cadence®

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

Product Version 23.1
September 2023

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

Contents

What's New in 23.1

This chapter describes the following enhancements and new features in the PSpice® A/D and PSpice®[1] 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

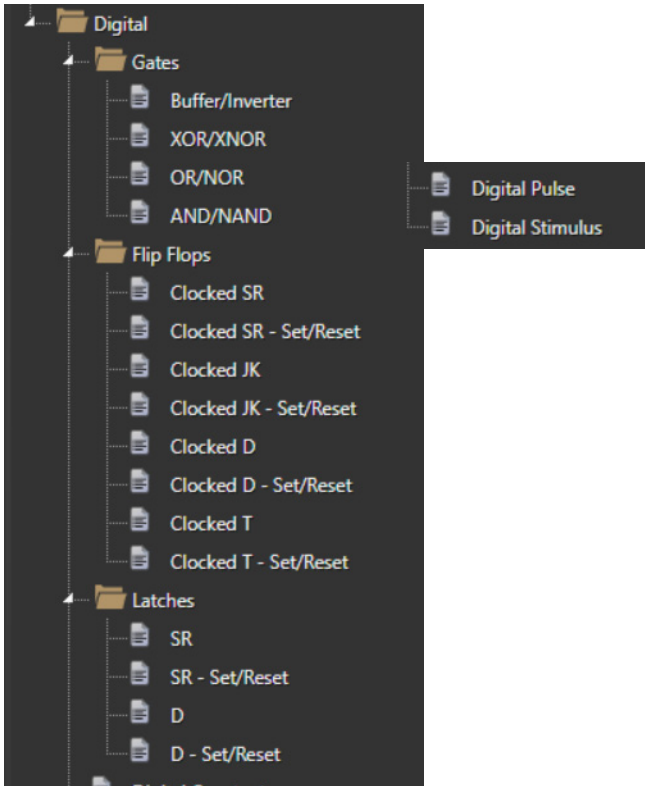
Release 23.1 supports digital devices and sources in *Modeling Application*. You can model various digital devices such as gates (Buffer, Inverter, AND, OR, and so on), flip flops (Clocked SR, Clocked JK, and so on), latches (SR, D, and so on), and sources (digital stimulus, digital clock) and place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

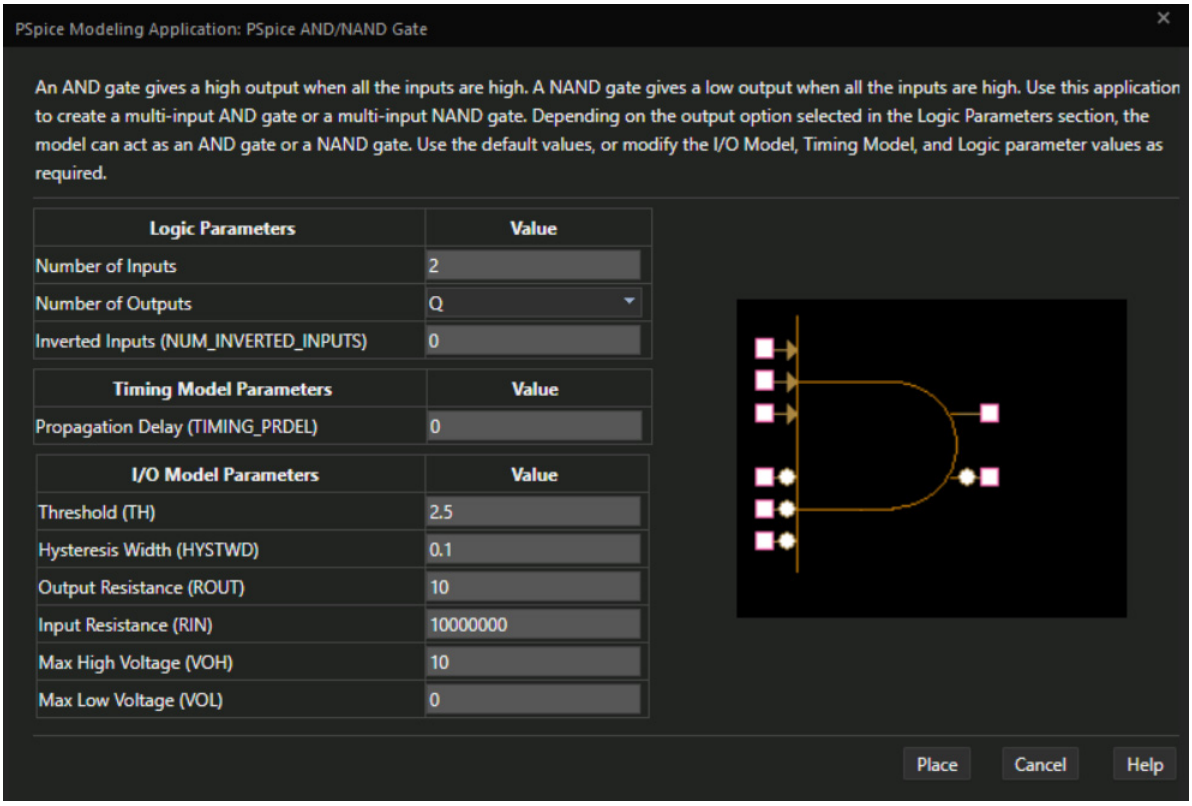
1. Depending on the license and installation, either PSpice or PSpice Simulator is installed.



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*.

- 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.

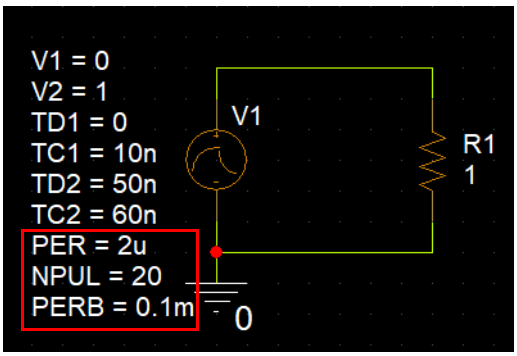
Related Documentation

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

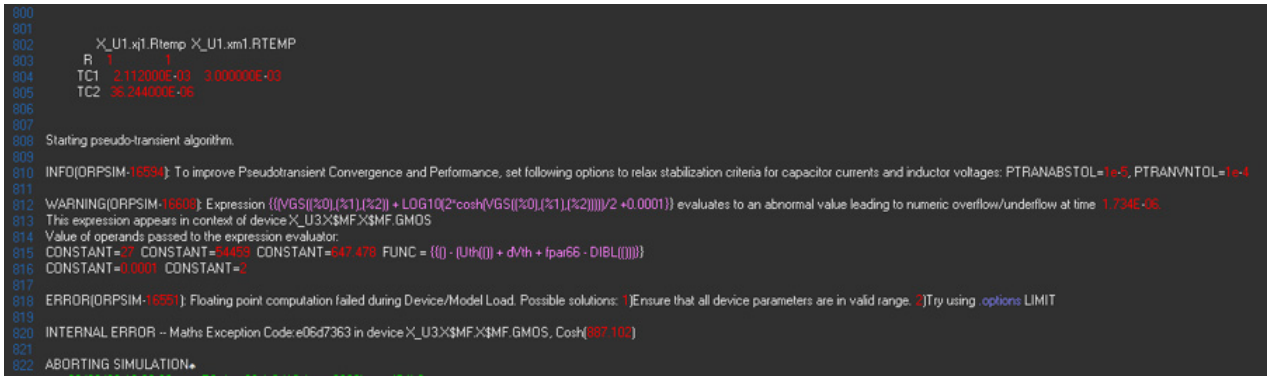
Enhanced Debugging of Convergence Error

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.

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:



Related Documentation

PSpice A/D Reference Guide