FlowCAD

What's New in PSpice 23.1

1 Support for Digital Modelling 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.

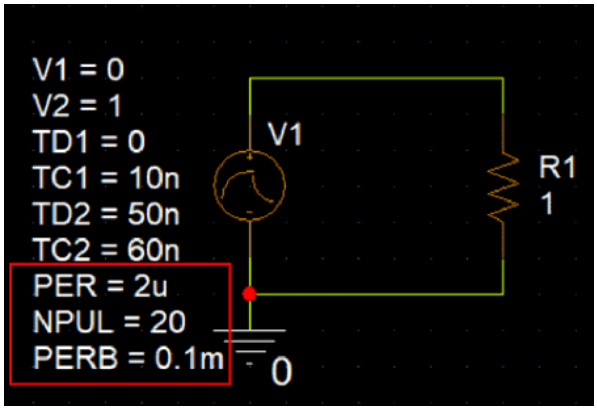
2 Temperature Variation in a Single Simulation

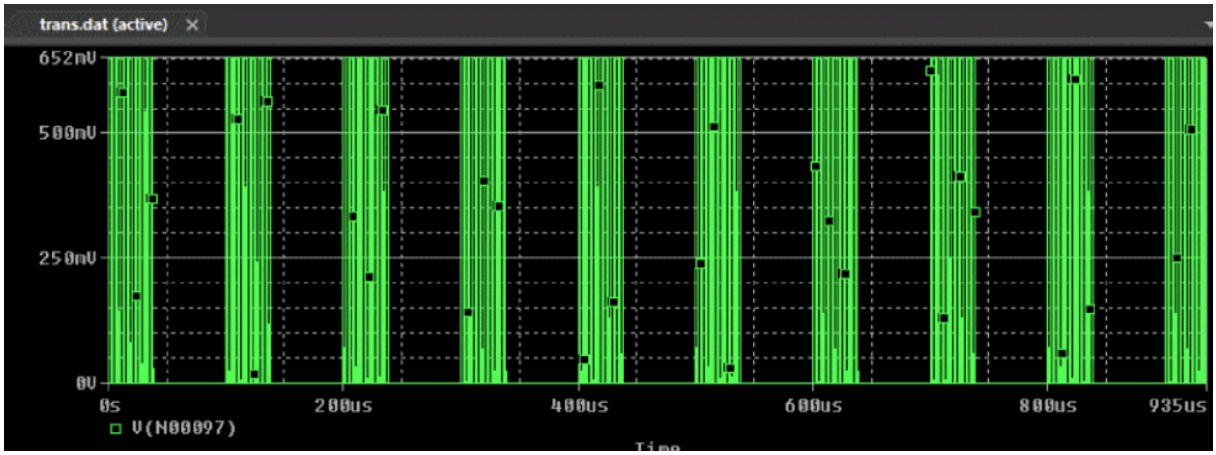
PSpice simulator is now enhanced to support temperature that can vary with time in a single transient run

3 Parameter Support for Exponential Sources

To model exponential voltage or a current source, two components, `IEXP_B` and `VEXP_B` are provided.

You can define pulse period (PER), number of pulse (NPUL) and repeat burst period (PERB) in an exponential voltage or a current source and use the model in a design.





4 Enhanced Debugging of Convergence Error

A new option allows to find the exact expression that causes abnormally high or low values or floating point errors in complex circuits, which often lead to convergence errors.

5 Support for Parallel Simulation to Improve Simulation Performance

PSpice simulator provides support for parallel simulation to generate faster simulation results and improve performance for the following analyses:

- Monte Carlo Analysis
- Parametric Sweep
- Temperature Sweep

6 Support for Transient Sources ISO 7637-2, ISO 16750-2

PSpice A/D now supports standard transient sources ISO 7637-2 and ISO 16750-2 used for automotive applications.

7 Ability to Control Noise Contributors in Noise Analysis

You can now exclude the noise contribution of a device from the Noise Analysis report.

8 Ability to Simulate Audio Input Files

- You can now specify an audio file as an input source in a PSpice A/D simulation.
- .wav formatted output files can be generated.

9 Persona Support in PSpice

- PSpice A/D introduces simulation personas, an easy and effective way to globally define and apply simulator options.
- Predefined default personas are available.