Digital Integrated Circuits

YuZhuo Fu

contact:fuyuzhuo@ic.sjtu.edu.cn

Office location：417 room

WeiDianZi building,No 800 DongChuan road,MinHang Campus

3.CMOS Inverter

outline

- CMOS at a glance
- CMOS static behavior
- CMOS dynamic behavior
- Power, Energy, and Energy Delay
- Perspective tech.
- SPICE Simulation

SPICE

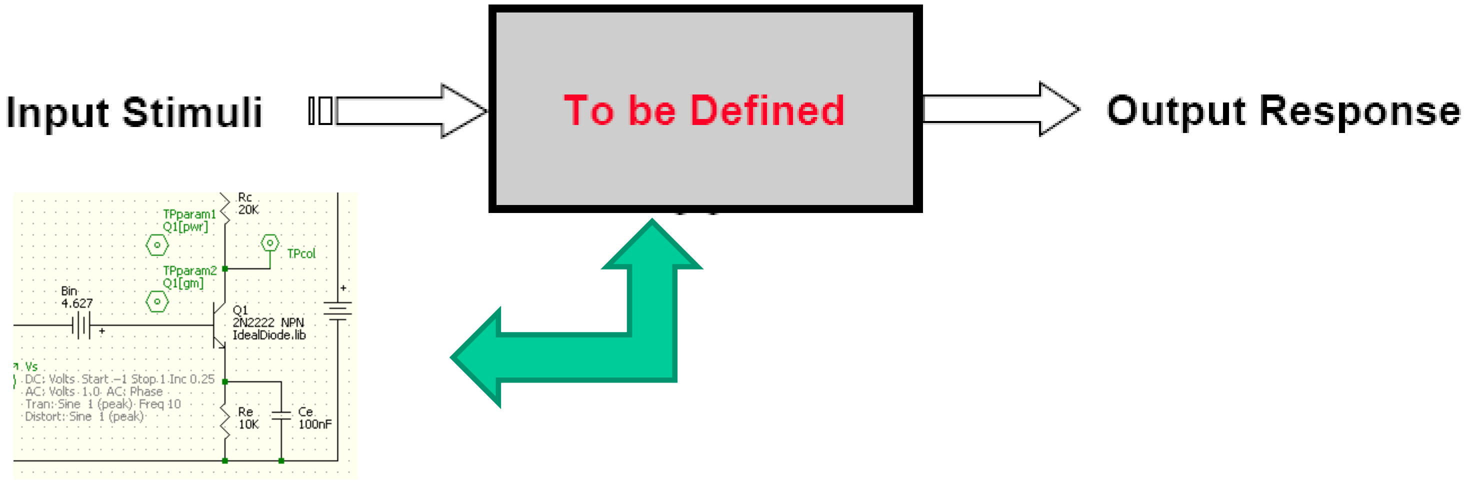
- Know Basic elements for circuit simulation
- Learn the basic usage of standalone spice simulators
- Know the concept of device models
- Learn the usage of waveform tools
- Advanced features of spice simulator

contents

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

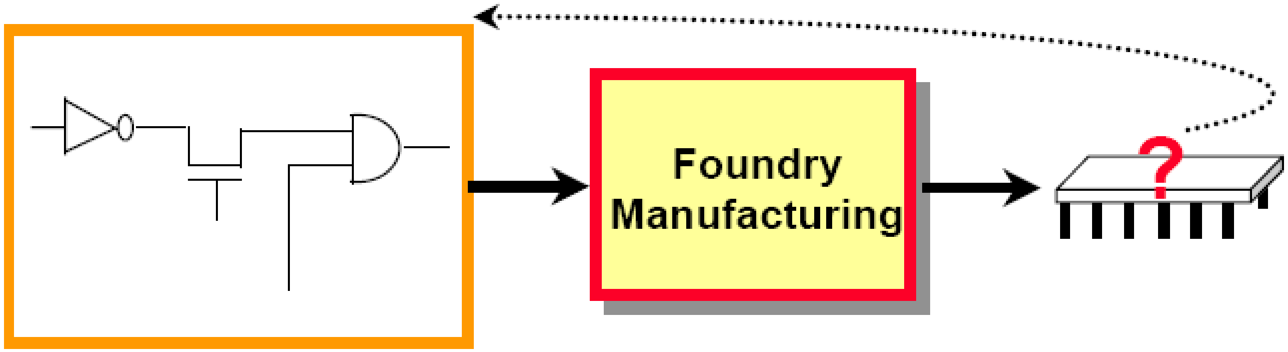
Overview of SPICE

- (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.



Overview of SPICE

- (2)What is Simulation
- Simulation:To predict the Circuit/System Characteristic after manufacturing



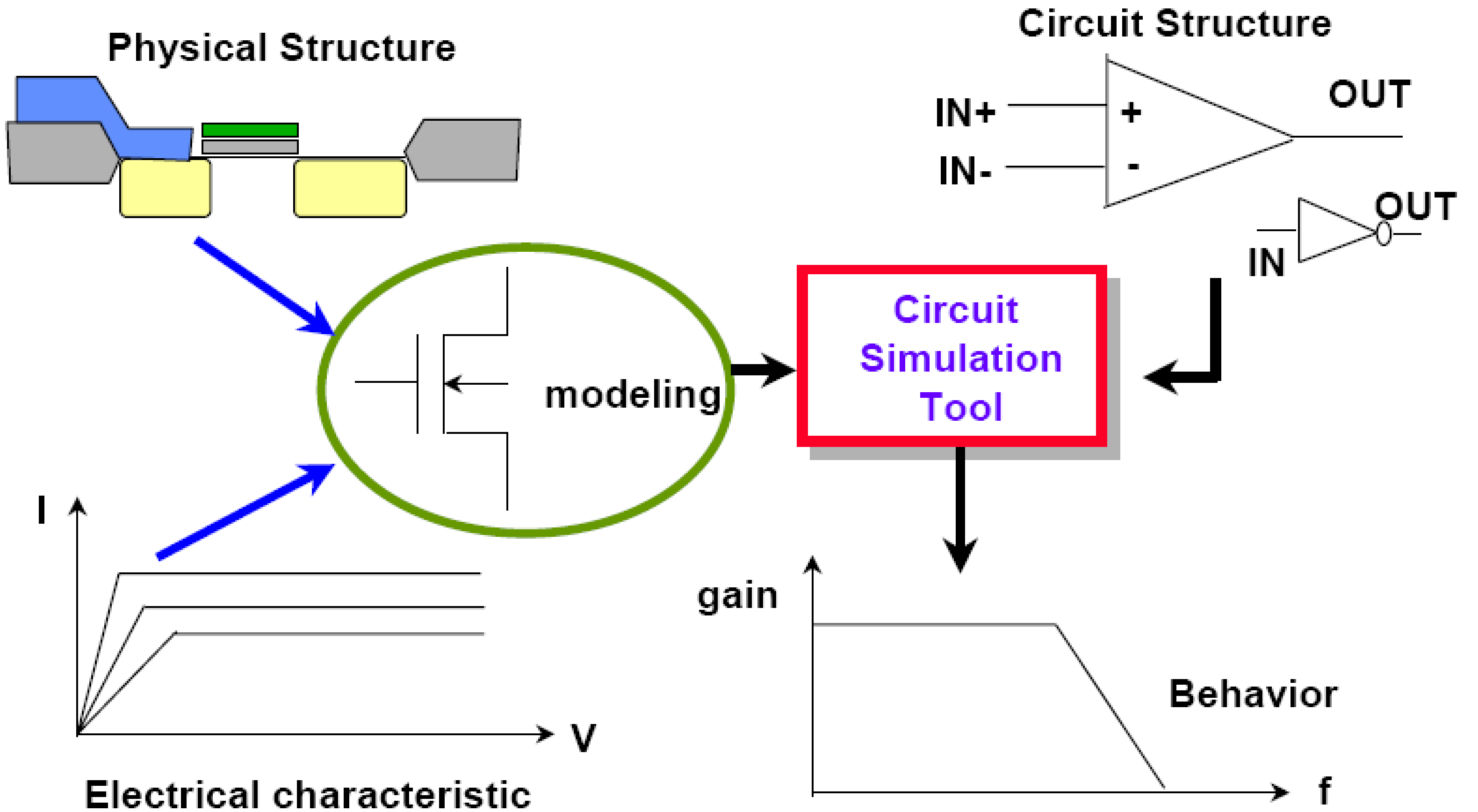
- Depends on the component behavior, simulation categories include :

Complexity Capacity

- Functional simulation
- Logic/Gate Level Simulation
- Switch/Transistor Level Simulation
- Circuit Simulation
- Device Simulation

Overview of SPICE

- (3). Circuit Simulation Background



Overview of SPICE

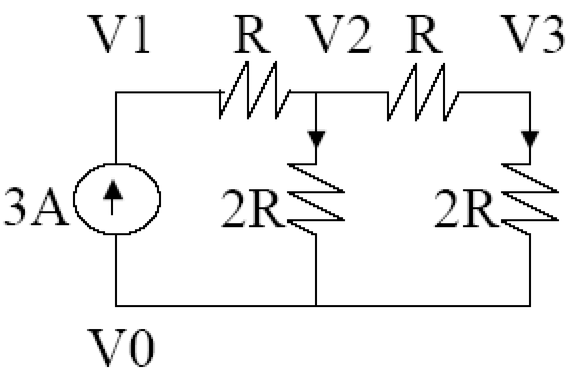
- (4). SPICE Background
- SPICE : Simulation Program with Integrated Circuit Emphasis
- Numerical Approach to Circuit Simulation
- Developed by University of California/Berkeley (UCB)
- Widely Adopted, Become *De Facto Standard*
- Circuit Node/Connections Define a Matrix
- Must Rely on Sub-Models for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

Overview of SPICE

- (5). SPICE Background
- 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.

Overview of SPICE

- (6). Solution for Linear Network



R=5ohm

$$\begin{pmatrix} 0.2 & 0 & -0.1 & -0.1 \\ 0 & 0.2 & -0.2 & 0 \\ -0.1 & -0.2 & 0.5 & -0.2 \\ 0 & 0 & -0.2 & 0.2 \end{pmatrix} \begin{bmatrix} V_0 \\ V_1 \\ V_2 \\ V_3 \end{bmatrix} = \begin{bmatrix} -3 \\ 3 \\ 0 \\ 0 \end{bmatrix}$$

$$\begin{pmatrix} 0.2 & -0.2 & 0 \\ -0.2 & 0.5 & -0.2 \\ 0 & -0.2 & 0.2 \end{pmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} = \begin{bmatrix} 3 \\ 0 \\ 0 \end{bmatrix}$$

V_0 ground

With Gaussian elimination

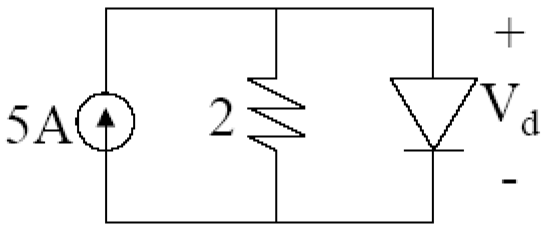
$$\begin{pmatrix} 0.2 & -0.2 & 0 \\ 0 & 0.3 & -0.2 \\ 0 & 0 & 0.25 \end{pmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} = \begin{bmatrix} 3 \\ 3 \\ 3 \end{bmatrix}$$

Results : $V_3 = 12V$, $V_2 = 18V$, $V_1 = 33V$

Overview of SPICE

- <u>(7).Iteration and approximation</u>

<u>-How solution is obtained</u>



$$I_d = 1pA*[\exp(40*V_d)-1]$$

$$5 = V_d/2 + I_d$$

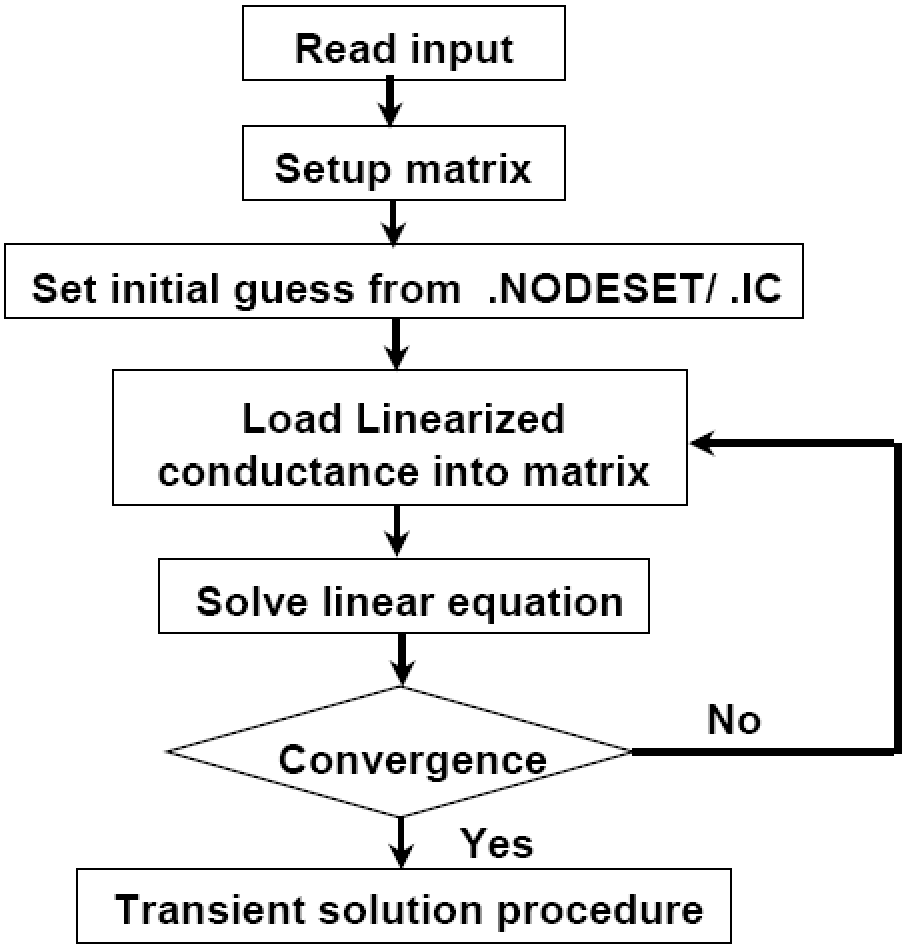
$$5 = V_d/2 + 1pA*[\exp(40*V_d)-1]$$

$$V_{d+1} = V_d - F(V_d)/F'(V_d)$$

	V_d	V_{d+1}	Delta V
1	1	0.975001	0.02499
2	0.975001	0.950002	0.02499
3	0.950002	0.925005	0.02499
4	0.925005	0.900015	0.02499
5	0.900015	0.875041	0.02497
6	0.875041	0.850113	0.02493
7	0.850117	0.825309	0.02481
8	0.825309	0.800838	0.02447
9	0.800838	0.777250	0.02359
10	0.777250	0.755885	0.02136
11	0.755885	0.739445	0.01644
12	0.739447	0.730983	0.00846
13	0.730983	0.729186	0.00179
14	0.729186	0.729119	0.00007

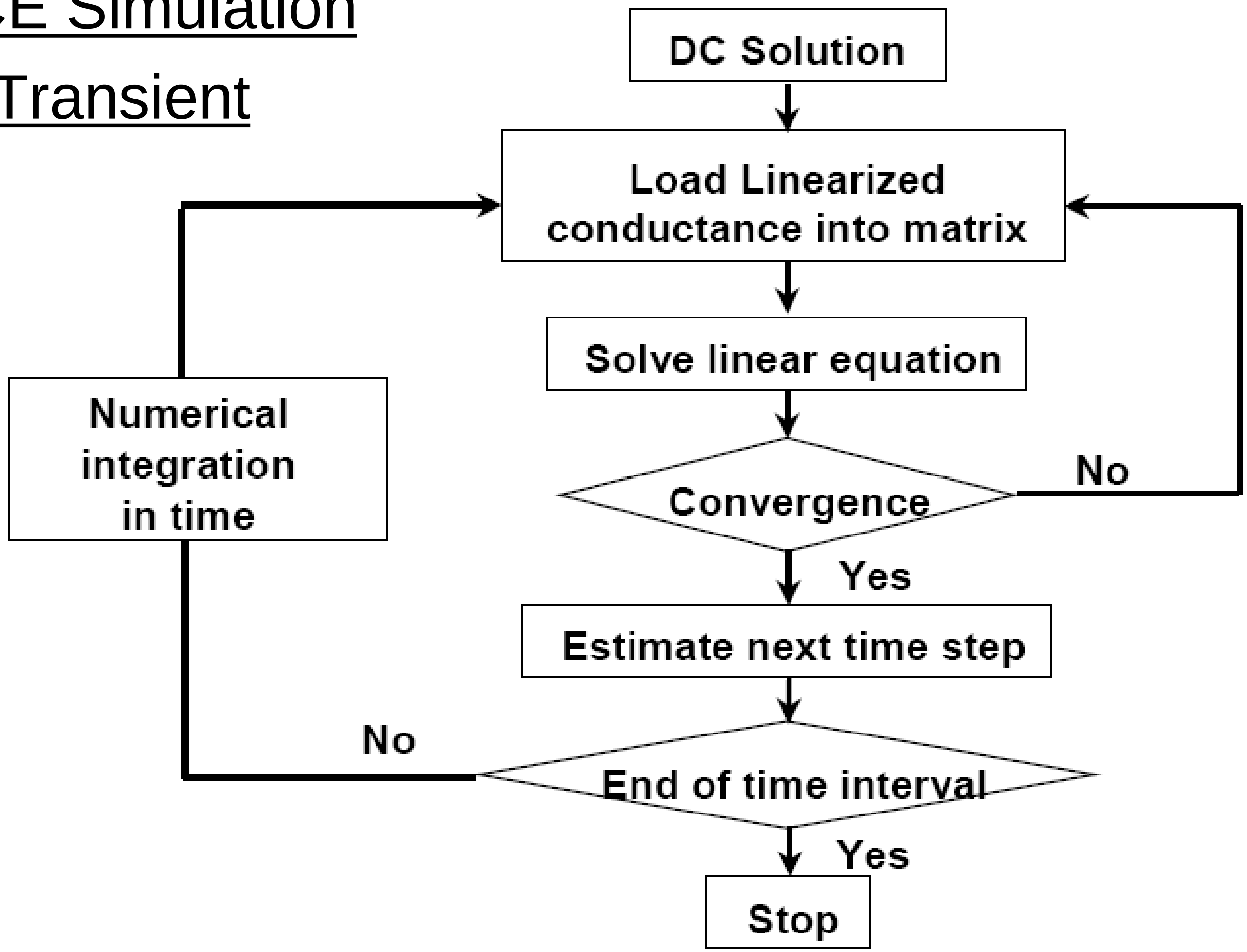
Overview of SPICE

- (8). SPICE Simulation Algorithm - DC



SPICE overview

- (9). SPICE Simulation Algorithm – Transient



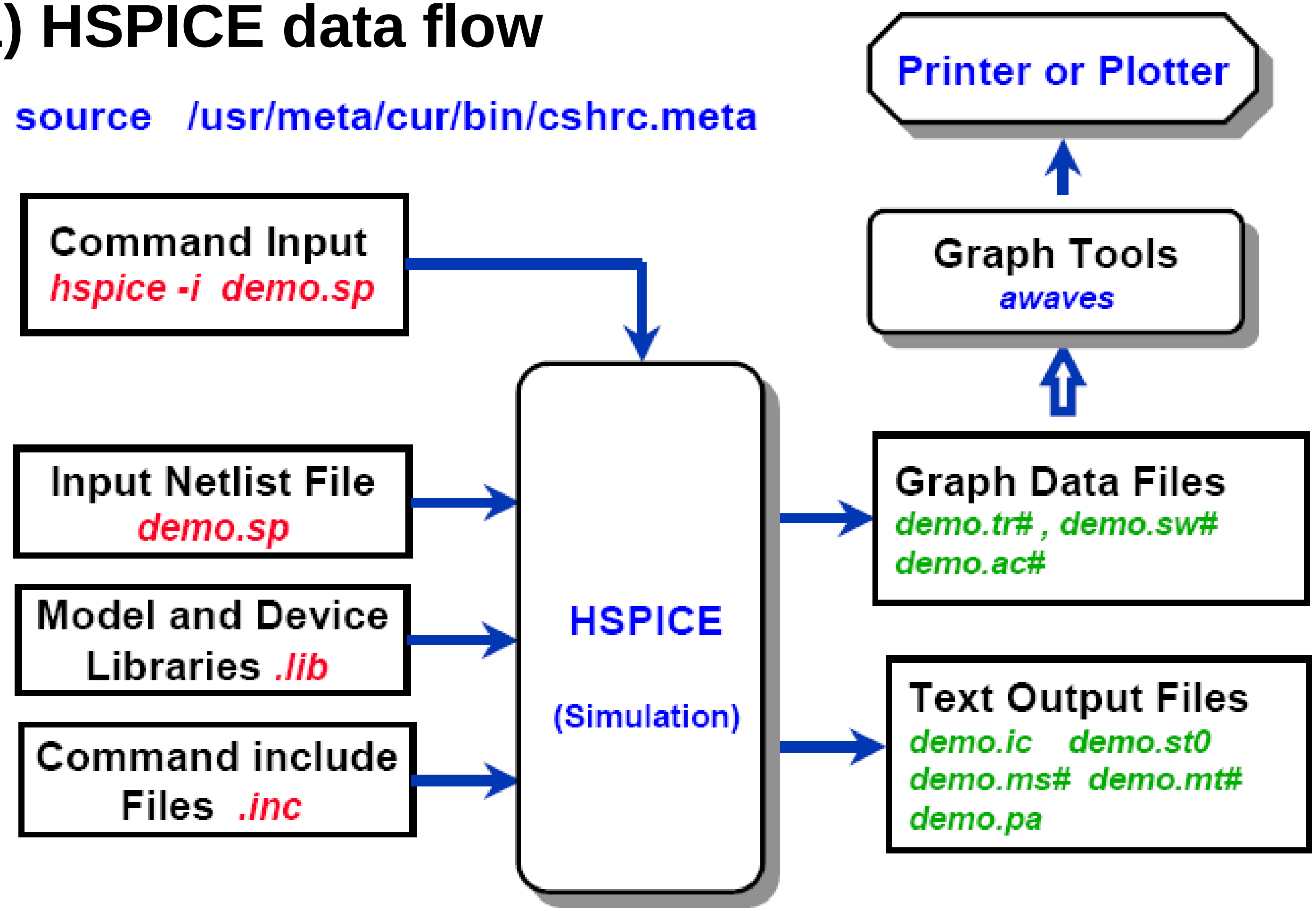
contents

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

Simulation input and control

- **(1) HSPICE data flow**

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



Simulation input and control

- **(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

Simulation input and control

- **(3) Netlist Structure (SPICE Preferred)**

Title	Title Statement - Ignored during simulation
Controls	.option nomod nopage
	.tran 1 10
	.print v(5) i(r1)
	.plot v(3) v(in)
	* voltage sources
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
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
Models & Subckts	.model... or .LIB or .Subckt
End file	.end

Simulation input and control

- **(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)

Simulation input and control

- **(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)

Simulation input and control

- **(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-9
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

Simulation input and control

- **(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.

Simulation input and control

- **(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
```

Simulation input and control

- **(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

Simulation input and control

- **(7). Input Control Statements: .DATA**
- .DATA Statement: Inline or Multiline .DATA Example

Inline .DATA Example

```
.TRAN 1n 100n SWEEP DATA=devinf
.AC DEC 10 Hz 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
```

Simulation input and control

- **(8). Input Control Statements: .TEMP**
- .TEMP Statement: Description
  - When TNOM is not Specified, it will Default to 25 ^oC for HSPICE
  - Example 1:

    .TEMP 30 *Ckt simulated at 30 ^oC
  - Example 2:

    .OPTION TEMP = 30 *Ckt simulated at 30 ^oC
  - Example 3:

    .TEMP 100

    D1 n1 n2 DMOD DTEMP=130 *D1 simulated at 130 ^oC

    D2 n3 n4 DMOD *D2 simulated at 100 ^oC

    R1 n5 n6 1K

Simulation input and control

- **(9). 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

Simulation input and control

- **(10). 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
.LIB "~users/model/tsmc/logic06.mod" TT
.UNPROTECT
```

Simulation input and control

- **(11)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
.ENDS [subname]

Simulation input and control

- **(11). Hierarchical Circuits, Parameters, and Models (Cont.)**
- .SUBCKT Statement : Examples

```
.PARAM VALUE=5V WN=2u WP=8u
*
.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
```

- 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
```

Simulation input and control

- **(12). Example Circuit**

subckt call → (points to the `x1 out inp vdd inv` line)

```
Invter gain
.lib 'logs353v.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
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

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

3. Sources and Stimuli

- Source / Stimuli : drive source of circuit
- Source types
  - 1. Independent DC Sources(supply fixed voltage/current)
  - 2. Independent AC/TRAN Sources(for input signal)
  - 3. dependent DC/AC/TRAN Sources(for models)
    - 压控电压源(VCVS-Voltage-Controlled Current Sources)
    - 压控电流源(VCCS)
    - 流控电压源(CCVS)
    - 流控电流源(CCCS)

3. Sources and Stimuli

- **(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*

3. Sources and Stimuli

- **(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)

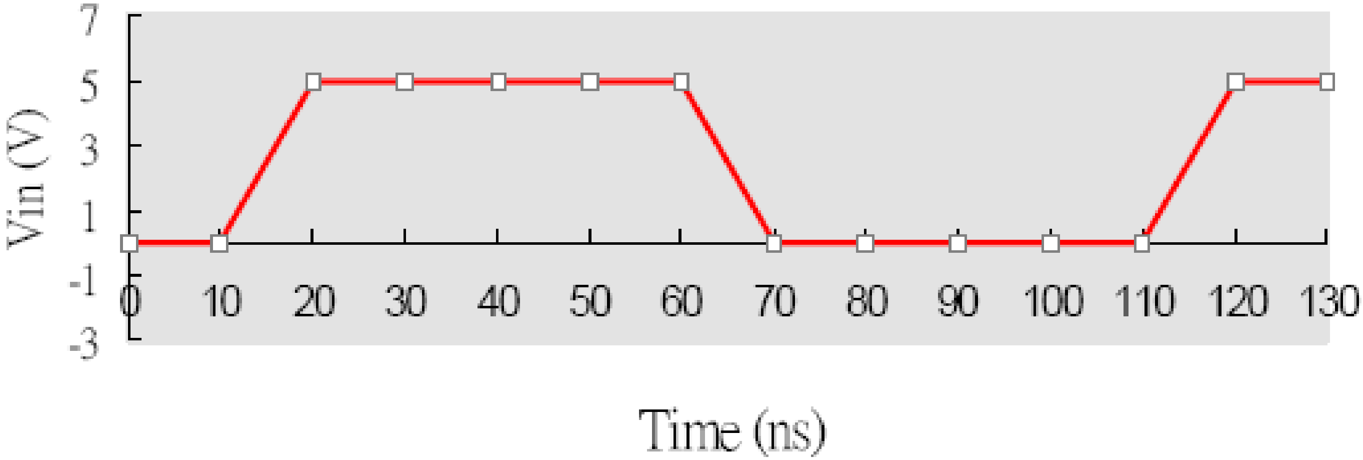
3. Sources and Stimuli

- **(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)



3. Sources and Stimuli

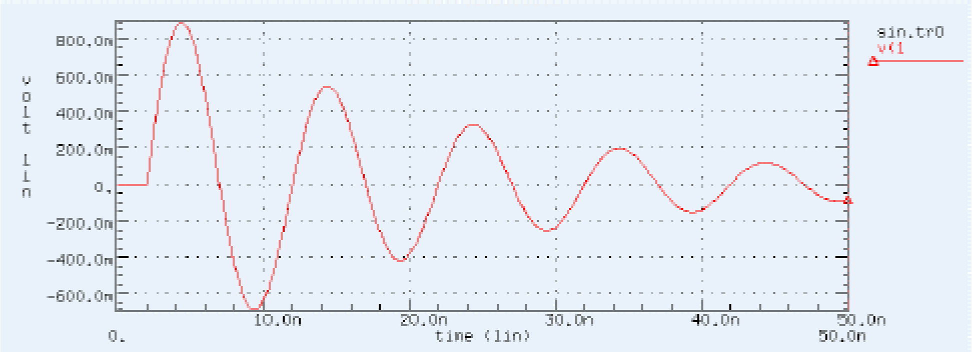
- **(2). Indep. Source Functions : Transient Sources(Cont.)**
- Sinusoidal Source Function : SIN
- Syntax :

SIN (Voffset Vacmag < Freq Tdelay Dfactor >)

Voffset + Vacmag e-(t-TD) *Dfactor * sin(2π Freq(t-TD))*

- Example :

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



3. Sources and Stimuli

- **(2). Indep. Source Functions : Transient Sources(Cont.)**
- Piecewise Linear Source Function : PWL or PL
- Syntax :

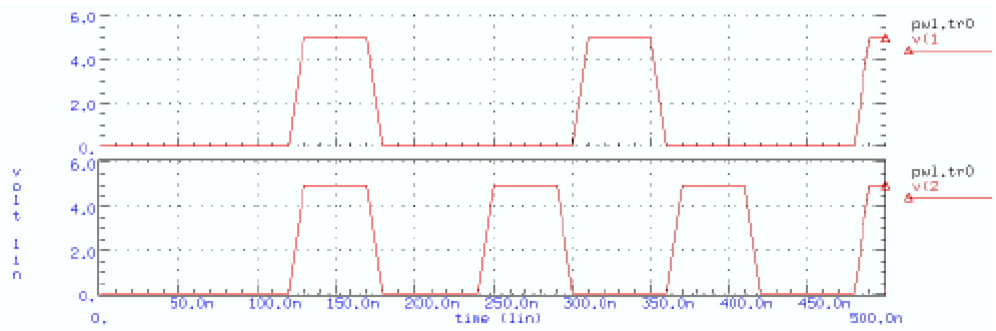
PWL (<t1 v1 t2 v2 ......> <R<=repeat>> <Tdelay=delay>)

$ 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



3. Sources and Stimuli

- **(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

contents

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

4. Analysis Types

- **(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

4. Analysis Types

- **(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

4. Analysis Types

- **(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
  - .PZ : Performs Pole/Zero Analysis (.OP is not Required)
  - .TF : Calculate DC Small-Signal Transfer Function (.OP is not Required)
- .DC Statement Sweep :
  - Any Source Value Any Parameter Value
  - Temperature Value
  - DC Circuit Optimization
  - DC Model Characterization

4. Analysis Types

- **(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

- **(4). Analysis Types : AC Sweep & Small Signal Analysis**
- AC Analysis Statements :
  - .AC : Calculate Frequency-Domain Response
  - .NOISE : Noise Analysis
- .AC Statement Sweep :
  - Frequency Element
  - Temperature
  - Optimization
  - .param Parameter

4. Analysis Types

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

4. Analysis Types

- **(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

- **(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

4. Analysis Types

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

4. Analysis Types

- **(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
```

4. Analysis Types

- **(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

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

5. Simulation Output and Controls

- **(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

5. Simulation Output and Controls

- **(1). Output Files Summary:**

Output File Type	Extension
Output Listing	.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

5. Simulation Output and Controls

- (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)
- Element Template :
- @x1.mn1[vth]
- @x1.mn1[gds]
- @x1.mn1[gm],@x1.mn1[gbs],@x1.mn1[cgd]

R : Real
I : Imaginary
M : Magnitude
P : Phase
DB : Decibels

5. Simulation Output and Controls

- **(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. Simulation Output and Controls

- **(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)...

5. Simulation Output and Controls

- **(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)
```

5. Simulation Output and Controls

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

- .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

5. Simulation Output and Controls

- **(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

5. Simulation Output and Controls

- **(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>
  - TARG ... : TARG targ_var VAL=targ_value <TD=time_delay>
  - + <CROSS=n|LAST> <RISE=r_n|LAST> <FALL=f_n|LAST>
  - TRIG ... : TRIG AT=value
  - <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
```

5. Simulation Output and Controls

- **(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
```

5. Simulation Output and Controls

- **(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>

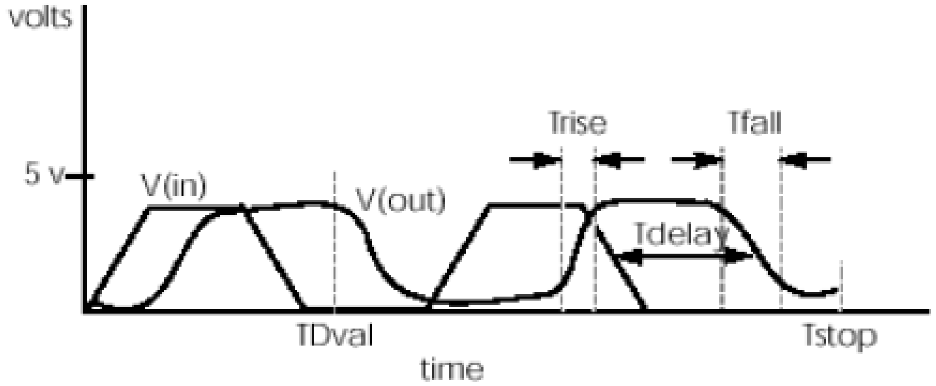
  - result : Name Given the Measured Value in HSPICE Output
  - WHEN ... : WHEN out_var2=val|out_var3 <TD=time_delay>
  - + <CROSS=n|LAST> <RISE=r_n|LAST> <FALL=f_n|LAST>
  - <Optimization Option> : <GOAL=val> <MINVAL=val> <WEIGHT=val>
- Example:

```
.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
```

5. Simulation Output and Controls

- **(12). MEASURE Statement : Application Examples**
- Rise, Fall, and Delay Calculations :

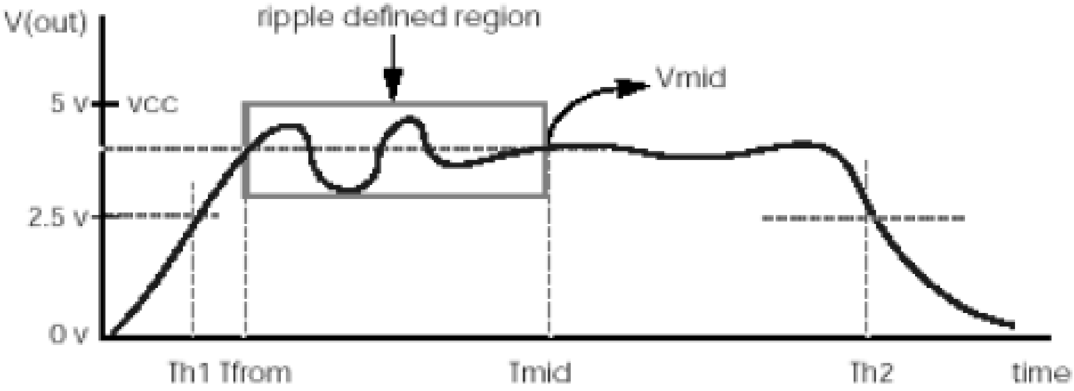
```
.meas TRAN Vmax MAX v(out) FROM=TDval TO=Tstop
.meas TRAN Vmin MIN v(out) FROM =TDval TO =Tstop
.meas TRAN Trise TRIG v(out) VAL='Vmin+0.1*Vmax' TD=TDval RISE=1
+ TARG v(out) VAl='0.9*Vmax' RISE=1
.meas TRAN Tfall TRIG v(out) VAL='0.9*Vmax' TD=TDval FALL=2
+ TARG v(out) VAl='Vmin+0.1*Vmax' FALL=2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



5. Simulation Output and Controls

- **(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'
```



5. Simulation Output and Controls

- **(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

6. Elements & Device Models

- **(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)

6. Elements & Device Models

- **(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

6. Elements & Device Models

- **(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

6. Elements & Device Models

- **(4). Active Device : MOSFET Introduction**
- MOSFET Model Overview :
  - MOSFET Defined by :
    - (1). MOSFET Model & Element Parameters
    - (2). Two Submodel : CAPOP & ACM
  - ACM : Modeling of MOSFET Bulk_Source & Bulk_Drain Diodes
  - CAPOP : Specifies MOSFET Gate Capacitance
- 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

6. Elements & Device Models

- **(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 MODN 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. Elements & Device Models

- **(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:

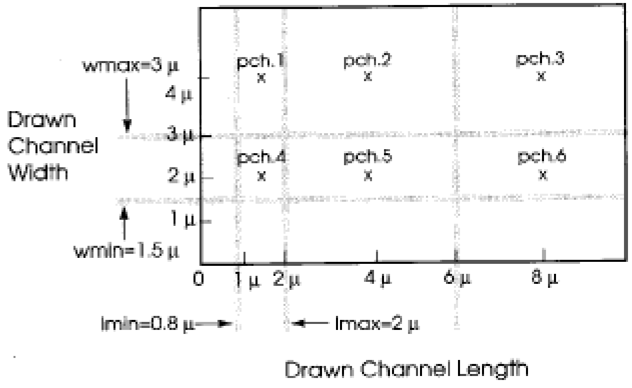
  .LIB TT or (FF|SS|FS|SF)
  .param toxn=0.0141 toxp=0.0148......
  .lib '~/simulation/model/cmos.l' MOS
  .ENDL TT or (FF|SS|FS|SF)

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

6. Elements & Device Models

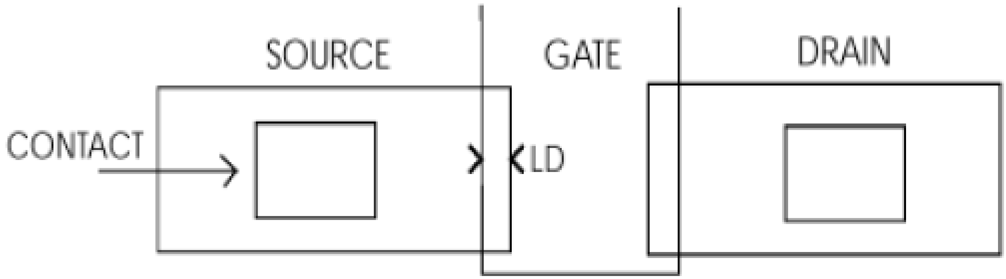
- **(7). MOSFET Introduction : Automatic Model Selection**
- Automatic Model Selection :
  - HSPICE can Automatically Find the Proper Model for Each Transistor Size by Using Parameters, LMIN,LMAX,WMIN, & WMAX in MOSFET Models

```
.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
```



6. Elements & Device Models

- **(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)

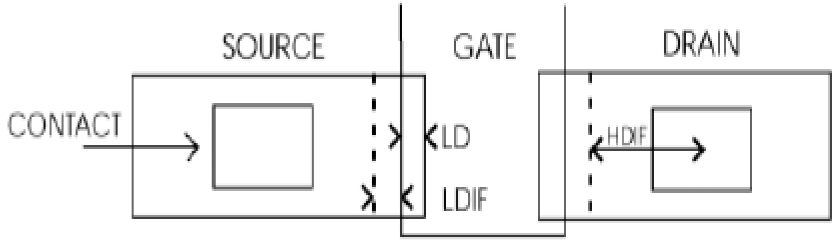


- ACM=0 : PN Bulk Junction of MOSFET are Modeled in the SPICE-style.
- ACM=0 : Not Permit Specifications of HDIF & LDIF.

ADeff = M • AD • WMLT2 • SCALE2

6. Elements & Device Models

- **(8). MOSFET Introduction : MOSFET Diode Model (Cont.)**
- ACM=2 MOSFET Diode: (MOSFET LDD Structure)

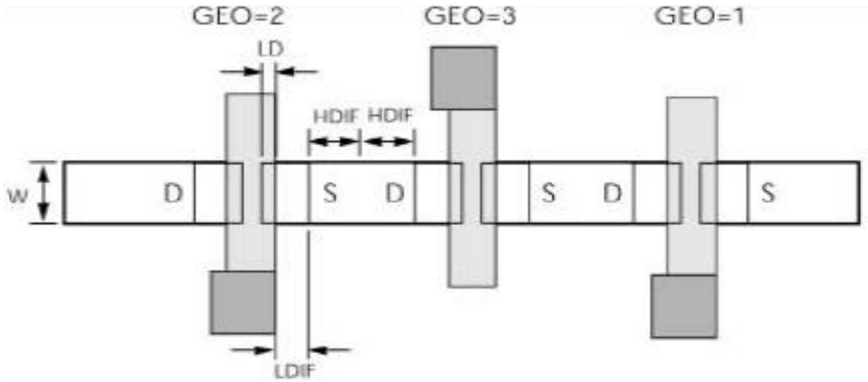


- ACM=2 : HSPICE_Style Diode Model, Combination of ACM=0 & 1.
- ACM=2 : Supports both Lightly & Heavily Doped Diffusions by Settling LD, LDIF, and HDIF Parameters.
- ACM=2 : Effective Areas and Peripheries can be Calculations by
- LDIF & HDIF (i.e. AS, AD, PS, & PD can be Omitted in MOS Element Statement)

ADeff = 2•HDIF•Weff   PDeff = 4•HDIF+2•Weff

6. Elements & Device Models

- **(8). MOSFET Introduction : MOSFET Diode Model (Cont.)**
- ACM=3 MOSFET Diode : (Stacked MOSFET Diode Model)



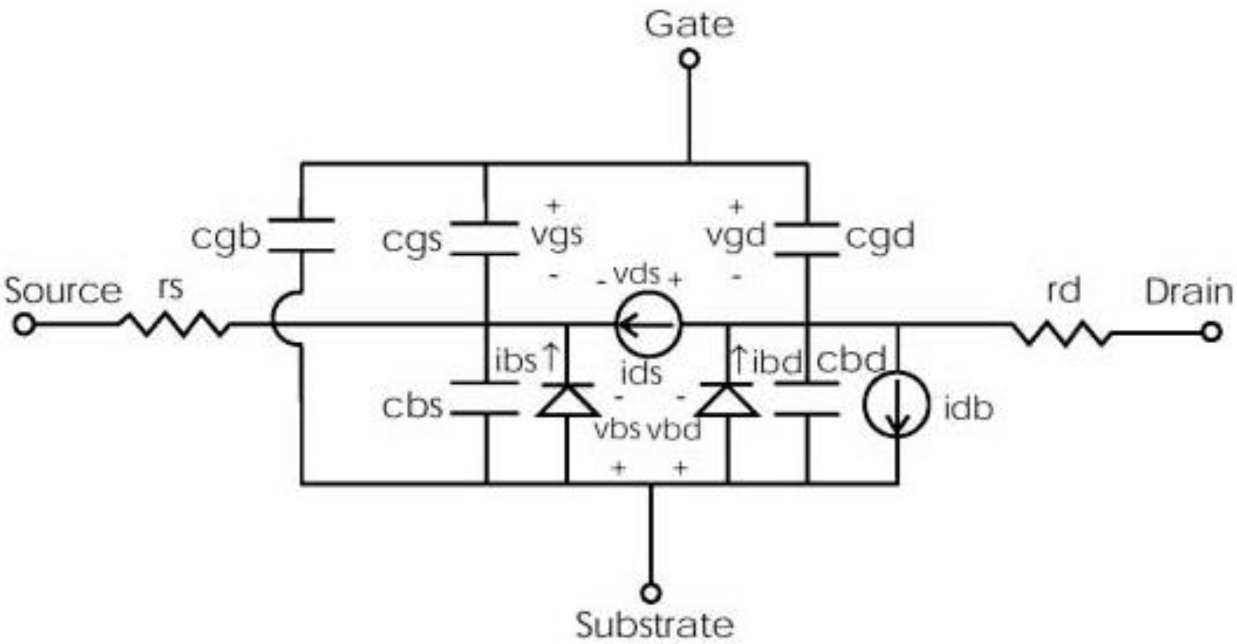
- ACM=3 : Extension of ACM=2 Model that Deals with Stacked Devices.
- ACM=3 : AS, AD, PS, & PD Calculations Depend on the Layout of the Device, which is Determined by the Value of Elememt Parameter GEO.
- ACM=3 : GEO=0 (Default) Indicates Drain & source are not Shared by other Devices

6. Elements & Device Models

- **(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

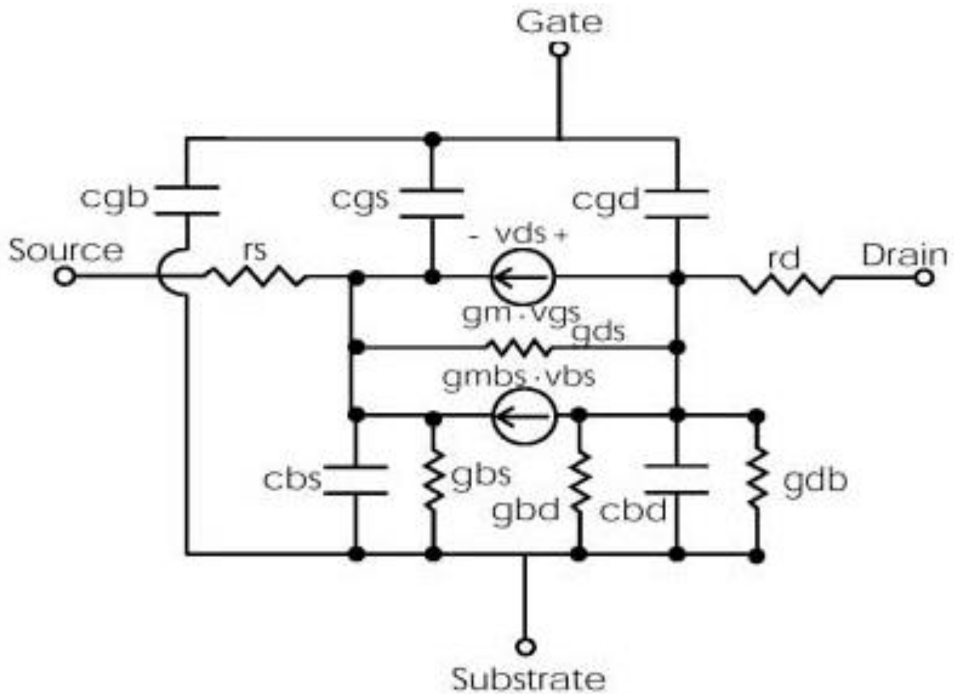
6. Elements & Device Models

- **(10). MOSFET Introduction : Equivalent Circuits**
- MOSFET Equivalent Circuit for Transient Analysis:



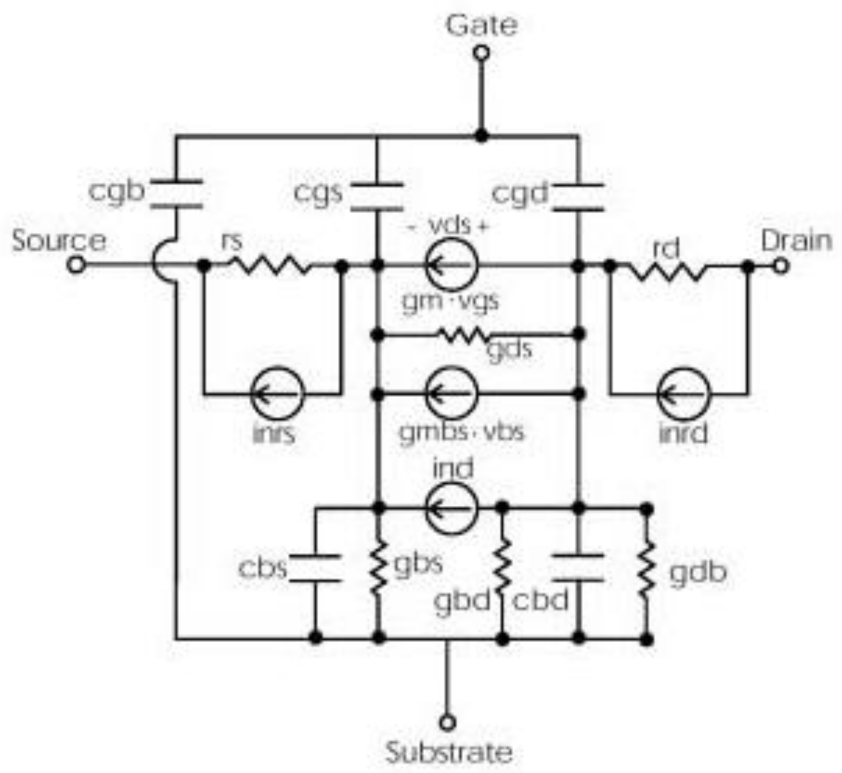
6. Elements & Device Models

- **(10). MOSFET Introduction : Equivalent Circuits (Cont.)**
- MOSFET Equivalent Circuit for AC Analysis:



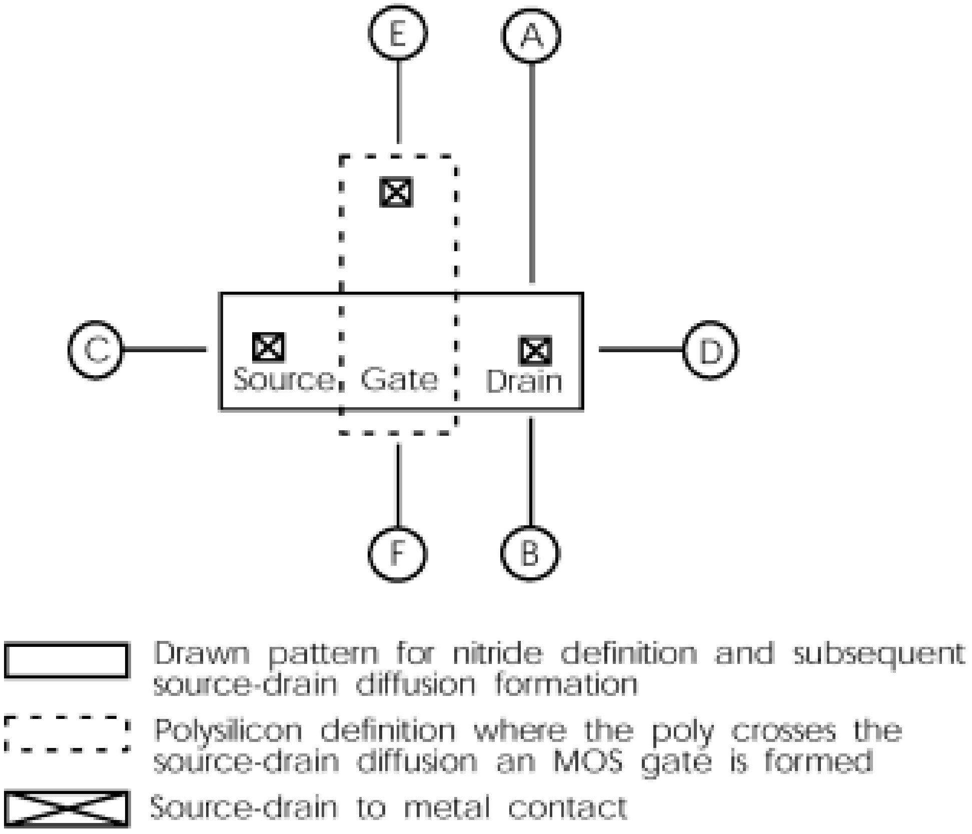
6. Elements & Device Models

- **(10). MOSFET Introduction : Equivalent Circuits (Cont.)**
- MOSFET Equivalent Circuit for AC Noise Analysis:



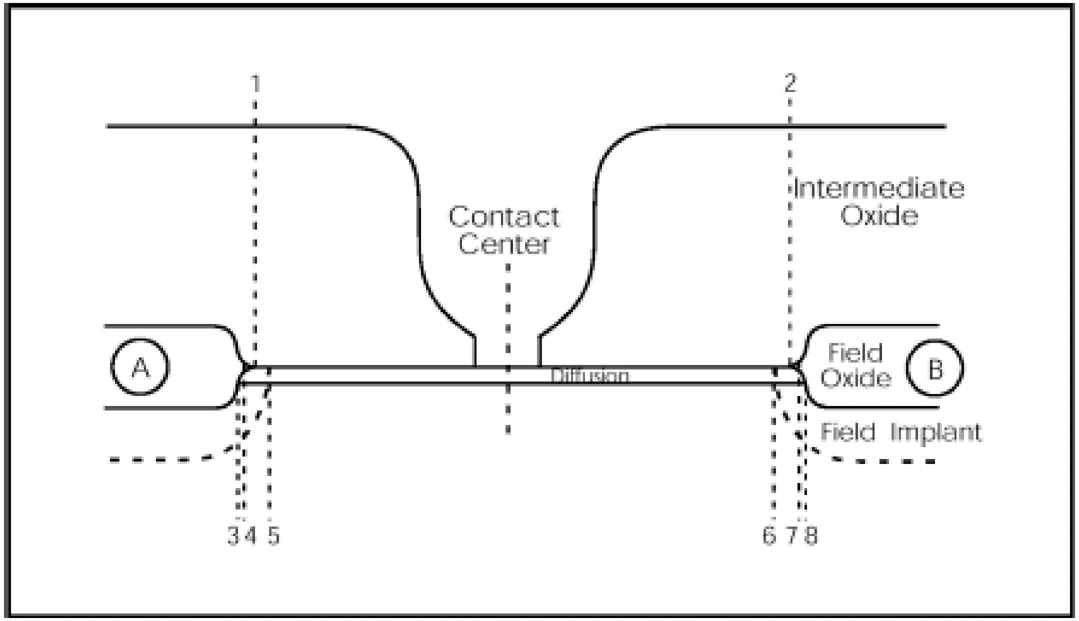
6. Elements & Device Models

- **(11). MOSFET Introduction: Construction of MOSFET**
- Isoplanar Silicon Gate Transistor :



6. Elements & Device Models

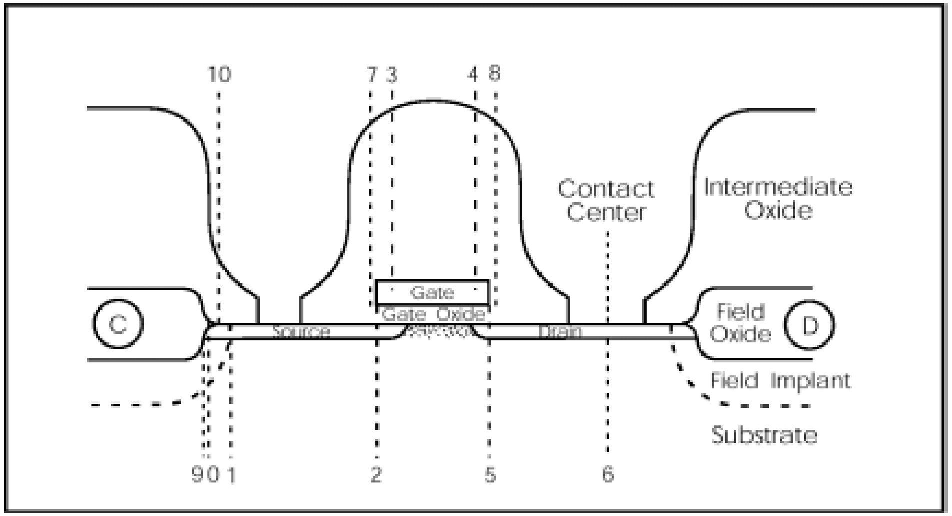
- **(11). MOSFET Introduction: Construction of MOSFET(Cont.)**
- Isoplanar MOSFET Construction : (Cut through A-B)



1 - 2	Diffusion drawn dimension for nitride
4 - 7	Nitride layer width after etch
3 - 1	Periphery of the diode

6. Elements & Device Models

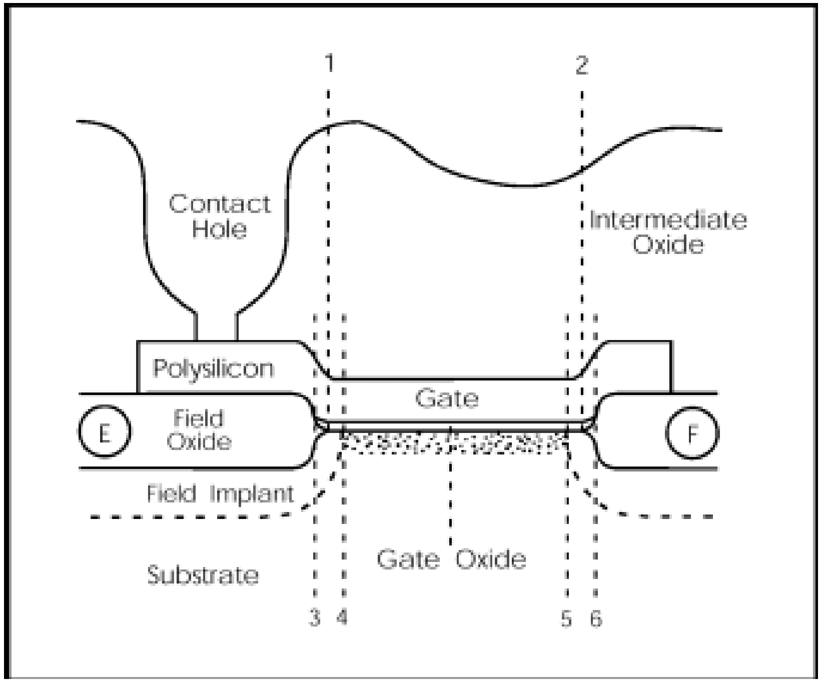
- ***(11). MOSFET Introduction:*** ***Construction of MOSFET(Cont.)***
- Isoplanar MOSFET Construction : (Cut through C-D)



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

6. Elements & Device Models

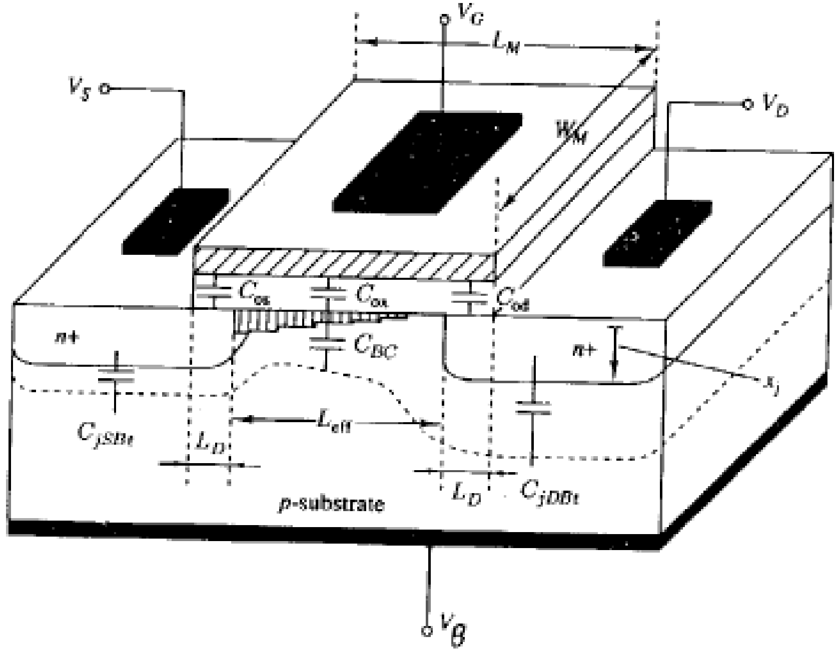
- ***(11). MOSFET Introduction:*** ***Construction of MOSFET(Cont.)***
- Isoplanar MOSFET Construction : (Cut through E-F)



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

6. Elements & Device Models

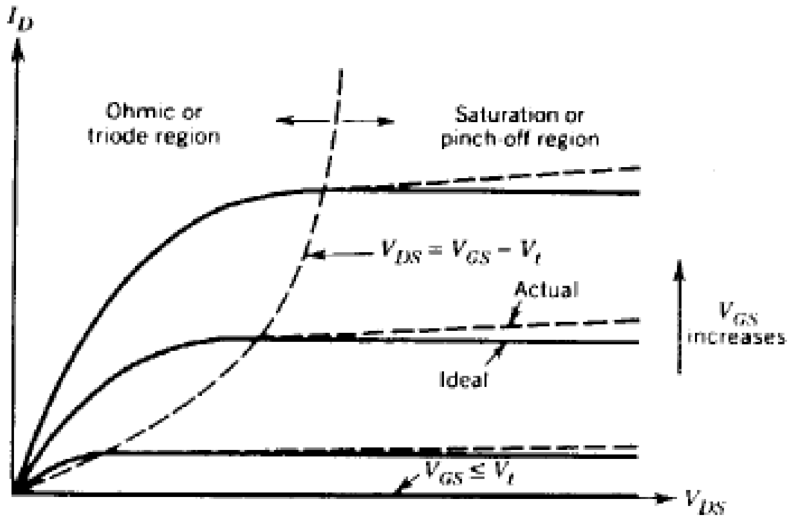
- ***(12). MOSFET Transistor Basics : Structure & Bias***
- MOSFET Structure : A Four Terminal Device (V_G, V_D, V_S, V_B)

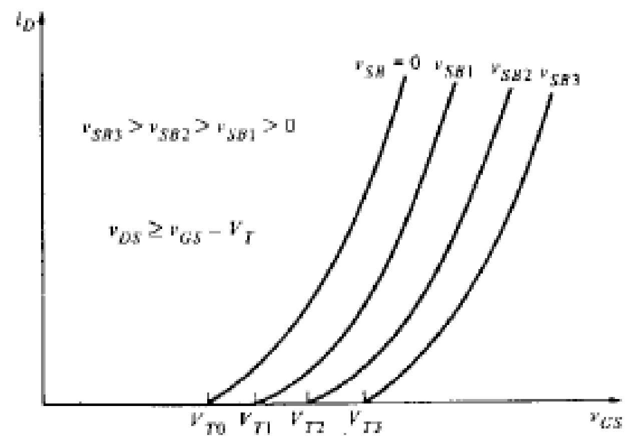


- Basic Parameters : Channel Length (L_M), Channel Width(W_M), Oxide Thickness(t_{ox}), Junction depth(x_j) & Substrate Doping(N_a)

6. Elements & Device Models

- **(13). MOSFET Transistor Basics : Transfer Characteristics**
- Transfer Characteristics of NMOS :





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

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

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

6. Elements & Device Models

- **(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)
  - Substrate Current Induced Body Effects (SCBE)
- Other Physical Effects :
  - Channel Mobility Degradation
  - Carrier Drift Velocity
  - Bulk Charge Effect
  - Parasitic Resistance
  - Subthreshold Current

6. Elements & Device Models

- **(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

6. Elements & Device Models

- **(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 ???

6. Elements & Device Models

- **(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
  - Advanced Version of Level 1 which Includes Additional Physical Effects
  - Applicable to Long Channel Device (~ 10 um)
  - Can Use either Electrical or Process Related Parameters

SPICE : Simulation Program with Integrated Circuit Emphasis

UCB : University of California at Berkeley

6. Elements & Device Models

- **(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
  - Empirical Approach to Small Geometry Effects
  - Emphasis on Mathematical Conditioning of Circuit Simulation

BSIM : Berkeley Short-Channel IGFET Model

6. Elements & Device Models

- **(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

6. Elements & Device Models

- **(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.

6. Elements & Device Models

- **(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

contents

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

7. Optimization

- **(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

7. Optimization

- **(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

7. Optimization

- **(3). Optimization Syntax : General Form**
- Variable Parameters and Components :

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

- Optimizer Model Statement :

.MODEL method_namd OPT <Parameter = val .....>

- Analysis Statement Syntax :

.DC|AC|TRAN ....<DATA=filement > SWEEP OPTIMIZE = OPTxxx

+ Results = meas_name MODEL = method_name

- Measure Statement Syntax :

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

7. Optimization

- **(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 (→ .param Tpw=opt1(0, 0, 15n))

Analysis type and optimization algorithm (→ .tran 0.1n 20n sweep optimize=opt1)

Optimization goal by measure command (→ .measure Tran Tprop ...)

contents

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

8. Control Options & Convergence

- **(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 onConvergence 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)

8. Control Options & Convergence

- **(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)"
  - " Internal 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

8. Control Options & Convergence

- **(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.

8. Control Options & Convergence

- **(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.

8. Control Options & Convergence

- **(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.

8. Control Options & Convergence

- **(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)

8. Control Options & Convergence

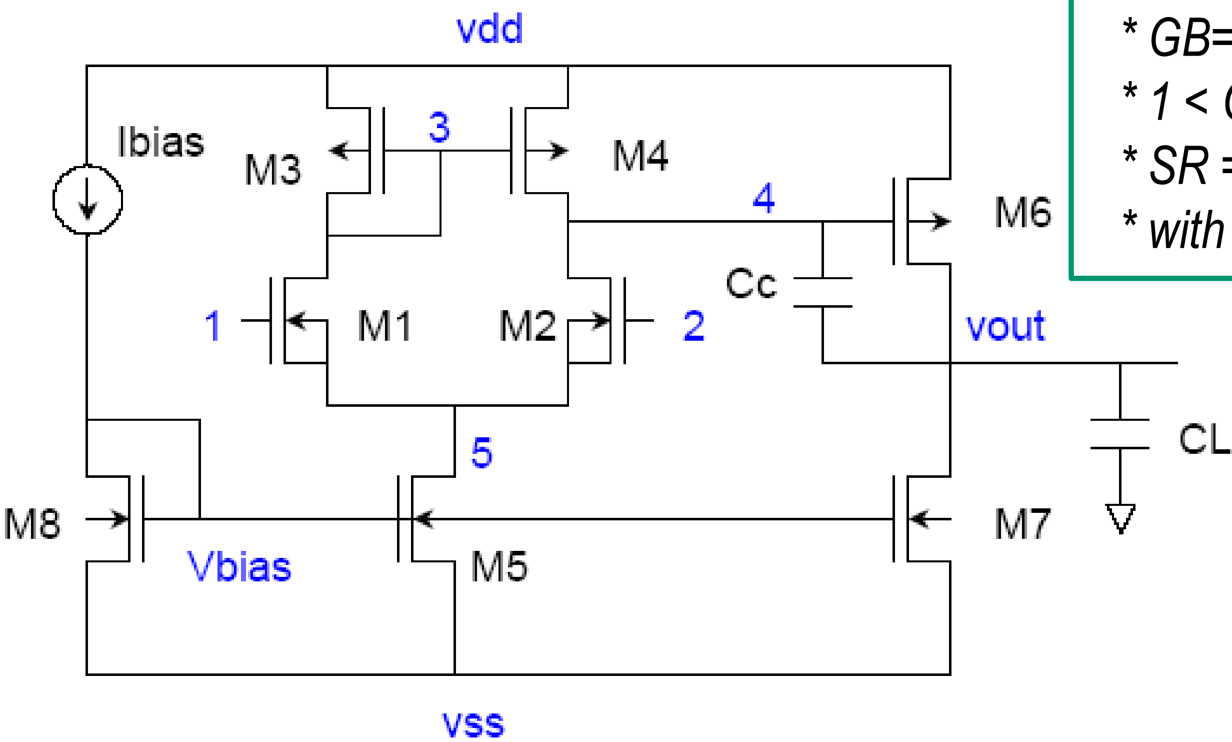
- **(7). Timestep Control Algorithms**
- Types of Dynamic Timestep Control Algorithm :
  - Iteration Count (Simplest):
    - If Iterations Required to Converge > MAX, Decrease the Timestep
    - If Iterations Required to Converge < MIN, Increase the Timestep
  - Local Truncation Error (LTE) :
    - Use 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

- SPICE Overview
- Simulation Input and Controls
- Sources and Stimuli
- Analysis Types
- Simulation Output and Controls
- Elements and Device Models
- Optimization
- Control Options & Convergence
- Applications Demonstration

9. Application Demo

- ***(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*

9. Application Demo

- ***(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

9. Application Demo

- **(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)

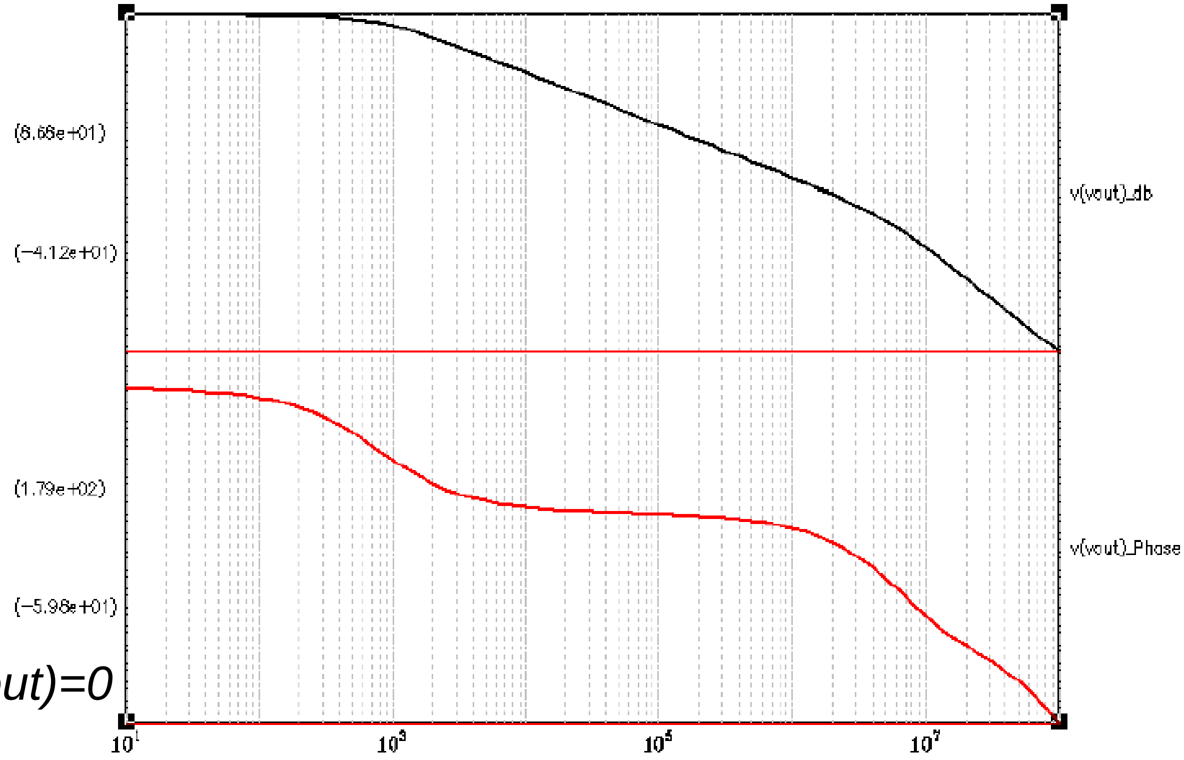
+ vp(vout) vdb(4) vp(4)

.meas ac Unit_gain

+ when vdb(vout)=0

.meas ac phase_mar

+ FIND vp(vout) when vdb(vout)=0



9. Application Demo

- **(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

.tran 5n 2u

.probe tran

.save all