SAND2020-2877R

Sandia National Laboratories

XYCE CASE STUDY

Unclassified Unlimited Release

IPO Integrated Partnerships Organizations

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology & 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.

U.S. DEPARTMENT OF ENERGY

NNSA National Nuclear Security Administration

BACKGROUND

With the elimination of underground nuclear testing and declining defense budgets, science-based stockpile stewardship requires increased reliance on high performance modeling and simulation of weapon systems. Today's weapon systems are comprised of various electrical components and systems. As a result, there is a need for tools that will allow the use of massively parallel modeling and simulation techniques on high performance computers in existing and future weapons' electrical systems models.[1]

The Xyce Parallel Electronic Simulator is a SPICE (Simulation Program with Integrated Circuit Emphasis)-compatible circuit simulator designed to run on large-scale parallel computing platforms, though it can also execute efficiently on a variety of architectures including single processor workstations.[2] As a mature platform for large-scale parallel circuit simulation, Xyce supports standard capabilities available in commercial simulators, in addition to various devices and models specific to Sandia's needs.[3] Specifically, Xyce aids in the design and verification of electrical and electronic circuits and systems prior to weapons' manufacturing and deployment.[4]

While designed to be SPICE-compatible, Xyce is not a derivative of SPICE. Built with a modern code base, Xyce was written from scratch in C++ to give it a modular, flexible design to enable the easy development of different analysis types, solvers, and models. This design enables Xyce to support large-scale parallel computing architectures through a message-passing implementation, which allows it to run on serial, shared-memory and distributed-memory parallel systems. Xyce can also leverage other Sandia-developed software tools, such as Trilinos, an open-source algorithm and solver framework.

Xyce goes beyond most SPICE-based codes in a number of ways, such as using differential-algebraic equation formulations to improve encapsulation (bundling of data) between models and solver layers, allowing the device model package to be better isolated from solver algorithms.[5] In addition to the standard analysis types found in most SPICE-based compatible codes-such as steady-state, transient, and noise analysis-Xyce also has advanced analysis types such as harmonic balance, multi-time PDE (partial differential equation), and model-order reduction methods.[6] There has been seven major versions of the software, with over 30 releases.[7]

RETURN ON INVESTMENT

PROGRAM DEVELOPMENT

Sandia developed Xyce internally with funding from the National Nuclear Security Administration (NNSA)'s Advanced Simulation and Computing (ASC) Campaign and the Defense Advanced Research Projects Agency (DARPA)'s Posh Open Source Hardware project.[8] Since 1999, Xyce has been in

[1] Report by Sandia PI Jason Verley
[2] https://info-ng.sandia.gov/xyce/#main-content
[3] Report by Sandia PI Jason Verley
[4] https://woset-workshop.github.io/PDFs/a15.pdf
[5] Report by Sandia PI Jason Verley
[6] https://xyce.sandia.gov/about_xyce/
[7] https://info-ng.sandia.gov/xyce/previous-releases/
[8] https://woset-workshop.github.io/PDFs/a15.pdf

continuous development at Sandia to support the evolving simulation needs of Sandia's electrical designers, and over time has matured into a platform for large-scale circuit simulation.[9] From the beginning, the focus of Xyce development has been to provide scalable, numerically accurate analog simulation for large-scale circuits through the development and improvement of algorithms at the core of SPICE-style simulation. Future developments are focused on making Xyce more compatible with industry standard simulators, such as HSPICE and Spectre. Specific areas of planned improvement include: feature compatibilities, netlist parsing compatibilities, and compact model support capabilities.

In 2003, Xyce solved a 14.3 million device circuit problem using 1,024 processors. Then, in 2004, Xyce became part of a commercial electronic design automation product for Fastrack Design Inc., a provider of advanced custom Application-Specific Integrated Circuit (ASIC) design solutions. In 2008, Sandia won an R&D 100 award for Xyce 5.0.2.[10]

LICENSES

Xyce became an open-source software released under the General Public License in October 2013. Xyce had also been licensed by several companies.[11]

LDRD

In the last 19 years, there has been Laboratory Directed Research and Development (LDRD) related to the development of Xyce.[12]

PUBLIC GOOD

Commercial simulation tools tend to be expensive, with limited models available for simulating intense radiation environments. In normal environment simulations, the high-cost of commercial tools makes conducting large uncertainty quantification studies increasingly expensive. Therefore, Sandia developed Xyce, in part, to address these concerns. Since October 2013, over 5,000 users have registered for the open-source version of Xyce.

Power Grid Modeling

Xyce has been used for power grid modeling, specifically for simulating electromagnetic pulse effects on the power grid to test and measure grid resiliency. For example, Sandia scientists used Xyce to simulate the effects of a nuclear blast on the power grid. The power of a nuclear blast can release electrical currents that can blow-out transformers and other technical equipment associated with the grid; thereby causing severe damage to the nation's electrical energy supply. Xyce has helped Sandia scientists to understand these vulnerabilities in the power grid so that they can inform and develop resiliency strategies and technologies in response to evolving threats.[13]

---

[9] https://info-ng.sandia.gov/xyce/#main-content
[10] https://www.rdmag.com/award-winners/2008/09/iteractive-solvers-know-millions-unknowns
[11] https://info-ng.sandia.gov/ESP/output.php
[12] https://ldrd.sandia.gov/cgi-bin/WebObjects/LDRDSearch.woa/wa/?ldrdKey=fc88a81f49eaacfba4dc12a361083fc4f4586595b75f1f46
[13] Interview with Jason Verley (Sandia PI on Xyce)

XYCE

ORIGIN

- A SPICE-compatible circuit simulator designed to run on large-sale parallel computing platforms.
- Aids in the design and verification of electrical and electronic circuits and systems prior to weapons' manufacturing and deployment.
- Written in C++ to give it a modular, flexible design to enable easy development of different analysis types, solvers, and models.

DEPLOYMENT

- Developed internally at Sandia with funding from NNSA's Advanced Simulation and Computing Campaign and DARPA's Posh Open Source Hardware project.
- In continuous development since 1999 to support the evolving simulation needs of Sandia's electrical engineers.
- Used for power grid modeling to assist in informing strategy on grid resiliency from evolving nuclear threats.

ROI

Program Development

LDRD Projects

Recognition & Credibility

1999 — ***Sandia develops Xyce with NNSA funding***

NNSA National Nuclear Security Administration

2003 — ***Xyce solves 14.3 million device circuit problem***

2004 — ***Xyce became part of Fastrack Design Inc.'s commercial electronic design automation product***

2008 — ***Sandia won R&D 100 award for Xyce***

R&D 100

2013 — ***Xyce became open source***

Xyce™

2016 — ***A defense contractor, national lab, and government agency licensed Xyce***

2018 — ***Sandia receives DARPA funding for Xyce***

DARPA