

TPS7A4501-SP Model User's Guide

This user guide is intended to demonstrate use of the Pspice model for the TPS7A4501-SP low-dropout linear regulator. Instructions on how to import the unencrypted model netlist into Cadence Pspice® are also provided. The first half of the guide outlines the modeled parameters and the second half of the guide addresses how to simulate the modeled parameters.

Contents

1	TPS7A4501-SP Model Specification	2
2	Example of using model with Cadence Pspice (17.2.0)	3
3	Simulation of the TPS7A4501-SP model.....	11

List of Figures

1	Model Schematic	10
2	Output Voltage Monte Carlo Histogram	14
3	Frequency Response Bode Plot	18
4	Frequency Response Histogram	19

Trademarks

Pspice, Capture are registered trademarks of Cadence.
All other trademarks are the property of their respective owners.

1 TPS7A4501-SP Model Specification

The netlist file (TPS7A4501-SP.lib) contains the spice model of the device TPS7A4501-SP. The model is intended for following types of simulation:

- Frequency response (Phase Margin, Phase Margin Crossover, and Gain Margin)
- Transient response
- DC characteristics

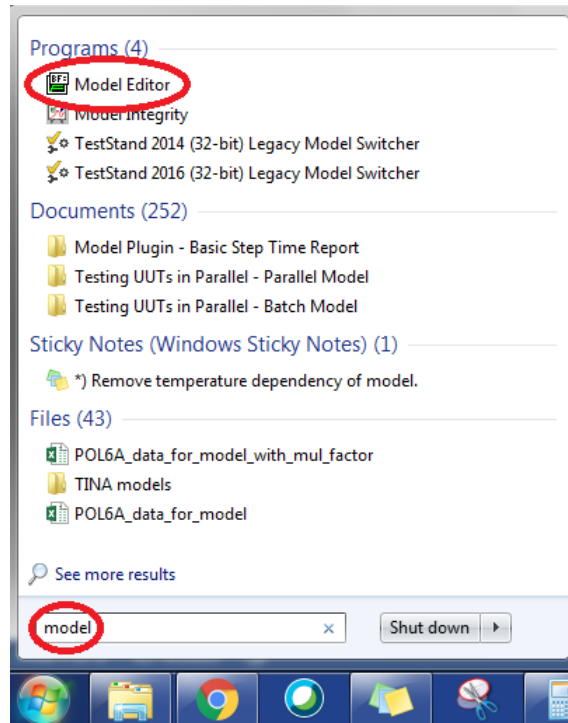
The model includes Monte Carlo to capture device-to-device variation and support simulation over the full military temperature range, -55°C to 125°C . As performance of the device is relatively independent of supply voltage, any variation due to supply voltage is modeled as part of Monte Carlo. The performance variation in the device due to radiation exposure is also modeled as part of Monte Carlo. Any convergence errors during simulation can be cleared by varying the simulation parameters. The supply current during shutdown condition has not been modeled. Please refer to the data sheet for precise information.

2 Example of using model with Cadence Pspice (17.2.0)

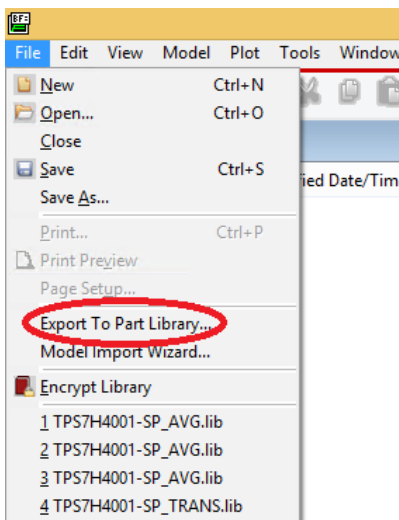
2.1 Creating a part from the netlist file

In order to run a simulation, the netlist file must be used to create a part that can be used in the schematic.

1. First, open the "Model Editor" Application.



2. Then select *File* → *Open* and choose the netlist file (*.lib).
3. Once the netlist opens, select *File* → *Export to Part Library...*

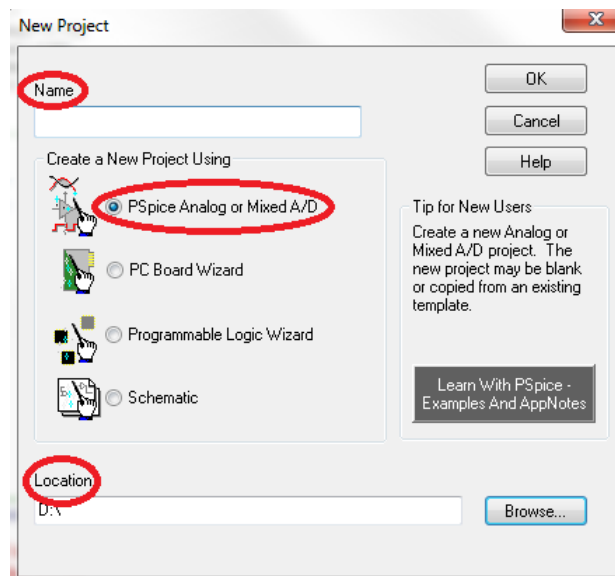


4. The *Create Parts for Library* dialogue box will open. Click "OK" to generate the part library in the default location. This will create a *.olb file with the same name and location as the netlist file.

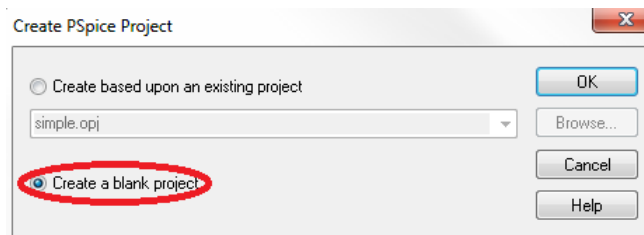
2.2 Steps to create project in Capture®

The following steps explain the procedure for creating a project in Capture and adding the part created from the netlist:

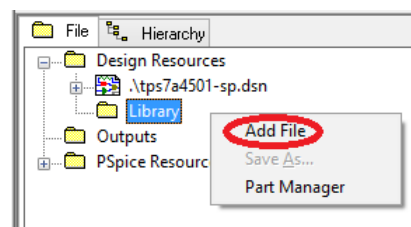
1. Open the Capture application from Cadence - v17.2.0 or above.
2. Click on *File* → *New* → *Project*.
3. Enter a project name and location, choose *PSpice Analog or Mixed A/D* from the options and click "OK".



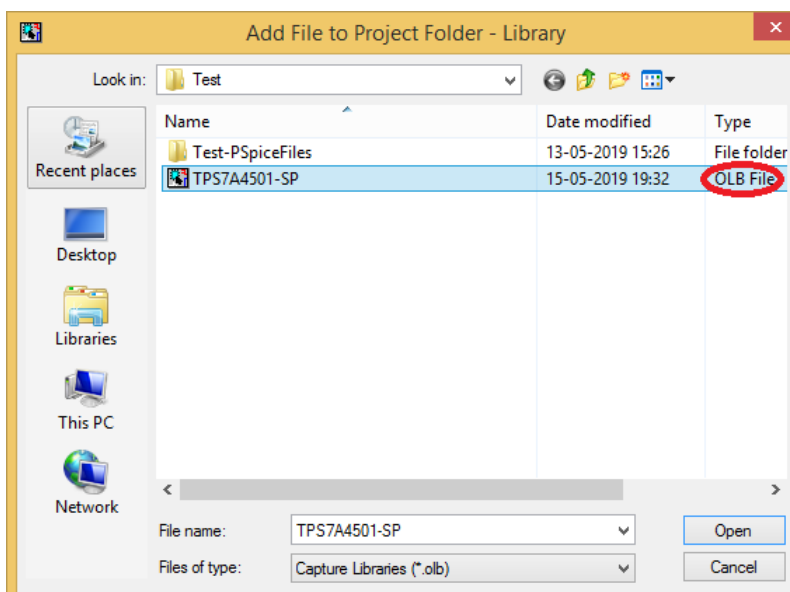
4. Once the *Create Pspice Project* dialogue box opens, select the "Create a blank project" option and click "OK".



5. A new project will be created and the project window will open. Right click on the *Library* folder and select "Add File".



- Choose the *****.olb** file that was previously created, add to the dialogue box, and click "Open".

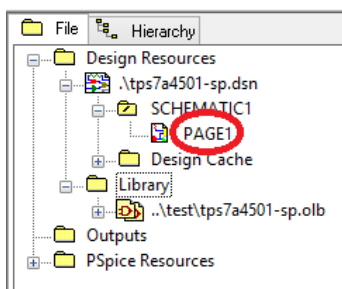


- This will add the part symbol to the project.

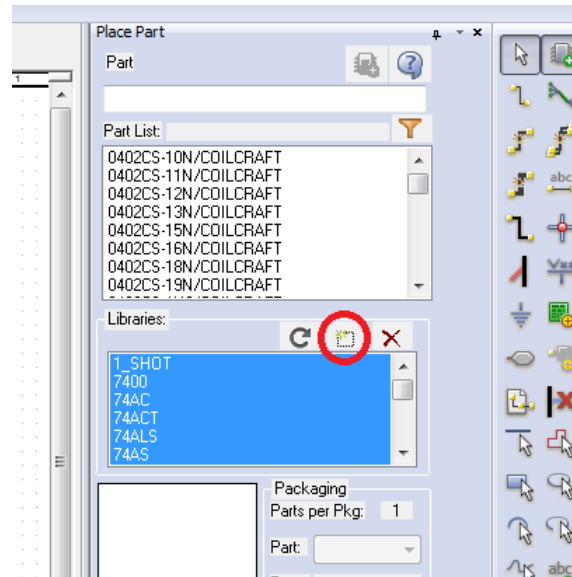
2.3 Creating the schematic in Capture

The following steps explain the procedure for developing a schematic in Capture using the part created from the netlist:

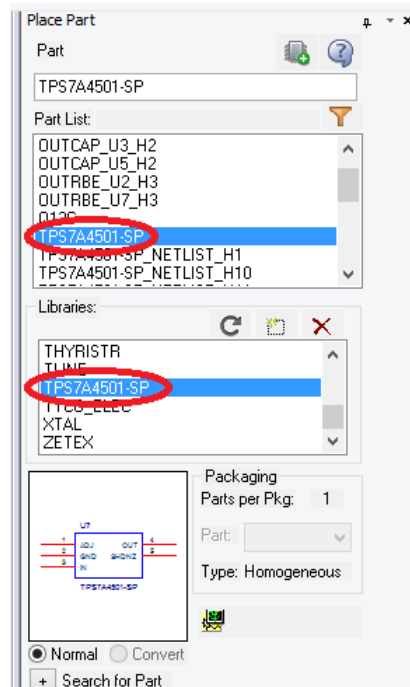
- Open **PAGE1** under the **SCHEMATIC1** folder below the **.dsn** file.



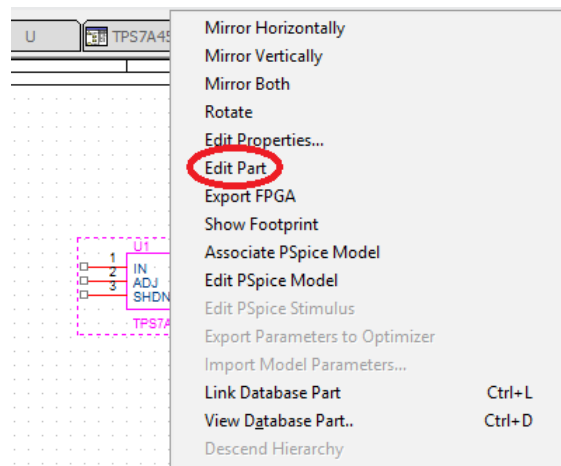
2. Select *Place* → *Part* to open the *Place Part* window. Click on the square icon shown in the following image to add libraries to the project.



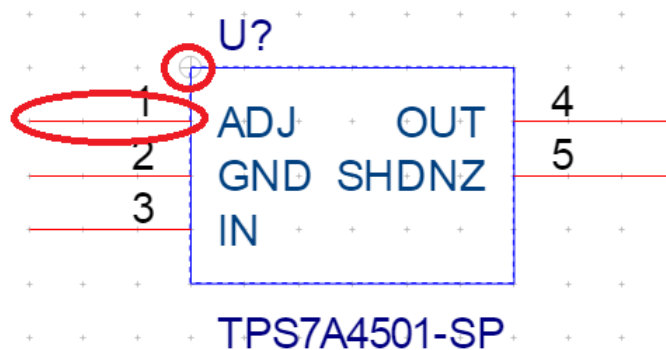
3. Select all default library files of PSPICE from installation directory and click "Open". (Default Location: "C:\Cadence\SPB_17.2\tools\capture\library\pspice") Note: This step can be omitted if the default libraries have already been added to Capture.
4. In the *Part* search box, type "TPS7A4501-SP" and select the model.



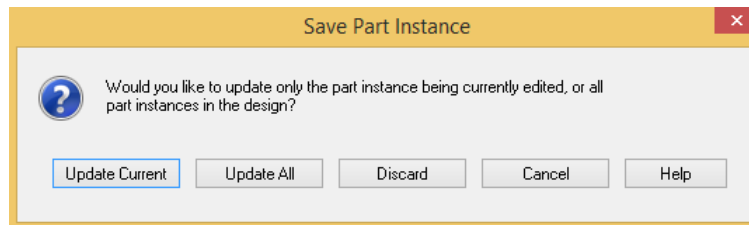
- Double click on the part in the *Part List* window and place it by left clicking the cursor on the PAGE1. Once the part is placed, the symbol can be edited by selecting the part, right clicking, and choosing "Edit Part".



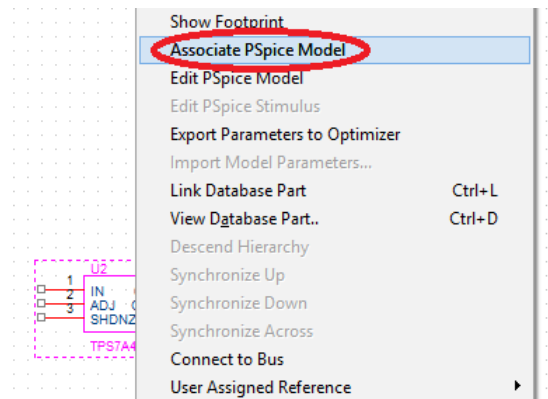
- The part can be resized and pins can be moved to more convenient locations by clicking and dragging them.



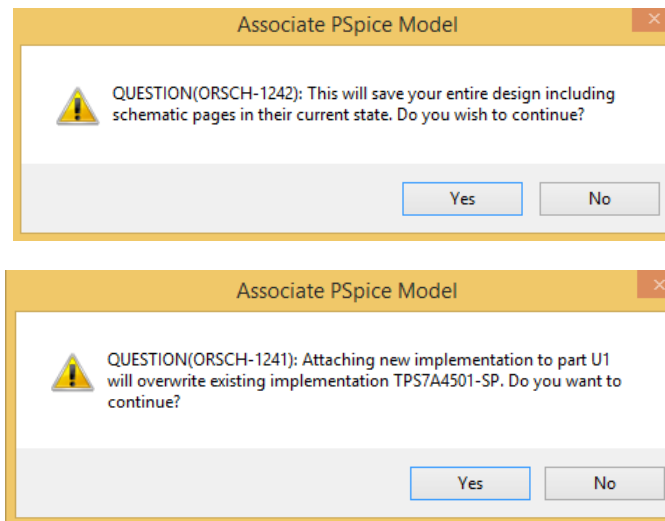
- After editing the part, close the tab to save changes. Choose "Update Current" in the pop-up.



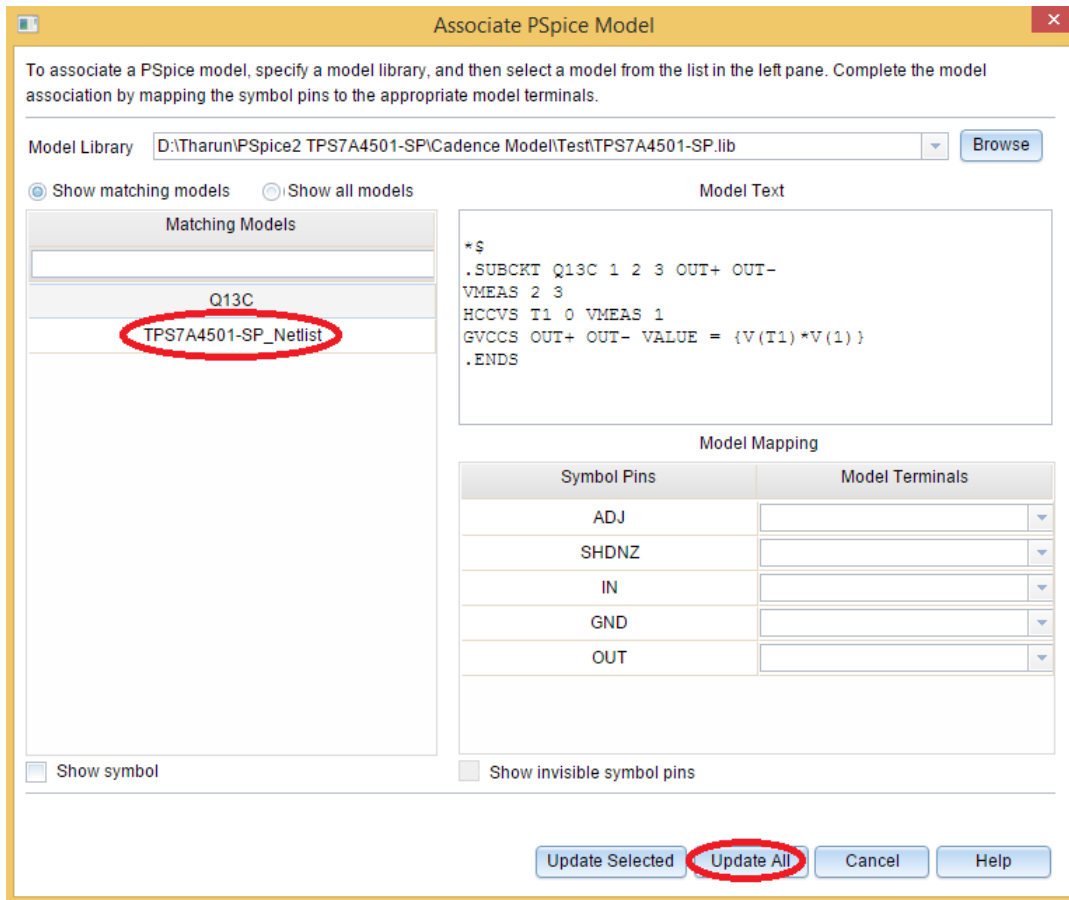
8. Now, the part must be associated with the netlist. Select the part, right click, and choose "Associate Pspice Mode".



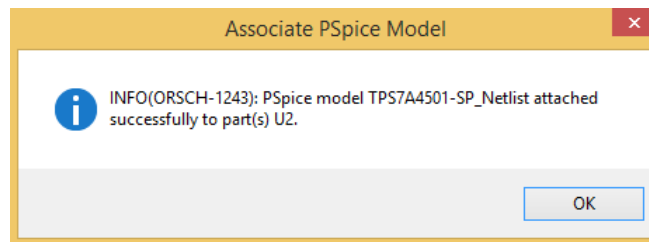
9. Two pop-ups will appear to confirm the operation. Click "Yes" for both.



10. In the *Associate Pspice Model* dialogue box, choose the netlist file (***.lib), select the model, and click "Update All."



11. A pop-up will appear with successful update message. Click "Ok".



12. Add the remaining components to complete the schematic as shown in [Figure 1](#).

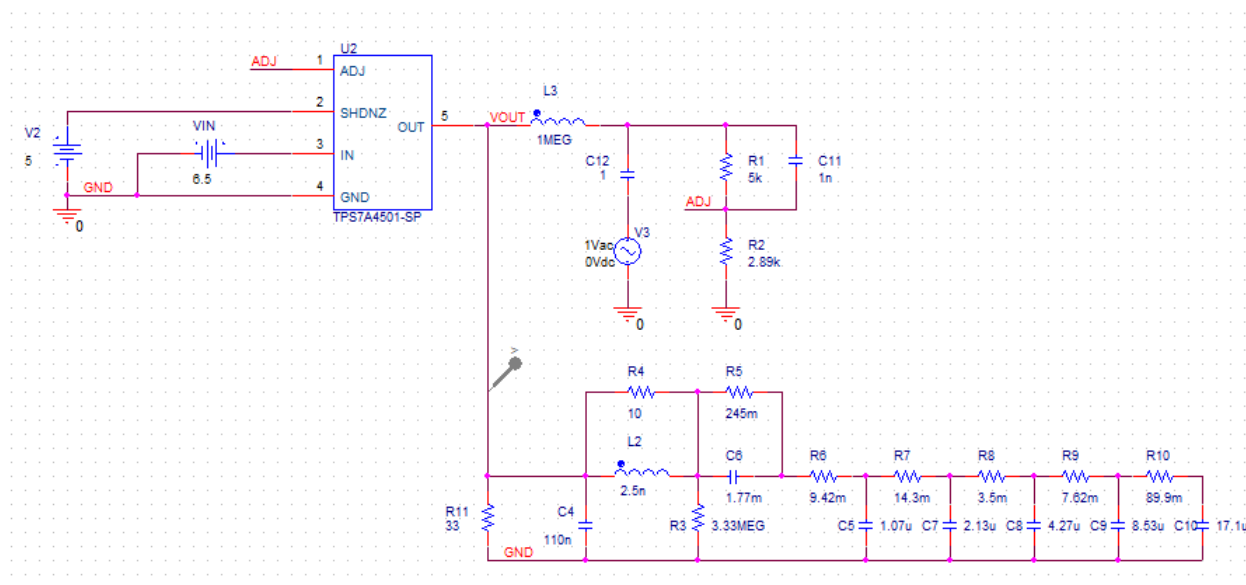
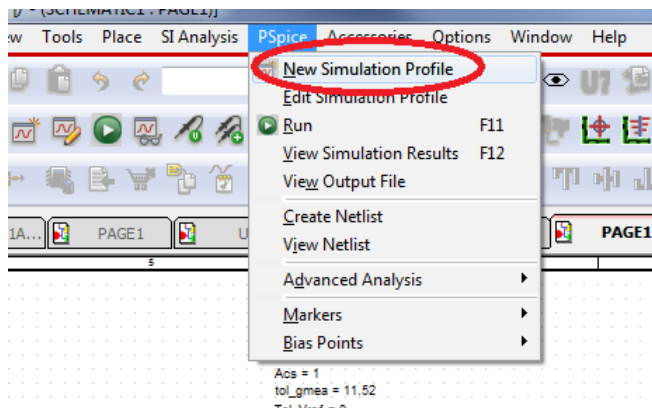


Figure 1. Model Schematic

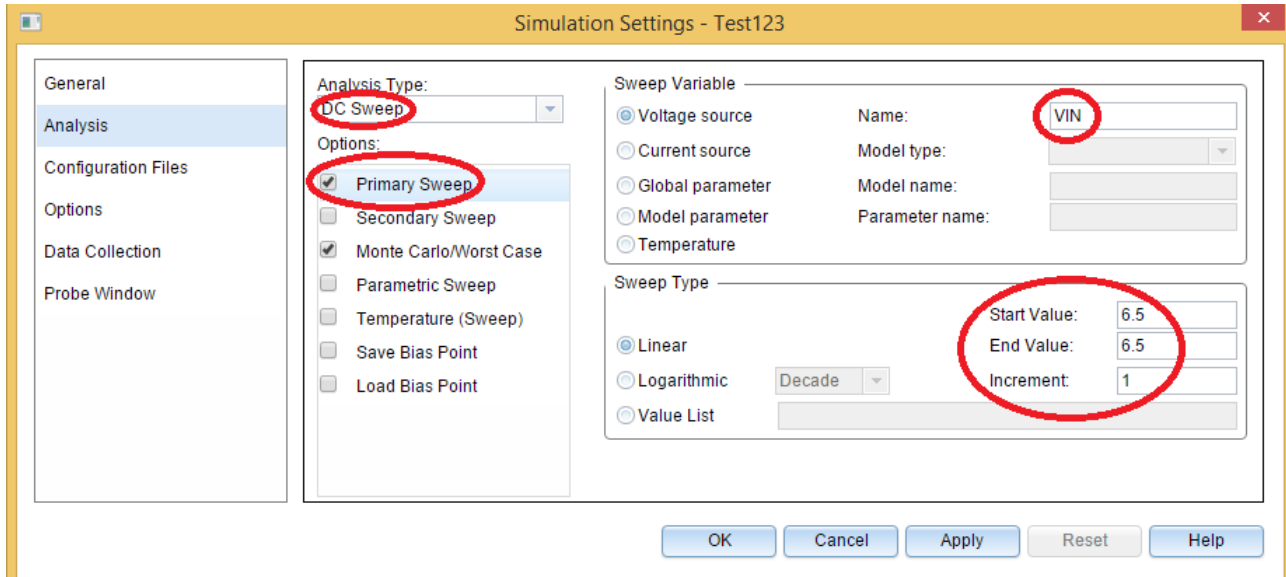
3 Simulation of the TPS7A4501-SP model

3.1 Monte Carlo analysis of output voltage

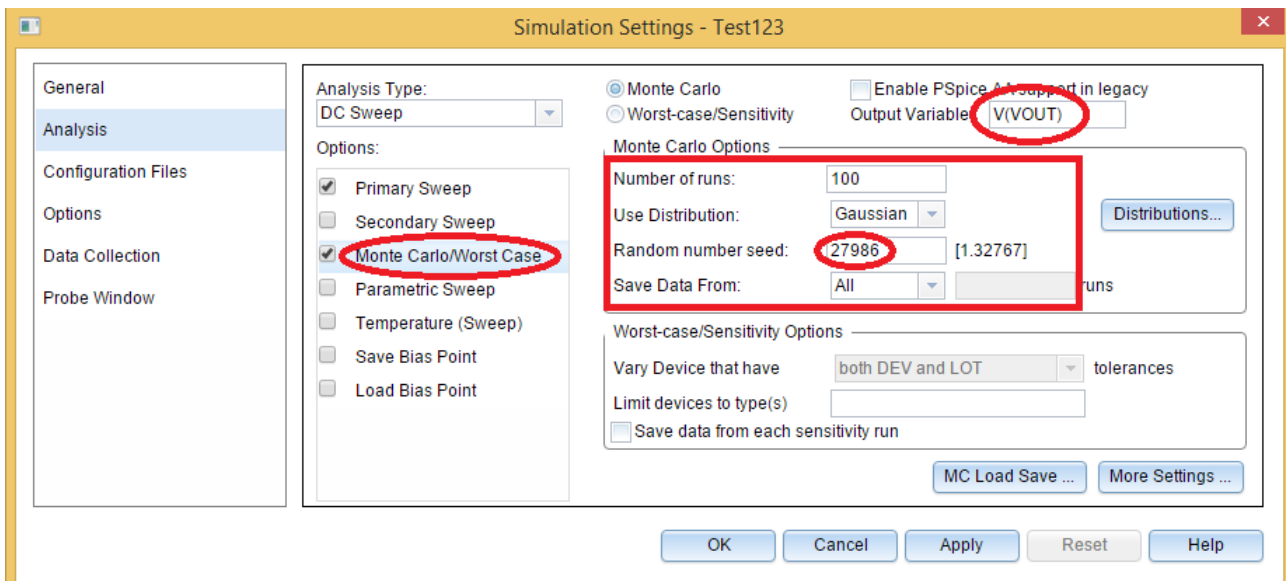
1. Create a new simulation profile by clicking on *PSpice* → *New Simulation Profile* and give it a name.



2. Set the parameters based on the image below using the following settings:
 - **Analysis Type** - DC-Sweep
 - **Primary Sweep** - VIN (Input Voltage) from 6.5 V to 6.5 V
 - **Monte Carlo** - 100 runs with Gaussian distribution
 - **Output Variable** - V(VOUT) - Output voltage



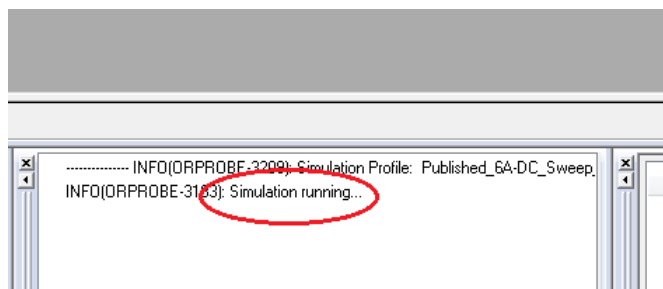
The image shows the 'Simulation Settings - Test123' dialog box. The 'Analysis Type' is set to 'DC Sweep'. Under 'Options', 'Primary Sweep' is checked. The 'Sweep Variable' is 'VIN' with a 'Start Value' of 6.5, 'End Value' of 6.5, and 'Increment' of 1. The 'Sweep Type' is 'Linear'.



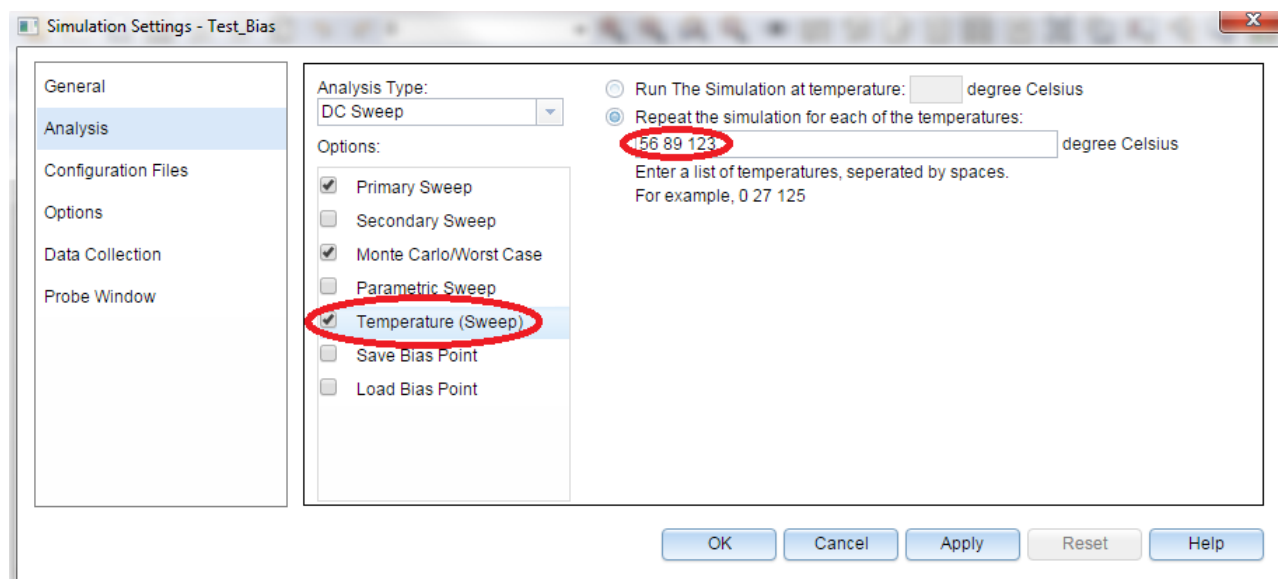
The image shows the 'Simulation Settings - Test123' dialog box. The 'Analysis Type' is set to 'DC Sweep'. Under 'Options', 'Monte Carlo/Worst Case' is checked. The 'Monte Carlo Options' section shows 'Number of runs' as 100, 'Use Distribution' as Gaussian, and 'Random number seed' as 27986. The 'Output Variable' is 'V(VOUT)'.

3. Set the random seed number for Monte Carlo within range shown to the right of the input box.
4. Run the simulation by pressing F11 or *PSPice* → *Run*.

- Wait for the simulation completion in the console window of AMS Simulator.



- The simulation can be performed at various temperatures by selecting temperature sweep in the simulation profile and setting the temperature values in ascending order.



7. The output window comes up once the simulation is completed. The result window shows the histogram of the output voltage (V(VOUT)) with Monte Carlo analysis. The mean, sigma, min, max, etc. are shown in the bottom of the window.

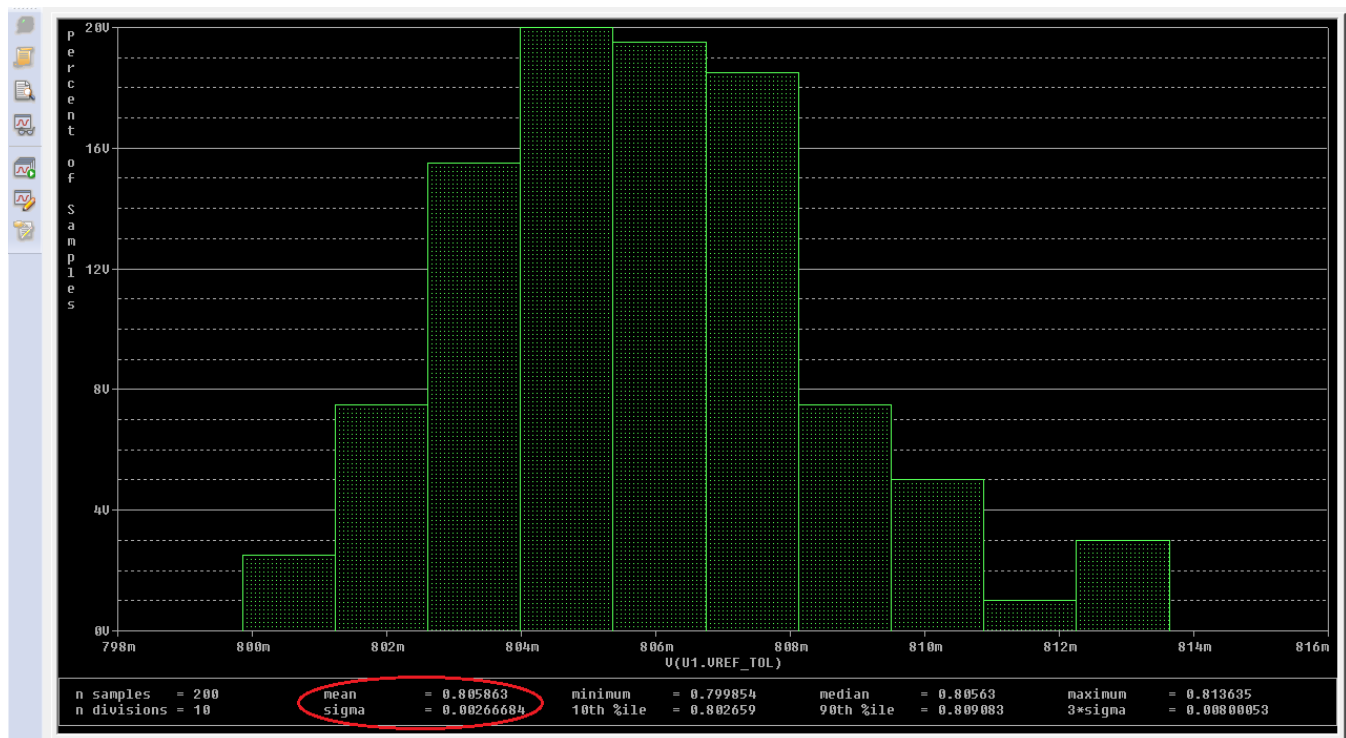
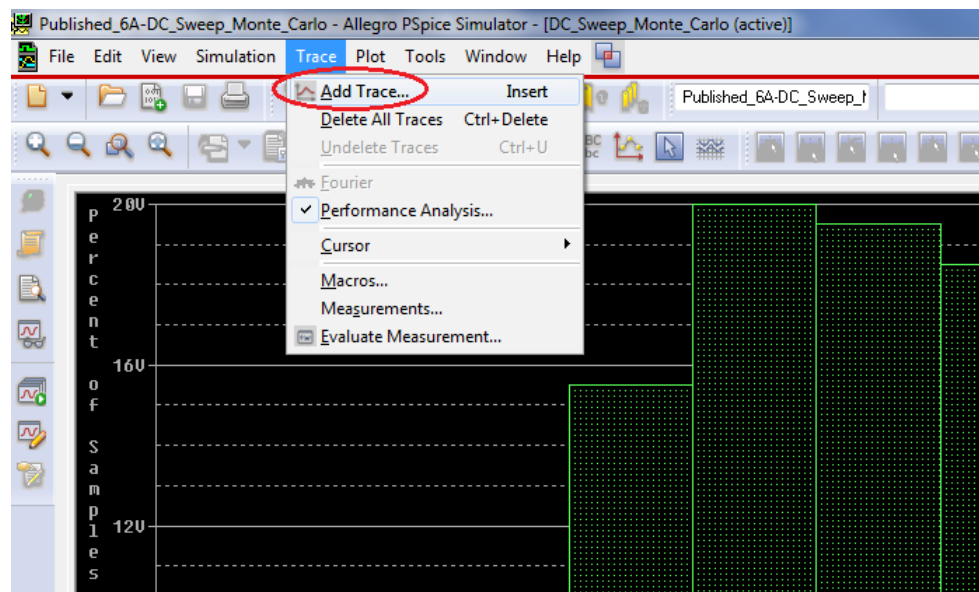
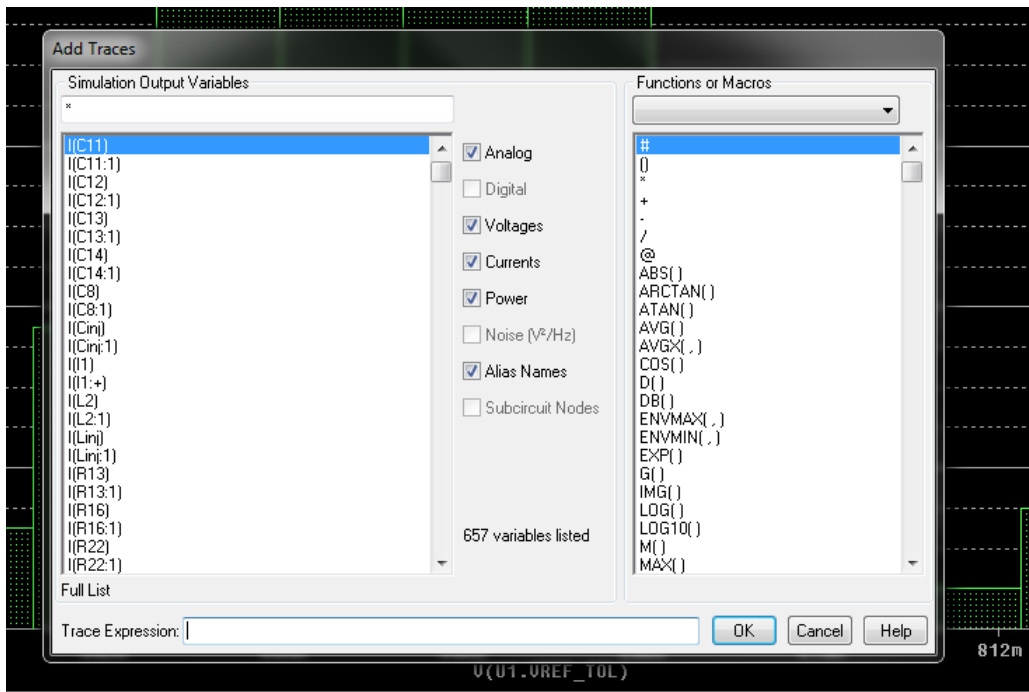


Figure 2. Output Voltage Monte Carlo Histogram

8. To analyze other nodes, click on *Trace* → *Add Trace*.



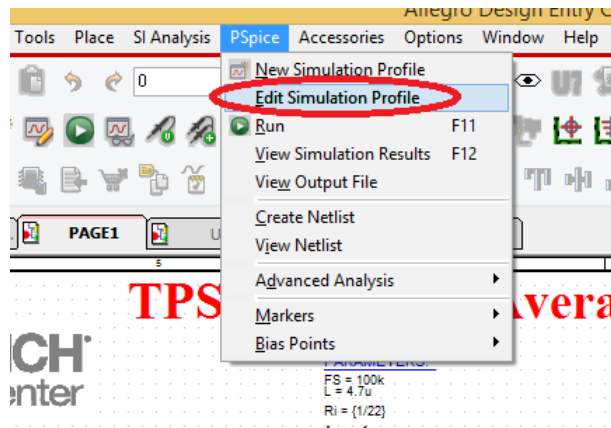
This will bring in another pop-up with the netlist present in the schematic.



Select the desired netlist and click "OK". The histogram of the selected node will be updated.

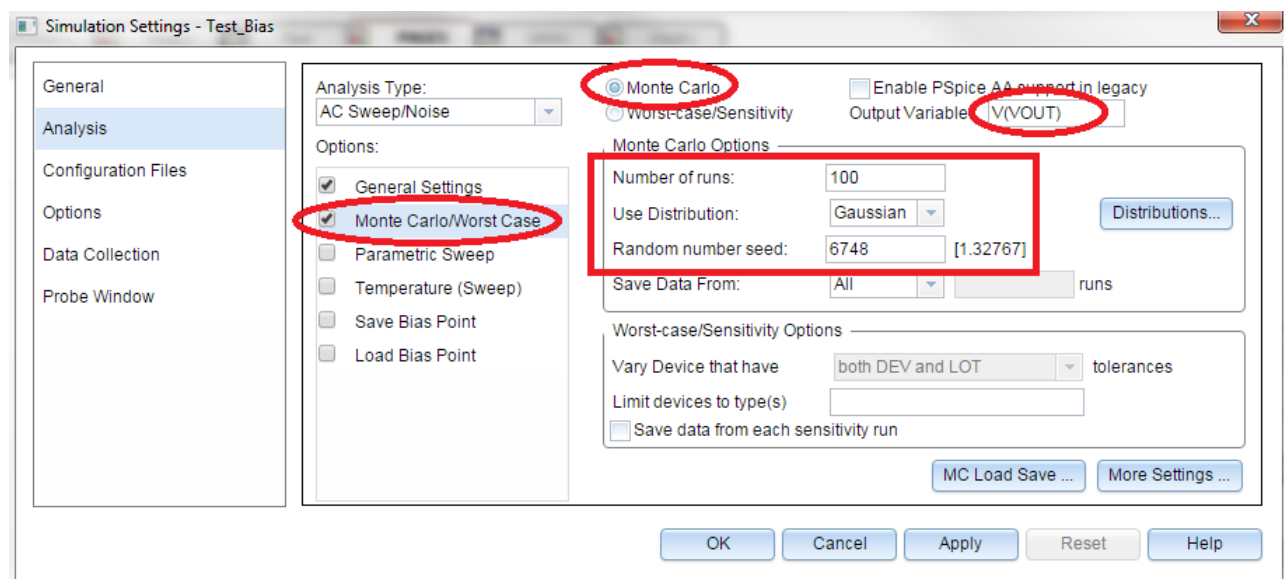
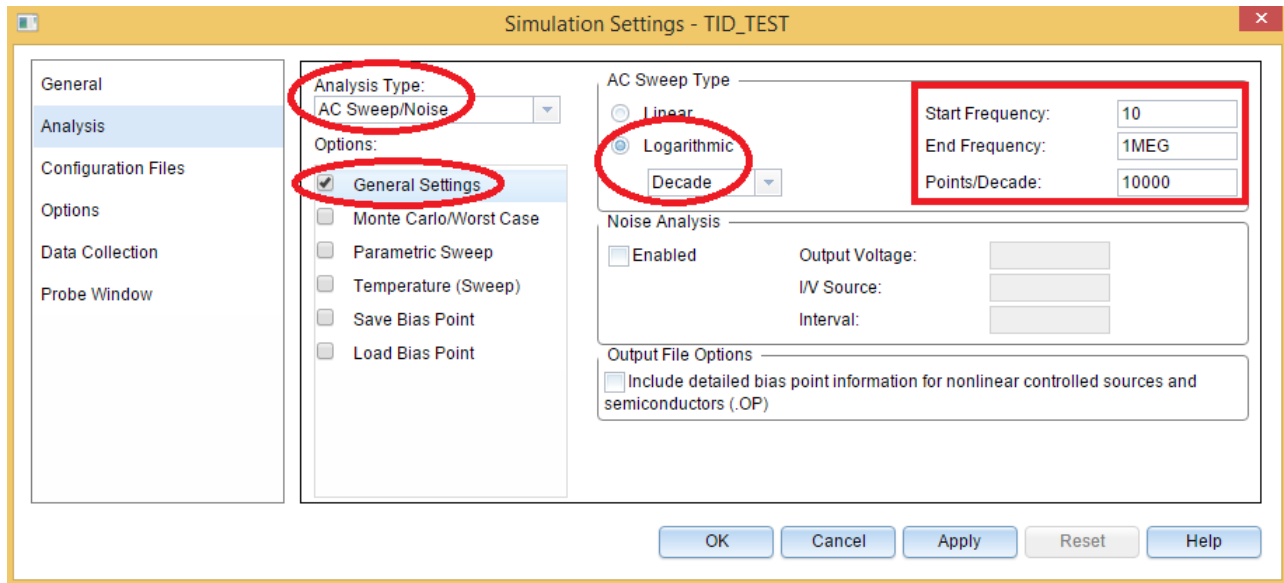
3.2 Performing frequency analysis

1. Create a new simulation profile or edit an existing simulation profile by clicking on *PSpice* → *Edit Simulation Profile*.



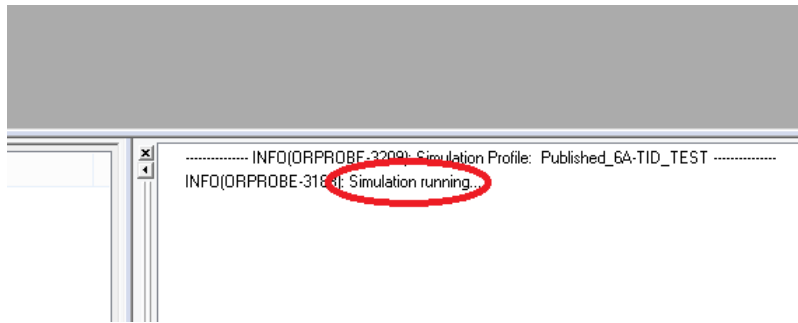
2. Set the following parameters as shown:

- **Analysis Type** - AC-Sweep
- **General Settings** - Set start and end frequency, number of points per decade and sweep type
- **Monte Carlo** - 500 runs with Gaussian distribution
- **Output Variable** - V(VOUT) - Output Voltage



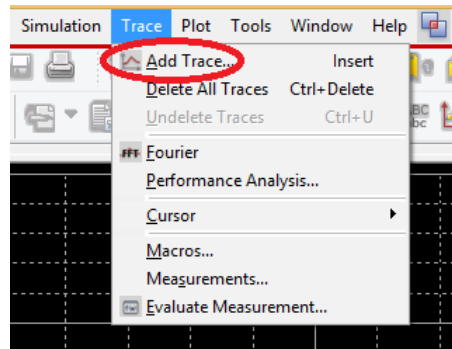
3. Run the Simulation by pressing F11 or *Pspice* → *Run*.

- Wait for the simulation completion in the console window of AMS Simulator.

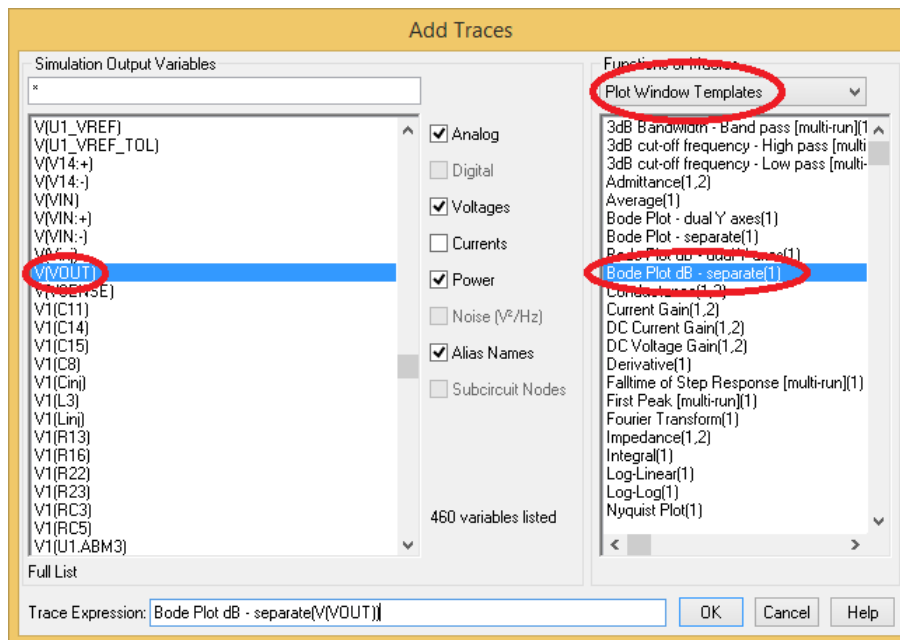


3.2.1 Analyzing frequency response with Bode plot

- To view the frequency response as Bode plot, click on *Trace* → *Add Trace*.



- This will bring in another pop-up with the netlist present in the schematic. Select the following options to get the Bode plot.



- Use the cursor to evaluate the plot.

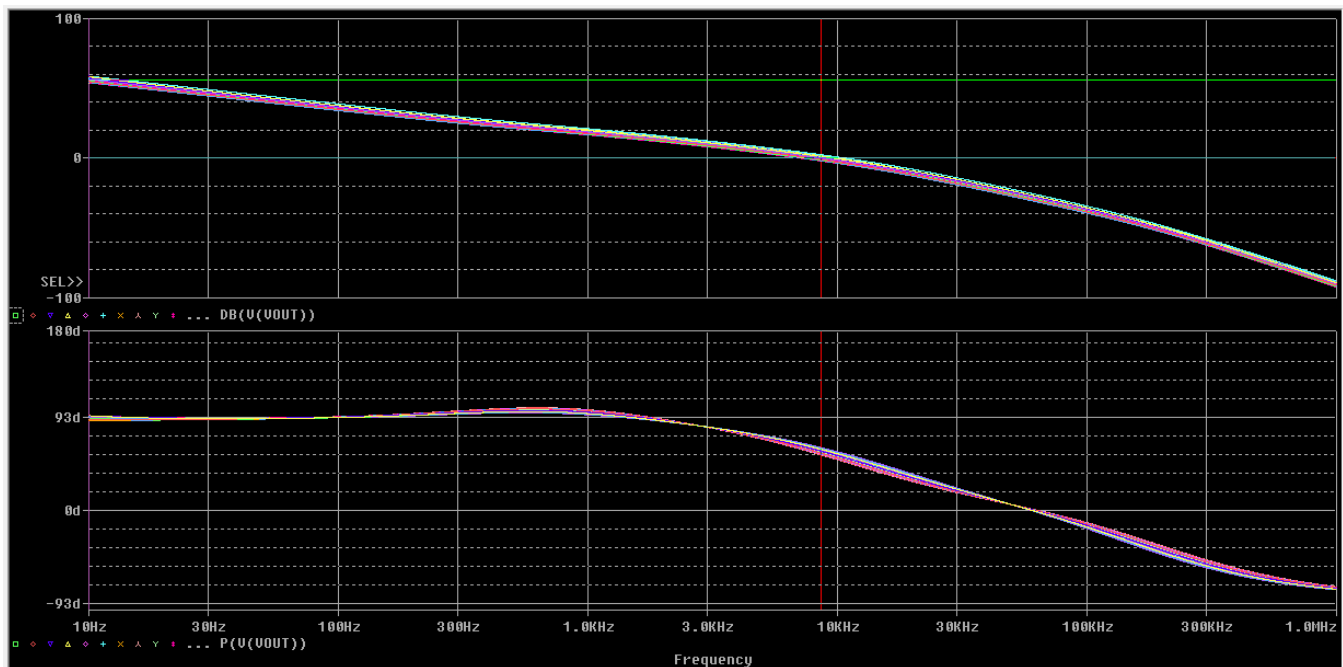
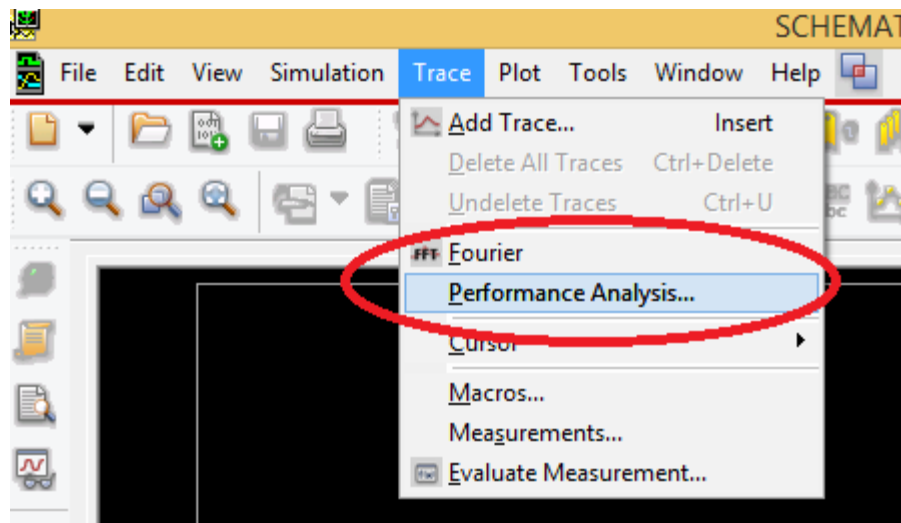


Figure 3. Frequency Response Bode Plot

3.2.2 Analyzing frequency response with histogram

- Select *Trace* → *Performance Analysis*. A pop-up will appear, click "OK".



- Right click on the plot area and click on "Add Trace".

- Copy the expression **PhaseMargin(DB(v(vout)),P(v(vout)))** into the *Trace Expression* field and click "OK". This will generate the histogram for the frequency response as shown in [Figure 4](#).

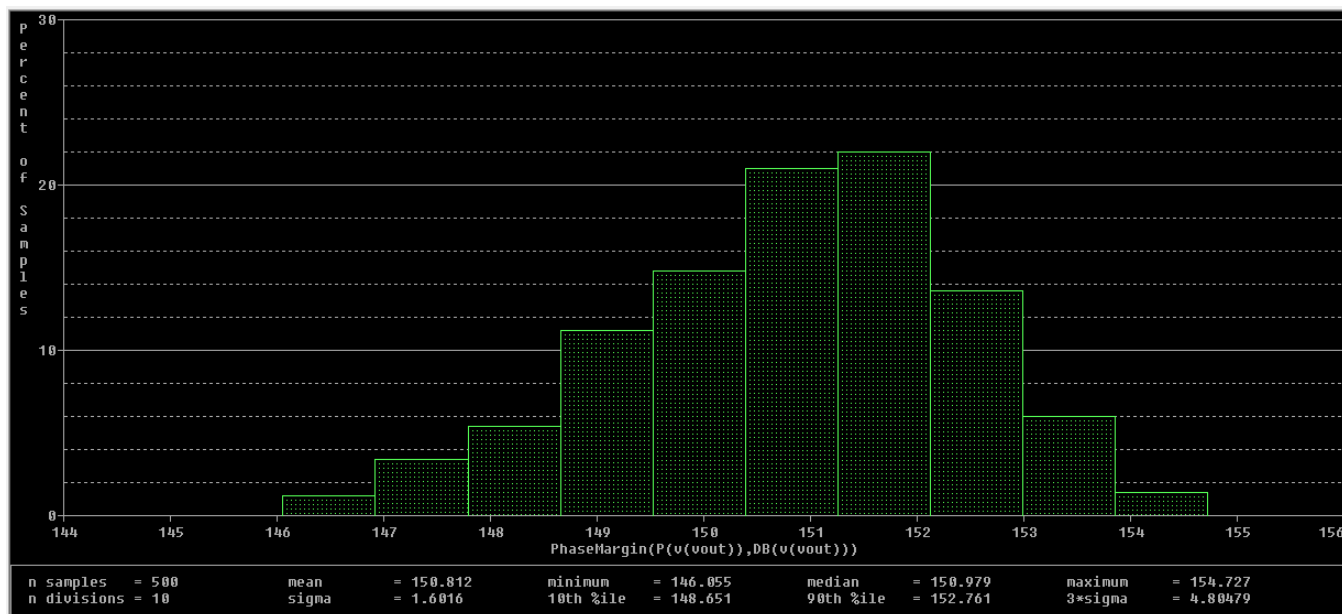


Figure 4. Frequency Response Histogram

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to TI's Terms of Sale (www.ti.com/legal/termsofsale.html) or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2019, Texas Instruments Incorporated