

BSIMProPlus Device Modeling Suite

The BSIMProPlus Device Modeling Suite provides device characterization and modeling capabilities for MOSFET, BJT, SOI, TFT, HVMOS/LDMOS, and many other device types. BSIMProPlus device modeling is the critical link between your design and the process that you have chosen, regardless of the fabrication source. The tools provide device modeling capabilities for baseband, RF, noise, reliability and etc.

BENEFITS

- ◆ Simulations that match silicon
 - BSIMProPlus device modeling suite provides the key link between circuit simulation and your silicon
- ◆ Assured equivalent results between BSIMProPlus modeling tools and major commercial SPICE simulation tools, including Spectre, HSPICE, and Eldo
- ◆ Model support for almost all mainstream, advanced and specialized processes
- ◆ Fast turn-around for your process characterization
 - Includes best-in-class modeling capabilities
- ◆ Silicon-accurate simulation
 - Links process characterization to IC design verification
- ◆ Measurement of your silicon characteristics
 - DC, RF, reliability, and noise characterization capabilities
- ◆ Peace of mind
 - Used by almost every foundry and major IDMs all over the world
 - Long history with track records since 1993

- The only total solution provider in the device modeling field

FEATURES

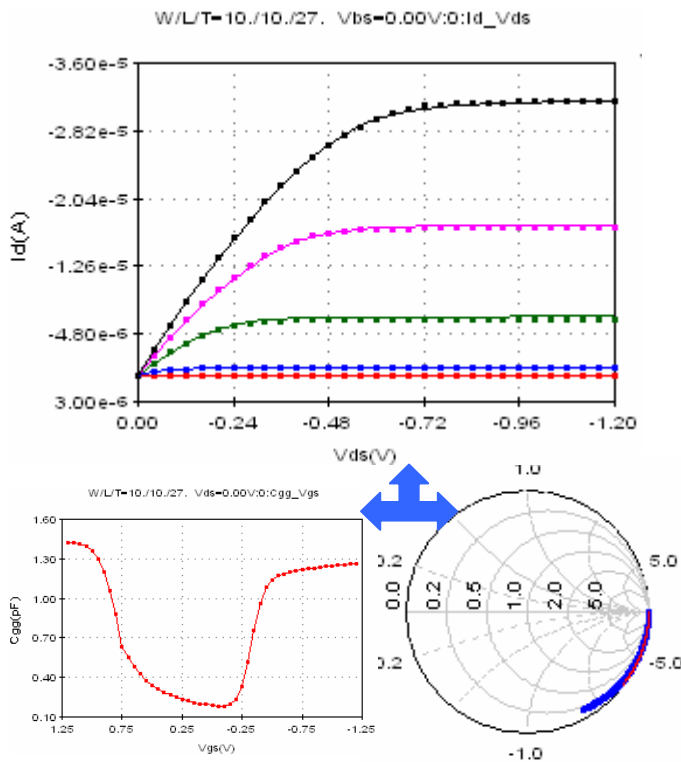
Wide Varieties of Model Types—MOSFET, BJT, SOI, TFT, HVMOS/LDMOS, Passive devices, Etc.

BSIMProPlus device modeling suite supports a comprehensive set of active and passive devices for measurement, model extraction and validation. The model support ranges from the most advanced models, such as PSP, HiSIM2, to the most widely used ones, including industry standard BSIM-based models and MOS9, MOS11, EKV for MOSFET/SOI, GP-BJT/Mextram/ VBIC/HiCUM for BJT/HBT, and TFT models, and to macro (sub-circuit) models as well as many proprietary models including the HVMOS, LDMOS & AgeMOS of Cadence. For each model, BSIMProPlus device modeling suite is capable of controlling test equipment to measure devices under test (DUT), extracting SPICE-level parameters for the specific model type, and validating the extracted model through simulation of the device using major commercially available SPICE simulators, e.g. Spectre, HSPICE and Eldo.

Baseband to RF Data Measurement, Model Extraction, and Model Validation

The BSIMProPlus device modeling suite covers designs that span the frequency range. Well-proven built-in extraction routines and highly efficient optimization algorithms enable the fastest and most accurate SPICE model extractions. Manual tuning of model parameters can be easily carried out in the equalizer during the optimization

process. Sensitivity analysis ranks the sensitivities of all selected parameters, through which the user can easily find out the importance of each parameter for a specific target. Auto-binning helps the user to generate binned models easily. Various corner-model generation flows are provided to model worse cases based on the typical model card. Model re-targeting flow enables the user to quickly re-target the typical model based on wafer-case data and typical device spec. In high-frequency applications, advanced de-embedded techniques are employed to remove extraneous parasitic introduced by the test structure, connections, and wiring. RF model templates and built-in extraction routines are provided to help quickly generate RF models for CMOS, on-chip inductors, etc. Various validation tools are embedded in BSIMProPlus for complete SPICE model validations, checking the model continuity, convergence, physical trend and rules, etc. Built-in circuit verification function helps the user to easily verify the model in circuit level against the measurement data, e.g., the delay per stage of various ring oscillator structures.



Macro Modeling

The macro (sub-circuit) modeling capability naturally

extends the scope of BSIMProPlus targeting the devices that can't be handled by any known compact models. The simulation engine used for macro modeling in BSIMProPlus is compatible with Spectre, HSPICE and Eldo, the *de-facto* golden SPICE simulators in the industry. The BSIMProPlus modeling platform with the help of greatly optimized simulation engine enables the user to run fast simulations and do parameter extractions for any macro model parameters by optimizations. The tool is also capable of using external SPICE simulator as a simulation engine to extract parameters for macro models or other proprietary models like Synopsys Level 66. BSIMProPlus can help users easily model the DC/CV/RF behaviors of high-voltage devices, flash devices, etc, and in the end, generate macro models in the format compatible with Cadence Spectre and/or Synopsys HSPICE.

Usability and Flexibility

The well-known usability of BSIMProPlus tools, through the intuitive GUI and powerful graphic functions, enables high productivity of modeling engineers. By using BSIMProPlus, users can easily convert limited measurement data to desired graphs by using powerful built-in conversions and many other graphical functions. Model engineers can easily obtain an initial guess of model card with reasonable good fittings by just several button-clicks, based on which the user can then do optimizations for any selected parameters by selecting fitting targets graphically. Users have the flexibility to define their own extraction/optimization strategy through C++ language, or API functions, extraction Workbench to build up their own model generation flow, or import user-developed SPICE3 model into BSIMProPlus to check the modeling result.

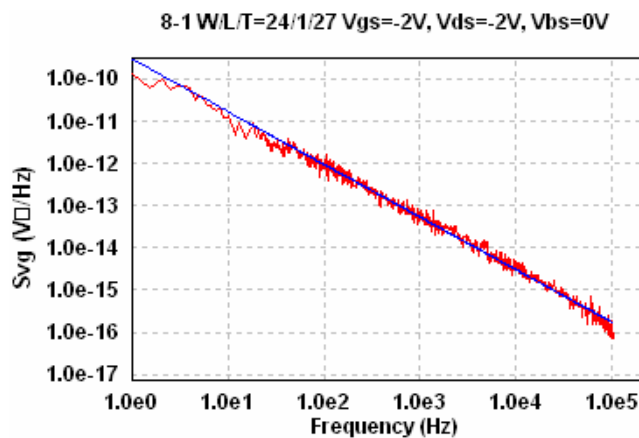
Hot Carrier Instability (HCI) and Negative Bias Temperature Instability (NBTI) Reliability Modeling

The BSIMProPlus device modeling tools provide unique, patented methodologies for the characterization of reliability problems such as HCI and NBTI that affect a wide variety of designs. By measuring devices under certain stress conditions, BSIMProPlus is able to characterize device reliability behavior, and generate reliability models based on

Cadence proprietary AgeMOS model. The use of AgeMOS can then be extended to circuit and/or whole-chip level, by using Cadence Virtuoso custom design tools e.g. RelXpert and UltraSim, to predict HCI and NBTI reliability induced failures that are critical in high-performance and DSM designs.

Flicker (1/f) Noise Simulation and Extraction

The noise modeling component of the BSIMProPlus device modeling suite provides the convenient noise simulation and extraction for BSIM3, BSIM4, BSIMSOI, BJT and resistor models. This feature also allows for the simultaneous simulation of DC and noise performance of the transistors. The measured noise data is obtained by using the NoisePro, which has been used by virtually every major foundry and IDM in the world. The NoisePro system is a combination of hardware and software tuned for accurate modeling of low-frequency noise – an issue that plagues a wide-range of designs, including RF designs, through up-converted noise components.



INSTRUMENT LIST

Platform/OS/Instruments

- ◆ Pentium 233MHz and above PC
 - Minimum of 128MB or larger RAM memory is recommended
 - Minimum of 1GB of hard disk space
 - Super VGA with 1024x768 or higher resolution display capability
- ◆ Windows 2000, Windows XP or later Windows operating

system (English, Chinese, and Japanese)

- ◆ National Instrument GPIB card and drivers (for measurement only)

Measurement (Drivers supported)

- ◆ GPIB
 - National Instrument GPIB card
- ◆ I-V Meter
 - Agilent 4142/4145/4155/4156, E5270, B1500A/B
 - Keithley SCS4200
- ◆ LCR Meter
 - Agilent 4284/4285/4294, B1500A/B
 - Keithley K590
- ◆ Switching Matrix
 - Agilent 4085, 5250, B2201A
 - Keithley 707/708
- ◆ Network Analyzer
 - Agilent 8510, 8720/8722/8753, E5070/E5071, E8358/E8361/E8362/E8363/E8364
 - Anritsu 37xxx
- ◆ Spectrum Analyzer
 - Agilent 3561, 35665, 35670, 89410, 89610
 - Stanford research 780/785
- ◆ Probe Station
 - Cascade Summit, Cascade with Nucleus
 - SUSS MicroTec probe station

© Copyright, 2007. ProPlus Design Solutions, Inc.

ProPlus Design Solutions, Inc. is the leading provider of semiconductor device modeling products that guarantees accurate transfer from semiconductor process technologies to integrated circuit designs. The Company, with more than 15 years experiences serving as the leader for the industry in the SPICE modeling field, offers full range of modeling tools, professional modeling and consulting services to electronic and semiconductor companies.

Company Headquarters

ProPlus Design Solutions, Inc.
2860 Zanker Road, Suite 204,
San Jose, CA 95134, USA

Phone: (408) 577-0378

Fax: (408) 321-8641

Email: info@ProPlusSolution.com

<http://www.ProPlusSolution.com>