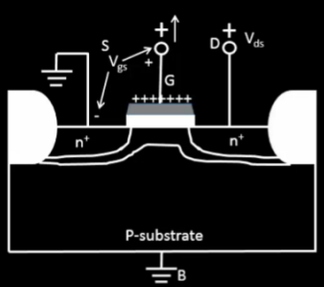
Introduction to SPICE

SPICE setup



Threshold Voltage Equation:

$Vt = Vto + \gamma(\sqrt{|-2\Phi_f + Vsb|} - \sqrt{|-2\Phi_f|})$

$\gamma = \frac{\sqrt{2qNA\epsilon_{si}}}{C_{ox}}$

$\Phi_f = -\Phi_T * \ln\frac{N_A}{n_i}$

Linear region:

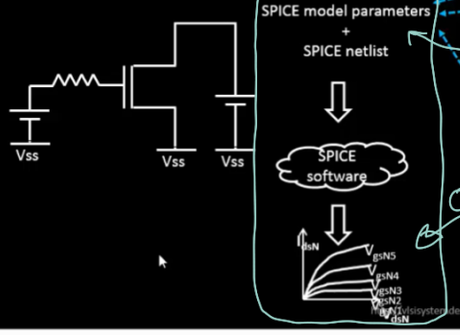
$Id = kn\,.\,[(Vgs - Vt)Vds - \frac{Vds^2}{2}$

Saturation region:

$Id = \frac{Kn}{2}\cdot\frac{W}{L}\cdot(Vgs - Vt)^2\,[1 + \lambda Vds]$

SPICE setup

Technology constants → Come from foundry



Threshold Voltage Equation:

$Vt = Vto + \gamma(\sqrt{|-2\Phi_f + Vsb|} - \sqrt{|-2\Phi_f|})$

$\gamma = \frac{\sqrt{2qNA\epsilon_{si}}}{C_{ox}}$

$\Phi_f = -\Phi_T * \ln\frac{N_A}{n_i}$

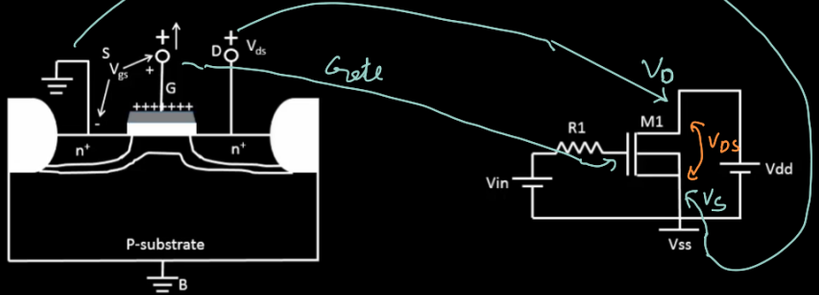
Linear region:

$Id = kn\,.\,[(Vgs - Vt)Vds - \frac{Vds^2}{2}$

Saturation region:

$Id = \frac{Kn}{2}\cdot\frac{W}{L}\cdot(Vgs - Vt)^2\,[1 + \lambda Vds]$

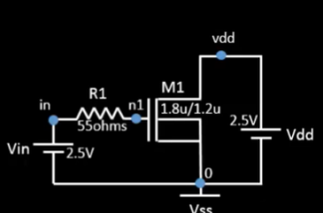
SPICE netlist



SPICE netlist

Technology file

```
M1 vdd n1 0 0 nmos W=1.8u L=1.2u
R1 in n1 55
Vdd vdd 0 2.5
Vin in 0 2.5
```



SPICE netlist
Technology file

Model parameters

Models

```
M1 vdd n1 0 0 nmos W=1.8u L=1.2u
R1 in n1 55
Vdd vdd 0 2.5
Vin in 0 2.5
```

nmos Model

```
.MODEL nmos  NMOS (….
+ …..)
.MODEL pmos  PMOS (….
+ …..)
.endl
```

Threshold Voltage Equation:

$Vt = Vto + \gamma(\sqrt{|-2\Phi_f + Vsb|} - \sqrt{|-2\Phi_f|})$

$\gamma = \frac{\sqrt{2qNA\varepsilon_{si}}}{C_{ox}}$

$\Phi_f = -\Phi_T * \ln\frac{N_A}{n_i}$

Linear region:

$Id = kn\,.\,[(Vgs - Vt)Vds - \frac{Vds^2}{2}$

Saturation region:

$Id = \frac{Kn'}{2}.\frac{W}{L}.(Vgs - Vt)^2\,[1 + \lambda Vds]$

SPICE netlist
Technology file

```
M1 vdd n1 0 0 nmos W=1.8u L=1.2u
R1 in n1 55
Vdd vdd 0 2.5
Vin in 0 2.5
```

same name

Model

```
.lib cmos_models (name of model)
.MODEL nmos  NMOS (TOX = ..
+ VTH0 = .. U0 = .. GAMMA1 = ..)
.MODEL pmos  PMOS (TOX = ..
+ VTH0 = .. U0 = .. GAMMA1 = ..)
.endl
```

Package in a file

xxxx_025um_model.mod

Threshold Voltage Equation:

$Vt = Vto + \gamma(\sqrt{|-2\Phi_f + Vsb|} - \sqrt{|-2\Phi_f|})$

$\gamma = \frac{\sqrt{2qNA\varepsilon_{si}}}{C_{ox}}$

$\Phi_f = -\Phi_T * \ln\frac{N_A}{n_i}$

Linear region:

$Id = kn\,.\,[(Vgs - Vt)Vds - \frac{Vds^2}{2}$

Saturation region:

$Id = \frac{Kn'}{2}.\frac{W}{L}.(Vgs - Vt)^2\,[1 + \lambda Vds]$

SPICE netlist
Simulation Commands

```
*** NETLIST Description ***
M1 vdd n1 0 0 nmos W=1.8u L=1.2u
R1 in n1 55
Vdd vdd 0 2.5
Vin in 0 2.5
*** .include xxxx_1um_model.mod  ***
.LIB "xxxx_025um_model.mod" CMOS_MODELS
```

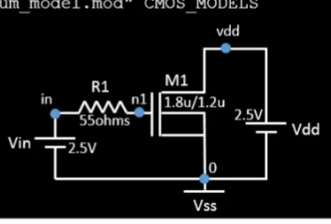
← Netlist description section

← Include section

← Simulation commands
eg. How to sweep voltages

SPICE netlist
Simulation Commands

```
*** NETLIST Description ***
M1 vdd n1 0 0 nmos W=1.8u L=1.2u
R1 in n1 55
Vdd vdd 0 2.5
Vin in 0 2.5
*** .include xxxx_1um_model.mod  ***
.LIB "xxxx_025um_model.mod" CMOS_MODELS
```



Question = How do we calculate Id for different values of 'Vgs' and at every value of Vgs, sweep Vds till (Vgs – Vt) using linear equation for Id ?

$Id = kn\,.\,[(Vgs - Vt)Vds - Vds^2/2$
$= kn\,.\,[(1 - 0.45)\,0.05 - 0.05^2/2$
$= kn\,.\,[0.0275 - 0.00125]$

≈ 0 for Vds <= (Vgs – Vt)

$Id = kn\,.\,0.0275$ → Linear or resistive region of operation

$Id = kn\,.\,(Vgs - Vt)\,Vds$ → linear f(Vds)

Vgs = 0.5V, 1V, 1.5V, 2V, 2.5V
(Vgs – Vt) = 0.05V, 0.55V, 1.05V, 1.55V, 2.05V
Vds = can be sweeped from 0 to 2.05
Vt = 0.45V