AND9004/D

Use of NCP1529 Pspice Model

Prepared by: Gang Chen
ON Semiconductor

ON Semiconductor®

http://onsemi.com

APPLICATION NOTE

Overview

The NCP1529 is a synchronous step−down DC−DC converter for portable applications powered by one cell Li−ion or three cell Alkaline/NiCd/NiMH batteries. The device is able to deliver up to 1.0 A on an output voltage range externally adjustable from 0.9 V to 3.9 V. The device also has a built−in 1.7 MHz (nominal) oscillator. Automatic switching PWM/PFM mode offers improved system efficiency.

To provide simulation results and an initial design of system parameters before a real board design in applications, a Pspice model of the NCP1529 has been developed. There are two typical applications using the NCP1529. Figure 1 shows a typical simulation circuit for a DC−DC application. Figure 2 shows another typical simulation circuit for a LED driver application. This simulation note is to illustrate how to use the NCP1529 Pspice model.

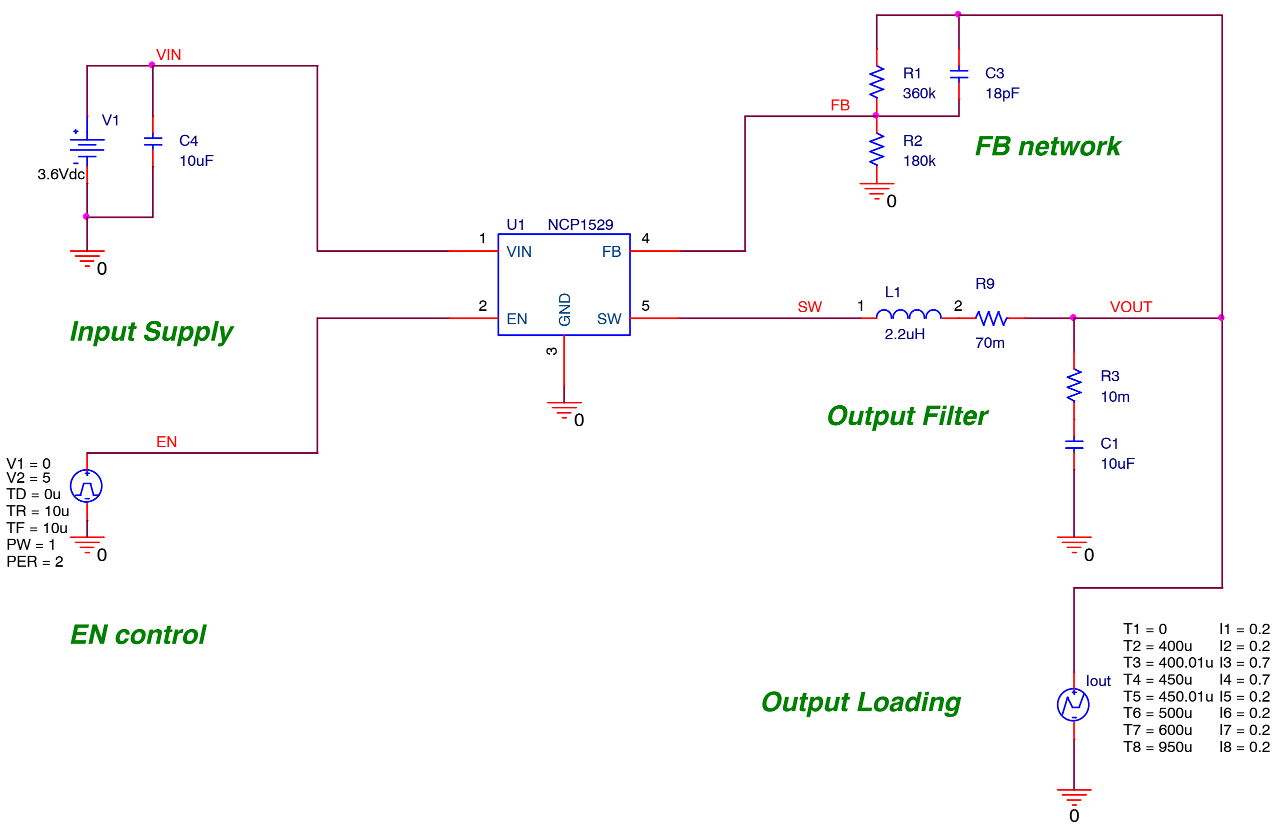

Figure 1. Typical Simulation Circuit of NCP1529 for DC−DC Applications

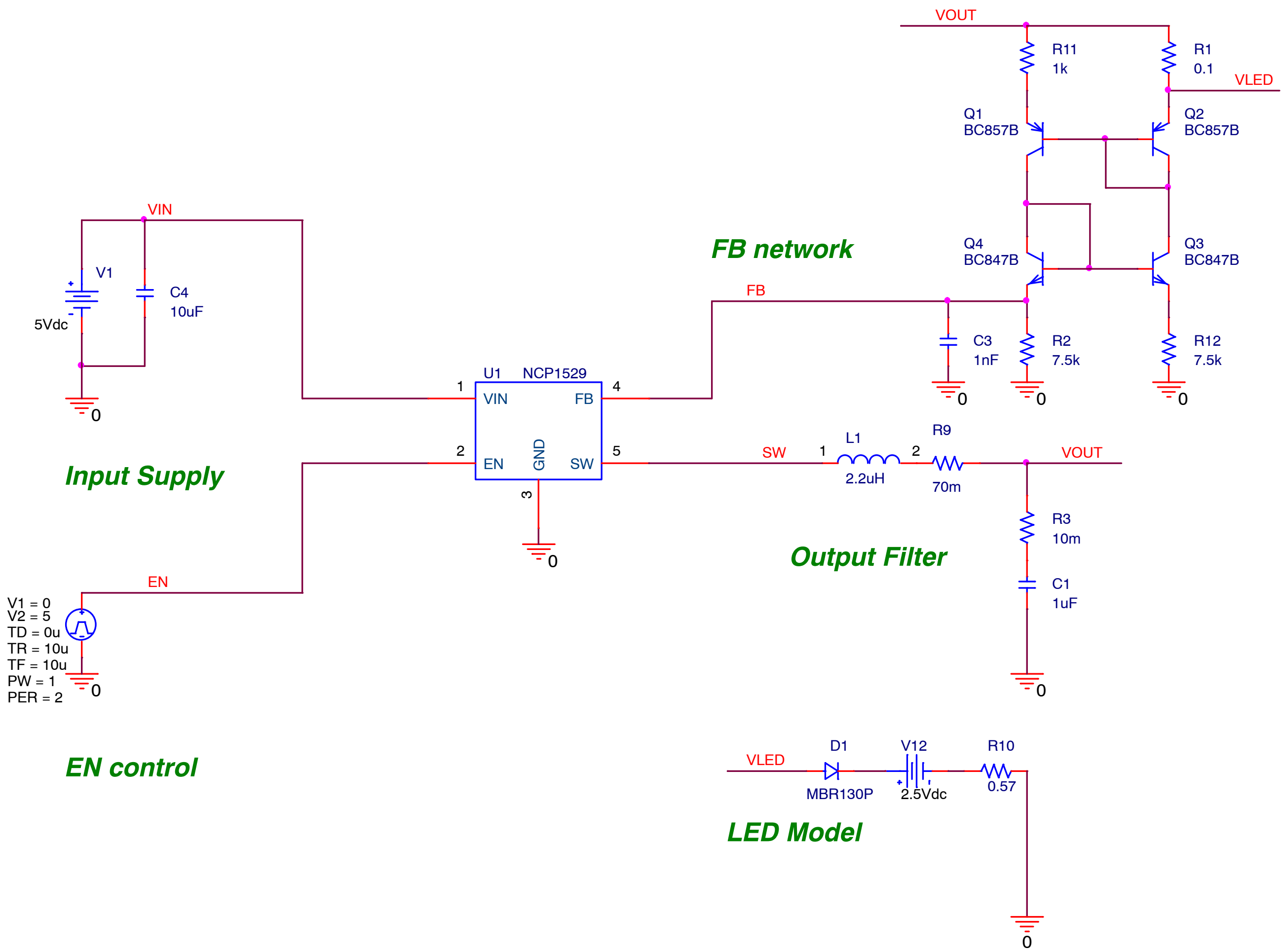

Figure 2. Typical Simulation Circuit of NCP1529 for LED Driver Applications

DETAILED DESCRIPTION

Download Pspice Model

Users can download the NCP1529 Pspice model from ON Semiconductor website, which is a zipped file (NCP1529_PSPICE.ZIP) including one Pspice model lib file (NCP1529_PSPICE.LIB), one schematic symbol olb file (NCP1529_PSPICE.OLB), and two design dsn files (NCP1529_DCDC.DSN and NCP1529_LED.DSN). Save all the extracted files in a folder.

Model Installation and Simulation

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.

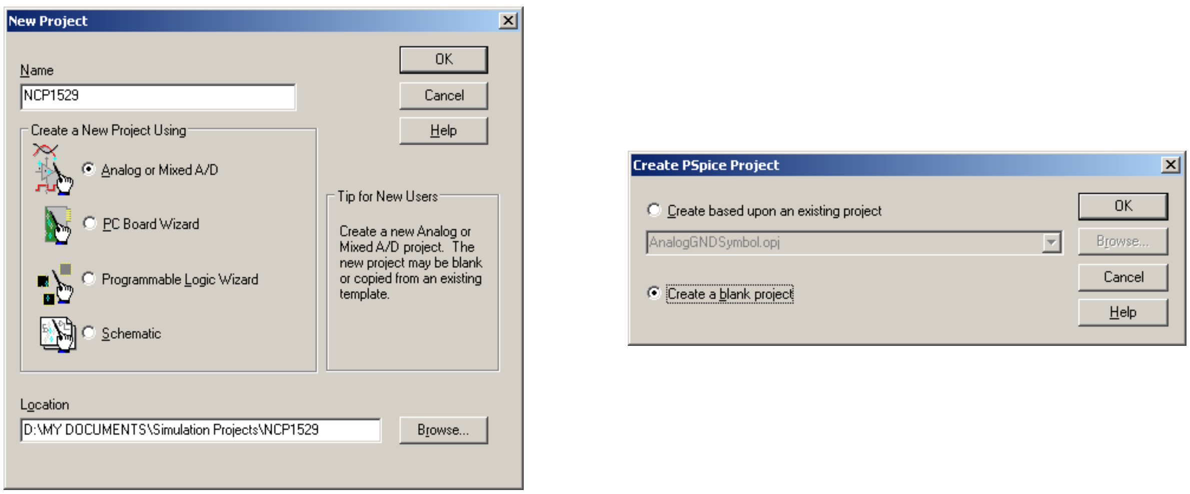

Figure 3. Create New Blank Simulation Project

2. Import Design File

Depending on the application to be simulated, users need to select either the design file "NCP1529_DCDC.DSN" or "NCP1529_LED.DSN" and add it into the Design Resources to replace the blank design.

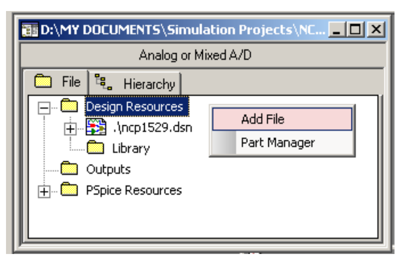

Figure 4. Import Design File into Design Resources

3. Import Symbol File

Add the symbol file "NCP1529_PSPICE.OLB" into the Design Resources as shown in Figure 5.

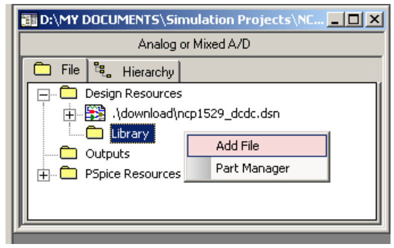

Figure 5. Import Symbol File into Design Resources

4. Open Schematic

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

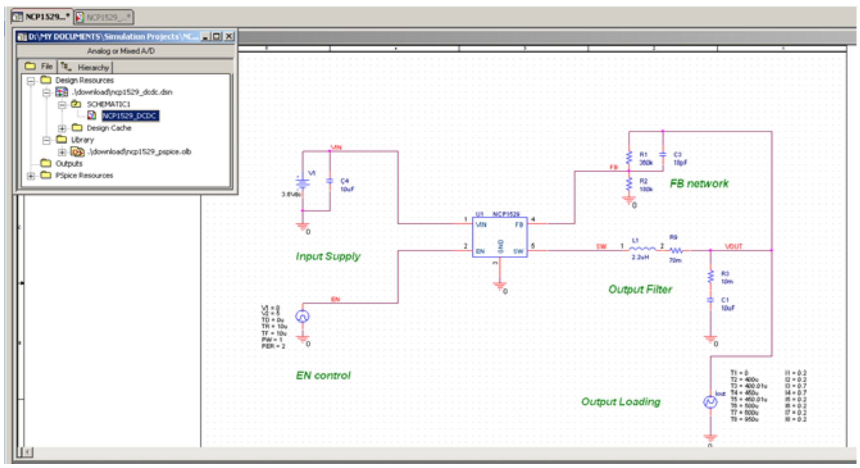

Figure 6. Open Schematic in Design File

5. Create Simulation Profile and Run Simulation

In order to run simulation, a simulation profile has to be created as shown in Figure 7. In the simulation setting of the simulation profile, users need to use browser to add the Pspice lib file “NCP1529_PSPICE.LIB” into the design library of the simulation configuration files, as shown in Figure 8.

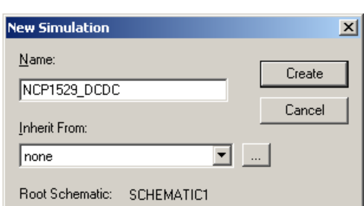

Figure 7. Create a New Simulation Profile

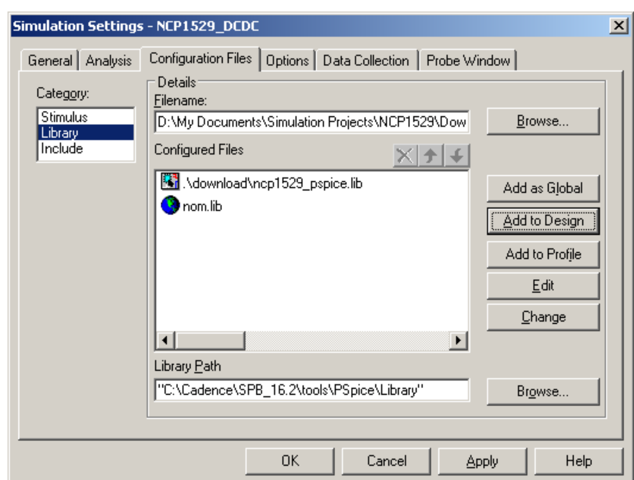

Figure 8. Add Pspice LIB File into Library of Configuration Files

To reduce simulation time, a 100 μs (instead of 310 μs in the NCP1529 datasheet) internal soft start has been implemented in the model. A typical time−domain simulation profile setting is shown in Figure 9. Users can review simulation waveforms in Pspice A/D after running a simulation.

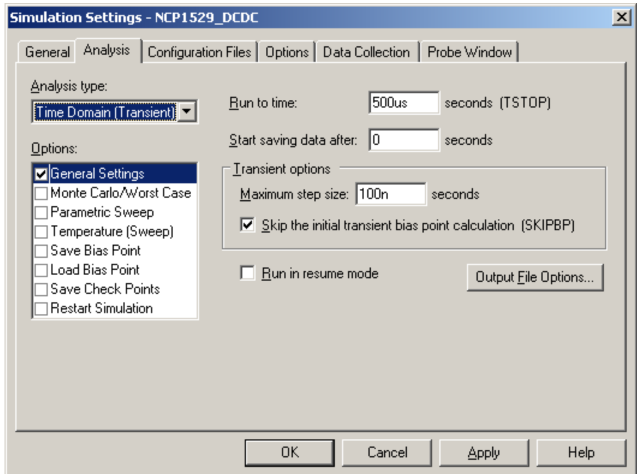

Figure 9. Simulation Setting for a Time Domain Simulation.

5.1 Simulation with DC−DC Application Circuit

Figure 1 shows the schematic included in the design file “NCP1529_DCDC.DSN”, which is a typical simulation circuit for a DC−DC application. To get detail application information, please refer to datasheet “NCP5219−D”. An IPWL current source “I_{out}” is employed to simulate a load current variation in the output of the DC−DC converter. Figure 10 shows an example of the simulation results regarding to a load transient event.

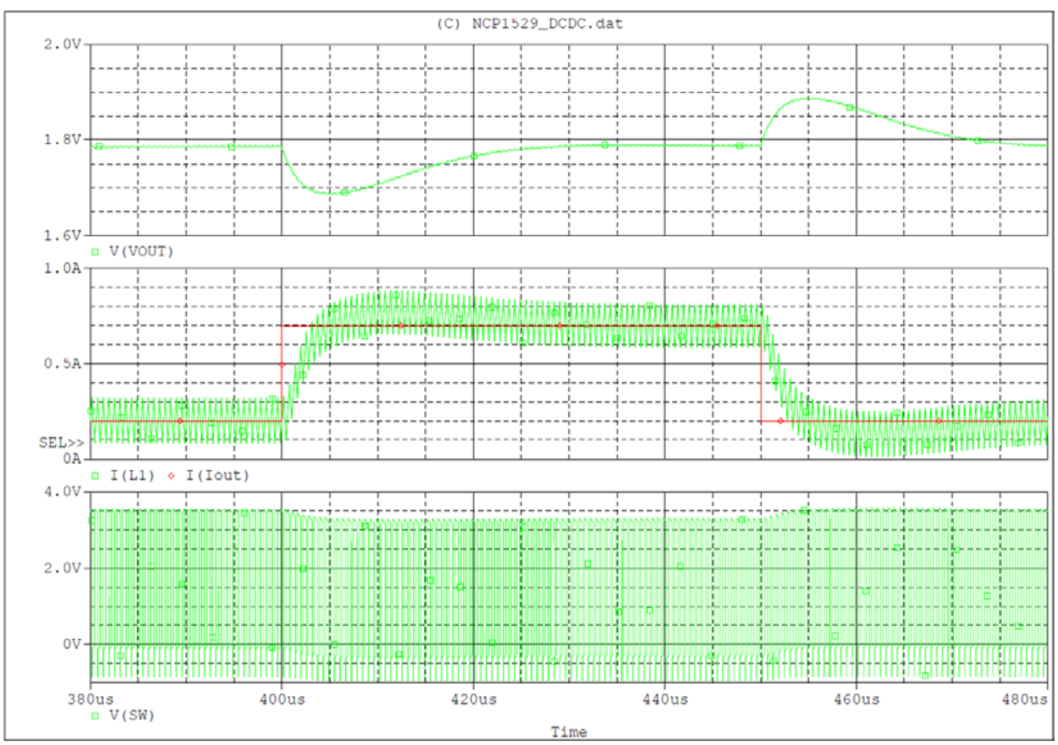

Figure 10. Typical Simulation Results of Time Domain Simulation in DC−DC Application

5.2 Simulation with LED Application Circuit

Figure 2 shows the schematic included in the design file “NCP1529_LED.DSN”, which is a typical simulation circuit for a LED driver application. The NCP1529 operates with an external current mirror to regulate LED current. To get detail application information, please refer to application note “AND8465/D”. In this simulation circuit, a LED diode is modeled by a circuitry incorporating a diode “D1”, a DC voltage source “V12”, and a resistor “R10”. Figure 11 shows an example of the simulation results regarding to the regulation of the LED current I(D1).

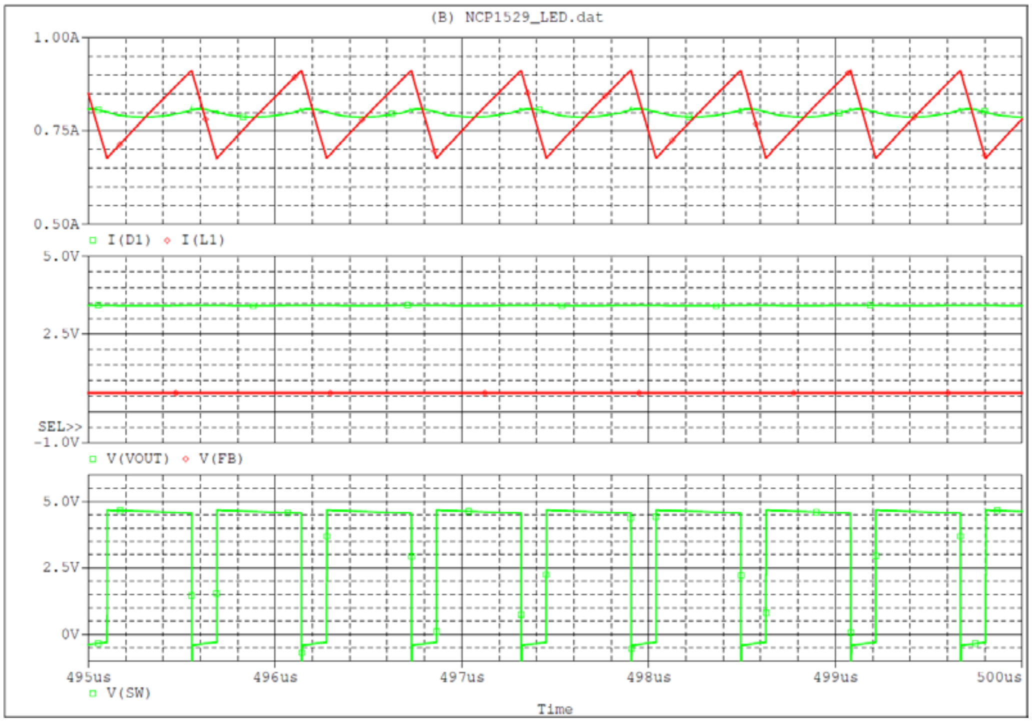

Figure 11. Typical Simulation Results of Time Domain Simulation in LED Application

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