Device Simulation for 65 nm

By
Harish B.P.

Microelectronics Lab,
Dept. of Electrical Communication Engg.,
Indian Institute of Science, Bangalore.
E-mail: harish@ece.iisc.ernet.in
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple \( I_d - V_g \) of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
Introduction

• Device simulation essential component of process design and development and device design.
• Device simulation provides quick feedback about device design before long and expensive fabrication.
• Commercially available computer simulation tools can solve all the device equations simultaneously with few or no approximations.
• Simulated results are as accurate as the models of physical effects included in simulation.
• Simulated output is to be compared with experimental measurements for
  1. model validity during model/tool development and
  2. design validity during tool usage.
Modeling & Simulation Approach

- Establish device equations (e.g. continuum based partial differential equations)
- Discretize PDEs into difference equations
- Solve difference equations (usually nonlinear)
- Post-process -- interpret simulation results
Device Simulation Approach

- In a device simulator Poisson’s eq. and Continuity eq. are solved.

- Poisson’s Eq. :
  \[ \nabla \cdot (-\varepsilon \nabla \Psi) = \rho \]
  where \( \rho = q [p - n + N_d^+ - N_a^-] \) (vol. charge density)

- Continuity Eq. :
  \[ \frac{\partial n}{\partial t} = \frac{1}{q} \frac{\partial J_n}{\partial x} - u \]
  \[ \frac{\partial p}{\partial t} = -\frac{1}{q} \frac{\partial J_p}{\partial x} - u \]
  where \( u = R - G \)
  \( G = \) electron/hole generation rate
  \( R = \) electron/hole recombination rate

These coupled PDEs can not be solved analytically and are solved numerically for self-consistent solution.
Device Eq.s (contd.)

• Total electron or hole current density:

\[ J_n = J_{ndrift} + J_{ndiff} \]

\[ J_n = n q \mu_n E + q D_n \frac{dn}{dx} \quad \text{for electrons} \]

\[ J_p = p q \mu_p E - q D_p \frac{dp}{dx} \quad \text{for holes} \]

Total conduction current density \( J = J_n + J_p \)
Discretization: Numerical techniques

- Time and space discretization to tackle coupled non-linear partial differential equations (PDE).

  Discretization
  \[ \text{PDE} \rightarrow \text{Difference Equations} \]

- Non-linear difference equations solved using iteration techniques like Newton-Raphson method etc.
Space discretization

- The device cross section is represented as the collection of small cells. The various quantities such as $\varphi$, $n$, $p$ etc are constant within the cell.
- Granularity of discretization: meshing - fine or coarse
- Mesh must be densest in device regions where
  - current density is high (MOS channels, bipolar base)
  - electric fields are high (MOS channels, drain, depletion regions)
  - charge generation is high (SEU alpha particle)
Time discretization

- Discretize the time instants at which each cell is evaluated for its physical and chemical composition.
Outline

• Introduction – Modeling and simulation approach
• **Device simulators – DESSIS**
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
Device Simulators

- Florida Object Oriented Device Simulator (FLOODS)
- Taurus-Medici from Synopsys
- ATLAS device simulation framework: S-Pisces/Device3D from Silvaco
- Sentaurus Device from Synopsys
- DESSIS from Synopsys
Introduction to DESSIS

- DESSIS is a multidimensional, electrothermal, mixed-mode device and circuit simulator for 1D, 2D, and 3D semiconductor devices.
- Advanced physical models and robust numerical methods for simulation.
- Simulates semiconductor devices from nano scale Si MOSFETs to large bipolar power structures.
- Supports SiC and III-V compound homostructure and heterostructure devices.
DESSIS (contd.)

• Simulates the electrical behaviour of a single SC device or of several devices combined in a circuit, numerically.
• Terminal currents, voltages, and charges are computed based on a set of physical device equations – poisson and continuity equations.
• MOS transistor represented in the simulator as a virtual device whose physical properties are discretized onto a non-uniform grid of nodes.
• Continuous properties like doping profiles are represented on a sparse mesh – defined at a finite no. of discrete points in space.
• Doping at any point between nodes is computed by interpolation.
Representation of Virtual Device

- Device structure described by 2 files:
  1. file_name.grd:
     - Grid or geometry file describes regions of device – boundaries, material types, location of elec. contacts.
     - contains the grid or locations of all discrete nodes and their connectivity.
  2. file_name.dat:
     - Data or doping file contains doping profiles in the form of data associated with discrete nodes.
Design Flow: DESSIS simulator

MDRAW

command
_mdrcmd
boundary
_mrdbnd

grid
_mdrgrd

doping
_mrdat

DESSIS

command
_descmd
parameter
.par

output
_mdrlag

current
_desplt

plot
.des.dat

nmos
 dio.grd
nmos
 dio.dat
output
_mrdlog

output
.des.log
Features of Dessis

- Supports 1D, 2D, 3D device structures.
- Set of non-linear solvers.
- Mixed-mode support for electrothermal netlists with mesh-based device and SPICE circuit models.
- Set of models for device physics and effects:
  - Drift-diffusion, Thermodynamic, Hydrodynamic models.
  - Monte Carlo
  - Tunneling through insulators
  - Hot carrier injection
  - Interface traps, bulk traps
  - Ferroelectrics
  - Optical generation (Single Event Upset – SEU)

- Analysis: DC, AC, Transient, Noise
Outline

- Introduction – Modeling and simulation approach
- Device simulators – DESSIS
- **Physical models for device simulation**
- Device simulation: Simple $I_d - V_g$ of NMOS
- Mixed-mode simulation
- Case study I: AC simulation
- Case study II: Transient simulation of inverter
- Some views on modeling and simulation
- Summary
# Physical Models: 65 nm devices

<table>
<thead>
<tr>
<th>Physical effects</th>
<th>Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carrier transport</td>
<td>Drift-diffusion, thermodynamic, hydrodynamic models</td>
</tr>
<tr>
<td>Mobility</td>
<td>Doping dependence, velocity saturation, transverse field dependence</td>
</tr>
<tr>
<td>Si bandgap narrowing</td>
<td>OldSlotboom, Slotboom (determines intrinsic carrier concentration)</td>
</tr>
<tr>
<td>Si bandgap widening</td>
<td>Channel quantization models – QC van Dort model, 1D Schrodinger eq., density gradient model</td>
</tr>
<tr>
<td>Recombination</td>
<td>Carrier generation and recombination – SRH recombination model – doping dependent lifetime</td>
</tr>
<tr>
<td>Gate current</td>
<td>Direct tunneling, FN-tunneling, Lucky electron injection, hot carrier injection</td>
</tr>
<tr>
<td>Interfaces</td>
<td>Interface charge, trapped charge</td>
</tr>
</tbody>
</table>

Harish B.P.  Microelectronics Lab, ECE, IISc  12 December 2006
Modeling Carrier Transport

• Drift-diffusion model:
  - widely used for simulation of carrier transport in semiconductors
  - defined by electron and hole current densities in terms of carrier mobility and electric field.
  - suitable for low power density devices with long active regions

• Thermodynamic model (non-isothermal):
  - to simulate effects of
    a. self-heating on temperature distribution and
    b. non-uniform temperature distribution on electrical characteristics.
  - suitable for high power density devices with long active regions
Modeling Carrier Transport (contd.)

- Hydrodynamic model:
  - accounts for energy transport of carriers.
  - to simulate velocity overshoot and correct estimation of impact ionization rates in deep sub-micron devices.
  - carrier temperatures not assumed to be equal to lattice temperature.
  - suitable for small active regions.
Selection of Physical Models

- understand the models available in the tool.
- understand the parameters in the model you select.
- know the default models and their parameters
- check for conflicts between various models
  (i.e. if model A is selected, model B can’t be used)
- Proper selection and specification of physical models is critical!
Outline

- Introduction – Modeling and simulation approach
- Device simulators – DESSIS
- Physical models for device simulation
- **Device simulation: Simple** $I_d - V_g$ **of NMOS**
- Mixed-mode simulation
- Case study I: AC simulation
- Case study II: Transient simulation of inverter
- Some views on modeling and simulation
- Summary
Structure of Simulation Program

- **Sections:**
  1. File
  2. Electrode
  3. Physics
  4. Plot
  5. Math
  6. Solve
MOSFET $I_d$-$V_g$ Simulation - Command File

• File {* I/O files:
  Grid = "nmos_mdr.grd"
  Doping = "nmos_mdr.dat"
  Plot = "n_des.dat"
  Current = "n_des.plt"
  Output = "n_des.log"
}

• Electrode {
  { Name="source" Voltage=0.0 }
  { Name="drain" Voltage=0.1 }
  { Name="gate" Voltage=0.0 Barrier=-0.55 }
  { Name="substrate" Voltage=0.0 }
}
Command File (contd.)

- Physics {
  Mobility( DopingDep HighFieldsat Enormal )
  EffectiveIntrinsicDensity(BandGapNarrowing (OldSlotboom))
}

- Plot {
  eDensity hDensity eCurrent hCurrent
  Potential SpaceCharge ElectricField
  eMobility hMobility eVelocity hVelocity
  Doping DonorConcentration AcceptorConcentration
}
Command File (contd.)

• Math {
    Extrapolate
    Derivatives
    RelErrControl
    NewDiscretization
}

• Solve {
    #-initial solution:
    Poisson
    Coupled { Poisson Electron }
    #-ramp gate:
    Quasistationary ( MaxStep=0.05
    Goal{ Name="gate" Voltage=2 } )
    { Coupled { Poisson Electron } }
}

END of Command File

Harish B.P.  Microelectronics Lab, ECE, IISc  12 December 2006
Id-V\textsubscript{g} and Id-V\textsubscript{ds} characteristics of 65 nm NMOS

a. Id - V\textsubscript{g} characteristics  
b. Id-V\textsubscript{ds} characteristics
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• **Mixed-mode simulation**
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
Mixed mode Simulation

Combining two simulations with different levels of abstraction

Device
DESSIS

Circuit
SPICE

Switch/Logic
IRSIM
Mixed-mode simulations

- Combining two simulations with different levels of abstraction
- Combining DESSIS & SPICE is also mixed mode Simulation
- Mixed mode is used in a much broader sense (can mean one or all of these)
  1. Mixed signal: analog & digital circuits with distinctively different waveforms (voltage Vs. logic state)
  2. Mixed level: same circuit described at different levels of abstraction
  3. Mixed precision: multiple precision used at different levels of abstraction
  4. Mixed method: different simulation algorithms for different parts of circuit.
Mixed device and circuit capabilities

a. single device simulations
b. single device with a circuit netlist
c. multiple devices with a circuit netlist

1. Different physical models applied on individual devices.
2. Supports devices of different dimensionality – 1D, 2D or 3D.
3. Combines DESSIS devices with other devices based on SPICE compact models.
Model classification

- DESSIS provides
  1. SPICE models – compact circuit models from BSIM3v3.2, BSIM4.1.0 and BSIMPDv2.2.2 Ex. R, L, C, VS, CS, BJT, diode, JFET, MOSFET, GaAs MESFET models.
  3. User models – compact model model interface (CMI) available for user-defined models in C++.
Structure of Simulation Program: Mixed-mode Environment

- **Device section:**
  1. Electrode
  2. File
  3. Physics
  4. Plot

- **General section:**
  1. Math
  2. File
  3. System
  4. Solve
Mixed-mode Simulations - Examples

- Two mixed-mode simulations:
  1. AC analysis: to obtain small signal admittance $Y$ matrix - current response at a node to a small signal voltage at another node of the form:
    $$ i = Y \times \nu = A \times \nu + j \omega C \times \nu $$
    $i =$ small signal current vector (at all nodes)
    $\nu =$ voltage vector
    DESSIS output is conductance matrix and capacitance matrix.
  2. Transient analysis of inverter circuit: DESSIS devices combined with other devices based on compact models like capacitor and voltage source (to obtain I/O and transfer characteristics).
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
I. MOSFET AC Analysis - Command File

- Device NMOS {
  Electrode {
    { Name="source" Voltage=0.0 }
    { Name="drain" Voltage=1.2 }
    { Name="gate" Voltage=0.0 Barrier=-0.55 }
    { Name="substrate" Voltage=0.0 }
  }
}

File {
  Grid = "@grid@"
  Doping = "@doping@"
  Current = "@plot@"
  Plot = "@dat@"
  Param = "mos"
}
Command File (contd.)

Physics {
    Mobility( DopingDep HighFieldSaturation Enormal )
    EffectiveIntrinsicDensity( BandGapNarrowing
    (oldSlotboom))
}

Plot {
    eDensity hDensity eCurrent hCurrent
    ElectricField eEparallel hEparallel
    eQuasiFermi hQuasiFermi
    Potential Doping SpaceCharge
    DonorConcentration AcceptorConcentration
}
}
Math {
    Extrapolate
    Derivatives
    RelErrControl
    NewDiscretization
    Notdamped=50
    Iterations=20}

File {
    Output = "@log@"
    ACEExtract = "@acplot@"}

System {
    NMOS trans (drain=d source=s gate=g substrate=b)
    Vsource_pset vd (d 0) {dc=2}
    Vsource_pset vs (s 0) {dc=0}
    Vsource_pset vg (g 0) {dc=0}
    Vsource_pset vb (b 0) {dc=0}}
Command File (contd.)

- Solve (  
  #-a) zero solution  
Poisson  
Coupled { Poisson Electron Hole }  
  #-b) ramp gate to negative starting voltage  
Quasistationary (  
  InitialStep=0.1 MaxStep=0.5 Minstep=1.e-5  
  Goal { Parameter=vg.dc Voltage=-2 }  
)  
  { Coupled { Poisson Electron Hole } }
Command File (contd.)

#-c) ramp gate -2V to +3V
Quasistationary (InitialStep=0.01 MaxStep=0.04 Minstep=1.e-5
Goal { Parameter=vg.dc Voltage=3 }
)
{ ACCoupled (StartFrequency=1e6 EndFrequency=1e6
NumberOfPoints=1 Decade
Node(d s g b) Exclude(vd vs vg vb)
)
{ Poisson Electron Hole }
}

END of Command File
CV-characteristics of 65 nm devices

a. NMOS

![NMOS Graph]

b. PMOS

![PMOS Graph]

Harish B.P.  Microelectronics Lab, ECE, IISc  12 December 2006
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
II. Inverter Transient Simulation
Inverter Simulation - Command File

Device NMOS {
  Electrode {
    { Name="source" Voltage=0.0 Area=1 }
    { Name="drain" Voltage=2.0 Area=1 }
    { Name="gate" Voltage=0.0 Area=1 Barrier=-0.55 }
    { Name="substrate" Voltage=0.0 Area=1 }
  }
  File {
    Grid = "@grid@"
    Doping = "@doping@"
    Current = "nmos"
    Plot = "nmos"
    Param = "mos"
  }
  Physics {
    Mobility( DopingDep HighFieldSaturation Enormal )
    EffectiveIntrinsicDensity(BandGapNarrowing (oldSlotboom))
  }
}

Harish B.P. Microelectronics Lab, ECE, IISc 12 December 2006
Command File (contd.)

Device PMOS {
  Electrode {
    { Name="source" Voltage=0.0 Area=1 }
    { Name="drain" Voltage=2.0 Area=1}
    { Name="gate" Voltage=0.0 Area=1 Barrier=-0.55 }
    { Name="substrate" Voltage=0.0 Area=1}
  }
  File {
    Grid = "@grid@"
    Doping = "@doping@"
    Current = “pmos"
    Plot = “pmos"
    Param = "mos“
  }
  Physics {
    Mobility ( DopingDep HighFieldSaturation Enormal )
    EffectiveIntrinsicDensity (BandGapNarrowing (oldSlotboom))
  }
}

Harish B.P. Microelectronics Lab, ECE, IISc 12 December 2006
Command File (contd.)

System {
    Vsource_pset v0 (n1 n0) { pw1 = (0.0e+00 0.0
        1.0e-11 0.0
        1.5e-11 2.0
        10.0e-11 2.0
        10.5e-11 0.0
        20.0e-11 0.0)
    }
    NMOS nmos ( "source"=n0 "drain"=n3 "gate"=n1 "substrate"=n0 )
    PMOS pmos ( "source"=n2 "drain"=n3 "gate"=n1 "substrate"=n2 )
    Capacitor_pset c1 ( n3 n0 ){ capacitance = 3e-14 }
    Set (n0 = 0)
    Set (n2 = 2)
    Set (n3 = 2)
    Plot "nodes.plt" (time() n0 n1 n2 n3 )
}
File {
    Current= "inv"
    Output = "inv"
}

Harish B.P.          Microelectronics Lab, ECE, IISc      12 December 2006
Command File (contd.)

Plot {
  eDensity hDensity eCurrent hCurrent
  ElectricField eEnormal hEnormal eQuasiFermi hQuasiFermi
  Potential Doping SpaceCharge
  DonorConcentration AcceptorConcentration
}

Math {
  Extrapolate
  RelErrControl
  Digits=4
  Iterations=12
  NewDiscretization
  NoCheckTransientError
}
Command File (contd.)

- Solve {
  #-build up initial solution
  Coupled { Poisson }
  Coupled { Poisson Electron Hole }
  Unset (n3)
  Transient (InitialTime=0 FinalTime=20e-11 InitialStep=1e-12 MaxStep=1e-11 MinStep=1e-15 Increment=1.3 )
  { Coupled { nmos.poisson nmos.electron nmos.contact pmos.poisson pmos.hole pmos.contact } }
}

END of Command File
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
Some views on modeling and simulation

• Many members of the SPICE generation merely hack away at design. They guess at circuit values, run a simulation, and then guess at changes before they run the simulation again…..and again…..and again.
Designers need an ability to create a simple and correct model to describe a complicated situation - designing on the back of an envelope. The back of the envelope has become the back of a cathode ray tube, and intuition has gone on vacation.

Paraphrased from:
another view on modeling and simulation

• “All software begins with some fundamental assumptions that translate into fundamental limitations, but these are not always displayed prominently in advertisements. Indeed, some of the limitations may be equally unknown to the vendor and to the customer. Perhaps the most damaging limitation is that software can be misused or used inappropriately by an inexperienced or overconfident engineer.”

stand up to a computer!

• “The use of sophisticated computer simulation tools is a growing component of modern engineering practice. These tools are unavoidably based on numerous assumptions and approximations, many of which are not apparent to the user and may not be fully understood by the software developer. But even in the face of these inherent uncertainties, computer simulation tools can be a powerful aid to the engineer”.

• “Engineers need to develop an ability to derive insight and understanding from simulations. They must be able to “stand up to a computer” and reject or modify the results of a computer-design when dictated to do so by engineering judgement.”

Paraphrased from:
How to use a simulation program?

• “The basic difference between an ordinary TCAD user and an true technology designer is that the former is relaxed, accepting on faith the program’s results, the latter is concerned and busy checking them in sufficient depth to satisfy himself that the software developer did not make dangerous assumptions. It takes years of training, followed by hands-on design practice to develop this capability. It cannot be acquired with short courses, or with miracle push-button simulation tools that absolve the engineer of understanding in detail what he is doing.”

Paraphrased from:
Final thought on modeling and simulation

• “The purpose of computing is insight, not numbers.”

R. W. Hamming
Outline

• Introduction – Modeling and simulation approach
• Device simulators – DESSIS
• Physical models for device simulation
• Device simulation: Simple $I_d - V_g$ of NMOS
• Mixed-mode simulation
• Case study I: AC simulation
• Case study II: Transient simulation of inverter
• Some views on modeling and simulation
• Summary
Summary

• Modeling and simulation of semiconductor devices introduced.
• Device simulators and their capability introduced by taking DESSIS simulator for a case study.
• A case study of mixed-mode simulation of 65 nm NMOS for DC and AC analysis and transient analysis of inverter circuit presented.
Thank You

Harish B.P.  Microelectronics Lab, ECE, IISc  12 December 2006