

TPS7H4001-SP Model

This user guide is intended to demonstrate use of the Pspice models for the TPS7H4001-SP synchronous buck converter. Both the average model and transient model are covered in this guide along with instructions on how to import the unencrypted model netlist into Cadence Pspice®. The first half of the guide focuses on using the models with Pspice. The second half of the guide addresses how to simulate the modeled parameters.

Contents

1	Example of using model with Cadence Pspice (17.2.0)	2
2	Simulation of Average Model for TPS7H4001-SP	11
3	Simulation of Transient Model for TPS7H4001-SP	15

List of Figures

1	Model Schematic	10
2	Frequency Response	14
3	Transient Response.....	16

Trademarks

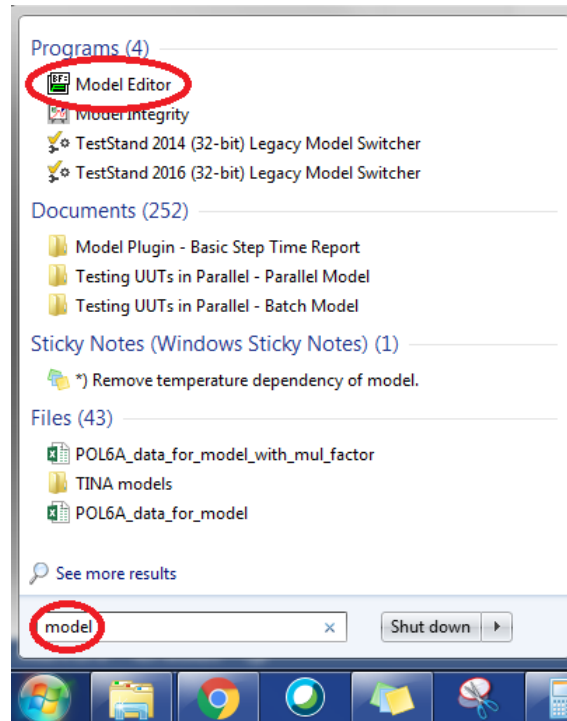
Pspice, Capture are registered trademarks of Cadence.
 All other trademarks are the property of their respective owners.

1 Example of using model with Cadence Pspice (17.2.0)

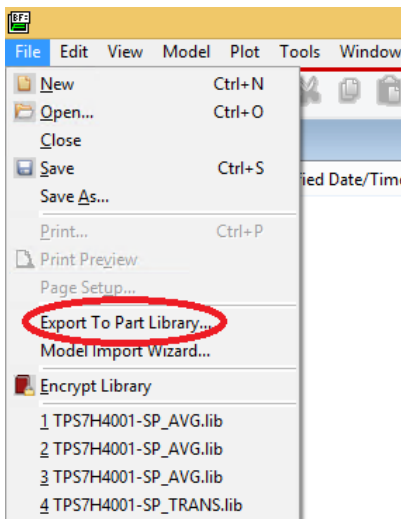
1.1 Creating a part from the netlist file

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...*

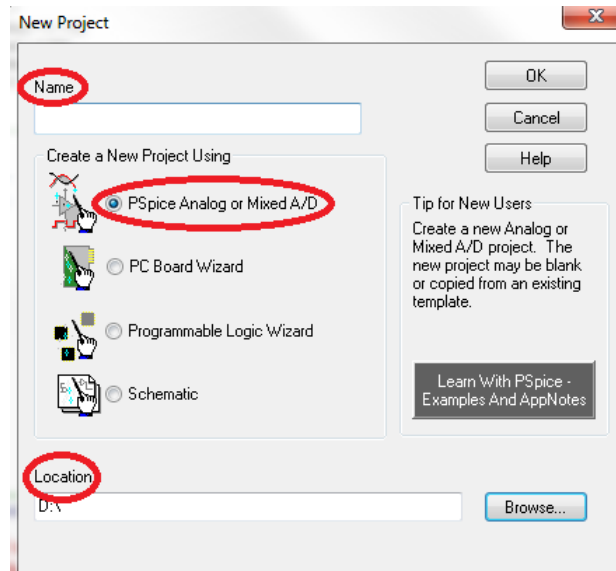


4. The *Create Parts for Library* dialogue box will open. Click "OK" to generate the part library in the default location. This will create a *.olb file with the same name and location as the netlist file.

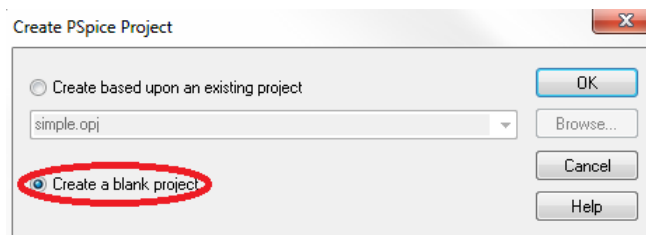
1.2 Steps to create project in Capture®

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".



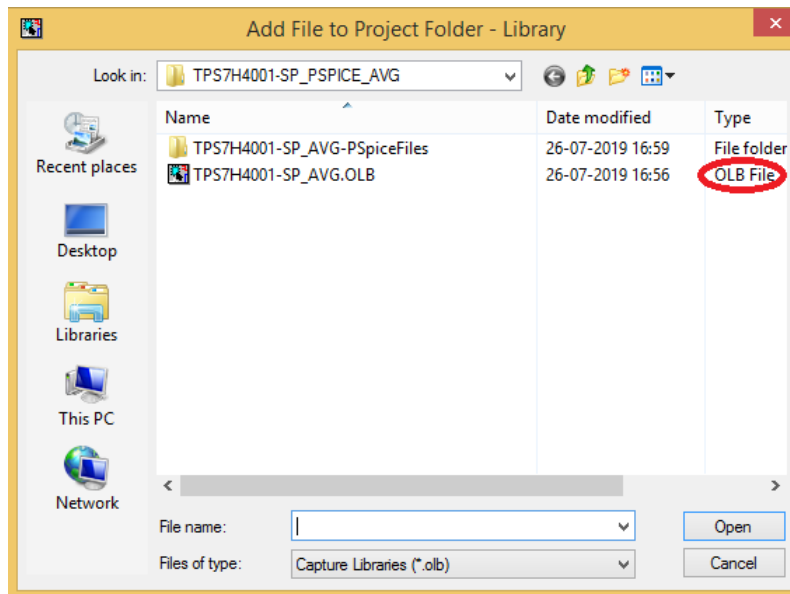
4. Once the *Create Pspice Project* dialogue box opens, select the "Create a blank project" option and click "OK".



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".

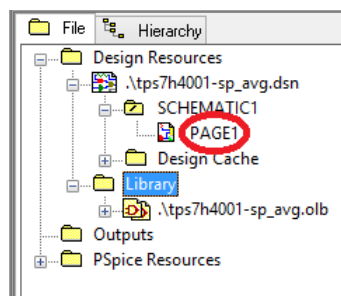


- This will add the part symbol to the project.

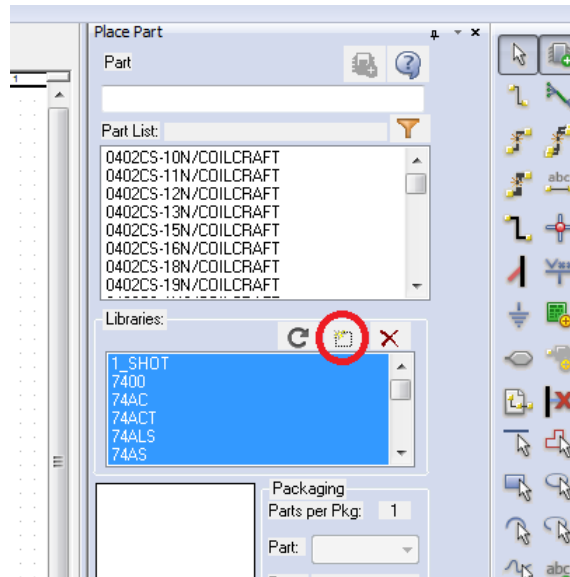
1.3 Creating the schematic in Capture

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

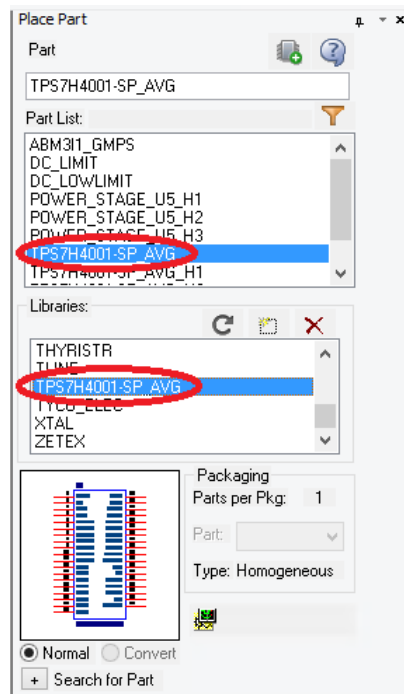
- Open **PAGE1** under the **SCHEMATIC1** folder below the **.dsn** file.



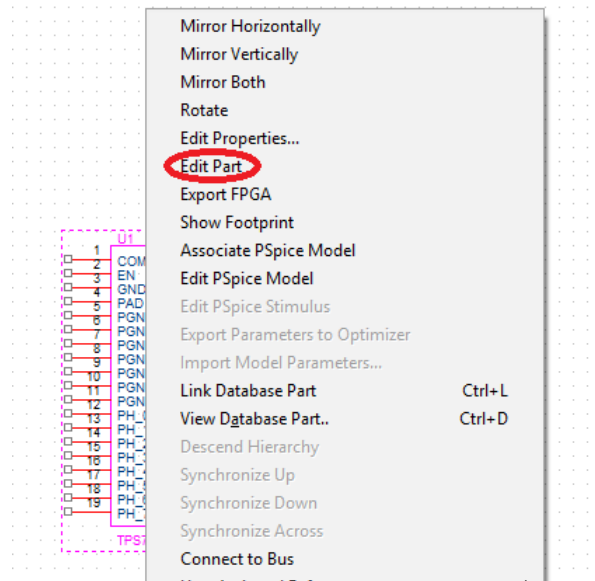
2. Select *Place* → *Part* to open the *Place Part* window. Click on the square icon shown in the following image to add libraries to the project.



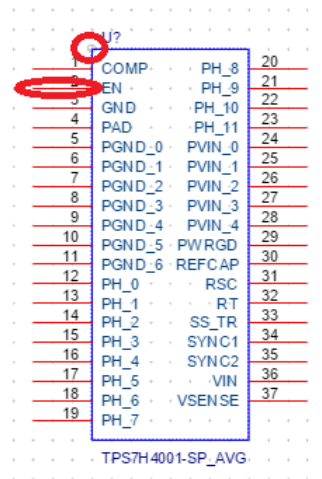
3. Select all default library files of PSPICE from installation directory and click Open. (Default Location: “C:\Cadence\SPB_17.2\tools\capture\library\pspice”) Note: This step can be omitted if the default libraries have already been added to Capture.
4. In the *Part* search box, type “TPS7H4001-SP” and select the model.



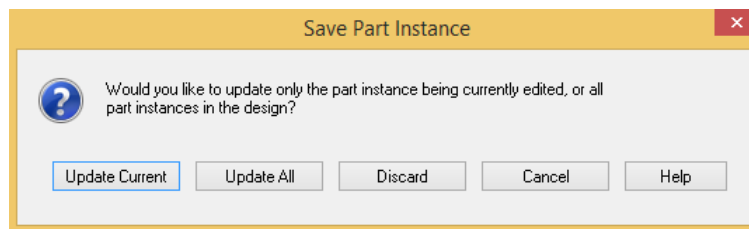
- Double click on the part in the *Part List* window and place it by left clicking the cursor on the *PAGE1* to place the part. Once the part is placed, the symbol can be edited by selecting the part, right clicking, and choosing "Edit Part".



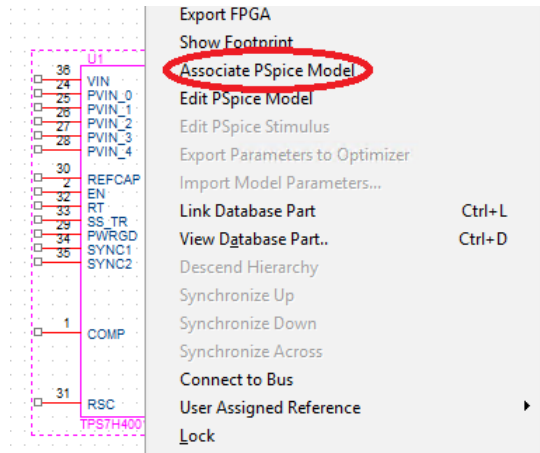
- The part can be resized and pin positions can be changed based on convenience by clicking and dragging them.



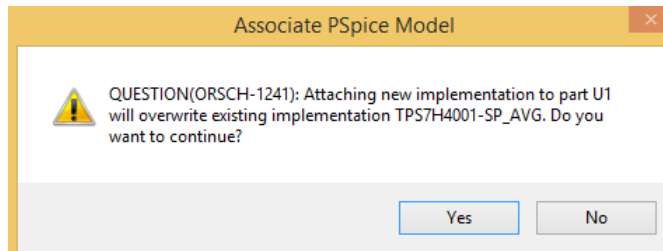
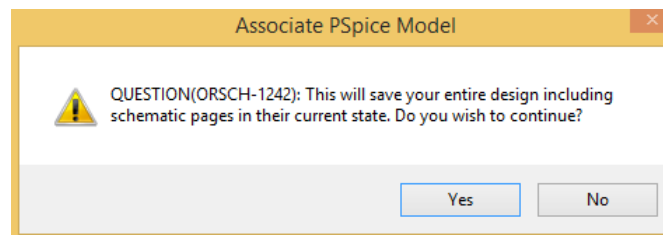
- After editing the part, close the tab to save changes. Choose "Update Current" in the pop-up.



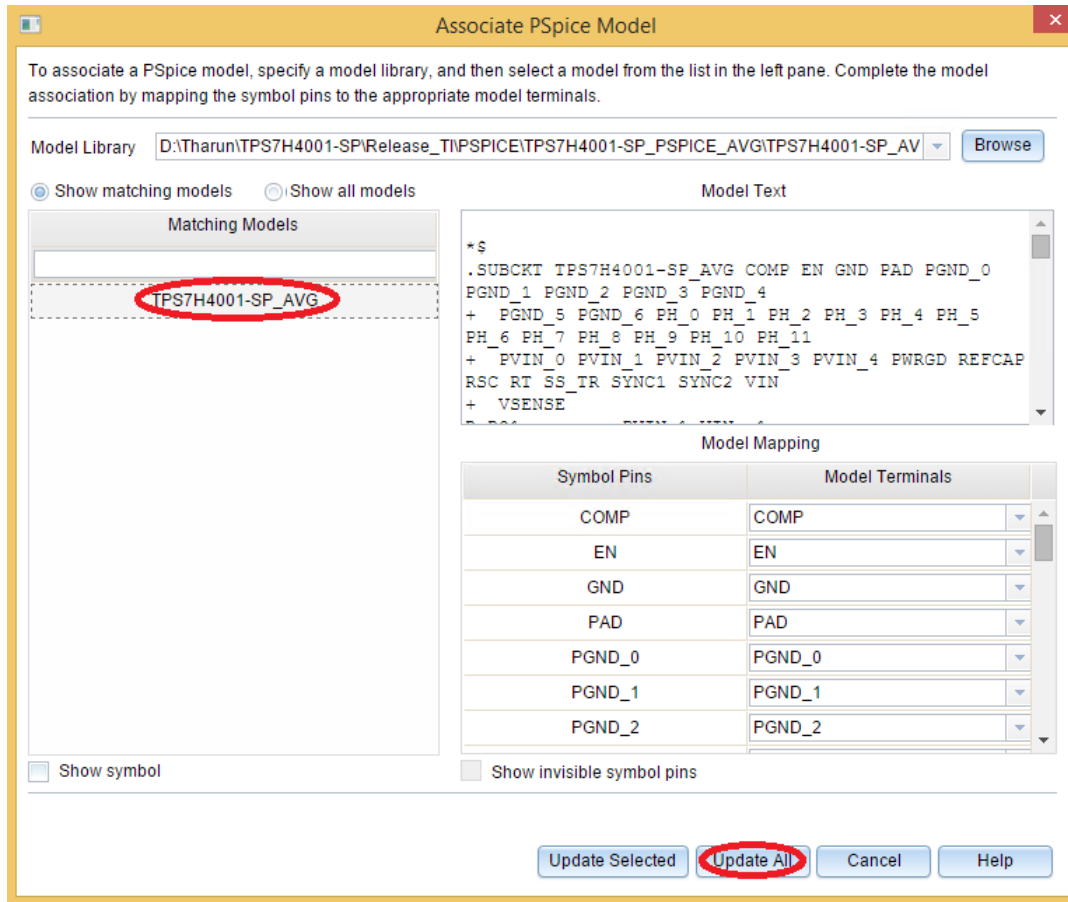
- Now, the part must be associated with the netlist. Select the part, right click, and choose "Associate Pspice Model".



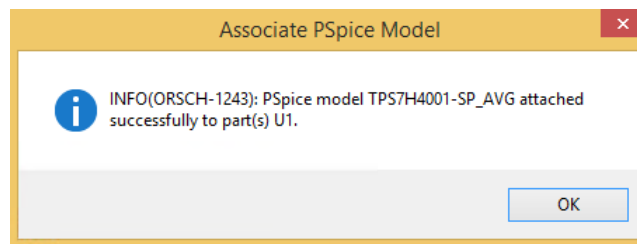
- Two pop-ups will appear to confirm the operation. Click "Yes" for both pop-ups.



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



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



- The average model has a couple controllable parameters, which can be varied to analyze the model functionality. The parameters and their default value are as follows:

- FS** - Switching frequency: 500k
- L** - Load inductance: 0.9 μ

- To enable easy access to control these parameters, make them visible by double clicking the part. The following window will become visible.

A	
	SCHEMATIC1 : PAGE1
Color	Default
Designator	FS 500k
Graphic	TPS7H4001-SP_AVG.Norma
ID	
Implementation	TPS7H4001-SP_AVG
Implementation Path	
Implementation Type	PSpice Model
L	0.9u
Location X-Coordinate	1130
Location Y-Coordinate	430
Name	INS16778231
Part Reference	U3
PCB Footprint	
Power Pins Visible	<input type="checkbox"/>
Primitive	DEFAULT
PSpiceTemplate	X*@REFDES %COMP %EN
Reference	U3
Source Library	D:\THARUN\TPS7H400
Source Package	TPS7H4001-SP_AVG
Source Part	TPS7H4001-SP_AVG.Norma
Value	TPS7H4001-SP_AVG

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

The screenshot shows the 'Display Properties' dialog box with the following settings:

- Name: (empty)
- Value: (empty)
- Font: Arial 7
- Color: Default
- Rotation: 0° (selected)
- Text Justification: Default
- Display Format:
 - Do Not Display
 - Value Only
 - Name and Value
 - Name Only
 - Both if Value Exists
 - Value if Value Exists

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

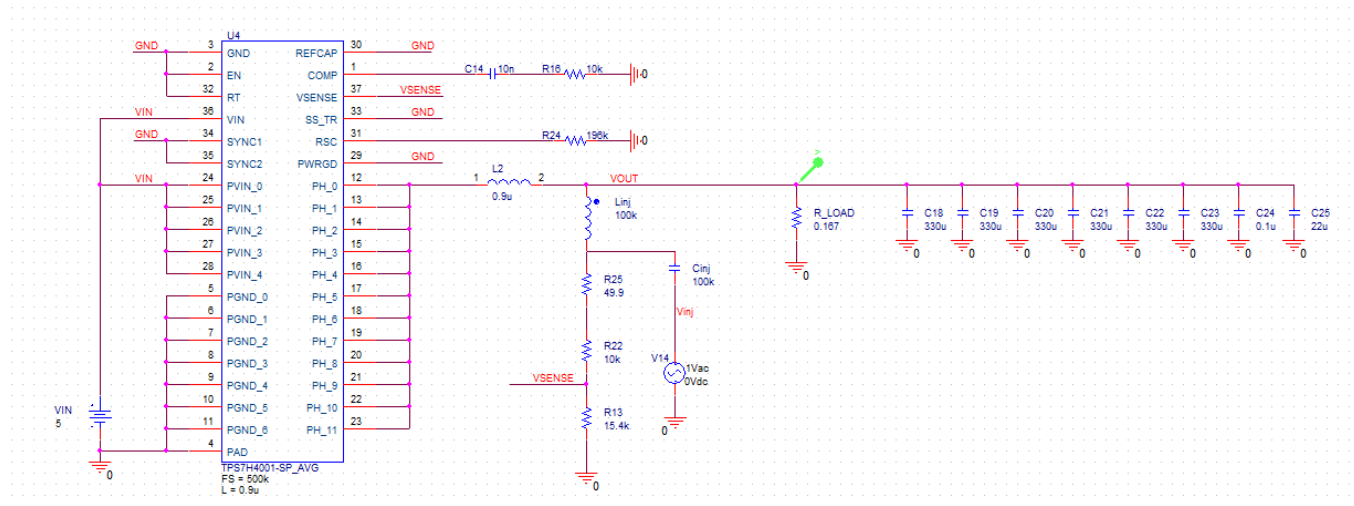


Figure 1. Model Schematic

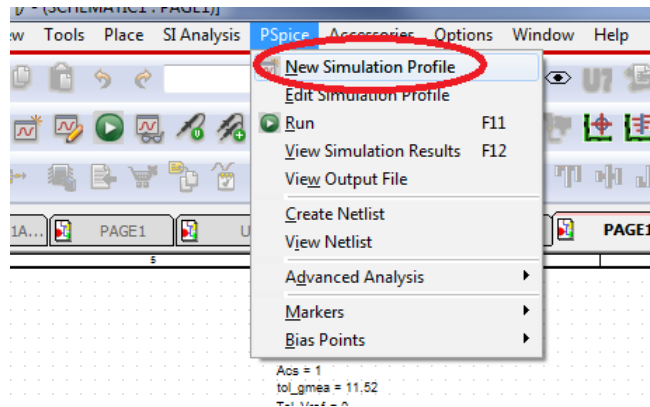
2 Simulation of Average Model for TPS7H4001-SP

The netlist file for the average model of TPS7H4001-SP is labeled as “TPS7H4001-SP_AVG.lib”. The average model is intended for following types of simulation:

- Frequency response (Phase Margin, Phase Margin Crossover, and Gain Margin)
- DC simulations for parameters like output voltage, load regulation, etc.

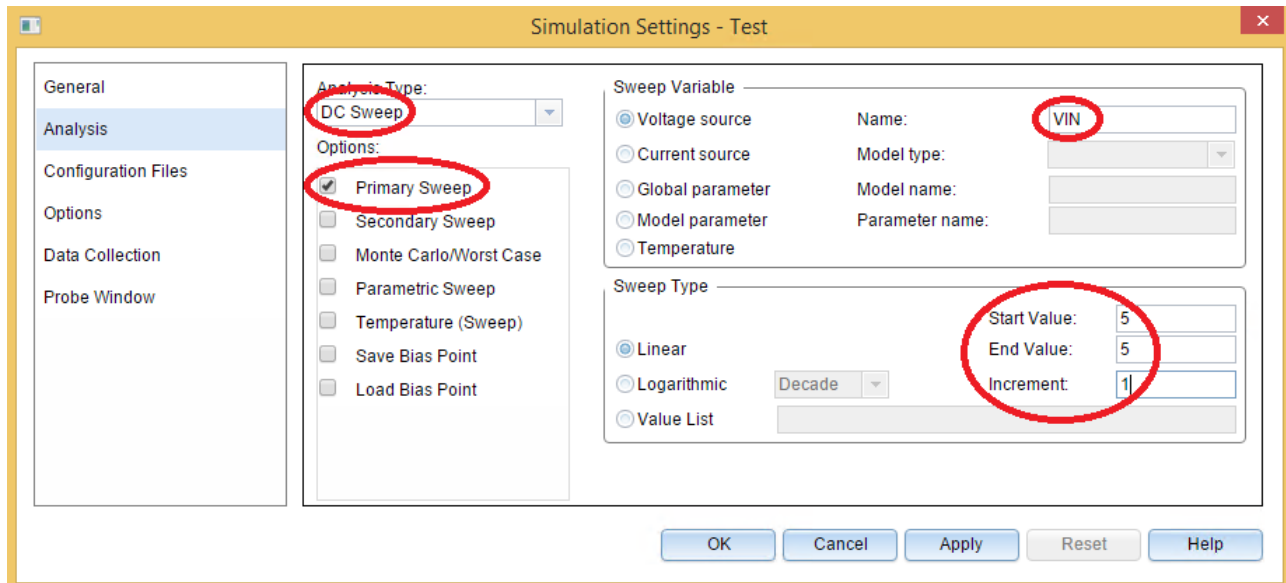
2.1 Analysis of output voltage

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



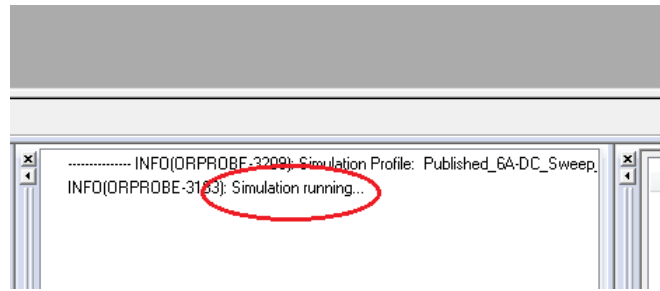
2. Set the parameters based on the image provided with the following settings:

- Analysis Type - DC-Sweep
- Primary Sweep - VIN (Input Voltage) from 5 V to 5 V



3. Run the simulation by pressing F11 or *PSpice* → *Run*.

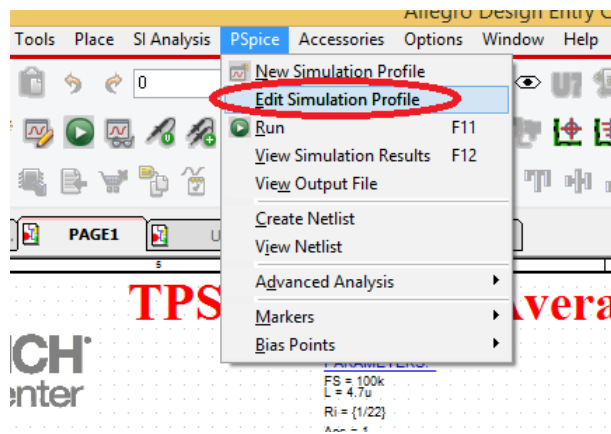
4. Wait for the simulation completion in the console window of AMS Simulator.



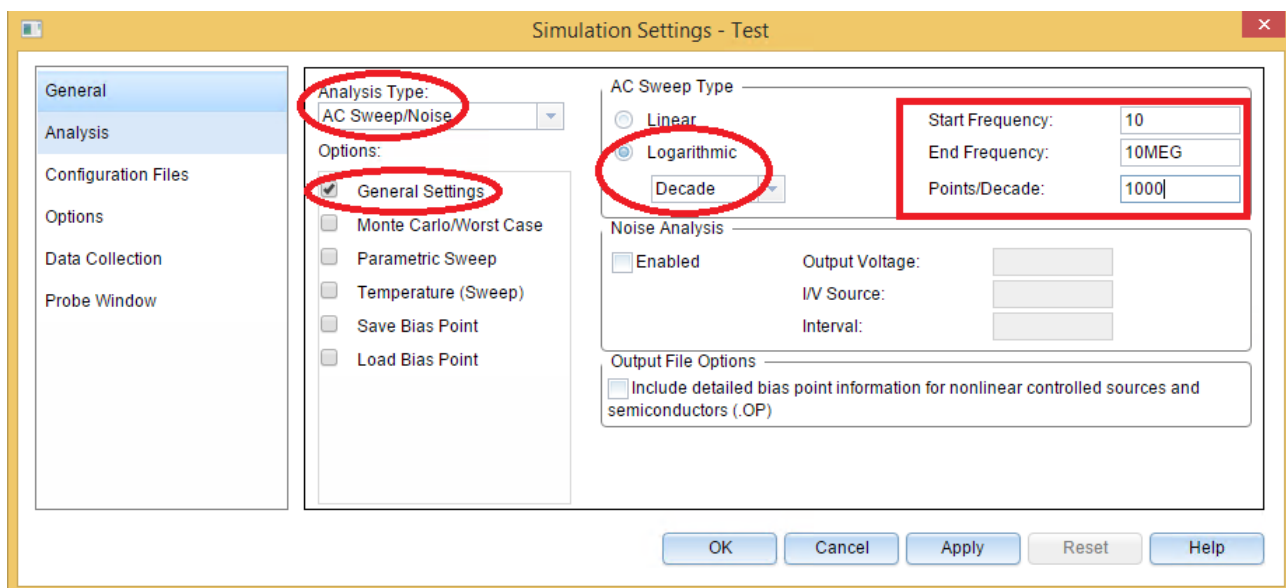
5. Once the simulation is complete, the data probed in the schematics will be displayed.

2.2 Performing frequency analysis

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

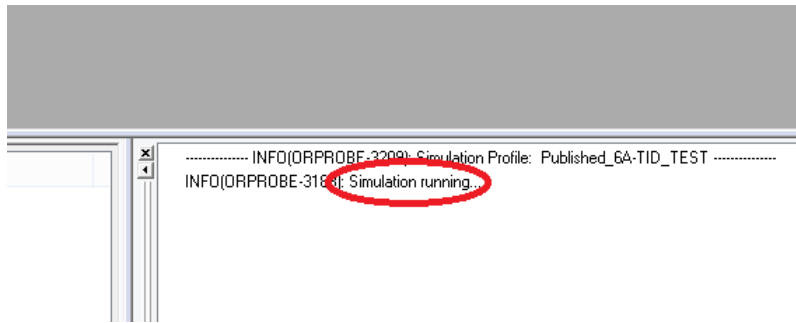


2. Set the parameters based on the image provided with the following settings:
 - Analysis Type - AC-Sweep
 - General Settings - Set start and end frequency, number of points per decade, and sweep type



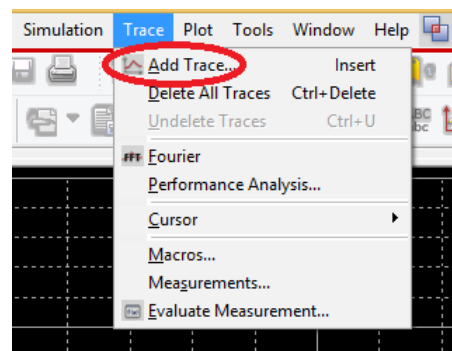
3. Run the Simulation by pressing F11 or *Pspice* → *Run*.

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

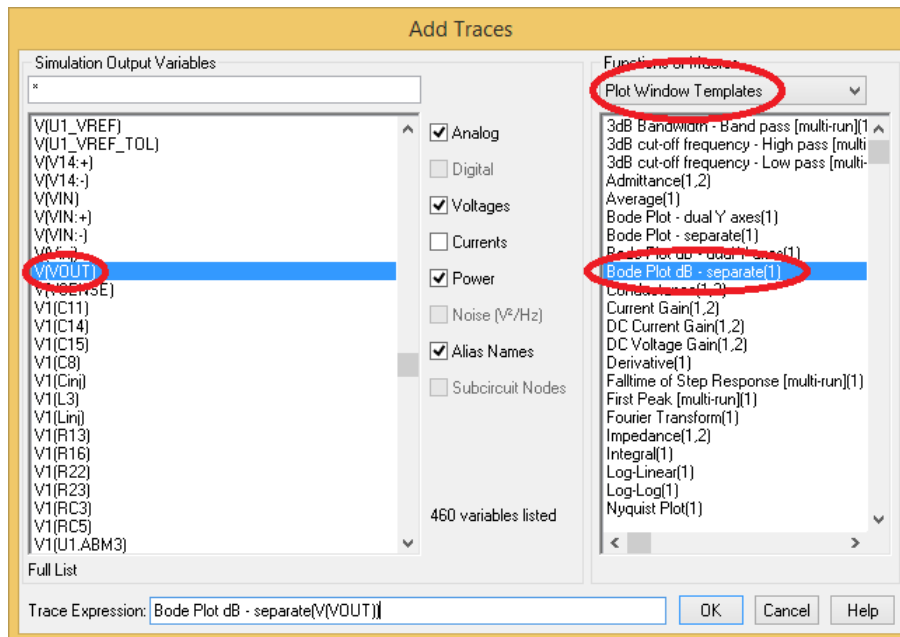


2.3 How to analyze frequency response results

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 schematics. Select the following options to get the Bode plot.



3. Use the cursor to evaluate the plot.

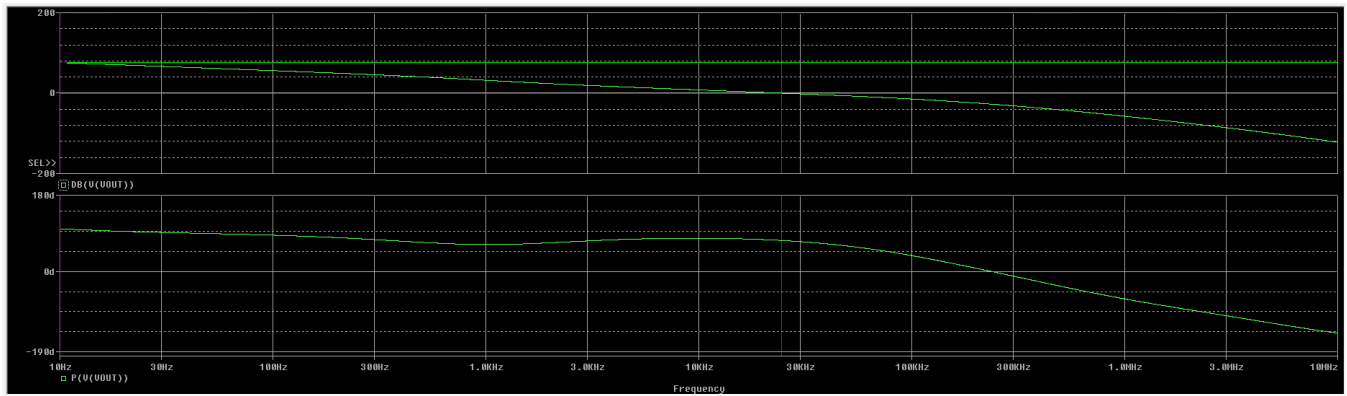


Figure 2. Frequency Response

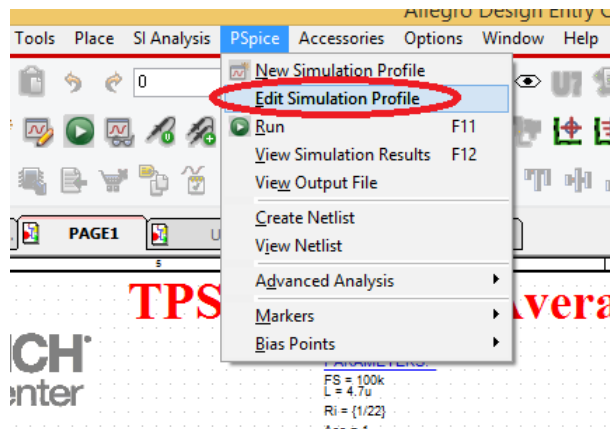
3 Simulation of Transient Model for TPS7H4001-SP

The netlist file for the transient model of TPS7H4001-SP is labeled as “TPS7H4001-SP_TRANS.lib”. The transient model incorporates the following behaviors:

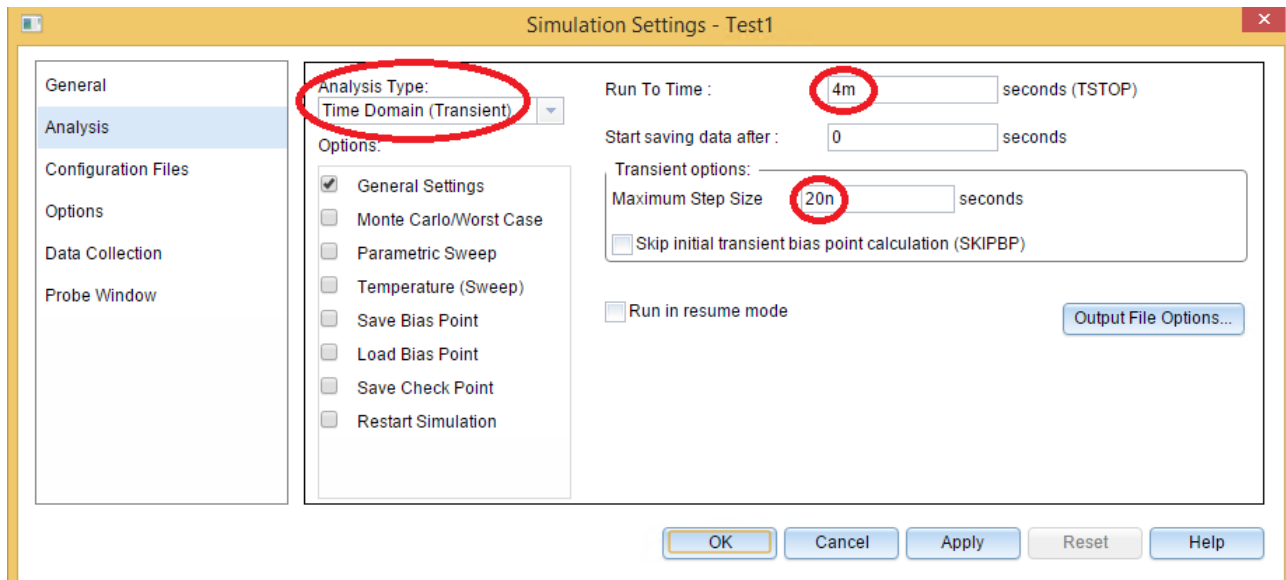
- Output voltage and current
- Soft-Start
- Slope compensation
- Switching frequency and external clock output
- UVLO and power good signal
- Overvoltage protection
- Overcurrent protection
- Master-Slave configuration
- Sequencing

To simulate the transient model:

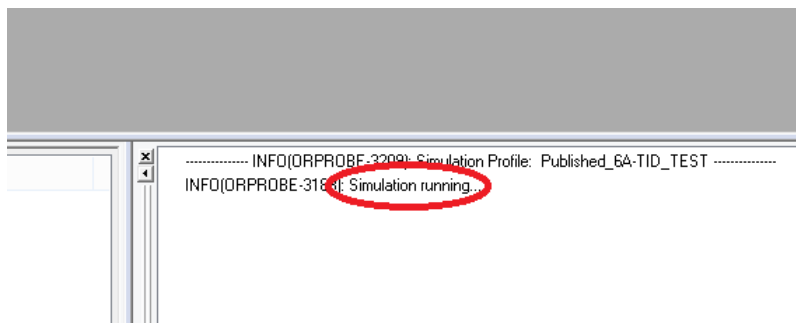
1. Create a new simulation profile, or edit existing simulation profile by clicking on *Pspice* → *Edit Simulation Profile*.



2. Set the parameters based on the image provided with the following settings:
 - Analysis Type - Time Domain (Transient)
 - General Settings - Set the required run time and restrict the maximum step size to 20 ns



3. Run the Simulation by pressing F11 or *Pspice* → *Run*.
4. Wait for the simulation completion in the console window of AMS Simulator.



5. Once the simulation is complete the data probed in the schematic will be displayed.

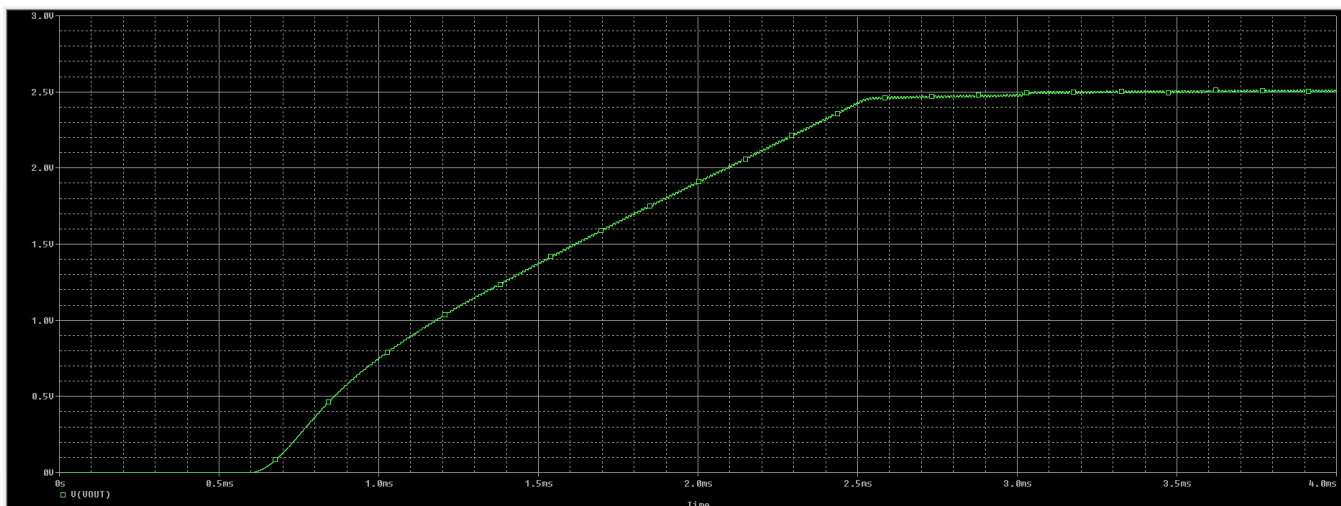


Figure 3. Transient Response

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 © 2020, Texas Instruments Incorporated