SILVACO

SPICE Modeling of Si, GaN and SiC Power FET Devices

2023 MOS-AK

Bogdan Tudor

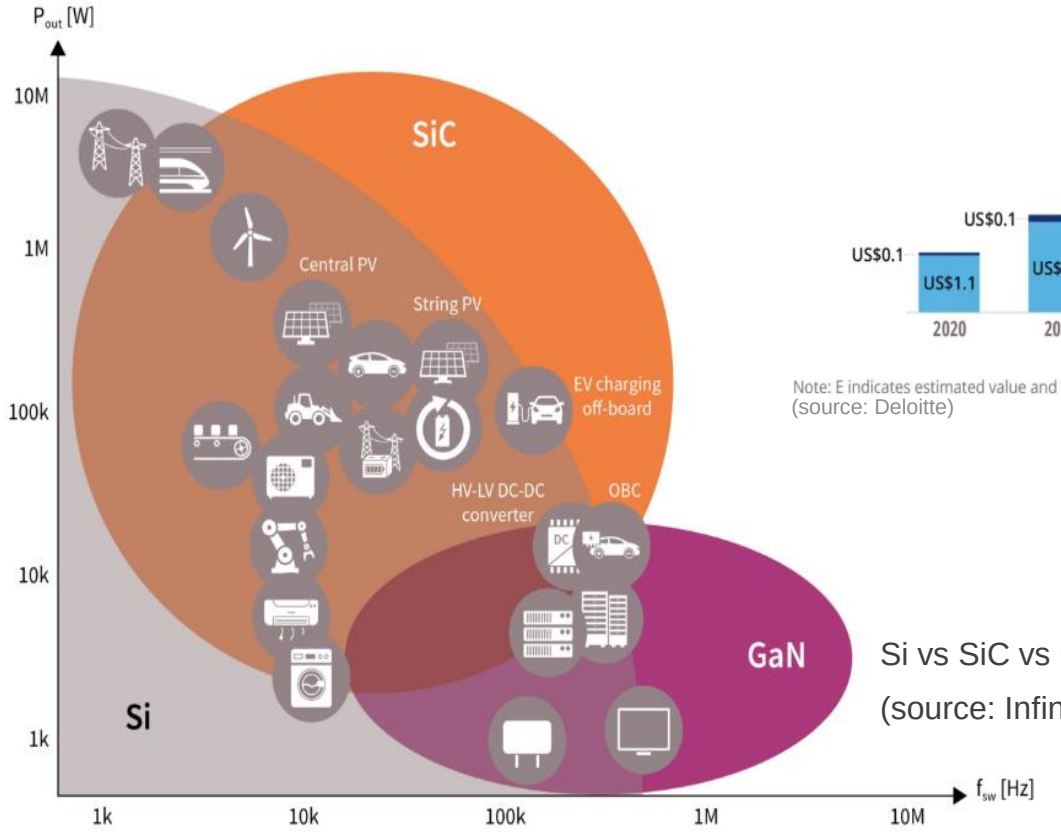
December 2023

Contents

- Introduction
- SPICE Models for GaN and SiC FET
- Modeling Power FET-specific Capacitances
- General Power FET Model Extraction Methodology
- Advantages of TCAD-based SPICE Modeling
- SPICE Modeling Examples of Si, GaN and SiC Devices
- Conclusions

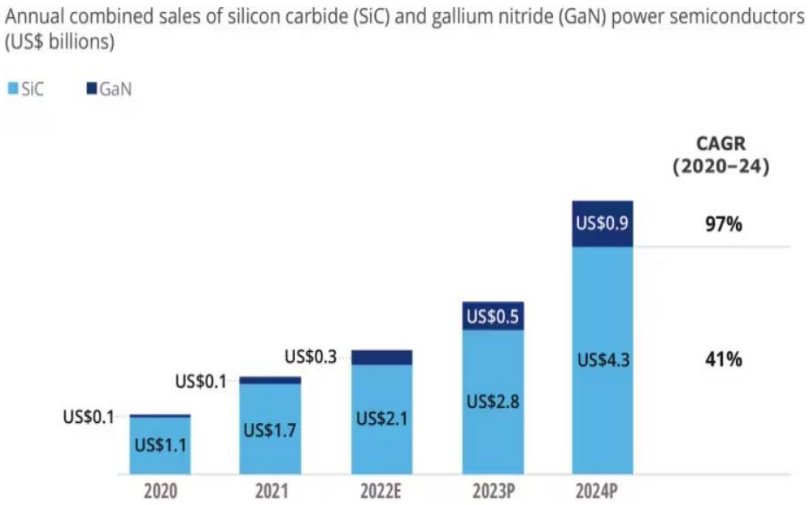
Introduction: Power Device Technologies

- Power applications: Si replaced by Wide Bandgap Semiconductors
- WBG advantages
  - Lower resistance
  - Higher frequency
  - Higher power
  - Higher operating temperature



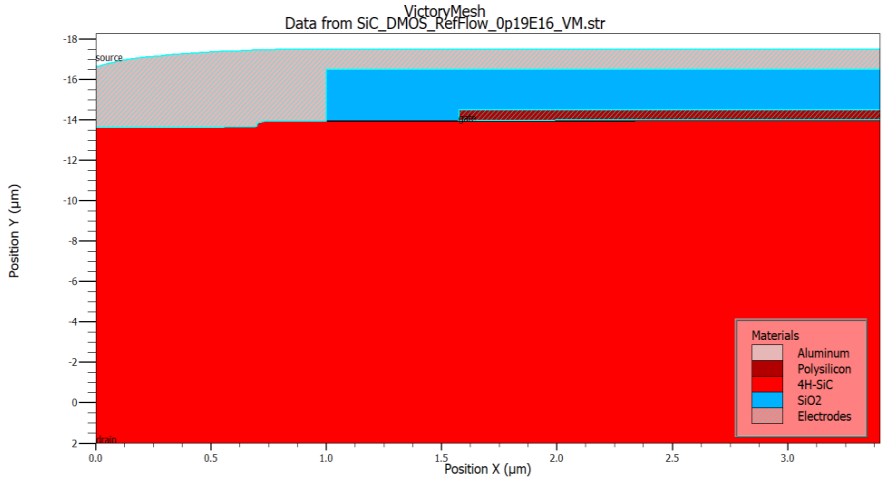
Si vs SiC vs GaN

(source: Infineon)

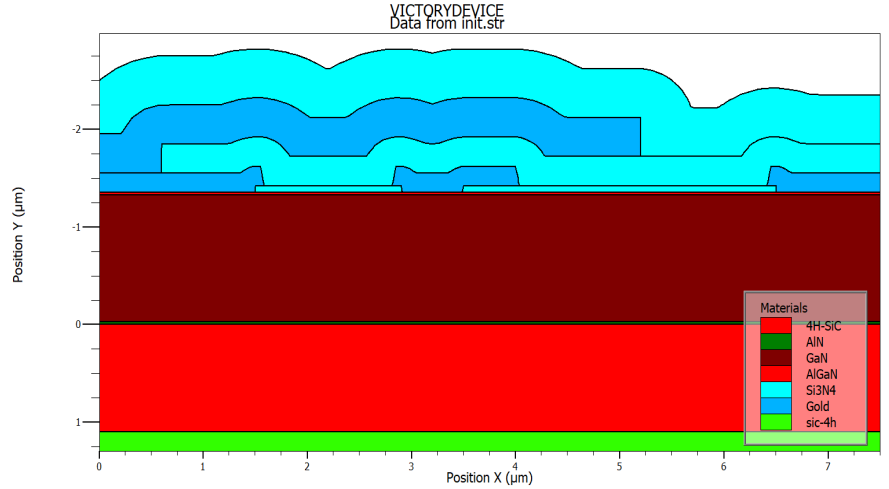


SiC and GaN

- SiC
  - Low thermal conductivity
  - High power
  - Main market: automotive/EV



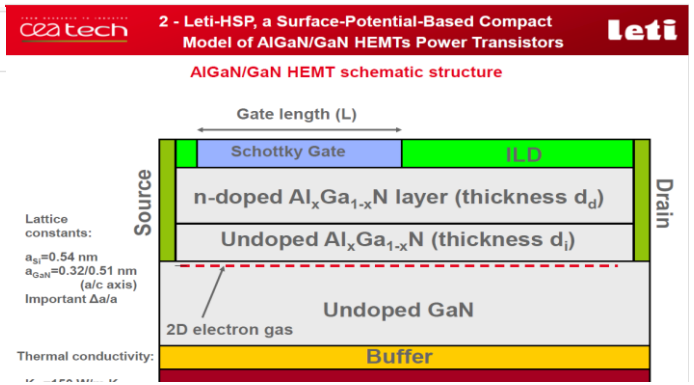
- GaN
  - RF applications
  - Switch-mode power supplies
  - Main market: consumer electronics chargers



SILVACO in Power FET Compact Model Development

CEA-Leti and Silvaco Look to Develop Innovative SPICE Models

Santa Clara, California



Left to Right Iliya Pesic, Silvaco Chairman, Emmanuel Sabonnadière, CEA-Leti CEO sign collaboration agreement to develop innovative

Si2 Approves Two IC Design Simulation Standards for Fast-Growing Gallium Nitride Market

Compact Model Coalition Models Expected to Reduce Costs, Speed Time-to-Market

For Immediate Release

AUSTIN–The Silicon Integration Initiative's (Si2) Compact Model Coalition market for gallium nitride semiconductors.

The approved standards are the 12th and 13th models currently funded (Simulation Program with Integrated Circuit Emphasis) models for IC des

John Ellis, president and CEO, said gallium nitride devices are used in ma broadband wireless systems, and automotive. "Although it's currently a s

To reduce research and developments costs and increase simulation acc models. Si2 is a research and development joint venture focused on IC d incorporated into design tools widely used by the semiconductor industr leading universities and national laboratories. The CMC directs and funds

Dr. Ana Villamor, technology and market analyst at Yole Développement We project an explosion of this market with 79% CAGR between 2017 an

Silicon Integration Initiative Targets New Silicon Carbide Standard SPICE Model

April 27, 2021 / in Compact Model, Frontpage / by Terry Berke

The Si2 Compact Model Coalition has voted to fund and standardize a SPICE model for silicon carbide-based metal-on-silicon field-effect transistors. Featuring high efficiency and fast operation with low switching losses, silicon carbide-based metal-on-silicon-field effect transistors are popular in high-growth semiconductor applications such as photovoltaic inverters and converters, industrial motor drives, electric vehicle powertrain and EV charging, and power supply and distribution.

Peter Lee

A CMC working group will oversee the model development as part of advancing Si2's mission to reduce interoperability costs, said Peter Lee, CMC chair. Participating companies include Analog Devices, Cadence Design Systems, Infineon, Qualcomm, Siemens EDA, Silvaco and Synopsys. The decision to launch the working group came after the CMC evaluated the model's ROI for members and interest by the industry at large. "I'd encourage companies with a stake in SiC devices to join this effort and help guide selection of the model which best represents their intended use," advised Lee. "They can benefit from both cost reduction that comes from shared model support and a standardized and qualified model that has ongoing bug fixes and requested feature enhancements from many like-minded companies."

Colin Shaw

"Next Generation SiC MOSFETS has many features that make them suitable, and even superior to legacy silicon solutions, for several high voltage applications. While the devices can handle high-temperature and voltage, its minimal ON-resistance allows smaller packages and better energy savings than comparable silicon devices," stated Colin Shaw from Silvaco, the working group chair.

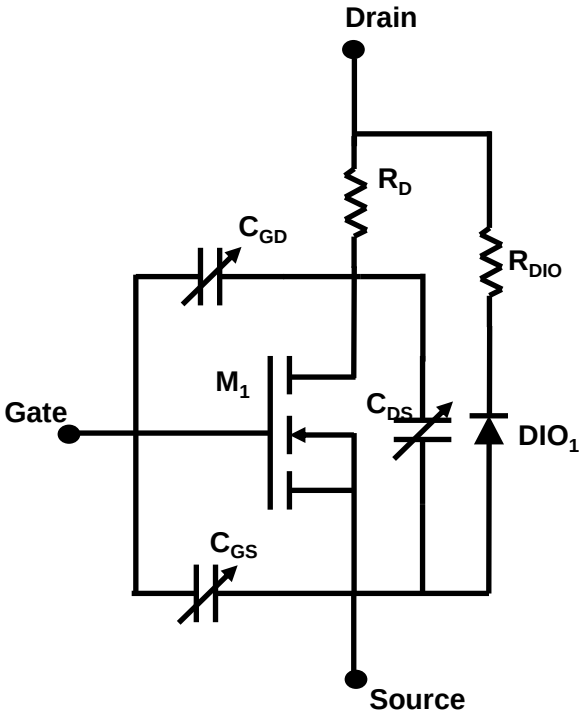
GaN FET: 2 CMC Standard Models

- GaN FET models in SmartSpice:
  - ASM (CMC Standard, SmartSpice Level 90)
  - MVSG (CMC Standard, SmartSpice Level 91)
  - LETI-HSP (SmartSpice Level 278)
- Model extraction initially based on Verilog-A, currently using SmartSpice built-in models
- GaN FET Model Extraction Examples:
  - MVSG: https://silvaco.com/examples/utmost4/section1/example18/index.html
  - ASM: https://silvaco.com/examples/utmost4/section1/example23/index.html
- GaN FET Model Extraction Webinars:
  - https://silvaco.com/webinar/tcad-based-model-extraction-flow-for-gan-hemt-devices-part-1-2/
  - https://silvaco.com/webinar/tcad-based-model-extraction-flow-for-gan-hemt-devices-part-2-2/

Template Categories	Quick-Start Templates
MOSFET	BSIM3v3, BSIM4, BSIM-BULK, HiSIM2, HiSIM_HV2, PSP
GaN HEMT	ASM, MVSG
TFT	RPI poly-Si TFT, RPI amorphous Si TFT
BJT	Gummel-Poon, VBIC, Mextram
IGBT	HiSIM_IGBT
Diode	Diode Level 1

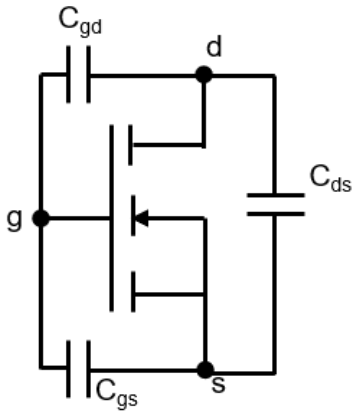
SiC MOSFET SPICE Model Solutions

- CMC SiC compact model standardization currently ongoing
  - Chaired by Silvaco
- Empirical Macromodels
  - Based on behavioral circuit elements (controlled sources, behavioral or PWL capacitors and resistors, etc.)
  - Limited accuracy and predictability
- Use a GaN HEMT model, such as ASM or MVSG
  - Device similarities resulting in a good accuracy and predictability
  - Accuracy can be further improved by using a macromodel
- MOSFET-based macromodel
  - Core FET (e.g., BSIM4)
  - Recovery diode
  - Additional resistors and bias dependent capacitors
  - Reasonable accuracy and predictability

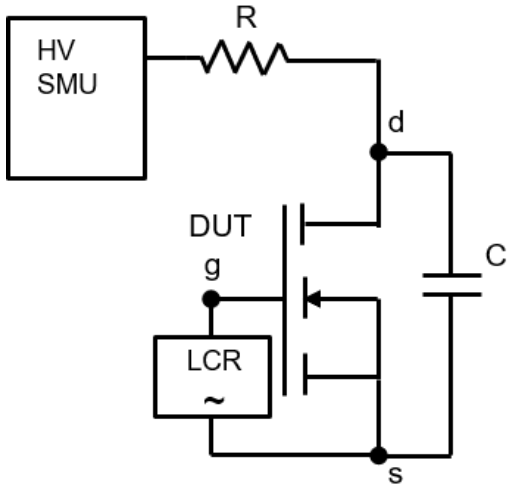


Specific Power MOSFET Capacitances

- 3-terminal MOSFET
  - The b and s nodes are always connected together
- Main bias sweep is Vds



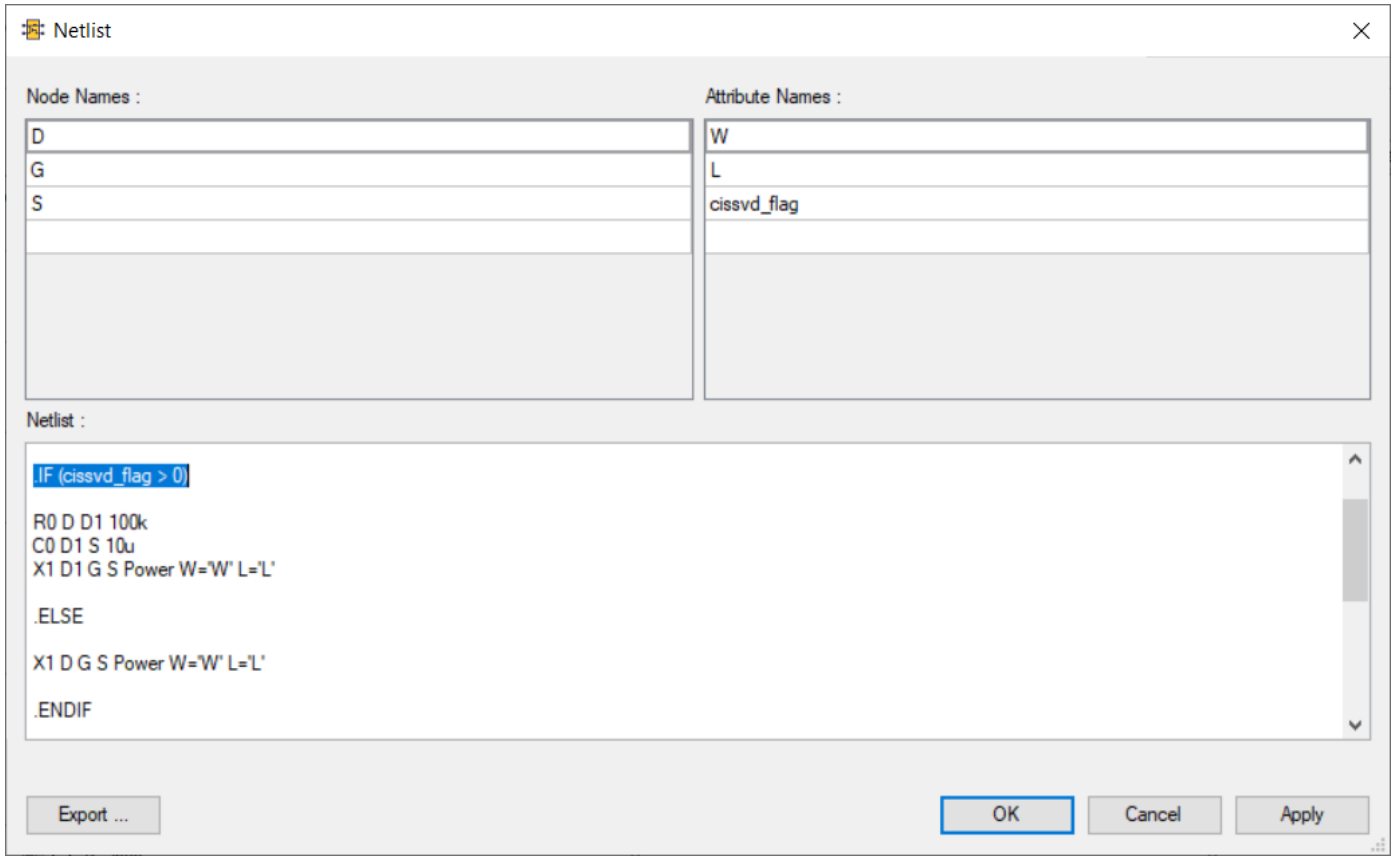
- Typical capacitances to measure:
  - Crss = Cgd
  - Coss = Cgd + Cds
  - Ciss = Cgs + Cgd



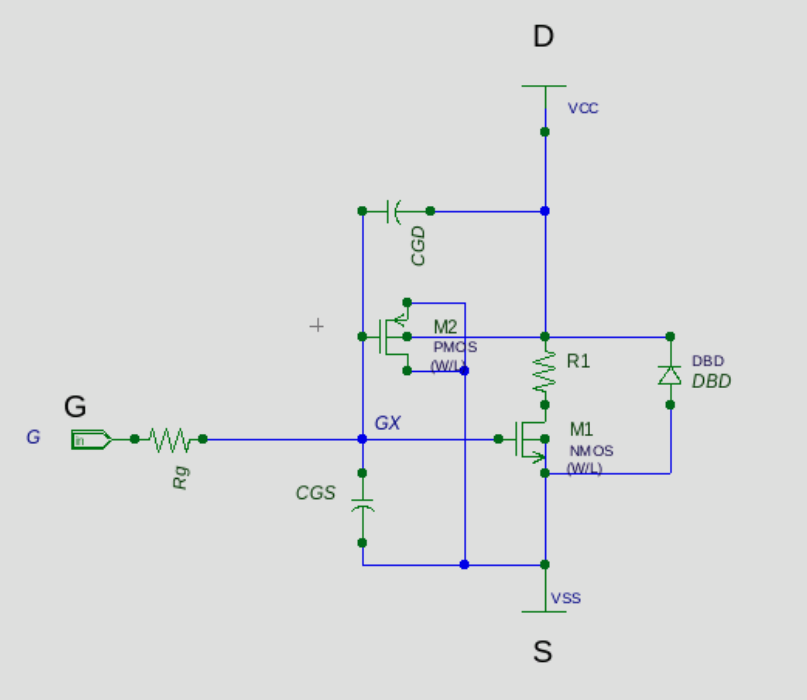
- Ciss requires a special test circuit
  - Need to use a corresponding netlist

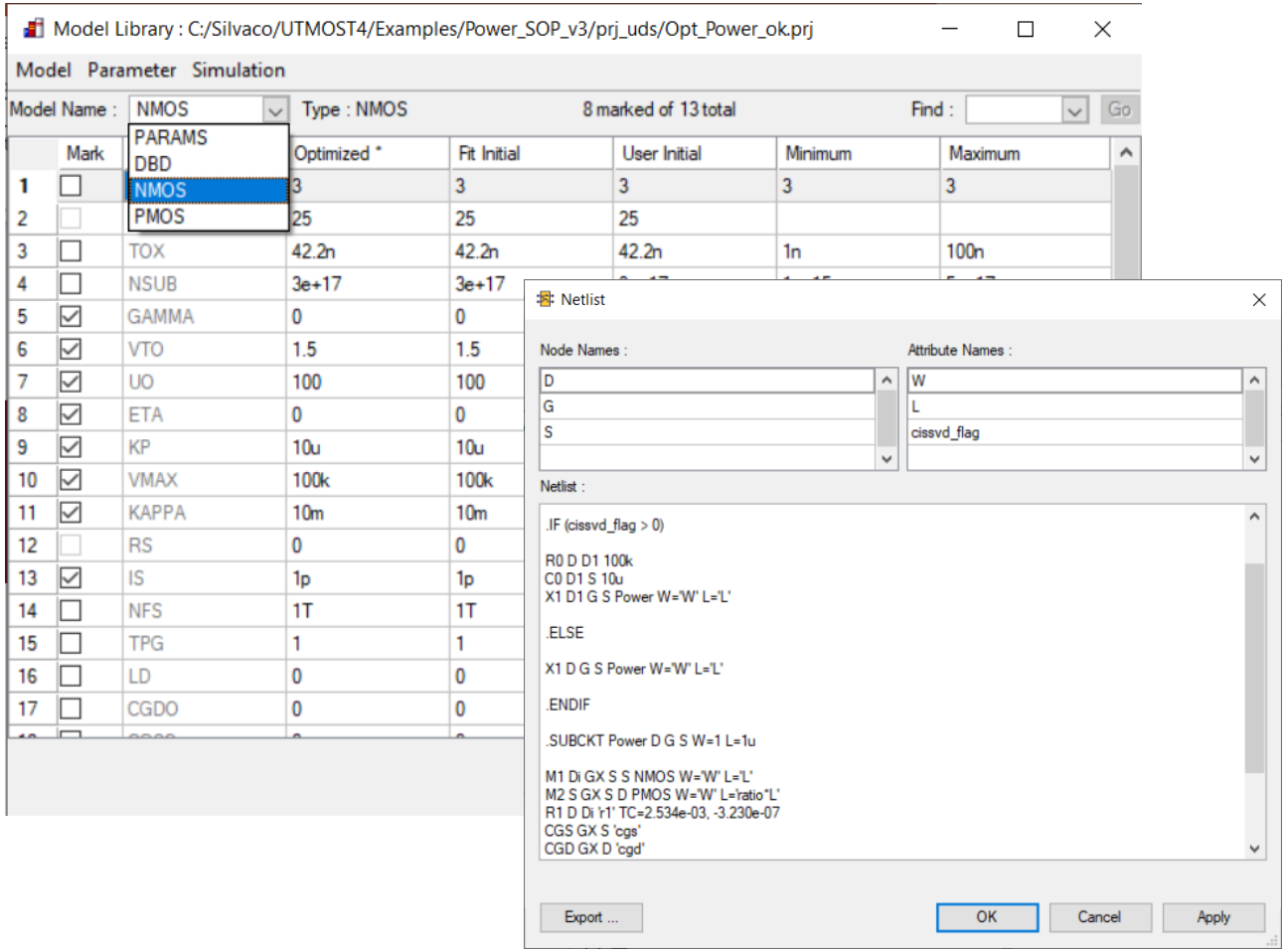
Single Netlist Supporting All Data

- Use attributes to help distinguish the netlist
  - The Ciss data set includes a special attribute set to 1
  - Conditional netlist based on the attribute value



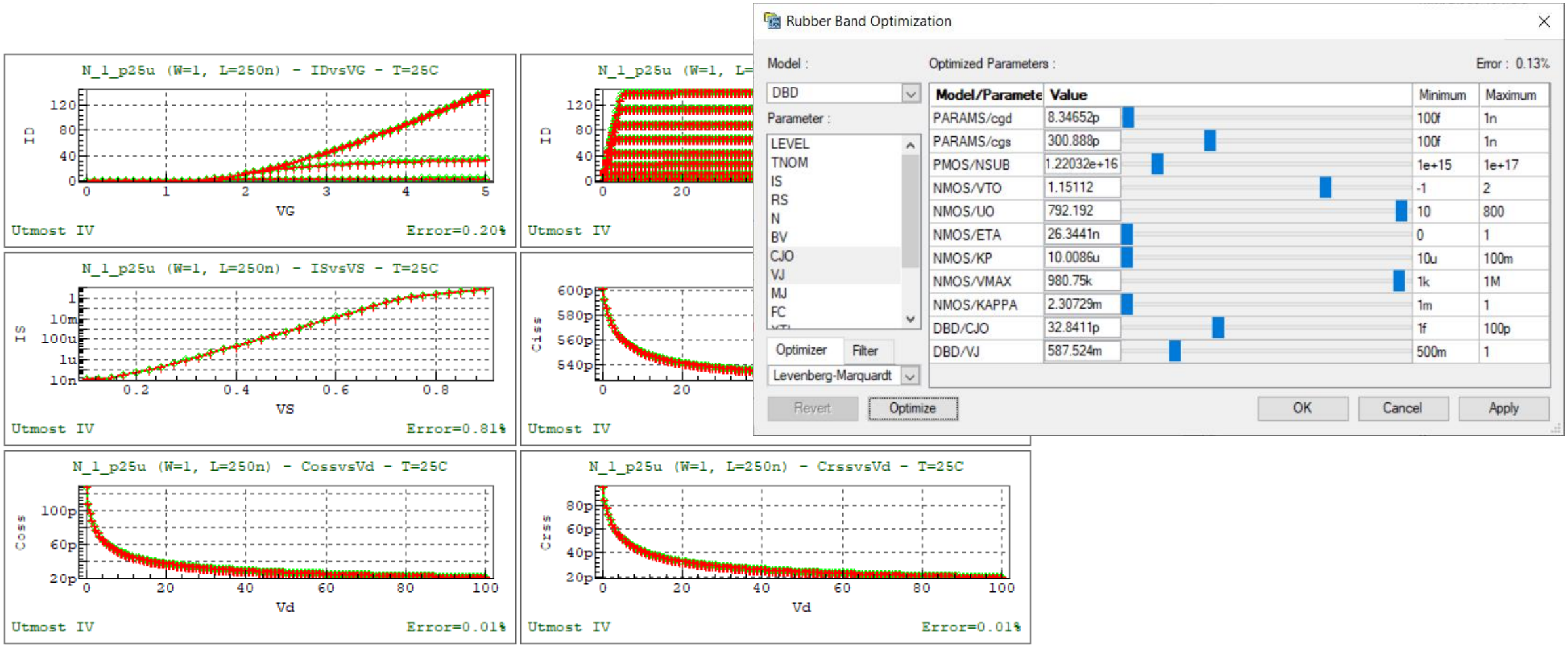
Power MOSFET Macromodel Example





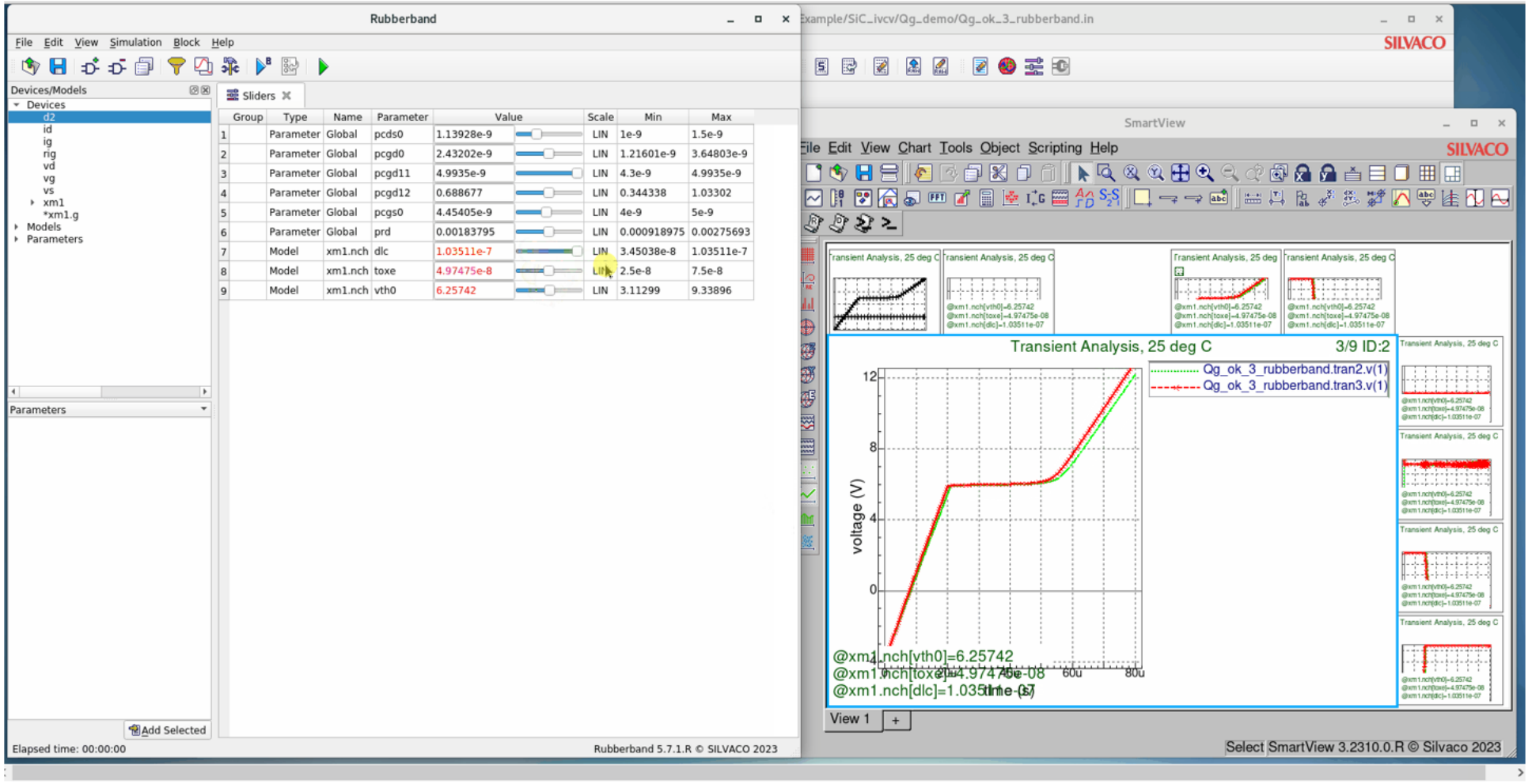
Macromodel I-V and C-V Parameter Extraction

- Global I-V and C-V parameter optimization using Utmost IV



Dynamic Characterization

- Using SmartSpice Rubberband to tune Qg



TCAD-to-SPICE DTCO Flow

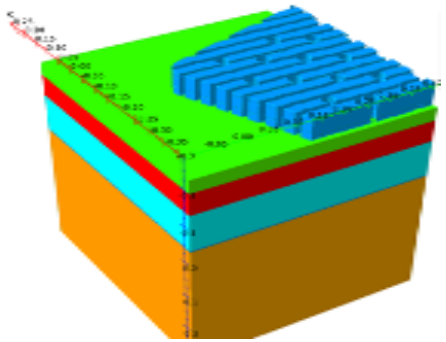
Process Simulation
Victory Process
Victory Mesh

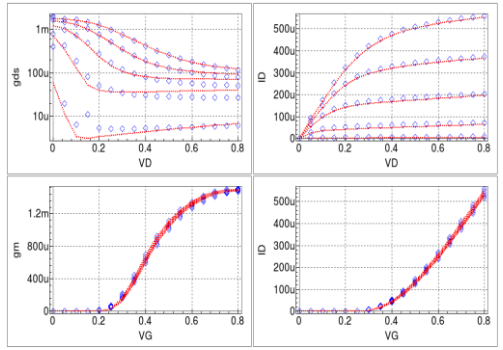
Device Simulation
Victory Device
Victory Atomistic

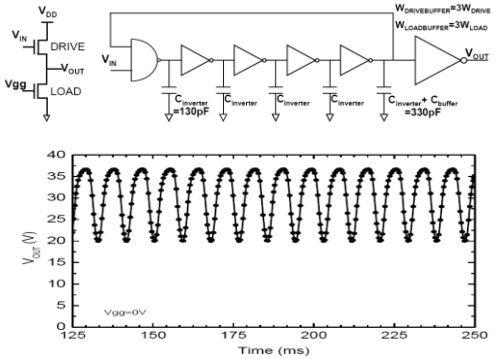
SPICE Modeling
Utmost IV

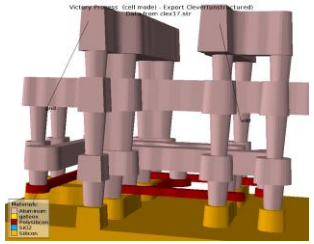
Circuit Simulation
SmartSpice

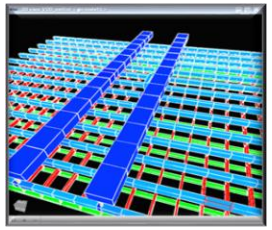
Parasitics Extraction
Victory RCx Pro
Jivaro





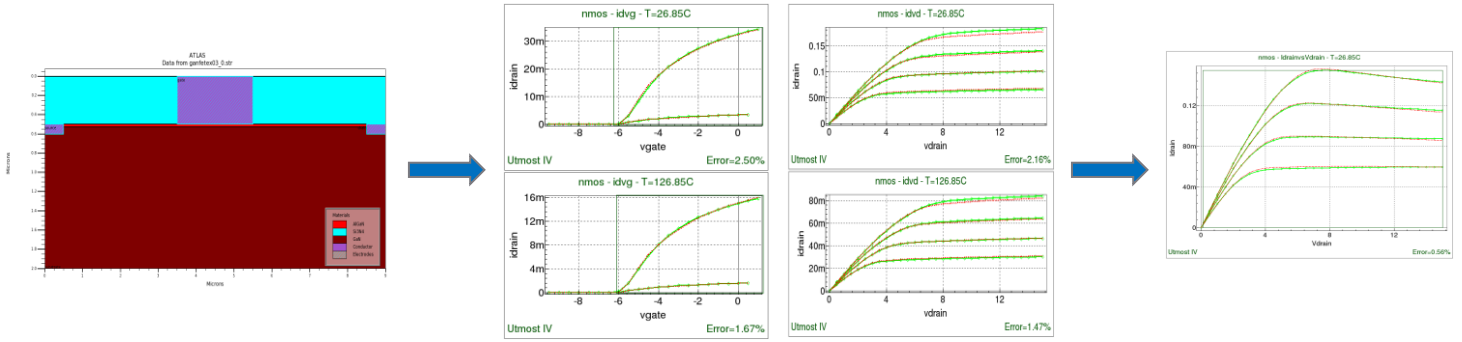




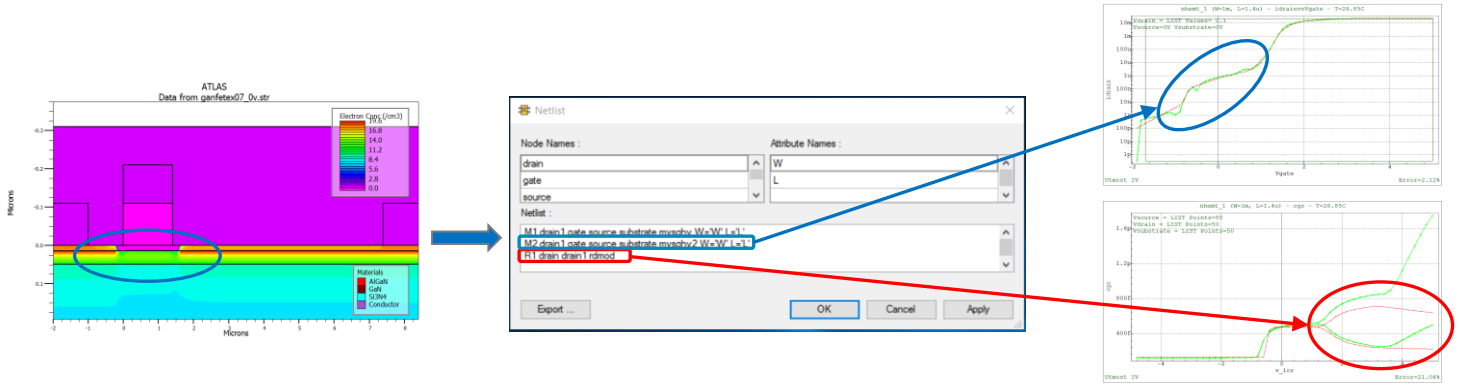


Advantages of TCAD-based SPICE Modeling

- TCAD simulations allows separation of physical effects (e.g., self-heating)

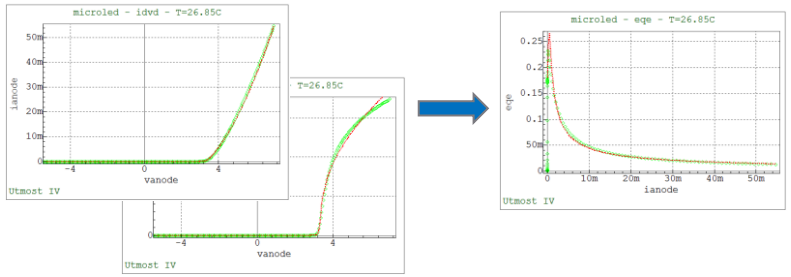


- Model development: identifying specific device effects (or parasitics) and addressing them as SPICE elements



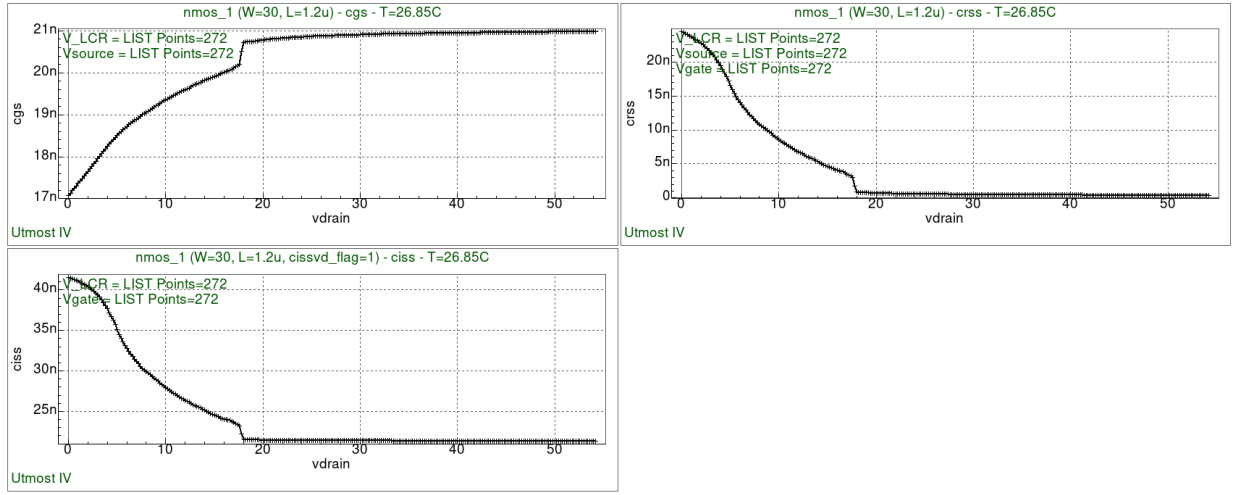
Advantages of TCAD-based SPICE Modeling (2)

- Simulate and calculate critical circuit design-related parameters, that are not easily measurable in real life, e.g.:
  - Probing optical outputs and calculating the Quantum Efficiency of MicroLEDs
  - Probing internal voltage nodes and thermal nodes



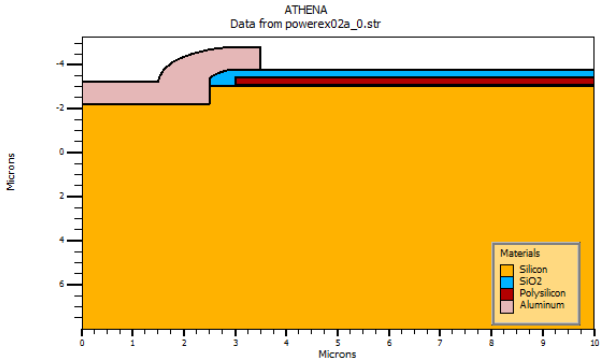
VO_2 thermal-driven memristor

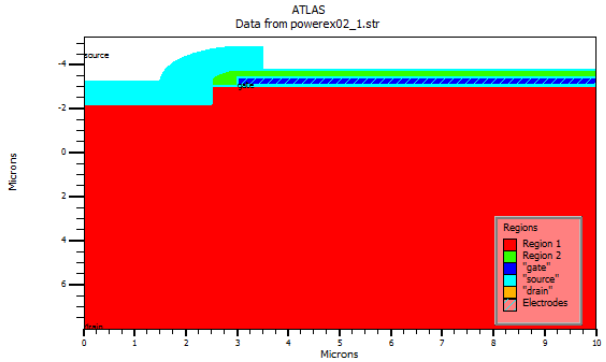
- Combining output components to calculate power device-specific quantities
  - Calculate Ciss(Vds) from Cgs(Vds) and Cgd(Vds)

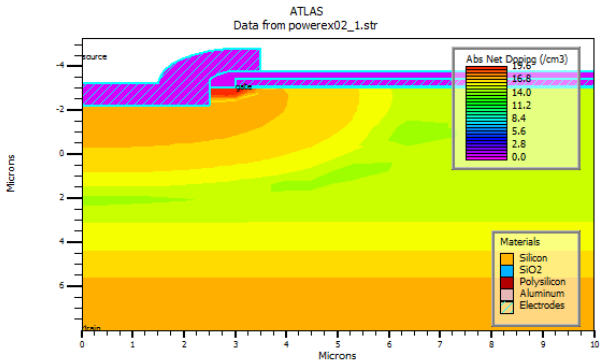


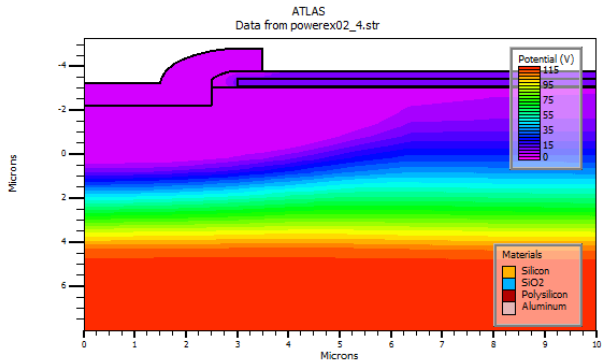
TCAD Si VDMOS Example

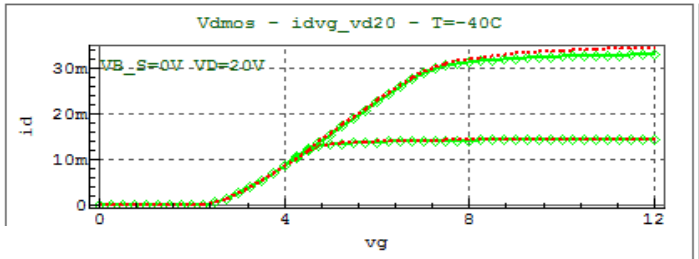
- HiSIM_HV2 SPICE Model
- https://silvaco.com/examples/utmost4/section1/example14/index.html

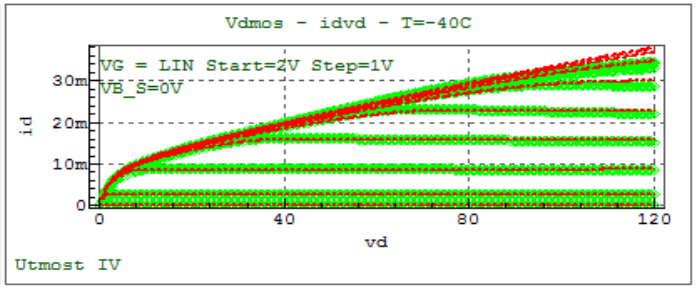


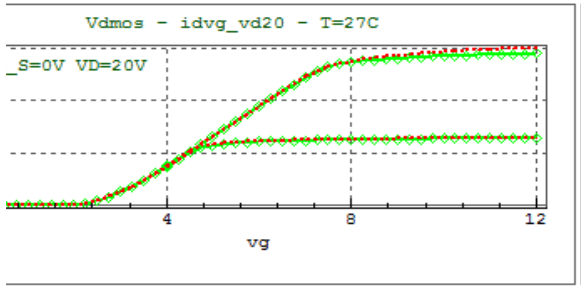


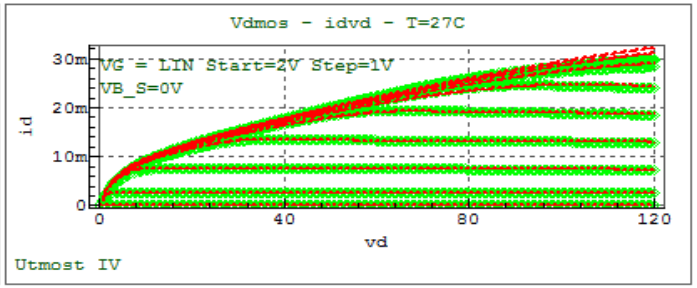


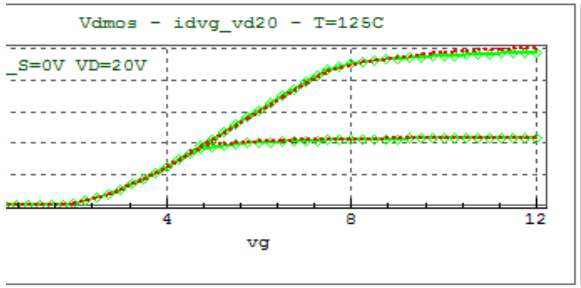


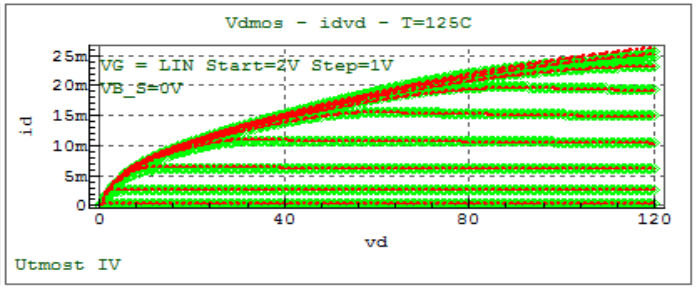






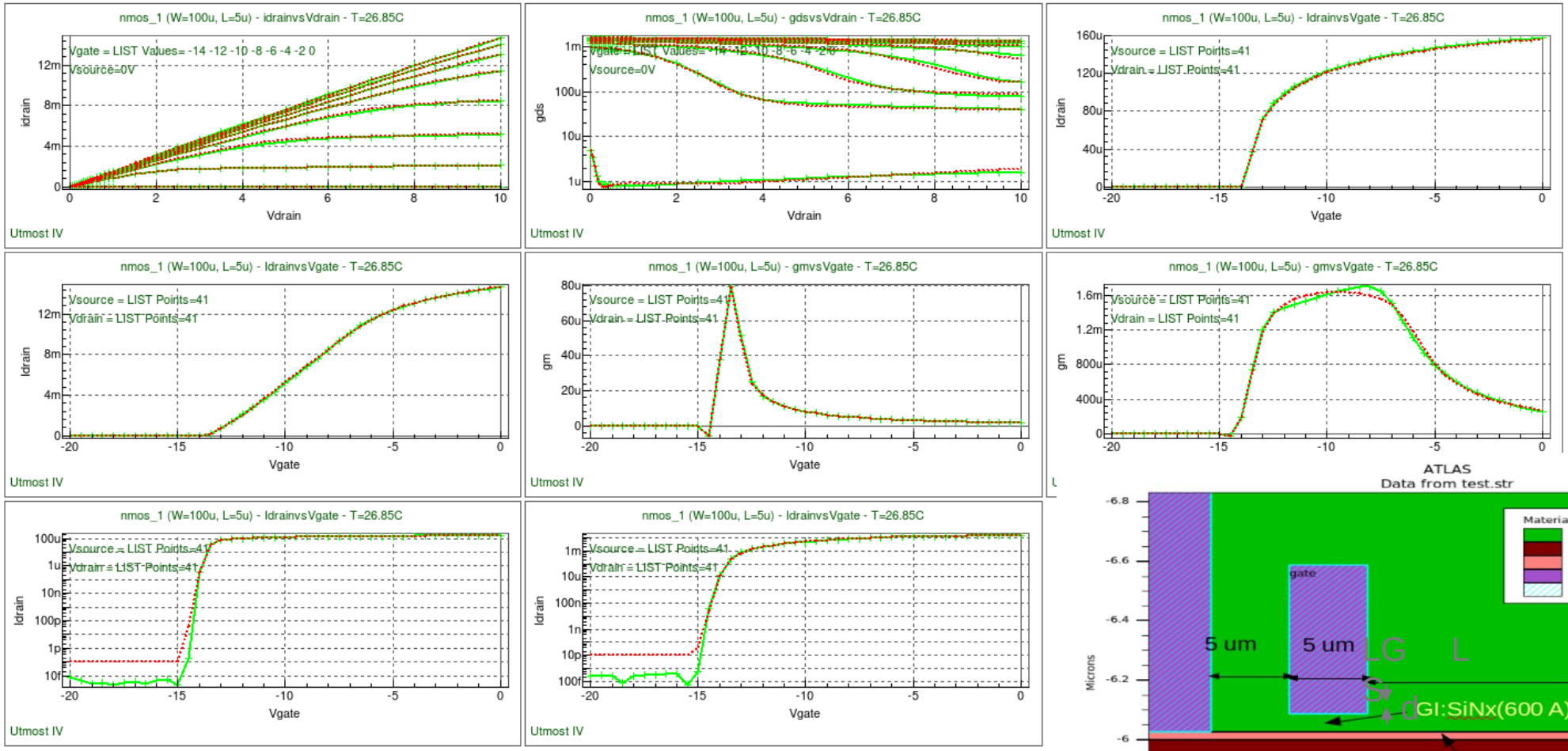






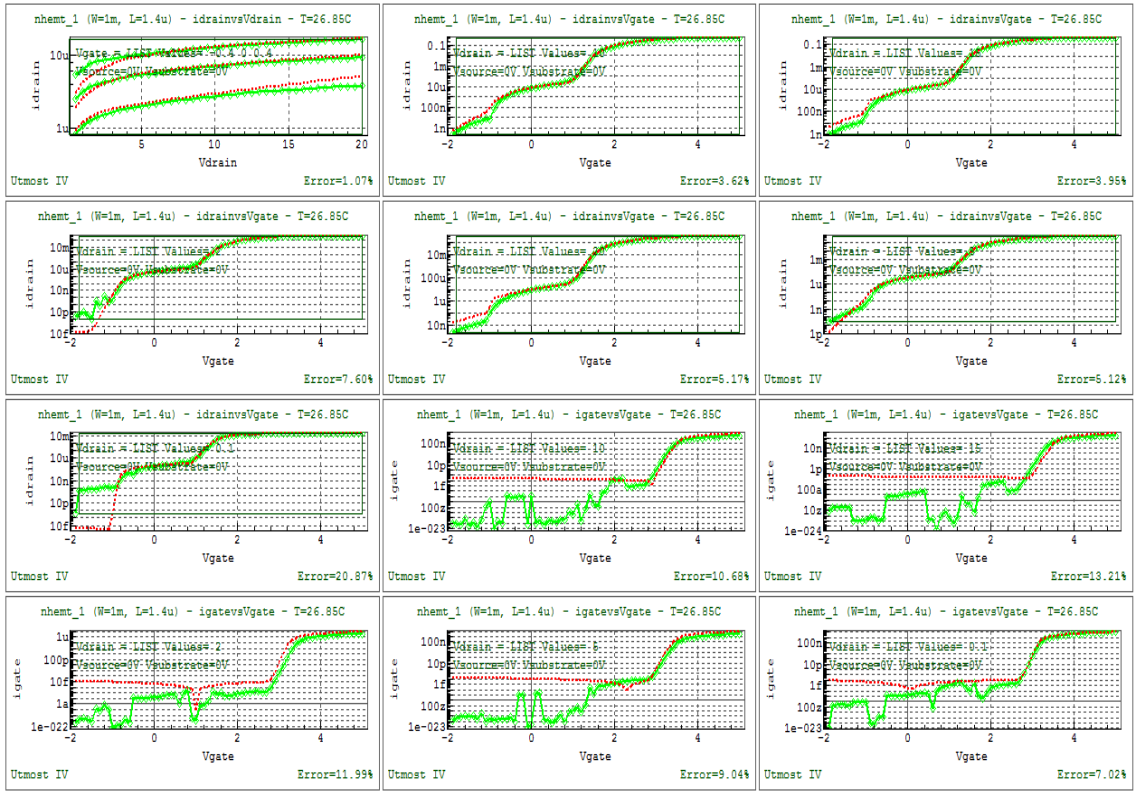
TCAD Generic GaN HEMT Device Structure

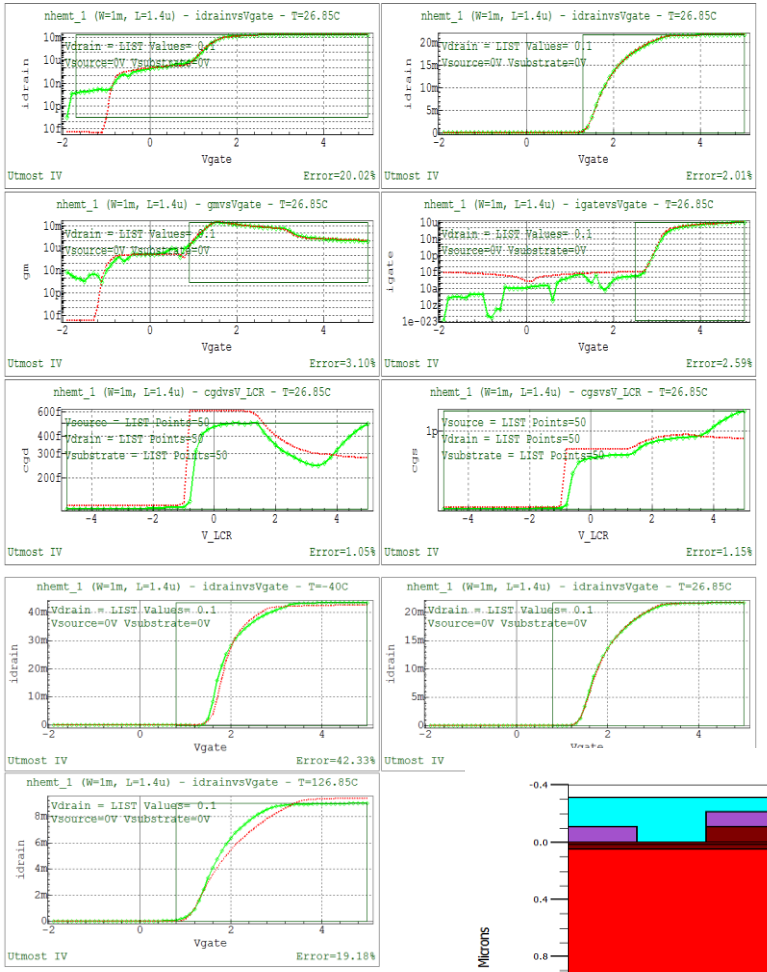
- SPICE ASM Compact Model

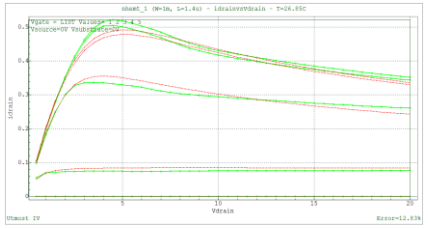


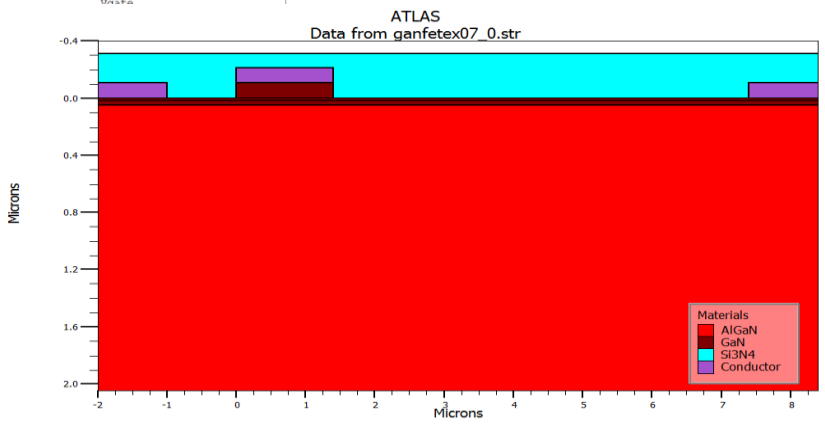
GaN HEMT Device Structure with P-type Gate

- TCAD structure calibrated based on published data
- P-type doped GaN gate resulting in Vth > 0 (“normally-off” device)



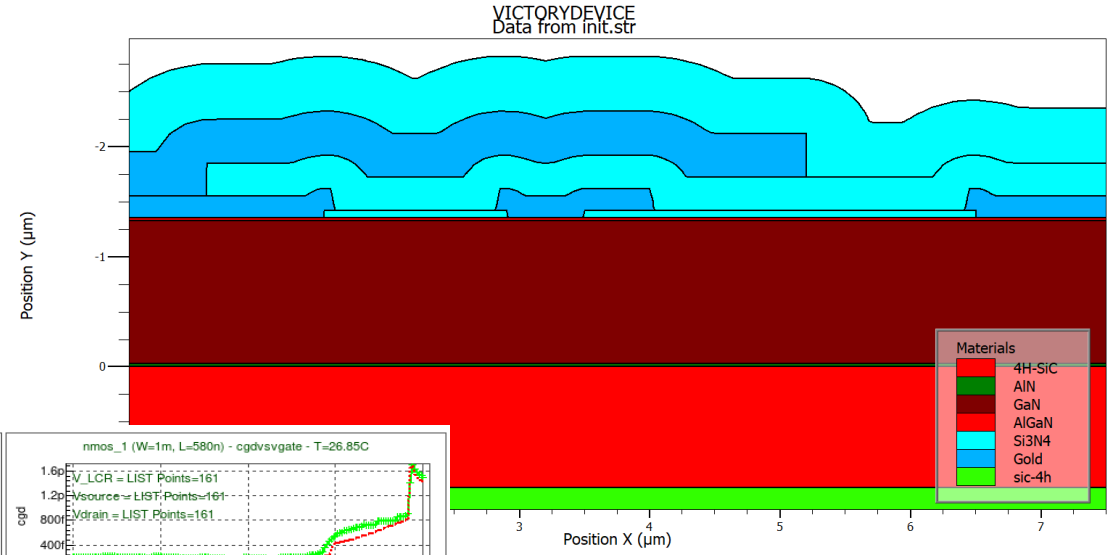


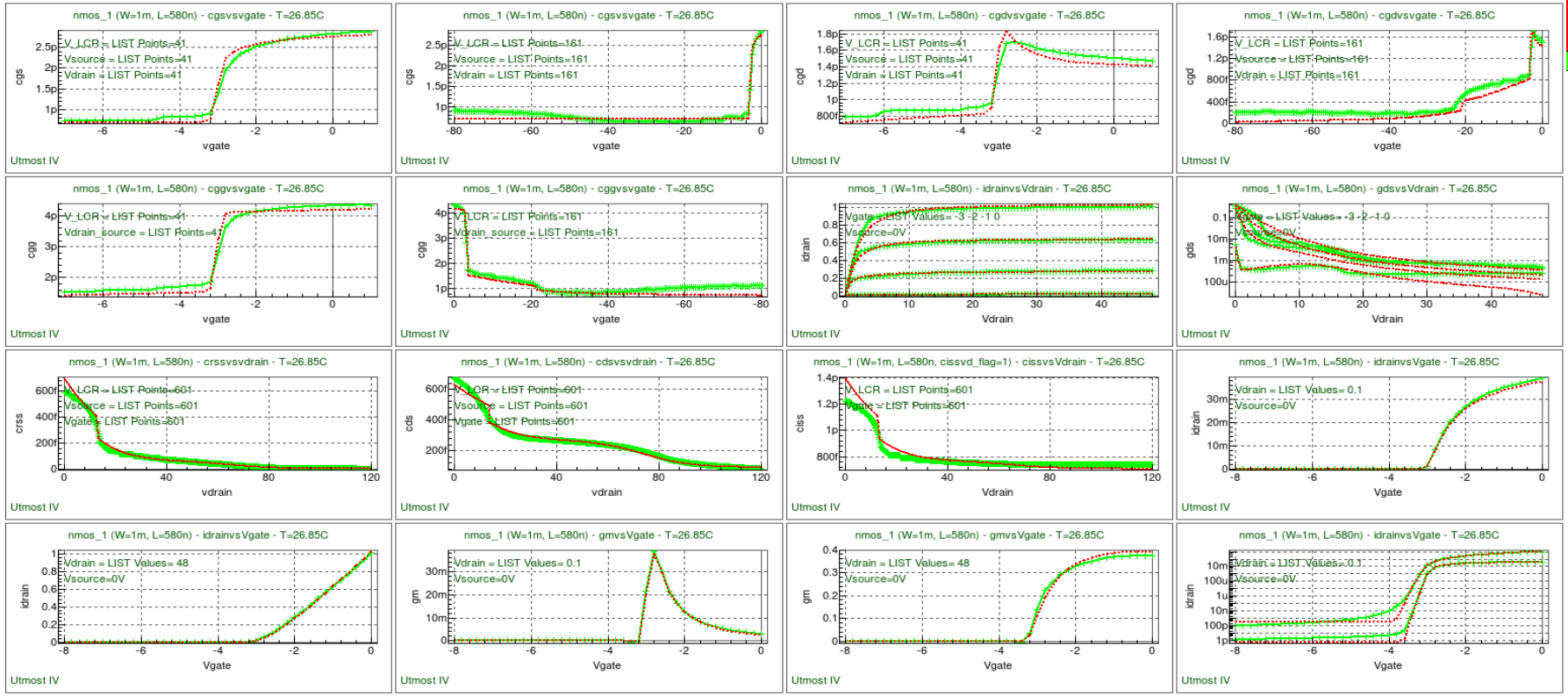




GaN HEMT Device With Field Plates

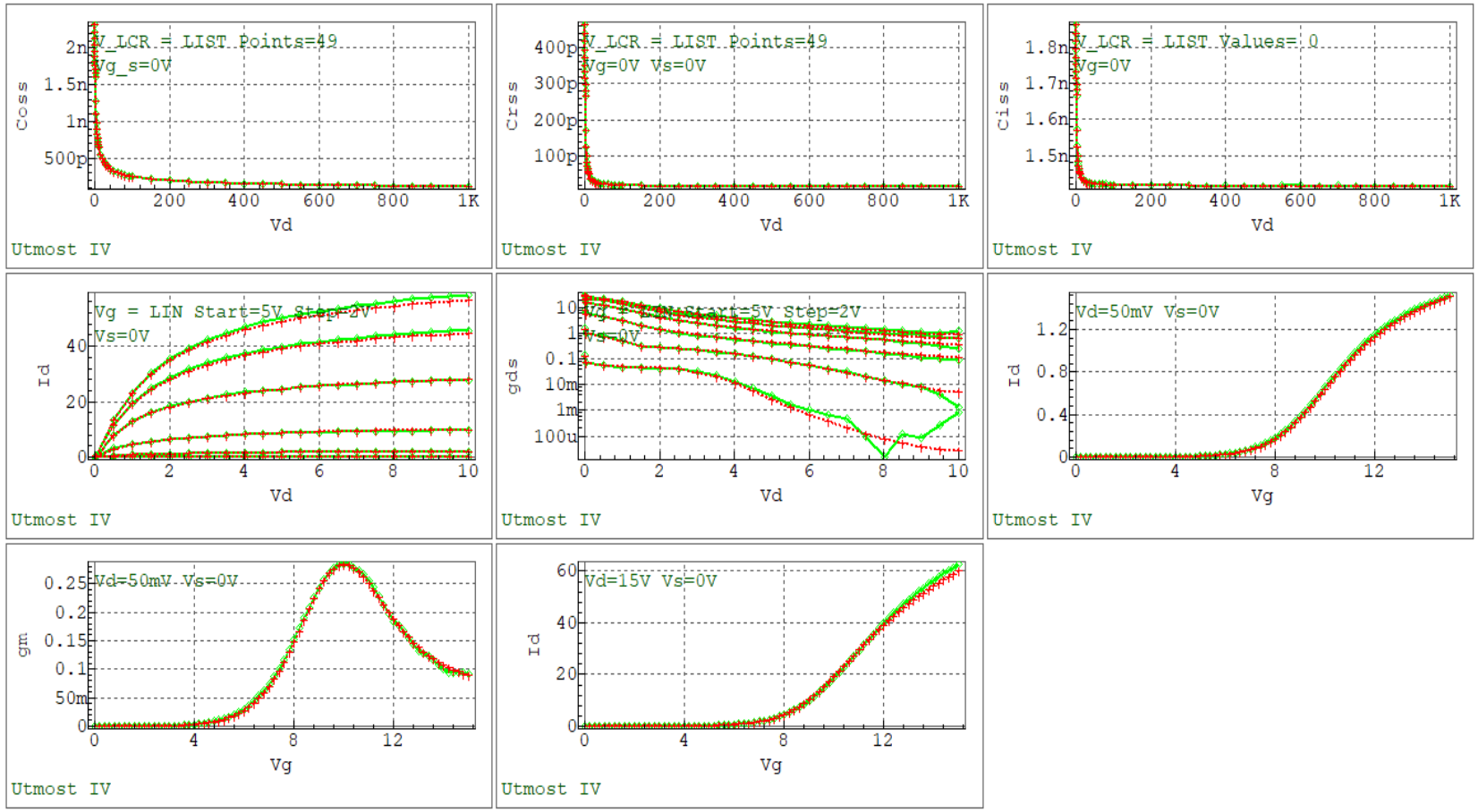
- TCAD device structure calibrated based on published data
- Gate and source field plates
- SPICE MVSG Compact Model





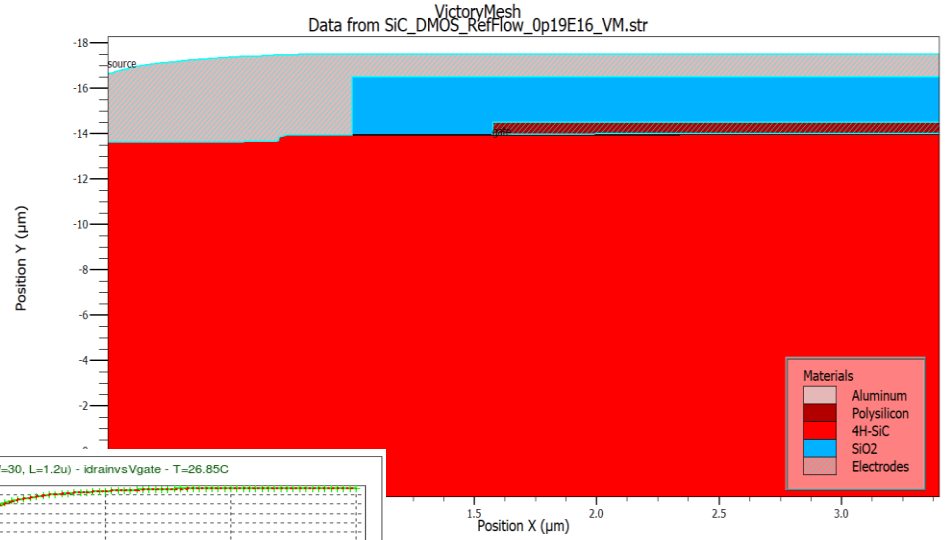
SiC Macromodel Example

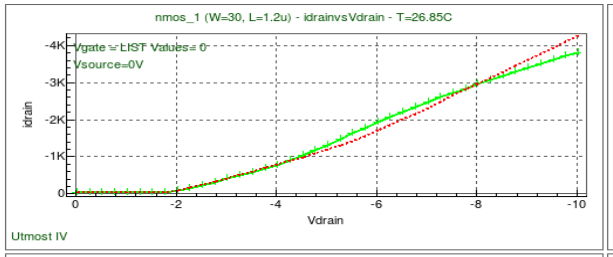
- Based on generic data:
  - C-V: Ciss(Vds), Crss(Vdg), Coss(Vds)
  - I-V: Id(Vgs), Id(Vds)

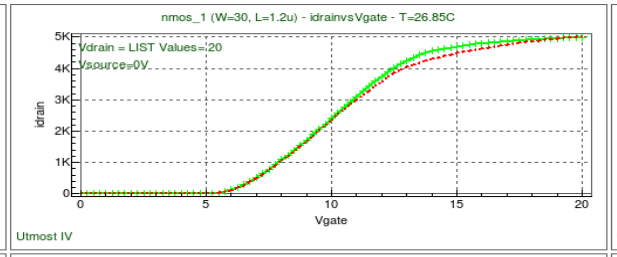


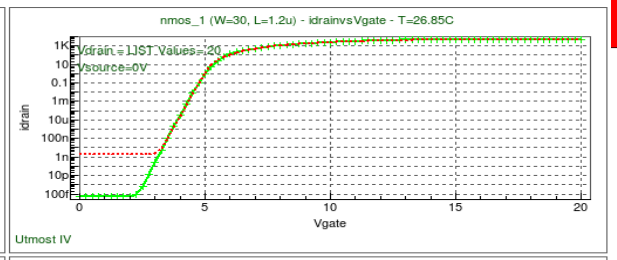
SiC TCAD Device

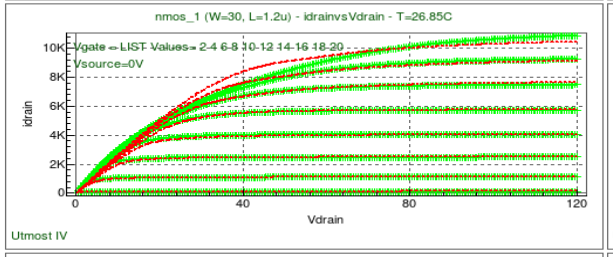
- TCAD device created using Victory Process, Victory Mesh and Victory Device
- SPICE BSIM4-based Macromodel

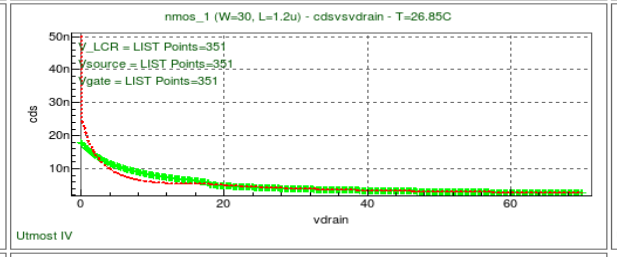


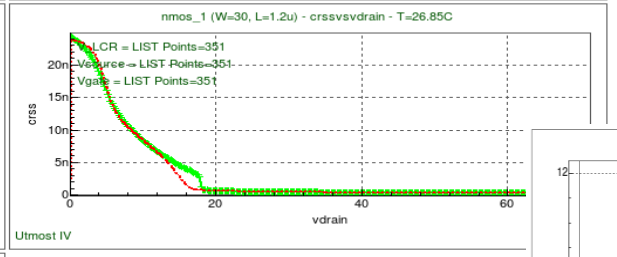


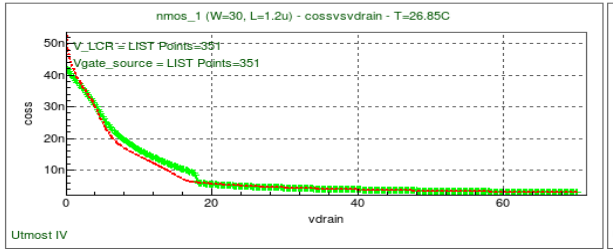


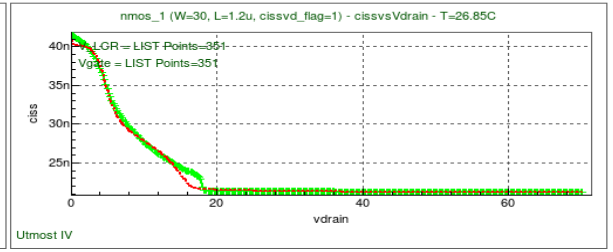


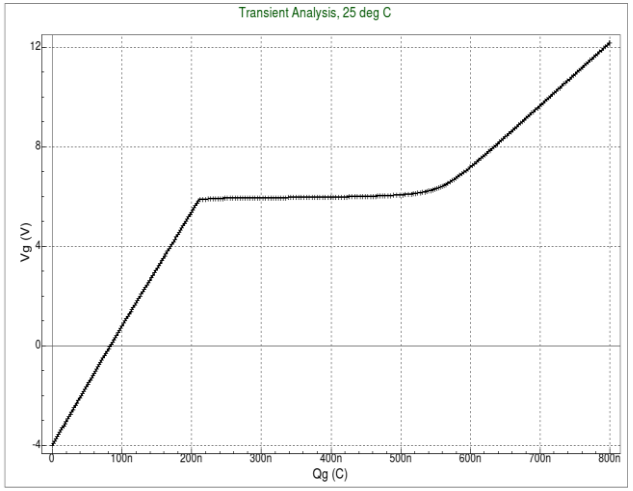












Conclusions

- Review of power device technologies
- Exploration of power FET SPICE models
- Power FET specific capacitance support using a single netlist
- Comprehensive modeling methodology of Power FETs, including dynamic characterization
- TCAD-based SPICE modeling carries unique advantages
- SPICE modeling capabilities addressing various power FET technologies: Si, GaN, SiC
- Silvaco provides a full set of software solutions, from TCAD to SPICE, to address the specific requirements of SPICE modeling for power devices


Thank you
SILVACO