What is the Simucad Model Library?

The Simucad Model Library (ModelLib) is a collection of all of the SPICE models (e.g. BSIM3, BSIM4, PSP, HiSIM) supported by Simcuad circuit simulators. Individual models are distributed online as pre-compiled, pre-linked, pre-tested binary models that simulator users can easily download and install when a required model update is available. ModelLib enables circuit simulator customers to enjoy simple, immediate and reliable access to the most up-to-date high performance SPICE models.

Why ModelLib?

When circuit designers need a model update that improves the model accuracy or fixes a bug, they typically wait 6-12 months for a simulator update from their vendor. The new simulator version typically needs to be completely re-qualified because traditional simulator design entwines the model code throughout the simulator. The new simulator version may also include many different changes. Engineers are typically on their next project by the time the update is available.

When a group of model developers from academia and the semiconductor industry collaborate on SPICE model development, there are only a few options available. The first option is to use a language interpreter that unfortunately is impractical for any circuit more than a few transistors. A second option has been to use the Verilog-A language for compact model development and accept the 3X to 5X runtime performance penalty as compared to standard Berkeley-style hand-coded "C" models. The third option is to hand-code the model changes and recompile the whole simulator. This third option may be tolerable for large semiconductor manufacturers who only use their own proprietary circuit simulators, but does not provide a viable solution for companies who use commercially available circuit simulators. The model must then be hand-coded in "C" by the simulator developer for any production use.

Now, ModelLib provides an ideal solution to these problems by enabling model developers to rapidly produce and test stand-alone "C" language models and place them online for immediate delivery. Circuit designers can now download these updated models and use them in Simucad commercial circuit simulators.

How ModelLib Works for Circuit Designers

ModelLib installation and configuration is simple for circuit designers. ModelLib-based simulator customers who subscribe will be notified by e-mail when updated models become available. Customers may also be notified by their Simcuad application engineer. Circuit designers can review the release notes for

the updated model to determine if they should download the model. Downloading from the Simucad website and simulator configuration is simple. The utility "sman" assures that the right file is placed in the correct place and the utility "slconfig" guides each step of the configuration process. You can restore to the original model configuration at any time. There is no simulation performance compromise with ModelLib. Once configured, there is no further action required.
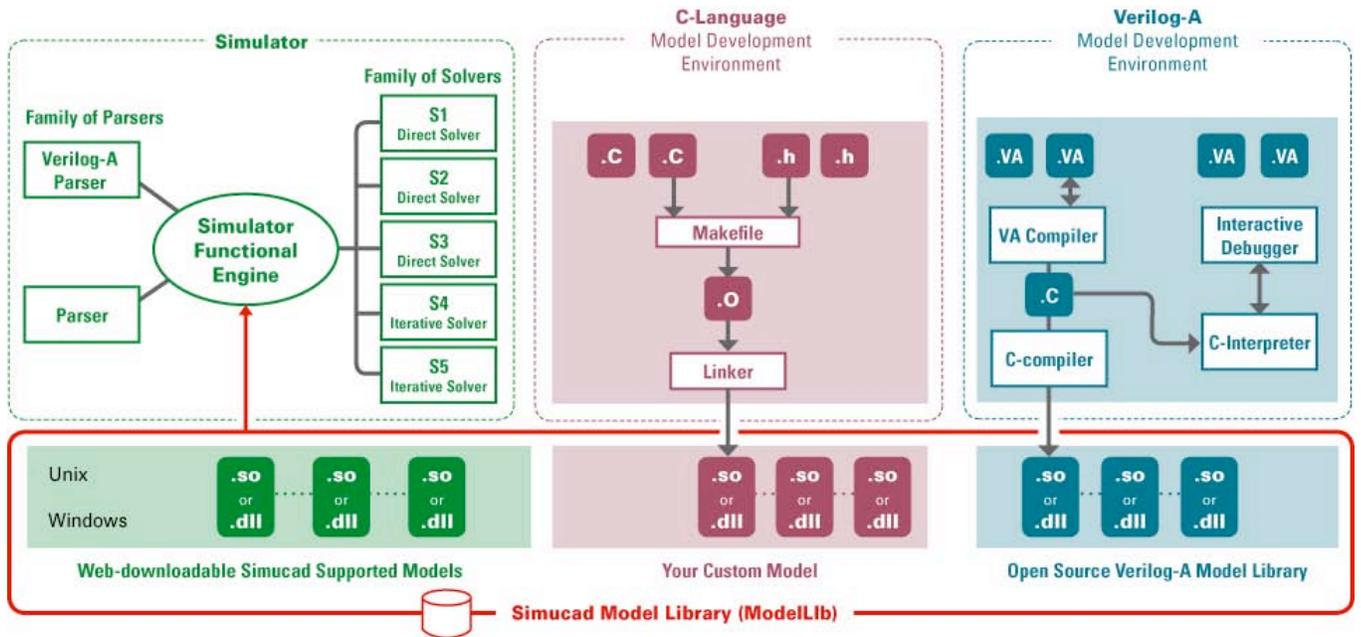

Fig. 1 Modular Architecture of Simucad Circuit Simulator Includes Separate Library of Solvers and Library of SPICE Models.

ModelLib "C" Language Development Environment

The Modellib Development Environment supports ModelLib Source Kits that include standard Berkeley-style "C" hand-coded models that conform to the basic ModelLib coding standards including thread-safe operation. Example source code kits for popular models include all of the header files, "C" code and makefiles for model developers to create customized models. Developers can modify the models, compile and link them to produce shared object files that can be configured to quickly bolt onto a circuit simulator at run time. Customer developed shared object files can be distributed and automatically configured for single user or site-wide use.

Benefits of Simucad Model Library for Circuit Designers

Fast turnaround for model updates
Same models available for SmartSpice Analog Circuit Simulator, SmartSpiceRF Hamonic Balance-based Simulator, Harmony Mixed-Signal Simulator, and Twister Full Chip Circuit Simulator
Easy to download and configure by circuit designers and CAD professionals
Reliable model updates are factory tested on all platforms and in multiple configurations
Safe update procedure does not delete anything. You can always restore to default settings

Benefits of Simucad Model Library for Model Developers

Convenient and fully independent environment for proprietary model development.
Sharable – different model development teams can share model source code and binaries on high-performance commercial circuit simulators.
Testable – it is easy to rapidly generate and run a complete set of regression tests for an independent and isolated model.
Traceable – the slconfig utility enables you to easily determine current model configuration that is also included in simulation output files
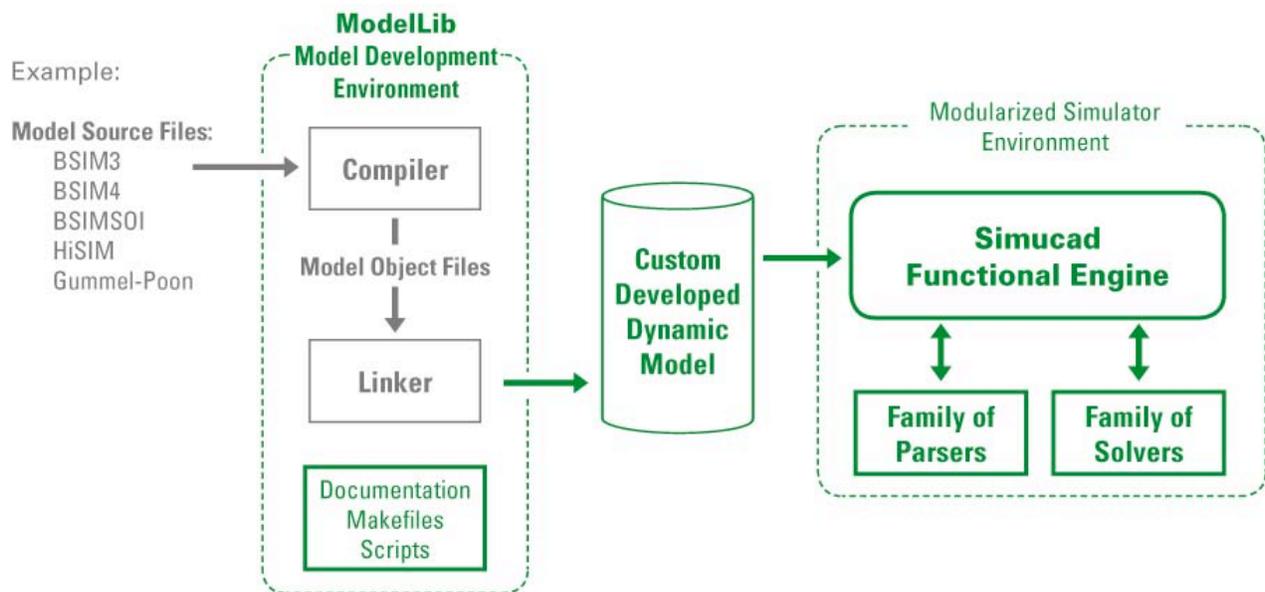Accuracy is preserved with no performance penalty

Fig. 2 The Simucad Model Library Development Environment Includes Examples of Model Source Files for Popular SPICE Models

Simucad Support of ModelLib Methodology

When circuit designers first hear about the ModelLib methodology they go through three predictable stages:

1. This is kind of odd, maybe more complex than it needs to be. Pre-compiled, pre-linked, pre-tested models? Dynamic linking of a circuit simulator at run-time?
2. Why do they DO it this way?
3. It saved my project again! This is amazing! I could never go back to the old way of waiting months for a new simulator release!

Simucad has put in place all of the support systems necessary to provide customers with what they need to use this new methodology. The Simucad Resource Center provides secure downloads of all of the latest models on Linux, Solaris and Windows. The slconfig utility manages all simulator and people configurations. Release notes describe what is different in each ModelLib version to make it easy to decide whether to download an updated model. Simucad runs exhaustive regression test suites before releasing a new ModelLib model. Simucad provides testing and qualification suites for customers to run after model installation so that circuit designers can be assured that model installation was performed correctly. ModelLib Developer Kits include all of the source files required (.c and..h) and Makefile to correctly build and test the examples models. Simucad provides a Verilog-A model development environment for those model developers who work with PSP and other Verilog-A distributed source models. Training and support for both circuit designers and model developers in the best practices in using ModelLib technology is available worldwide.

Conclusion

The new ModelLib methodology enables circuit simulation customers to enjoy fast turnaround of SPICE model updates. ModelLib methodology enables rapid collaborative development and exchange of custom SPICE models between partners