Module #3 – SPICE Modeling

- Agenda
  1. SPICE Modeling

- Announcements
  1. Read Chapter 4

SPICE Modeling

- Simulation Program with Integrated Circuit Emphasis
- Simulation Program for the Integrated Circuit Environment
- 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 free 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, Synopsys, Cadence,...)

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

- 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

- 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

<table>
<thead>
<tr>
<th>DECK</th>
<th>Model</th>
<th>Parameters</th>
<th>Model Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ri, Re, Vcc</td>
<td>Vcc, Ri, Re</td>
<td>Vcc, Ri, Re</td>
<td>NMOS or PMOS</td>
</tr>
<tr>
<td>L1, L2, R1, R2</td>
<td>R1, R2</td>
<td>R1, R2</td>
<td>Inductor</td>
</tr>
<tr>
<td>C1, C2, C3, C4, C5</td>
<td>C1, C2</td>
<td>C1, C2</td>
<td>Capacitor</td>
</tr>
<tr>
<td>V1, h, v, DC, Vcc, Vgs</td>
<td>Vcc, Vgs</td>
<td>Vcc, Vgs</td>
<td>Diode</td>
</tr>
<tr>
<td>R1, n, Rn</td>
<td>R1, n</td>
<td>R1, n</td>
<td>Current source</td>
</tr>
</tbody>
</table>

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 dependent)
    3) Reverse-Bias behavior of Junction Diodes

- **Design Parameters**
  - These parameters are under the designer's control
  - Some have default values, but if we are doing design, this is what we change
  - Parameter Description
  - Length of Channel:
  - Width of channel:
  - Area of Source/DRAIN:
  - Perimeter of Source/DRAIN:

- **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 a sensitivity analysis
  - Parameter Description
    - $K_P$: transconductance
    - $V_{T0}$: Threshold voltage
    - $\gamma$: Substrate bias coefficient
    - $|\Phi_F|$: Surface potential
    - $\lambda$: Channel length modulation coefficient

- **Physical Parameters**
  - These are parameters that describe the shape and material properties of the device
  - Notice that these parameters are redundant with the Electrical parameters since these quantities are used to calculate $K_P$, $V_{T0}$, $\gamma$, $|\Phi_F|$, and $\lambda$
  - 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 parameter on the performance of the device. You would need to remove the electrical parameter.

- **Parasitic Parameters**
  - These parameters are the capacitances and resistances of the material
  - Parameter Description
    - $C_J$: Zero-bias bulk capacitance per area
    - $C_{JSW}$: Zero-bias sidewall capacitance per area
  - 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 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 (BSIM)
  - Berkeley Short-Channel IGFET Model
  - What is an IGFET?
    - The term MOSFET implies a "metal" contact for the gate. Some say that is 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 really can’t)
    - Typically, we can use Level 1 models for quick, functional simulations
      - eg) If I hook it up like this, does it do what I think it is supposed to?
    - Then we move to BSIM models for accurate simulations which tell us speed, power, etc…