AND9002/D

Use of NCP5217A Pspice Model

Prepared by: Gang Chen
ON Semiconductor

ON Semiconductor®

http://onsemi.com

APPLICATION NOTE

Overview

The NCP5217A is a synchronous step−down controller for high performance battery−powered systems likes notebook applications. The IC is packaged in QFN14. Figure 1 shows a typical application circuit. The range of the input voltage VIN is from 4.5 V to 27 V. The range of the output voltage VOUT is from 0.8 V to 3.3 V. The internal reference voltage (FB voltage) is 0.8 V.

To provide very fast simulation results and an initial design of system parameters before a real board design in applications, a NCP5217A averaged behavior model in Pspice has been developed to simulate transient response and loop stability. Figure 2 shows a typical simulation circuit with the NCP5217A Pspice model. An inherent input voltage feed−forward function of the NCP5217A makes transient response and stability almost independent to input voltage variation, so that users do not need to provide input voltage information in simulation. Also the behavior of the external MOSFETs has been embedded into the NCP5217A Pspice model to simplify the simulation system since the MOSFETs have little effect on either transient response or stability. This simulation note is to illustrate a simulation procedure with the NCP5217A Pspice model.

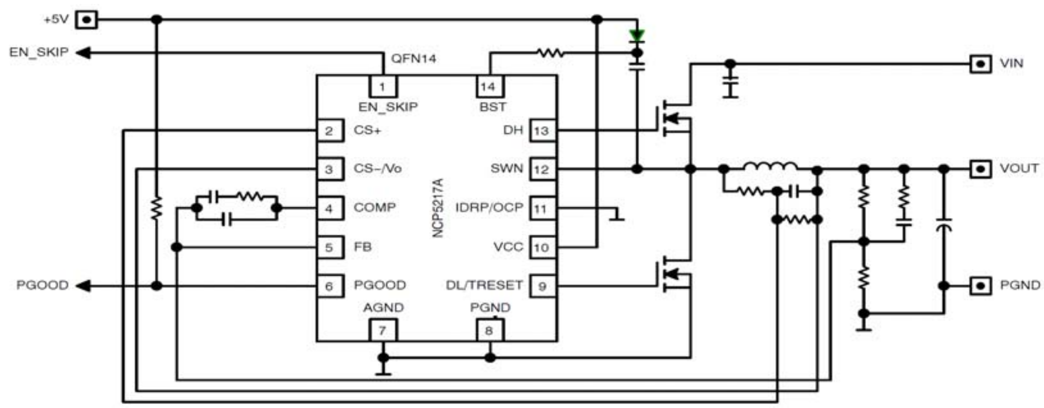

Figure 1. Typical Application Circuit

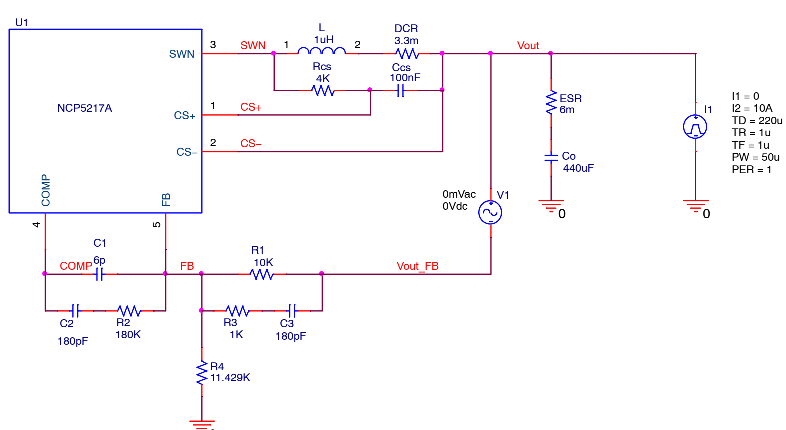

Figure 2. Typical Simulation Circuit

DETAILED DESCRIPTION

Download Pspice Model

Users can download the NCP5217A Pspice model from ON Semiconductor website, which is a zipped file “NCP5217_PSPICE.ZIP” including one Pspice model lib file “NCP5217_PSPICE.LIB”, one schematic symbol olb file “NCP5217_PSPICE.OLB”, and one design dsn file “NCP5217A.DSN”. Save all the extracted files in a folder.

Model Installation

1. Create New Project

Users need to run ORCAD Capture or Allegro Design Entry CIS first, and then create a new blank project in Capture as shown in Figure 3.

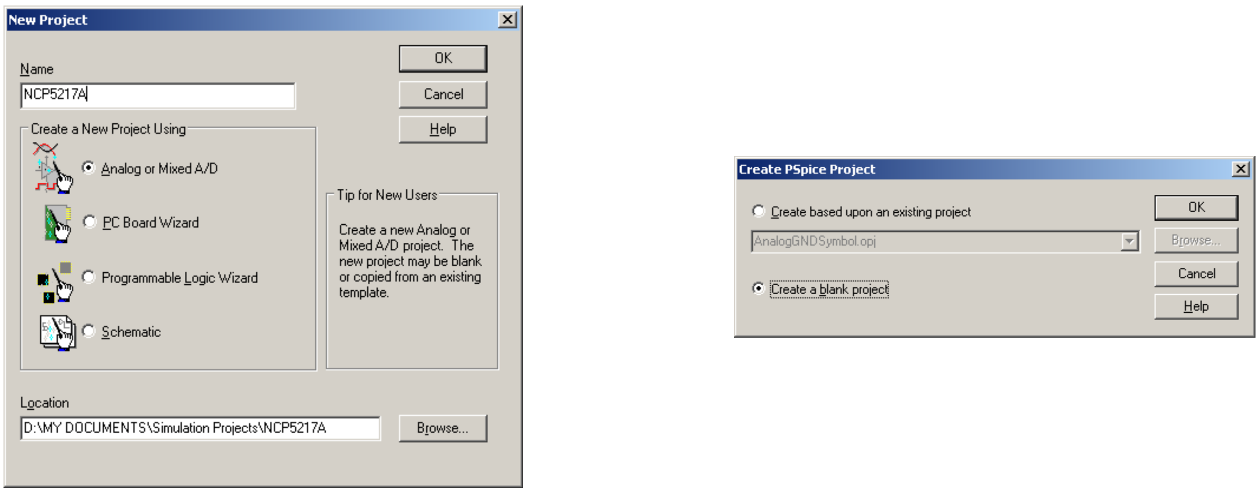

Figure 3. Create New Blank Simulation Project

2. Import Design File

Add the design file “NCP5217A.DSN” into the Design Resources to replace the blank design.

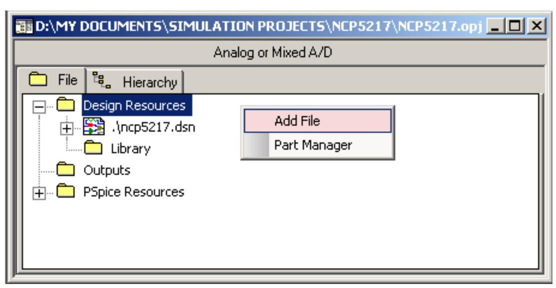

Figure 4. Import Design File into Design Resources

3. Import Symbol File

Add the symbol file “NCP5217_PSPICE.OLB” into the Design Resources as shown in Figure 5.

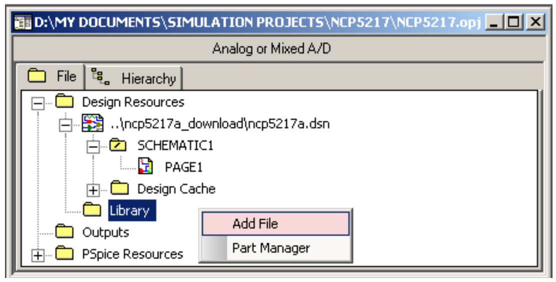

Figure 5. Import Symbol File into Design Resources

4. Open Schematic

Open the schematic “Page 1” in the “Schematic1” under the design file as shown in Figure 6. Users can edit the schematic according to real applications.

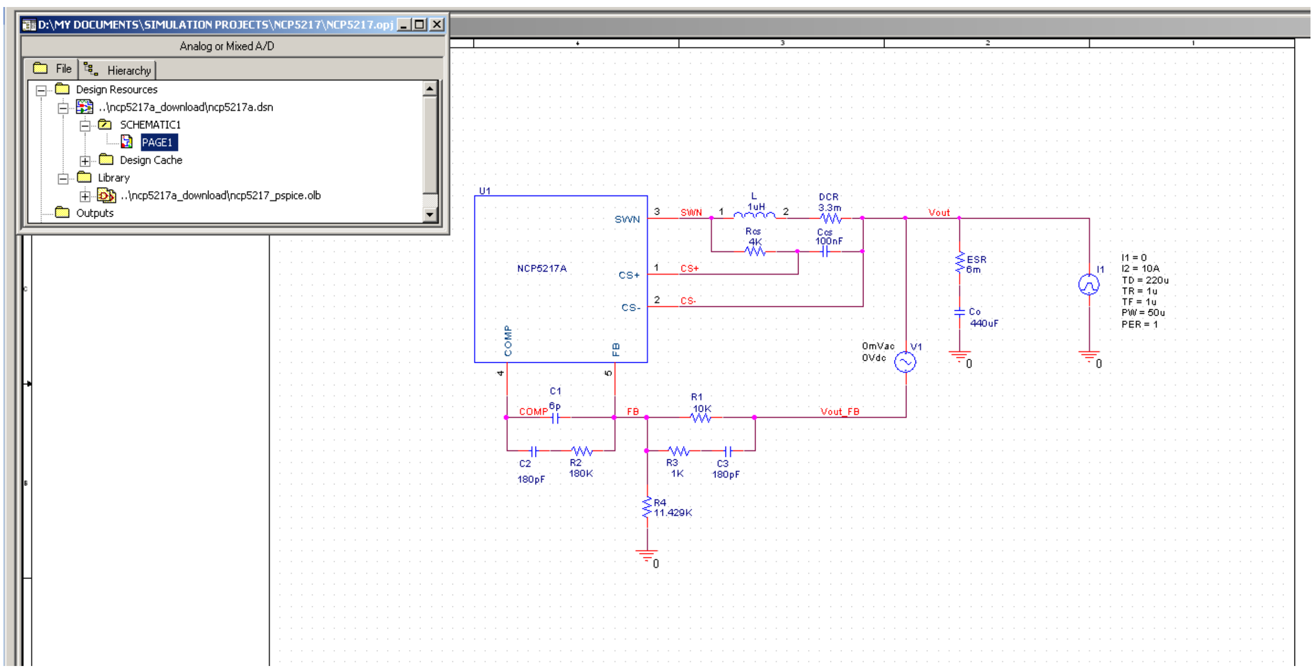

Figure 6. Open Schematic in Design File

5. Create Simulation Profile and Run Simulation

In order to run simulations, a new simulation profile has to be created. In the simulation setting of the simulation profile, users need to use browser to add the Pspice lib file “NCP5217_PSPICE.LIB” into the design library of the simulation configuration files, as shown in Figure 8. The Pspice model of the NCP5217A is able to support both time domain transient simulation and AC frequency domain simulation. Users can set both configurations in the simulation profile.

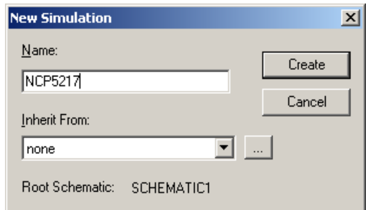

Figure 7. Create a New Simulation Profile

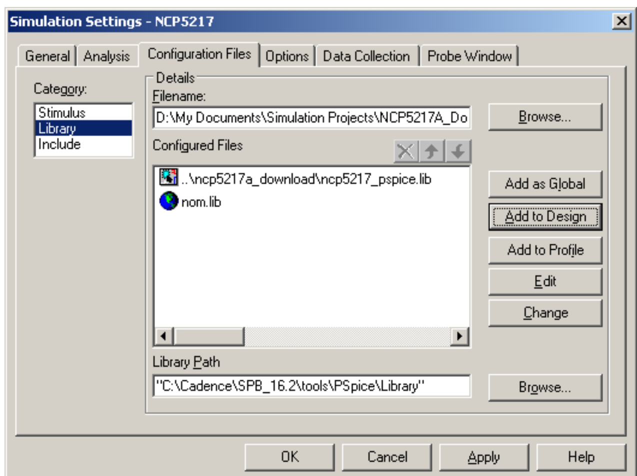

Figure 8. Add Pspice Lib File into the library of the Configuration Files

Time Domain Transient Simulation

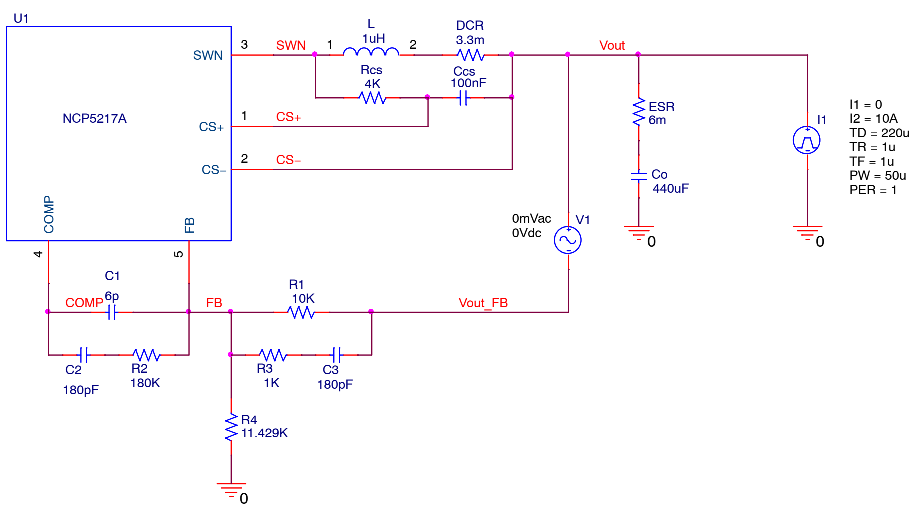

Figure 9. Typical Schematic for Time Domain Transient Simulation

Figure 9 shows a typical schematic for a time domain transient simulation. An AC source V1 is set to 0 V as its AC and DC components. Users can edit parameters of the pulse current source I1 to simulate load transient in the output V_{OUT}. In order to reduce simulation time, a 100 μs (instead of 1.1 ms in the NCP5217A datasheet) internal soft start has been implemented in the model. A typical time-domain simulation profile setting is shown in Figure 10. Users can review simulation waveforms in Pspice A/D after running a simulation. Figure 11 shows an example of the simulation results regarding to a load transient event.

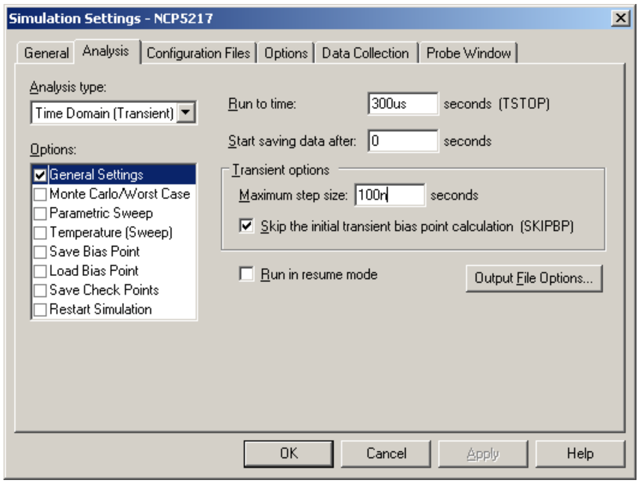

Figure 10. Simulation Setting for Time Domain Simulation

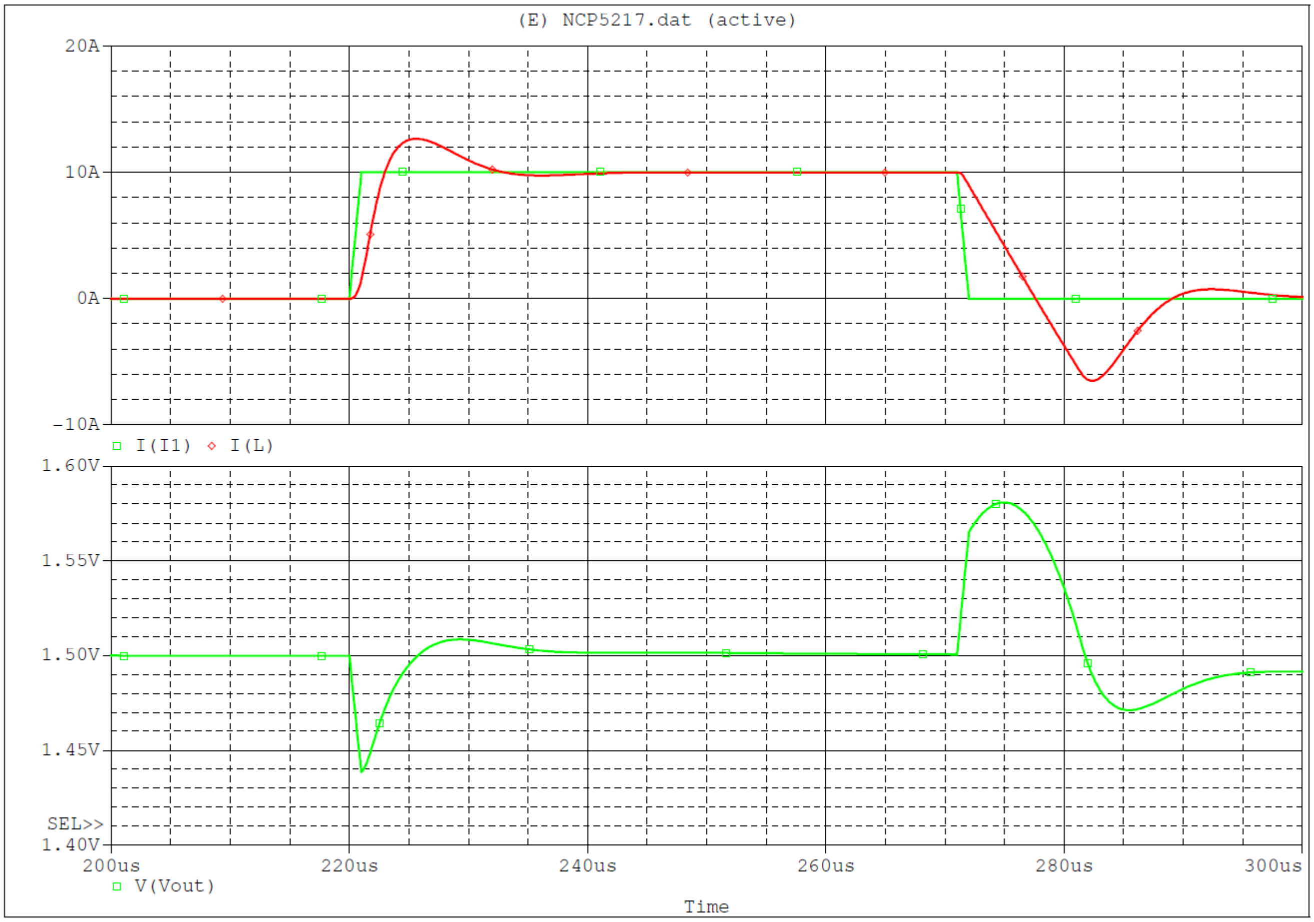

Figure 11. Typical Simulation Results of Time Domain Simulation

Due to a benefit from the averaged behavior model, the total simulation time is just a few seconds and thus it is good for users to optimize the system by running a parameter sweep simulation. Before running a parameter sweep simulation, at least one “PARAM” part from the “SPECIAL.OLB” needs to be added in the schematic. Figure 12 shows an example schematic that is able to be used to run parameter sweep for the capacitor C3 in the compensation network. Users can program a pattern of the parameter sweep in the simulation profile as shown in Figure 13.

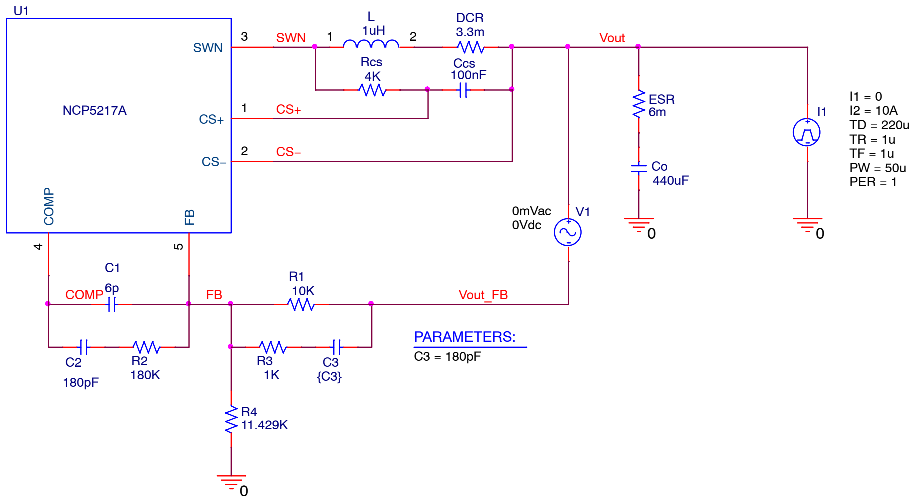

Figure 12. Typical Schematic for Parametric Sweep in Time Domain Transient Simulation

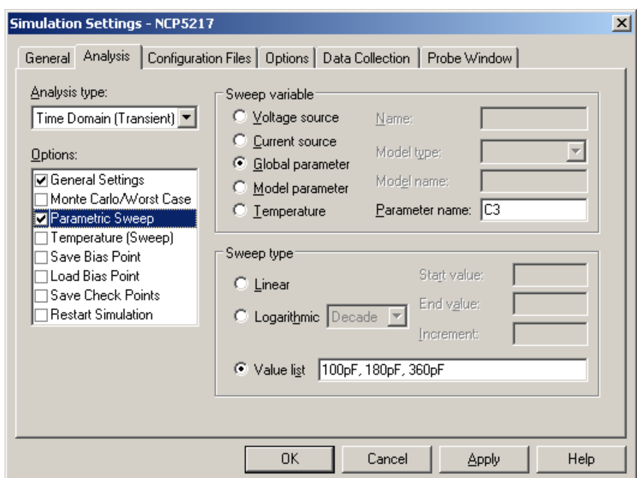

Figure 13. Simulation Setting for Parametric Sweep in Time Domain Simulation

Figure 14 shows multiple simulation results after the parameter sweep simulation. It is very easy for users to see the parameter impact on the transient response.

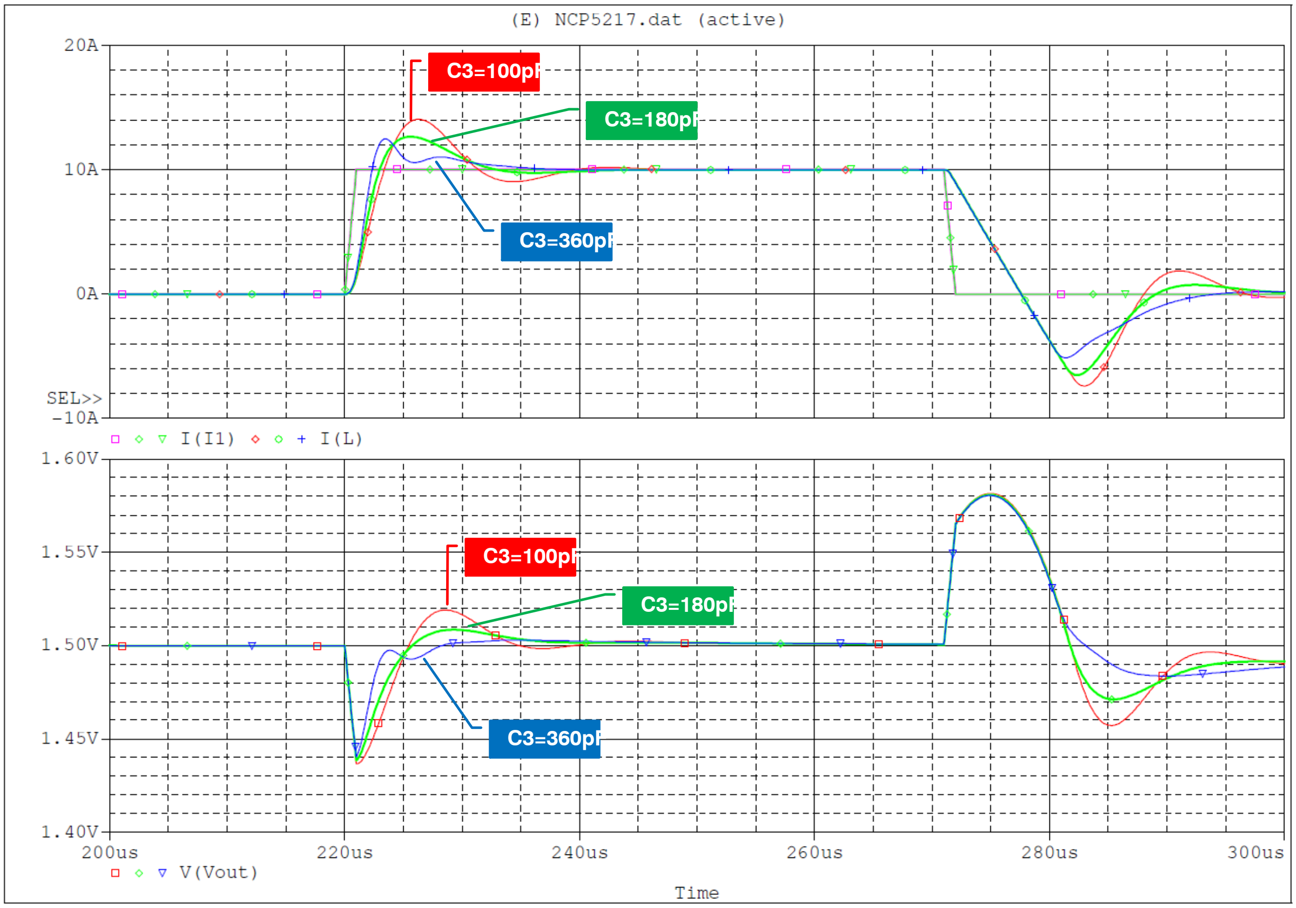

Figure 14. Typical Simulation Results of Parametric Sweep in Time Domain Simulation

AC Frequency Domain Simulation

With the NCP5217A Pspice model, users are able to use almost the same schematic to do AC frequency domain simulation as what is used in the time domain simulation. The main difference is in the setting of the AC voltage source V1 shown in Figure 15. In the frequency domain simulation, usually a small AC voltage such as 10 mV ~ 100 mV is used.

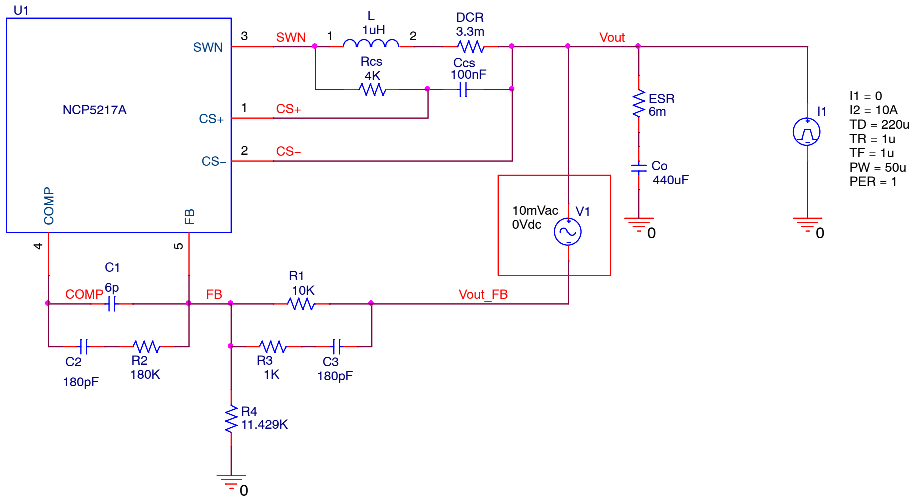

Figure 15. Typical Schematic for AC Frequency Domain Simulation

Figure 16 shows a simulation setting for an AC frequency domain simulation and Figure 17 shows typical simulation results. Users can read close-loop bandwidth and phase margin from the bode plot.

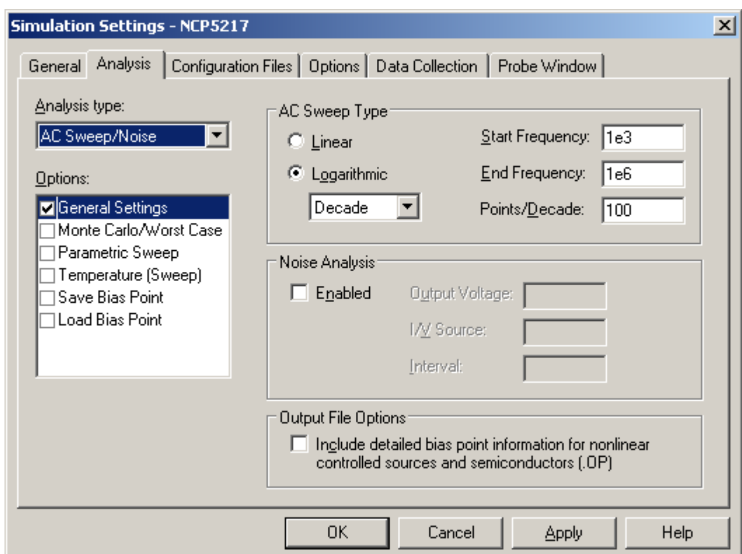

Figure 16. Simulation Setting for AC Frequency Domain Simulation

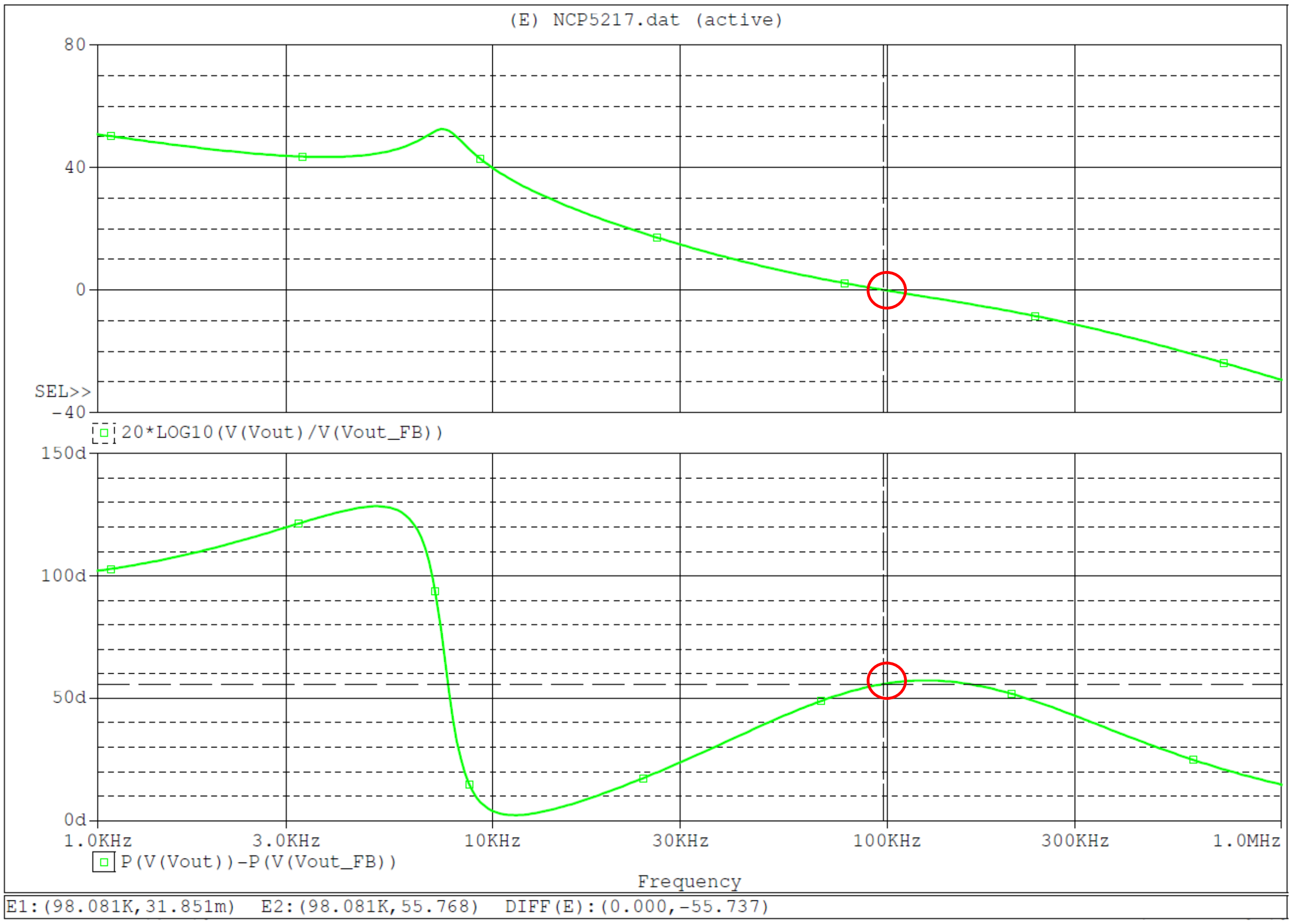

Figure 17. Typical Simulation Results of AC Frequency Domain Simulation

Users also can run parametric sweep in an AC frequency domain simulation. Figure 18 is an example schematic which is very similar to the schematic shown in Figure 12 but with a 10mV AC component in the AC voltage source V1. Figure 19 shows a corresponding simulation setting, and Figure 20 shows typical simulation results. Users can see the parameter impact on the close-loop stability.

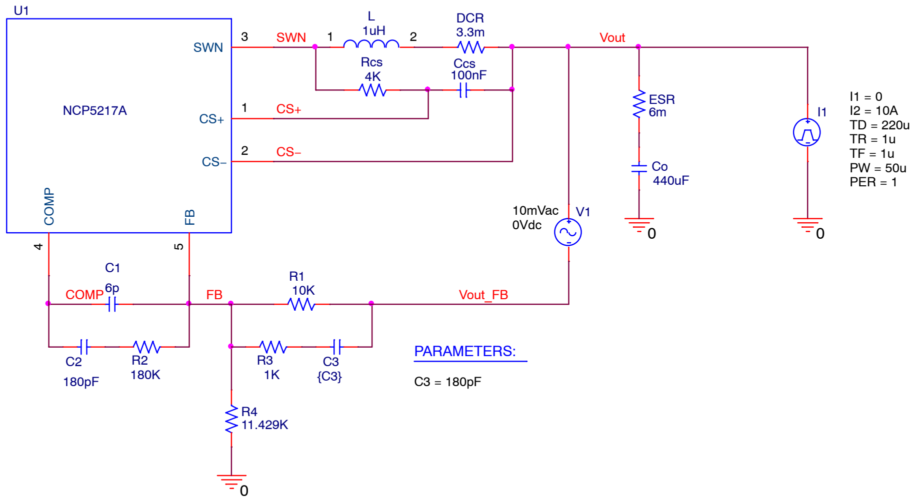

Figure 18. Typical Schematic for Parametric Sweep in AC Frequency Domain Simulation

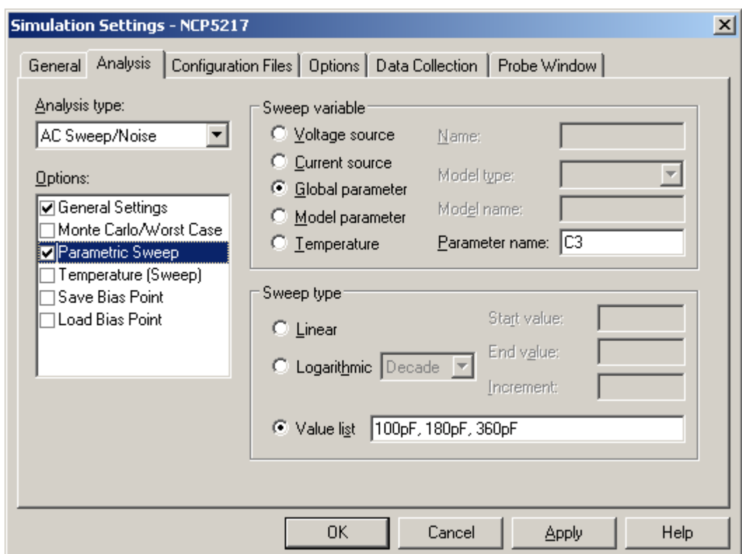

Figure 19. Simulation Setting for Parametric Sweep in AC Frequency Domain Simulation

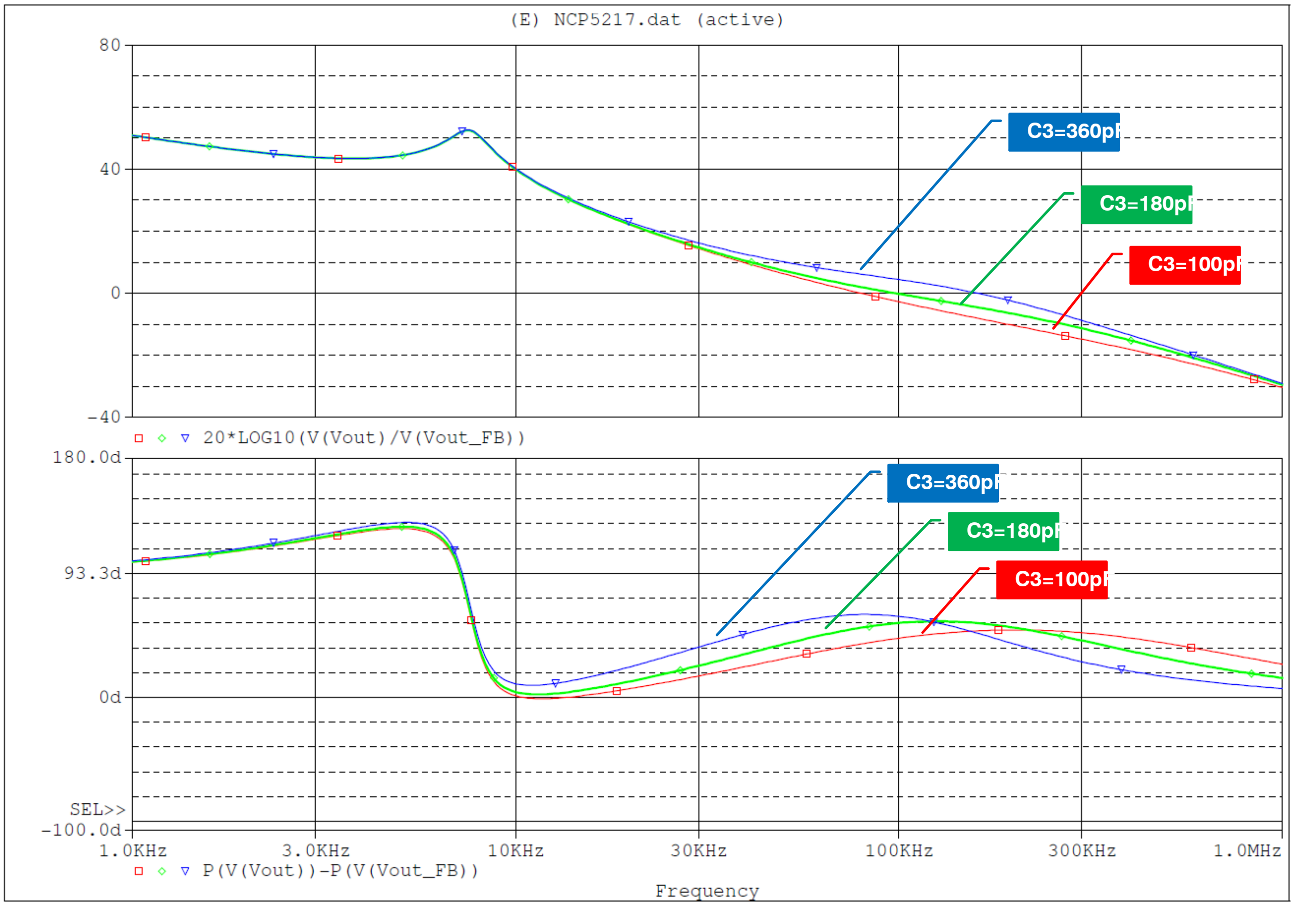

Figure 20. Typical Simulation Results of Parametric Sweep in AC Frequency Domain Simulation

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
P.O. Box 5163, Denver, Colorado 80217 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5773−3850

ON Semiconductor Website: www.onsemi.com

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative