

Xyce™ Parallel Electronic Simulator

Version 7.10 Release Notes

Sandia National Laboratories

May 15, 2025

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

New Features and Enhancements	2
Defects Fixed in this Release	2
Supported Platforms	3
Xyce Release 7.10 Documentation	3
External User Resources	4



Sandia National Laboratories



New Features and Enhancements

Interface Improvements

- The **TABLE** expression feature now allows the data to be specified out of order. The expression code in Xyce will now automatically sort tabular data. This works for all of the interpolation features, such as **SPLINE** and **AKIMA**.

Important Announcements

- Xyce has deprecated the autotools configuration and build system for release 7.10. Support for autotools will be removed and the CMake configuration and build system will be the default for release 7.11. This change is to facilitate planned integrations with new features and capabilities from Trilinos that are not available in the older autotools compliant versions. This notice is only relevant for users that build Xyce from source code. Please consult the CMake building guide for updated information:
<https://github.com/Xyce/Xyce/blob/master/INSTALL.md>.

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/841: Valgrind reports memory leaks with Intel MKL	Versions of Xyce built with the oneAPI IntelMKL library can report memory leaks when run under valgrind. This was due to memory allocated by the Intel MKL and has been fixed.
Xyce Project Backlog/858: Include Python, Matlab, Simulink Examples	Examples of calling Xyce from Python, Matlab and Simulink are now included in the installed version of Xyce within the share directory.
Xyce Project Backlog/861: Diode model led to Inf/NaN	When the saturation current was set to zero, it would cause the diode model to have Inf/NaN errors. This is a use case that needs to work, and it has been fixed.
Xyce Project Backlog/884: Fix doping interpolations to for 1D PDE (TCAD) device to be more robust	The interpolation of doping profiles was not working correctly, when the parameter "nmaxchop" was used. This resulted in unphysical noisy solutions in regions of high doping. This has been fixed.
Xyce Project Backlog/953: Uninitialized return variables can cause Xyce to exit.	Xyce's parameter class could return an uninitialized value in some instances when netlist parsing was expecting a numeric value and none was found. With some newer compilers this caused a premature exit of the code. This issue has been fixed.

Supported Platforms

Certified Support

The following platforms have been subject to certification testing for the Xyce version 7.10 release.

- Red Hat Enterprise Linux[®] 8, x86-64 (serial and parallel)
- Microsoft Windows 11[®], x86-64 (serial)
- Apple[®] macOS, ARM64, M-series CPUs (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
- Apple[®] macOS, x86-64 (serial and parallel)

Xyce Release 7.10 Documentation

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

- Xyce Version 7.10 Release Notes (this document)
- Xyce Users' Guide, Version 7.10
- Xyce Reference Guide, Version 7.10
- 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 multitechnology 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.