Georgia Tech

Research News

Georgia Institute of Technology
Research Communications Office
Atlanta, Georgia 30332-0800
USA
404•894•3444

*For Immediate Release*
*February 2, 1993*

# SIMULATOR FOR CIRCUITS & HIGHER-LEVEL SYSTEMS AVAILABLE FOR NO-COST LICENSE; HAS NON-ELECTRONIC APPLICATIONS

A software package developed at the Georgia Institute of Technology for simulating circuits and higher-level electronic systems is being made available to users through a no-cost license. Known as XSPICE, the program can also be used to simulate the operation of some non-electronic systems.

"XSPICE is unusual in that it provides very powerful analog simulation at the circuit card level as well as being useful for system-level simulation," explained Fred L. Cox, senior research scientist with the Georgia Tech Research Institute (GTRI). "It is especially appropriate when you want to mix system level simulation with analog simulation."

Introduced at the 1992 International Symposium on Circuits and Systems (ISCAS), XSPICE is an extended and enhanced version of the popular SPICE analog circuit simulation program developed at the University of California at Berkeley. XSPICE permits users to simulate analog, digital and even non-electronic designs from the circuit level through the system level in a single simulator.

"We can tie together the digital and the analog worlds as well as model arbitrary kinds of events," Cox said. "XSPICE integrates an event-driven simulation capability with the traditional SPICE analog simulator. This should be useful for a wide variety of applications."

A special code modeling feature allows users to add new models directly into the simulator for maximum simulation speed and accuracy. Code models are written in the C programming language, allowing arbitrarily complex behavior to be described. Code model development tools are provided to simplify the process of creating new models, compiling them and linking them with the XSPICE core.

XSPICE provides a rich set of predefined code models in addition to the standard discrete device models available in SPICE. The XSPICE code model library contains more than 40 new functional blocks, including summers, multipliers, integrators, magnetics models, limiters, S-domain transfer functions, digital gates, digital storage elements, and a generalized digital state machine.

Digital functions are simulated in XSPICE through the embedded event-driven algorithm added to the SPICE core. This algorithm is coordinated with the analog simulation algorithm to provide fast and accurate simulation of mixed-signal circuits and systems.

The event-driven algorithm supports a new "user-defined node" capability, allowing

- OVER -

*FOR MORE INFORMATION:*

*ASSISTANCE: John Toon or Lea McLees, (404) 894-3444; CompuServe at 71045,164, or Internet at jt7@prism.gatech.edu.*
*RESEARCHER: Fred L. Cox, (404) 894-7046.*
*WRITER: John Toon*

additional event-driven data types to be defined and used. XSPICE comes with a 12-state digital data type as well as a user-defined node library that includes 'real' and 'integer' types useful in simulating sampled-data systems such as digital signal processing algorithms.

Cox expects the software will be helpful to electrical engineers in a wide range of applications, though it also has uses outside of electronics.

> *"SPICE was developed for electrical engineers, but we have extended it so powerfully that it can be used for a wide range of systems."*
>
> *-- Fred Cox, senior research scientist*

"SPICE was developed for electrical engineers, but we have extended it so powerfully that it can be used for a wide range of systems," he added.

Other potential applications include modeling dynamic systems such as processing facilities, hydraulic networks, and other phenomena that can be modeled with differential equations -- or as event-driven systems. XSPICE is already being used to model the design and operation of wastewater treatment facilities.

The XSPICE software was developed at Georgia Tech to meet a U.S. Air Force need for simulating the operation of mixed digital-analog avionics equipment to aid development of test program sets. Cox said distribution of the software through a no-cost license fulfills the Air Force's desire to transfer its technology into civilian applications that can have a positive economic impact.

The extension of SPICE is fully compatible with the original code, which has been distributed under the same type of no-cost license from the University of California at Berkeley.

XSPICE is currently available for UNIX workstations and is supplied in source code form, allowing users to customize and extend both the simulator and models to their own particular needs. To date, the simulator has been successfully compiled and used on HP/Apollo and Sun workstations.

The XSPICE simulator and user's manual are available with a cost-free license arrangement from the Georgia Tech Research Corporation for a distribution charge of U.S.$200, including first class postage within the United States. An additional $25 is required for air delivery overseas.

The license agreement allows users to copy, distribute and extend the software, and to use it in commercial products. The only requirements are acknowledgement to the developers and compliance with federal regulations on software distribution.

For further information about licensing the product, contact the Office of Technology Licensing, Georgia Tech Research Corporation, Georgia Institute of Technology, 400 Tenth Street, Atlanta, Georgia 30332-0415, or phone (404) 894-6287, or fax (404) 894-9728. Internet users may send e-mail to XSPICE@GTRI.GATECH.EDU to obtain copies of the order form and license agreement. (Please include the word "LICENSE" in the subject header when mailing to the Internet address.)

# # #