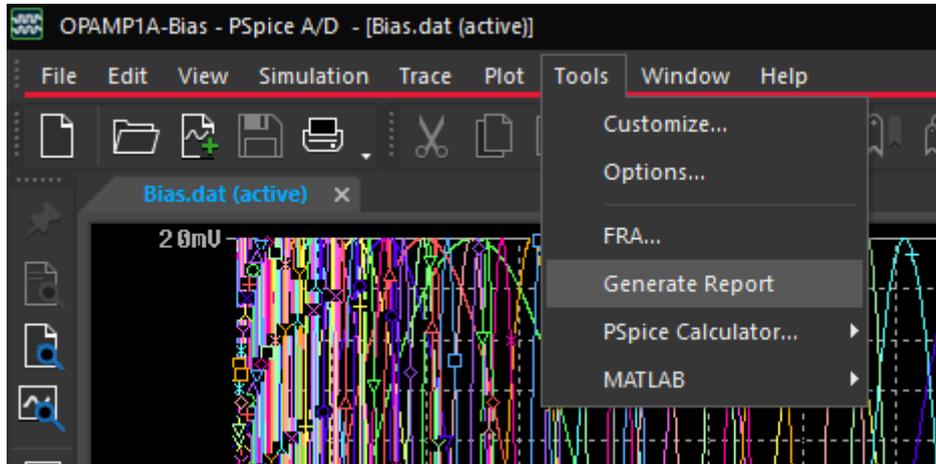




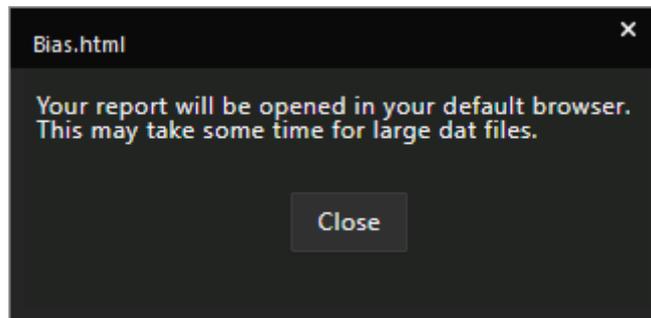
Creating Simulation Reports from PSpice AD

Creating Simulation Reports from PSpice AD

Once a simulation has completed, use Tools>Generate Report from the Probe, results, menu.



If the simulation produced a large data file, you would see the message about processing the output data to generate the formatted report:



There may be some minutes of background activity while the report is generated. Sections of the generated report are as follows, starting with the report header:

Transient Simulation Report

1 General Information

1.1 Dat file

E:\SPB_Data\PSpice\TPS54160_PSPICE_TRANS\tps54160-PSpiceFiles\SCHEMATIC1\trans\trans.dat

1.2 Analysis Name

Transient Analysis

1.3 Circuit Name

** Profile: "SCHEMATIC1-trans" [E:\SPB_Data\PSpice\TPS54160_PSPICE_TRANS\tps54160-pspicefiles\schematic1\trans.sim]

1.4 Simulation Time

Then the Nets section:

How to access Cadence software Updates (Hotfixes)

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.