User's Guide

SLVUBT1A–October 2019–Revised June 2020

# TPS50601A-SP Worst Case Analysis Unencrypted PSpice Average Model User's Guide

This user's guide is intended to demonstrate use of the average Pspice model for the TPS50601A-SP synchronous buck converter. Instructions on importing the unencrypted model netlist into Cadence Pspice® are also provided. The first half of the guide outlines the modeled parameters and specifications 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 TPS50601A-SP WCA Model Specification

The netlist file (TPS50601A-SP_AVG.LIB) contains the average model of the device TPS50601A-SP. The model incorporates the following parameters:

- Frequency response (Phase Margin, Phase Margin Crossover and Gain Margin)
- Reference voltage

These parameters are modeled over the full military temperature range, –55°C to 125°C, as well as over the rated Total Ionizing Dose from 0 to 100 krad(Si). Monte Carlo analysis can be performed with the model to include device-to-device variation and changes in supply voltage.

Monte Carlo analysis models device variation for the following parameters:

- **TOL_GMea -** This parameter is the transconductance tolerance (%) of the error amplifier.
- **TOL_GMps -** This parameter is the transconductance tolerance (%) of the power stage.
- **TOL_Vref -** This parameter defines the tolerance (%) of the reference voltage.

The default tolerance values are set based on design simulations. The user has flexibility to change these values.

There are two additional variables that need to be set based on the use case:

- **L -** Output inductor value.
- **FS** - Switching frequency of the device.

Environmental parameters like TID and operating temperature can also be adjusted.

# 2 Default Parameters

The default model parameters are as follows:

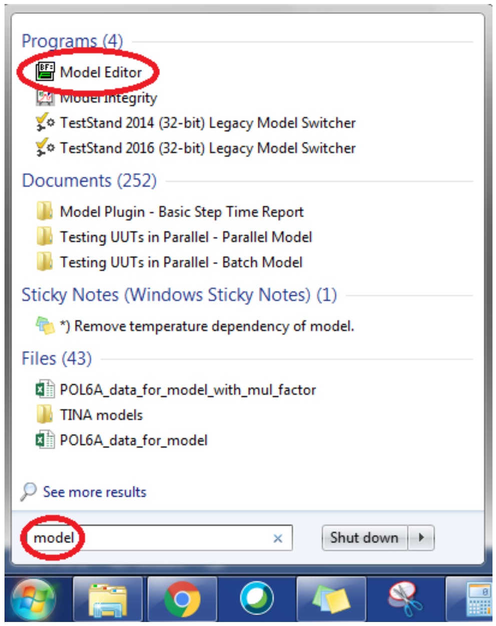
- **FS** - Switching frequency: 100k
- **L -** Load inductance: 4.7 µ
- **Tol_GMea -** Error amplifier tolerance (%): 11.52
- **Tol_Vref** - Reference voltage tolerance (%): 0.3
- **Tol_GMps -** Power stage gain tolerance (%): 5.58
- **TID** - Radiation exposure (krad(Si)): 0

# 3 Example of using model with Cadence Pspice (17.2.0)

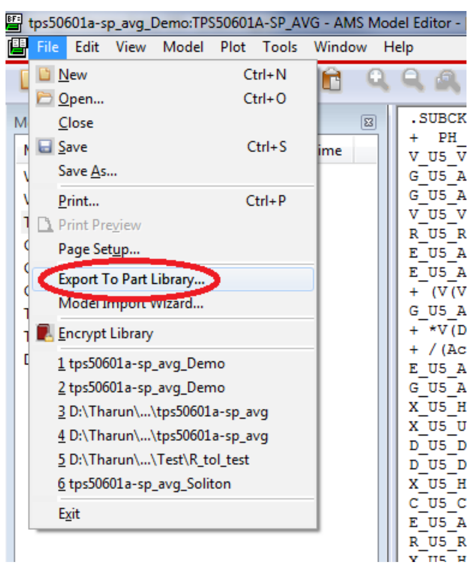
## 3.1 Creating a part from the netlist file

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...

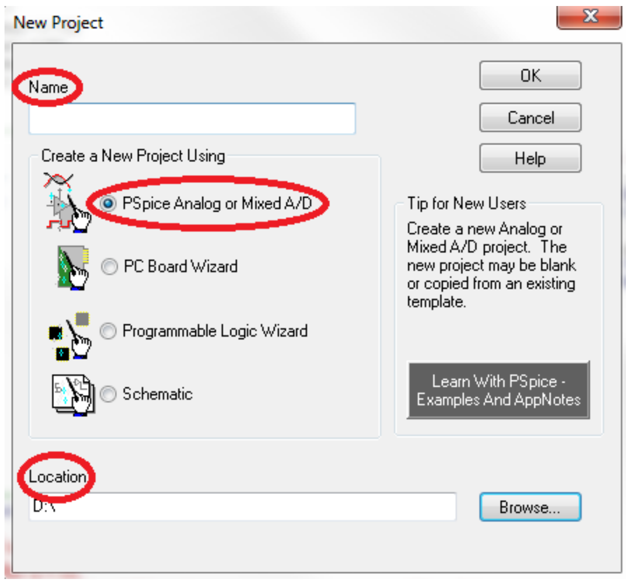


4. The *Create Parts for Library* dialogue box will open. Click "OK" to generate the part library in the default location. This will create a ***.olb file with the same name and location as the netlist file.

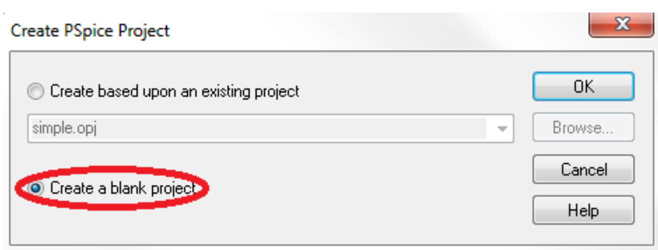
## 3.2 Steps to create project in Capture®

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

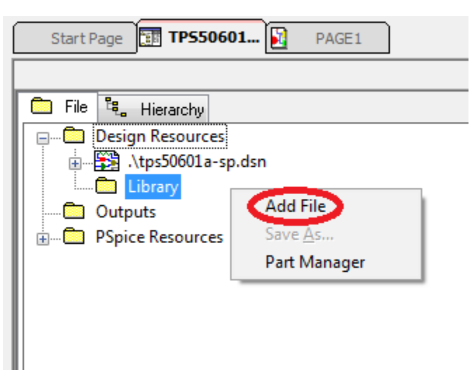
1. Open the Capture application from Cadence - v17.2.0 or above.
2. Click on File → New → Project.
3. Enter a project name and location, choose *PSpice Analog or Mixed A/D* from the options, and click "OK".



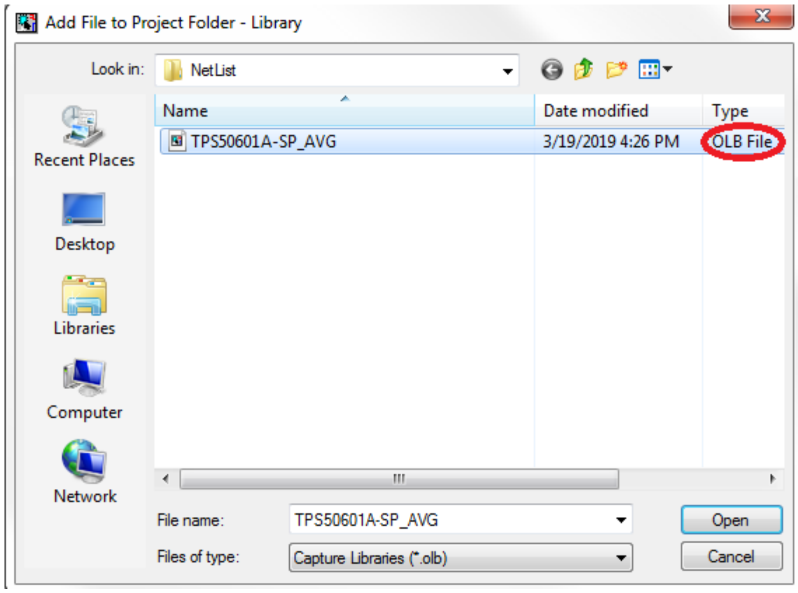

4. Once the *Create Pspice Project* dialogue box opens, select the "Create a blank project" option and click "OK".




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



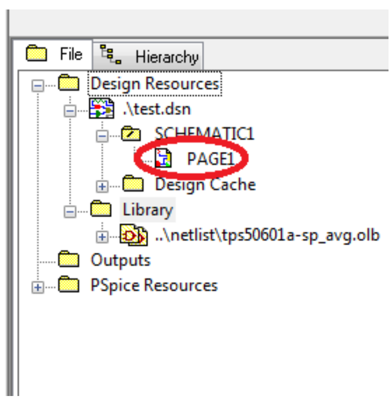
6. Choose the ***.olb file that was previously created, add to the dialogue box, and click "Open". This will add the part symbol to the project.



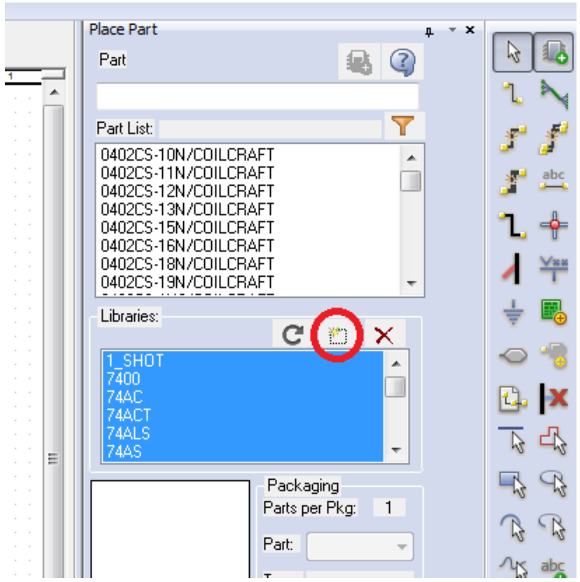

## 3.3 *Creating the schematic in Capture*

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

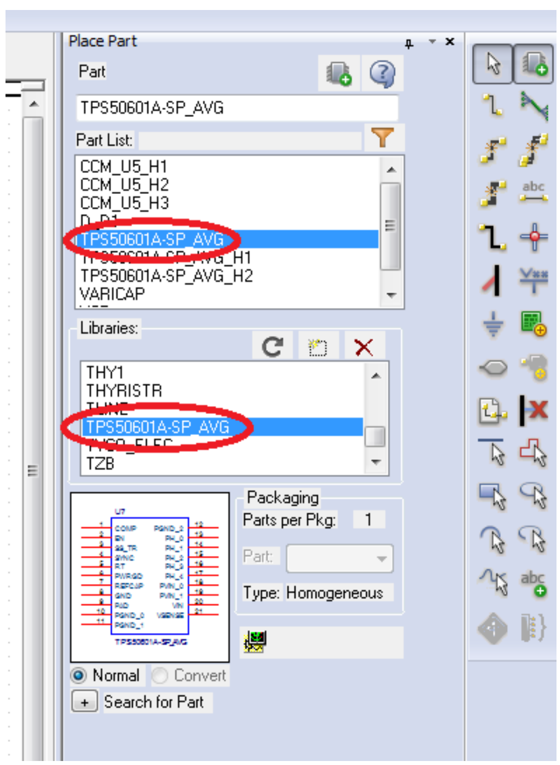
1. Open *PAGE1* under the *SCHEMATIC1* folder below the .dsn file.



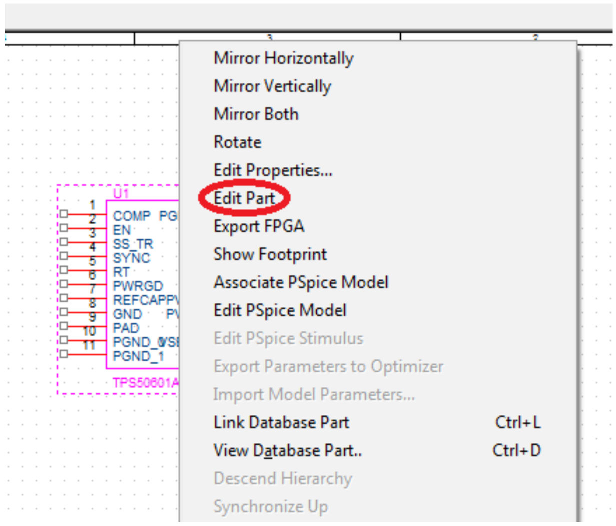
2. Select *Place* → *Part* to open the *Place Part* window. Click on the square icon shown in the following image to add libraries to the project.



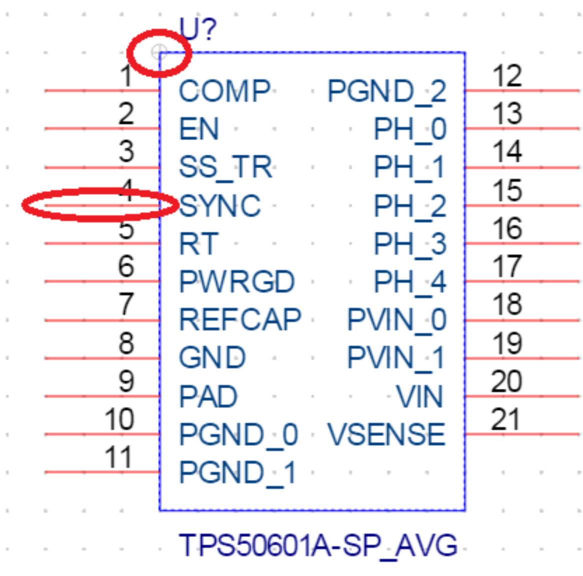

3. Select all default library files of PSPICE from installation directory and click "Open". (Default Location: “C:\Cadence\SPB_17.2\tools\capture\library\pspice”) Note: This step can be omitted if the default libraries have already been added to Capture.
4. In the *Part* search window, type “TPS50601A-SP_AVG” and select the model.



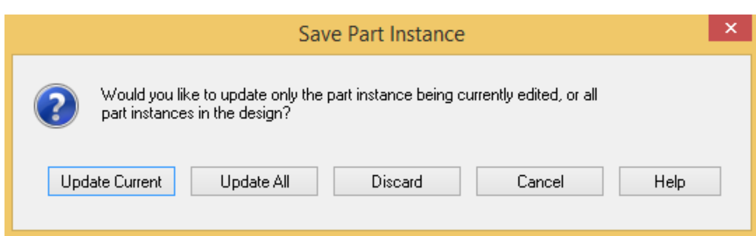
5. Double click on the part in the *Part List* window and place it by left clicking the cursor on the PAGE1 to place the part. Once the part is placed, the symbol can be edited by selecting the part, right clicking, and choosing "Edit Part".



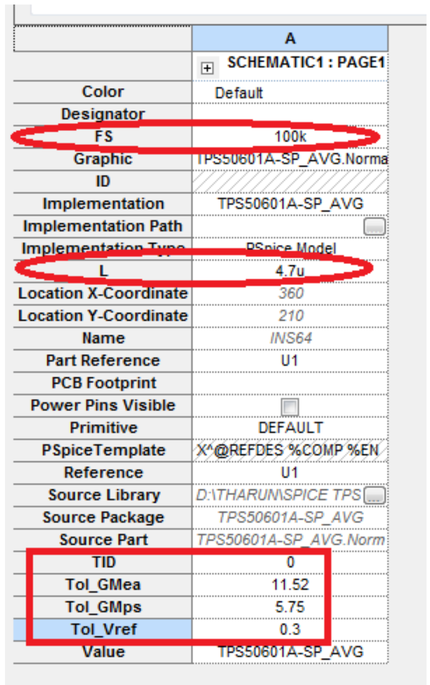

6. The part can be resized and pins can be moved to more convenient positions by clicking and dragging them.



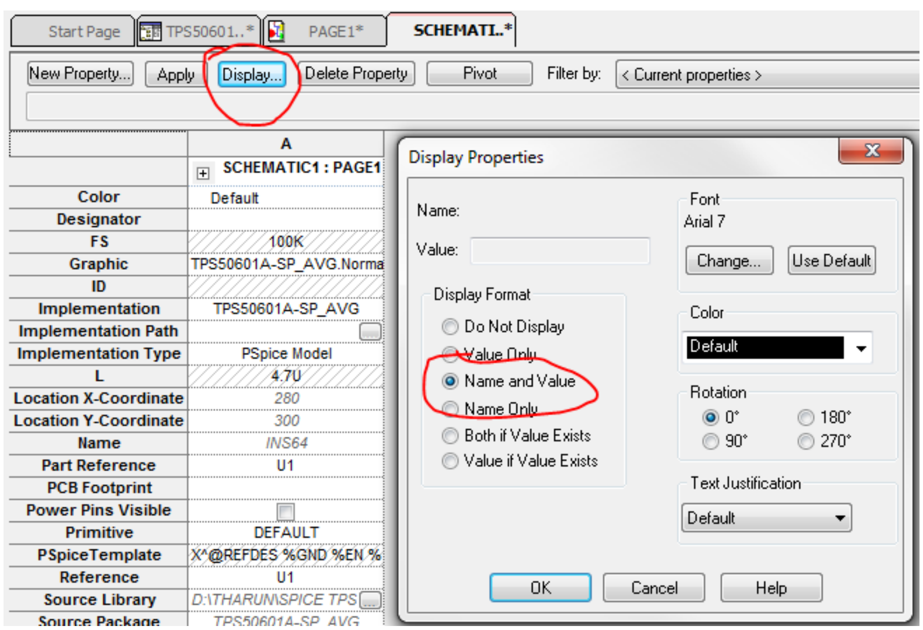

7. After editing the part, close the tab to save changes. Choose "Update Current" in the pop-up.



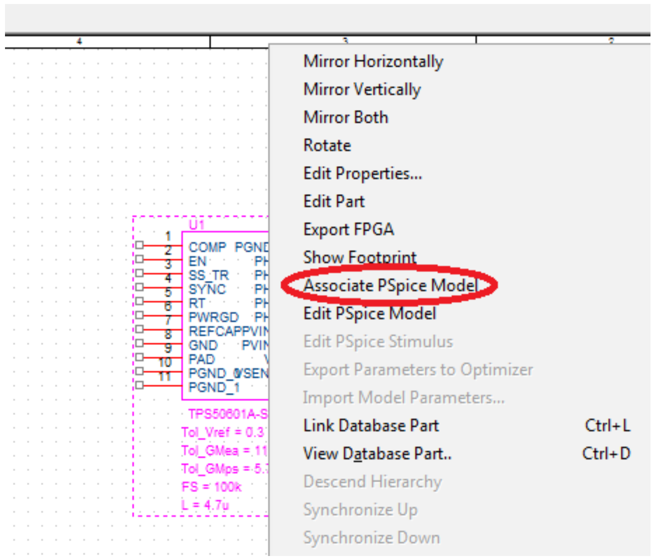
8. Multiple controllable parameters are present in the model, which can be varied to analyze the model functionality. The default values for these parameters can be found in Section 2. To enable easy access to these parameters, make them visible by double clicking the part. The following window will then appear:



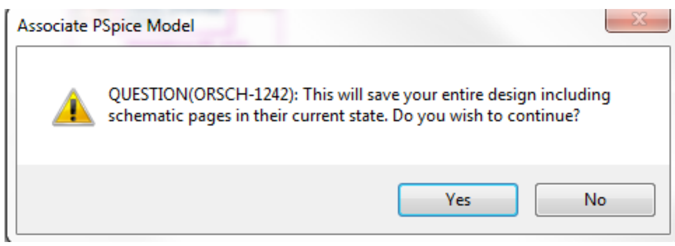

9. Select all the parameters highlighted above and click the "Display" button. In the pop-up, select "Name and Value" and click "OK".

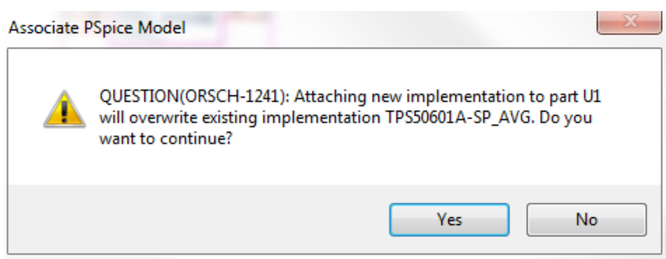


10. Now, the part must be associated with the netlist. Select the part, right click, and choose "Associate Pspice Model".

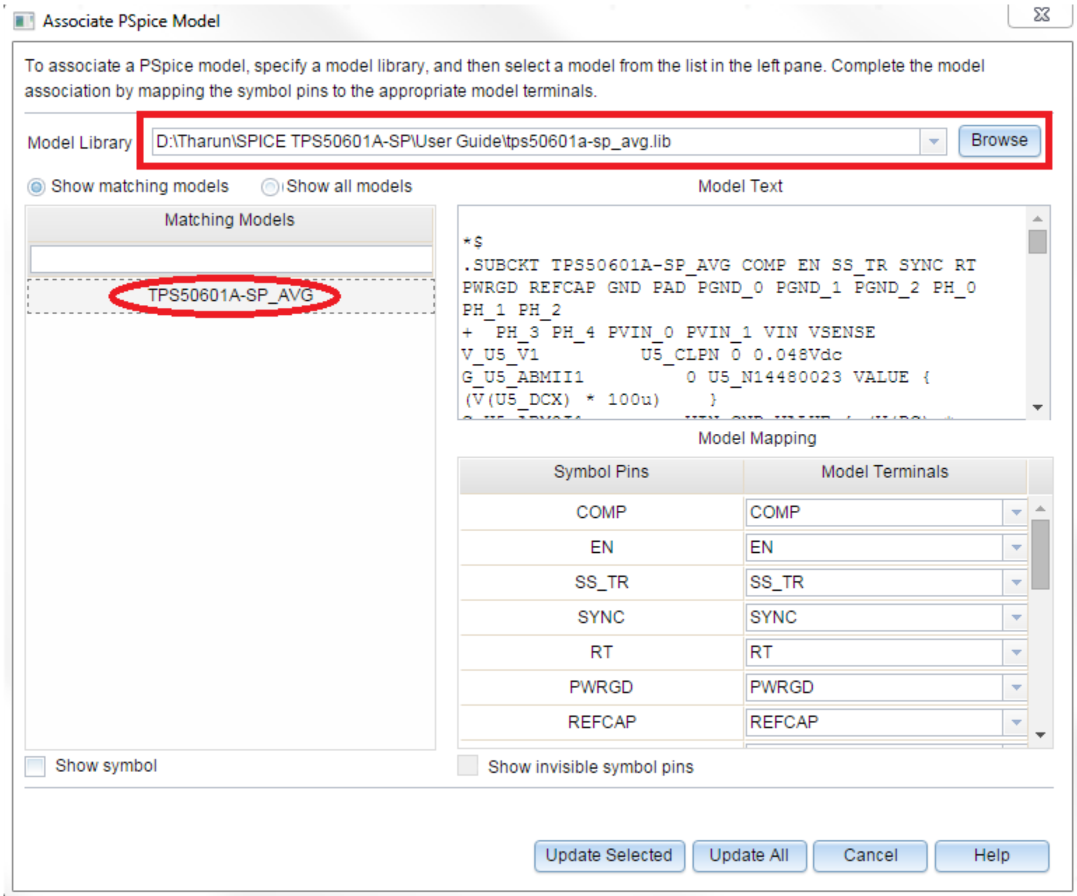



11. Two pop-ups will appear to confirm the operation. Click "Yes" for both.

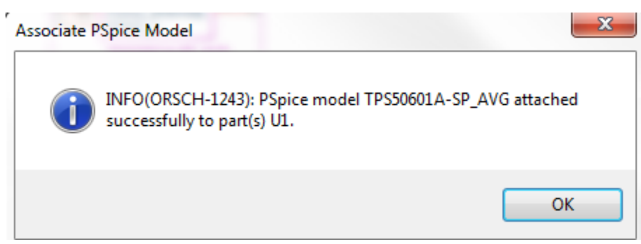


12. In the *Associate Pspice Model* dialogue box, choose the netlist file (***.lib), select the model, and click "Update All".




13. A pop-up will appear with successful update message. Click "OK".



14. Add the remaining components to complete the schematic as shown in Figure 1.

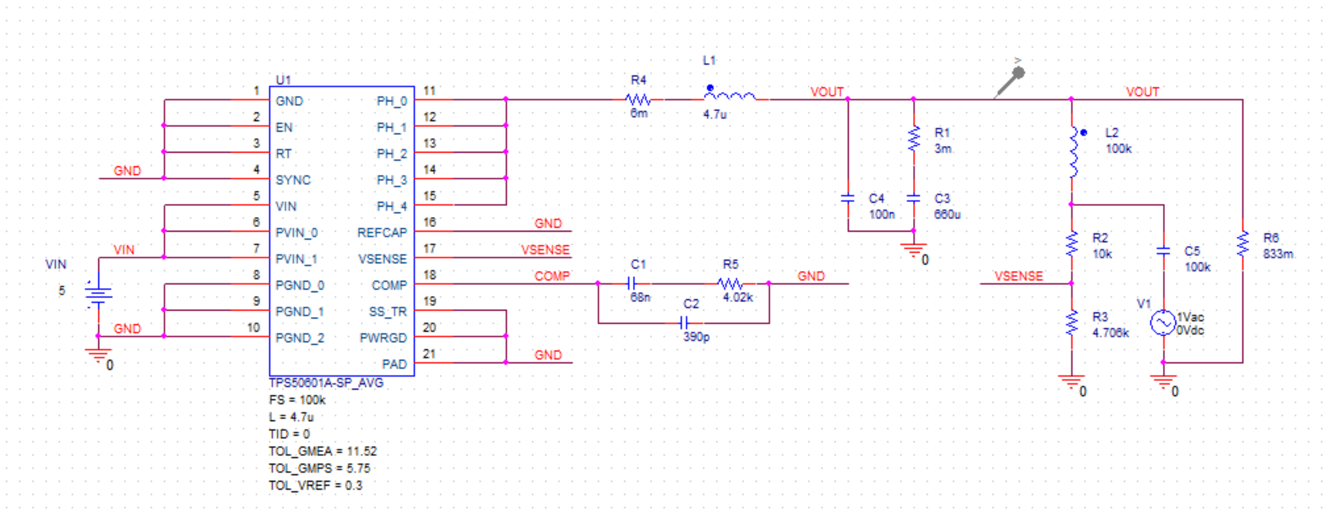
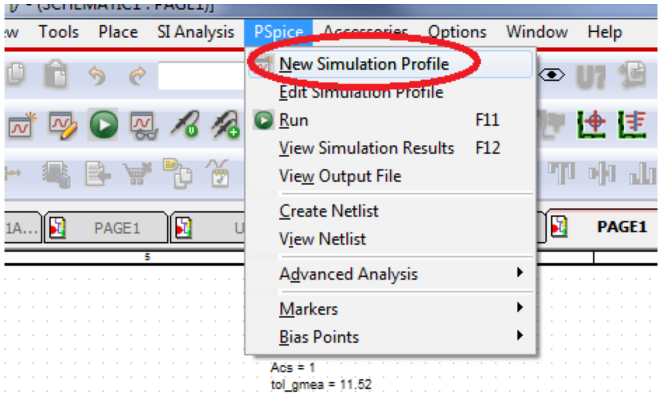



**Figure 1. Model Schematic**

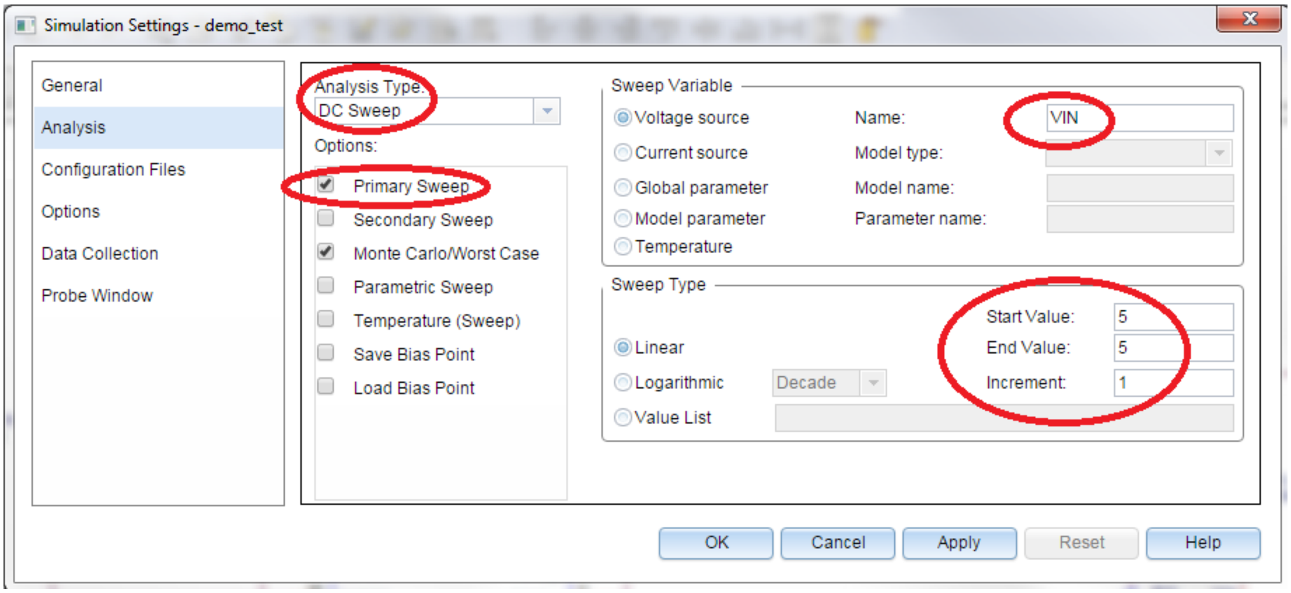
# 4 Simulating the Model

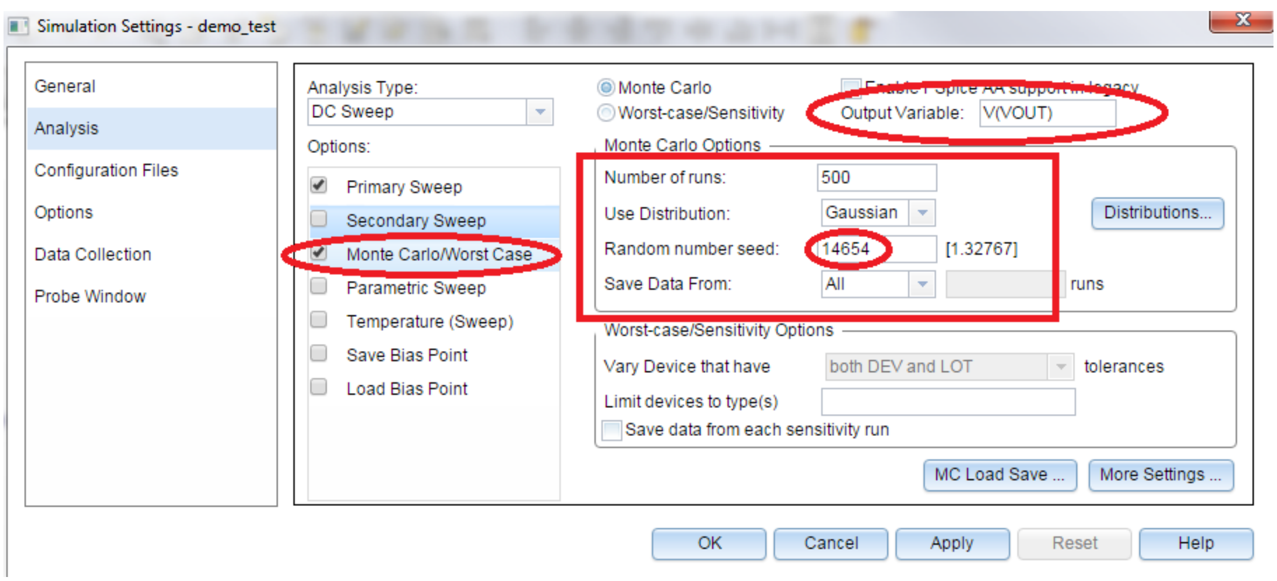
## 4.1 Monte Carlo Analysis of Output Voltage

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



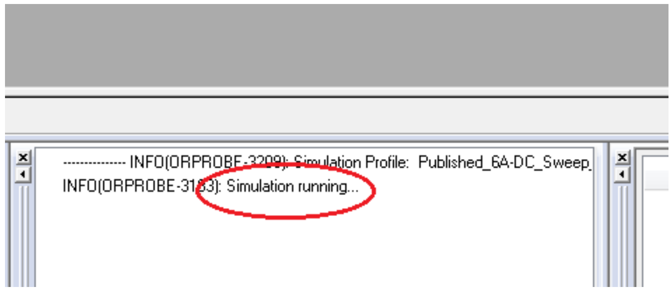
2. Set the parameters based on the image below using the following settings:
   - **Analysis Type** - DC-Sweep
   - **Primary Sweep** - VIN (Input Voltage) from 5 V to 5 V
   - **Monte Carlo** - 500 runs with Gaussian distribution
   - **Output Variable** - V(VOUT) - Output Voltage



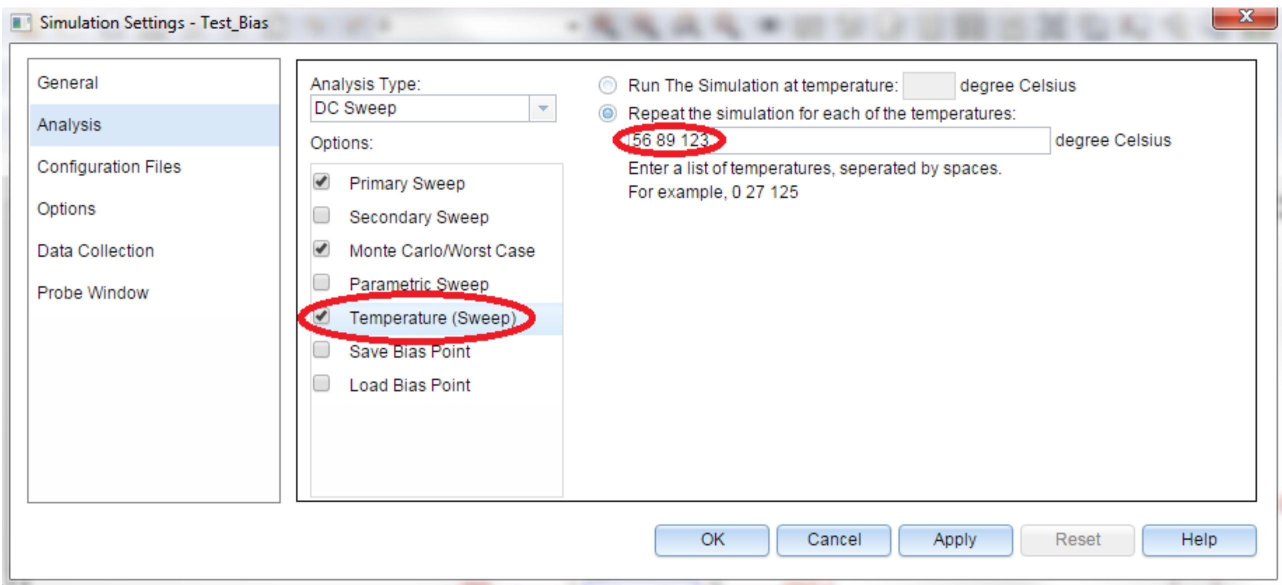




3. Set the random seed number for Monte Carlo within the range shown to the right of the entry box.
4. Run the simulation by pressing F11 or *PSpice* → *Run*.

5. 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.




The output windows come up once the simulation is completed. The results window shows the histogram of the output voltage (V(VOUT)) with Monte Carlo analysis. The mean, sigma, min, max, etc. are displayed at the bottom of the window.

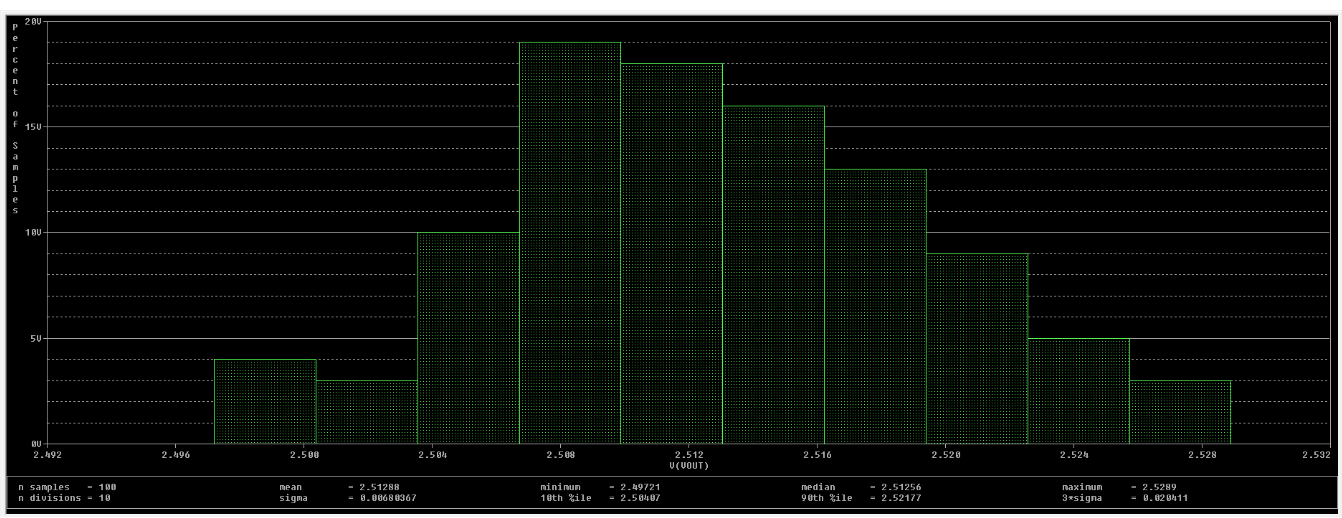
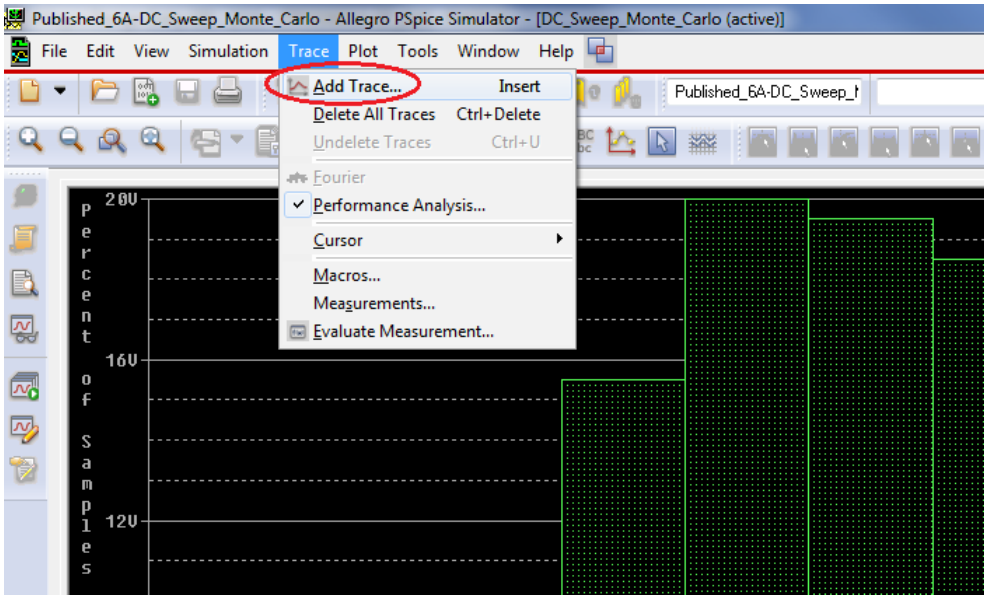
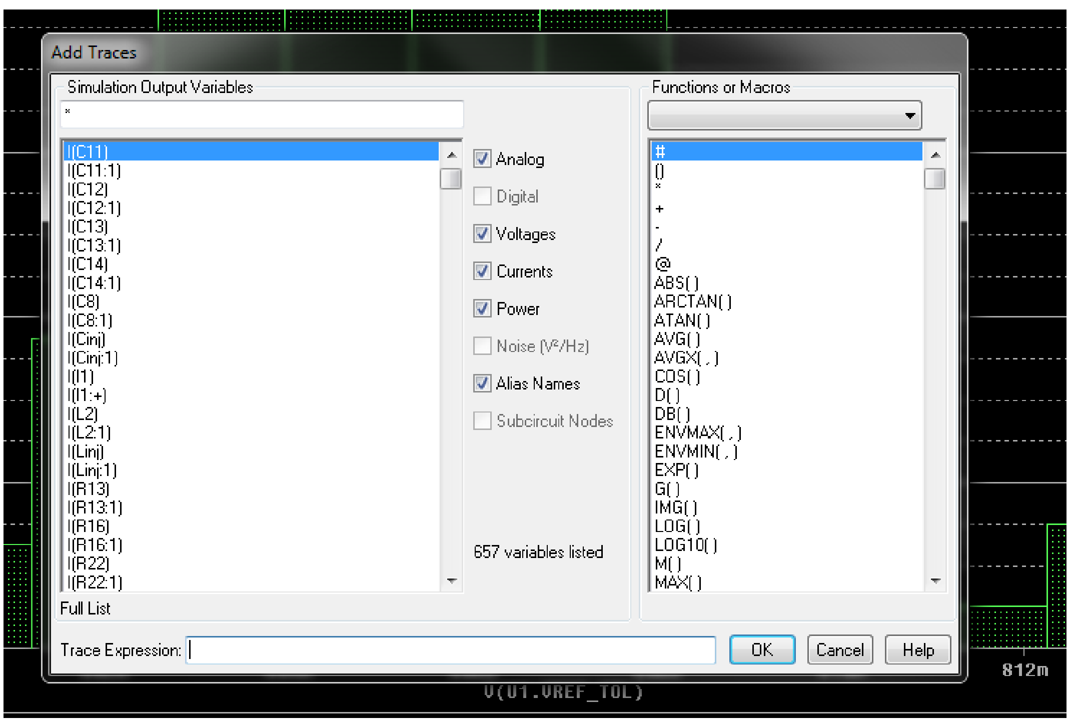



**Figure 2. Output Voltage Monte Carlo Analysis**

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

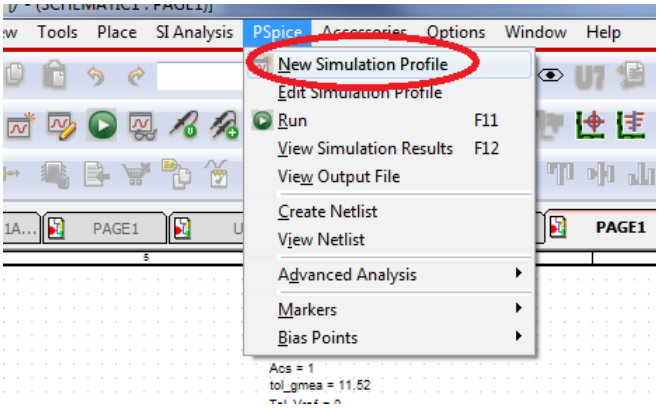



This will bring in another pop-up with the netlist present in the schematics. Select the desired trace and click "OK". The histogram of the selected node will be updated.

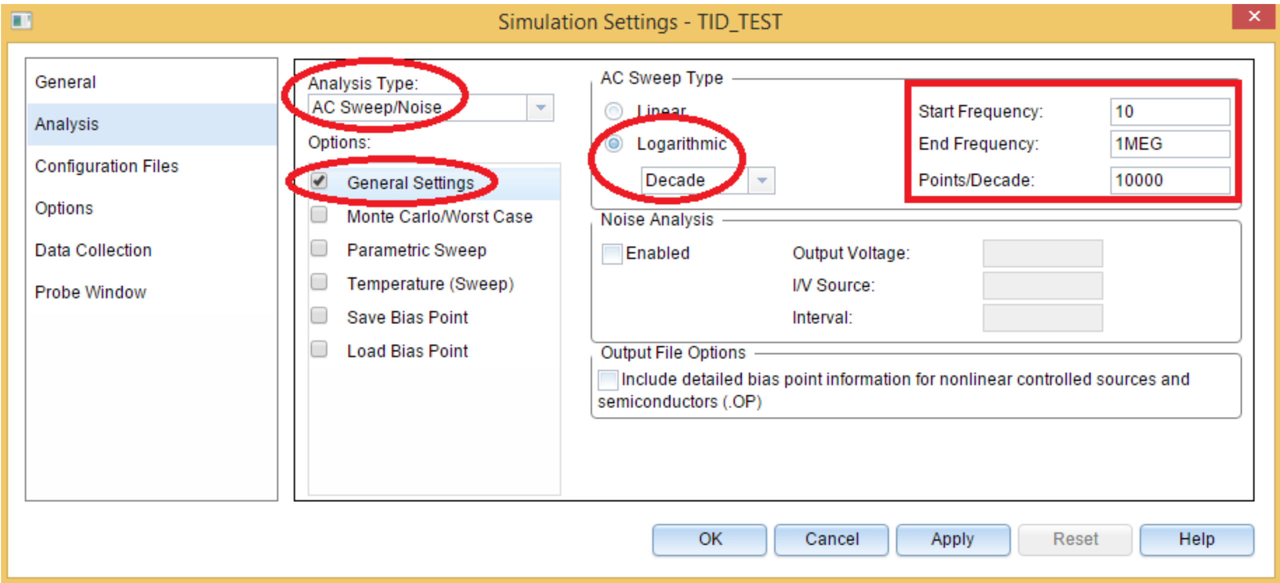


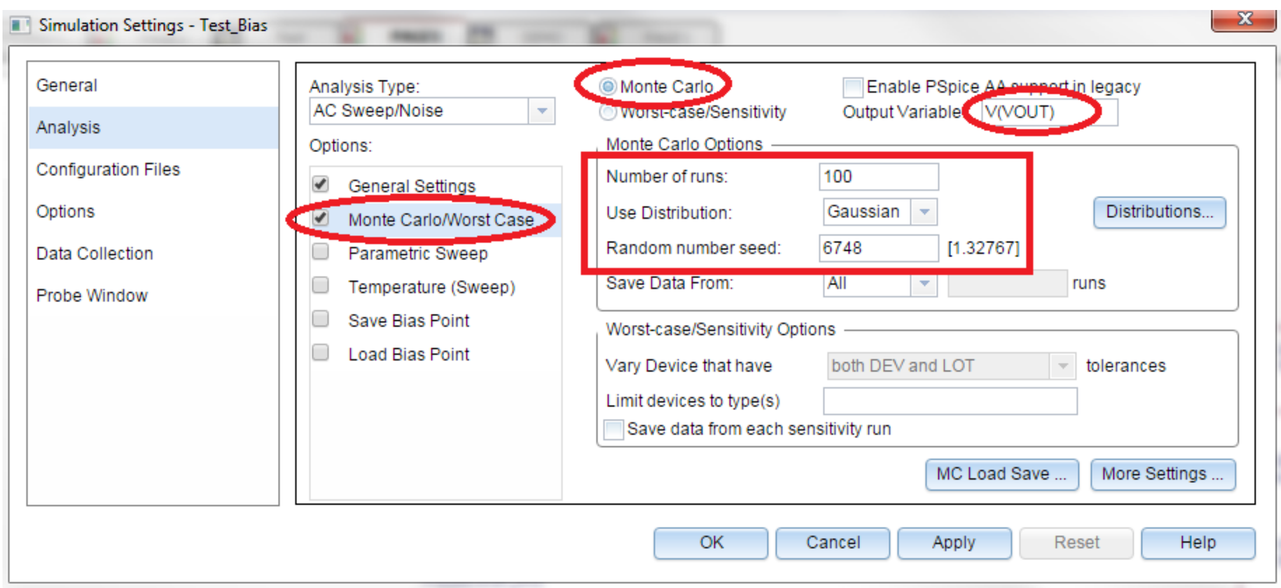
### 4.2 Frequency Response Analysis

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



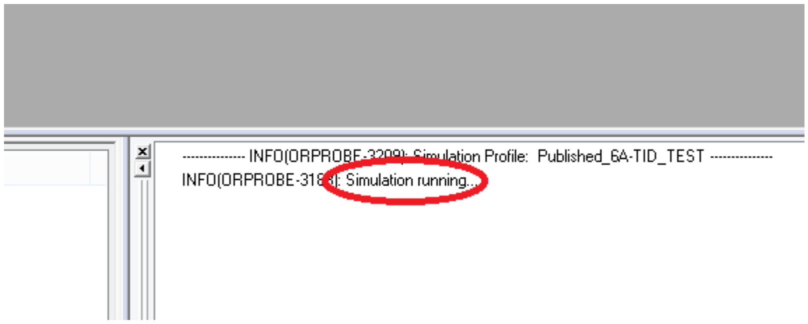
2. Set the parameters based on the image provided below using the following settings:
   - **Analysis Type** - AC-Sweep
   - **General Settings** - Set start and end frequency, number of points per decade, and sweep type
   - **Monte Carlo** - 500 runs with Gaussian distribution
   - **Output Variable**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



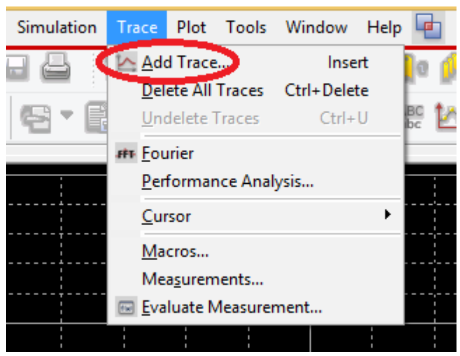

3. Run the Simulation by pressing F11 or *Pspice* → *Run*.

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

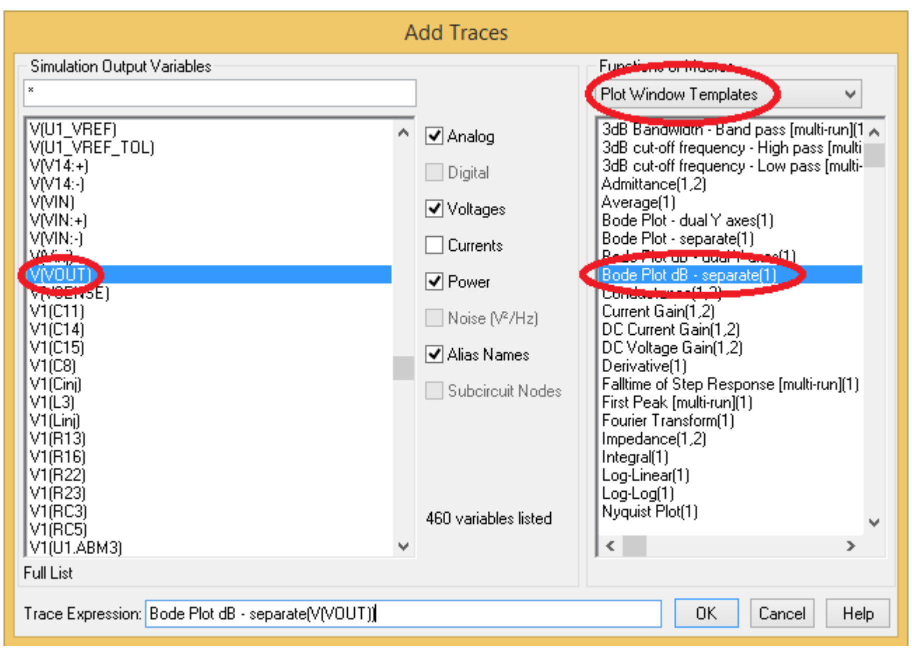



### 4.2.1 Analyzing Frequency Response with Bode Plot

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.

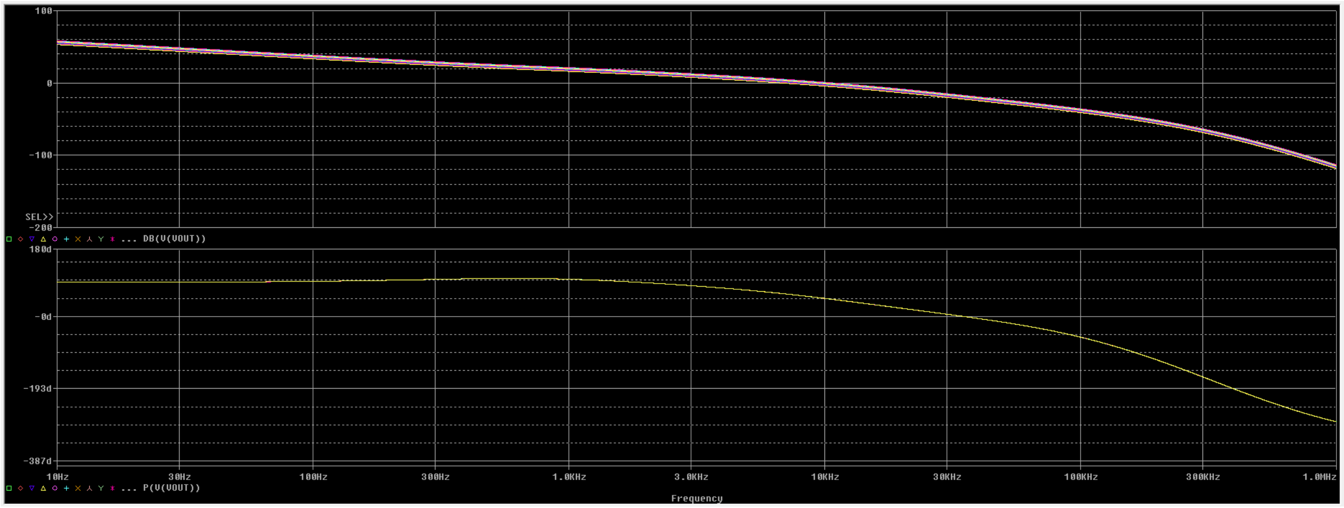
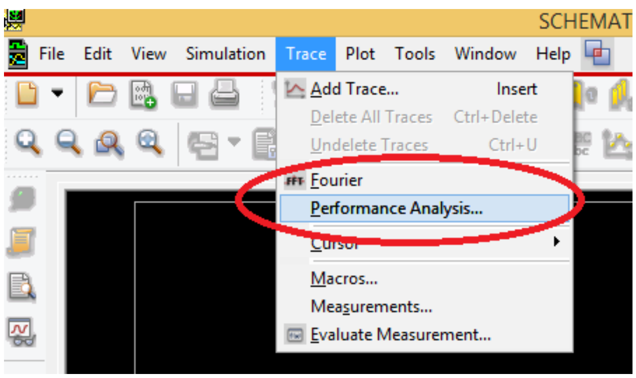



**Figure 3. Frequency Response Bode Plot**

### 4.2.2 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 "Add Trace".
3. Copy the expression **PhaseMargin(DB(v(vout)),P(v(vout)))** into the *Trace Expression* field and click "OK". The histogram will be displayed in the trace window.

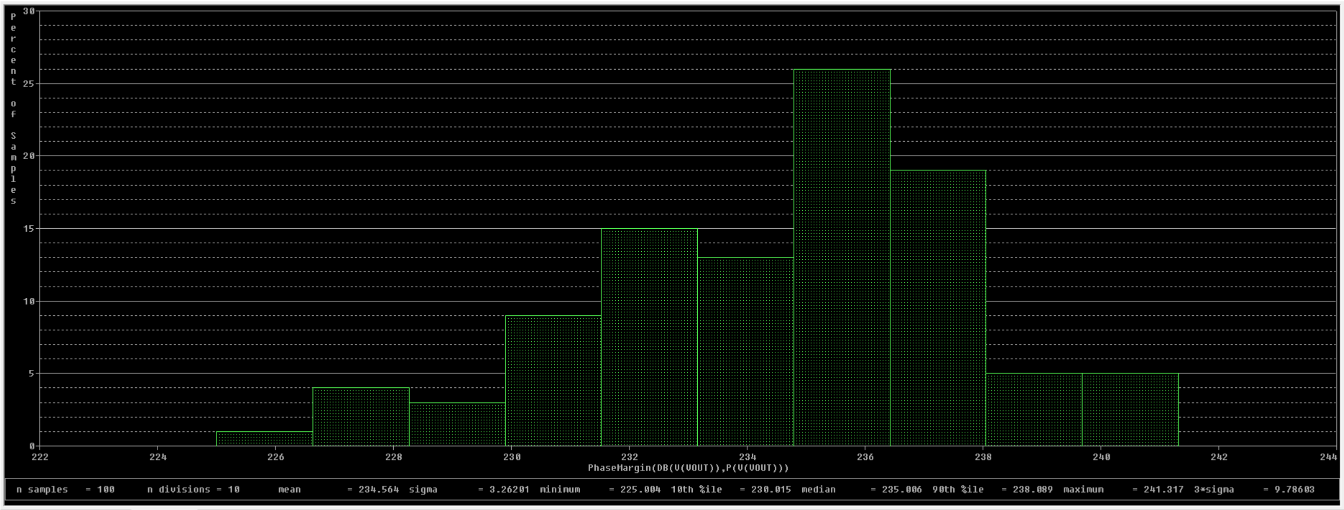



**Figure 4. Frequency Response Histogram**

# Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

**Changes from Original (October 2019) to A Revision** **Page**