Xyce™ Parallel Electronic Simulator Version 7.8 Release Notes

Sandia National Laboratories

December 15, 2023

The Xyce™ Parallel Electronic Simulator has been written to support the simulation needs of Sandia National Laboratories' electrical designers. Xyce™ is a SPICE-compatible simulator with the ability to solve extremely large circuit problems on large-scale parallel computing platforms, but also includes support for most popular parallel and serial computers.

For up-to-date information not available at the time these notes were produced, please visit the Xyce™ web page at http://xyce.sandia.gov.

Contents

Sandia National Laboratories

Xyce™ Parallel Electronic Simulator

New Features and Enhancements

XDM

- XDM did not change in this release, so new installers were not generated.

New Devices and Device Model Improvements

- The PSP103 model has been modified so that the DTA parameter can be specified on the instance line. Previously it was exclusively a model parameter.

Enhanced Solver Stability, Performance and Features

- Xyce can now handle the sources that involve transient specifications during source stepping. Source stepping is one of the solver techniques that Xyce automatically attempts when trying to solve the DC operating point (DCOP). Previously, source stepping would only apply to DC sources that had no transient specifications. It now applies to sources that involve transient specifications, with or without DC specifications. When a source has inconsistent DC and transient specifications, the value used in the source stepping depends on the analysis type. See the Xyce Users Guide for details. This leads to improved robustness and convergence of source stepping in circuits using these sources for the DCOP calculation.

- Xyce now provides a sequential source stepping algorithm that will ramp up sources with non-zero DC value in sequence, which can be more robust than the default simultaneous source stepping algorithm. This new source stepping approach can be called directly using `.options nonlin continuation=35` or `.options nonlin continuation=sourcestep2`. It will not replace simultaneous source stepping in the default DCOP strategy at this time.

Interface Improvements

- Xyce can now handle multiple `.OPTIONS` statements for the same keyword. So, for example, if the netlist contains more than one `.OPTIONS TIMEINT` command, Xyce will combine all the statements into a single, combined `.OPTIONS TIMEINT` statement. Previously, Xyce's behavior in this respect was undefined. For example, if the netlist contained multiple `.options timeint` statements, Xyce would only use one of them and silently ignore the others. Xyce now treats multiple statements as a combined single statement, consistent with other simulators, and issues warnings for duplicate parameters.

- The `.GLOBAL` statement now supports multiple global nodes on the same line. Previously each specified node needed its own unique `.GLOBAL` statement.

Important Announcements

- The model interpolation technique previously described in the Xyce Reference Guide has been removed from Xyce.

Defects Fixed in this Release

Table 1: Fixed Defects. The Xyce team has multiple issue trackers, and the table below indicates fixed issues by indentifying both the tracker and the issue number. Further, some issues are reported by open source users on GitHub and these issues may be tracked using multiple issue numbers.

Defect	Description
Xyce Project Backlog/xxx: Desciption	Details
Xyce Backlog Bugs/64: LTRA device does not properly initialize history vector when UIC/NOOP is used	The lossy transmission line device (LTRA) was not bein intialized correctly, for transient simulations that skipped the DCOP calculation. As a result, transient simulations using this device would crash if the netlist contained a `UIC` or `NOOP` keyword on the `.TRAN` line. This has been fixed.
Xyce Project Backlog/571: Modify Xyce parser so that it can handle multiple `.options` statements for the same component (for example, support `.options timeint` statements)	Most SPICE-style simulators use the `.option` (non-plural) command, and allow the netlist to have multiple `.option` comamnds. Xyce historically was different in two respects. One difference is that Xyce uses a plural `.options` command, and another difference is a required keyword to identify the part of the code to which the block of options is applied. Also, finally, Xyce was designed to only have a single `.options` statement of each type. So, for example, at most one `.options device` statement, at most one `.options timeint` statement, etc. Unforutunately, recent versions of Xyce silently accepted multiple `.options` statements, and behaved differently depending on the keyword. For example, if the netlist contained multiple `.options timeint` statements, Xyce would only use one of them and silently ignore the others. In contrast, if the netlist contained multiple `.options device` statements, it would use them all, but would ignore duplicate parameters. This was confusing for users of other simulators, and has been fixed. Xyce now treats multiple statements as a combined single statement, and issues warnings for duplicate parameters.
Xyce Project Backlog/609: Global parameters that are applied to subcircuit instance parameters don't work in parallel	If a subcircuit instance parameter (on the `X` line) depended on a `.param` or `.global_param`, and that parameter was a variable, this dependency worked in serial, but not parallel. In this case, "variable" means parameters that are allowed to change as part of a `.STEP` sweep, or as part of a sampling method. This was related to the order of operations in the parser. This has been fixed.
Xyce Project Backlog/631: Expression-based local variation doesn't always work with subcircuit arguments	When expression-based local variation was applied to a subcircuit argument, it was handled incorrectly. This meant that subcircuit arguments that should have had a random distribution were fixed to the mean value during sampling. This has been fixed.

Table 1: Fixed Defects. Note that we have two multiple issue tracking systems for Sandia Users. SON and SRN refer to our legacy open- and restricted-network Bugzilla system, and Gitlab refers to issues in our gitlab repositories.

Defect	Description
Xyce Project Backlog/637: Support shared library build of Xyce under Windows.	Modify the cmake build system to support building a shared library version of xyce, `xyce-share.dll`, so that Xyce functionality can be accessed as a library from other applications.

Supported Platforms

Certified Support

The following platforms have been subject to certification testing for the Xyce version 7.8 release.

- Red Hat Enterprise Linux® 7, x86-64 (serial and parallel)
- Microsoft Windows 11®, x86-64 (serial)
- Apple® macOS, x86-64 (serial and parallel)

Build Support

Though not certified platforms, Xyce has been known to run on the following systems.

- FreeBSD 12.X on Intel x86-64 and AMD64 architectures (serial and parallel)
- Distributions of Linux other than Red Hat Enterprise Linux 6
- Microsoft Windows under Cygwin and MinGW

Xyce Release 7.8 Documentation

The following Xyce documentation is available on the Xyce website in pdf form.

- Xyce Version 7.8 Release Notes (this document)
- Xyce Users' Guide, Version 7.8
- Xyce Reference Guide, Version 7.8
- Xyce Mathematical Formulation
- Power Grid Modeling with Xyce
- Application Note: Coupled Simulation with the Xyce General External Interface
- Application Note: Mixed Signal Simulation with Xyce 7.2

Also included at the Xyce website as web pages are the following.

- Frequently Asked Questions
- Building Guide (instructions for building Xyce from the source code)
- Running the Xyce Regression Test Suite
- Xyce/ADMS Users' Guide
- Tutorial: Adding a new compact model to Xyce

External User Resources

- Website: http://xyce.sandia.gov
- Google Groups discussion forum: https://groups.google.com/forum/#!forum/xyce-users
- Email support: xyce@sandia.gov
- Address:

Electrical Models and Simulation Dept.
Sandia National Laboratories
P.O. Box 5800, M.S. 1177
Albuquerque, NM 87185-1177

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC, a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.