

1

# 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<sup>®</sup> 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

1	TPS50601A-SP WCA Model Specification	2
2	Default Parameters	2
3	Example of using model with Cadence Pspice (17.2.0)	3
4	Simulating the Model	12

#### List of Figures

1	Model Schematic	11
2	Output Voltage Monte Carlo Analysis	14
3	Frequency Response Bode Plot	19
4	Frequency Response Histogram	20

#### 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

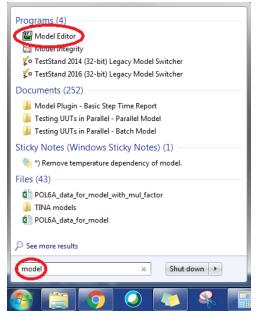


# 3 Example of using model with Cadence Pspice (17.2.0)

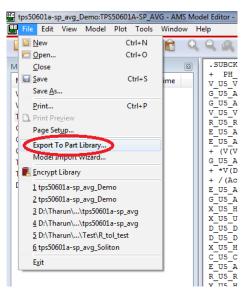
### 3.1 Creating a part from the netlist file

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  $\rightarrow$  Open and choose the netlist file (\*\*\*.lib).
- 3. Once the netlist opens, select File  $\rightarrow$  Export to Part Library...



3



Example of using model with Cadence Pspice (17.2.0)

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  $\rightarrow$  New  $\rightarrow$  Project.
- 3. Enter a project name and location, choose *PSpice Analog or Mixed A/D* from the options, and click "OK".

×
OK
Cancel
Help
Tip for New Users
Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
Learn With PSpice - Examples And AppNotes
Browse

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".





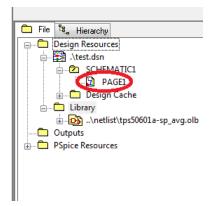
6. Choose the \*\*\*.olb file that was previously created, add to the dialogue box, and click "Open". This will add the part symbol to the project.

👪 Add File to Project Folder - Library					×
Look in:	퉬 NetList		•	G 🤌 📂 🛄 -	
Recent Places	Name TPS50601A-	SP_AVG		Date modified 3/19/2019 4:26 PM	Type OLB File
Desktop					
Libraries					
Computer					
Network	•				4
NEWOK	File name: Files of type:	TPS50601A-SP_AVC		- [	Open Cancel

# 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:

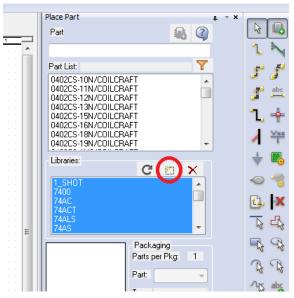
1. Open PAGE1 under the SCHEMATIC1 folder below the .dsn file.



5

Example of using model with Cadence Pspice (17.2.0)

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.

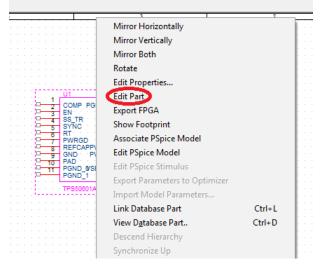


- Select all default library files of PSPICE from installation directory and click "Open". (Default Location: "C:\Cadence\SPB\_17.2\tools\capture\library\pspice") Note: This step can be omitted if the default libraries have already been added to Capture.
- 4. In the *Part* search window, type "TPS50601A-SP\_AVG" and select the model.

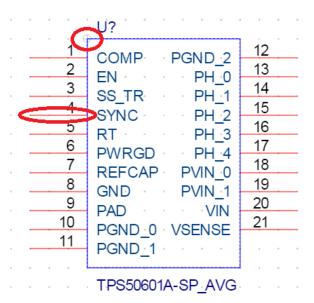
	Place Part 📮 👻 🗙	
	Part 🔬 🔇	
	TPS50601A-SP_AVG	1 🔨
	Part List:	* *
:	CCM_U5_H1 CCM_U5_H2	ahc
		J
	TPS50601A-SP_AVG	Ղ 🔶
	TPS50601A-SP_AVG_H2 VARICAP	/ 1
	Libraries: C 🐑 🗙	÷ 📭
		~ 👘
:	TUNE	🔁, 🗙
· · · =		下小
	Packaging	<b>R</b>
		NR
		AK abc
:	7 spece part 18 6 sho part 10 10 part 10	- 13 and
		<b>(</b> ]
	Normal      Convert	
	+ Search for Part	



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.

	Save Part Instance	2	×
Would you like to update only the part instance being currently edited, or all part instances in the design?			
Update Current Updat	e All Discard	Cancel	Help

7



Example of using model with Cadence Pspice (17.2.0)

www.ti.com

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:

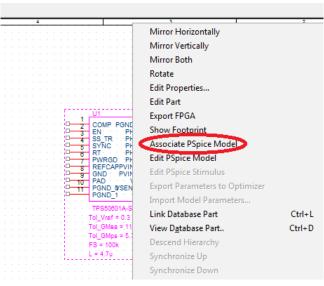
	Α	
	SCHEMATIC1 : PAGE1	
Color	Default	
Designator		
FS	100k	
Graphic	TPS50601A-SP_AVG.Norm	
ID		
Implementation	TPS50601A-SP_AVG	
Implementation Path		
Implementation Type	PSpice Model	
	4.7u	
Location X-Coordinate	360	
Location Y-Coordinate	te 210	
Name	INS64	
Part Reference	U1	
PCB Footprint		
Power Pins Visible		
Primitive	DEFAULT	
PSpiceTemplate	X^@REFDES %COMP %EN	
Reference	U1	
Source Library	D:\THARUN\SPICE TPS	
Source Package	TPS50601A-SP_AVG	
Source Part	TPS50601A-SP_AVG.Norm	
TID	0	
Tol_GMea	11.52	
Tol_GMps	5.75	
Tol_Vref	0.3	
Value	TPS50601A-SP_AVG	

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

Start Page TPS50601* PAGE1* SCHEMATL.*					
New Property Apply Display Delete Property Pivot Filter by: < Current properties >					
	$\bigcirc$				
	A SCHEMATIC1 : PAGE1	Display Properties			
Color	Default	Nama	Font		
Designator		Name:	Arial 7		
FS	100K	Value:			
Graphic	TPS50601A-SP_AVG.Norma	Value.	Change Use Default		
ID		Display Format			
Implementation	TPS50601A-SP_AVG		Color		
Implementation Path		🔘 Do Not Display			
Implementation Type	PSpice Model	Value Only	Default 👻		
L	4.70	Name and Value			
Location X-Coordinate	280		Rotation		
Location Y-Coordinate	300	Name Only	O <sup>*</sup> ○ 180 <sup>*</sup>		
Name	INS64	🔘 Both if Value Exists	90° 270°		
Part Reference	U1	💿 Value if Value Exists			
PCB Footprint			Text Justification		
Power Pins Visible			Default 👻		
Primitive	DEFAULT		Derdak		
PSpiceTemplate	X^@REFDES %GND %EN %				
Reference	U1	ОК	Canaal Hala		
Source Library	D:\THARUN\SPICE TPS		Cancel Help		
Source Package	TPS50601A-SP_AVG	<u></u>			



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



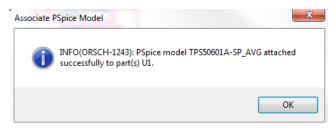
11. Two pop-ups will appear to confirm the operation. Click "Yes" for both.

Associate	PSpice Model
4	QUESTION(ORSCH-1242): This will save your entire design including schematic pages in their current state. Do you wish to continue?
	Yes No
Associate F	PSpice Model
<b></b>	QUESTION(ORSCH-1241): Attaching new implementation to part U1 will overwrite existing implementation TPS50601A-SP_AVG. Do you want to continue?
	Yes No

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

	in the left pane. Complete the model
er Guide\tps50601a-sp_avg.lib	Browse
М	odel Text
PWRGD REFCAP GND PAD         PAD         PC           PH_1 PH_2         +         PH_3 PH_4 PVIN_0	VG COMP EN SS_TR SYNC RT GND_0 PGND_1 PGND_2 PH_0 IN_1 VIN VSENSE N 0 0.048Vdc US_N14480023 VALUE { }
M	odel Mapping
Symbol Pins	Model Terminals
COMP	COMP
EN	EN 💌
SS_TR	SS_TR 🗸
SYNC	SYNC 💌
RT	RT 💌
PWRGD	PWRGD 🔻
REFCAP	REFCAP
Show invisible symbol pins	I <b></b>
	iate model terminals. er Guide\tps50601a-sp_avg.lib *\$ .SUBCKT TPS50601A-SP_A' PWRGD REFCAP GND FAD PO PH_1 PH_2 + PH_3 PH_4 PVIN_0 PV: V_US_V1 US_CLPI G_US_ABMII1 0 T (V(US_DCX) * 100u) M Symbol Pins COMP EN SS_TR SS_TR SYNC RT PWRGD REFCAP

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





Example of using model with Cadence Pspice (17.2.0)

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

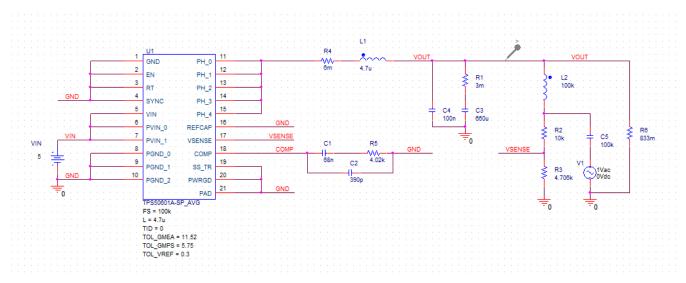


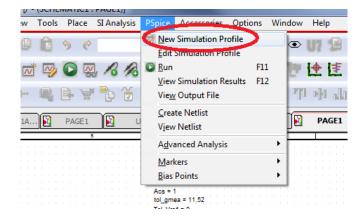
Figure 1. Model Schematic



## 4 Simulating the Model

### 4.1 Monte Carlo Analysis of Output Voltage

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





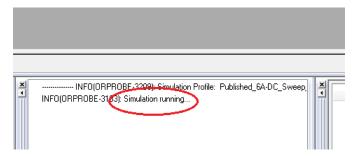
- 2. Set the parameters based on the image below using the following settings:
  - Analysis Type DC-Sweep
  - Primary Sweep VIN (Input Voltage) from 5 V to 5 V
  - Monte Carlo 500 runs with Gaussian distribution
  - Output Variable V(VOUT) Output Voltage

Simulation Settings - demo_test	TWENT IN	×
General Analysis Configuration Files Options Data Collection Probe Window	Analysis Type. DC Sweep v Options: Primary Sweep Secondary Sweep Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	Sweep Variable         Image: Voltage source       Name:         Voltage source       Model type:         Current source       Model type:         Global parameter       Model name:         Model parameter       Parameter name:         Model parameter       Parameter name:         Temperature       Start Value:         Sweep Type       Start Value:         Incement:       1         Value List       Increment:
		OK Cancel Apply Reset Help
Simulation Settings - demo_test	TWENT PI	×
General Analysis Configuration Files	Analysis Type: DC Sweep v Options: Primary Sweep	Monte Carlo  Worst-case/Sensitivity  Monte Carlo Options  Number of runs:  500

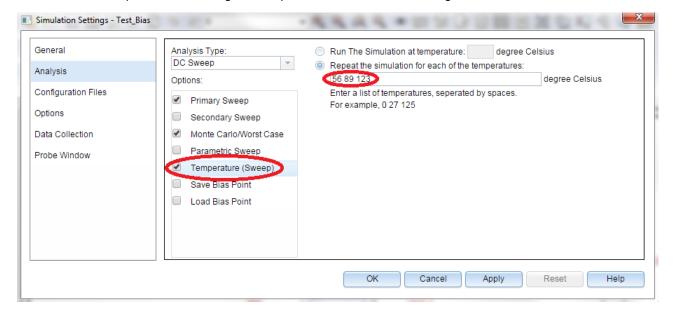
Options	Secondary Sweep Use Distribution: Gaussian 💌 Distributions
Data Collection	Monte Carlo/Worst Case Random number seed: [1.4654] [1.32767]
Probe Window	Parametric Sweep Save Data From: All 💌 runs
	Temperature (Sweep)     Worst-case/Sensitivity Options
	Save Bias Point Vary Device that have both DEV and LOT v tolerances
	Limit devices to type(s)
	Save data from each sensitivity run
	MC Load Save More Settings
	OK Cancel Apply Reset Help

- 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  $PSpice \rightarrow Run$ .

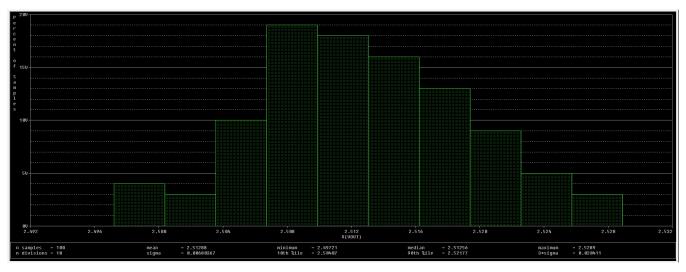
5. Wait for the simulation completion in the console window of AMS Simulator.

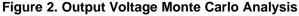


The simulation can be performed at various temperatures by selecting temperature sweep in the simulation profile and setting the temperature values in ascending order.



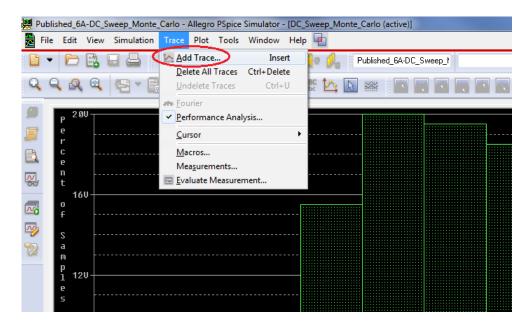
The output windows come up once the simulation is completed. The results window shows the histogram of the output voltage (V(VOUT)) with Monte Carlo analysis. The mean, sigma, min, max, etc. are displayed at the bottom of the window.







To analyze other nodes, click on Trace  $\rightarrow$  Add Trace.

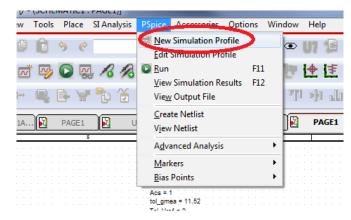


This will bring in another pop-up with the netlist present in the schematics. Select the desired trace and click "OK". The histogram of the selected node will be updated.

	Add Traces			)
	Simulation Output Variables	_	Functions or Macros	
			<b></b>	
-	I(C11)	<ul> <li>Analog</li> </ul>	#	
	I(C11:1) I(C12)	Digital	0 ×	
	I(C12:1)		+	_
	l(C13) l(C13:1)	Voltages	· /	
	I(C14)	Currents	@ ABS()	
	I(C14:1) I(C8)	V Power	ARCTAN()	
	l (C8:1) l (Cini)	_	ATAN()	
	I(Cinj:1)	Noise (V²/Hz)	AVGX(,)	
	[[1]   [[1:+]	📝 Alias Names	COS()	
	1(L2)	Subcircuit Nodes	D() ) DB()	
	l(L2:1) l(Linj)		ENVMAX(,) ENVMIN(,)	
	l(Linj:1)		EXP()	
	l (R13) l (R13:1)		G() IMG()	
	l(R16)		LOG()	
	I(R16:1) I(R22)	657 variables listed	LOG10() M()	
	I(B22:1)	<del>.</del>	MAX()	
	Full List			
	Trace Expression:		OK Cancel Help	
				<b>81</b> 2m
		V(U1.VREF_TOL)	)	

### 4.2 Frequency Response Analysis

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





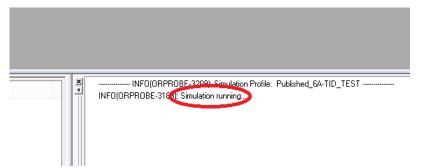
- 2. Set the parameters based on the image provided below using the following settings:
  - Analysis Type AC-Sweep
  - General Settings Set start and end frequency, number of points per decade, and sweep type
  - Monte Carlo 500 runs with Gaussian distribution
  - Output Variable V(VOUT) Output Voltage

	Sinulati	on Settings - TID_TEST		
General	Analysis Type:	АС Ѕweep Туре ———		
Analysis	AC Sweep/Noise	Linear	Start Frequer	
Configuration Files	Options:	Logarithmic	End Frequen	
Options	General Settings	Decade	Points/Decad	le: 10000
	Monte Carlo/Worst Case	Noise Analysis —		
Data Collection	Parametric Sweep	Enabled Out	tput Voltage:	
Probe Window	Temperature (Sweep)		Source:	
	Save Bias Point		erval:	
	Load Bias Point	Output File Options — Include detailed bias poin	t information for poplinger	controlled courses and
		semiconductors (.OP)	it mormation for nonlinear	controlled sources and
		@Monte Carlo	Enable PSnice A4-e	
Simulation Settings - Test_ General Analysis	Bias Analysis Type: AC Sweep/Noise	Monte Carlo	Enable PSpice AA o Output Variable	
General Analysis	Analysis Type:	Worst-case/Sensitivity	Output Variable	
General Analysis Configuration Files	Analysis Type: AC Sweep/Noise	Worst-case/Sensitivity Monte Carlo Options — Number of runs:	Output Variable	upport in legacy DUT)
General Analysis Configuration Files	Analysis Type: AC Sweep/Noise	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution:	Output Variable V(VC	Distributions
General Analysis Configuration Files Options	Analysis Type: AC Sweep/Noise Options: General Settings	Worst-Case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed:	Output Variable V(VC 100 Gaussian v 6748 [1.32767]	DUT)
General Analysis Configuration Files Options Data Collection	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep)	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution:	Output Variable V(VC	Distributions
General Analysis Configuration Files Options Data Collection	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point	Worst-Case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed:	Output Variable V(VC 100 Gaussian • 6748 [1.32767] All •	DUT)
General Analysis Configuration Files Options Data Collection	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep)	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed: Save Data From:	Output Variable V(VC 100 Gaussian • 6748 [1.32767] All •	DUT)
General	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed: Save Data From: Worst-case/Sensitivity Optio	Output Variable         V(V0           100	Distributions
General Analysis Configuration Files Options Data Collection	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed: Save Data From: Worst-case/Sensitivity Optio Vary Device that have	Output Variable V(VC Gaussian = 6748 [1.32767] All = both DEV and LOT	Distributions
General Analysis Configuration Files Options Data Collection	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point	Worst-case/Sensitivity Monte Carlo Options Number of runs: Use Distribution: Random number seed: Save Data From: Worst-case/Sensitivity Optio Vary Device that have Limit devices to type(s)	Output Variable V(VC Gaussian = 6748 [1.32767] All = both DEV and LOT	Distributions

3. Run the Simulation by pressing F11 or  $Pspice \rightarrow Run$ .

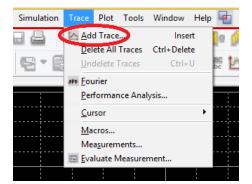
#### Simulating the Model

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

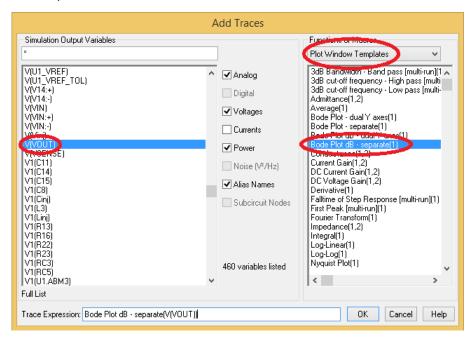


### 4.2.1 Analyzing Frequency Response with Bode Plot

1. To view the frequency response as Bode plot, click on  $Trace \rightarrow Add Trace$ .



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.

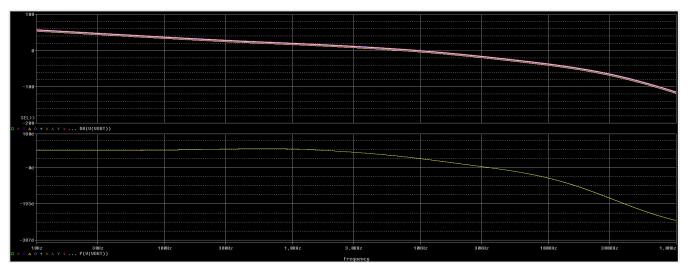
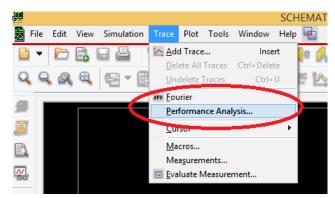


Figure 3. Frequency Response Bode Plot

### 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



Page

# **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

Changed title from "TPS50601A-SP Model User's Guide" to "TPS50601A-SP Worst Case Analysis Unencrypted PSpice
 Average Model User's Guide"
 1

# IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), 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, regulatory 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 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.

TI objects to and rejects any additional or different terms you may have proposed.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265 Copyright © 2022, Texas Instruments Incorporated