EELE 414 – Introduction to VLSI Design

Module #3 – SPICE Modeling

- **Agenda**
  1. SPICE Modeling

- **Announcements**
  1. Read Chapter 4

SPICE Modeling

- **SPICE Modeling**

- <u>S</u>imulation <u>P</u>rogram with <u>I</u>ntegrated <u>C</u>ircuit <u>E</u>mphasis

or

<u>S</u>imulation <u>P</u>rogram for the <u>I</u>ntegrated <u>C</u>ircuit <u>E</u>nvironment

- Developed by UC Berkeley in the late 1970's

- SPICE is an electric circuit simulator (R,L,C,V,I,…)

- Its main contribution at the time was the ability to support transistor **Models**

- Today…

  - Berkeley SPICE is <u>free</u> for Unix/Linux. It is a text based program.

  - CAD vendors take the free SPICE engine and add features like graphical entry and additional components

  - The CAD vendors then sell it for big money (Mentor, Synopsis, Cadence,…)

SPICE Modeling

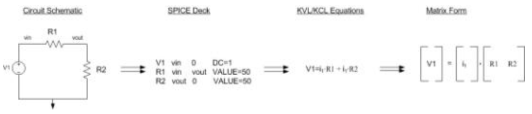
- **SPICE Modeling**

- How does SPICE work?

  - for a given circuit, KCL and KVL equations can be written

  - Just like in EE206, these equations can be solved using Matrix math

  - SPICE does the same thing, except on the front-end it is able to take the entered circuit and create the KCL/KVL equations for us

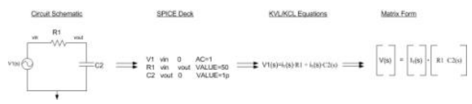


SPICE Modeling

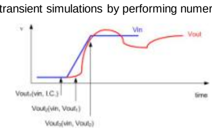
- **SPICE Modeling**

- This can also be extended to AC analysis since the matrix math can handle Complex numbers

- We can create Bode Plots by sweeping the frequency (i.e., running a simulation at each frequency)



- SPICE can also perform transient simulations by performing numerical integration



SPICE Modeling

- **SPICE Modeling**

- The source file for a SPICE simulation is called a **DECK**
  (from the days of punch cards… what is a punch card?... I'm not that old…)

- The DECK can be thought of as a text netlist of the circuit.

- Even when using a graphical entry tool for the schematic, the first thing the tool does when you click "simulate", is create a text-based DECK that is plugged into the SPICE engine.

- the first letter of a component instantiation in the DECK tells SPICE what the component is.

- devices are then followed by the net names they connect to followed by their parameters

```
R1   n1   n2    VALUE=75          * resistor
L1   n2   n3    VALUE=1n          * inductor
C1   n3   n4    VALUE=1p          * capacitor

V1   n4   n5    DC=1v             * DC voltage source
I1   n5   n6    ACmag=1           * AC current source
```


SPICE Modeling

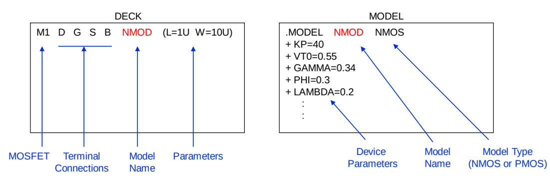
- **SPICE Modeling**

- SPICE allows the use of **MODELS** to represent components with complex, non-linear responses such as Diodes and Transistors

- Models are present in their own file (starting with the .MODEL keyword)

- A component is instantiated in the DECK, but then references the MODEL to describe its behavior

- MOSFETS are denoted with an "M" as their first letter



SPICE Modeling

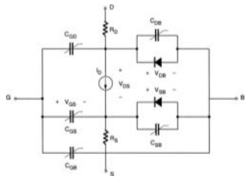
- **SPICE Modeling (Level 1)**

  - There are different *levels* of accuracy and complexity that a model can have.

  - We give these different types of models the description of *Level* (i.e., Level 1 model, Level 2 model..)

  - Increasing model accuracy increases simulation time

  - Let's start by looking at the simplest model for a MOSFET, **Level 1**



- Level 1 uses the basic IV equations

- it also includes:

  1) Resistance of Source & Drain

  2) Capacitance (bias dependant)

  3) Reverse-Bias behavior of Junction Diodes

SPICE Modeling

- **SPICE Modeling (Level 1)**

  - parameters can exist in the Model file. However, we can pass in parameters (i.e., override) the parameters by putting them in the Deck instantiation

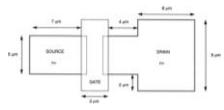
  - there are different types of parameters for the model

"Design Parameters"

- these parameters are under the designer's control

- these *sometimes* have default values, but if we are doing design, this is what we change

Parameter	Description
L	length of channel (drawn)
W	width of channel
AS / AD	area of Source/Drain
PS / PD	perimeter of Source/Drain



SPICE Modeling

- **SPICE Modeling (Level 1)**

"Electrical Parameters"

- there are 5 parameters that fully characterize the base model

- these will have default values in the model based on the fab process

- we can overwrite these from the DECK if we want to perform sensitivity analysis

Parameter	Description
KP	k', transconductance
VTO	V_{T0}, zero substrate bias threshold
GAMMA	γ, substrate-bias coefficient
PHI	$\|2\phi_F\|$, surface potential
LAMBDA	λ, channel length modulation coefficient

SPICE Modeling

- **SPICE Modeling (Level 1)**

"Physical Parameters"

- there are parameters that describe the shape and material properties of the device

Parameter	Description
U0	u_n, electron mobility
TOX	t_{ox}, oxide thickness
NSUB	N_A, doping concentration
LD	L_D, lateral diffusion

- notice that these parameters are redundant with the Electrical parameters since these quantities are used to calculate k', V_{T0}, γ, $|2\phi_F|$, and λ

- these allow you to get further into the details of the fabrication to see its effect on performance

- however, the "Electrical Parameters" **OVERRIDE** the "Physical Parameters"

- this means you wouldn't supply both if you really want to see the effect of a physical parameters on the performance of the device. You would need to remove the electrical parameter.

SPICE Modeling

- **SPICE Modeling (Level 1)**

"Parasitic Parameters"

- these are the capacitances and resistances of the material

Parameter	Description
CJ	C_{J0}, zero-bias bulk capacitance per area
CJSW	C_{J0sw}, zero-bias sidewall capacitance per area

- there parameters scale with the size of the device provided by W,L,AS,AD,PS, and PD.

- there are many more parameters in table 4.1 in the textbook, take a look and you'll see why we need SPICE to properly predict the behavior of a transistor.

SPICE Modeling

- **SPICE Modeling (Level 2)**

  - Level 2 adds the following behavior to the Level 1 model

    1) Variation of the bulk depletion charge dependence on the channel voltage (we assumed it was constant in Level 1).

    2) Variation of electron mobility (u_n) with the applied E-field

    3) Variation of effective Channel Length in Saturation model

    4) Carrier Velocity Saturation

    5) Subthreshold Conduction

  - we also have the ability to indicate which level we want to use. For example, you can have a Level 2 model, but in the instantiation you say:

```
M1  D  G  S  B  NMOD          (Level=1 L=1U  W=10U)
```

this will tell the simulator to ignore all the parameters associated with Level 2 or higher accuracy.

  - we can also put the "Level=1" as the first parameters in the model

SPICE Modeling

- **SPICE Modeling (Level 3)**

  - Level 3 was developed to specifically address small geometry effects.

  - instead of trying to come up with an expression for each and every *bump and wiggle* on the IV curve, Level 3 instead moves toward a more empirical approach.

  - curve-fitting parameters are added to the IV equations from Level 1 and Level 2.

  - these parameters are dialed-in based on measurement data from a test run of transistors.

SPICE Modeling

- **SPICE Modeling (BSIM)**

  - Berkeley Short-Channel IGFET Model

  - what is an IGFET?

    - the term MOSFET implies a "metal" contact for the gate. Some say that is in not accurate for transistors that use *polysilicon* for the gate contact since polysilicon is not considered a true metal. Of course polysilicon is a conductor, just not a pure metal.

    - so the term "Insulated Gate FET" is used which describes any type of conducting gate used.

  - this is a totally empirical model which reduces the # of curve fitting parameters

  - this actually reduces simulation time over the Level 3 models, and sometimes over Level 2 due to moving away from IV equations with many coefficients.

  - there have been many versions of the BSIM models, but the most current is **BSIM3**

  - this is the most commonly used model for accurate simulations.

SPICE Modeling

- **SPICE Modeling**

  - Which Model should I use?

    - simulation is always a tradeoff of accuracy vs. simulation time

    - simulation time is a big problem. You'll never ship if each simulation takes a month, (and they easily can!!!)

    - typically, we can use Level 1 models for quick, functional simulations

      ex) if I hook it up like this, does it do what I think it is supposed to?

    - then we move to BSIM3 models for accurate simulations which tell us speed, power, etc…