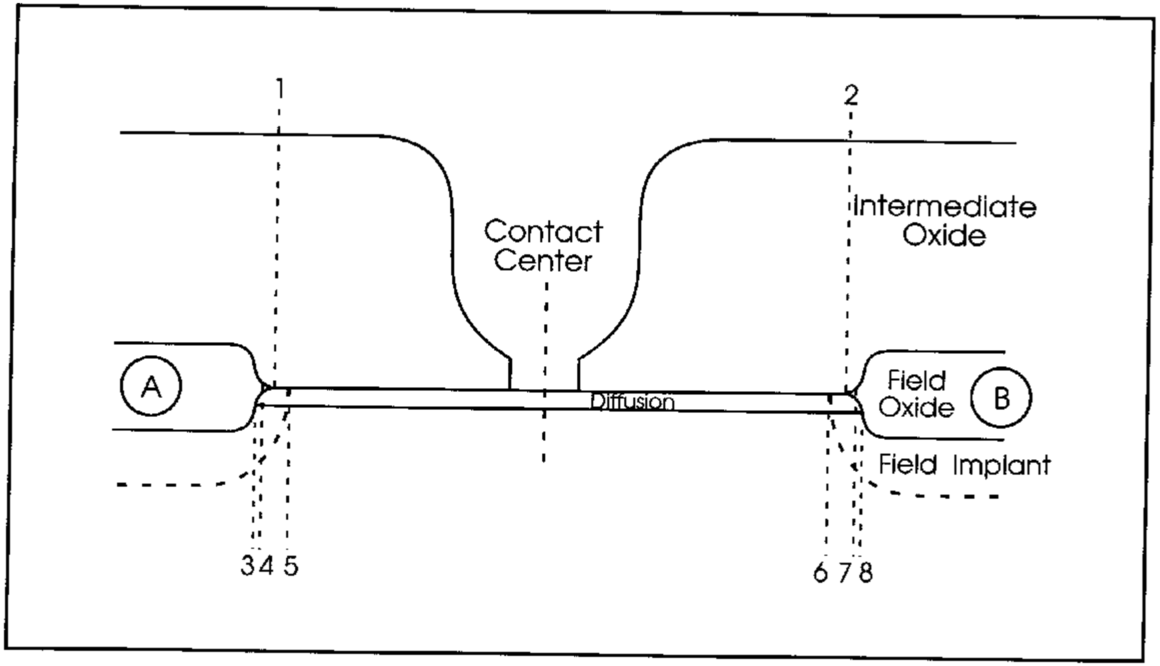
(11). MOSFET Introduction : Construction of MOSFET

- Isoplanar Silicon Gate Transistor :


E
A
C
D
F
B
Source
Gate
Drain
Drawn pattern for nitride definition and subsequent source-drain diffusion formation
Polysilicon definition where the poly crosses the source-drain diffusion an MOS gate is formed
Source-drain to metal contact

(11). MOSFET Introduction : Construction of MOSFET (Cont.)

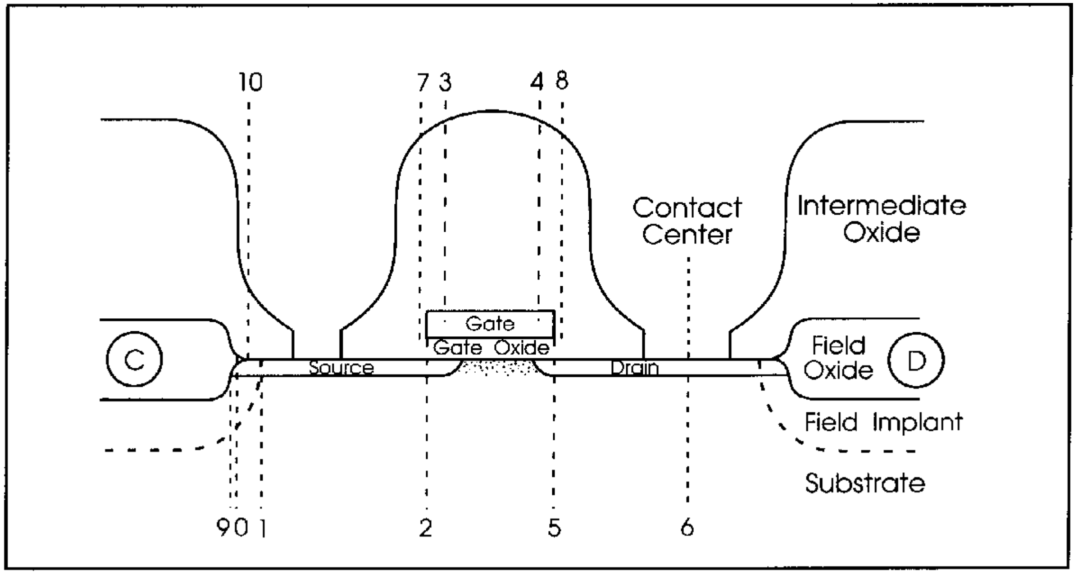
- Isoplanar MOSFET Construction : (Cut through A-B)



1 - 2 diffusion drawn dimension for nitride
4 - 7 nitride layer width after etch
3 - 1 periphery of the diode

(11). MOSFET Introduction : Construction of MOSFET(Cont.)

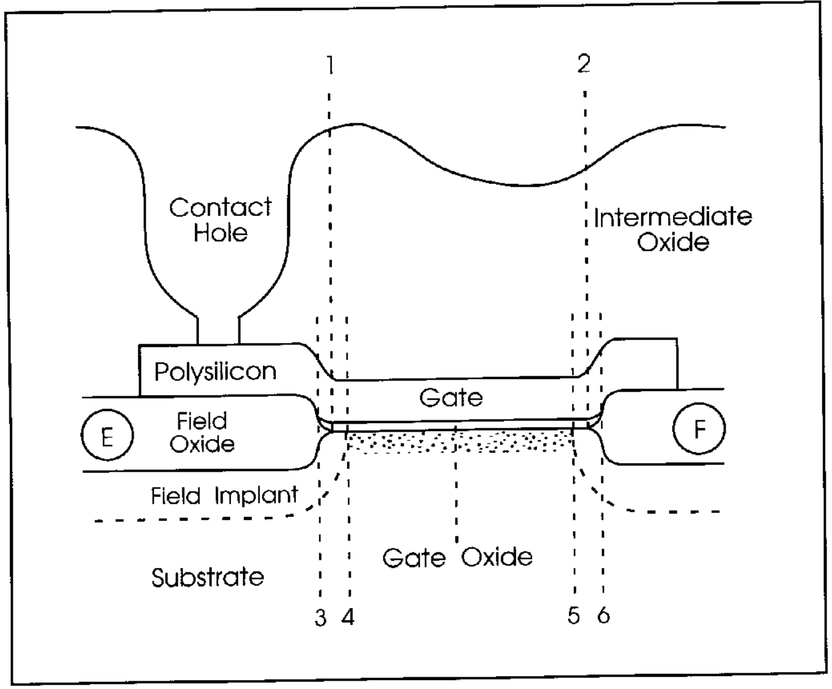
● Isoplanar MOSFET Construction : (Cut through C-D)



7 - 8	drawn channel length L
2 - 5	actual poly width after etching L + XL where XL<0
3 - 4	effective channel length after diffusion L + XL - LD
4 - 5	lateral diffusion LD
9 - 10	diffusion periphery for diode calculations
5 - 6	gate edge to center contact for ACM=1 and ACM=2 calculations

(11). MOSFET Introduction : Construction of MOSFET(Cont.)

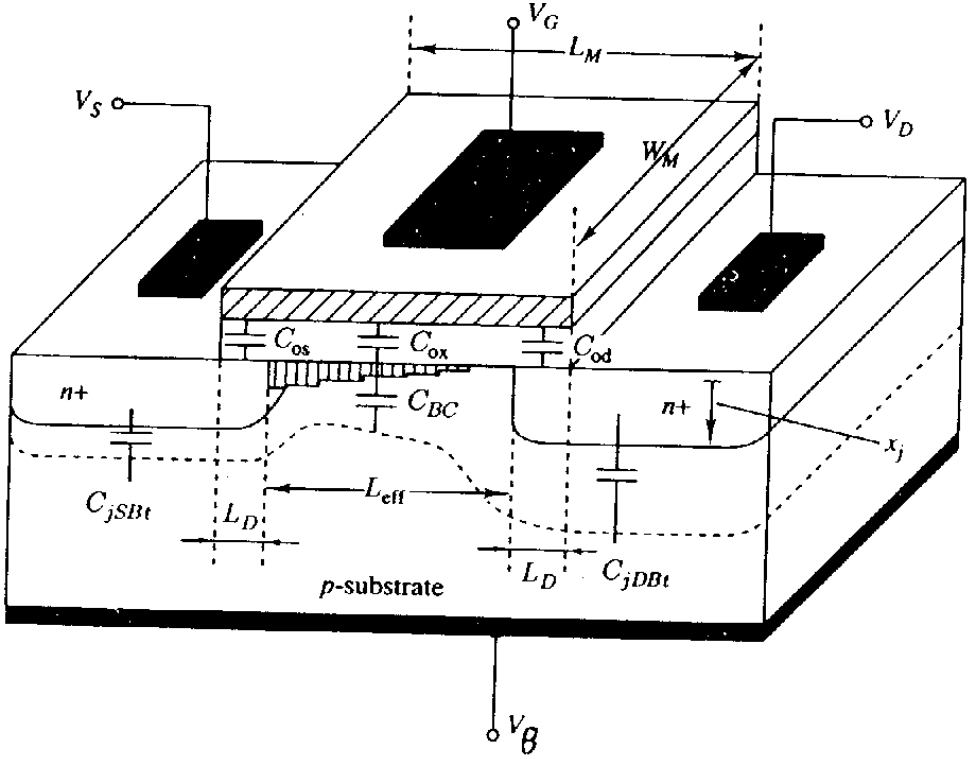
● Isoplanar Silicon Gate Transistor : (Cut through E-F)



1 - 2	drawn width of the gate W
3 - 4	depleted or accumulated channel (parameter WD)
4 - 5	effective channel width W+ XW -2 WD
3 - 6	physical channel width W + XW

(12). MOSFET Transistor Basics : Structure & Bias

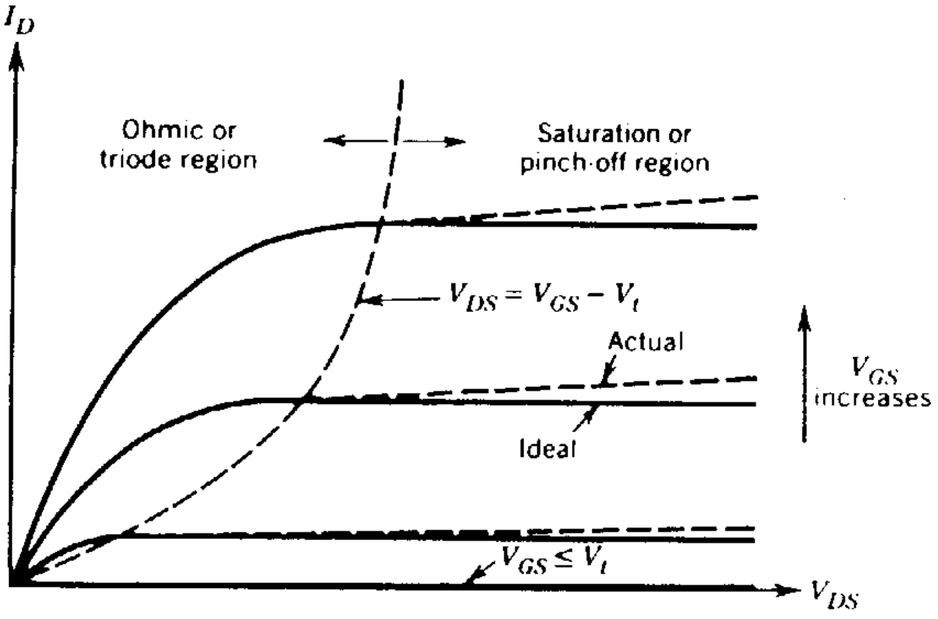
- MOSFET Structure : A Four Terminal Device (V_G, V_D, V_S, V_B)

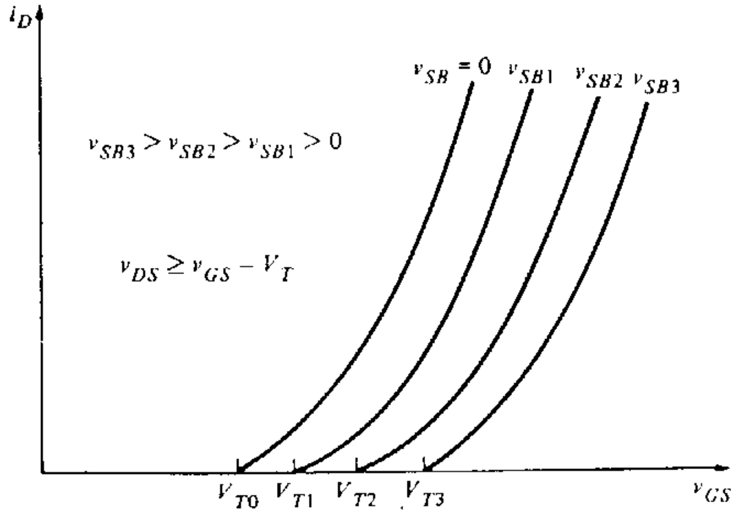


- Basic Parameters : Channel Length (L_M), Channel Width(W_M), Oxide Thickness(tox), Junction depth(x_j) & Substrate Doping(Na)

(13). MOSFET Transistor Basics : Transfer Characteristics

● Transfer Characteristics of NMOS :





- **Basic Operations : Saturation, Linear , & Subthreshold Regions**
- **Basic Characteristics : Channel Length Modulation & Body Effects**

$$I_D = \frac{\mu_n C_{ox}}{2}\frac{W}{L}(V_{GS} - V_T)^2(1 + \lambda V_{DS})$$

$$V_{TN} = V_{TN0} + \gamma\left(\sqrt{|2\Phi_F| + V_{SB}} - \sqrt{|2\Phi_F|}\right)$$

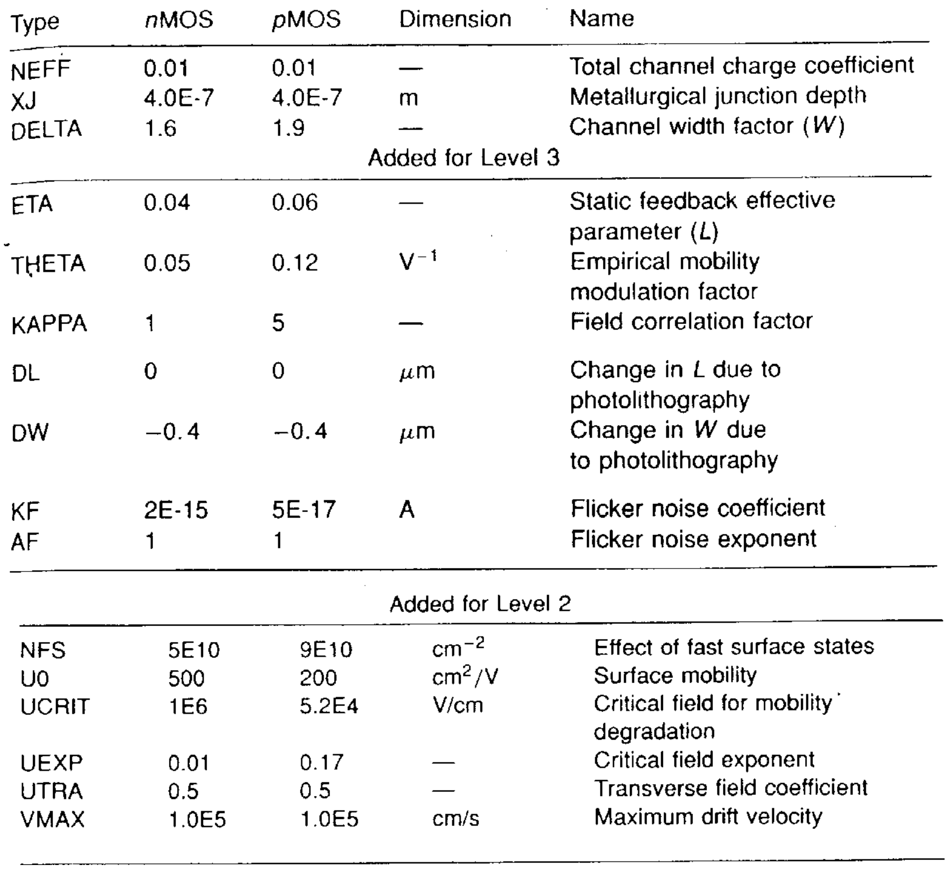
(14). MOSFET Transistor Basics : SPICE Parameters

● Normal SPICE Model Parameters :

Type	*n*MOS	*p*MOS	Dimension	Name
			Level 1	
VT0	0.6	−0.77	V	Zero bias threshold voltage
KP	4.0E-5	1.5E-5	A/V^2	Transconductance parameter
GAMMA	0.92	0.54	$V^{1/2}$	Bulk threshold parameter
LAMBDA	0.022	0.047	V^{-1}	Channel length modulation parameter
CGS0	5.2E-10	4.0E-10	F/m	*GS* overlap capacitance per meter channel width
CGD0	5.2E-10	4.0E-10	F/m	*GD* overlap capacitance per meter channel width
CGB0			F/m	*GB* overlap capacitance per meter channel width
CJ	4.5E-4	3.6E-4	F/m^2	Zero bias junction capacitance
MJ	0.5	0.5	—	Grading coefficient
CJSW	6E-10	6E-10	F/m	Zero bias junction sidewall capacitance
MJSW	0.33	0.33	—	Grading coefficient
PB	0.6	0.6	V	Bulk junction potential
FC	0.5	0.5	—	Coefficient forward bias depletion capacitance
JS	1E-3	1E-3	A/m^2	Bulk junction saturation current
TPG	1.0	−1.0	—	Type of gate
LD	3.2E-7	3.2E-7	m	Lateral diffusion
RSH	20	50	Ω/□	Sheet resistance diffusion of *D* and *S*
RS	15	20	Ω	Ohmic source resistance
RD	15	20	Ω	Ohmic drain resistance
			Process parameters	
TOX	5.2E-8	5.2E-8	m	Oxide thickness
PHI	0.60	0.60	V	Surface potential ($2\phi_F$)
NSUB	4.6E15	2.5E14	cm^{-3}	Substrate doping
NSS	5E10	5E10	cm^{-2}	Effect of surface charge (Q_{ox})

(14). MOSFET Transistor Basics : SPICE Parameters(Cont.)

● Normal SPICE Model Parameters(Cont.) :



Type	*n*MOS	*p*MOS	Dimension	Name
NEFF	0.01	0.01	—	Total channel charge coefficient
XJ	4.0E-7	4.0E-7	m	Metallurgical junction depth
DELTA	1.6	1.9	—	Channel width factor (*W*)
Added for Level 3				
ETA	0.04	0.06	—	Static feedback effective parameter (*L*)
THETA	0.05	0.12	V^{-1}	Empirical mobility modulation factor
KAPPA	1	5	—	Field correlation factor
DL	0	0	μm	Change in *L* due to photolithography
DW	−0.4	−0.4	μm	Change in *W* due to photolithography
KF	2E-15	5E-17	A	Flicker noise coefficient
AF	1	1		Flicker noise exponent
Added for Level 2				
NFS	5E10	9E10	cm^{-2}	Effect of fast surface states
U0	500	200	cm^{2}/V	Surface mobility
UCRIT	1E6	5.2E4	V/cm	Critical field for mobility degradation
UEXP	0.01	0.17	—	Critical field exponent
UTRA	0.5	0.5	—	Transverse field coefficient
VMAX	1.0E5	1.0E5	cm/s	Maximum drift velocity

(15). MOSFET Transistor Basics : *Higher-Order Effects*

- **Geometry and Doping Effects on Vth :**
  - ■ Short Channel Effect (Small L)
  - ■ Narrow Channel Effect (Small W)
  - ■ Non-Uniform Doping Effect
- **Physical Effects on Output Resistance :**
  - ■ Channel Length Modulation (CLM)
  - ■ Drain Induced Barrier Lowering (DIBL)
  - ■ Substrate Current Induced Body Effects (SCBE)
- **Other Physical Effects :**
  - ■ Channel Mobility Degradation
  - ■ Carrier Drift Velocity
  - ■ Bulk Charge Effect
  - ■ Parasitic Resistance
  - ■ Subthreshold Current

Source : HP Eesof Device Modeling Seminar, March 1996

(16). MOSFET Models : Historical Evolution

- Can Define Three Clear Model “Generations”
- First Generation :
  - “Physical” Analytical Models
  - Geometry Coded into the Model Equations
  - Level 1, Level 2, & Level 3
- Second Generation :
  - Shift in Emphasis to Circuit Simulation
  - Extensive Mathematical Conditioning
  - Individual Device Parameters & Separate Geometry Parameter
  - Shift “Action” to Parameter Extraction (Quality of Final Model is Heavily Dependent on Parameter Extraction)
  - BSIM1, Modified BSIM1, BSIM2

Source : IEEE 1997 CICC Educational Sessions E3.3

(16). MOSFET Models : Historical Evolution (Cont.)

- Third Generation :

  - "Original Intent" was a Return to Simplicity
  - Scalable MOSFET model
  - 1-st derivative is continuous
  - Attempt to Re-Introduce a Physical Basis While Maintaining "Mathematical Fitness"
  - BSIM3, MOS-8, Other ???

Source : IEEE 1997 CICC Educational Sessions E3.3

(17). Overview of Most Popular MOSFET Models :

- **UCB Level 1 : (Level = 1)**
  - **Shichman-Hodges Model (1968)**
  - **Simple Physical Model, Applicable to L> 10um with Uniform Doping**
  - **Not Precise Enough for Accurate Simulation**
  - **Use only for Quick, Approximate Analysis of Circuit Performance**
- **UCB Level 2 : (Level = 2)**
  - **Physical/Semi-Empirical Model**
  - **Applicable to Long Channel Device (~ 10 um)**
  - **Advanced Version of Level 1 which Includes Additional Physical Effects**
  - **Can Use either Electrical or Process Related Parameters**

SPICE* : *Simulation Program with Integrated Circuit Emphasis

UCB* : *University of California at Berkeley

(17). Overview of Most Popular MOSFET Models(Cont.) :

- **UCB Level 3 : (Level = 3)**
  - **Semi-Empirical Model Model (1979)**
  - **Applicable to Long Channel Device (~ 2um)**
  - **Includes Some New Physical Effects (DIBL, Mobility Degradation by Lateral Field)**
  - **Very successful Model for Digital Design (Simple & Relatively Efficient)**
- **BSIM : (Level = 13)**
  - **First of the "Second Generation" Model (1985)**
  - **Applicable to Short Channel Device with L~ 1.0um**
  - **Emphasis on Mathematical Conditioning of Circuit Simulation**
  - **Empirical Approach to Small Geometry Effects**

BSIM : Berkeley Short-Channel IGFET Model

(17). Overview of Most Popular MOSFET Models (Cont.) :

● Modified BSIM1 LEVEL 28 :

- Enhanced Version of BSIM 1, But Addressed most of the Noted Shortcomings
- Empirical Model Structure --> Heavy Reliance on Parameter Extraction for Final Model Quality
- Applicable to Deep Submicron Devices (~ 0.3 - 0.5um)
- Suitable for Analog Circuit Design

● BSIM 2 : (HSPICE Level = 39)

- “Upgraded” Version of BISM 1 (1990)
- Applicable to Devices with (L~ 0.2um)
- Drain Current Model has Better Accuracy and Better Convergence Behavior
- Covers the Device Physics of BSIM 1 and Adds Further Effects on Short Channel Devices

(17). Overview of Most Popular MOSFET Models (Cont.) :

● **BSIM 3v3 : (Level = 49)**

- ■ **Newly Advanced Submicron MOSFET Model (Third Generation)**
- ■ **Latest Physics-Based, Deep-Submicron Model (BSIM3v3.1)**
- ■ **Use a Single Equation for a Device Property for All Regions of Device Operation, But Slow & Inefficient Behavior During Circuit Simulation**
- ■ **Use of Smoothing Functions Greatly Improves Final Results**
- ■ **Attempt to Impose “Global” Capability Creates a Number of Problems**

In HSPICE, level=53 is BSIM 3V3 Berkeley compliance
, level=49 is Avanti implemented BSIM3V3

● **MOS8 Model : (SBTSPICE Level = 8)**

- ■ **Developed by SBT**
- ■ **4-Terminals NQS charge-conservation model**
- ■ **Physical-oriented scalable model**

(17). Overview of Most Popular MOSFET Models (Cont.) :

● EKV Model :

- Developed at Swiss Federal Institute of Technology in Lausanne (EPFL)
- A Newly "Candidate" Model for Future Use
- Description of Small Geometry Effects is Currently Being Improved
- Developed for Low Power Analog Circuit Design
- Fresh Approach to FET Modeling
  - Use Substrate (not Source) as Reference
  - Simpler to Model FET as a Bi-Directional Element
  - Can Treat Pinch-Off and Weak Inversion as the same Physical Phenomenon
- First "Re-Thinking" of Analytical FET Modeling Since Early 1960s.

Source : IEEE 1997 CICC Educational Sessions E3.3

(18). MOSFET Model Comparison :

- **Model Equation Evaluation Criteria :** (Ref: HSPICE User Manual 1996, Vol._II)
  - Potential for Good Fit to Data
  - Ease of Fitting to Data
  - Robustness and Convergence Properties
  - Behavior Follows Actual Devices in All Circuit Conditions
  - Ability to Simulate Process Variation
  - Gate Capacitance Modeling
- **General Comments :**
  - Level 3 for Large Digital Design
  - HSPICE Level 28 for Detailed Analog/Low Power Digital
  - BSIM 3v3 & MOS Model 9 for Deep Submicron Devices
  - All While Keeping up with New Models

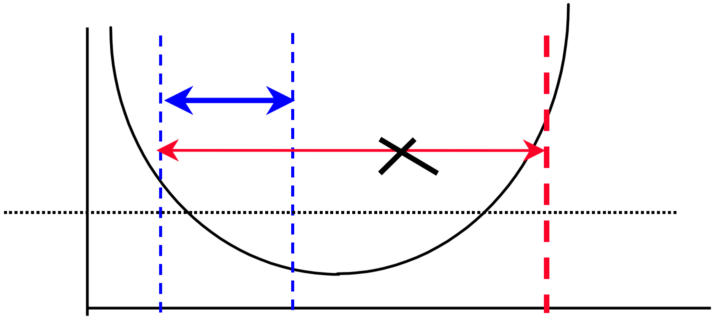
Source : IEEE 1997 CICC Educational Sessions E3.3

Contents

(1). SPICE Optimization

● Circuit Level Goal Optimization:

- A procedure for automatic searching instance parameters to meet design goal
- Can be applied for both .DC , .AC and .TRAN analysis
- Optimization implemented in SBTSPICE can optimize one goal
- Optimization implemented in HSPICE can optimize multi-goal circuit parameter/device model parameter
- The parameter searching range must differentiate the optimization goal



(2). Optimization Preliminaries

- Circuit Topology Including Elements and Models
- List of Element to be Optimized
  - Initial Guess, Minimum, Maximum
- .Measure Statements for Evaluating Results
  - Circuit Performance Goals
  - Selection of Independent or Dependent Variables Measurement Region
- Specify Optimizer Model

(3). Optimization Syntax : General Form

- Variable Parameters and Components :

```
.PARAM  parameter = OPTxxx (init, min, max)
```

- Optimizer Model Statement :

```
.MODEL  mod_namd   OPT   <Parameter = val .....>
```

- Analysis Statement Syntax :

```
.DC|AC|TRAN .......<DATA=filement >  SWEEP   OPTIMIZE = OPTxxx
+                       Results = meas_name    METHOD = method_name
```

- Measure Statement Syntax :

```
.MEASURE meas_name .......<GOAL=val>  <MINVAL=val>
```

(4). Optimization Example

```
.lib “ls35_4_1.l” tt
.option post probe
.param Cload =10p
.param Tpw=opt1(0, 0, 15n)
.model optmod opt method=passfail
.tran 0.1n 20n sweep optimize=opt1
+                        result = Tprop
+                        model = optmod
.measure Tran Tprop Trig V(in) Val=2.5 Rise = 1
+                    Targ v(out) Val=2.5 Fall = 1

vcc 1 0 5
vin in 0 pulse(0 5 1n 1n 1n Tpw 20n)
....
.end
```

Specify parameter range

Analysis type and optimization algorithm

Optimization goal by measure command

Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Control Options : Output Format

- Output Format : General (LIST, NODE, ACCT, OPTS, NOMOD)

  - .OPTION LIST : Produces an Element Summary Listing of the Data to be Printed. (Useful in Diagnosing Topology Related NonConvergence Problems)
  - .OPTION NODE : Prints a Node Connection Table. (Useful in Diagnosing Topology Related NonConvergence Problems)
  - .OPTION ACCT : Reports Job Accounting and Run-Time Statistics at the End of Output Listing. (Useful in Observing Simulation Efficient)
  - .OPTION OPTS : Prints the Current Settings of All Control Options.
  - .OPTION NOMOD : Suppress the printout of Model Parameters (Useful in Decreasing Size of Simulation Listing Files)

(2). Simulation Controls & Convergence

- Definition of "Convergence" :
  - The Ability to Obtain a Solution to a Set of Circuit Equations Within a Given Tolerance Criteria & Specified Iteration Loop Limitations.
  - The Designer Specifies a Relative & Absolute Accuracy for the Circuits Solution and the Simulator Iteration Algorithm Attempts to Converge onto a Solution that is within these Set Tolerance
- Error Messages for "NonConvergence" :
  - " No Convergence in Operating Point (or DC Sweep)"
  - " Iteration TimeStep is Too Small in Transient Analysis"
- Possible Causes of "NonConvergence" :
  - Circuit Reasons : (1). Incomplete Netlist; (2). Feedback; (3). Parasitics
  - Model Problems : (1). Negative Conductance (2). Model Discontinuity
  - Simulation Options : (1). Tolerances; (2). Iteration Algorithm

(3). AutoConvergence for DC Operating Point Analysis

● AutoConvergence Process:

- If Convergence is not Achieved in the Number of Iteration set by ITL1, HSPICE Initiates an AutoConvergence Process, in which it Manipulates DCON, GRAMP, and GMINDC, as well as CONVERGENCE in some Cases.

- ITL1= *x* : Set the Maximum DC Iteration Limit, Default=100. Increasing Values as High as 400 Have Resulted in Convergence for Certain Large Circuits with Feedback, such as OP Amp & Sense Amplifiers.

- GMINDC= *x* : A Conductance that is Placed in Parallel with All PN Junction and All MOSFET Nodes for DC Analysis. It is Important in Stabilizing the Circuit During DC Operating Point Analysis.

(4).Steps for Solving DC Operating Point NonConvergence

- (1). Check Topology :
  - Set .OPTIONS NODE to Get a Nodes Cross Reference Listing if You are in Doubt
  - Check if All PMOS Substrates Connected to VDD or Positive Supplies ?
  - Check if All NMOS Substrates Connected to GND or Negative Supplies ?
  - Check if All Vertical NPN Substrates Connected to GND or Negative Supplies ?
  - Check if All Lateral PNP Substrates Connected to Negative Supplies ?
  - Check if All Latches Have Either an OFF Transistor or a .NODESET or an .IC on one side ?
  - Check if All Series Capacitors Have a Parallel Resistance, or is .OPTION DCSTEP Set ?

(4).Steps for Solving DC Operating Point NonConvergence (Cont.)

● (2). Check Your .MODEL Statements :

- Be Sure to Check your Model Parameter Units.
- Check if Your MOS Models had Subshreshold Parameter Set ? (NFS= 1e11 for HSPICE Level 1,2 &3 and N0=1 for HSPICE BSIM1,2,3 Models & Level 28)
- Use MOS ACM=1, ACM=2, or ACM=3 Source and Drain Diode Calculation to Automatically Generate Parasitics.

● (3). General Remarks :

- Circuits that Converge Individually and Fail when Combined are Almost Guaranteed to have a Modeling Problem.
- Schmitt Triggers are Unpredictable for DC Sweep and Sometimes for Operating Point for the same Reasons Oscillators and Flip-Flops are. Use Slow Transient.

(4).Steps for Solving DC Operating Point NonConvergence (Cont.)

● (3). General Remarks (Cont.):

- Open Loop OP Amp have High Gain, which can Lead to Difficulties in Converging. ==> Start OP Amp in Unity-Gain Configuration and Open them Up in Transient Analysis with a Voltage-Variable Resistor or a Resistor with a Large AC Value for AC Analysis.

● (4). Check Your Options :

- Remove All Convergence-Related Options and Try First with No Special Options Setting.
- Check NonConvergence Diagnostic Table for NonConvergence Nodes. Look up NonConvergence Nodes in the Circuit Schematic. They are Generally Latches, Schmitt Triggers or Oscillating Nodes.
- SCALE and SCALM Scaling Options Have a Significant Effect on the Element and Model Parameters Values. Be Careful with Units.

(5).Solutions for Some Typical NonConvergence Circuits

● Poor Initial Conditions :

- Multistable Circuits Need State Information to Guide the DC Solution. For Example, You must Initialize Ring Oscillator or Flip-Flops Circuits Using the .IC Statement.

● Inappropriate Model Parameters :

- It is Possible to Create a Discontinuous Ids or Capacitance Model by Imposing Nonphysical Model Parameters
- Discontinuities Most Exits at the Intersection of the Linear & Saturation Regions

● PN Junctions (Diodes, MOSFETs, BJTs) :

- PN Junctions Found in Diodes, BJTs, and MOSFET Models can Exhibit NonConvergence Behavior in Both DC and Transient Analysis.==> Options GMINDC and GMIN Automatically Parallel Every PN Junction with a Conductance.

(5).Solutions for Some Typical NonConvergence Circuits (Cont.)

● Rapid Transitions in DC Sweep :

- Increase the ITL2 , which is the Number for Iteration Allowed at Each Steps in DC Sweep Analysis, Value to 100, 200 or more. Default=50. (ITL1 is for the Initial Operating Point)

● DC Options for High Power Circuits :

- For High Power Bipolar Transistors, Set Options ABSI up from Default 1 nA to Perhaps 10 mA. Also, Minimum Voltage Tolerance Should be Raised from 50 uV to 10mV.
- If the Accuracy Given by Tight Tolerance is Necessary, Be Sure to Increase Option ILT1 to 300~~400.

● DC Options for Op Amp and Comparator :

- High Gain Op Amp and Comparator can Cause Convergence Problems, as well as Possible Violations of KCL Law. It is Sometimes Necessary to Tighten the Options RELI to 0.005 and ABSI to 1 pA

(6).Numerical Integration Algorithm Controls

● Types of Numerical Integration Methods :

- Trapezoidal Algorithm (Default in HSPICE)
- GEAR Algorithm

.OPTION METHOD = TRAP

.OPTION METHOD = GEAR

● Trapezoidal Algorithm :

- Highest Accuracy
- Lowest Simulation Time
- Best for CMOS Digital Circuits

● GEAR Algorithm :

- Most Stable
- Highly Analog, Fast Moving Edges

● One Limitation of Trapezoidal Algorithm :

- It can Results in Unexpected Computational Oscillation. (Also Produces an Usually Long Simulation Time)
- For Circuits are Inductive in Nature, such as Switching Regulator, Use GEAR Algorithm. (Circuit NonConvergent with TRAP will often Converge with GEAR)

(7). Timestep Control Algorithms

- Types of Dynamic Timestep Control Algorithm :
  - Iteration Count (Simplest):
    - If Iterations Required to Converge > MAX, Decrease the Timestep
    - If Iterations Required to Converge < MIN, Increase the Timestep
  - Local Truncation Error (LTE) :
    - Use a Taylor Series Approximation to Calculate Next Timestep
    - Timestep is Reduced if Actual Error is > Predicted Error
- Timestep Control Algorithm vs. Numerical Integration Algorithm :
  - For GEAR is Selected => Defaults to Truncation Timestep Algorithm; For TRAP is Selected => Defaults to ITERATION Algorithm

Contents

(1). Window environment for Graphic tools

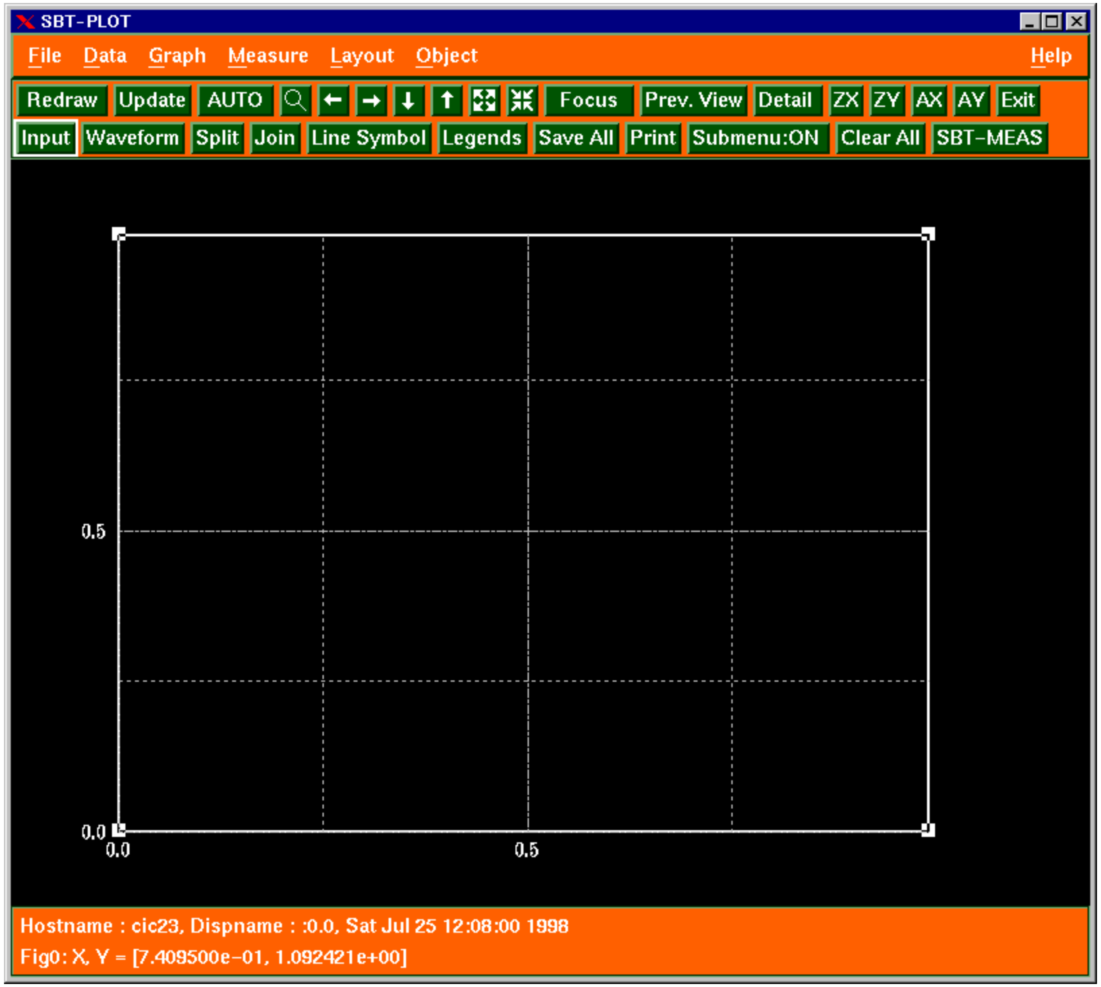
- All the Graphic tools are X-window Based, Must be used within X-window environment
- Use SETENV DISPLAY *your_xhost_server*:0 to redirect the graphic window from a remote host to your working host.
- Use xhost + *remote_host* to allow remote host display window on your machine
- The waveform tool define its own color table, the color table may conflict with other programs, so some colors in waveform window might not be visible, quit other application

(2).SBTPLOT

Environment setup

source /usr/sbt/sbt.cshrc

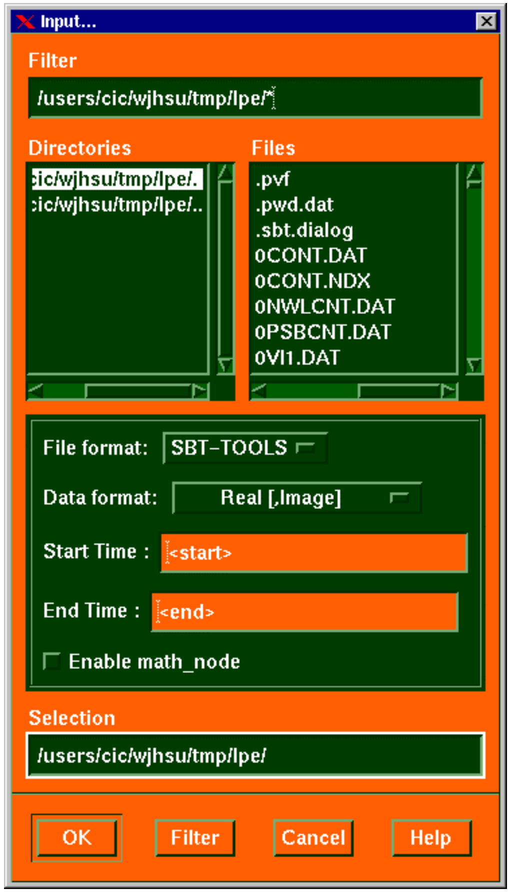
sbtplot



```
Hostname : cic23, Dispname : :0.0, Sat Jul 25 12:08:00 1998
Fig0: X, Y = [7.409500e-01, 1.092421e+00]
```

(3)Select waveform file

Select waveform file



Input...

Filter

/users/cic/wjhsu/tmp/lpe/*

Directories

- cic/wjhsu/tmp/lpe/.
- cic/wjhsu/tmp/lpe/..

Files

- .pvf
- .pwd.dat
- .sbt.dialog
- 0CONT.DAT
- 0CONT.NDX
- 0NWLCNT.DAT
- 0PSBCNT.DAT
- 0VI1.DAT

File format: SBT-TOOLS

Data format: Real [,Image]

Start Time : <start>

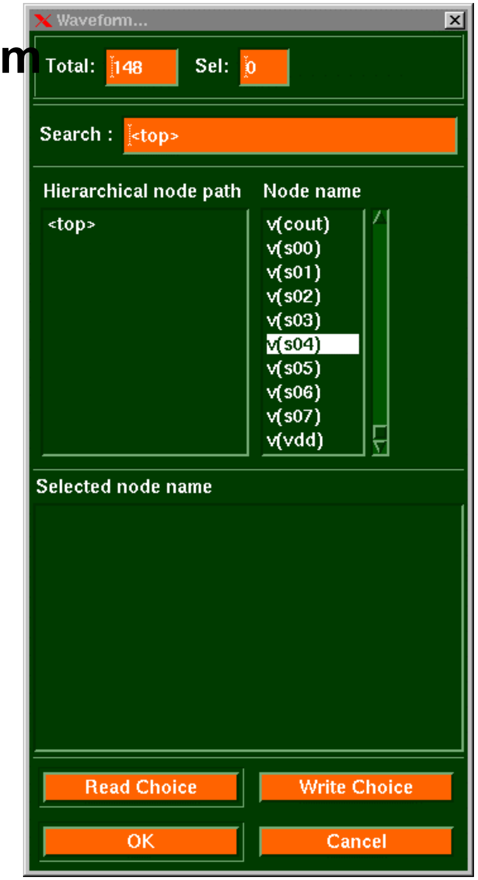
End Time : <end>

☐ Enable math_node

Selection

/users/cic/wjhsu/tmp/lpe/

Select waveform to display



Waveform...

Total: 148 Sel: 0

Search : <top>

Hierarchical node path

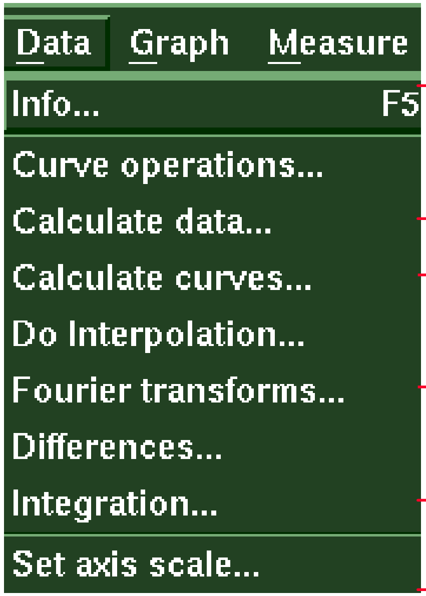
<top>

Node name

- v(cout)
- v(s00)
- v(s01)
- v(s02)
- v(s03)
- v(s04)
- v(s05)
- v(s06)
- v(s07)
- v(vdd)

Selected node name

(4). Data calculation



Hide/kill displayed curves

Evaluates a formula defined by user

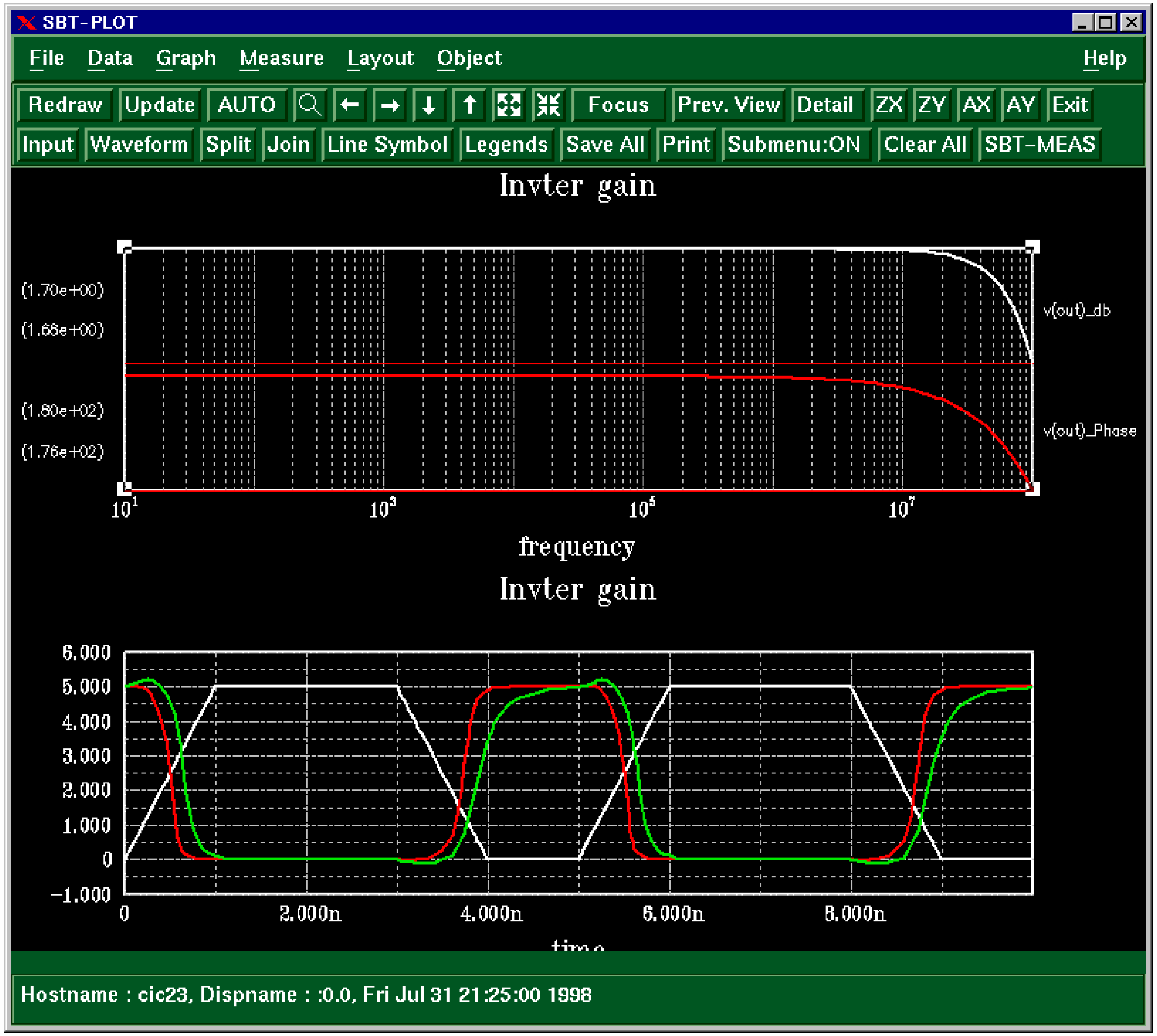
Evaluates equation on curve data

Discrete Fourier Transform < 1000 data point
Fast Fourier Transform - 2^x data points

Numerical integration/Differentiation on curve

Set the current figure type to
linear-linear , log-linear, linear-log, log-log

(5).Waveform



Hostname : cic23, Dispname : :0.0, Fri Jul 31 21:25:00 1998

HSPICE 模擬波形觀測介面-awaves簡介

- HSPICE 係模擬程式，模擬結果依選項指示存於檔案中。
- 原波形界面 gsi 及 hsplot 已由 awaves(avanwaves)取代，98.4版中不提供此兩者之執行檔。
- Gsi 仍可使用awaves 之license
- awaves係視窗程式，需在Xwindow環境下使用。
- 在輸入netlist檔中需指定graphic輸出，hspice才會儲存waveform資料，即加入 .option post 的指令項
- 99.4版的輸出資料，gsi無法接受，需於netlist 檔中加入 .option post_version-9007 的指令項
- 執行awaves 之指令

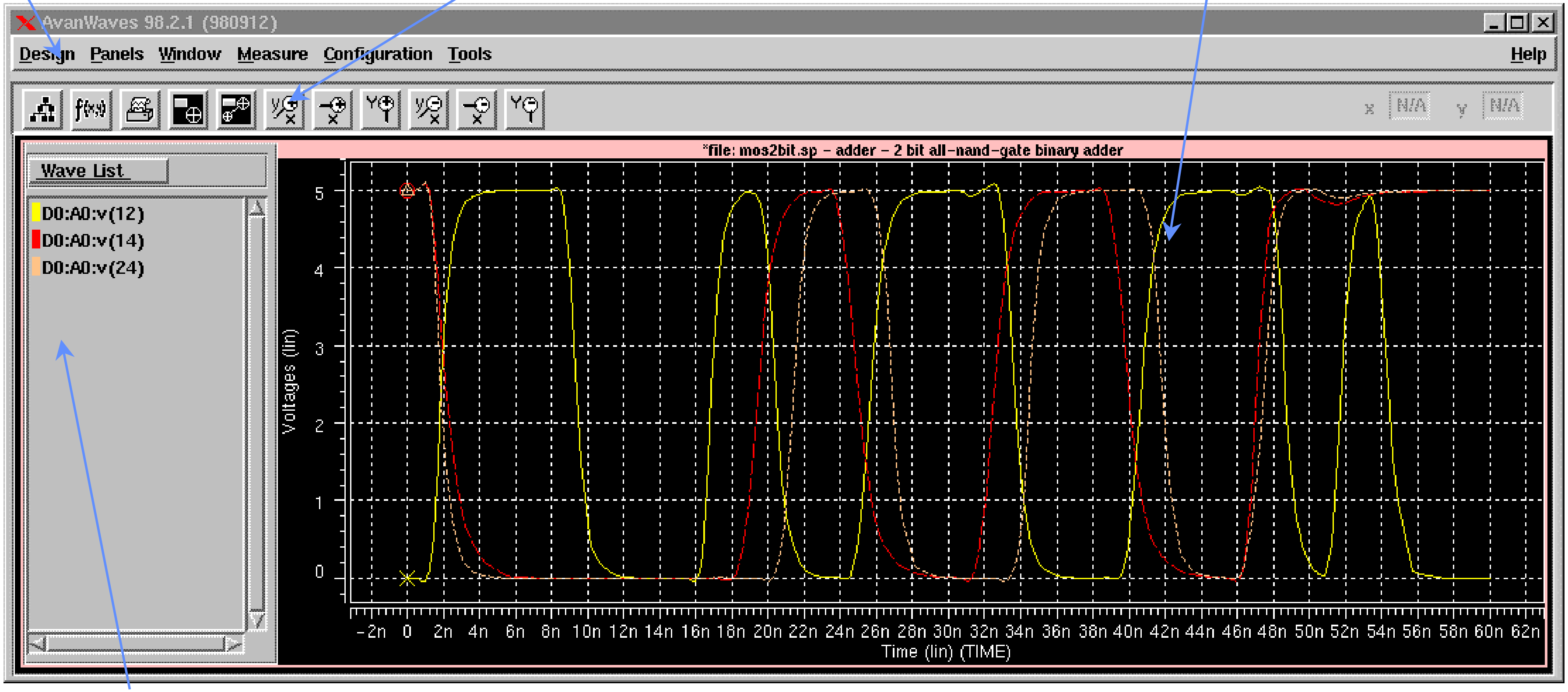
  /usr/meta/cur/bin/awaves

Awaves - Awaves Window

menu

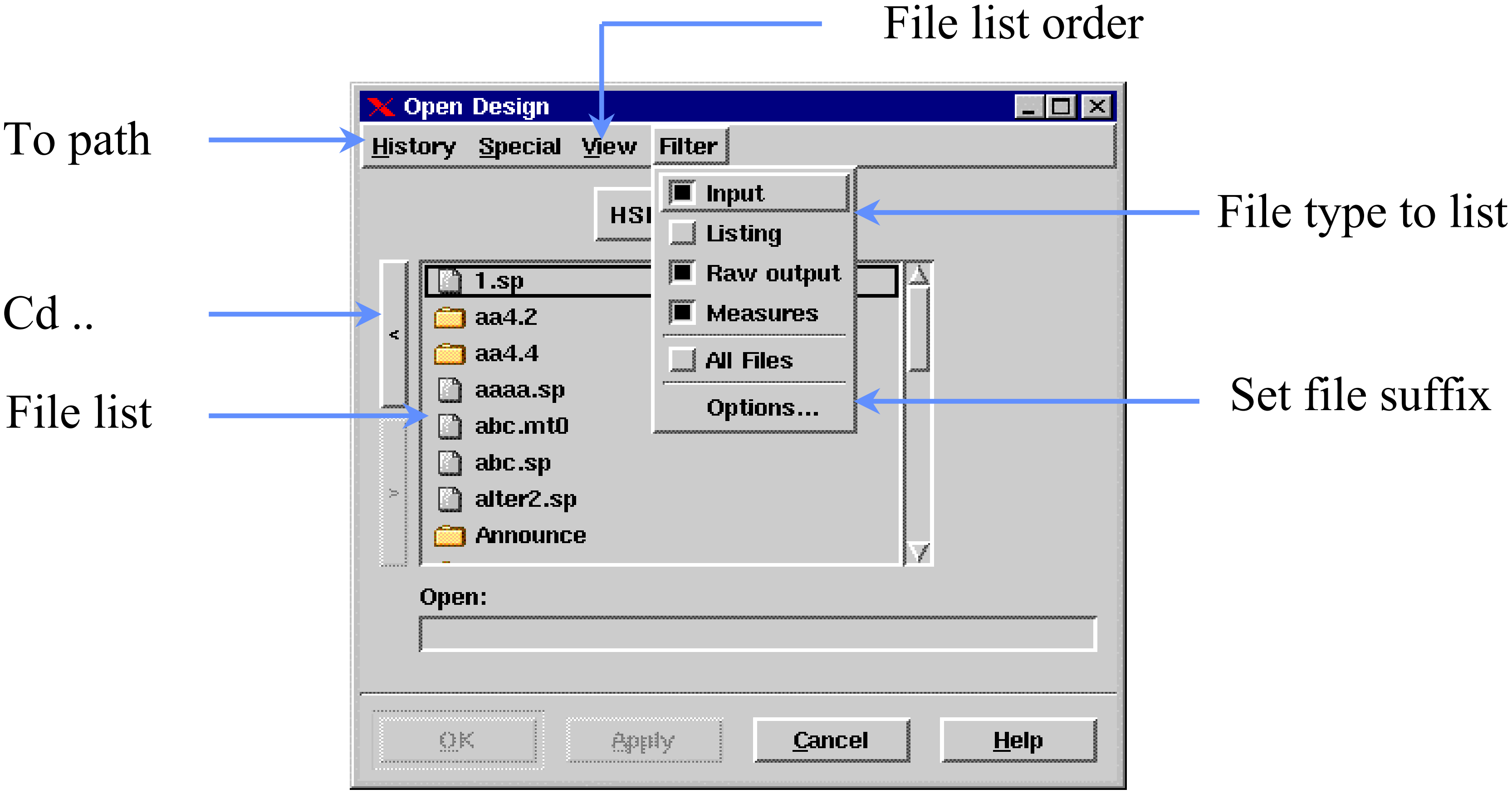
Tool button

waveform


AvanWaves 98.2.1 (980912)
Design
Panels
Window
Measure
Configuration
Tools
Help
x N/A
y N/A
Wave List
🟨 D0:A0:v(12)
🟥 D0:A0:v(14)
🟧 D0:A0:v(24)
*file: mos2bit.sp – adder – 2 bit all–nand–gate binary adder
Voltages (lin)
0
1
2
3
4
5
-2n 0 2n 4n 6n 8n 10n 12n 14n 16n 18n 20n 22n 24n 26n 28n 30n 32n 34n 36n 38n 40n 42n 44n 46n 48n 50n 52n 54n 56n 58n 60n 62n
Time (lin) (TIME)

Data names

Awaves- Open Design



Filter:
- ■ Input
- □ Listing
- ■ Raw output
- ■ Measures
- □ All Files
- Options...

Open:

To path

Cd ..

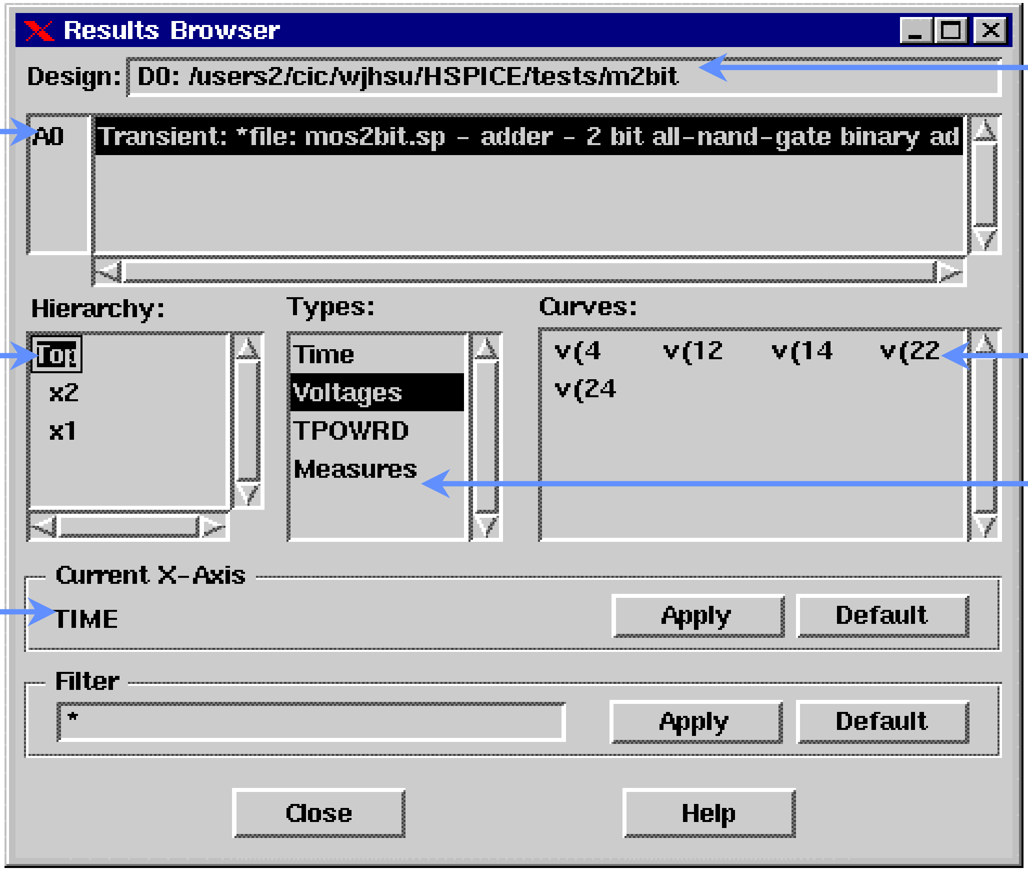
File list

File list order

File type to list

Set file suffix

Awaves- Result Browser



Results Browser

Design: D0: /users2/cic/wjhsu/HSPICE/tests/m2bit

A0 Transient: *file: mos2bit.sp - adder - 2 bit all-nand-gate binary ad

Hierarchy:
Top
x2
x1

Types:
Time
Voltages
TPOWRD
Measures

Curves:
v(4 v(12 v(14 v(22
v(24

Current X-Axis
TIME

Filter
*

Design name

Analysis type in design

Instance name and hierarchy

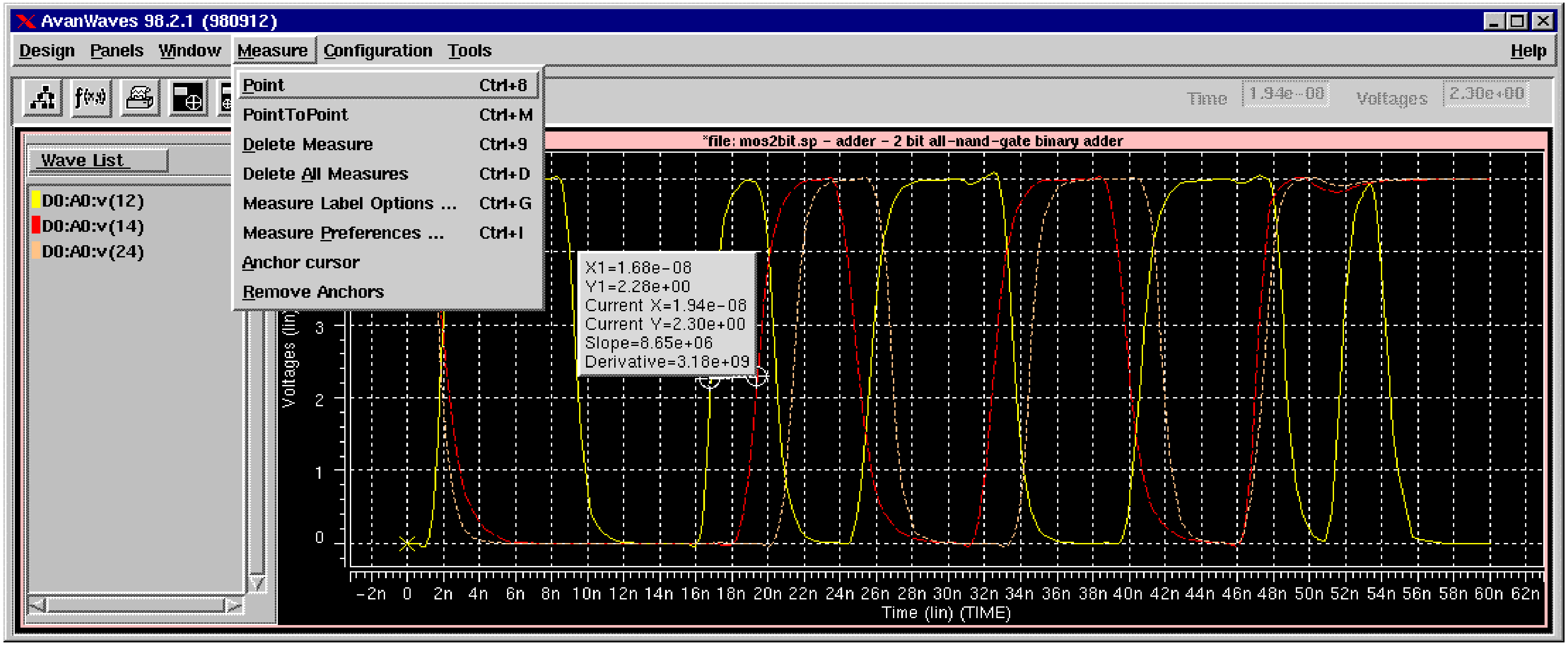
Data to be shown

Data type

Set X axis

Awaves - Measurement

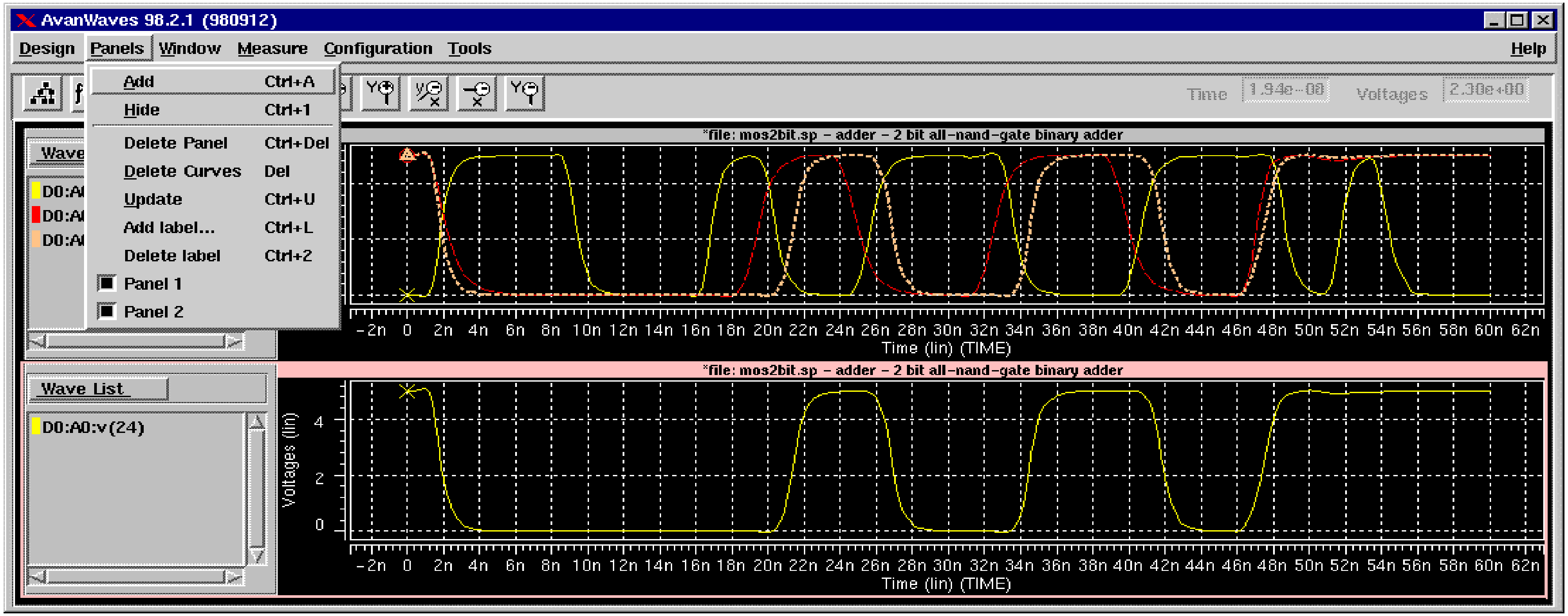
On window measurement function



```
X1=1.68e-08
Y1=2.28e+00
Current X=1.94e-08
Current Y=2.30e+00
Slope=8.65e+06
Derivative=3.18e+09
```

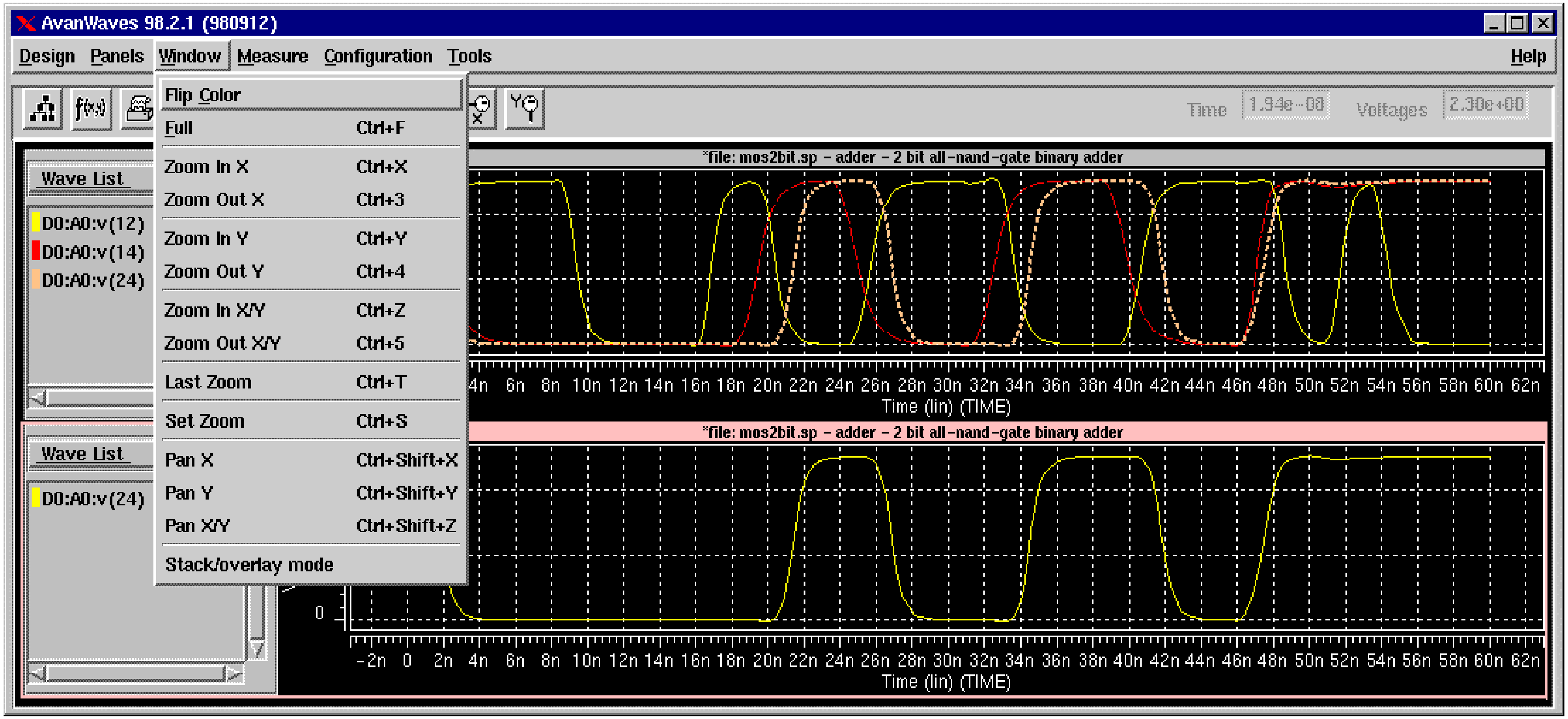
Awaves - Multiple Panels

Multiple panels can be displayed on window
Maximum number of panels displayed depends on window size
Waveforms can be dragged and drop into other panels


AvanWaves 98.2.1 (980912)
Design
Panels
Window
Measure
Configuration
Tools
Help
Add Ctrl+A
Hide Ctrl+1
Delete Panel Ctrl+Del
Delete Curves Del
Update Ctrl+U
Add label... Ctrl+L
Delete label Ctrl+2
■ Panel 1
■ Panel 2
Time 1.94e-08
Voltages 2.30e+00
Wave
D0:A
D0:A
D0:A
Wave List
D0:A0:v(24)
*file: mos2bit.sp – adder – 2 bit all-nand-gate binary adder
-2n 0 2n 4n 6n 8n 10n 12n 14n 16n 18n 20n 22n 24n 26n 28n 30n 32n 34n 36n 38n 40n 42n 44n 46n 48n 50n 52n 54n 56n 58n 60n 62n
Time (lin) (TIME)
*file: mos2bit.sp – adder – 2 bit all-nand-gate binary adder
Voltages (lin)
4
2
0
-2n 0 2n 4n 6n 8n 10n 12n 14n 16n 18n 20n 22n 24n 26n 28n 30n 32n 34n 36n 38n 40n 42n 44n 46n 48n 50n 52n 54n 56n 58n 60n 62n
Time (lin) (TIME)

Awaves - View Port Management

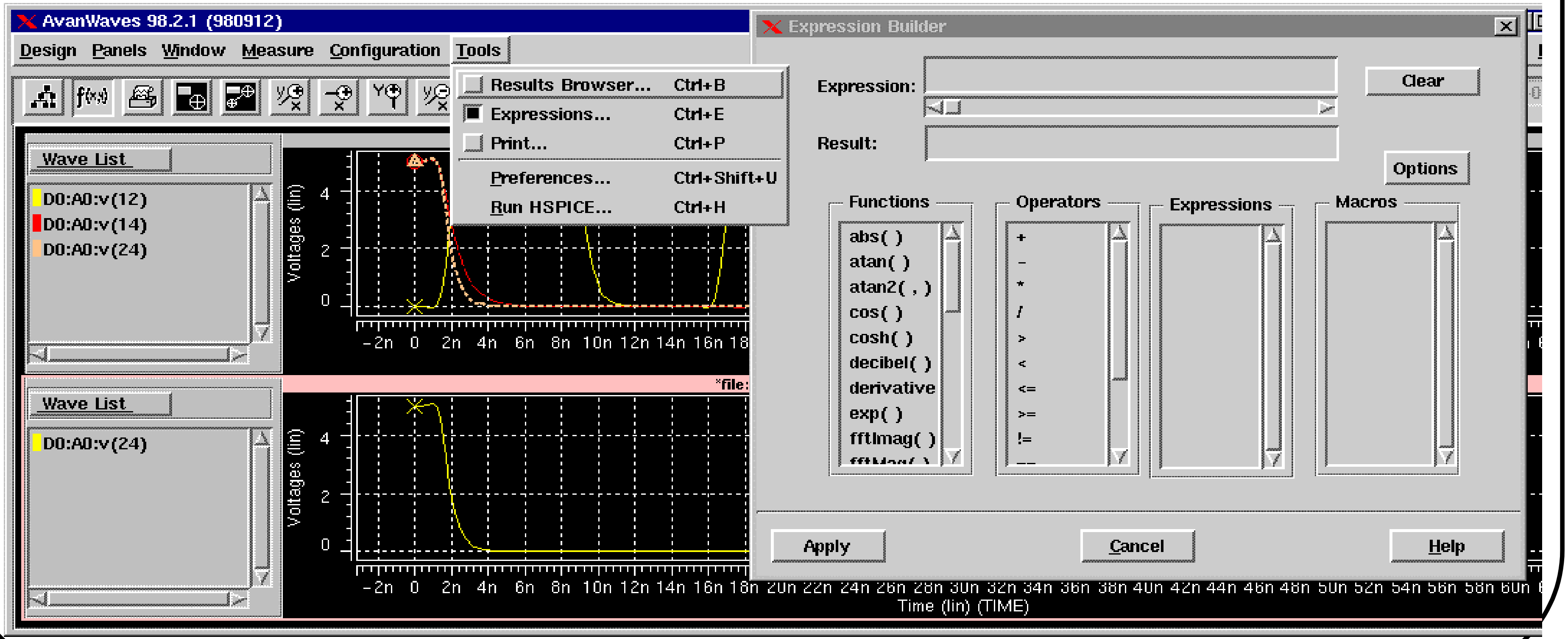
Zoom size is independent between panels



Awaves - Expressions

Expressions provide capability for data calculation

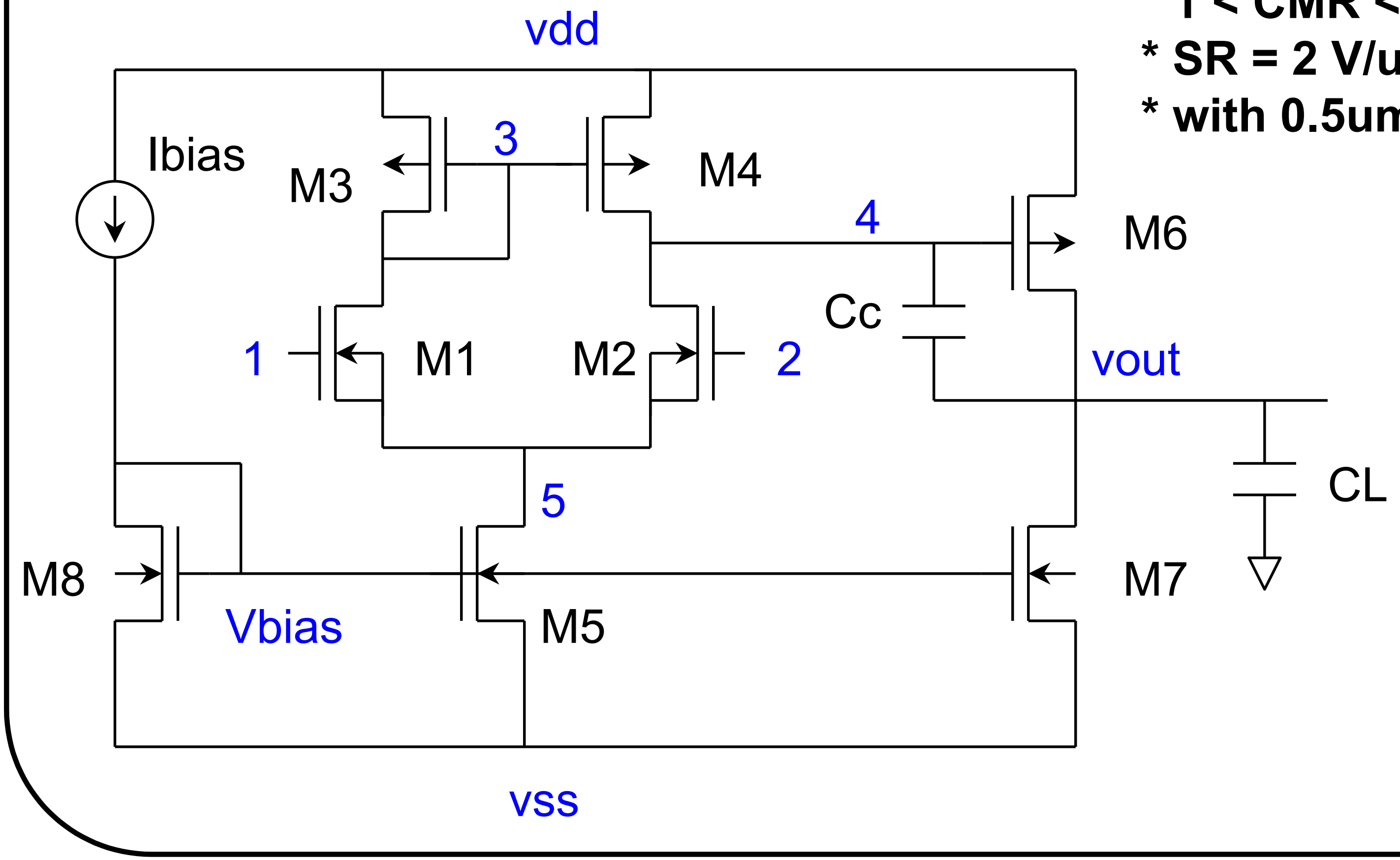
Can use drag&drop to type expression



Contents

1. SPICE Overview
2. Simulation Input and Controls
3. Sources and Stimuli
4. Analysis Types
5. Simulation Output and Controls
6. Elements and Device Models
7. Optimization
8. Control Options & Convergence
9. Graphic Tools
10. Applications Demonstration

(1). Two-stage OP AMP Design



* Target specification :
* CL = 4pF, Av>4000,
* GB=2MHz
* 1 < CMR < 4 , 0.8 < Vout < 4.2
* SR = 2 V/us , Pdiss < 10mW ,
* with 0.5um UMC process

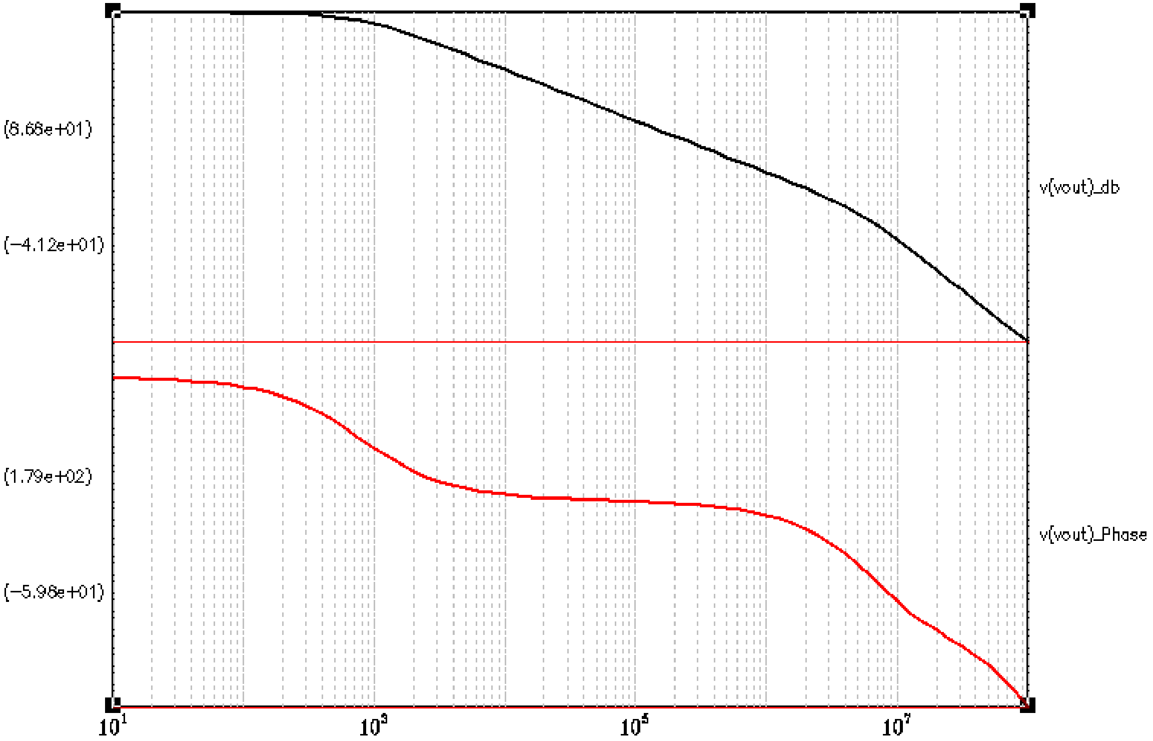
(2). Netlist

```
*Two stage OP design
.lib ”9905spice.model" mos_tt
.option post nomod
.TEMP 27
* Netlist information
M1 3 1 5 0 nmos L=2u W=8u AS=18p AD=18p
+                    PS=18u PD=18u
M2 4 2 5 0 nmos L=2u W=8u AS=18p AD=18p
+                    PS=18u PD=18u
M3 3 3 vdd vdd pmos L=10u W=10u AS=12p AD=12p PS=16u PD=16u
M4 4 3 vdd vdd pmos L=10u W=10u AS=12p AD=12p PS=16u PD=16u
M5 5 vbias vss vss nmos L=2u W=7u AS=49p AD=49p PS=26u PD=26u
M6 vout 4 vdd vdd pmos L=2u W=70u AS=490p AD=490p PS=150u PD=150u
M7 vout vbias vss vss nmos L=2u W=130u AS=930p AD=930p
+                                   PS=260u PD=260u
M8 vbias vbias vss vss nmos L=2u W=7u AS=49p AD=49p PS=26u PD=26u
```

```
* Feedback CAP
Cc vout 4 0.44pF
Cl vout 0 4pF
Ibias vdd vbias 8.8u

* Voltage sourses
vdd vdd 0 5v
vss vss 0 0v
```

(3). AC Frequency Analysis

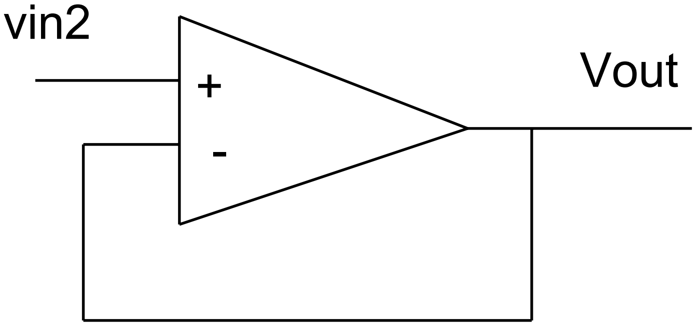
```
vin2 2 0 2.5v
vin1 1 0 DC 2.5v AC 1

*.OP

* AC Analysis function
.ac dec 10 10 100MEG
.probe ac vdb(vout)
+ vp(vout) vdb(4) vp(4)
.meas ac Unit_gain
+ when vdb(vout)=0
.meas ac phase_mar
+ FIND vp(vout) when vdb(vout)=0
```

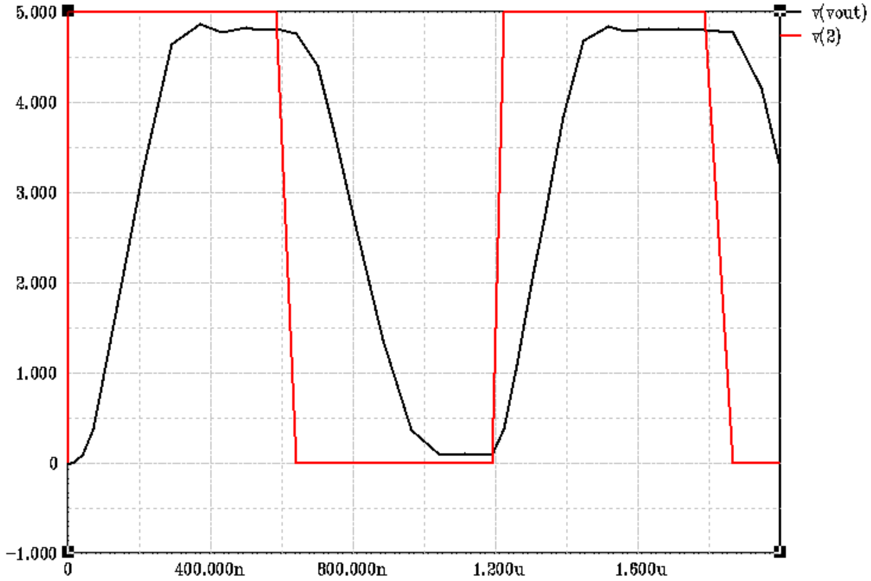


(4).Transient Analysis

Slew Rate analysis

```
* Transient analysis section

M1 3 vout 5 0 nmos L=2u W=8u AS=18p AD=18p PS=18u PD=18u
vin2 2 0 pulse(0v 5v 1n 1p 1p 600n 1200n)
.tran 5n 2u
.probe tran
.save all
```




400.000n
800.000n
1.200u
1.600u
v(vout)
v(2)