Xyce™ Parallel Electronic Simulator Version 6.11 Release Notes

Sandia National Laboratories

May 23, 2019

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 5

Interface Changes in this Release 9

Known Defects and Workarounds 10

Supported Platforms 16

Xyce Release 6.11 Documentation 16

External User Resources 17

New Features and Enhancements

New Devices and Device Model Improvements

- The BSIM-CMG version 110 (level 110 MOSFET) now provides its drain-source conductance (`GM`), junction voltages (`Vds`, `Vbs`, and `Vgs`), drain-source saturation voltage (`Vdsat`), and threshold voltage (`Vth`) for output on `.PRINT` lines using the "N()" syntax.

- The VBIC version 1.2 (BJT level 10) has been removed in this release of Xyce. Users of VBIC must use the VBIC 1.3 3T device instead. See the Xyce Reference Guide Q device (BJT) section for details on how legacy VBIC 1.2 netlists may be converted to use VBIC 1.3 instead.

- The unoptimized versions of VBIC 1.3 3-terminal and BSIM-CMG 110 (level 111 BJT and level 1110 MOSFET) have been removed. Only the optimized versions of these devices (level 11 BJT and level 110 MOSFET) remain in Xyce 6.11.

- A new "charge expression" variant of the capacitor has been implemented. Users may specify a solution-dependent expression for the capacitor's "Q" parameter, and this expression will be used to evaluate the capacitor's charge instead of by computing it from the capacitance.

- Several of the devices that had been generated from Verilog-A sources have been regenerated in such a manner as to realize a significant performance improvement over the versions in Xyce 6.10. The models that have been regenerated for this release are:
  - VBIC 1.3 4-terminal (BJT level 12)
  - HICUM levels 0 and 2 (BJT levels 230 and 234)
  - PSP version 102 and 103 (MOSFET levels 102 and 103)
  - JUNCAP 200 (Diode level 200)
  - MEXTRAM (BJT level 504)
  - MVS 2.0.0 (MOSFET levels 2000 (ETSOI) and 2001 (HEMT))

  Performance improvements of up to a factor of four in the evaluation of model equations have been observed in these devices as a result of this change.

  Furthermore, the VBIC 1.3 3-terminal model, which had previously had its derivatives optimized by hand in Xyce 6.10 (with a resulting performance improvement of approximately a factor of five) has been regenerated in this manner, which has realized a further improvement in device evaluation of approximately 25% over Xyce 6.10.

- A bug was found in the HEMT version of MVS 2.0.0 (level 2001 MOSFET) as it was obtained from its authors. The bug has been fixed in this release, but does change the results of simulation slightly.

- A new pattern (PAT) source function was added for the V and I devices.

- An error in the SPICE-compatible level 1 and level 2 diode model that could result in convergence problems or incorrect results at high temperatures was corrected. See the "Fixed Defects" table under the heading for bug 1144-SON for details.

- The BSIM3 device now supports analytic parameter sensitivities.

- Device-level parameter finite-difference sensitivities are now much more efficient for transient adjoints.

Enhanced Solver Stability, Performance and Features

- The time step recovery algorithm is now considered deprecated. It will be removed in a future version of Xyce.
- The S-parameter analysis is now supported in Xyce. It can be enabled by using a `.LIN` command with a `.AC` command in the netlist. The port device was developed to support S-parameter analysis and it can be used for DC, AC and transient analyses.The results of S-parameter analysis can be output to a file in either Touchstone 1 or Touchstone 2 format. See the Xyce Users' and Reference Guides for more details.
- The initial guess method based on DC op solutions for HB is now supported and it can be enabled by using the TAHB option.

Interface Improvements

- The Xyce expression library now accepts C-style ternary conditional operators. Please see section 2.2 of the Xyce Reference Guide for important information about restrictions on this operator and details about its behavior.
- Expressions may now contain curly braces in most contexts. They are treated as identical to parentheses.
- "X" is a now an allowed scaling factor for 1e6. In prior Xyce releases, only "MEG" was recognized as a scaling factor for 1e6.
- The `-remeasure` command line option now supports the use of the I() operator in the re-measured output files.
- .MEASURE is now supported for .AC analyses.
- Improved support for expressions on .MEASURE lines. See the "Expression Support" subsection of the .MEASURE section of the Xyce Reference Guide for more details.
- For improved HSPICE compatibility, the MEASFAIL, MEASDGT and MEASOUT options are now supported for .OPTIONS MEASURE.
- Improved support for "arithmetic operators" (e.g., + - or *) in node names and device names that are used in expressions. See the "Legal Characters in Node and Device Names" subsection of the Xyce Reference Guide for more details.
- Transient direct sensitivities can now be processed using `.FOUR`.
- Parameter sensitivities are now supported for `.AC` analysis.

Important Announcements

- The model interpolation technique described in the Xyce Reference Guide in section 2.1.17 has been marked as deprecated, and will be removed in a future release of Xyce.
- It has been determined that some distributions of Linux have broken builds of OpenMPI in their package repositories. Building Xyce from source code in parallel with these OpenMPI installs will result in a version of Xyce that may crash on some problems. This is not a bug in Xyce, but a packaging

error of the OpenMPI package on those operating systems. Please see commentary in the "Known Defects" section of these release notes under bug number "967-SON".

- Xyce has deprecated the default conversion of quoted-string file names to a table of x,y pairs of data. The old convention of `PARAMETER="file.dat"` which worked in some model statements and in behavioral sources will now generate a warning in the Xyce output. The correct way to specify a file of data for a paramater is to use the new `tablefile` keyword as in `PARAMETER=tablefile("file.dat")`. While this release of Xyce will accept both the old and new syntax, future releases will only accept the syntax of `PARAMETER=tablefile("file.dat")`. Also, a new syntax of `PARAMETER=string("string value")` has been introduced to specify parameters that are pure strings. This will be deprecated a future release and the simpler syntax of `PARAMETER="string value"` will be used to specify string valued parameters.

- The "Xygra" device, which was written as a special-purpose coupling mechanism to ALEGRA but which has occasionally been used for other coupling problems, has been marked as deprecated. The new, more flexible "General External" device was created to take its place, and has supplanted the use of Xygra in ALEGRA. The Xygra device and the API that enables it will be removed in Xyce 6.12. If your code has been using the Xygra capability to couple to Xyce, you must replace your usage with the new capability. The "General External" coupling mechanism is documented thoroughly in an application note available on the Xyce web site.

Defects Fixed in this Release

Table 1: Fixed Defects. Note that we have two different Bugzilla systems for Sandia users. SON, which is on the open network, and SRN, which is on the restricted network.

Defect	Description
1162-SON: Xyce behavior is counter-intuitive when `.DC` or `.STEP` parameters are inconsistent with syntax	Xyce's .DC and .STEP can sweep linearly, by decade, or by octave in addition to other sweep types. When a linear, decade, or octave sweep was specified in such a manner that the "stop" value could not be reached from the "start" value by applying the requested step algorithm, Xyce would not perform the same simulation that other SPICE-like simulators would have. For the linear case, if the stop value was smaller than start, but the step value was positive, Xyce would perform no simulation at all. For the decade and octave sweeps if stop was smaller than start, Xyce would perform an incorrect upward sweep starting at the start value. Xyce now performs better error checking on the arguments of `.DC` and `.STEP`, and will now always compute only the first DC or STEP value, even if the stepping algorithm cannot advance from "start" to "stop". This brings Xyce's behavior in line with those of other SPICE-like simulators.
1160-SON .DATA did not work with -remeasure for .TRAN and .DC analyses	The `.DATA` netlist command, which was introduced in release 6.10, did not work with the `-remeasure` command line option. This is fixed now.
1144-SON Fix discontinuity in diode model at high temperature	Since 2005, the Xyce diode model has supported a parameter called **IRF**, which allowed users to apply a scaling factor to the reverse current of the diode in order to match experimental data. Not only was this feature never properly validated, it was also not correctly falling back to SPICE3F5 compatibility as the documentation claimed it should have done, nor was its temperature dependence correctly documented. This has been corrected in Xyce 6.11, and the feature is now disabled correctly if **IRF** is not specified in the diode's model card, and the diode does in fact match SPICE3F5 as originally intended. The reference guide discussion of the reverse current equations in the level 1 and 2 diode models now correctly documents how **IRF** is actually used and what temperature dependence is applied to it. Furthermore, since it has been observed that use of **IRF** can introduce a discontinuity that could have significant impact on simulation results, even resulting in simulation failure, the Xyce team intends to remove this feature in a future release. Any use of the **IRF** parameter in a diode model card will now result in Xyce emitting a deprecation warning even if the parameter is being set to its default value of 1.0. That is because when **IRF** is specified at all, a temperature-dependent factor is applied to it that could also cause a discontinuity. For maximum compatibility with other SPICE-like products, users should never specify **IRF** in any diode model.

Table 1: Fixed Defects. Note that we have two different Bugzilla systems for Sandia Users. SON, which is on the open network, and SRN, which is on the restricted network.

Defect	Description
1146-SON Fix and test ill-defined behavior from ill-formed .AC lines	Some ill-formed `.AC` lines would produce the "sub-optimal" behavior of running the simulation to completion without any errors or warnings. However, no output files would be made because the .AC line had invalid parameters. Xyce now correctly emits a parsing error if the `<start frequency value>`, `<end frequency value>` or `<points value>` parameters specifed on the `.AC` line are not allowed. This also applies to the error handling for `.DATA` commands that are used to specify the frequency steps for a `.AC` analysis.
1112-SON: Support for HSpice/LTSpice charge-based capacitor needed	Xyce now supports a "Q" parameter for its capacitor. This parameter should be set to a solution-dependent expression that will be used to evaluate the capacitor charge instead of using a capacitance value.

1087-SON: Incorrectly formatted V and I device lines would cause Xyce to segfault	These netlist lines, where the value is missing after the `DC` parameter at the end of the netlist line, would cause Xyce to segfault.
	
	`V1 1 0 DC`
	`I2 2 0 DC`
	
	This is fixed now.

1037-SON: Issues with the use of TIME, TEMP, or VT in .PARAM statements	The use of the reserved words `TIME`, `TEMP`, or `VT` on `.PARAM` lines would result in the simulation running, but the resultant parameters would evaluate to 0 without generating any warning or error message. Parameters defined on `.PARAM` lines are supposed to evaluate to constants in Xyce, so the use of `TIME`, `TEMP` or `VT` should not be allowed on `.PARAM` lines. That usage will now cause a parsing error.
1033-SON: Issues with use of reserved words TEMP and VT in expressions in device instance parameters and .GLOBAL PARAM statements	Prior to Release 6.10 of Xyce, if the reserved words `TEMP` or `VT` were used in expressions for device instance parameters or in `.GLOBAL_PARAM` statements then their values would always be 0. For Release 6.10, it was fixed for netlists that had a `.STEP` command. It is now also fixed for the "non-step case" and for the case where `.OPTION DEVICE TEMP=<val>` is used to set the circuit temperature.
1034-SON: Issues with processing expressions that contain arithmetic operators (e.g., + - or *) within an otherwise legal node or device name	Something like this G-source instance line would produce a parsing error: `G1 IOUT+ IOUT- VALUE={V(VIN+)}` The issue was that "arithmetic operators", like +, were not valid in device and node names in as many expression contexts as in other Spice-like simulators. They are now allowed in device and node names in expressions, when they are enclosed within Xyce operators such as V() and I(). See the "Legal Characters in Node and Device Names" subsection of the Xyce Reference Guide for more details.

Table 1: Fixed Defects. Note that we have two different Bugzilla systems for Sandia Users. SON, which is on the open network, and SRN, which is on the restricted network.

Defect	Description
944-SON: Xyce-generated .csd Files for .AC with .STEP do not open correctly in the PSpice A/D waveform viewer	Some of the header fields (e.g., `XBEGIN` and `XEND`) were incorrect. Also, an attempt to view the signals in the `<netlistName>.TD.csd` file would cause the PSpice A/D waveform viewer to crash. These issues are fixed now.
855-SON: Missing error message when a netlist uses an operator (e.g., IR or P) that is not supported for .AC analyses	Xyce would output all zeroes or all NaNs, for the requested quantity, when a netlist used an operator (e.g., `IR` or `P`) that was unsupported for .AC or .NOISE analyses. For .AC and .NOISE analyses, Xyce now emits a parsing error for the `P` operator. It also emits a parsing error for the `I` operator for all devices, except the V, E, H and L devices and the voltage-form of the B-device. Those five devices do support branch currents for .AC and .NOISE analyses.
697-SON Re-measure did not support the I() operator	In previous Xyce releases, the `-remeasure` command line option only worked with the V() and N() operators. It is now also supported for the I() operator.
649-SON: Xyce does not support traditional ternary operators in expressions	Most other simulators accept C-style ternary operators ("?:") in expressions, and until release 6.11 Xyce did not. These operators are now valid in Xyce expressions.
2039-SRN: Xyce will not accept expressions that contain curly braces	Until release 6.11, Xyce would not accept expressions that contained curly braces (other than those delimiting the expression itself). As of this release, these internal braces are now treated as equivalent to parentheses. The only remaining exception is expressions used in table definitions for E and G sources.
1173-SON: Breakpoint handling is really inefficient when there is a large number of breakpoints	It was observed that when an independent source is specified from a PWL file, and that file contains a really large number of points (thousands), that the circuit runs much more slowly than it should. This happened because breakpoints were stored and sorted using inefficient methods. This has been fixed for this release.
1092-SON: Analytic and finite difference sensivities for the BSIM6 and BSIM3 (and possibly others) don't match well in transient	The formula used for finite difference device parameter derivatives was not accurate. It has been fixed and now it produces much more accurate results.
1086-SON: .OP and .PRINT SENS, without a .PRINT DC present, causes Xyce to exit with error	This was due to a small logic mistake in the parser and has been fixed.
1080-SON: Adjoint transient sensitivities fail if using finite difference derivatives	This was due to the finite difference derivatives being computed and stored in an inefficient manner. For transient adjoints, a time history of device parameter derivatives has to be stored, and this storage must be sparse in order to work well. The code has been rewritten to use sparse storage and works correctly with transient adjoints now.

Table 1: Fixed Defects. Note that we have two different Bugzilla systems for Sandia Users. SON, which is on the open network, and SRN, which is on the restricted network.

Defect	Description
895-SON: A user reported a discrepancy between Xyce and other spice-like simulators on an AC test case	This was due to the different DC op calculated by Xyce and the other SPICE simulators. For this test case, the DC data was not specified in the voltage source. So, Xyce used its default DC value of 0 for its DC op calulcation. The other SPICE simulators used the value of the transient data at time 0 as their DC values, which was not zero in this case. This led to different DC biases calculated and different AC results. This has been fixed
1183-SON: The AC analysis did not run correctly in parallel for some AC test cases	The AC analysis works in parallel when the AC source is a voltage source. If the AC source is a current source, the results could be incorrect depending on how the devices and the solution variables are distributed in processors. This has been fixed.

Interface Changes in this Release

Table 2: Changes to netlist specification since the last release.

Change	Detail
The behavior of the `PULSE` transient source has changed	The `V1` parameter, which is the initial value, is now a required parameter. Previously, a `PULSE` source that was specified with no parameters would default to a 0V (or 0A) DC source. Note that in other simulators, like PSpice and ngspice, both the `V1` and `V2` parameters may be required for a `PULSE` source.
The behavior of `TIME`, `TEMP` and `VT` on `.PARAM` lines has changed	That usage is now a parsing error.

Known Defects and Workarounds

Table 3: Known Defects and Workarounds.

Defect	Description
1085-SON: Expression library mishandles `.FUNC` definitions of functions that begin with "I" and are two characters long	Xyce's expression library assumes that all terms of the form "`Ix(<arguments>)`" are lead current expressions, where "x" is either a lead designator such as "D", "G" , or "S" for a MOSFET or "C", "B", "E" for a BJT, or a digit indicating the pin number of the device associated with the lead. This assumption makes it impossible for users to define a function with a two-character name starts with "I". Unfortunately, the parser does not warn of this problem should a user define such a function, and the first indication of something being wrong is an unhelpful error message about an "undefined parameter or function" where the problematic function is used. ***Workaround***: Do not use function names of two character length that begin with the letter "I". If you are making use of a vendor-supplied library that includes definitions of functions such as "I0", you will have to modify the library to change the function name and all the instances of its use.
1031-SON: .OP output is incomplete in parallel	When Xyce is run in parallel, the `.OP` output may be incomplete. ***Workaround***: One workaround is to run the netlist in serial. Another one is to use these command line options: `-per-processor -l output`. In that case, the per-processor log files will have the `.OP` information for the devices that were instantiated on each processor.
1009-SON: Transient adjoint sensitivities don't work with `.STEP`	Transient adjoint sensitivities require backward integrations to be performed after the primary transient forward integration. Doing this properly requires information to be stored during the forward solve, and for certain bookkeeping to be performed. Currently, these extra operations to support transient adjoints are not properly set up for `.STEP` analysis. ***Workaround***: None
1006-SON: SDT (expression library time integration) derivatives are not supported, so SDT can't be used for sensitivity analysis objective functions	SDT is a function supported by the Xyce expression library to compute numerical time integration. When this function is used, the expression library does not produce correct derivatives. This impacts Jacobian matrix entries, when SDT is used with a Bsrc, and it also impacts sensitivity analysis, when SDT is used in an objective function. For the former case, this can result in a lack of robustness for circuits that contain SDT-Bsrc devices. For the latter case, the objective function will simply be incorrect. ***Workaround***: None

Table 3: Known Defects and Workarounds.

Defect	Description

1004-SON: Ill-defined .STEP behavior for "default parameters" for transient sources (SIN, EXP, PWL, PULSE and SFFM)	If, for example, these netlist lines are used in a transient (`.TRAN`) simulation:
	
	`V1 1 0 SIN(0 1 1)`
	`.STEP V1 1 2 1`
	
	then Xyce will run the simulation without warnings or errors, but no instance parameter of source `V1` will be stepped.
	Workaround: Explicitly use the desired stepped parameter (e.g., `V0`) on the `.STEP` line. For example, `.STEP V1:V0 1 2 1` would work correctly.

991-SON: Non-physical BH Loops in non-linear mutual inductor	Nonlinear mutual inductors that have high coupling coefficients (i.e. model parameter `ALPHA` over 1.0e-4) and low loss characteristics (i.e. zero `GAP`) can produce B-H loops with nonphysical hysteresis. ***Workaround***: Lower `ALPHA` values or larger `GAP` values can ameliorate this issue, but the root cause is still under investigation.
989-SON: I(*) will not print branch currents that are part of a Y device	Bug 715-SON (I(*) for subcircuit devices does not work properly on .PRINT lines) was fixed for the Xyce 6.9 release. The caveat is that I(*) still does not work for branch currents that are part of a Y device. ***Workaround***: Explicitly request those Y-device branch currents on the `.PRINT` line.
800-SON: Use of global parameters in expressions on .MEASURE lines will yield incorrect results	The use of global parameters in expressions on .MEASURE lines is not allowed, as documented in the Xyce Reference Guide. However, instead of producing a parsing error the measure statement will be evaluated with the specified qualifier value (e.g., `FROM`) being left at its default value. ***Workaround***: None, other than not doing this.
971-SON: Use of default device parameter syntax on a .PRINT line causes Xyce to print 0 for that parameter	This line (`.PRINT TRAN R1`) will cause Xyce to print 0 for the resistance value of R1. ***Workaround***: Use `.PRINT TRAN R1:R` instead.
970-SON: Some devices do not work in frequency-domain analysis	Devices that may be expected to work in AC or HB analysis do not at this time. For AC this includes, but is not limited to, the lossy transmission line (LTRA) and lossless transmission line (TRA). For HB, the transmission lines do work but the nonlinear dependent sources (B source and nonlinear E, F, G, or H source) do not. ***Workaround***: The LTRA and TRA models will need to be replaced with lumped transmission line models (YTRANSLINE) for AC analysis. There is not yet a workaround for the B source in harmonic balance.

Table 3: Known Defects and Workarounds.

Defect	Description
967-SON: Zoltan segmentation fault with OpenMPI 2.1.x and 3.0.0 on some systems	It has been observed that when Xyce and Trilinos are built with OpenMPI 2.1.x or 3.0.0 on certain unsupported operating systems, a small number of test cases in the regression suite crash with a segmentation fault inside the Zoltan library. The Xyce team has determined that this is not a bug in either Xyce or Zoltan, but is instead due to some pre-packaged OpenMPI binaries on some operating systems having been built with an inappropriate option. This option, "–enable-heterogeneous" is explicitly documented in OpenMPI documentation as broken and unusable since 2013, but some package managers have OpenMPI binaries built with this option explicitly enabled. Turning on this option causes the resulting OpenMPI build to perform certain communication operations in a way that does not adhere to the MPI standard. There is nothing that can be done in Xyce or Zoltan to fix this issue — it is entirely a bug in the OpenMPI library as built on that system. A new test case has been added to the Xyce test suite in order to detect this problem. The test is "MPI_Test/bug_967", and it will be run whenever the test suite is invoked with the "+parallel" tag as described in the documentation for the test suite at `https://xyce.sandia.gov/documentation/RunningTheTests.html`. If this test fails, your system has a broken OpenMPI build that cannot be used with Xyce. At the time of this writing, this issue is present in Ubuntu Linux versions 17.10 and later, and there is an open bug report for it at `https://bugs.launchpad.net/ubuntu/+source/openmpi/+bug/1731938`. The issue may be present in other distros of Linux that are derived from Debian (as is Ubuntu), but we cannot confirm this. ***Workaround***: The only workaround for this problem is to build OpenMPI from source yourself, and not to include "–enable-heterogeneous" in its configure options. You should also post a bug report in your operating system's issue tracker requesting that they rebuild their OpenMPI binaries without the "–enable-heterogeneous" option. If you are using Ubuntu, you should register with that issue tracking system and add yourself to the list of people it affects in the existing bug report (doing so increases the "heat" of the bug, which may increase the likelihood of it being fixed).
964-SON: Compatibility of .PRINT TRANADJOINT with .STEP	The use of `.PRINT TRANADJOINT` is not compatible with `.STEP`. The resultant Xyce output will not be correct. ***Workaround***: There is none.

Table 3: Known Defects and Workarounds.

Defect	Description
939-SON: Invalid fields (XBEGIN, XEND and SUBTITLE) in Xyce-generated HOMOTOPY.csd files	The fields in the #H header block of the .HOMOTOPY.csd files are currently hard-coded to 0 and 1, respectively. The `SUBTITLE` field is incorrect for `.STEP` data. It is missing the values for the stepped parameters. ***Workaround***: There is no workaround for the `XBEGIN` and `XEND` issue. However, it should not affect the "viewability" of those files in the PSpice A/D viewer. The workaround for the `SUBTITLE` issue is to put the stepped parameters on the `.PRINT HOMOTOPY` line.
932-SON: Analysis lines do not support expressions for their operating parameters	The Xyce parser and analysis handlers do not yet support the use of expressions on netlist analysis lines such as `.TRAN`. The parameters of these analysis lines (such as stop time for `.TRAN` or fundamental frequency for `.HB`) may only be expressed as literal numbers. ***Workaround***: There is no workaround internal to Xyce. Use of an external netlist preprocessor would be required.
928-SON: The .hb_ic.prn file can be incorrect when .STEP is used with .HB	Xyce should only output the initial condition (IC) data for the accepted tolerance in the `<netlist-name>.hb_ic.prn` file. However, it currently outputs all of the intermediate IC data while harmonic balance tries to find a good tolerance if `.STEP` is used with `.HB`. ***Workaround***: There is no workaround.
883-SON .PREPROCESS REPLACEGROUND does not work on nodes referenced in expressions	The `.PREPROCESS REPLACEGROUND` feature does not replace ground synonyms if they appear in B source expressions. ***Workaround***: Do not use ground synonyms (`GND`, `GROUND`, etc.) in expressions. Use a literal "0" when referring to the ground node in expressions.
812-SON: Undocumented limitations on, and bugs with, parameter and global parameter names	Based on external customer input and pre-release testing, there are some bugs and undocumented limitations on parameter and global parameter names in Xyce. Parameters and global parameters should start with a letter, rather than with a number or "special" character like #. In addition, the use of a single character V as a global parameter name can result in either netlist parsing failures or incorrect results from `.PRINT` lines.
807-SON: BSIM4 convergence problems with non-zero rgatemod value	There have been reports of convergence problems (e.g., the Xyce simulation fails part way through and says that the "time step is too small") when the `rgatemod` parameter is non-zero.
794-SON: Bug in TABLE Form of Xyce Controlled Sources	In some case, a Xyce netlist that contains a controlled source that uses the TABLE form will get the correct answer at first. However, it may then "stall" (e.g, keep taking really small time-steps) and never complete the simulation run. ***Workaround***: In some cases, the TABLE specification for the controlled source can be replaced with a Piecewise Linear (PWL) source that uses nested IF statements.
783-SON: Use of ddt in a B-Source definition may produce incorrect results	The `DDT()` function from the Xyce expression package, which implements a time derivative, may not function correctly in a B-Source definition. ***Workaround***: None.

Table 3: Known Defects and Workarounds.

Defect	Description
727-SON: Xyce parallel builds hang randomly on OS X	During Sandia's internal nightly testing of the OSX parallel builds, we see that Xyce "hangs on exit" with an estimated frequency of less than 1-in-5000 simulation runs. We have not seen this issue with parallel builds for either RHEL6 or BSD. The hang is on exit, whether on a successful exit or on an error exit. The hang occurs after all of the Xyce output has occurred though. So, the user will get their sim results, but might have trouble if the individual Xyce runs are part of a larger script. ***Workaround***: None.
661-SON Lead currents and power accessors (I(), P() and W()) do not work properly in .RESULT Statements	There are two issues. First, `.RESULT` statements will fail netlist parsing if the requested lead current is omitted from the `.PRINT TRAN` line. As an example, this statement (`.RESULT I(R1)`) requires either `I(R1)`, `P(R1)` or `W(R1)` to be on the `.PRINT TRAN` line. Second, the output value, in the `.res` file, for the lead current or power calculation will always be zero.
652-SON: HB output is buggy	While a straightforward use of `.print HB` works as described in the Xyce Users' Guide and Reference Guide, several of the documented features do not work as intended. More details are provided by other entries in this table.
583-SON: Switch with RON=0 leads to convergence failure.	The switch device does not prevent a user from specifying `RON=0` in its model, but then takes the inverse of this value to get the "on" conductance. The resulting invalid division will either lead to a division by zero error on platforms that throw such errors, or produce a conductance with "Not A Number" or "Infinity" as value. This will lead to a convergence failure. ***Workaround***: Do not specify an identically zero resistance for the switch's "on" value. A small value of resistance such as 1e-15 or smaller will generally work well as a substitute.
469-SON: Belos memory consumption on FreeBSD and excessive CPU on other platforms	Memory or thread bloat can result when using multithreaded dense linear algebra libraries, which are employed by Belos. If this situation is observed, either build Xyce with a serial dense linear algebra library or use environment variables to control the number of spawned threads in a multithreaded library.
468-SON: It should be legal to have two model cards with the same model name, but different model types.	SPICE3F5 and ngspice only require that model cards of the same type have unique model names. They accept model cards of different types with the same name. Xyce requires that all model card names be unique.
250-SON: NODESET in Xyce is not equivalent to NODESET in SPICE	As currently implemented, `.NODESET` applies the initial conditions given throughout a full nonlinear solve for the operating point, then uses the result as an initial guess for a second nonlinear solve with no constraints. This is not the same as SPICE, which merely applies the given initial conditions to a single nonlinear solve for the first two iterations, then lets the problem converge with no further constraints. This can lead to a Xyce `.NODESET` failing where the same netlist in SPICE might not, if the initial conditions are such that a full nonlinear solve cannot converge with those constraints in place. There is no workaround.

Table 3: Known Defects and Workarounds.

Defect	Description
247-SON: Expressions don't work on .options lines	Expressions enclosed in braces ({ }) are handled specially throughout Xyce, and may only be used in certain contexts such as in device model or instance parameters or on `.PRINT` lines.
49-SON Xyce BSIM models recognize the model TNOM, but not the instance TNOM	Some simulators allow the model parameter TNOM of BSIM devices to be specified on the instance line, overriding the model parameter TNOM. Xyce does not support this.
27-SON: Fix handling of .options parameters	When specifying .options for a particular package, what gets applied as the non-specified default options might change.

2119-SRN: Voltages from interface nodes for subcircuits do not work in expressions used in device instance parameters	This bug can be illustrated with this netlist fragment:
	
	`X1 1 2 MySub`
	`.SUBCKT MYSUB a c`
	`R1  a b 0.5`
	`R2  b c 0.5`
	`.ENDS`
	`B1 3 0 V={V(X1:a)}`
	
	This fragment will produce the netlist parsing error `Directory node not found: X1:A`. The workaround is to use `V={V(1)}` in the B-source expression instead. This bug also affects the solution-dependent capacitor.

1923-SRN: LC lines run out of memory, even if equivalent (larger) RLC lines do not.	In some cases, circuits that run fine using an RLC approximation for a transmission line, exit with an out-of-memory error if the (supposedly smaller) LC approximation is used.
1595-SRN: Xyce won't allow access to inductors within subcircuits for mutual inductors external to subcircuits	It is not possible to have a mutual inductor outside of a subcircuit couple to inductors in a subcircuit. ***Workaround***: Put all inductors and mutual inductance lines that couple to them together at the same level of circuit hierarchy.

Supported Platforms

Certified Support

The following platforms have been subject to certification testing for the Xyce version 6.11 release.

- Red Hat Enterprise Linux® 7, x86-64 (serial and parallel)
- Red Hat Enterprise Linux® 6, x86-64 (serial and parallel)
- Microsoft Windows 10®, x86-64 (serial)
- Apple® OS X Sierra, x86-64 (serial and parallel)

Build Support

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

- FreeBSD 11.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 6.11 Documentation

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

- Xyce Version 6.11 Release Notes (this document)
- Xyce Users' Guide, Version 6.11
- Xyce Reference Guide, Version 6.11
- Xyce Mathematical Formulation
- Power Grid Modeling with Xyce
- Application Note: Coupled Simulation with the Xyce General External Interface
- Application Note: Syntax, Parsing and Feature Differences Between HSPICE and Xyce 6.11

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.