

# **PSpice A/D and PSpice Advanced Analysis: What's New in Release 23.1**

**Product Version 23.1  
September 2023**

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

**Restricted Permission:** This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

---

# Contents

---

What’s New in 23.1 ..... 3

Support for Digital Modeling Application ..... 3

Temperature Variation in a Single Simulation ..... 5

Parameter Support for Exponential Sources ..... 6

Enhanced Debugging of Convergence Error ..... 6



---

# What's New in 23.1

---

This chapter describes the following enhancements and new features in the PSpice® A/D and PSpice®<sup>1</sup> Advanced Analysis 23.1 release:

- [Support for Digital Modeling Application](#)
- [Temperature Variation in a Single Simulation](#)
- [Parameter Support for Exponential Sources](#)
- [Enhanced Debugging of Convergence Error](#)

## Support for Digital Modeling Application

Release 23.1 supports digital devices and sources in *Modeling Application*. You can model various digital devices such as gates (Buffer, Inverter, AND, OR, and so on), flip flops (Clocked SR, Clocked JK, and so on), latches (SR, D, and so on), and sources (digital stimulus, digital clock) and place them on the schematic.

You can access modeling applications from *Place – PSpice Part* menu in OrCAD X Capture.

The integrated Capture – PSpice flow supports the following digital devices and sources:

Category	Digital Devices
Sources	Digital Pulse, Digital Stimulus
Gates	Buffer/Inverter, XOR/XNOR, OR/NOR, AND/NAND
Flip Flops	Clocked SR, Clocked SR - Set/Reset, Clocked JK, Clocked JK - Set/Reset, Clocked D, Clocked D - Set/Reset, Clock T, Clocked T- Set/Reset
Latches	SR, SR - Set/Reset, D, D - Set/Reset
	Digital Constant

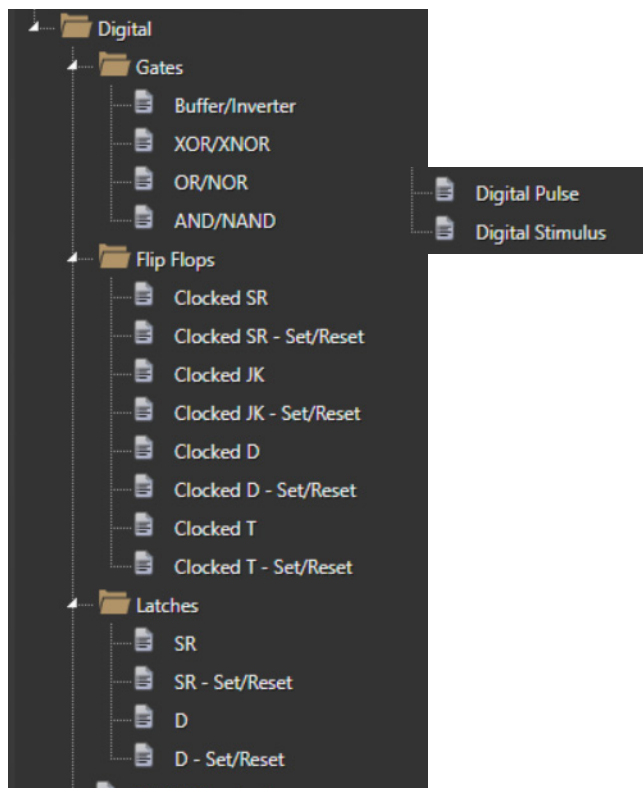
---

1. Depending on the license and installation, either PSpice or PSpice Simulator is installed.

## PSpice A/D and PSpice Advanced Analysis: What's New in Release 23.1

### What's New in 23.1

---



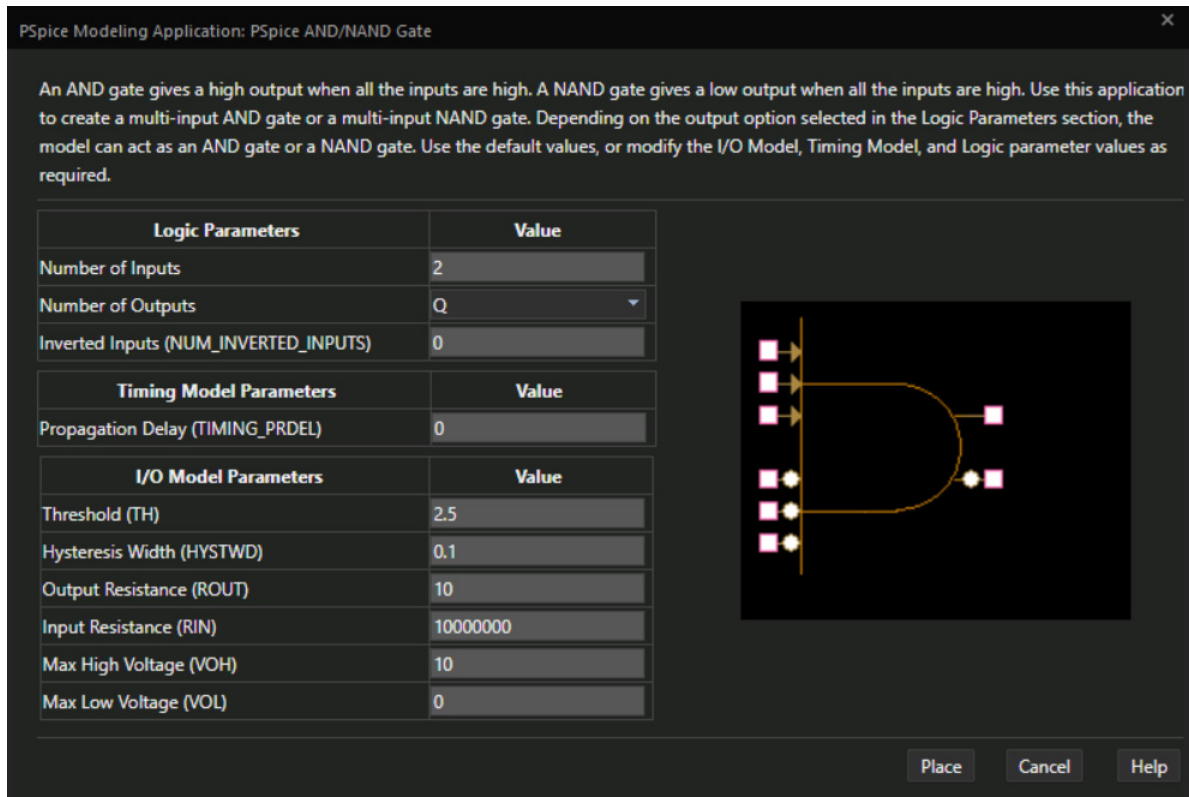
In release 23.1, digital devices also support logic, timing, and I/O model parameters in *Modeling Application*. You can modify the default values of the parameters before placing the digital devices on the schematic.

- Model logic gates, flip-flops, latches with parameters such as, *Threshold*, *Setup/Hold Time*, *Number of Input/Output*.
- Model digital pulse sources with parameters such as, *Output Resistance (ROUT)*, *Max High/Low Voltage (VOH/VOL)*, *Delay*, and *Clock ON/OFF Time*.

## PSpice A/D and PSpice Advanced Analysis: What's New in Release 23.1

### What's New in 23.1

- Model digital stimulus with parameters such as, *Number of Output Nodes*, *Output High/Low Voltage*, and *Output Resistance*.



### Related Documentation

[PSpice A/D Modeling Application](#)

## Temperature Variation in a Single Simulation

PSpice simulator is now enhanced to support temperature that can vary with time in a single transient run. To vary temperature as a function of time in a single run, use an option `.OPTIONS ENABLE_TIME_VARYING_TEMPERATURE`.

Model parameters such as `T_ABS`, `T_REL_GLOBAL`, and `T_REL_LOCAL` are used for device temperatures for expression in the model definition.

In the expression, use `CDN_CUR_TIME` (alias of time) with temperature-dependent parameters of the components.

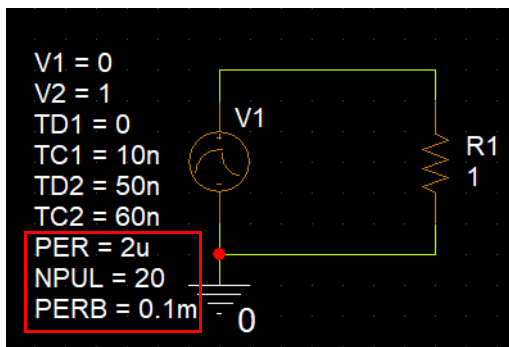
### ***Related Documentation***

[PSpice A/D Reference Guide](#)

## **Parameter Support for Exponential Sources**

To model exponential voltage or a current source, two components, `IEXP_B` and `VEXP_B` are provided in `source.olb`. These components support the following parameters that you can configure, if required:

- `PER`: Defines pulse period.
- `NPUL`: Defines the number of pulse.
- `PERB`: Defines the repeat burst period.



### ***Related Documentation***

[PSpice A/D Reference Guide](#)

## **Enhanced Debugging of Convergence Error**

Abnormally high or low values or floating point errors in complex circuits often lead to convergence errors. However, it is difficult to find the exact expression that causes the error.

As an enhancement in the 23.1 release, you can now use the `EXPR_DEBUG` option to include the expressions in the warning message that cause convergence errors.



## PSpice A/D and PSpice Advanced Analysis: What's New in Release 23.1

### What's New in 23.1

---

Syntax to use the option: `.OPTIONS EXPR_DEBUG`.

The following image shows a warning message (ORPSIM-16608) that lists the expression leading to the convergence error:

```
800
801      X_U1.x1.Rtemp X_U1.xm1.RTEMP
802      R 1 1
803      TC1 2.112000E+03 3.000000E+03
804      TC2 36.244000E+06
805
806
807 Starting pseudo-transient algorithm.
808
809 INFO(ORPSIM-16594): To improve Pseudotransient Convergence and Performance, set following options to relax stabilization criteria for capacitor currents and inductor voltages: PTRANABSTOL=1e-5, PTRANVNTOL=1e-4
810
811 WARNING(ORPSIM-16608): Expression (((VGS((%0),(%1),(%2))) + LOG10(2*cosh(VGS((%0),(%1),(%2)))))/2 +0.0001)) evaluates to an abnormal value leading to numeric overflow/underflow at time 1.734E+06
812 This expression appears in context of device X_U3$MFX$MFGMOS
813 Value of operands passed to the expression evaluator:
814 CONSTANT=27 CONSTANT=54459 CONSTANT=647.478 FUNC = ((( - (Uth(( ))) + d/Th + (pa66 - DIBL(( )))))
815 CONSTANT=1.0001 CONSTANT=2
816
817 ERROR(ORPSIM-16591): Floating point computation failed during Device/Model Load. Possible solutions: 1)Ensure that all device parameters are in valid range. 2)Try using options LIMIT
818
819 INTERNAL ERROR -- Maths Exception Code:e06d7363 in device X_U3$MFX$MFGMOS, Cosh(887.102)
820
821
822 ABORTING SIMULATION.
```

### Related Documentation

[PSpice A/D Reference Guide](#)

## **PSpice A/D and PSpice Advanced Analysis: What's New in Release 23.1**

### **What's New in 23.1**

---