How to Use PSIM-HyperSpice Link

Altair PSIM Tutorial

1. Overview

HyperSpice™ is a high-performance circuit simulator using the SPICE syntax. Its language is based on the Spice3 spice standard circuit description language. Its architecture is based on a hierarchical view of circuit description and is using a complex elaboration that offers the possibility to simulate huge circuits providing accuracy and good simulation speed while consuming less memory. HyperSpice is a matrix-based simulator. The solving is made by using a specific solver using the latest powerful techniques of the market for the system resolution.

The PSIM-HyperSpice link is a new feature starting from the v2022.3 release. This new feature has following highlights:

- With PSIM-HyperSpice link, the transient, AC, and DC Spice operations can be performed in PSIM environment.
- PSIM can generate a HyperSpice netlist based on the schematic.
- PSIM can pass the netlist to HyperSpice, and initiate the simulation.
- PSIM calls the HyperSpice DLL at each time step and receives simulation results from HyperSpice.
- Upon completion of the simulation, simulation results are displayed in SIMVIEW.

This tutorial is written for users who already have some experience in both PSIM and SPICE simulations. It provides a quick guide on how to use the HyperSpice unit in PSIM. For further details, please refer to SPICE Module User Manual.

2. Getting Started with HyperSpice Simulation

Running a HyperSpice simulation in PSIM:

To run HyperSpice simulation, click on the **Run HyperSpice Simulation** button on the toolbar or select **Simulate** >> **Run HyperSpice Simulation** from the pull-down menu, as indicated below in the figure.

HyperSpice simulation is run in the HyperSpice engine. Only the content of the .log file generated from HyperSpice will be displayed in PSIM's message window. During the time while HyperSpice simulation is running, there is no status or progress information passed from HyperSpice to PSIM for

display. At the end of HyperSpice simulation, PSIM's message window will refresh the message from the .log file generated by HyperSpice, and the content of the .raw file can be displayed in SimView the same way as the PSIM simulation.

The transient analysis will be performed with the following settings for the above example, as shown below.
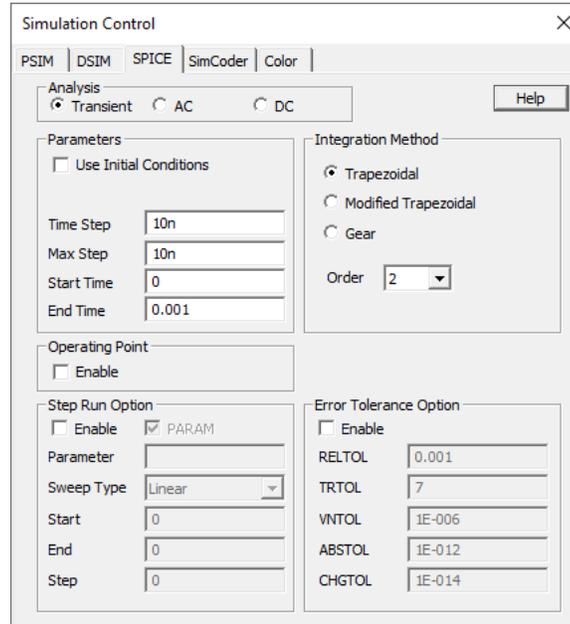
3. Modes of HyperSpice simulation

HyperSpice can perform simulation in three modes:

- Transient: time domain simulation

- AC: frequency response simulation

- DC: Operating point simulation

3.1 Transient: Time domain simulation

Users can open an example from the Spice Link example folder to simulate the transient-analysis-based SPICE simulation. To keep the original examples unchanged, we will create a new folder "*C:\PSIM_SPICE_Link Tutorial*", and use this folder as the working folder in this tutorial.

Click "File >> Open Examples…" then go to the subfolder "*C:\PSIM 2022.3\examples\SPICE Link\dc-dc\Phase-shifted full-bridge converter (SPICE).psimsch*". Select the example file "*Phase-shifted full-bridge converter (SPICE).psimsch*" and use "File >> Save As…" to save the file into the new folder.

The example circuit is shown below. It is a dc-dc phase-shifted full-bridge converter circuit.

In this circuit,

- A SPICE Directive Block "XSPD1" contains a SPICE netlist for the Transformer model of TI's LM5046.

- A SPICE Directive Block "XSPD2" contains a Subcircuit SPICE netlist of primary-side MOSFET IRFH5053PBF.

- A SPICE Directive Block "XSPD2" contains a Subcircuit SPICE netlist of secondary-side synchronous MOSFET AUIRFR8401.
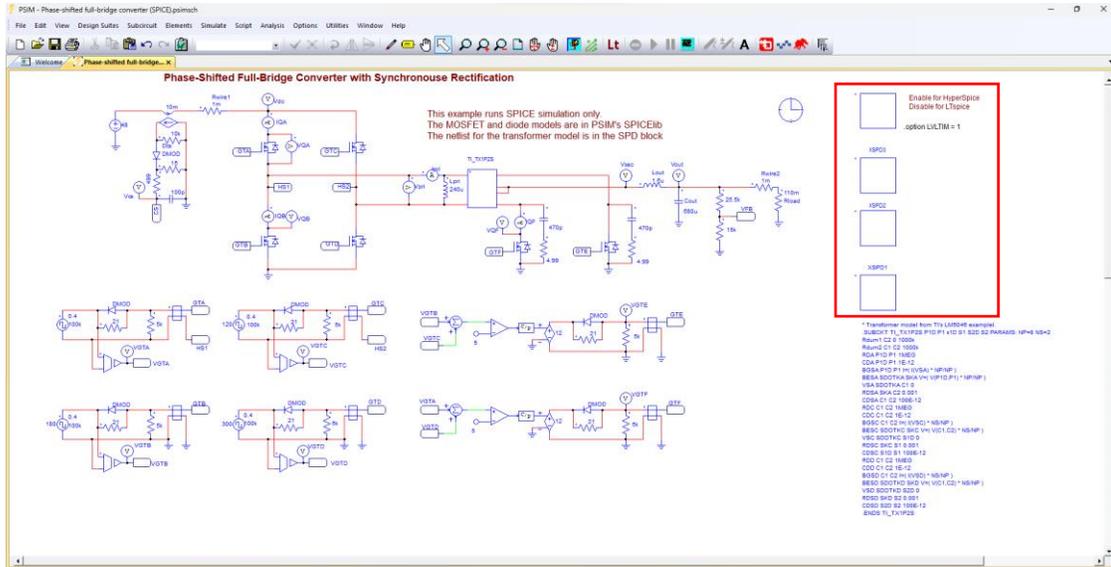
For this example, the transient analysis will be performed with the settings shown below.
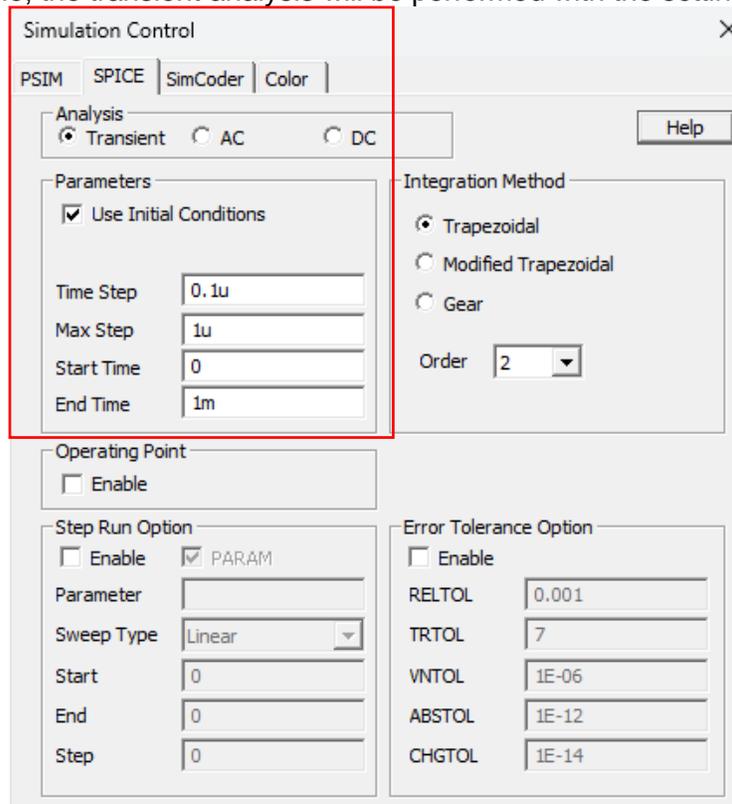
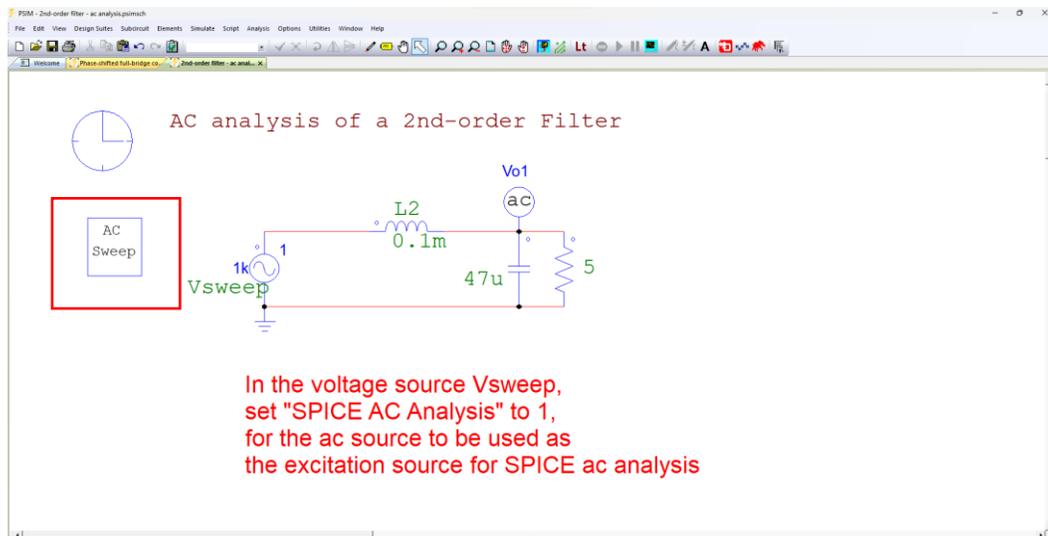Run HyperSpice Simulation, the results will be as shown below.
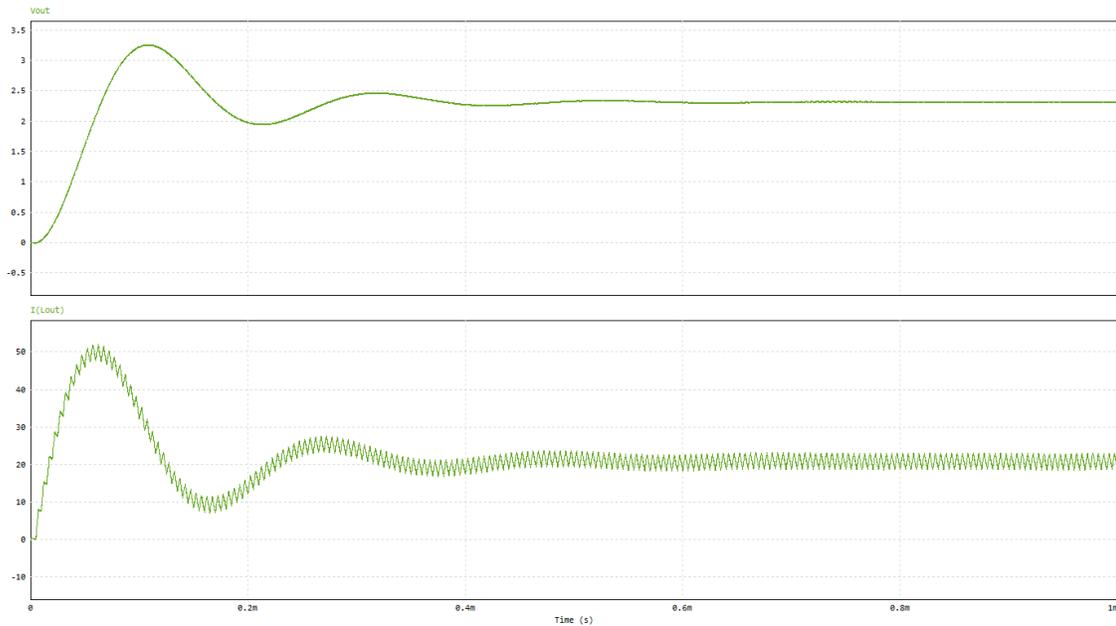
Please Note: Most SPICE examples in PSIM are supported in both HyperSpice and LTSpice. The simulation performance is related to the time-step control algorithm which can be managed by Simulation Control. In some SPICE examples, it is possible to accelerate the simulation by adjusting the tolerances which can reduce the simulation time.

3.2 AC: Frequency response simulation

To run .AC HyperSpice simulation; click "File >> Open Examples…" then go to the subfolder *"C:\PSIM 2022.3.0.52\examples\SPICE Link\ac and dc analysis\2nd-order filter - ac analysis.psimsch"*. Select the example file "*2nd-order filter - ac analysis.psimsch*" and use "File >> Save As…" to save the file into the new folder.

The example circuit is shown below. It is a basic 2nd-order LC filter circuit with ac sweep Spice simulation. The sweep type used is Decade from 100Hz to 100kHz.

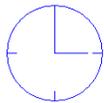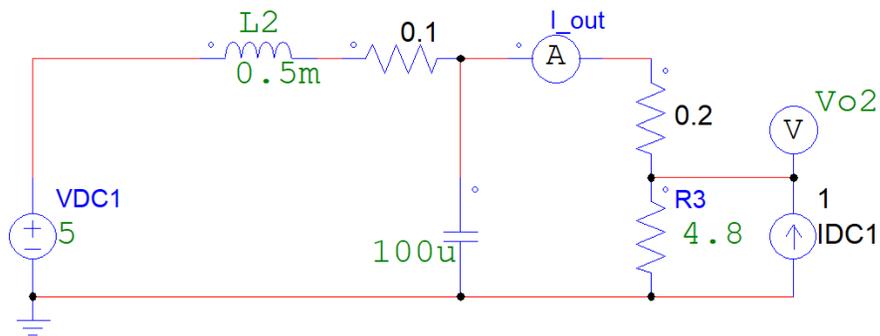For this example, the .AC analysis will be performed with the settings shown below.
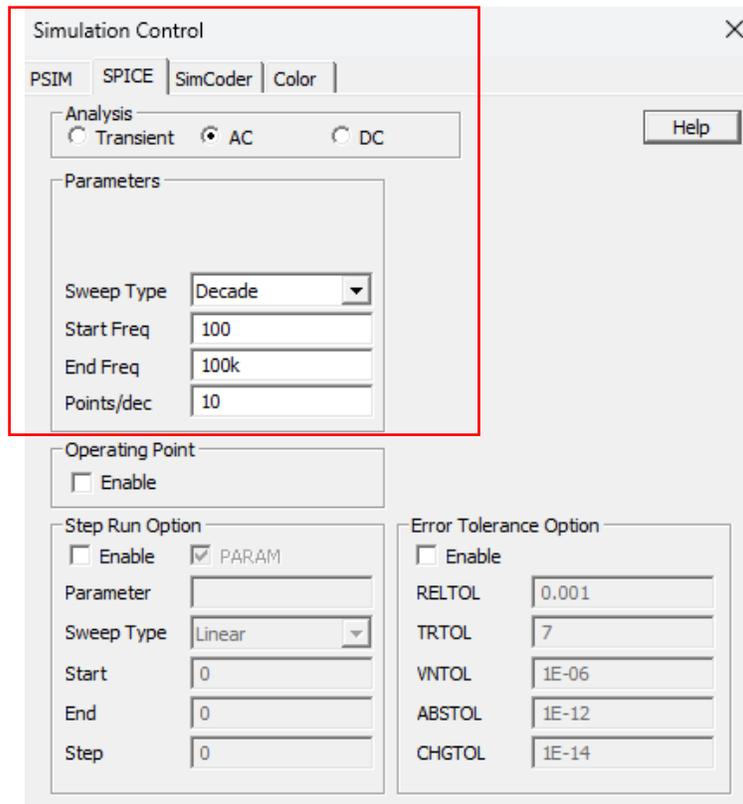
Run HyperSpice Simulation, the ac sweep results will be as shown below.
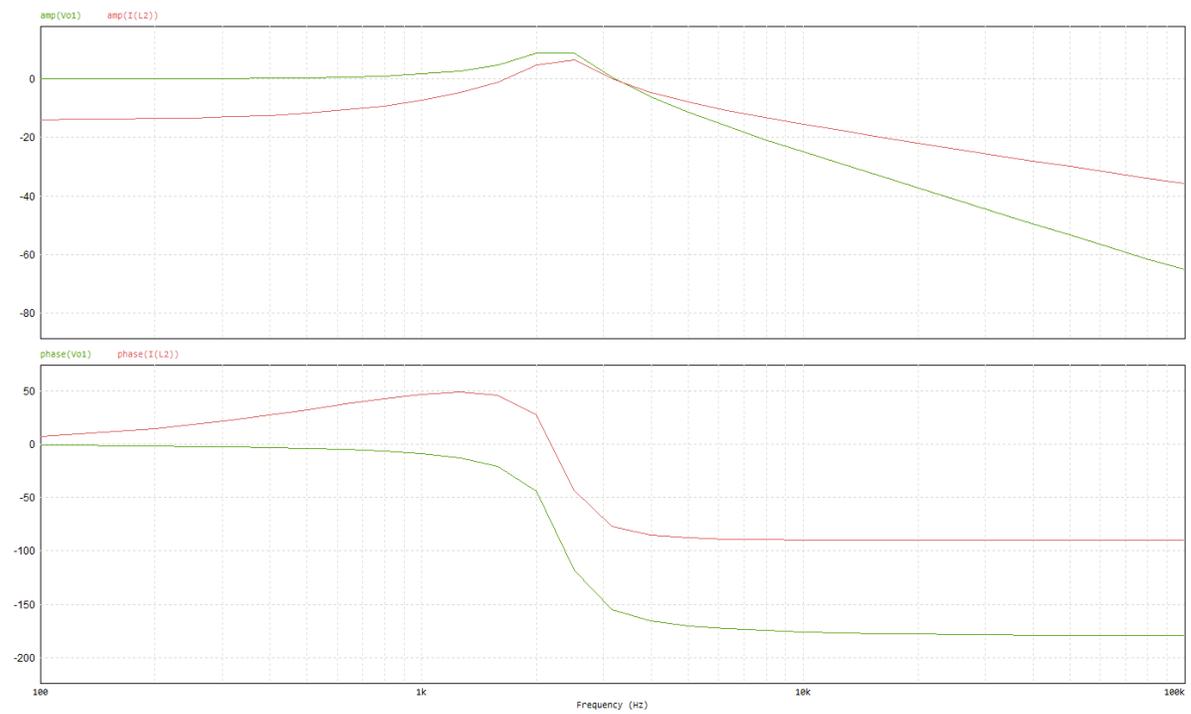
3.3 DC: Operating point simulation

To run .DC HyperSpice simulation; click "File >> Open Examples…" then go to the subfolder *"C:\PSIM 2022.3.0.52\examples\SPICE Link\ac and dc analysis\2nd-order filter - dc analysis V and I.psimsch"*. Select the example file "*2nd-order filter - dc analysis V and I (SPICE).psimsch*" and use "File >> Save As…" to save the file into the new folder.

The example circuit is shown below. It is a basic 2nd-order LC filter circuit with ac sweep Spice simulation. The sweep type used is Linear for both Source 1 (V) and Source 2 (I).
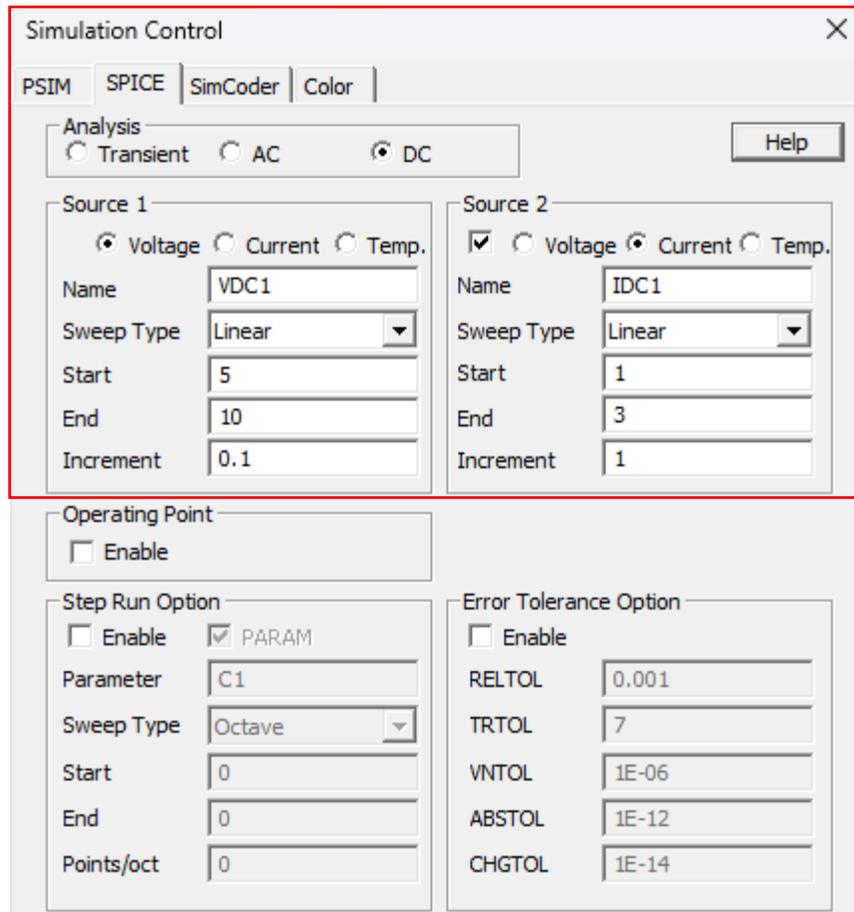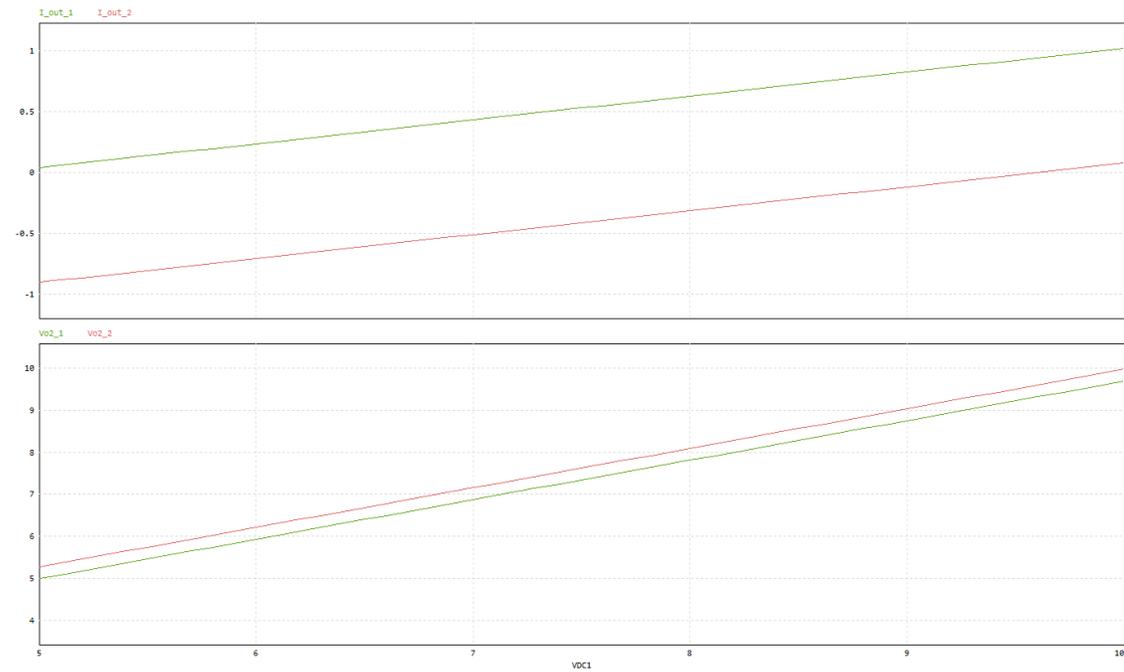

For this example, the .DC analysis will be performed with the settings shown below.

Run HyperSpice Simulation, the DC sweep results will be as shown below.

4. Export SPICE Netlists

Users can capture a PSIM schematic and export it as a SPICE netlist by clicking **Simulate** >> **Generate SPICE Netlist (.cir)**. The netlist file thus generated can be loaded into HyperSpice for simulation.

The netlist file with the same name as the PSIM schematic would be overridden each time "Generate SPICE Netlist" is clicked or each time the HyperSpice simulation is run.

5. Import SPICE Netlists

Pre-existing netlists, either written by hand or generated from other schematic capture software, can be loaded into PSIM and run HyperSpice simulation.

On the PSIM menu, click "**File >> Open SPICE Netlist File**" to load the Netlist file into PSIM, and then, run the HyperSpice simulation.

6. SPICE Netlist Format

PSIM provides the convenience of running HyperSpice simulation with PSIM schematics. The SPICE netlist syntax and functions must be supported by HyperSpice.

Some SPICE netlist files from other sources may contain syntaxes and/or functions which are not supported by HyperSpice. In such a case, one needs to convert these statements manually.