GNU Tools for CM Standardization

Paolo Nenzi (Presenter), Fabrizio Ramundo and Mauro Olivieri
Università di Roma “La Sapienza”
Summary

- CM Standardization
  - Model implementation issues
  - A modeling standard
- Open tools for CM:
  - GSS-TCAD
  - ExtractCAD
  - NGSpice
  - ADMS & SimCAS
- Conclusions
Simulator implementation of compact models

- Implementing model into a circuit simulator requires:
  - Writing model equations into simulator’s language.
  - Writing “glue” code to link the model to the rest circuit simulator:
    - Code to setup the model (memory allocation, state vector space reservation)
    - Code to load coefficients to the system matrix and vector (MNA formulation).
    - Code to set parameters from user circuit description (netlist).
    - Code to send simulation results to the simulator.
    - Code to verify convergence of the Newton iteration (not all simulators handle this at device level).
Simulator implementation of compact models

Ngspice compact model interface for BSIM4 model

- 20 .c files of which only 7 contains numerical code
- 4 .h files (not shown)
## Simulator implementation of compact models

<table>
<thead>
<tr>
<th>Model</th>
<th>C - Lines Of Code</th>
<th>VA – Lines Of Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>BSIM4</td>
<td>24,215</td>
<td></td>
</tr>
<tr>
<td>BSIM3</td>
<td>12,563</td>
<td></td>
</tr>
<tr>
<td>MOS3</td>
<td>8,529</td>
<td></td>
</tr>
<tr>
<td>CAP</td>
<td>1,601</td>
<td></td>
</tr>
<tr>
<td>RES</td>
<td>1,556</td>
<td></td>
</tr>
<tr>
<td>EKV 2.6</td>
<td>6,328</td>
<td>661</td>
</tr>
<tr>
<td>HICUM0</td>
<td>11,286</td>
<td>856</td>
</tr>
<tr>
<td>HICUM2</td>
<td>31,840</td>
<td>1,664</td>
</tr>
<tr>
<td>MEXTRAM</td>
<td>14,692</td>
<td>1,712</td>
</tr>
<tr>
<td>PSP 102</td>
<td>40,651</td>
<td>4,348</td>
</tr>
<tr>
<td>RES (VA)</td>
<td>1,414</td>
<td>25</td>
</tr>
</tbody>
</table>
Simulator implementation of compact models

- BSIM 4 consists of more than 20000 lines of code.
- Coding effort grew over time: MOS3 (~1980) has only 8000 lines, BSIM3 (BSIM4 predecessor) has 12000.
- The simple linear resistor (in ngspice) circuit simulator has ~1500 lines of code!
- Most of a model code is interface code:
  - this code is simulator specific,
  - this code has no numerical relevance,
  - this code has a regular structure.
Simulator implementation of compact models

- Problems for the model developer:
  - **Correctness**
    - Derivatives - Many simulators API require explicit coding of the Jacobian matrix entries.
    - Code dimension – It is usual to overlook something in 20000 lines of code.
  - **Performance**
    - Coding language - Model developer have to develop in-depth knowledge of the language in which the code is written.
    - API – Model developer have to develop in-depth knowledge of simulators API for compact models.
  - **Distribution**
    - Slow model dissemination - Each implementation is unique to each simulator. Model fixes and updates (new releases) propagates slowly.
Simulator implementation of compact models

- Possible solutions:
  - Correctness
    - Do not code derivatives explicitly. Leave this task to an automatic differentiator.
    - Reduce the lines of code that the model developer has to write to the numerical code only.
  - Performance
    - Leave the task of code optimization to the compiler and the simulator developer (who should know his own APIs).
  - Distribution
    - Standardize compact model development flow.
A modeling standard

Benefits:

For model developer:
- “Code once, run everywhere” – W. Grabinski.
- Focus on model equations and not implementation.

For the software vendors:
- Simplified implementation of the standard models.
- Proprietary models can be supported.

For the silicon fabs:
- Standardized model parameter set

For the designer (end user):
- Standard libraries and design kits
Open tools for CM
## Needed tools

<table>
<thead>
<tr>
<th>Task</th>
<th>Tools and Software</th>
</tr>
</thead>
</table>
| Write model equations        | - Tool to represent ODE equations:  
                               - CAS tools, numerical software.                                                |
| Get data from device         | - Simulated data:  
                               - TCAD simulation software  
                               - Measured data from wafer  
                               - Instrumentation driving software                                              |
| Extract parameters for model | - Parameter extraction tool                                                       |
| Simulate                      | - Circuit simulation tool                                                          |
Internet survey for open tools

- There are ~600 EDA tools available on the Internet (sourceforge.net web site).
- Almost 90% of them are abandoned or not working.
- Need to test the quality of the remaining 10%.
Open TCAD tools


- **CIDER**: 1D ad 2D simulator (integrated into ngspice).
Open extraction tools

- ExtractCAD - www.ymlab.org/software/ecad/extractcad.htm - Extract CAD (ECAD) provides a environment for users to extract parameters in compact models to fit the measured data.
- Octave - www.gnu.org/software/octave - Octave is a general purpose numerical analysis software.
Open circuit simulation tools

- Espice – espice.ugr.es
- NGSpice – www.ngspice.org
- QUCS – www.qucs.org
- GNUcap - www.gnu.org/software/gnuicap
- .... many others.
- The proliferation of free circuit simulator tools is due to the original spice2 and spice3 open distribution policy.
Flow

The following tools have shown to work together (with little gluing effort).

**TCAD**
- GSS-TCAD for device simulation.

**Extraction**
- ExtractCAD for BSIM4 model parameter extraction.

**Simulator**
- NGSpice + ADMS+Verilog-A
25nm Well Tempered NMOS

- Chosen as test device:
  - Structure TCAD file and simulation results are available on the web.
  - It is used as example model in many TCAD simulators.

D.A. Antoniadis -http://www- mtl.mit.edu/researchgroups/Well/
GSS TCAD Simulation

GSS-TCAD

Commercial tool
GSS TCAD Simulation

- Error is near 100% for low bias.
ExtractCAD

- ExtractCAD tool can extract the following MOSFET model:
  - BSIM3 model
  - BSIM4 model
  - PSP Model (Version 102.1)
- ExtractCAD was used to extract a BSIM4 model card of the 25 nm device.
NGSpice circuit simulator

**NGSpice** is a mixed-level/mixed-signal circuit simulator. Its code is based on three open source software packages: Spice3f5, Cider1b1 and Xspice and is one of the simulators in the **gEDA project**.

- Verilog-A devices are compiled **statically** into the simulator and code must be present at **configure** time.
- ADMS templates and codemodels devices are grouped under a common directory.
- ADMS templates are used to translate Verilog-AMS code and fill with the appropriate code NGSpice model structure.

http://www.ngspice.org
Simulation in ngspice

- Extracted netlist is imported into ngspice.
### Needed tools

<table>
<thead>
<tr>
<th>Task</th>
<th>Tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>Write model equations</td>
<td>- Tool to represent ODE equations: CAS tools, numerical software.</td>
</tr>
<tr>
<td>Get data from device</td>
<td>- Simulated data: TCAD simulation software, measured data from wafer, instrumentation driving software</td>
</tr>
<tr>
<td>Extract parameters for model</td>
<td>- Parameter extraction tool</td>
</tr>
<tr>
<td>Simulate</td>
<td>- Circuit simulation tool</td>
</tr>
</tbody>
</table>
ADMS (Lemaitre)

Simulator-Specific ADMS-XML Interfaces

Verilog-A Model Code

Testing prior implementation

http://mot-adms.sourceforge.net

ADMS Parser

XML Internal Data

Code Generator

Other applications

Other ADMS-XML Tools

c-code for:
ADS, Eldo, Mica, PStar
hspice, Spectre, Titan
zspice, ngspice, QUCS

• Documentation
• Circuit Test Benches

Verilog-A

ADMS (Lemaitre)
SimCAS (Cyril Collineau)

- A different approach to electric simulation:
  - Simcas is a symbolic analog simulator. SimCAS uses symbolic equations to define components and solves the net system by using a “CAS” algorithm.
  - EKV Model (2.6 – long channel version) has been implemented into SimCAS.
SimCAS

http://sourceforge.net/projects/simcas/
SimCAS – EKV model

```plaintext
SimCAS version 0.3, Copyright (C) 2007 SimCAS
SimCAS comes with ABSOLUTELY NO WARRANTY.
This is free software, you are welcome to redistribute it.
```
Remarks about open tools

- Open source tools can be used for modeling but:
  - Accuracy and reliability must be verified.
  - Tools are not integrated.
    - Passing data between tools usually requires the intervention of conversion software (in this flow OpenOffice spreadsheet was used to exchange data).
Thank you!
WELCOME TO ROME
APRIL 8-9 2010