TEXAS INSTRUMENTS

User's Guide
SLVUBT2–October 2019

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

List of Figures

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

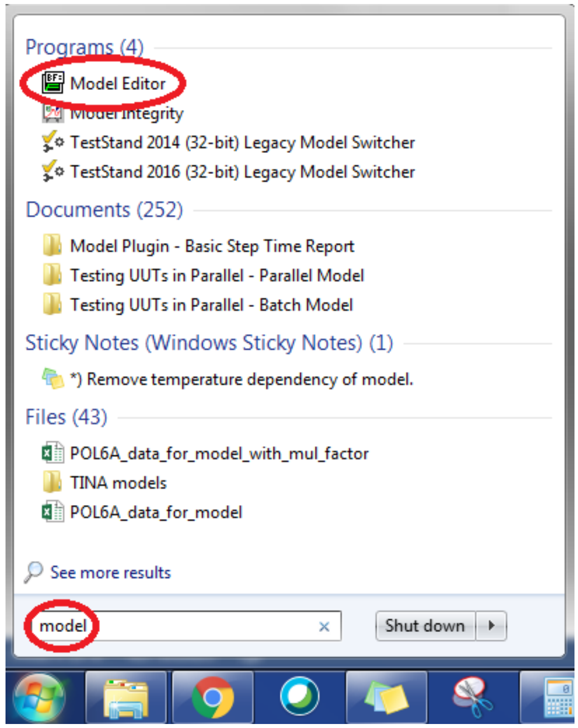
The model includes Monte Carlo to capture device-to-device variation and support simulation over the full military temperature range, –55°C to 125°C. As performance of the device is relatively independent of supply voltage, any variation due to supply voltage is modeled as part of Monte Carlo. The performance 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)

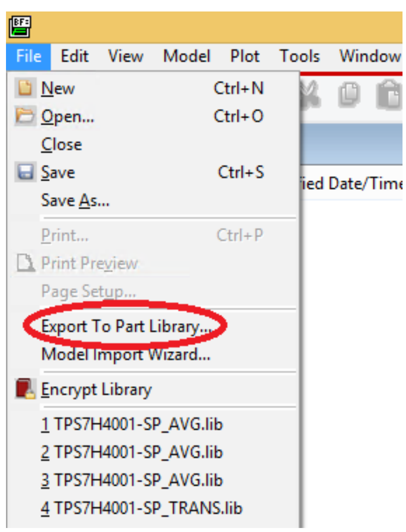
2.1 Creating a part from the netlist file

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...*

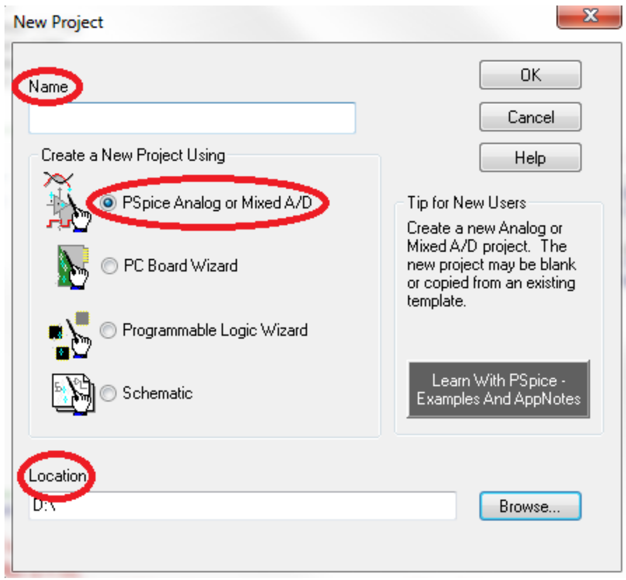


4. The *Create Parts for Library* dialogue box will open. Click "OK" to generate the part library in the default location. This will create a ***.olb file with the same name and location as the netlist file.

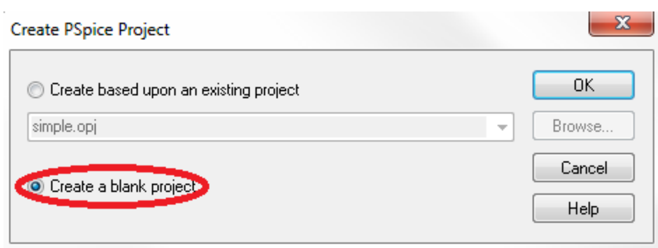
2.2 Steps to create project in Capture®

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".

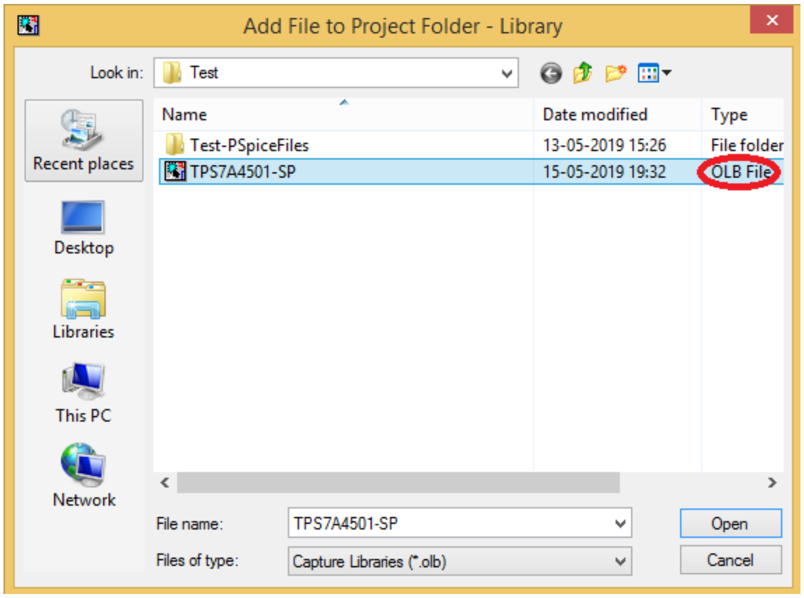


4. Once the *Create Pspice Project* dialogue box opens, select the "Create a blank project" option and click "OK".



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

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

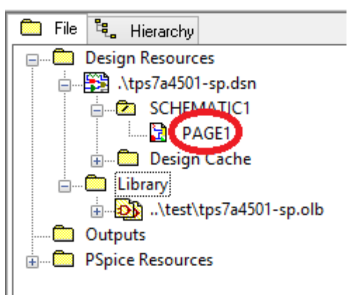


7. This will add the part symbol to the project.

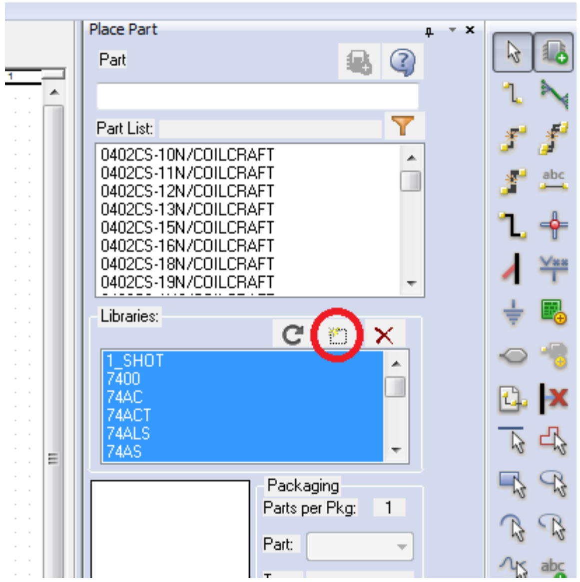
2.3 Creating the schematic in Capture

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

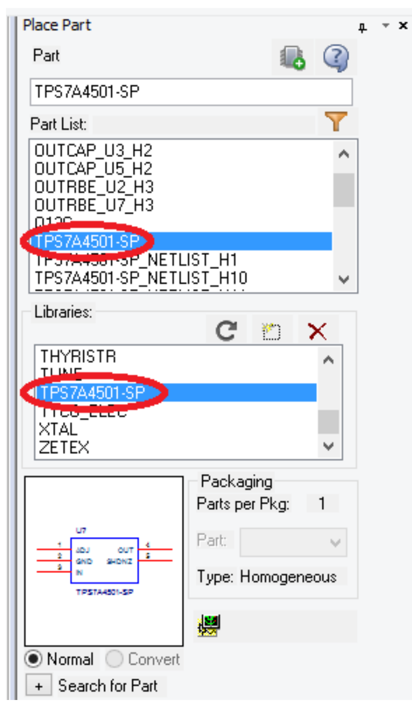
1. Open *PAGE1* under the *SCHEMATIC1* folder below the .dsn file.



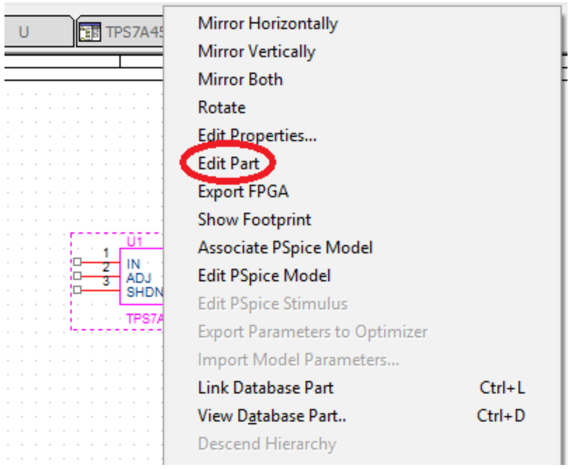
2. Select *Place* → *Part* to open the *Place Part* window. Click on the square icon shown in the following image to add libraries to the project.



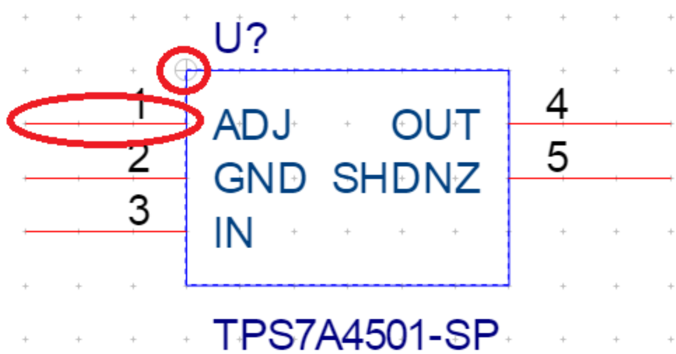
3. Select all default library files of PSPICE from installation directory and click "Open". (Default Location: “C:\Cadence\SPB_17.2\tools\capture\library\pspice”) Note: This step can be omitted if the default libraries have already been added to Capture.
4. In the *Part* search box, type “TPS7A4501-SP”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



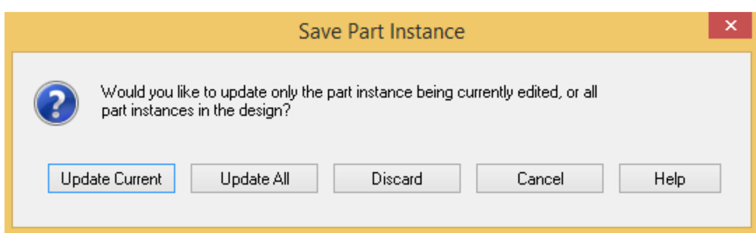
5. Double click on the part in the *Part List* window and place it by left clicking the cursor on the PAGE1. Once the part is placed, the symbol can be edited by selecting the part, right clicking, and choosing "Edit Part".



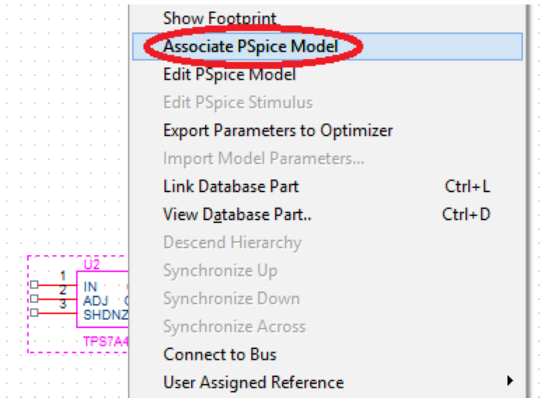
6. The part can be resized and pins can moved to more convenient locations by clicking and dragging them.



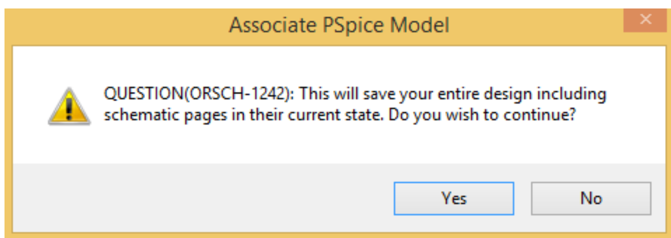
7. After editing the part, close the tab to save changes. Choose “Update Current” in the pop-up.

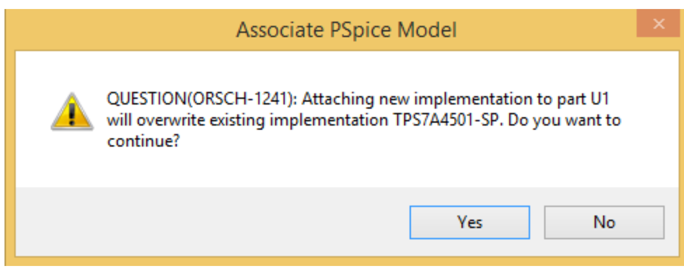


8. Now, the part must be associated with the netlist. Select the part, right click, and choose "Associate Pspice Mode".

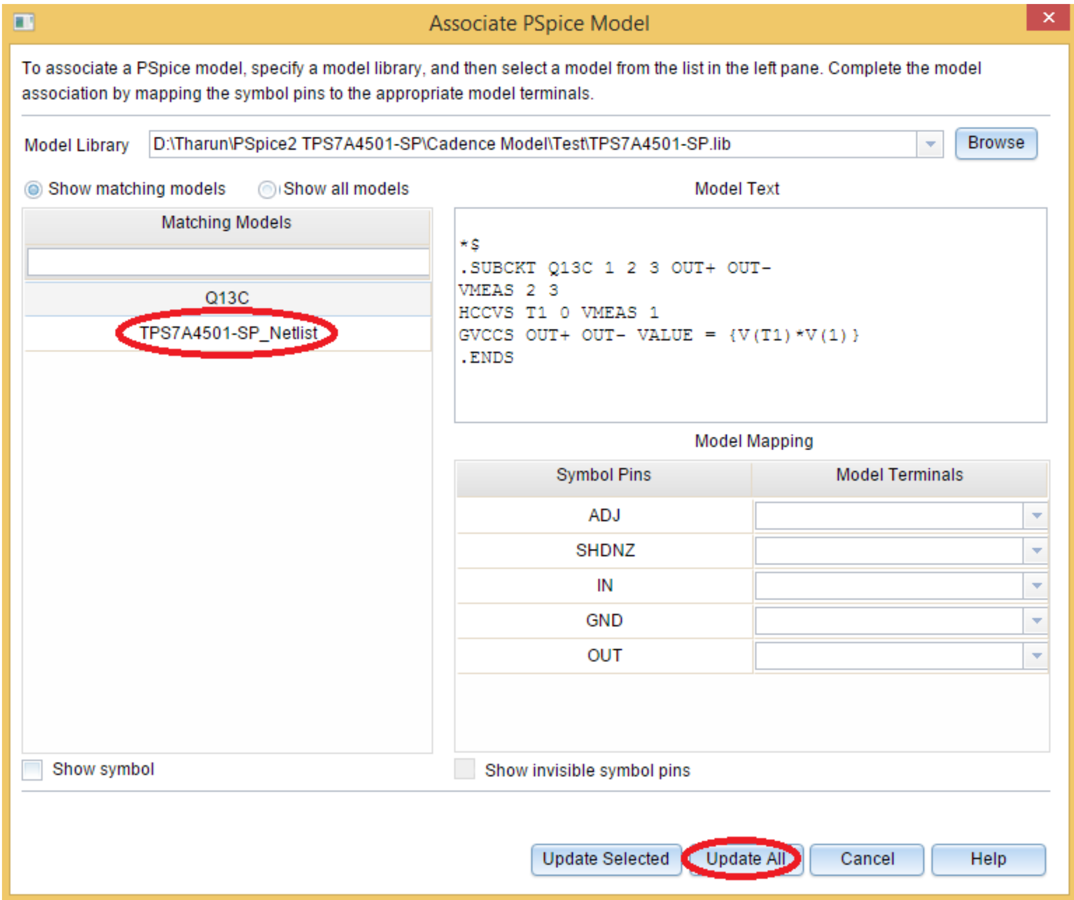


9. Two pop-ups will appear to confirm the operation. Click "Yes" for both.

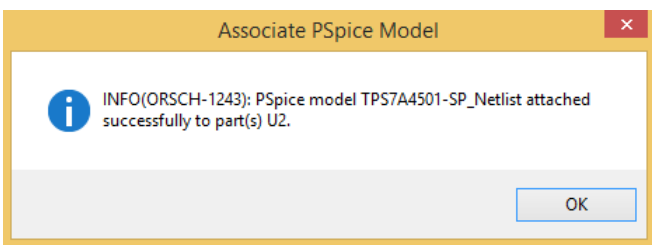




10. In the *Associate Pspice Model* dialogue box, choose the netlist file (***.lib), select the model, and click "Update All."



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.

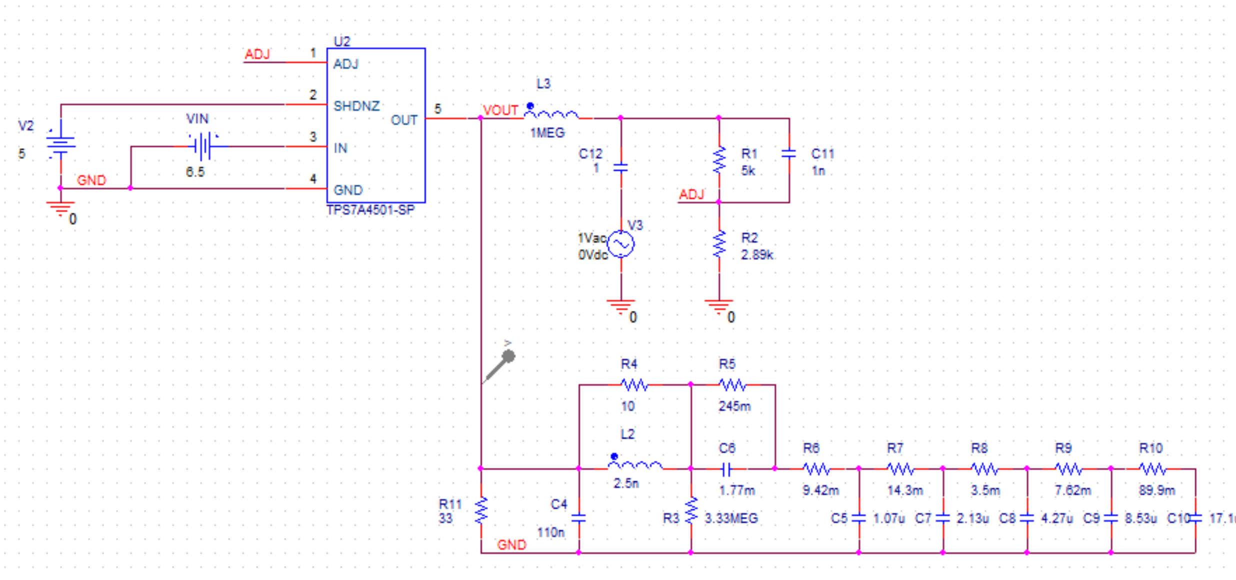
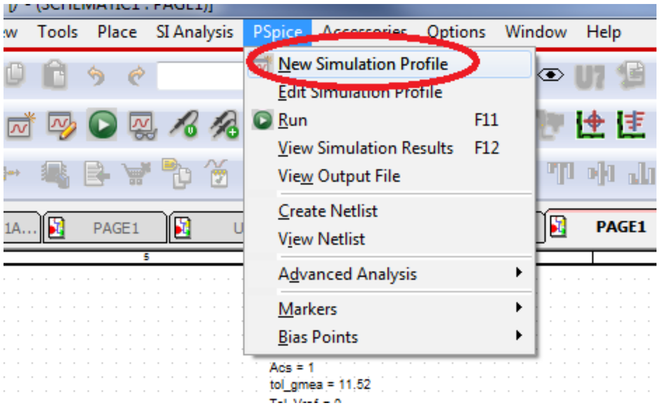


Figure 1. Model Schematic

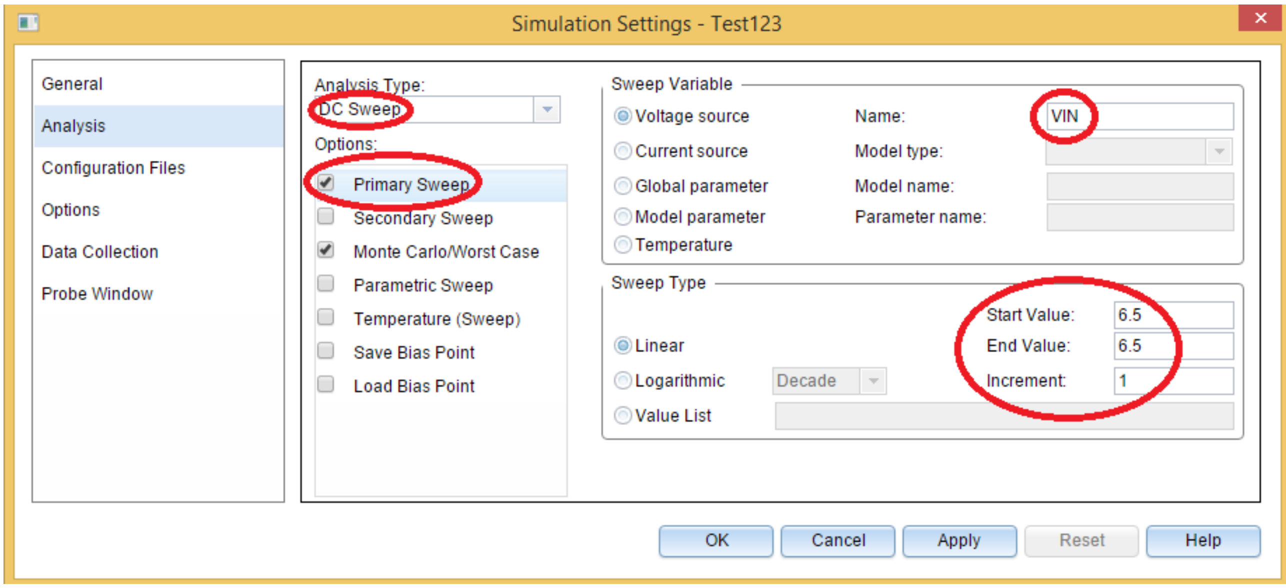
3 Simulation of the TPS7A4501-SP model

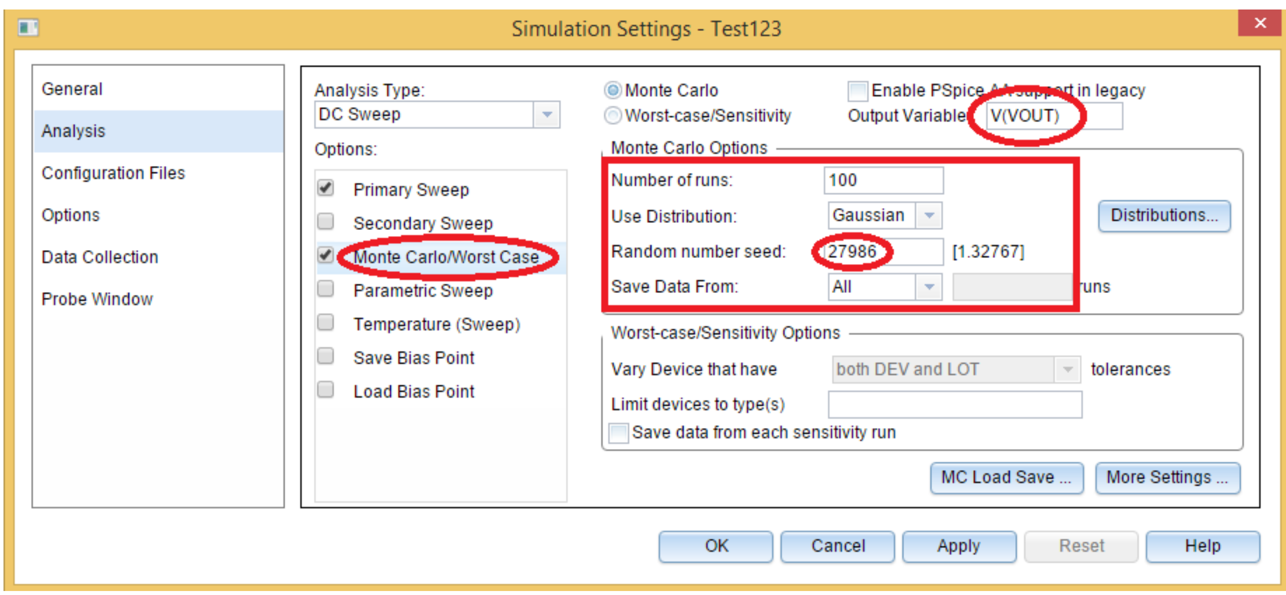
3.1 Monte Carlo analysis of output voltage

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



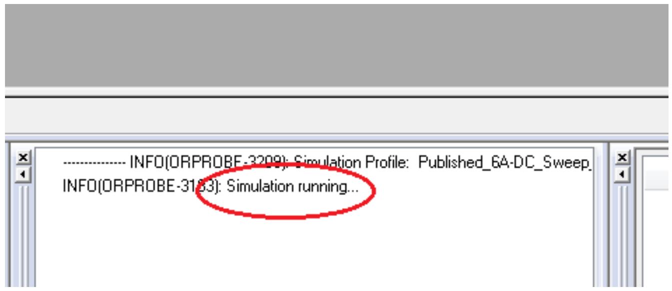
2. Set the parameters based on the image below using the following settings:
   - **Analysis Type** - DC-Sweep
   - **Primary Sweep** - VIN (Input Voltage) from 6.5 V to 6.5 V
   - **Monte Carlo** - 100 runs with Gaussian distribution
   - **Output Variable** - V(VOUT) - Output voltage



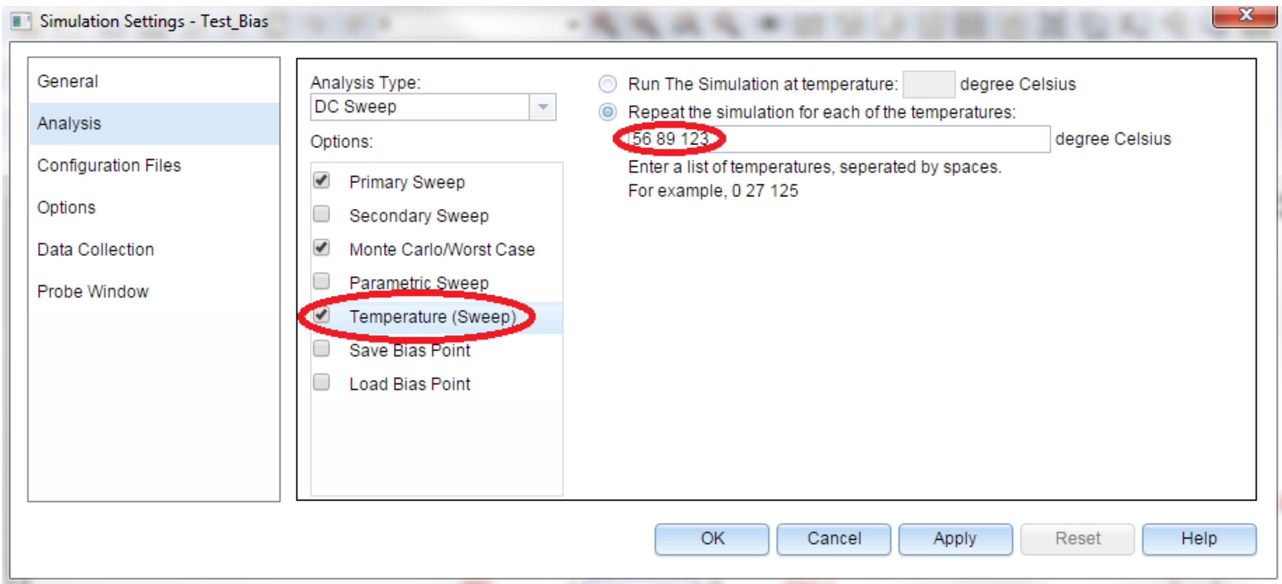


3. Set the random seed number for Monte Carlo within range shown to the right of the input box.
4. Run the simulation by pressing F11 or *PSpice* → *Run*.

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



6. 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.

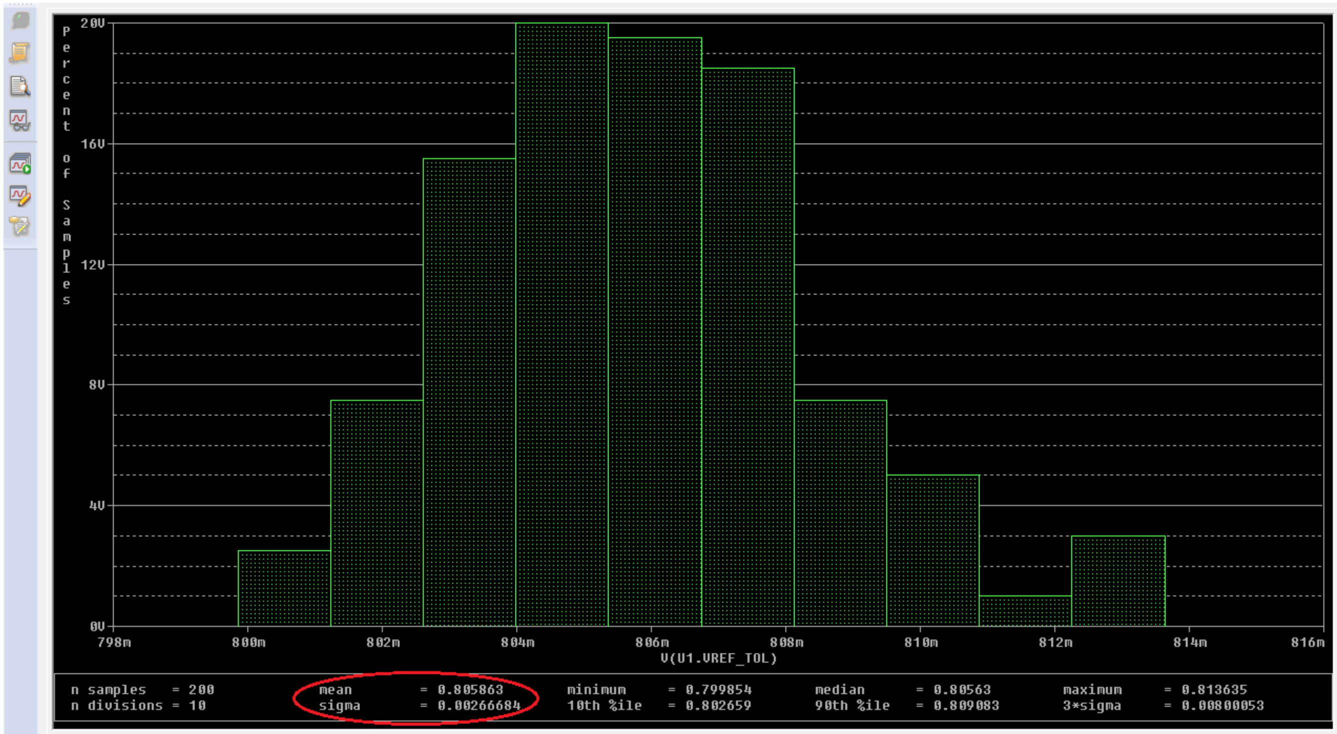
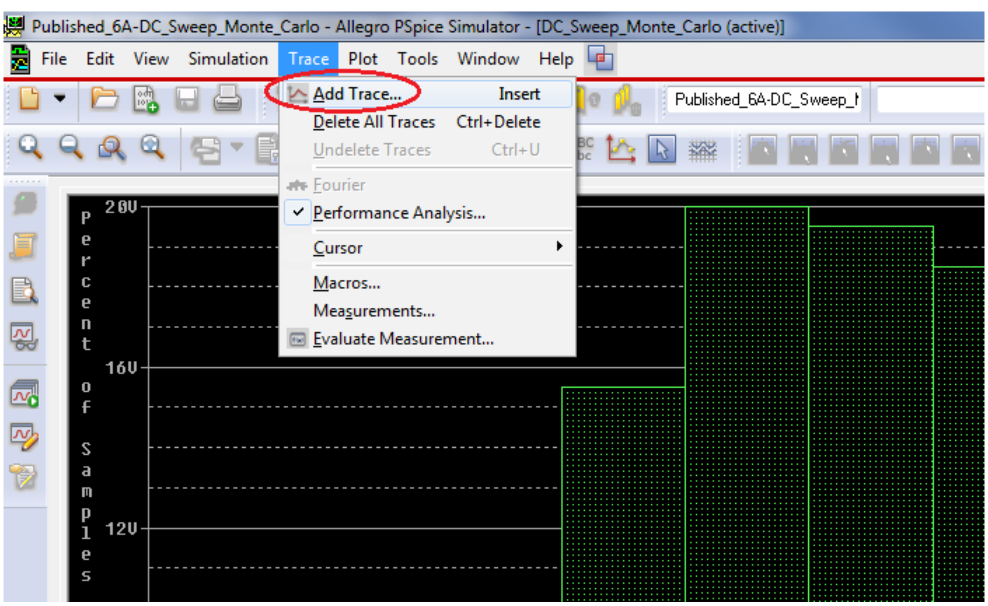
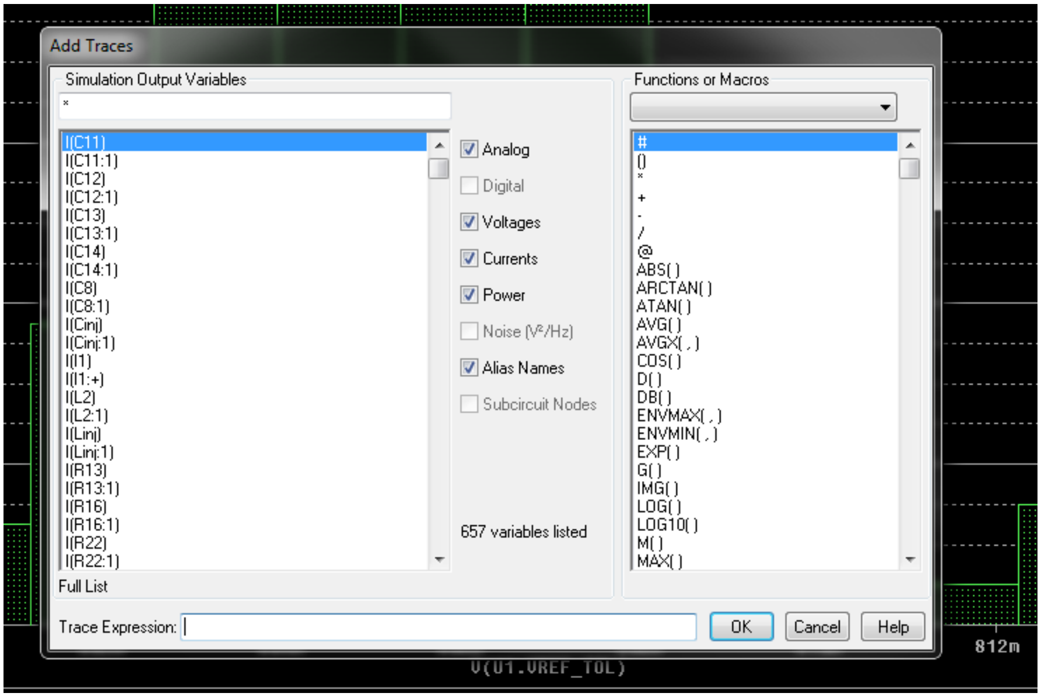


Figure 2. Output Voltage Monte Carlo Histogram

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



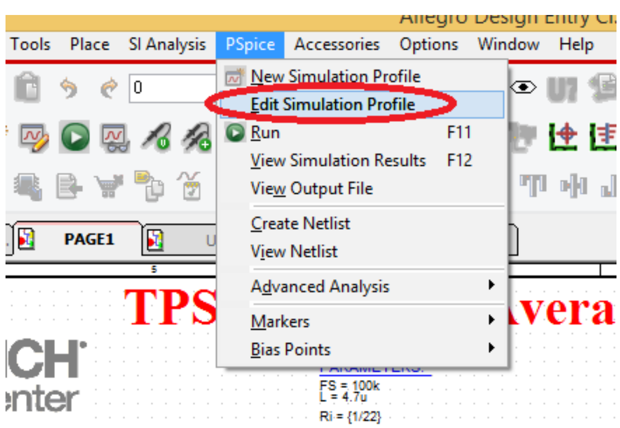
This will bring in another pop-up with the netlist present in the schematic.



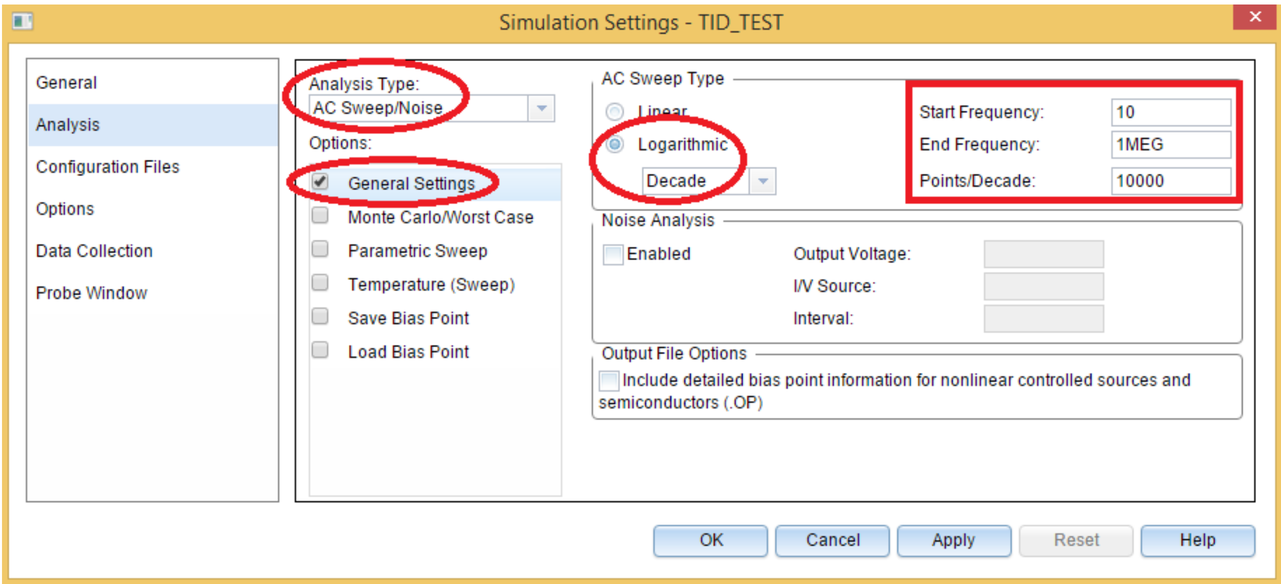
Select the desired netlist and click "OK". The histogram of the selected node will be updated.

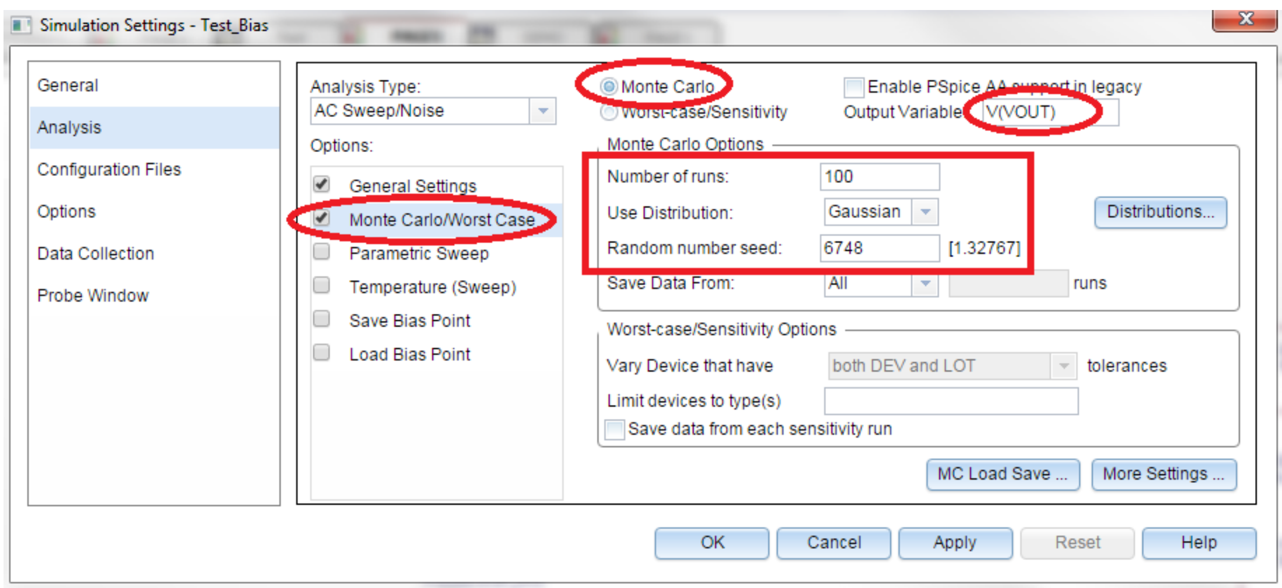
3.2 Performing frequency analysis

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



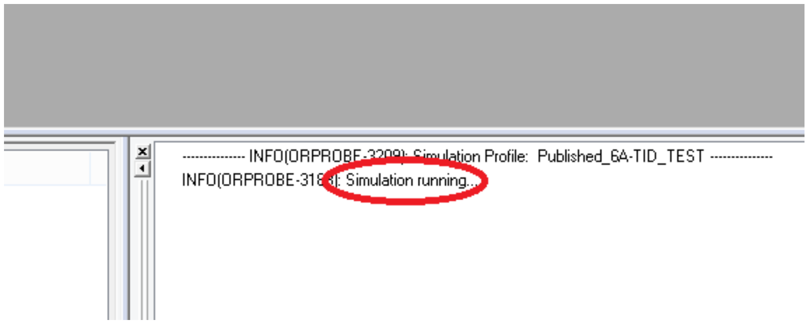
2. Set the following parameters as shown:
   - **Analysis Type** - AC-Sweep
   - **General Settings** - Set start and end frequency, number of points per decade and sweep type
   - **Monte Carlo** - 500 runs with Gaussian distribution
   - **Output Variable** - V(VOUT) - Output Voltage





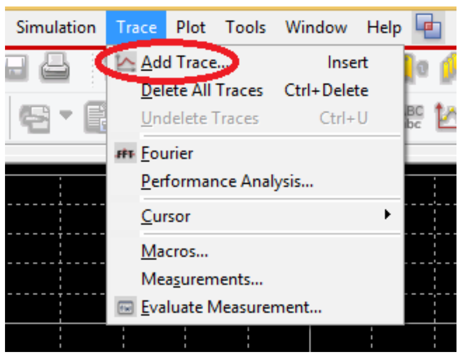
3. Run the Simulation by pressing F11 or *Pspice* → *Run.*

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

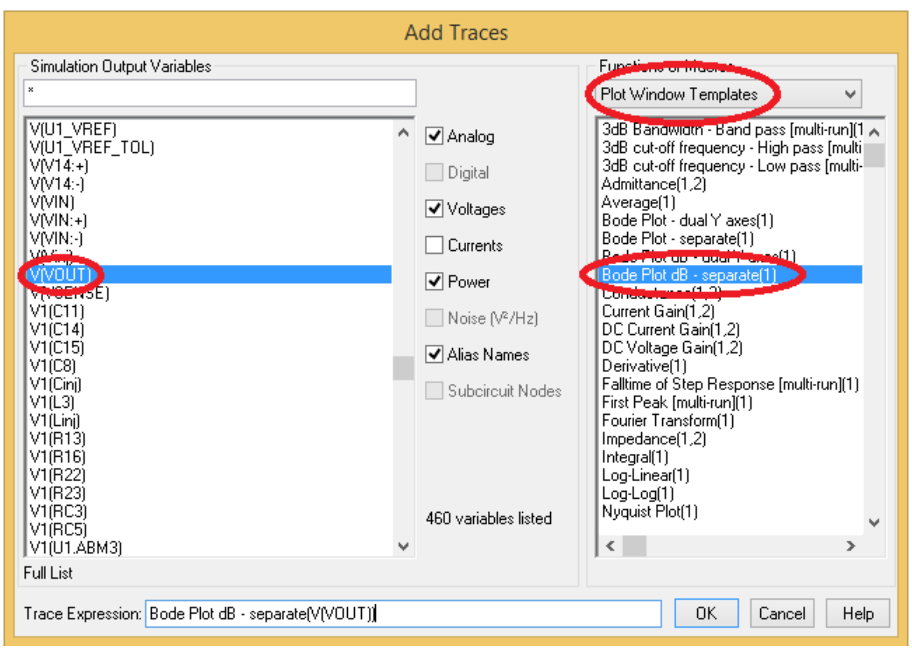


3.2.1 Analyzing frequency response with Bode plot

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



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



3. Use the cursor to evaluate the plot.

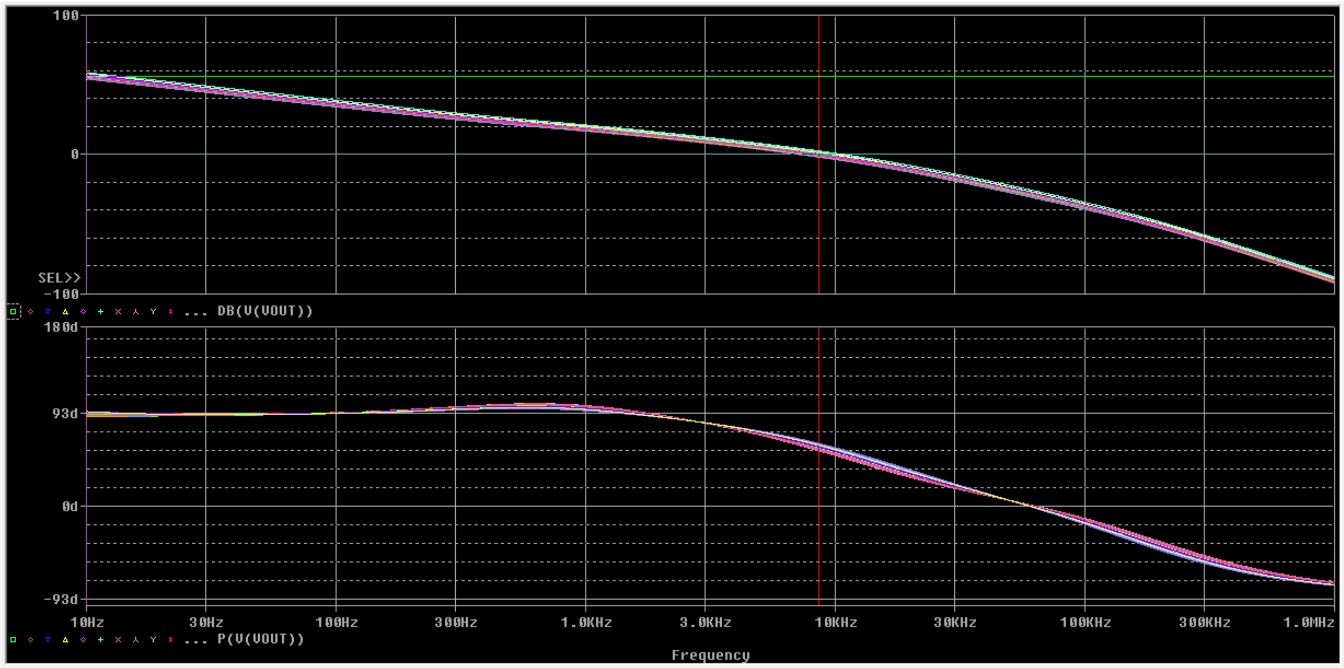
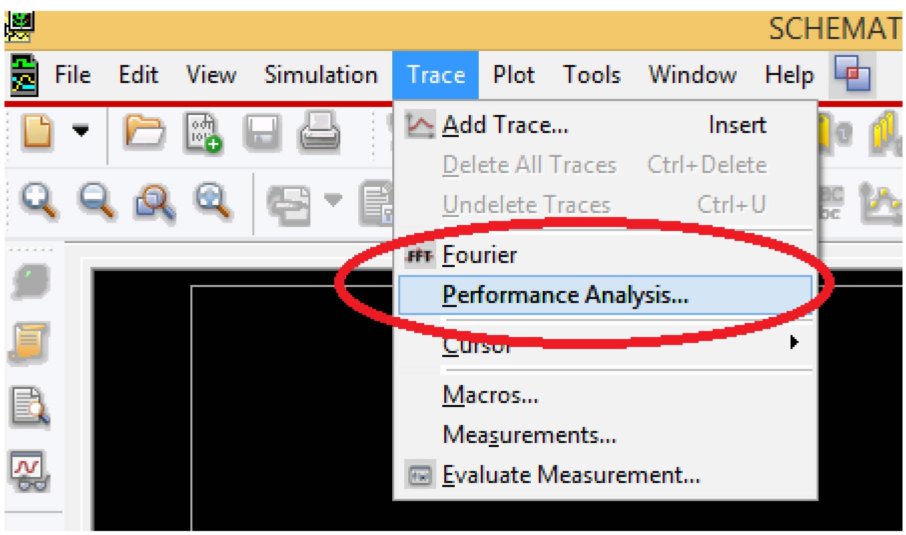


Figure 3. Frequency Response Bode Plot

3.2.2 Analyzing frequency response with histogram

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



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

3. 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.

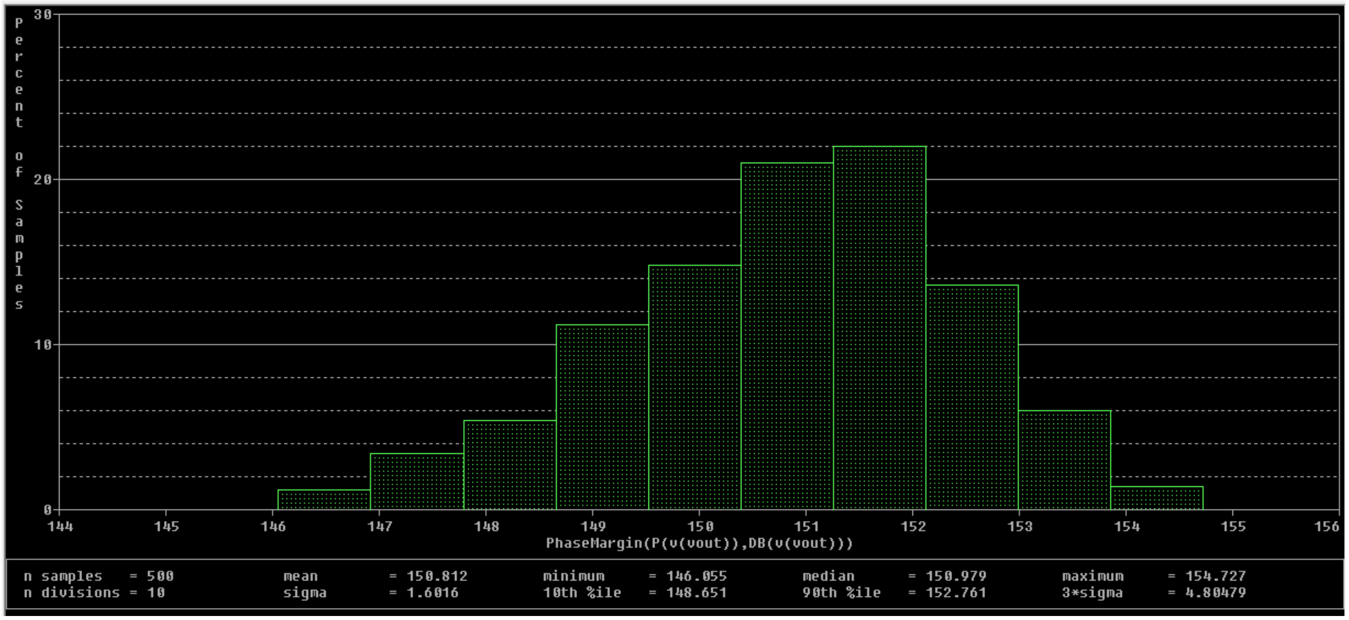


Figure 4. Frequency Response Histogram