User’s Guide

TPS7H4001-SP Worst Case Analysis Model

ABSTRACT

This user's guide is intended to define the Worst Case Analysis (WCA) Pspice model for the TPS7H4001-SP. Instructions on setting up the unencrypted model for simulation Cadence Pspice® are also provided. The first section of the guide outlines the model parameters and development. The second section covers different options for setting the model up to run simulations. Finally, the third section addresses how to run different types of simulations.

Table of Contents

List of Figures

Trademarks

Pspice® are registered trademarks of Cadence.
All trademarks are the property of their respective owners.

1 TPS7H4001-SP WCA Model Specification

The TPS7H4001-SP WCA model is a behavioral model intended for simulation of frequency response characteristics, such as phase margin, phase margin crossover frequency, and gain margin. Monte Carlo analysis can also be performed to observe the distribution of behavior across a specified sample size of devices. The following sources of variation are incorporated into the model:

Process Device-to-device manufacturing differences

Temperature Over full military range: -55°C to 125°C

Load Up to the maximum recommended load: 18 A

Radiation Up to the maximum rated Total Ionizing Dose (TID) of the device: 100 krad

Aging Represented by data taken at 25°C before and after 1,000 hours of testing at 125°C, and meant to emulate 15 years operating at 65-95°C as defined by Group C specifications in MIL-PRF-38535

In the model, the specific device parameters influenced by the above variation include:

gm_{ea} Error amplifier transconductance

gm_{ps} Power stage transconductance

V_{REF} Reference voltage

1.1 Model Parameters

The following parameters are used within the model to define the internal characteristics, operating environment, and configuration of the device.

Parameter	Description	Default Model Value
Mean_GMea	The mean value of the error amplifier transconductance	1800 µS
Mean_GMps	The mean value of the power stage transconductance	40 S
Mean_Vref	The mean value of the reference voltage	0.605 V
Tol_GMea	The tolerance of the error amplifier transconductance [1]	8.72 %
Tol_GMps	The tolerance of the power stage transconductance [1]	5.22 %
Tol_Vref	The tolerance of the reference voltage [1]	0.31 %
TID	The Total Ionizing Dose the device is subjected to, within the range of 0 to 100 krad	0 krad
AGING	Binary value indicating the presence of aging effects, where: 0 = No aging effects added 1 = Aging effects added	0
L	Output inductor value	0.9 µF
FS	Switching frequency	500 kHz
TEMP	Operating temperature, which is set within the simulation profile and takes the default value unless otherwise specified.	27°C

[1] For Gaussian distributions, tolerance is defined as the standard deviation (1-sigma) divided by the mean and is expressed as a percentage.

In addition to the listed model parameters, external component selection will also influence device behavior. The external components used in the default schematic take nominal values and, as such, users may see fit to add tolerances to them to model real-world variation.

1.2 Model Assumptions and Development

Process variation for a given parameter is represented by its mean and tolerance values, which have been given default values based on statistical design simulations that are run at room temperature (27°C) and encompass variation caused by input voltage. If desired, the user may alter these values.

The influence of temperature and load on the listed device parameters is provided by nominal-case design simulations which exclude radiation and aging effects.

Radiation and aging effects are incorporated into the model in two ways. First, for the specified device parameters, the worst shifts observed between pre- and post-test measurements of approximately 50 units are determined for both TID testing and Life testing. The shifts associated with radiation and aging are applied independently to the mean parameter values in the model when it is set to simulate these conditions. If desired, users can incorporate similar shifts for non-modeled device parameters using Group C (Aging) and Group E (Radiation) reports. The model is also tuned so that the frequency response characteristics produced by simulations more closely match frequency response data collected using EVMs.

The EVM data used to tune the frequency response results of the TPS7H4001-SP WCA model was taken at two specific output voltages (1 V and 2.5 V) at 25°C using the default configuration seen on the 1-channel TPS7H4001-SP EVM, with the exception of changing the bottom feedback resistor to adjust the output voltage. Data collection was done using a total of 10 EVMs: three boards for High Dose Rate (HDR) TID testing, three boards for Low Dose Rate (LDR) TID testing, three boards for Life Test, and one control board.

2 Model Setup

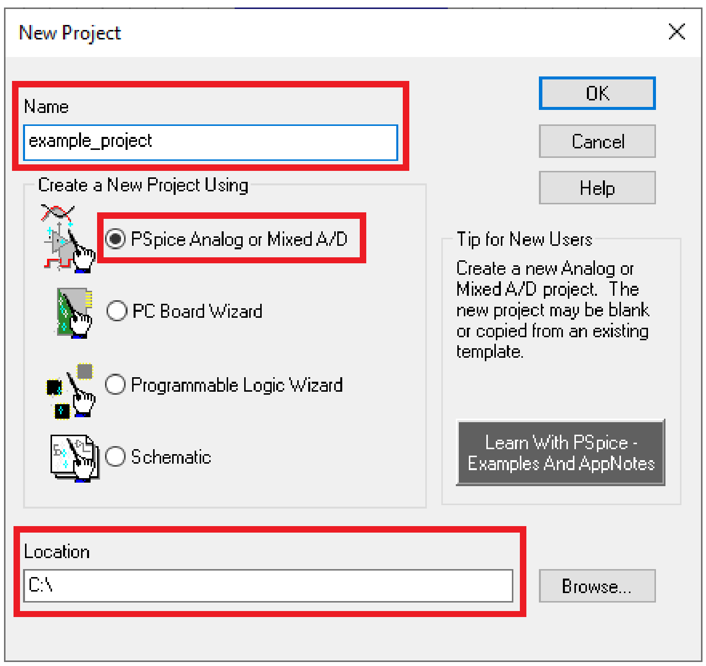
The TPS7H4001-SP WCA model comes with a default schematic contained in the PSpice project file (tps7h4001-sp_wca.opj) that can be used to run simulations with minimal effort needed to set up the model. Users will need to set up their own Simulation Profiles in order to run simulations. Instructions for doing so are provided in Section 3.

If desired, the user can also import the TPS7H4001-SP model into a new or pre-existing PSpice project. The instructions for doing so can be found in Section 2.1.

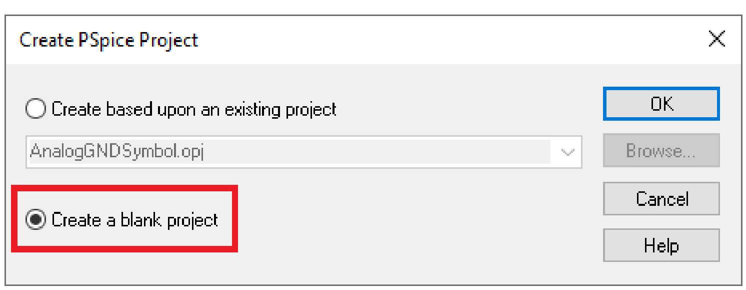
Section 2.2 contains instructions for locating and using the model within Pspice® for TI, which is a free version of Pspice® that includes an automatically updated library of TI PSpice models.

2.1 Importing the TPS7H4001-SP WCA Model into PSpice projects

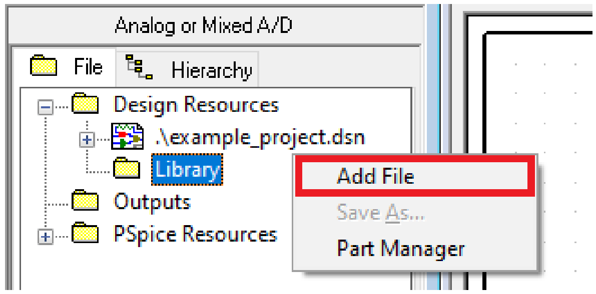
1. Open the Capture application from Cadence - v17.2.0 or above.
2. Click **File** → **New** → **Project**.
3. Enter a project name and location, choose **PSpice Analog or Mixed A/D** from the options, and then click **OK**.



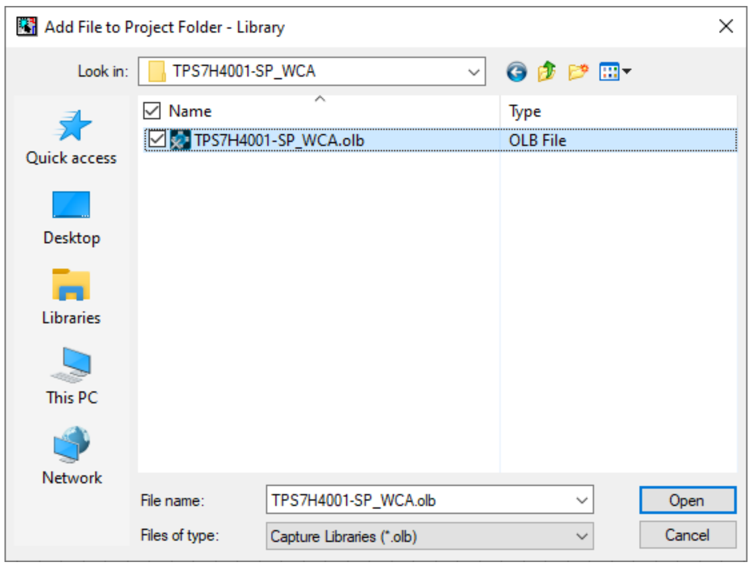
4. When the *Create Pspice Project* dialogue box appears, select the **Create a blank project** option and click **OK**.



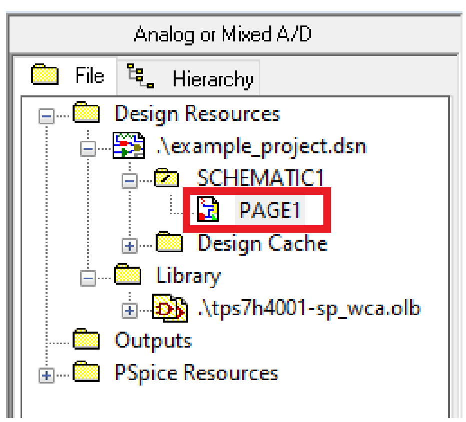
5. A new project is created and the project window displays. Right click the **Library** folder and select **Add File**.



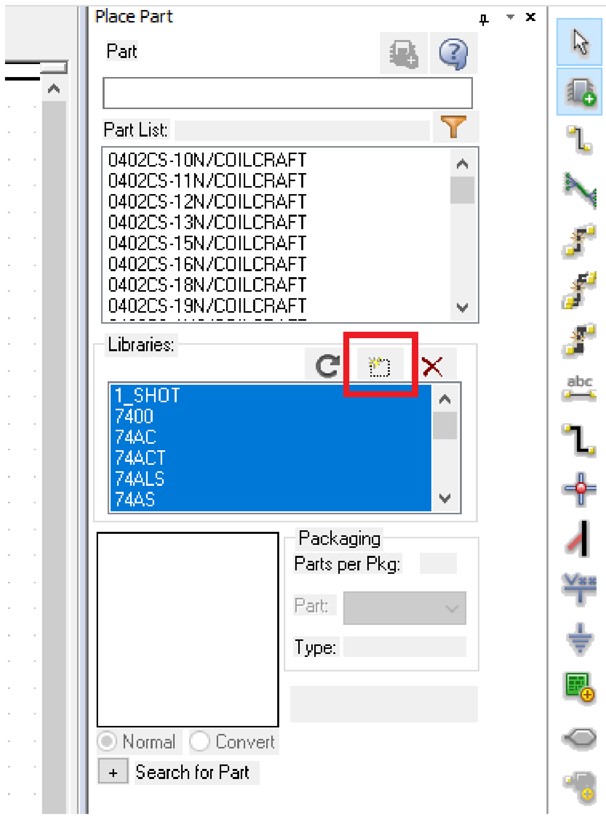
6. Choose the **tps7h4001-sp_wca.olb** file from the downloaded model, add to the dialogue box, and click **Open**. This adds the part symbol to the project.



7. In the project navigation window, open *PAGE1* under the *SCHEMATIC1* folder below the .dsn file.

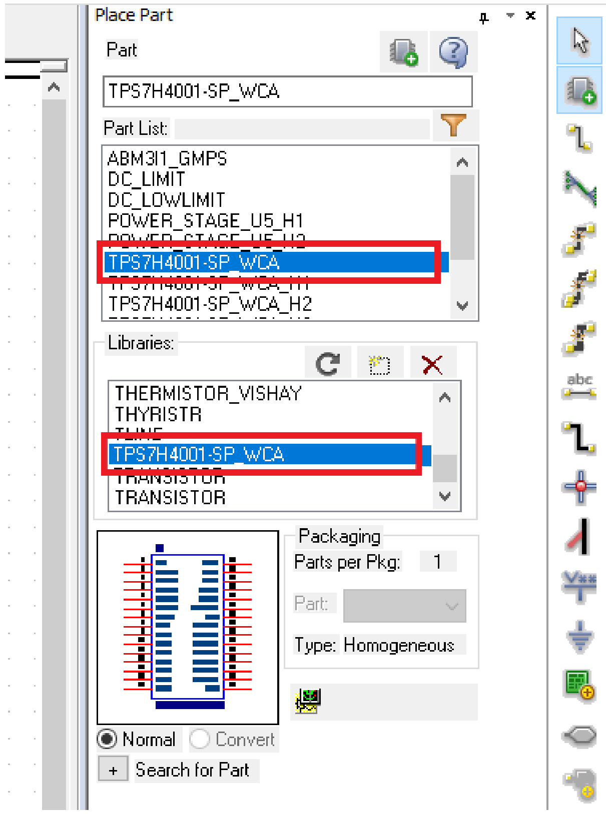


8. Select ***Place*** → ***Part*** to open the *Place Part* window. Click on the square icon shown in the following image to add libraries to the project.

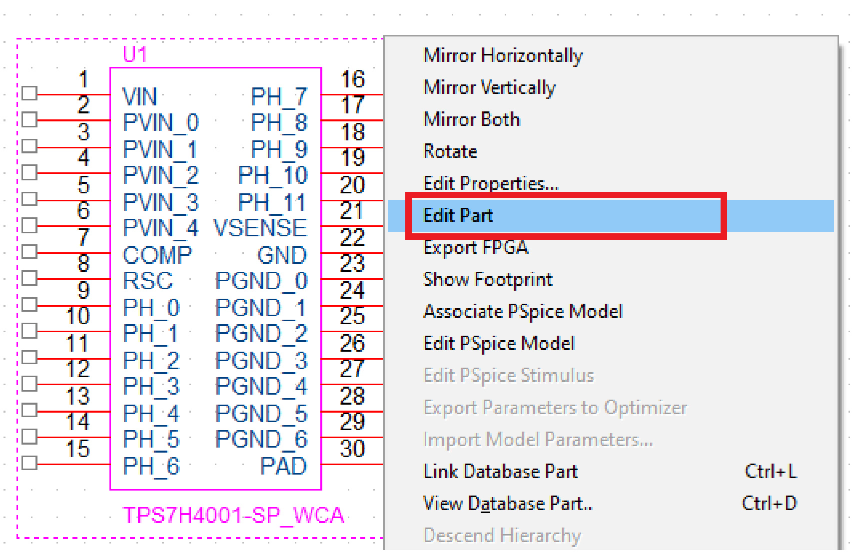


9. If the default library files of PSpice do not appear in the *Libraries* window as shown above in blue, they can be located in the installation directory. To add them, highlight them all and click **Open**. (Default installation directory location: *C:\Cadence\SPB_17.2\tools\capture\library\pspice*)

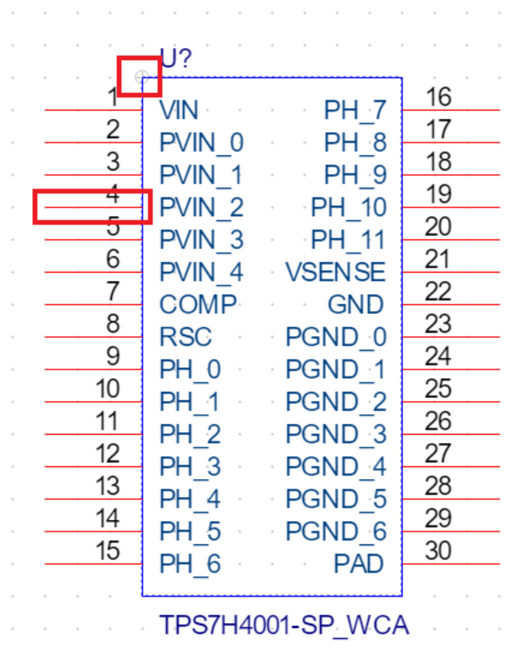
10. In the *Part* search field, type **TPS7H4001-SP_WCA** and select the model in the *Libraries* window.



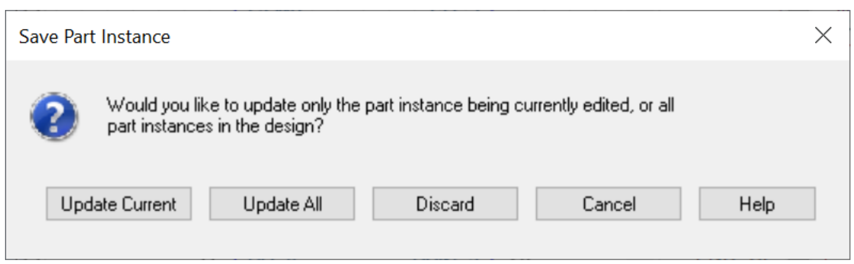
11. In the *Part List* window, double-click on **TPS7H4001-SP_WCA** and place it by clicking the cursor on *PAGE1* to place the part. Once the part is placed, the symbol can be edited by selecting the part, right-clicking, and choosing **Edit Part**.



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

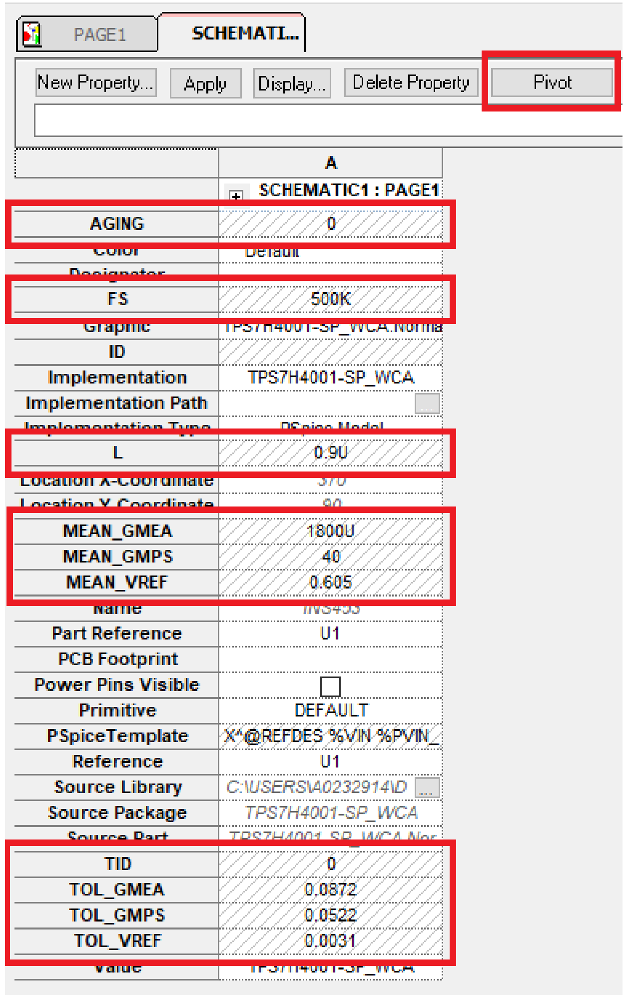


13. After editing the part, close the tab to save changes. Choose **Update Current** in the pop-up.

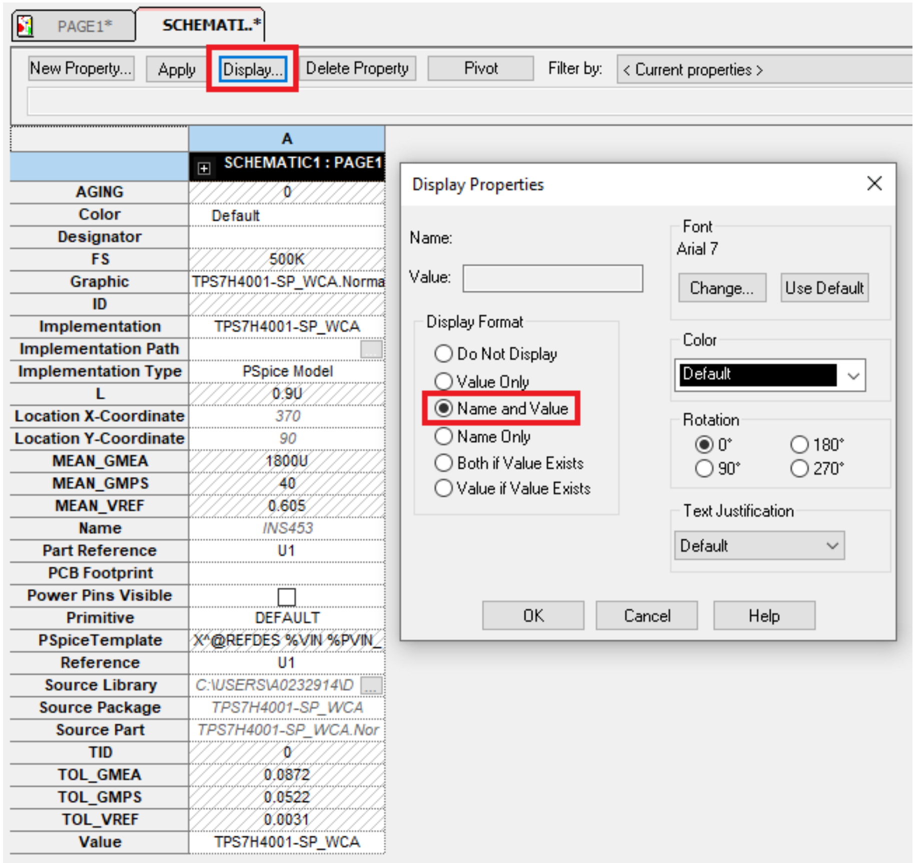


14. The model contains multiple controllable parameters that can be displayed near the part on the schematic to make them easier to view and alter. The default values for these parameters can be found in Section 1.1.

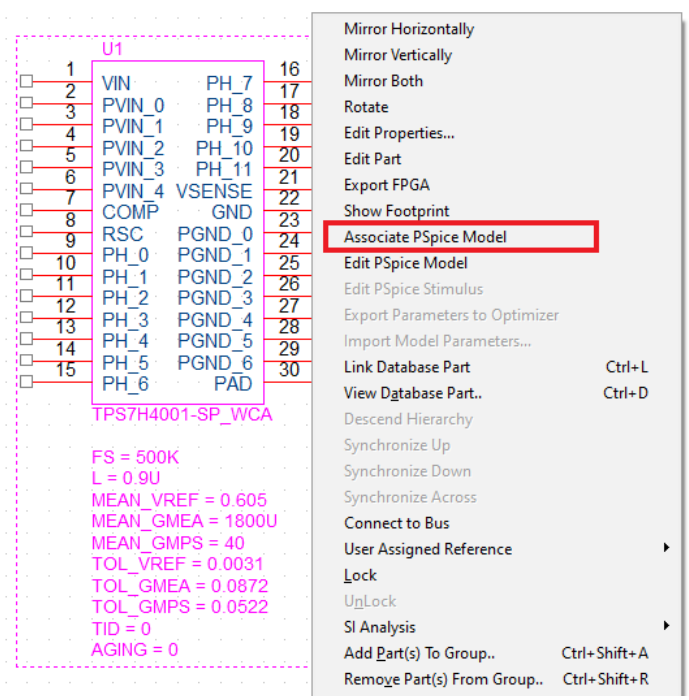
15. To make the parameters visible, double-click the part to open the properties table. If it appears to be transposed compared to the image shown below, clicking **Pivot** will reorient it.



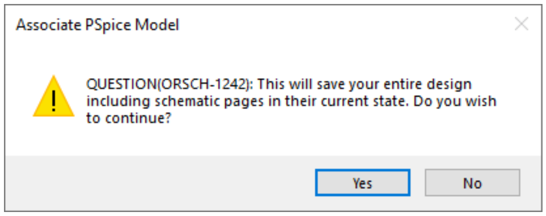
16. Select all the parameters highlighted in the previous image and click the **Display** button. In the pop-up, select **Name and Value** and click **OK**.

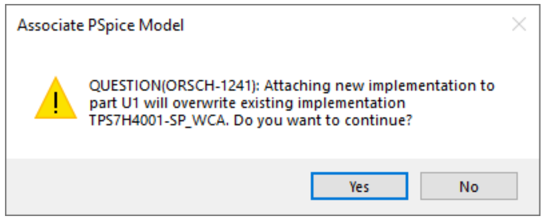


17. Now, the part must be associated with the netlist. Select the part, right-click, and then choose **Associate Pspice Model**.

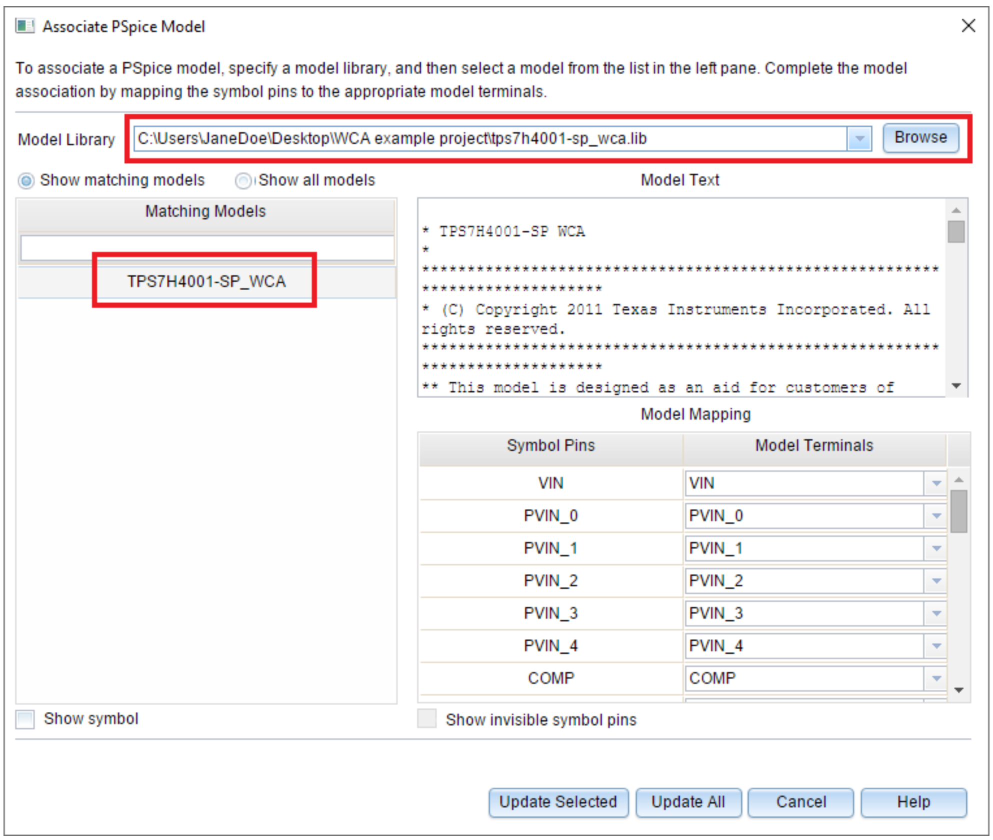


18. Two pop-ups will appear to confirm the operation. Click **Yes** for both.

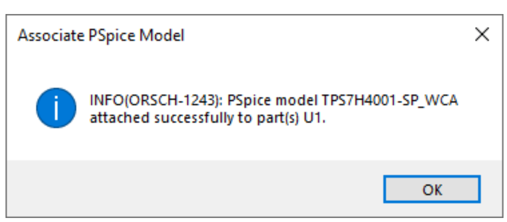




19. In the *Associate Pspice Model* dialogue box, choose the netlist file (***.lib), select the model, and click **Update All**.



20. A pop-up will appear with successful update message. Click **OK**.



21. Add the remaining components to complete the schematic as shown in the schematic below.

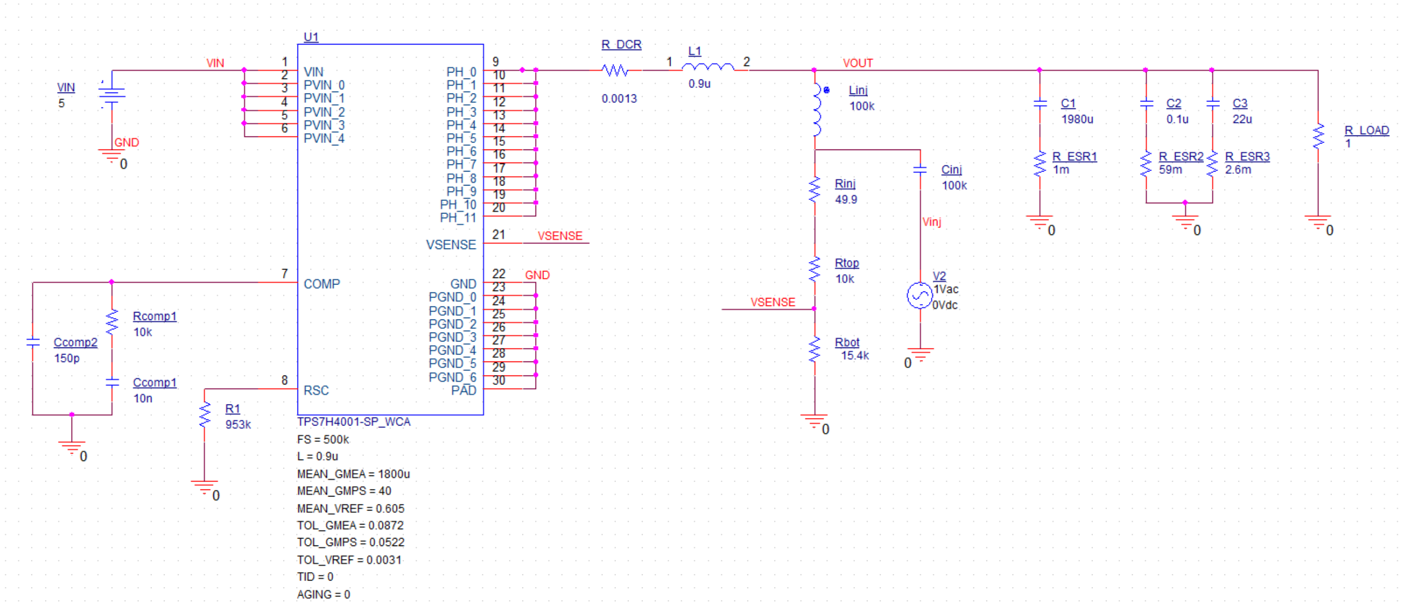
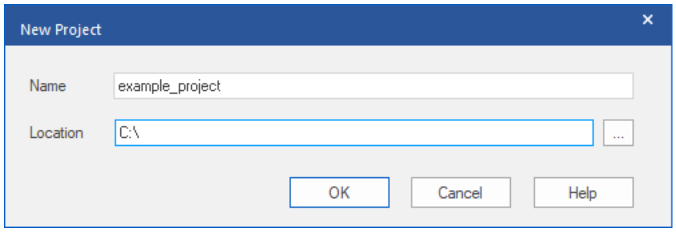
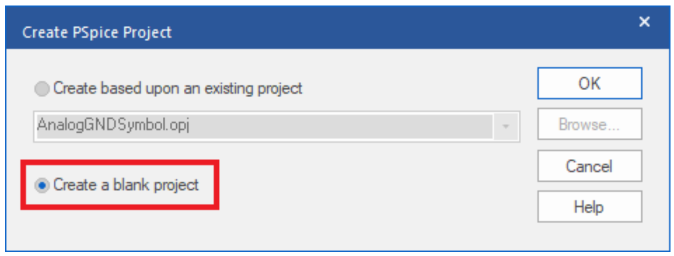


Figure 2-1. Model Schematic

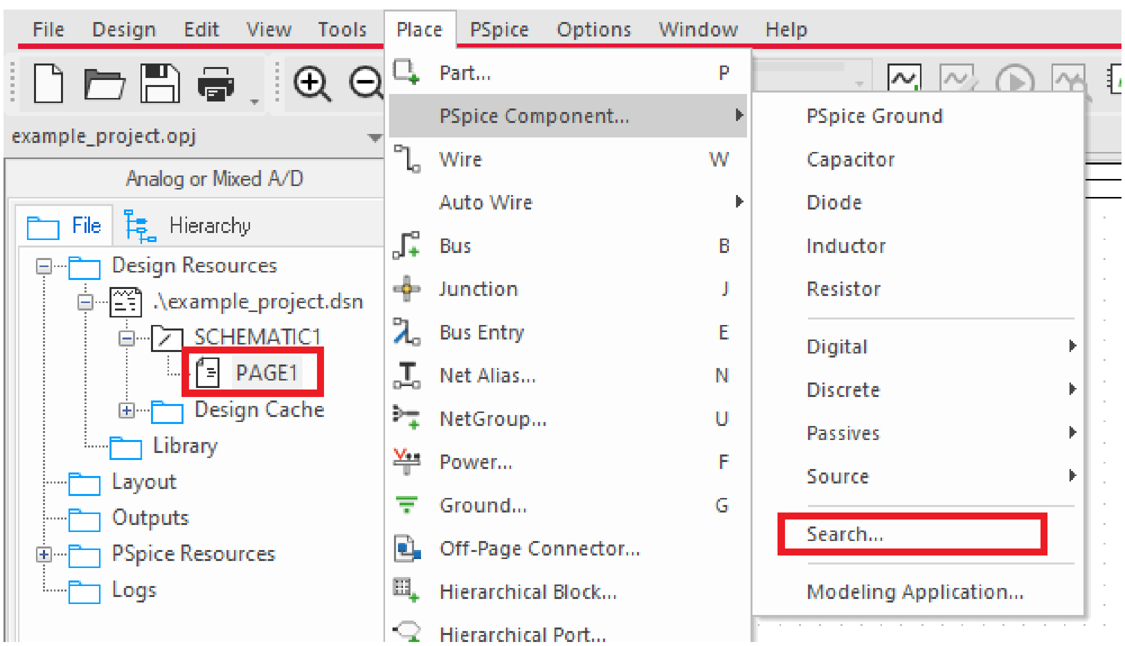
2.2 Using Pspice® for TI

1. Download, install, and open Pspice® *for TI*.
2. Open an existing project, or click on **File→New→Project** to create a new project.div
3. If creating a new project, enter a project name and location and click **OK**. Then select the ***Create a blank project*** option and click **OK** again. A new project will be created and the project window appears.

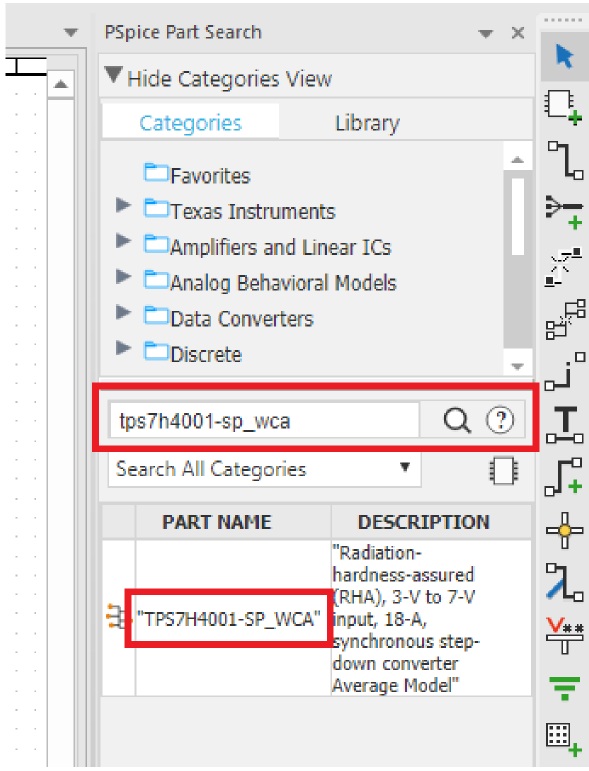




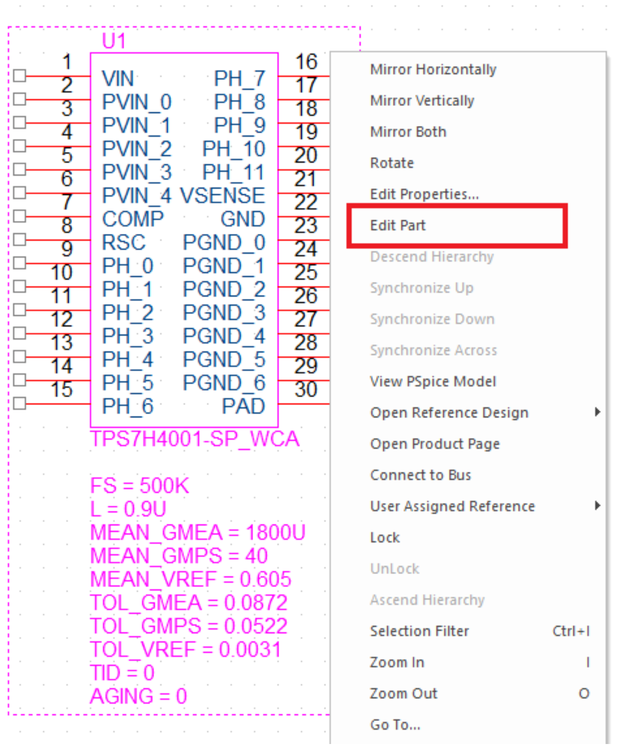
4. Open the schematic page where the part will be placed ("PAGE1" in the example below) and then open the *PSpice Part Search* window by selecting **Place→PSpice Component→Search**



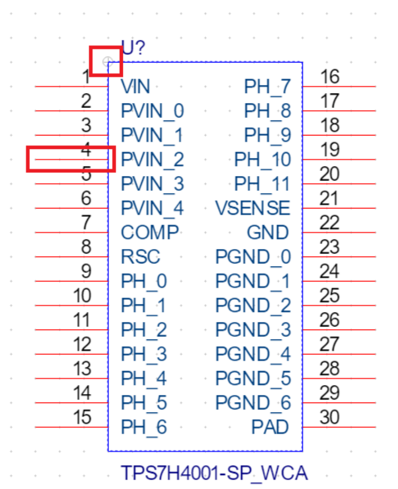
5. Type **tps7h4001-sp_wca** into the search field and press the enter key to display the search results. Double-click on the WCA model that appears in the search results window and then click in the schematic page to place the part.



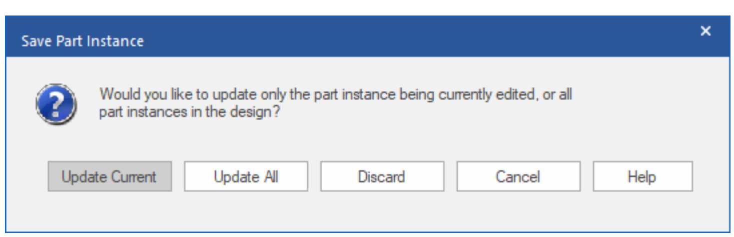
6. Once the part is placed, the symbol can be edited by selecting the part, right-clicking, and choosing **Edit Part**.



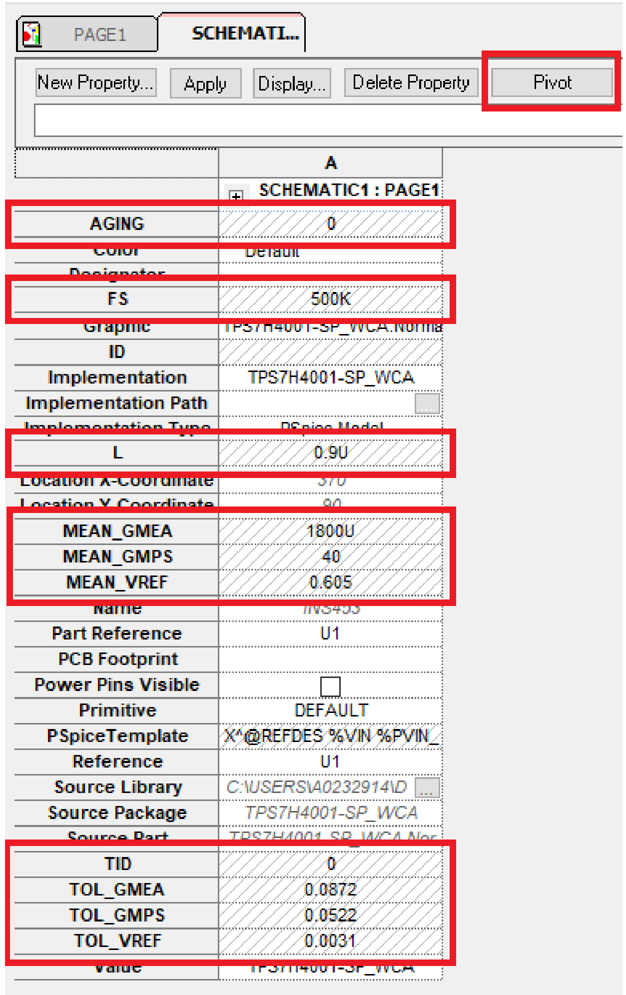
7. The part can be resized and pins can be moved to more convenient positions by clicking and dragging them.



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

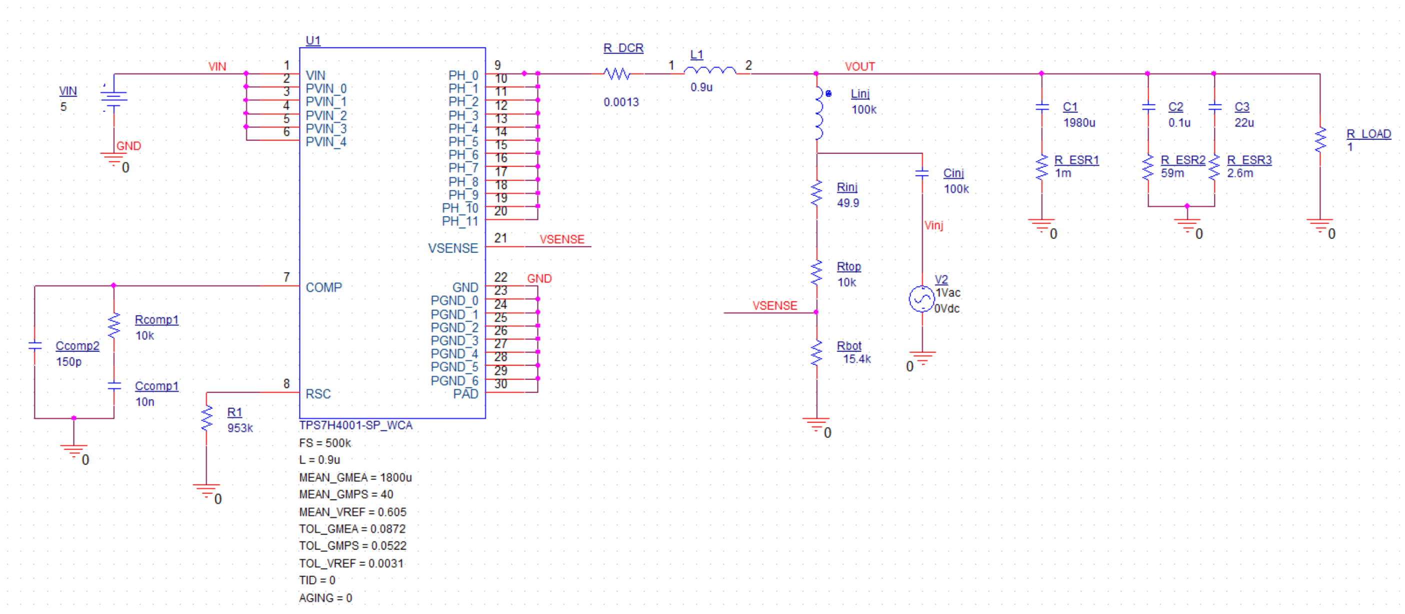


9. To make the parameters visible, double-click the part to open the properties table. If it appears to be transposed compared to the image shown below, clicking **Pivot** will reorient it.



10. The model contains multiple controllable parameters that can be displayed near the part on the schematic to make them easier to view and alter. The default values for these parameters can be found in Section 1.1.
11. Select all the parameters highlighted above and click the **Display** button. In the pop-up, select **Name and Value** and click **OK**.
12. Close out of the *Property Editor* window. The parameters should now be displayed underneath part on the schematic page and their values can be altered by double-clicking on them.

13. Add the remaining components to complete the schematic as shown below.



3 Simulation

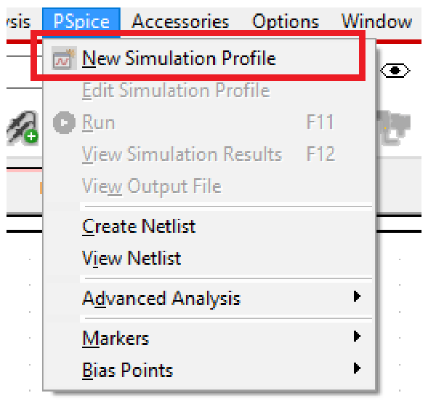
The TPS7H4001-SP WCA Model is intended to simulate the frequency response of the TPS7H4001-SP. It can also be used for Monte Carlo simulation to observe the distribution of simulated behavior observed across a specified sample size of devices.

Section 3.1 describes how to set up simulation of the frequency response and how to view the results as a bode plot.

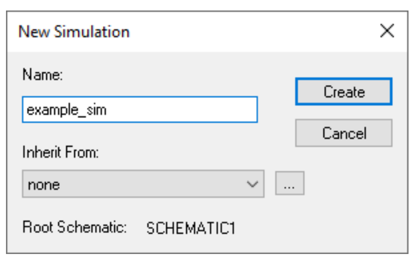
Section 3.2 describes the additional steps needed to set up Monte Carlo simulation and how to view the multiple simulation runs on a bode plot as well as on a histogram.

3.1 Frequency Response

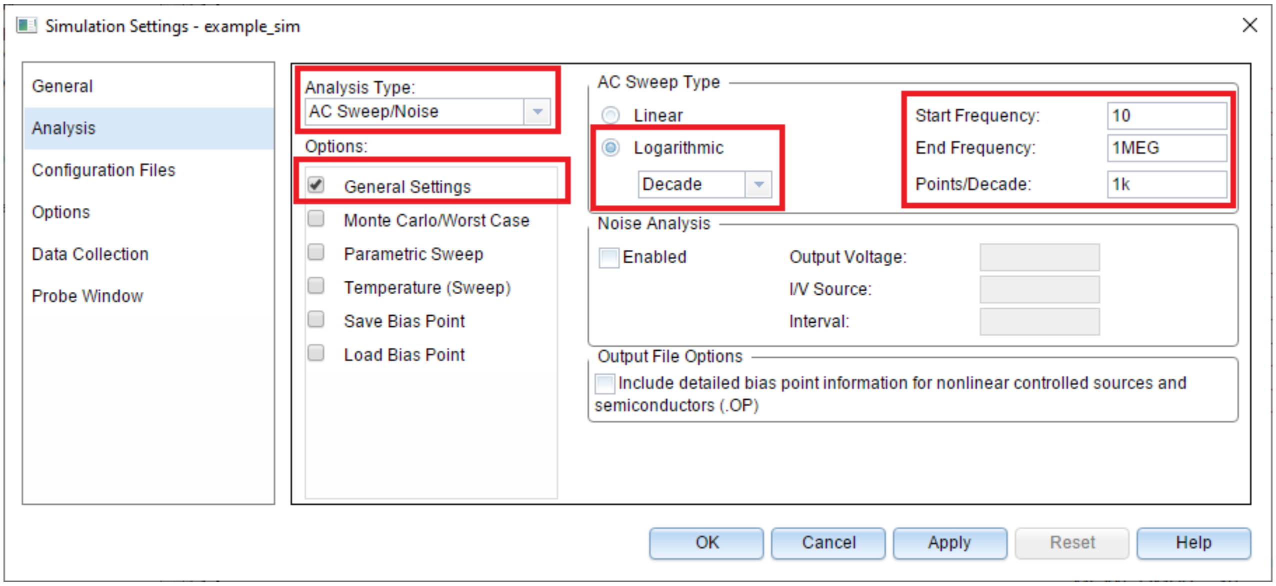
1. Create a new simulation profile simulation profile by clicking on **PSpice** → **New Simulation Profile**.



2. Name the simulation profile and then click **OK**.

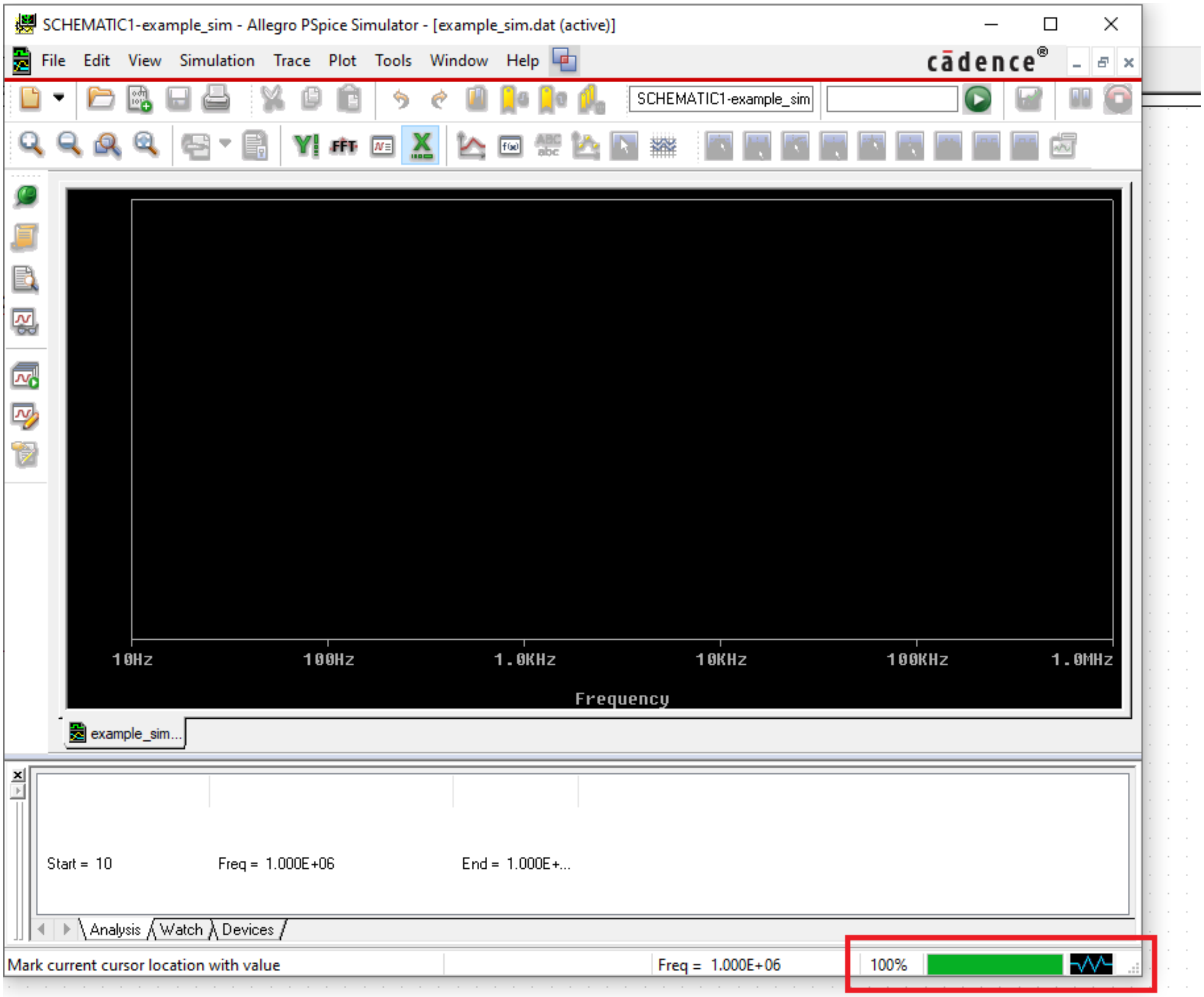


3. The bullets below highlight the parameters that must be filled out in each section of the simulation settings. The image below shows where the parameters are located and gives an example for how they can be set.
   - **Analysis Type**
     – Select "AC Sweep/Noise"
   - **General Settings**
     – Chose the desired start and end frequency
     – Set the number of points per decade to determine simulation resolution
     – Select the desired sweep type ("Logarithmic" is chosen in the example since a Bode plot will be used to display the simulation results)
   - **Temperature (Sweep)**
     – This section can be used to set the simulation temperature

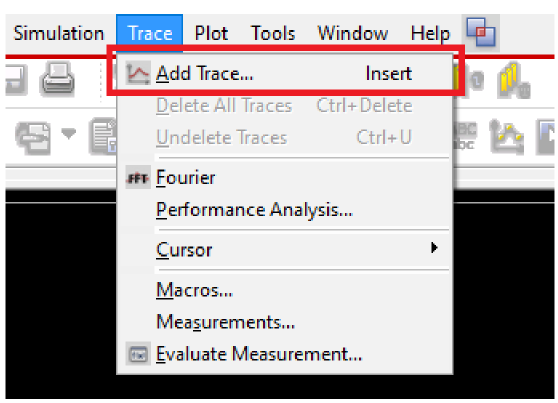


4. Once the parameters have been defined, click **OK** to save and close the simulation settings.

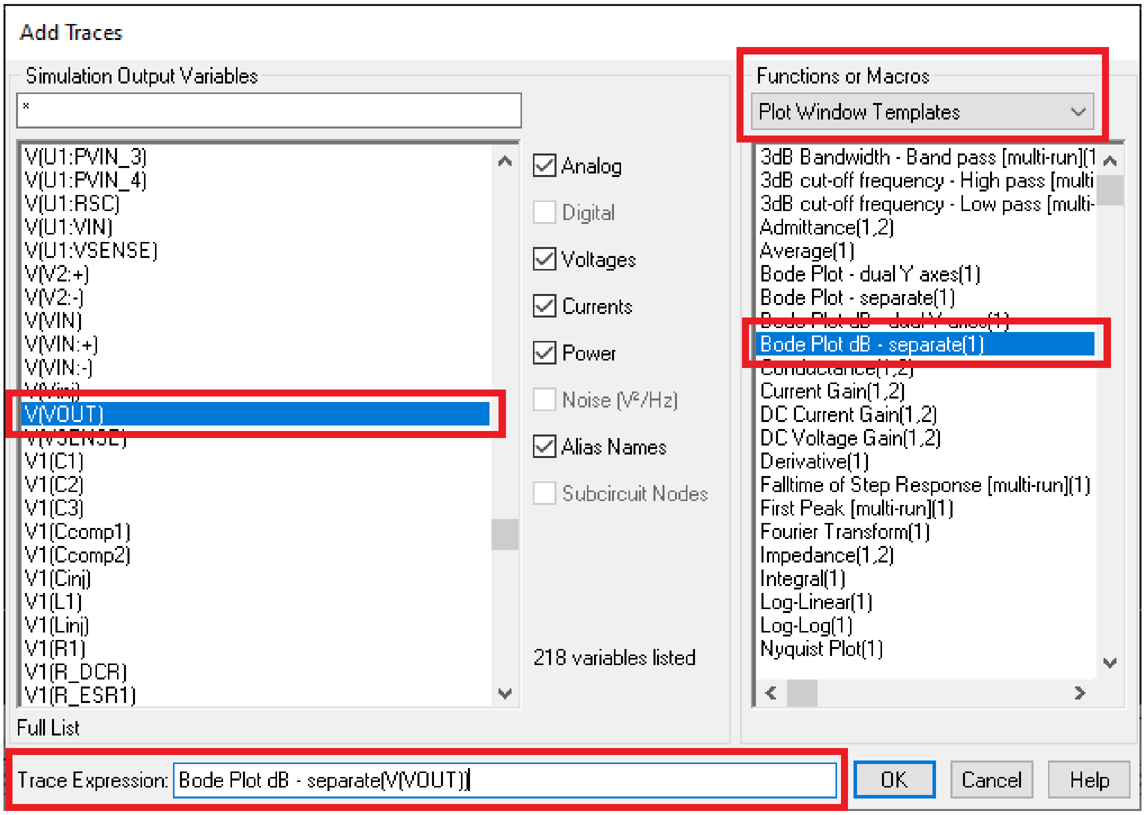
5. Run the Simulation by pressing F11 or selecting **Pspice** → **Run**. The simulation window will open and simulation progress will be shown in the bottom-right corner.



6. After the simulation is complete, the frequency response results can be viewed as a Bode plot by clicking on **Trace** → **Add Trace**.



7. This will bring up another window that lists the signals present in the schematic on the left side and gives options for displaying signals on the right side. Select the options shown in the image below to display the Bode plot or type the following expression directly into the *Trace Expression* field: **Bode Plot dB - separate(V(VOUT))**



8. Press **OK** and the Bode plot will be displayed.

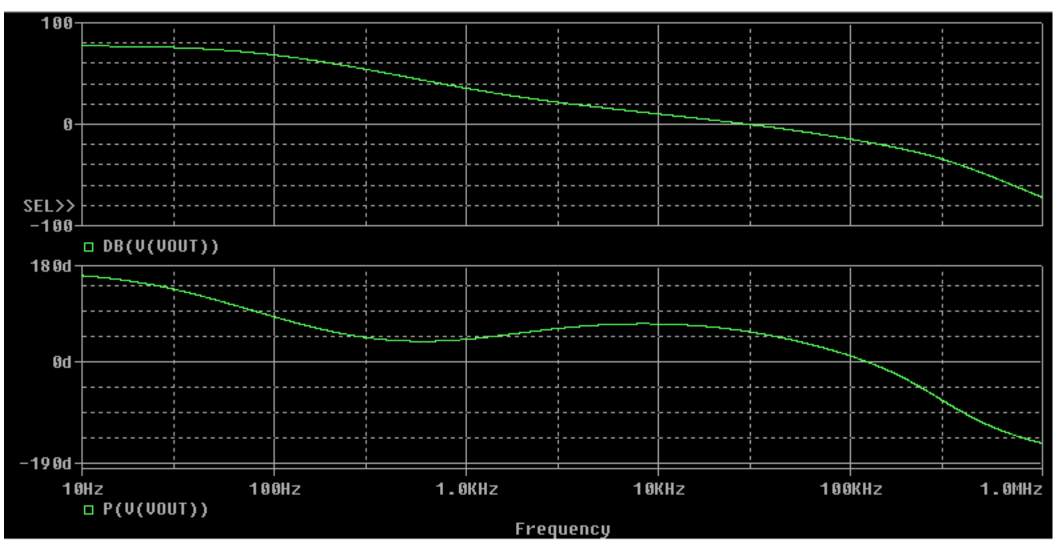
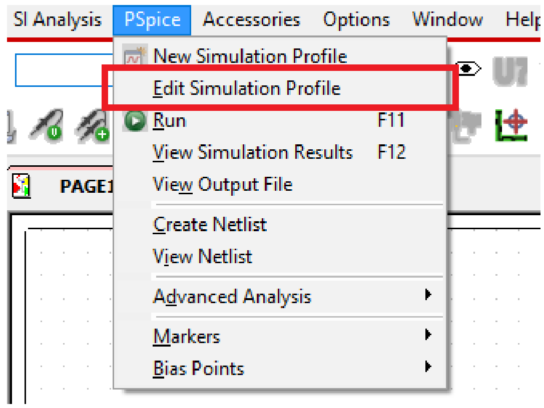


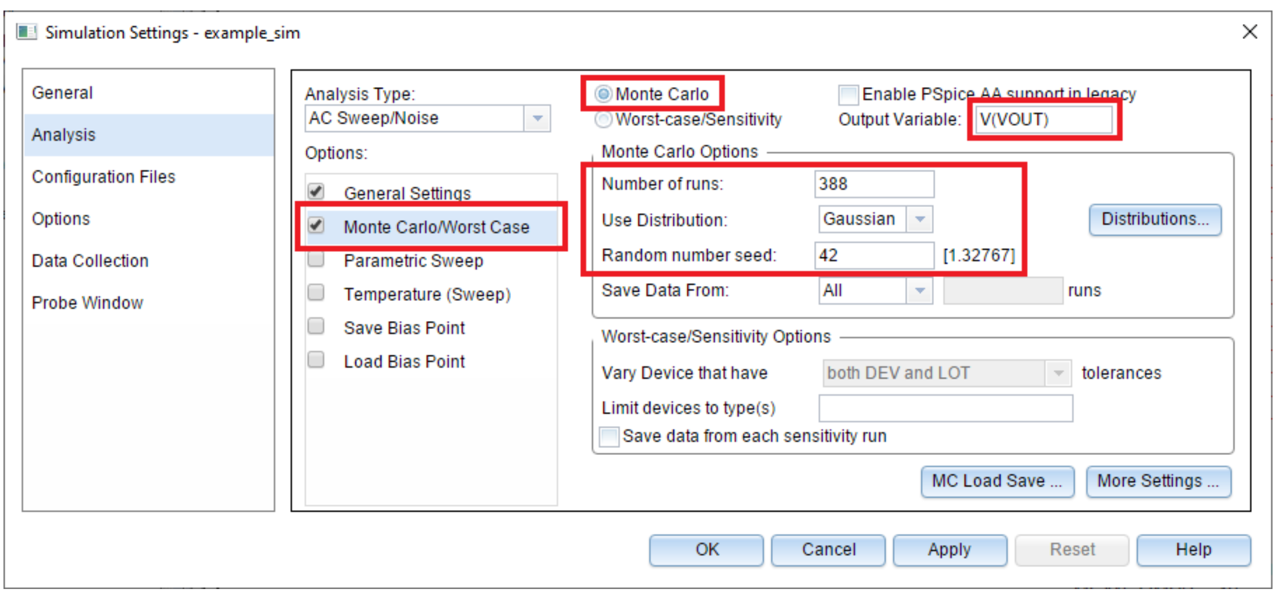
Figure 3-1. Frequency Response Bode Plot

3.2 Monte Carlo

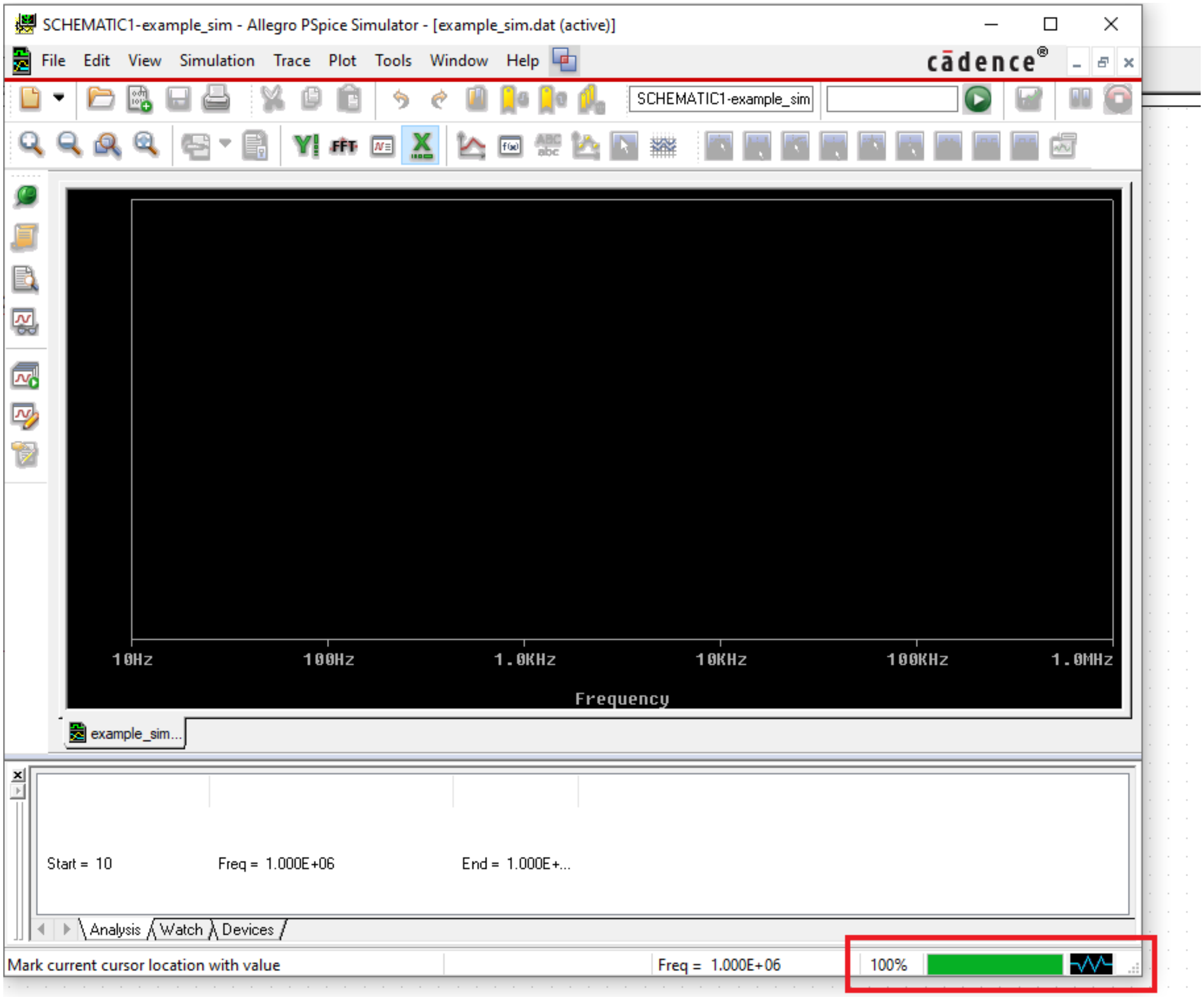
1. Edit the frequency response simulation profile created in Section 3.1 by clicking on **PSpice** → **Edit Simulation Profile**.



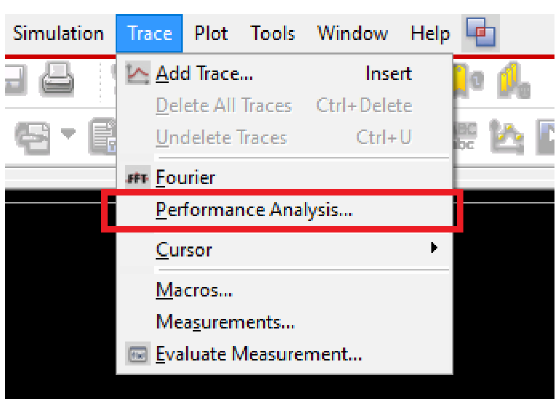
2. In addition to the parameters that were previously set in the **General Settings** section, the following parameters will also need to be set in order to run Monte Carlo simulation:
   - **Monte Carlo/Worst Case**
     - Select the "Monte Carlo" simulation option
     - Define the signal that will be analyzed as the Output Variable (VOUT is used in the example below)
     - Set the number of simulations to run
     - Select "Gaussian" for the distribution
     - Enter a value for the random number seed between 1 and 32,767 if a specific set of random simulations is desired (otherwise this field can be left blank)



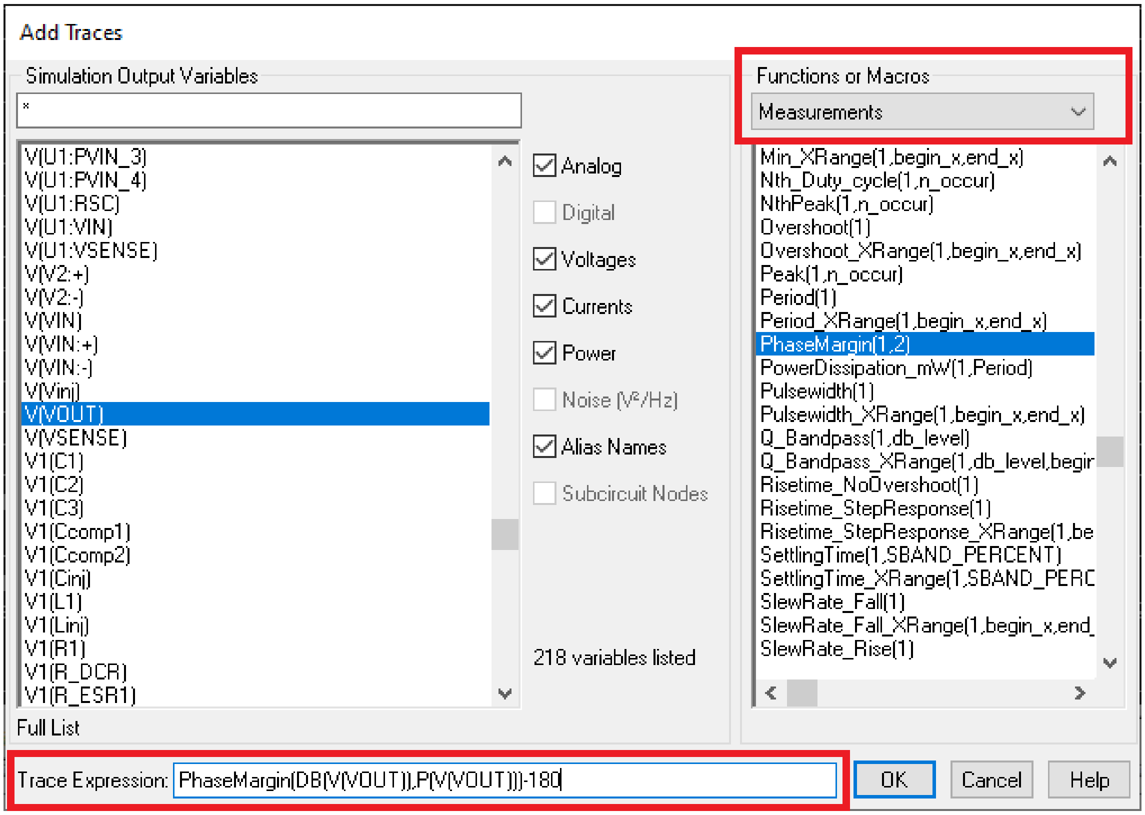
3. Run the simulation by pressing F11 or **PSpice** → **Run**. The simulation window will open and simulation progress will be shown in the bottom-right corner.



4. After the simulations are complete, a window will appear that lists them all. Press **OK**. The frequency response results can be viewed as a Bode plot by following the directions in Section 3.1 starting from step 6.
5. In order to view the results as a histogram, select **Trace** → **Performance Analysis**. A pop-up will appear. Click **OK**.



6. Right click on the plot area and click **Add Trace**.
7. Once again, the desired signal and display method can be chosen. For example, phase margin can be displayed by entering the following expression into the *Trace Expression* field: **PhaseMargin(DB(V(VOUT)),P(V(VOUT)))-180**



8. Click **OK** and the histogram will be displayed along with statistical information that describes the distribution of the simulation results, such as min, max, standard deviation, and mean.

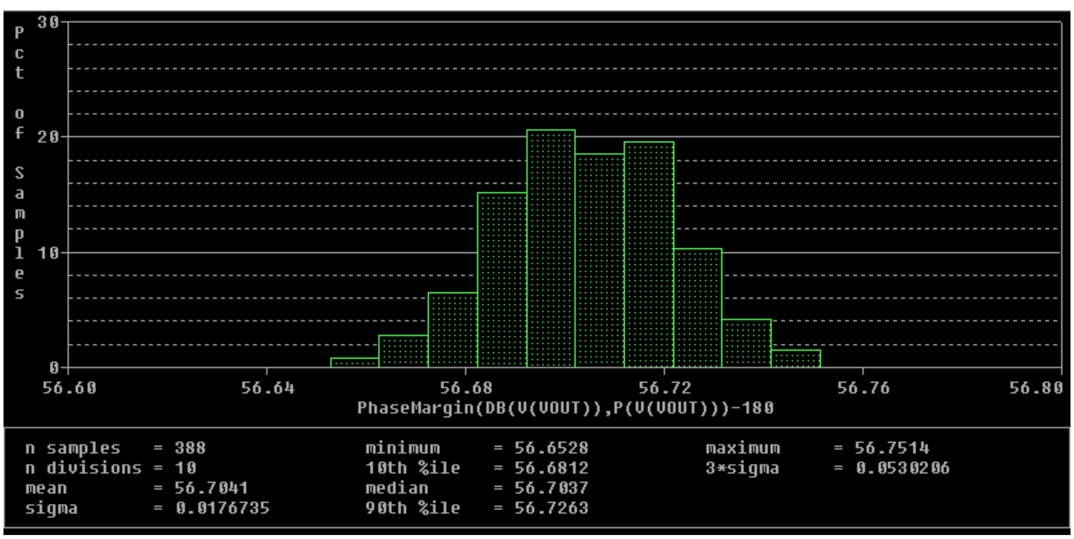


Figure 3-2. Monte Carlo Histogram of Phase Margin