JIJEL FACULTY OF TECHNOLOGY
ELECTRONIC DEPARTMENT

Introduction to Modeling MOSFETS in SPICE

By: Mr. C. ZIGHA

OUTLINE

INTRODUCTION

PSpice Lite 9.2 is one of the OrCAD family of products, from Cadence Design Systems, Inc., offering a complete suite of electronicdesign tools. It is free and includes limited versions of OrCAD Capture, for schematic capture, PSpice for analog circuit simulation and Pspice A/D for mixed analog and digital circuit simulation.
PSpice Lite 9.2 is limited to 64 nodes, 10 transistors, two operational amplifiers and 65 primitive digital devices. See page 35 (xxxv) of the PSpice Users Guide.

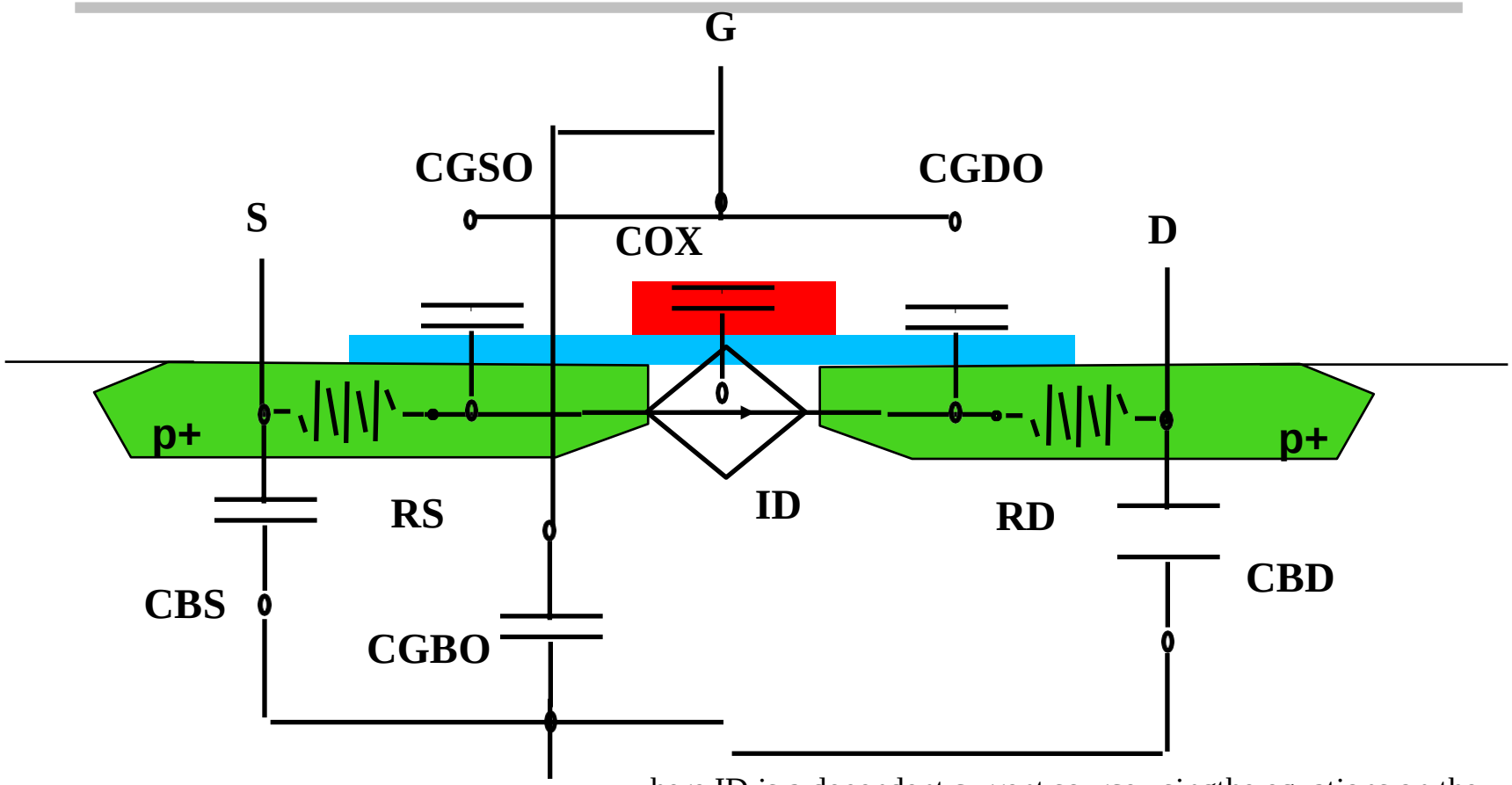
LT SPICE – is a free SPICE simulator with schematic capture from Linear Technology. It is quite similar to PSpice Lite but is not limited in the number of devices or nodes. Linear Technology (LT) is one of the industry leaders in analog and digital integrated circuits. Linear Technology provides a complete set of SPICE models for LT components.

MOSFET DEVICE MODELS

MOSFET Device models used by SPICE (Simulation Program for Integrated Circuit Engineering) simulators can be divided into three classes: First Generation Models (Level 1, Level 2, Level 3 Models), Second Generation Models (BISM, HSPICE Level 28, BSIM2) and Third Generation Models (BSIM3, Level 7, Level 8, Level 49, etc.) The newer generations can do a better job with short channel effects, local stress, transistors operating in the sub-threshold region, gate leakage (tunneling), noise calculations, temperature variations and the equations used are better with respect to convergence during circuit simulation.

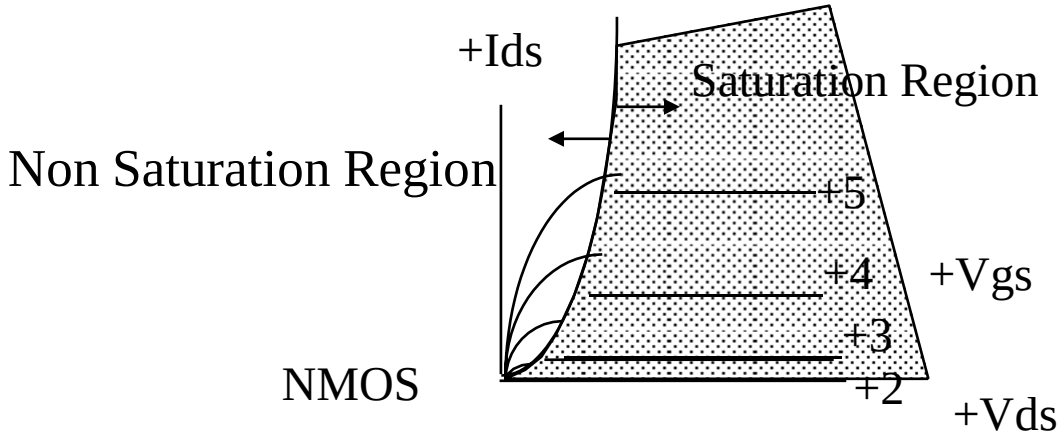
In general first generation models are recommended for MOSFETs with gate lengths of 10um or more. If not specified most SPICE MOSFET Models default to level=1 (Shichman and Hodges)

SPICE LEVEL-1 MOSFET MODEL



where ID is a dependent current source usingthe equations on the next page

SPICE LEVEL 1 - SHICHMAN AND HODGES



$$I_{Dsat} = \frac{\mu W \, Cox' \, (Vg-Vt)^2}{2L}$$

Saturation Region

$$I_D = \frac{\mu W \, Cox' \, (Vg-Vt-V_d/2)V_d}{L}$$

Non Saturation Region

where μ, Cox'and Vt are given in equations on the next pages

SPICE LEVEL-1 EQUATIONS FOR UO, VTO AND COX'

Mobility: (cm2/V-s)

$$\mu = \mu_{min} + \frac{(\mu_{max} - \mu_{min})}{\{1 + (N/N_{ref})^{\alpha}\}}$$

Parameter	Arsenic	Phosphorous	Boron
μ_{min}	52.2	68.5	44.9
μ_{max}	1417	1414	470.5
N_{ref}	9.68X10^16	9.20X10^16	2.23X10^17
α	0.680	0.711	0.719

Threshold Voltage: +/- nmos/pmos

$$\mathbf{VTO = \Phi ms - q\ NSS/Cox' +/ -2[\Phi F] +/-2\ (q\varepsilon o\varepsilon rsi\ NSUB\ [\Phi F])^{0.5}/Cox'}$$

$[\Phi F] = (KT/q)\ \ln (NSUB/ni)$ where ni = 1.45E10 and KT/q = 0.026

Absolute value

PHI = 2 [ΦF]

Gate Capacitance per unit area Cox'

$$\mathbf{Cox' = \varepsilon rox\ \varepsilon o/TOX = 3.9\ \varepsilon o/TOX}$$

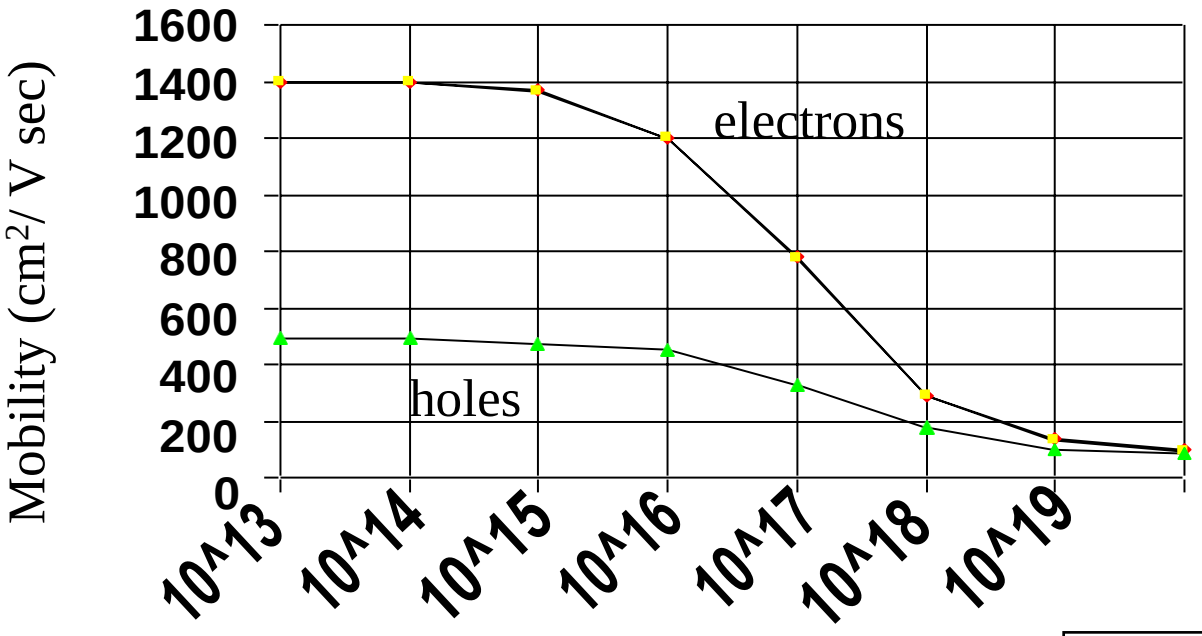
= 3.9

where εr si = 11.7 and εr ox

εo = 8.85E-12 F/m or 8.8eE-14F/cm

q = 1.6E-19

MOBILITY MODEL



Electron and hole mobilities in silicon at 300 K as functions of the total dopant concentration (N). The values plotted are the results of the curve fitting measurements from several sources. The mobility curves can be generated using the equation below with the parameters shown:

$$\mu(N) = \mu_{mi} + \frac{(\mu_{max} - \mu_{min})}{\{1 + (N/N_{ref})^{\alpha}\}}$$

LONG CHANNEL THRESHOLD VOLTAGE, VT

Flat-band Voltage $V_{FB} = \phi_{ms} - \frac{Q_{ss}}{C'_{ox}} - \frac{1}{C'_{ox}} \int_0^{Xox} \frac{X \ \rho(x)\, dx}{X_{ox}}$

p-type substrate (n-channel)	n-type substrate (p-channel)	$Q_{ss} = q\, N_{ss}$
Bulk Potential : $\phi_p = -KT/q \ \ln (N_A / n_i)$	$\phi_n = +KT/q \ \ln (N_D / n_i)$	
Work Function Difference: $\phi_{M\,S} = \phi_M - (X + Eg/2q + [\phi_p])$	$\phi_{M\,S} = \phi_M - (X + Eg/2q - [\phi_n])$	
Maximum Depletion Width: (W_{dmax})	$\sqrt{\frac{4\, \varepsilon s [\phi_p]}{qNa}}$	$\sqrt{\frac{4\, \varepsilon s [\phi_n]}{qNd}}$

Threshold Voltage: $VT = V_{FB} + 2[\phi_p] + \frac{1}{C'_{ox}} \sqrt{2\, \varepsilon s\, q\, Na\, (2[\phi_p] + Vsb)}$

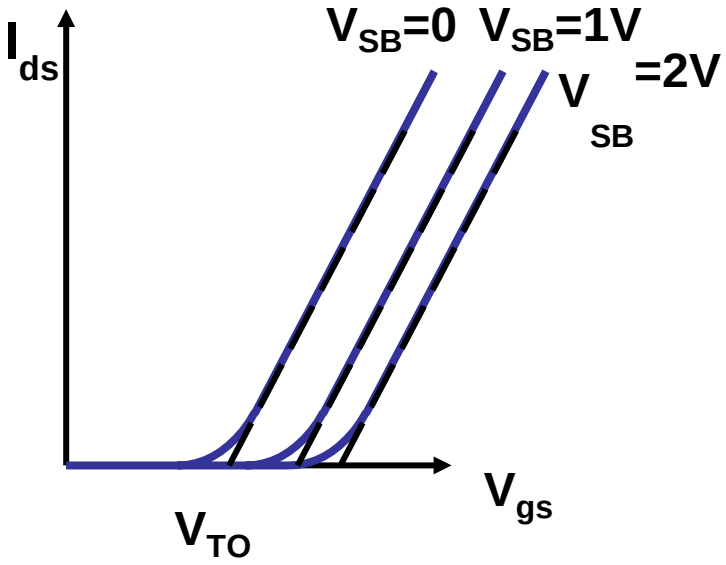
p-type substrate

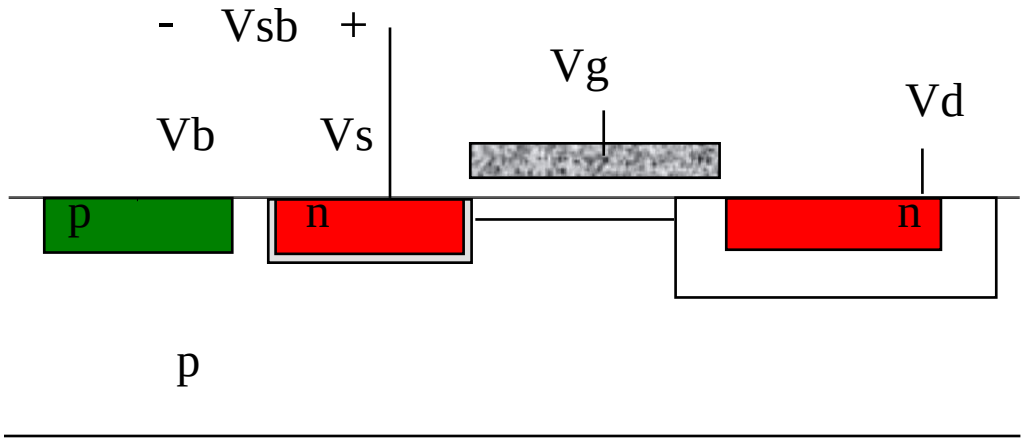
Threshold Voltage: $VT = V_{FB} - 2[\phi_n] - \frac{1}{C'_{ox}} \sqrt{2\, \varepsilon s\, q\, Nd\, (2[\phi_n] + Vbs)}$

n-type substrate

BACK-BIASING EFFECTS – GAMMA

Body Effect coefficient GAMMA or γ :





$$\gamma = \frac{1}{C'_{ox}}\sqrt{2q\varepsilon_{Si}N_{sub}}$$

$$V_T = \Phi_{MS} - \frac{Qss}{C'_{ox}} + 2\phi_F + \gamma$$

εo = 8.8eE-14F/cm q = 1.6E-19

VT ADJUST IMPLANT

The threshold voltage can be adjusted with an ion implant. If total implant dose is shallow (within W_{dmax}) then the change in Vt is:

$$+/- \Delta Vt = q\ Dose^*/Cox'$$

where Dose* is the dose that is added to the Si
Cox' is gate oxide capacitance/cm^2

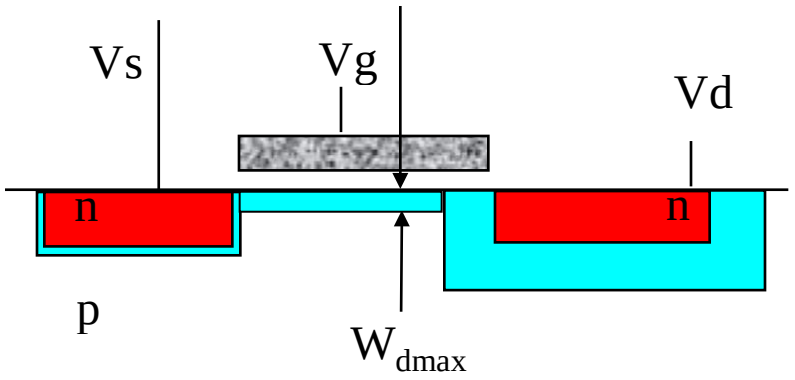
Boron gives + shift

Phosphorous gives - shift

$$Cox' = \varepsilon o \varepsilon r\ /\ Xox$$

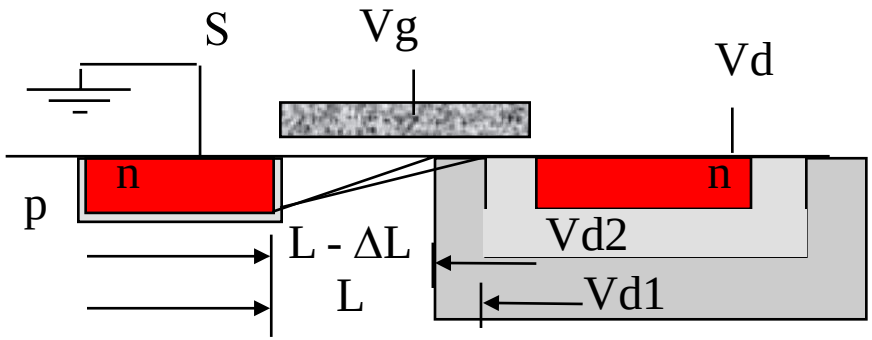
Maximum Depletion Width:

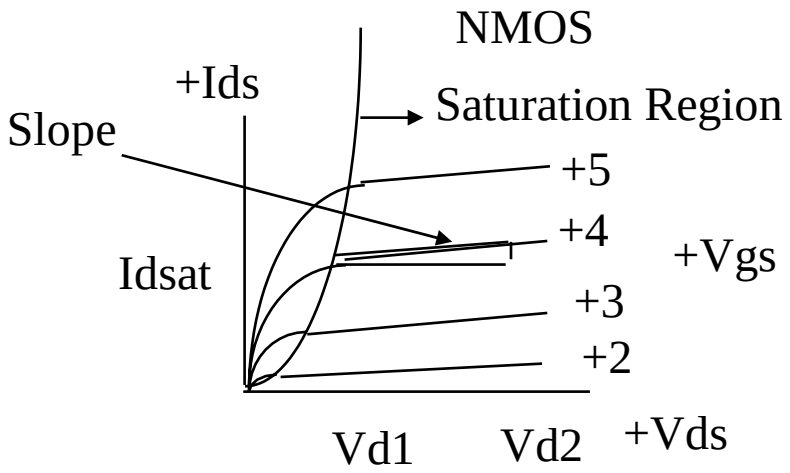
$$Wdmax = \sqrt{\frac{4\ \varepsilon s[\phi_p]}{qN}}$$



CHANNEL LENGTH MODULATION - LAMBDA

Channel Length Modulation
Parameter λ
λ = Slope/ Idsat





$$I_{Dsat} = \frac{\mu W Cox'}{2L}(Vg-Vt)^2 (1+ \lambda Vds)$$

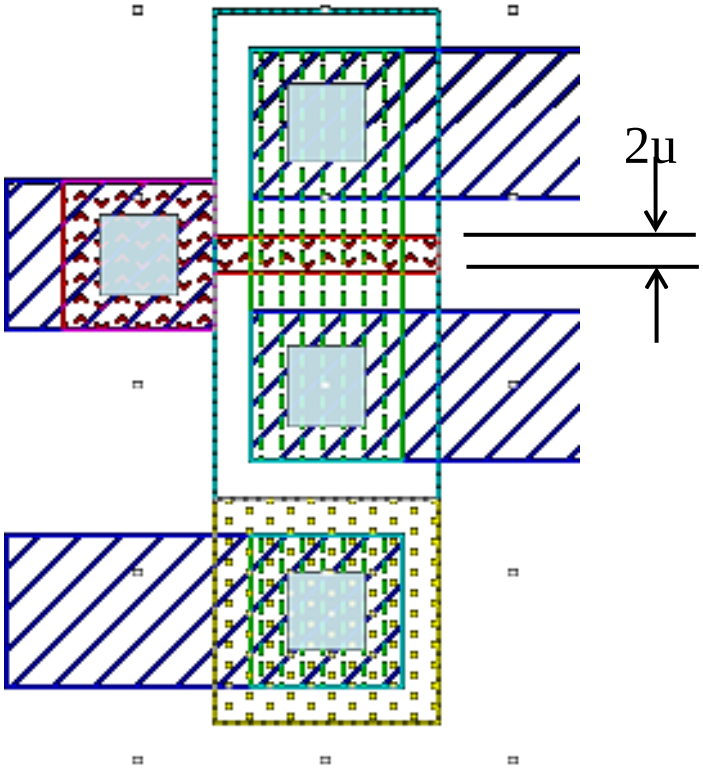
Saturation Region

$$I_D = \frac{\mu W Cox'}{L}(Vg-Vt-V_d/2)V_d (1+ \lambda Vds)$$

Non Saturation Region

NMOS Transistor
DC Model, λ is the channel length modulation parameter and is different for each channel length, L. Typical value might be 0.02

TRANSISTOR PROPERTIES OR ATTRIBUTES



NMOS 2/8

L= 2u
W = 8u
Ad = 8u x10u = 80p
As = Ad = 80p
Pd = 8u+10u+8u+10u = 36u
Ps = Pd = 36u
Nrs = 1
Nrd = 1

LTSPICE MOSFET ATTRIBUTES

MOSFETS are four terminal devices (Drain, Gate, Source and Substrate). L and W are channel length and width in meters, Ad and As are area of drain and source in square meters. If not specified default values are used. (see next page) Perimeter of Drain and source(PD and PS) in meters is used to calculate drain and source side wall capacitance. If PD and PS are not given the default is zero. NRD and NRS are multiplied by the drain and source sheet resistance to give series resistance RD and RS. The default value for NRD and NRS is one.

MOSFET DEFINITION - LTSPICE

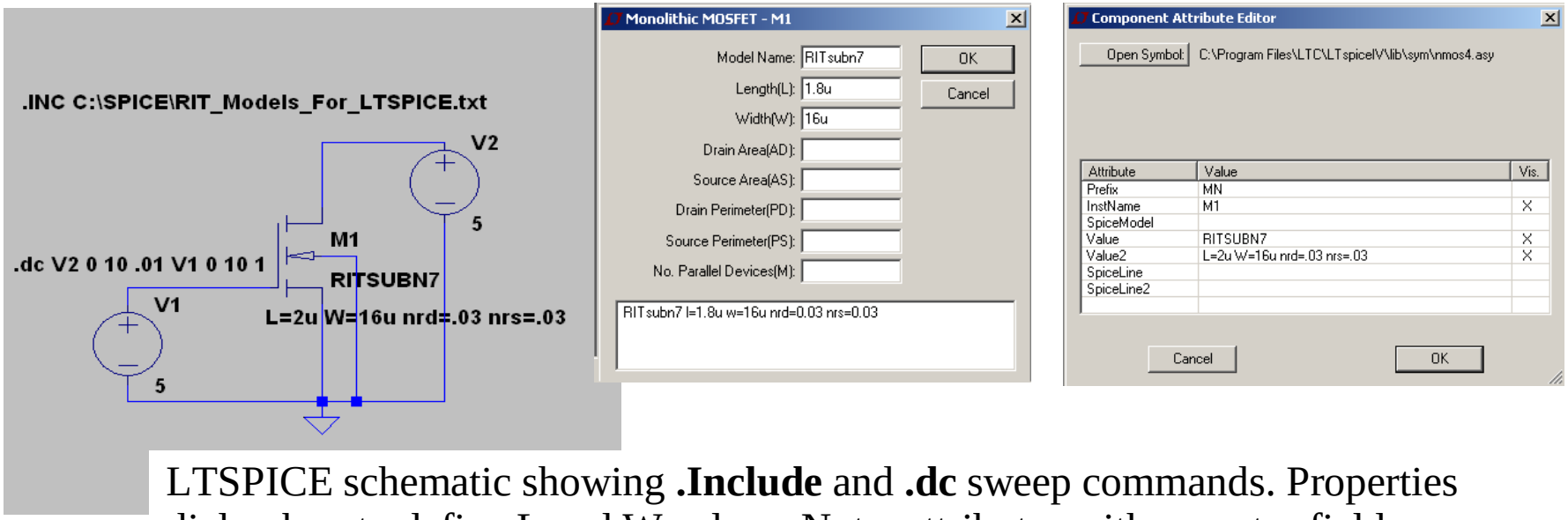
For example:

* SPICE Input File
* MOSFET names start with M…. M2 is the name for the MOSFET below and its drain, gate, source
* and substrate is connected to nodes 3,2,0,0 respectively. The model name is RITSUBN7.
* The parameters/attributes is everything after that.

M2 3 2 0 0 RITSUBN7 L=2U W=16U ad=96e-12 as=96e-12 pd=44e-6 ps=44e-6 nrd=1.0 nrs=1.0

*

*



LTSPICE schematic showing **.Include** and **.dc** sweep commands. Properties dialog box to define L and W values. Note: attributes with no entry field **nrs** and **nrd** are typed in bottom box. Attribute Editor (CTRL R-click on the transistor) allows attributes with Vis.=X to be displayed on the schematic.

MOSFET DEFINITION - PSPICE

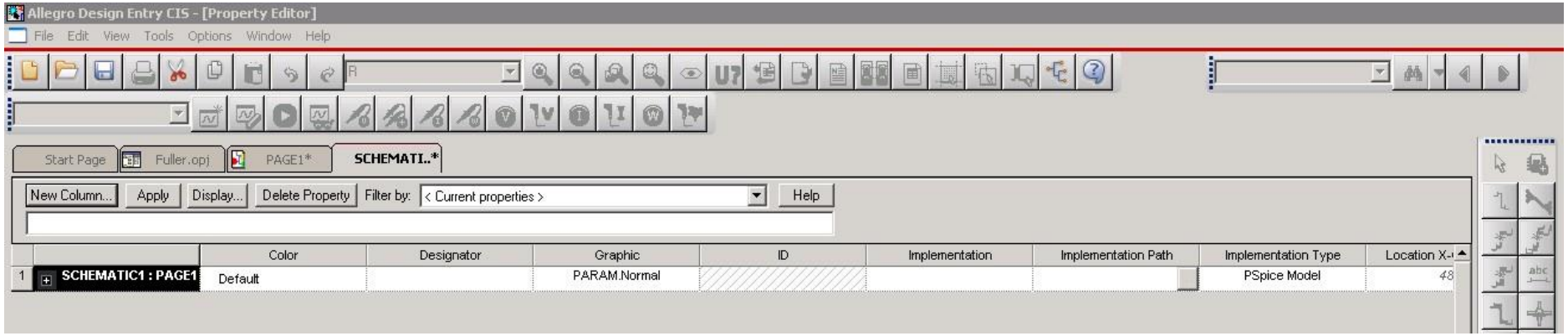
For example:

* SPICE Input File
* MOSFET names start with M…. M2 is the name for the MOSFET below and its drain, gate, source
* and substrate is connected to nodes 3,2,0,0 respectively. The model name is RITSUBN7.
* The parameters/attributes is everything after that.

M2 3 2 0 0 RITSUBN7 L=2U W=16U ad=96e-12 as=96e-12 pd=44e-6 ps=44e-6 nrd=1.0 nrs=1.0

*

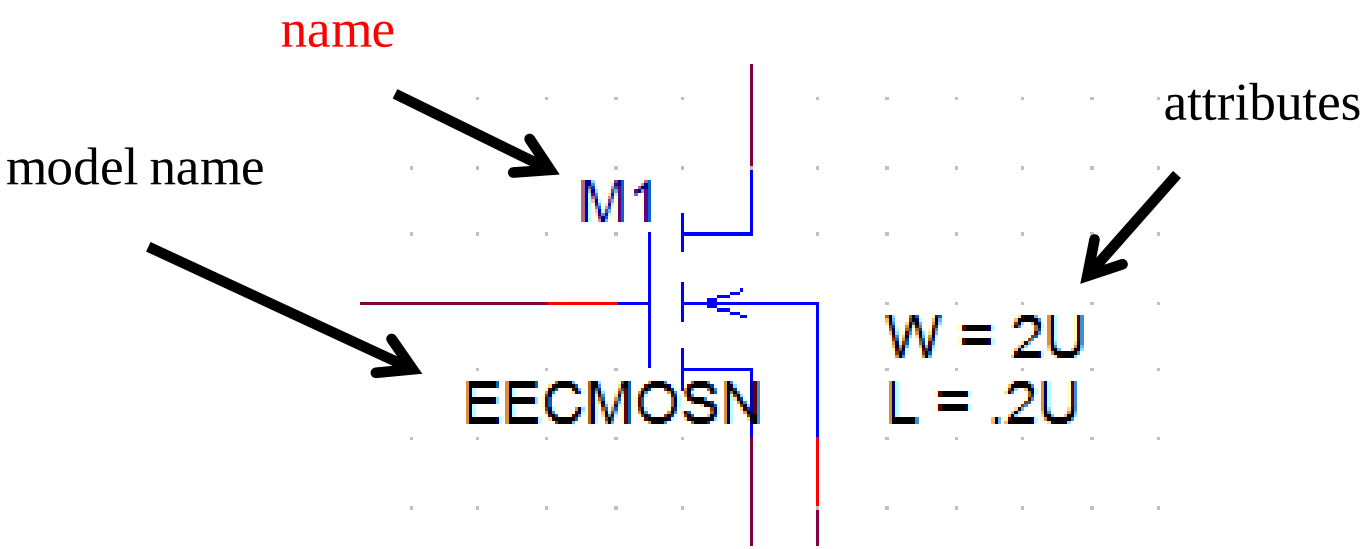
*



In PSPICE the Attribute Editor (CTRL R-click on the transistor) allows attributes values to be set, new attribute columns to be created, and attributes can be selected to be displayed on the schematic..

MOSFET DEFINITION - PSPICE

In SPICE a transistor is defined by its **name** and associated **properties or attributes** and its **model**. MOSFET names start with M, attributes (L, W, Ad, As, etc.) are specified by the user and shown in the input filenet list. Some attributes can be displayed on the schematic. The model is specified in a file in a given location or is defined in a library.



SIMPLE AND ADVANCED SPICE MODELS

```
* From Electronics II EEEE482 FOR ~100nm Technology
.model EECMOSN     NMOS (LEVEL=8
+VERSION=3.1 CAPMOD=2 MOBMOD=1
+TOX=5E-9 XJ=1.84E-7 NCH=1E17 NSUB=5E16 XT=5E-8
+VTH0=0.4 U0= 200 WINT=1E-8 LINT=1E-8
+NGATE=5E20 RSH=1000 JS=3.23E-8 JSW=3.23E-8 CJ=6.8E-4 MJ=0.5 PB=0.95
+CJSW=1.26E-10 MJSW=0.5 PBSW=0.95 PCLM=5
+CGSO=3.4E-10 CGDO=3.4E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 FOR ~100nm Technology
.model EECMOSP      PMOS (LEVEL=8
+TOX=5E-9 XJ=0.05E-6 NCH=1E17 NSUB=5E16 XT=5E-8
+VTH0=-0.4 U0= 100 WINT=1E-8 LINT=1E-8
+NGATE=5E20 RSH=1000 JS=3.51E-8 JSW=3.51E-8 CJ=5.28E-4 MJ=0.5 PB=0.94
+CJSW=1.19E-10 MJSW=0.5 PBSW=0.94 PCLM=5
+CGSO=4.5E-10 CGDO=4.5E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 SIMPLE MODEL
.model EENMOS       NMOS (VTO=0.4 KP=432E-6 GAMMA=0.2 PHI=.88)
*
* From Electronics II EEEE482 SIMPLE MODEL
.model EEPMOS       PMOS (VTO=-0.4 KP=122E-6 GAMMA=0.2 PHI=.88)
```

CHANGING THE MOSFET SPICE MODEL IN PSPICE

FULLERMOSFET.lib:Mbreakn - AMS Model Editor - [Model Text]

File Edit View Model Plot Tools Window Help

cadence

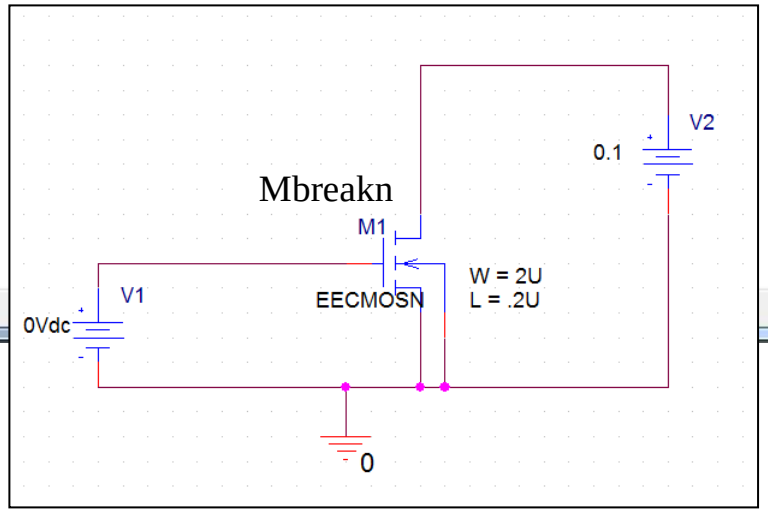
Models List

Model Name	Type
Mbreakn	MOS

```
.model Mbreakn NMOS vto=0.4 kp=432u gamma=0.2 phi=0.88 l=0.1u
```

Right click and select EDIT SPICE MODEL to type in underlined parameters

L and W shown on the schematic override default values



View output file to see this listing of spice model parameters including default values for L and W

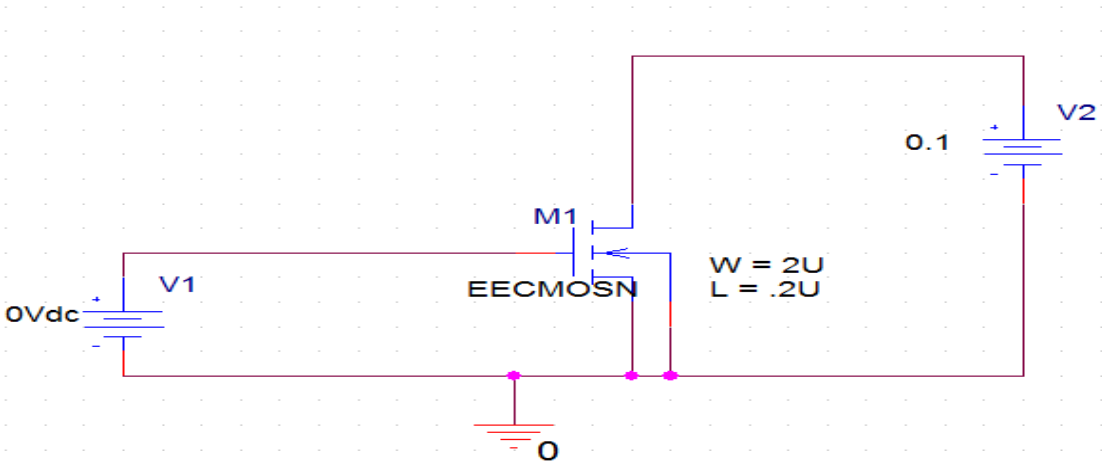
```
                NMOS
      LEVEL      1
          L   100.000000E-09
          W   100.000000E-06
        VTO       .4
         KP   432.000000E-06
      GAMMA       .2
        PHI       .88
     LAMBDA     0
         IS    10.000000E-15
         JS     0
         PB       .8
       PBSW       .8
         CJ     0
       CJSW     0
       CGSO     0
       CGDO     0
       CGBO     0
        TOX     0
         XJ     0
      UCRIT    10.000000E+03
     DIOMOD     1
        VFB     0
       LETA     0
       WETA     0
         U0     0
       TEMP     0
        VDD     5
      XPART     0
```

COMPARISON OF MOSFET CHARACTERISTICS

The circuit shown can be used to see the transistor family of Ids-Vds curves, Ids-Vgs plot and Ids-Vgs (Ids on log scale) Subthreshold plot. We can investigate the effect of changing attributes, SPICE model and model parameters.

V1 is steped to get family of curves or isswept to get Ids-Vgs and Sub-Vt plots

V2 is swept to get family of curves or isheld constant to get Ids-Vgs plots



PSPICE MOSFET MODEL PARAMETERS

95 mosfet model parameters used by cadence PSPICE for Level 8 BSIM

Start Page | FullerMOS..* | PAGE1*

```
280:                EECMOSP
281:                PMOS
282:         LEVEL    8
283:             L  100.000000E-06
284:             W  100.000000E-06
285:           VTO    -.4
286:            KP  414.377300E-06
287:         GAMMA    0
288:           PHI
289:        LAMBDA    0
290:           RSH    1.000000E+03
291:            IS    1.000000E-15
292:            JS   35.100000E-09
293:          JSSW   35.100000E-09
294:            PB     .94
295:          PBSW     .94
296:            CJ  528.000000E-06
297:          CJSW  119.000000E-12
298:          MJSW     .5
299:          CGSO  450.000000E-12
300:          CGDO  450.000000E-12
301:          CGBO  575.000000E-12
302:          NSUB   50.000000E+15
303:           TOX    5.000000E-09
304:            XJ   50.000000E-09
305:         UCRIT   10.000000E+03
306:        DIOMOD    1
307:           VFB   -1
308:          LETA    0
309:          WETA    0
310:            U0  100
311:          TEMP
312:           VDD
313:         XPART    0
314:          VTH0    -.4
315:            UA    1.000000E-09
316:            UB  100.000000E-21
317:            UC  -46.500000E-12
318:          VSAT   80.000000E+03
319:          RDSW  200
320:          VOFF    -.08
321:          PCLM    5
322:            A0    1
323:            A1    0
324:            A2    1
325:         NPEAK  100.000000E+15
326:            XT   50.000000E-09
327:           LDD    0
328:          LITL   27.386130E-09
329:           UA1    1.000000E-09
330:           UB1   -1.000000E-18
331:           UC1     .025
332:          DSUB     .56
333:         NGATE  500.000000E+18
334:        MOBMOD    0
335:          PRWG    1
336:          LINT   10.000000E-09
337:          WINT   10.000000E-09
338:           DLC   10.000000E-09
339:           DWC   10.000000E-09
340:            CF  107.725800E-12
341:          NOIA    9.900001E+18
342:          NOIB    2.400000E+03
343:          NOIC    1.400000E-12
344:       VERSION    4.1
345:          PBSWG     .94
346:          MJSWG     .5
347:          CJSWG  119.000000E-12
348:          JTSCD   35.100000E-09
349:         JSTSCD   35.100000E-09
350:           TOXM    3.000000E-09
351:            LLC    0
352:            LWC    0
353:           LWLC    0
354:            WLC    0
355:            WWC    0
356:           WWLC    0
357: BSIM4oxideTrapDensityC    8.750000E+09
358:           toxp    3.000000E-09
359:             eu    1
360:           aigc     .31
361:           bigc     .024
362:           cigc     .03
363:          aigsd     .31
364:          bigsd     .024
365:          cigsd     .03
366:          dlcig   10.000000E-09
367:            dwj   10.000000E-09
368:         CJSWGD  500.000000E-12
369:         PBSWGD    1
370:         CJSWGS  500.000000E-12
368:         CJSWGD  500.000000E-12
369:         PBSWGD    1
370:         CJSWGS  500.000000E-12
371:         PBSWGS    1
372:           coxe     .01151
373:           coxp     .01151
374: BSIM4factor1   94.868330E-06
```

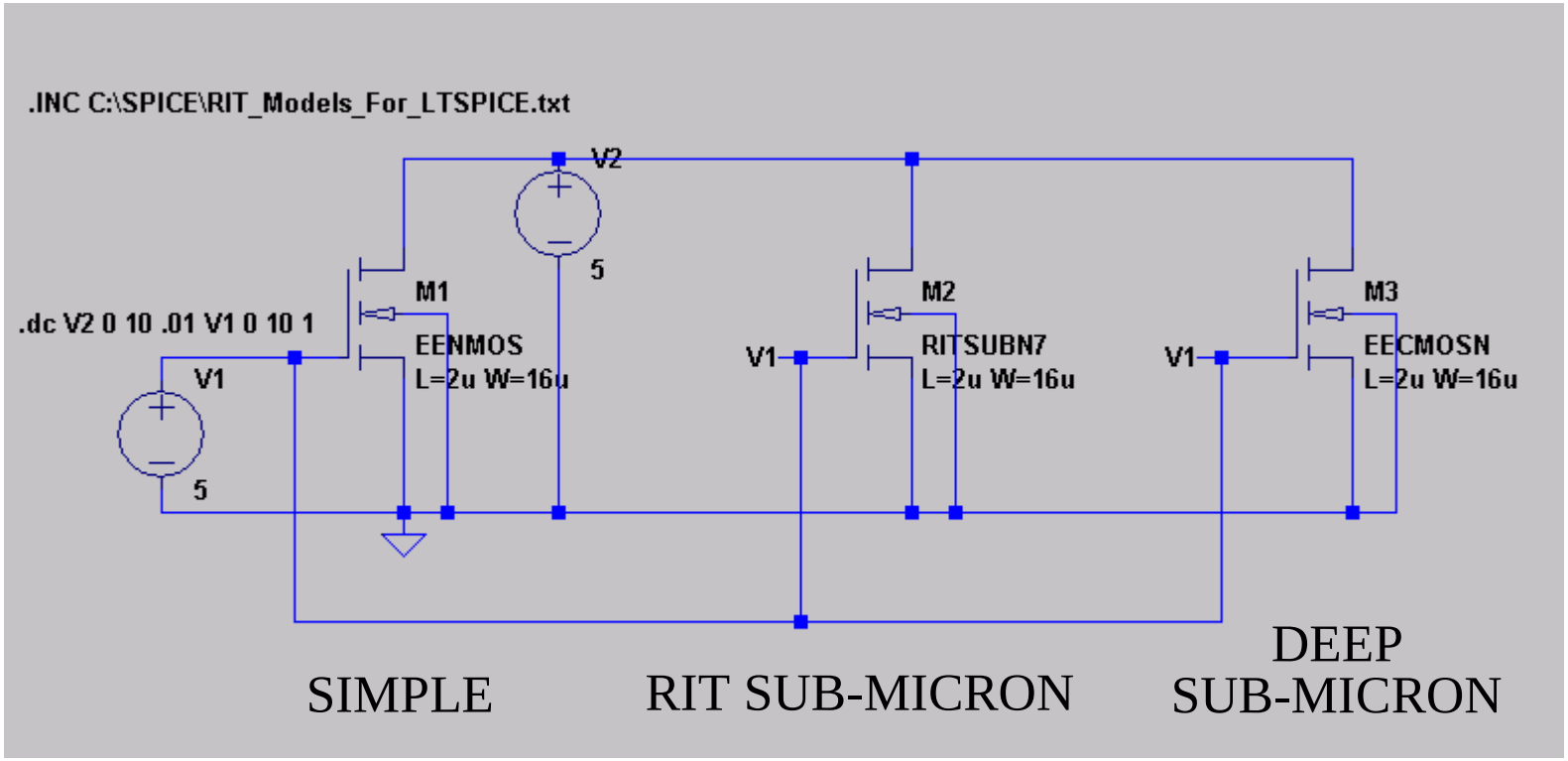
View Output File

31 mosfet model parameters used by cadence PSPICE for Level 1 Shichman and Hodges

FullerMOS..* | PAGE1* | ID-VGS-SW... | SCHEMATI..* | ID-VGS-S...

	EENMOS NMOS	EEPMOS PMOS
LEVEL	1	1
L	100.000000E-06	100.000000E-06
W	100.000000E-06	100.000000E-06
VTO	.4	-.4
KP	432.000000E-06	122.000000E-06
GAMMA	.2	.2
PHI	.88	.88
LAMBDA	0	0
RSH		
IS	10.000000E-15	10.000000E-15
JS	0	0
JSSW		
PB	.8	.8
PBSW	.8	.8
CJ	0	0
CJSW	0	0
MJSW		
CGSO	0	0
CGDO	0	0
CGBO	0	0
NSUB		
TOX	0	0
XJ	0	0
UCRIT	10.000000E+03	10.000000E+03
DIOMOD	1	1
VFB	0	0
LETA	0	0
WETA	0	0
U0	0	0
TEMP	0	0
VDD	5	5
XPART	0	0

LTSPICE CIRCUIT SCHEMATIC

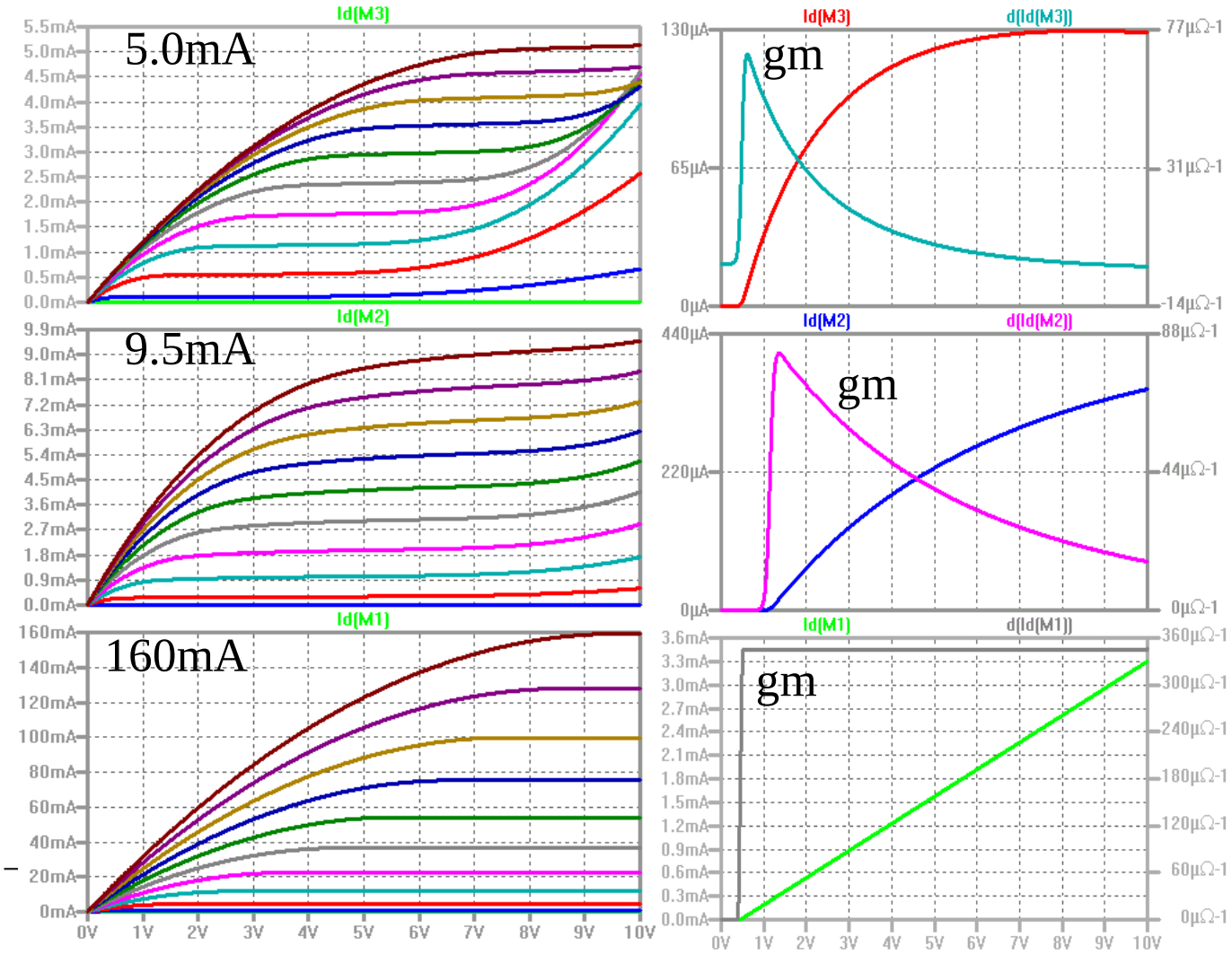


Three transistor all the same L=2u and W=16u but with different SPICE models. (SIMPLE, RIT SUB-MICRON and 100nm DEEP SUB-MICRON

THREE DIFFERENT NMOS SPICE MODELS

From Sub-Micron CMOS Manufacturing Classes in MicroE
.MODEL RITSUBN7 NMOS (LEVEL=7
+VERSION=3.1 CAPMOD=2 MOBMOD=1
+TOX=1.5E-8 XJ=1.84E-7 NCH=1.45E17 NSUB=5.33E16 XT=8.66E-8
+VTH0=1.0 U0= 600 WINT=2.0E-7 LINT=1E-7
+NGATE=5E20 RSH=1082 JS=3.23E-8 JSW=3.23E-8 CJ=6.8E-4 MJ=0.5 PB=0.95
+CJSW=1.26E-10 MJSW=0.5 PBSW=0.95 PCLM=5
+CGSO=3.4E-10 CGDO=3.4E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 FOR ~100nm Technology Deep Sub-Micron
.model EECMOSN NMOS (LEVEL=8
+VERSION=3.1 CAPMOD=2 MOBMOD=1
+TOX=5E-9 XJ=1.84E-7 NCH=1E17 NSUB=5E16 XT=5E-8
+VTH0=0.4 U0= 200 WINT=1E-8 LINT=1E-8
+NGATE=5E20 RSH=1000 JS=3.23E-8 JSW=3.23E-8 CJ=6.8E-4 MJ=0.5 PB=0.95
+CJSW=1.26E-10 MJSW=0.5 PBSW=0.95 PCLM=5
+CGSO=3.4E-10 CGDO=3.4E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 SIMPLE MODEL
.model EENMOSNMOS (VTO=0.4 KP=432E-6 GAMMA=0.2 PHI=.88)

LTSPICE OUTPUT FOR ID-VDS AND ID-VG



Model is EECMOSN
L=2u W=16u

Model not good.
Current low and only good out to 3 volts.

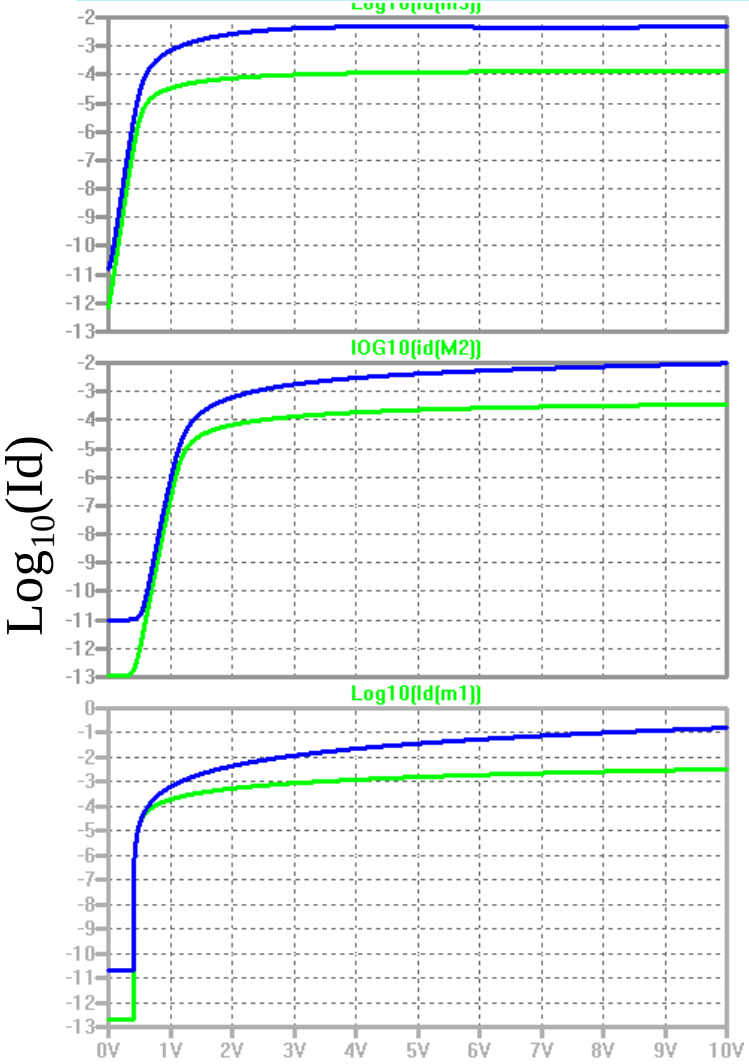
Model is RITSUBN7
L=2u W=16u

Model good for RIT Sub-Micron MOSFETs

Model is EENMOS
L=2u W=16u

Model not good current too large

LTSPICE OUTPUT FOR SUBTHRESHOLD ID-VGS



Model is EECMOSN
L=2u W=16u

Model not good MOSFET does not turn off, Vt too low

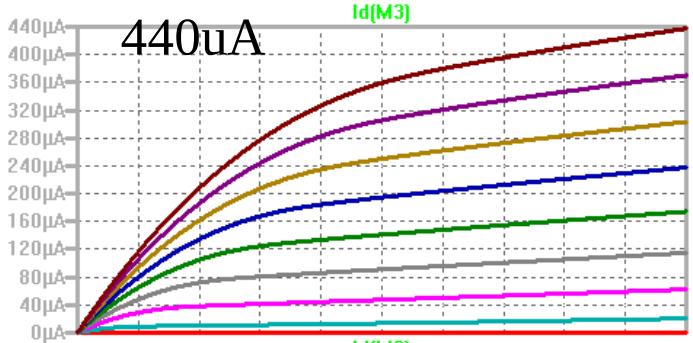
Model is RITSUBN7
L=2u W=16u

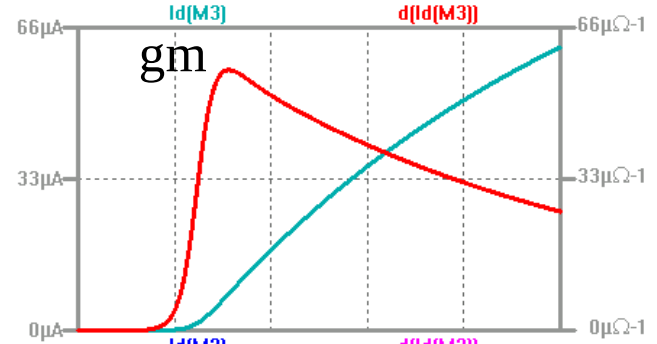
Model good

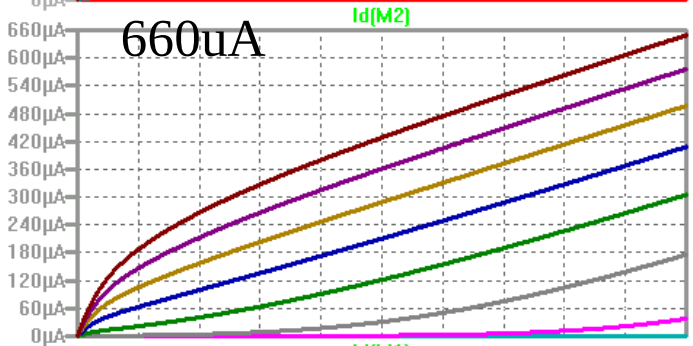
Model is EENMOS
L=2u W=16u

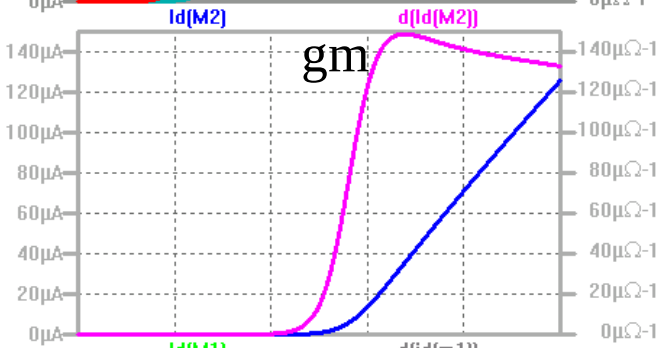
Model incorrect in subthreshold region.
Subthreshold slope not possible.

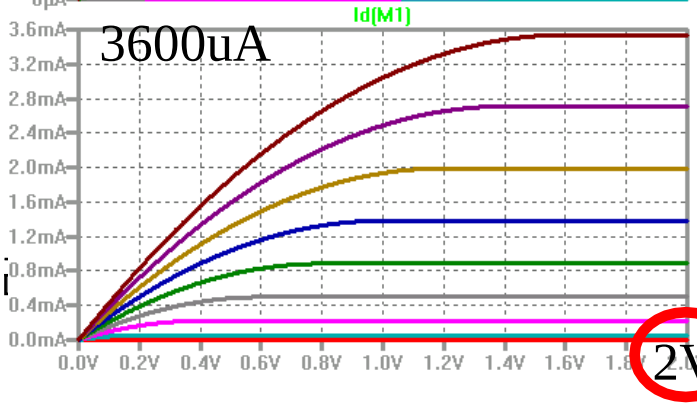
DEEP SUB-MICRON TRANSISTOR MODELS

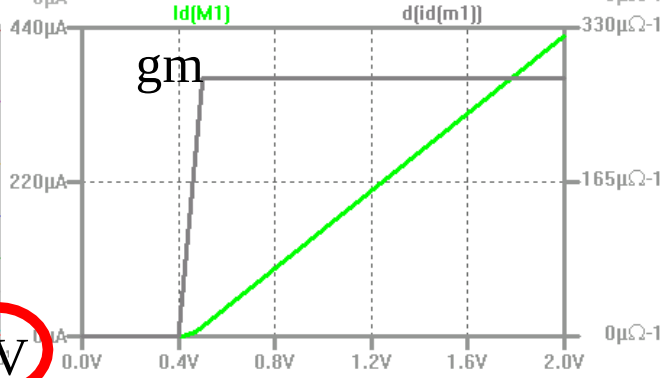












Model is EECMOSN
L=0.25u W=1.6u

Model good for Deep Sub-Micron MOSFETs

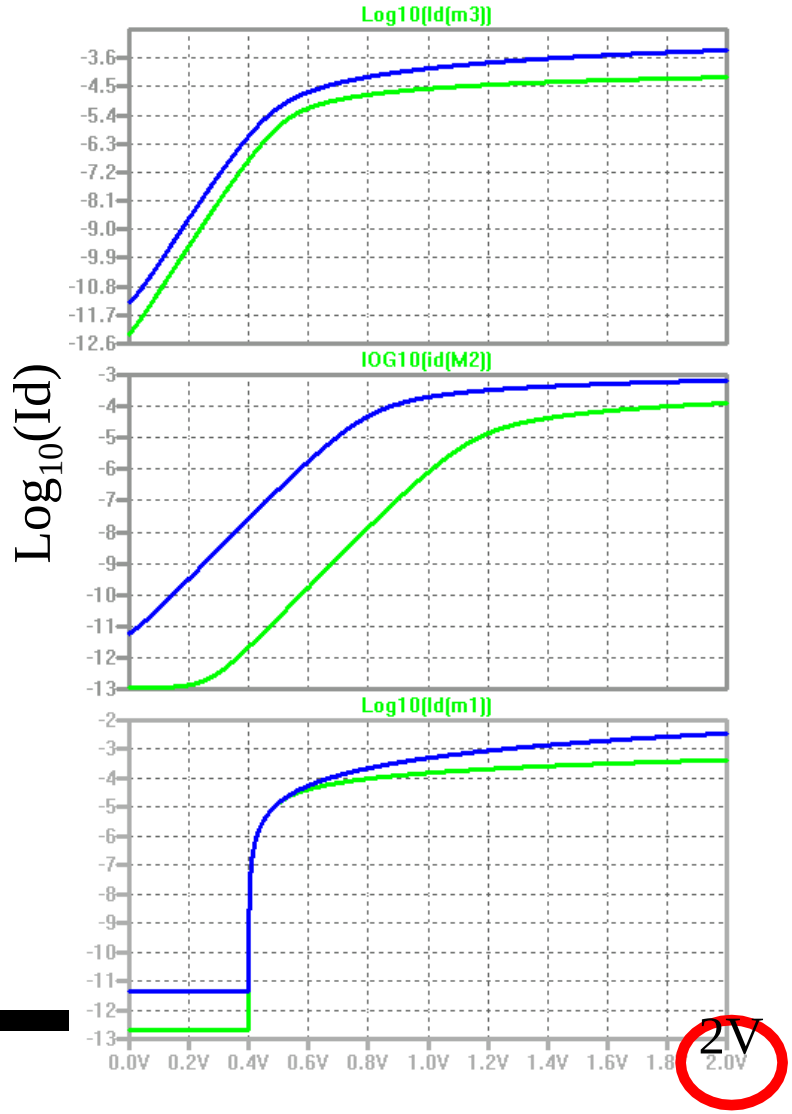
Model is RITSUBN7
L=0.25u W=1.6u

Model not good too much short channel effects

Model is EENMOS
L=0.25u W=1.6u

Model not good current too large does not show mobility degradation

DEEP SUB-MICRON TRANSISTOR MODELS



Model is EECMOSN
L=0.25u W=1.6u

Model good for Deep Sub-Micron MOSFETs

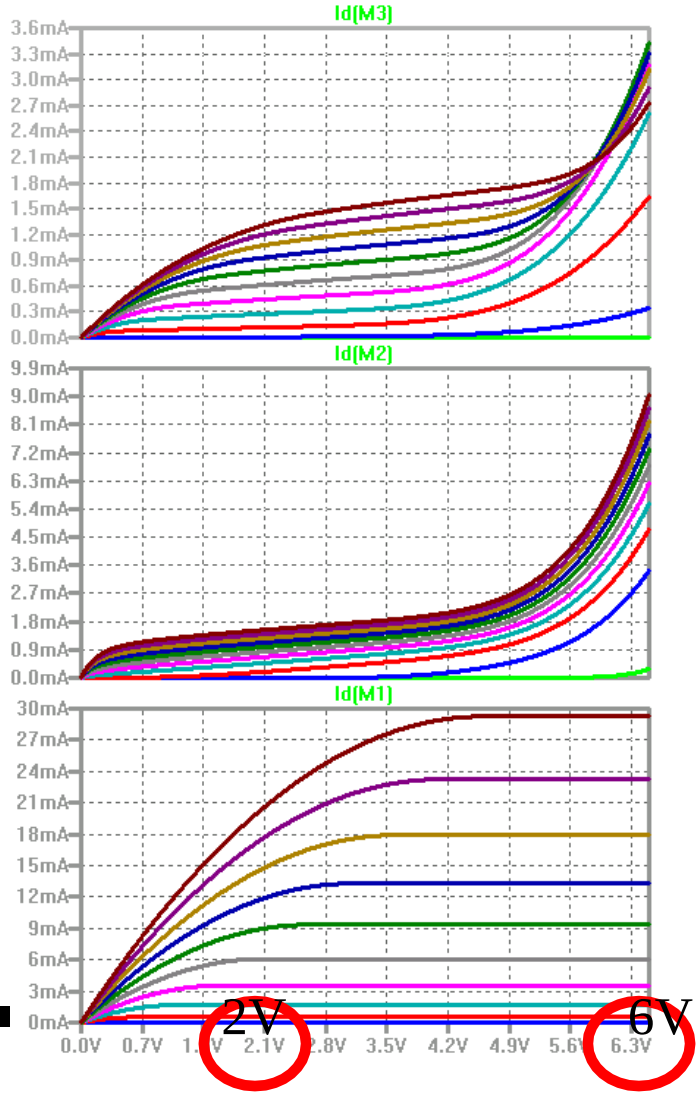
Model is RITSUBN7
L=0.25u W=1.6u

Model not good too much DIBL

Model is EENMOS
L=0.25u W=1.6u

Model incorrect in subthreshold region

DEEP SUB-MICRON TRANSISTOR MODELS



Deep sub-micron transistors show punch through at drain voltages over 3.3 volts. Which is correct.

Problem is worse in the sub-micron transistor because the channel is lighter doped.

Simple model is incorrect.

MOSFET MODELS USED BY LTSPICE

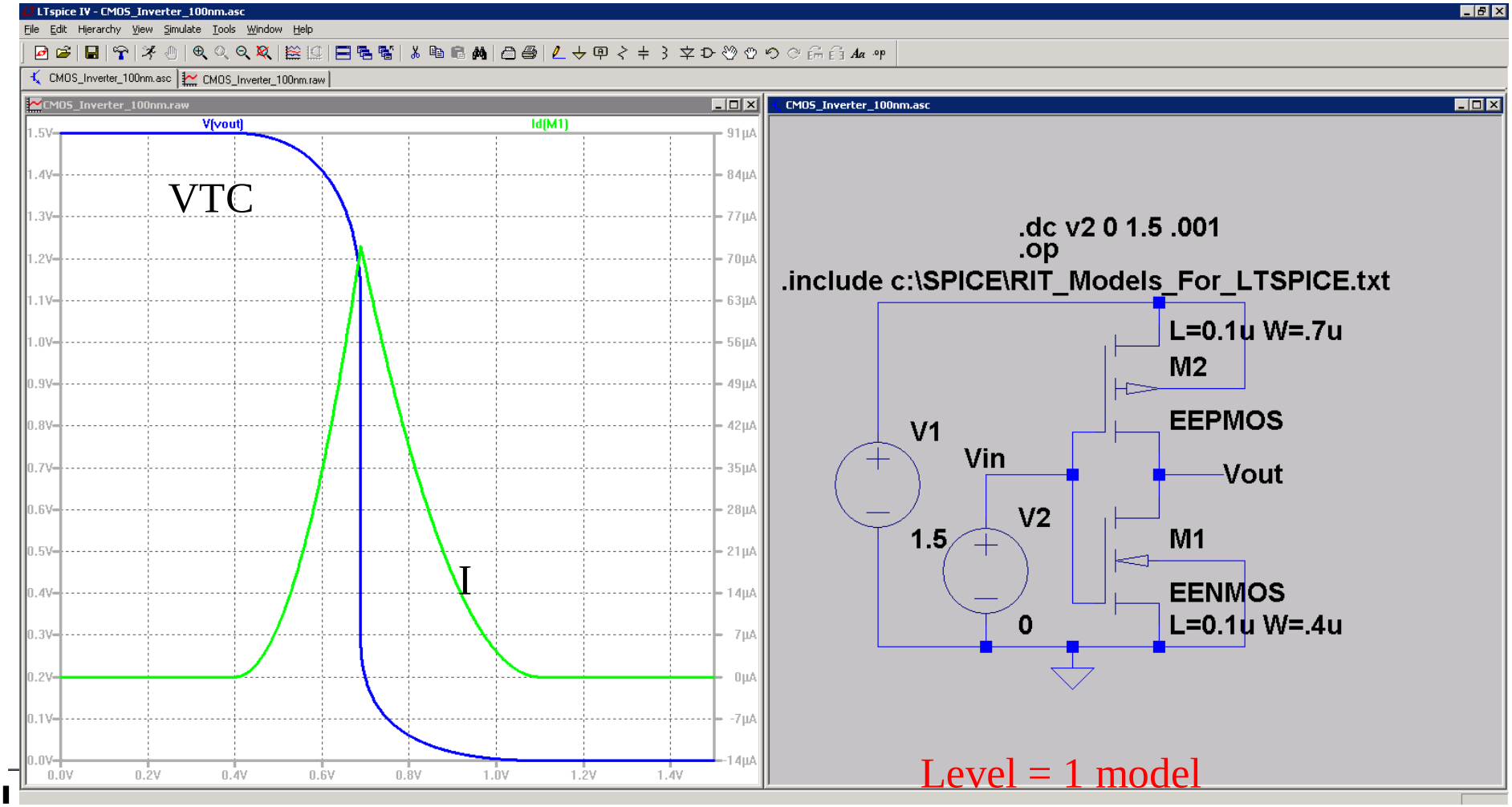
LTSPICE uses several different types of MOSFET models including simple, deep submicrometer, Silicon On Insulator (SOI), Vertical double diffused Power MOSFET. Level = 1 is the default if a model level is not specified.

Level		
1	Shichman and Hodges	1st generation models
2	MOS2, Vladimirescu and Liu, UC Berkeley, October 1980	
3	MOS3, a semi-emperical model, UC Berkeley	
4	BSIM UC Berkeley, May 1985	2nd generation models
5	BSIM2, UC Berkeley, October 1990	
6	MOS6, UC Berkeley, March 1990	
8	BSIM3V3.3.0, UC Berkeley 2005	
9	BSIMSOI3.2, Silicon on Insulator (SOI), UC Berkeley 2004	3rd generation models
14	BSIM4.6.1, UC Berkeley 2007	
	more….	

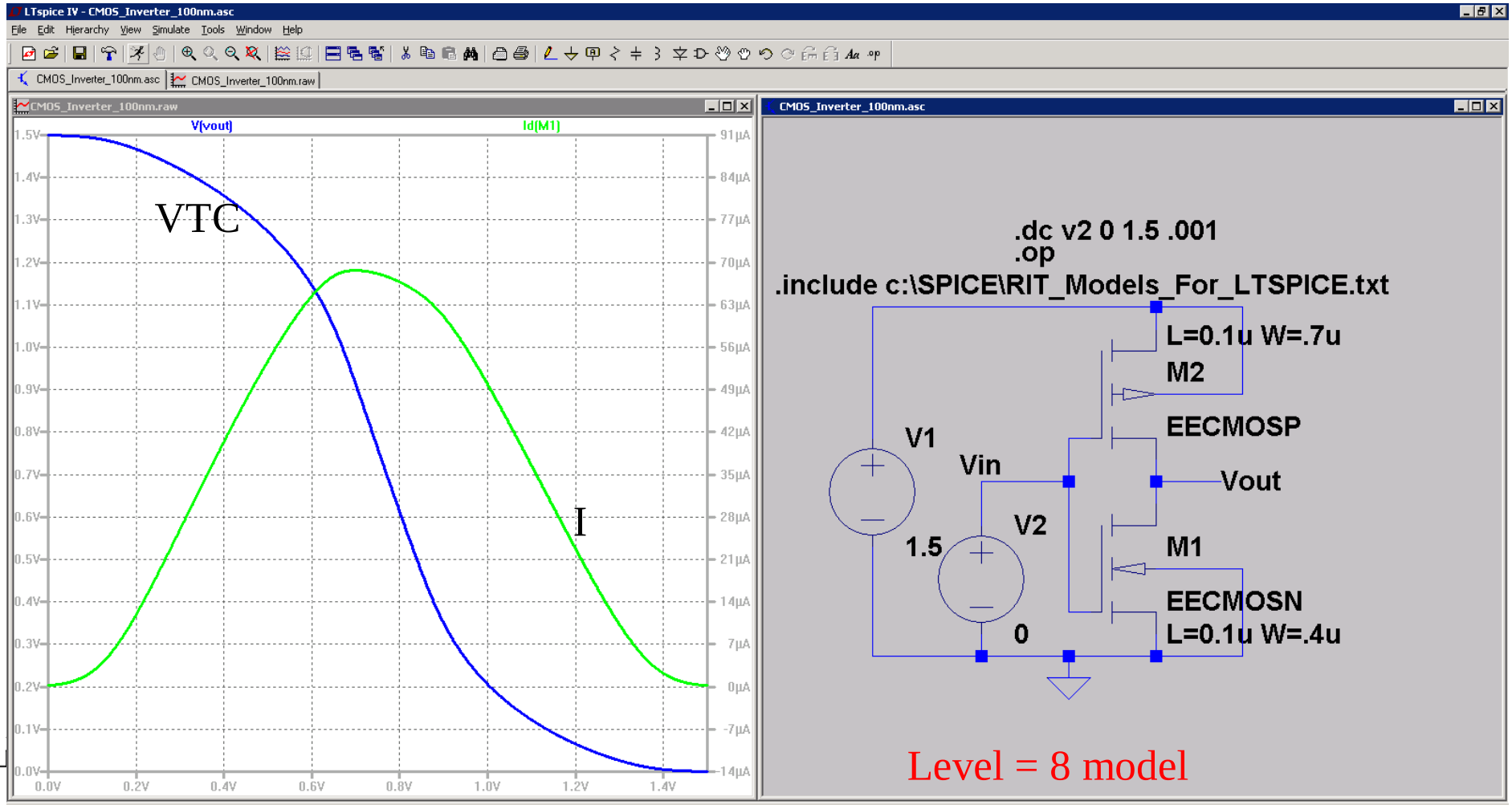
SIMPLE AND ADVANCED SPICE MODEL

```
* From Electronics II EEEE482 FOR ~100nm Technology
.model EECMOSN     NMOS (LEVEL=8
+VERSION=3.1 CAPMOD=2 MOBMOD=1
+TOX=5E-9 XJ=1.84E-7 NCH=1E17 NSUB=5E16 XT=5E-8
+VTH0=0.4 U0= 200 WINT=1E-8 LINT=1E-8
+NGATE=5E20 RSH=1000 JS=3.23E-8 JSW=3.23E-8 CJ=6.8E-4 MJ=0.5 PB=0.95
+CJSW=1.26E-10 MJSW=0.5 PBSW=0.95 PCLM=5
+CGSO=3.4E-10 CGDO=3.4E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 FOR ~100nm Technology
.model EECMOSP      PMOS (LEVEL=8
+TOX=5E-9 XJ=0.05E-6 NCH=1E17 NSUB=5E16 XT=5E-8
+VTH0=-0.4 U0= 100 WINT=1E-8 LINT=1E-8
+NGATE=5E20 RSH=1000 JS=3.51E-8 JSW=3.51E-8 CJ=5.28E-4 MJ=0.5 PB=0.94
+CJSW=1.19E-10 MJSW=0.5 PBSW=0.94 PCLM=5
+CGSO=4.5E-10 CGDO=4.5E-10 CGBO=5.75E-10)
*
* From Electronics II EEEE482 SIMPLE MODEL
.model EENMOS       NMOS (VTO=0.4 KP=432E-6 GAMMA=0.2 PHI=.88)
*
* From Electronics II EEEE482 SIMPLE MODEL
.model EEPMOS       PMOS (VTO=-0.4 KP=122E-6 GAMMA=0.2 PHI=.88)
```

CMOS INVERTER WITH LEVEL 1 SPICE MODEL



CMOS INVERTER WITH LEVEL 8 SPICE MODEL



CONCLUSION

All of these examples are for DC characteristics but similar results would be shown for examples that depend on internal capacitors and resistors such as a study of rise-time, fall time, gate delay, oscillators, multi-vibrators, etc.

In general the third generation SPICE models for MOSFETS give better results.

Level=1 models are not good for MOSFETS with L less than 10um.

Large MOSFETS, SUB-MICRON MOSFETS and DEEP SUB MICRON MOSFET models have been introduced.

REFERENCES

1. MOSFET Modeling with SPICE, Daniel Foty, 1997, Prentice Hall, ISBN-0-13-227935-5
2. Operation and Modeling of the MOS Transistor, 2nd Edition, Yannis Tsividis, 1999, McGraw-Hill, ISBN-0-07-065523-5
3. UTMOST III Modeling Manual-Vol.1. Ch. 5. From Silvaco International.
4. ATHENA USERS Manual, From Silvaco International.
5. ATLAS USERS Manual, From Silvaco International.
6. Device Electronics for Integrated Circuits, Richard Muller and Theodore Kamins, with Mansun Chan, 3rd Edition, John Wiley, 2003, ISBN 0-471-59398-2
7. ICCAP Manual, Hewlet Packard
8. PSpice Users Guide.