

**Non-isolated Synchronous Buck
DC-DC Power Supply
Basic Simulation Circuit
Reference Guide**

RD047-RGUIDE-01

TOSHIBA ELECTRONIC DEVICES & STORAGE CORPORATION

Table of Contents

| | |
|---|-----------|
| 1. Introduction | 3 |
| 2. Overview of the non-isolated synchronous rectification buck DC-DC power supply..... | 5 |
| 2.1. Power supply specifications..... | 5 |
| 2.2. Circuit configuration..... | 5 |
| 3. Simulation results..... | 9 |
| 4. Product Overview | 13 |
| 4.1. TPH6R003NL | 13 |
| 4.2. TPH2R903PL | 13 |
| 5. Using the simulation circuit..... | 14 |

1. Introduction

Most information and communication systems, home appliances, and other electric devices operate on DC voltage. Since they cannot operate on mains AC voltage, it is necessary to convert AC voltage to DC voltage. An AC-DC power supply is used for this purpose. It might be necessary to further convert the DC voltage from an AC-DC power supply to different DC voltage levels, depending on the requirement of each load in a system.

The device that converts one DC voltage level to another is called a DC-DC power supply. Fig. 1.1 shows an example of a system power line configuration. There are several power lines that supply different DC voltages to different loads. One load is directly supplied from the AC-DC power supply whereas other loads are supplied through one or two stages of DC-DC power supplies following the AC-DC power supply.

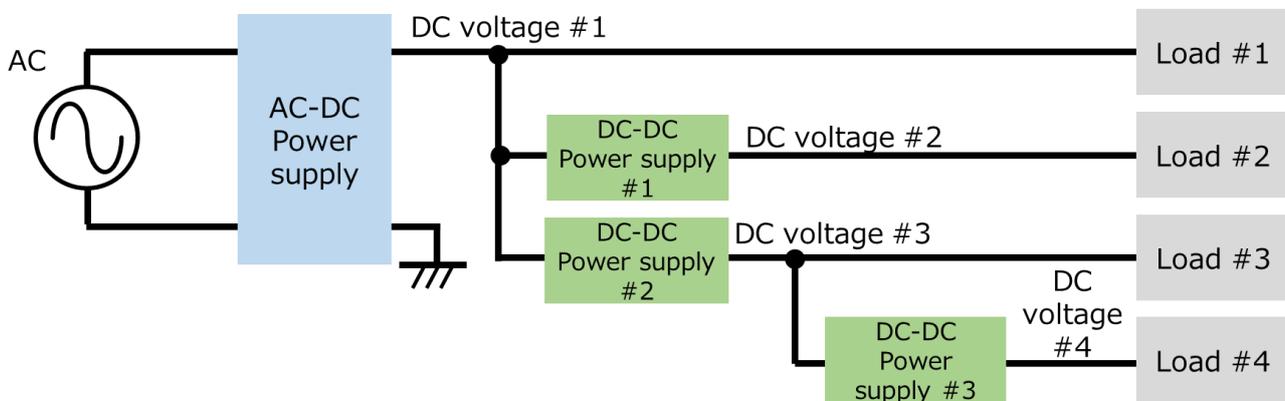


Fig. 1.1 Example of a system power line configuration

DC-DC power supplies are broadly divided into two categories: series regulators and switched-based regulators. This Reference Guide discusses switched-based regulators that are the mainstream of DC-DC power supplies. Switched-based regulators achieve voltage regulation by varying the on-off time ratio of switching devices. Switched-based regulators require more complicated control circuitry than series regulators, but generally exhibit less power loss.

There are two types of switched-based regulators: isolated DC-DC power supplies that incorporate a transformer to provide electrical isolation between the input and output sides and non-isolated DC-DC power supplies that do not provide electrical isolation between the input and output sides. This Reference Guide discusses non-isolated DC-DC power supplies.

A non-isolated DC-DC power supply is composed of an inductor, switching devices, and rectifier diodes. The switching devices operate at several hundreds of kilohertz to convert an input DC voltage to a regulated output DC voltage.

Three types of DC-DC power supplies are available: (a) buck (step-down) power supplies that step down voltage from input to output, (b) boost (step-up) power supplies that step up voltage from input to output, and (c) buck-boost power supplies with an output voltage that is either greater than or less than the input voltage. Fig. 1.2 shows the basic configurations of buck, boost, and buck-boost DC-DC power supplies. Because diodes cause a significant power loss during conduction, some DC-DC power supplies use MOSFETs instead of diodes. This technique is called synchronous

rectification. Also, a step-down non-isolated DC-DC power supply is sometimes called a buck converter.

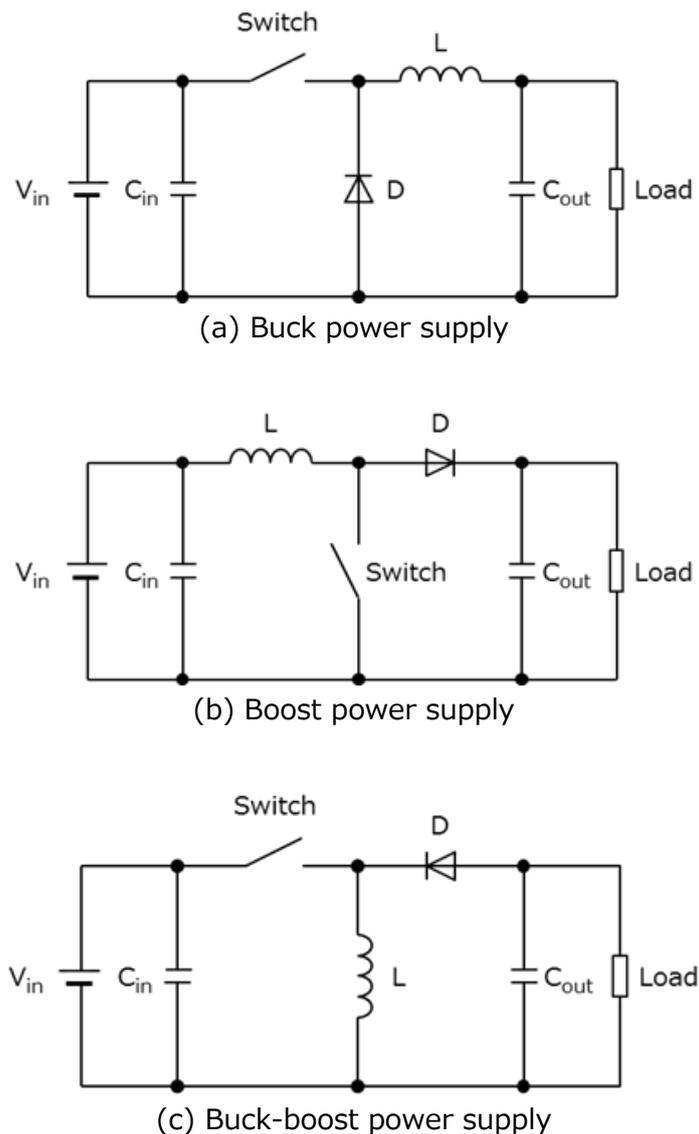


Fig. 1.2 Examples of block diagrams of non-isolated DC-DC power supplies

This Reference Guide discusses a buck synchronous rectification DC-DC power supply that generates an output voltage lower than an input voltage and uses MOSFETs for rectification instead of diodes. Synchronous rectification requires more complicated control than diode rectification. However, despite the added MOSFET driver circuit, synchronous rectification helps improve efficiency under high-load conditions. Therefore, synchronous rectification is commonly used for power supplies with large output current. A basic simulation circuit of a non-isolated synchronous rectification buck DC-DC power supply (RD047-SPICE-01) is available for download on Toshiba's website, which will help you understand its operation.

The Reference Guide provides an overview of this simulation circuit and shows how to simulate. OrCAD® Capture and PSpice® A/D from Cadence are necessary to simulate this circuit. Both the simulation circuit and the Reference Guide are based on OrCAD® 17.2.

2. Overview of the non-isolated synchronous rectification buck DC-DC power supply

The basic simulation circuit (RD047-SPICE-01) is a 40 W non-isolated synchronous rectification DC-DC converter.

2.1. Power supply specifications

The specifications of the DC-DC converter are as follows:

- Input voltage: 12 V
- Output voltage: 5 V
- Output power: 40 W
- Operating frequency: 300 kHz
- Inductor current ripple ratio: 40 %
- Output inductor: 3.3 μ H

2.2. Circuit configuration

Fig. 2.1 shows the simulation circuit for OrCAD[®]. It is a non-isolated synchronous rectification DC-DC power supply, which mainly consists of a DC-DC converter and a PWM controller. The low side of the power supply section is a MOSFET synchronous rectifier. The PWM controller is a general-purpose controller with a MOSFET gate driver, which was prepared to create this DC-DC power supply. As an example, the TPH6R003NL is used as the high-side MOSFET whereas the TPH2R903P is used as the low-side MOSFET.

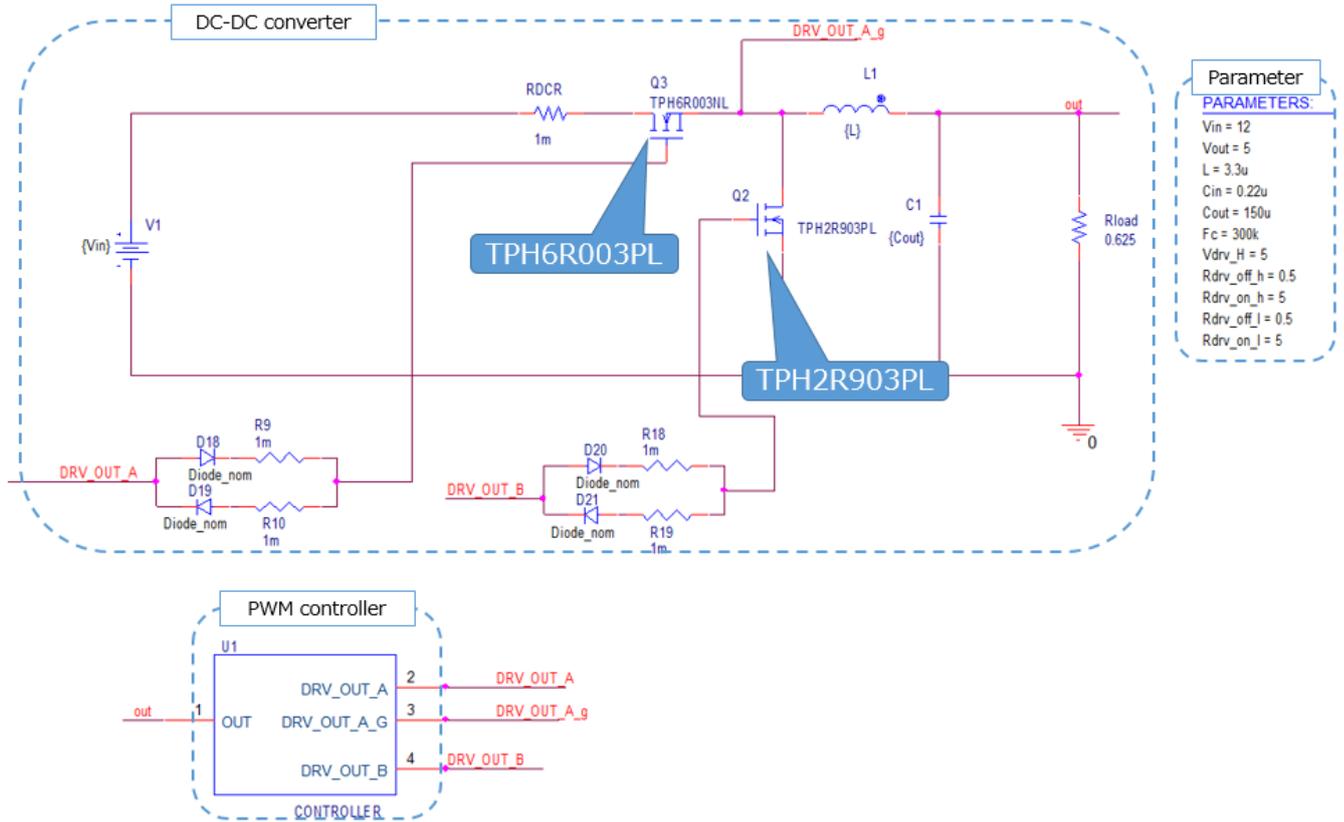


Fig. 2.1 Simulation circuit for a 40 W non-isolated synchronous rectification buck DC-DC power supply

Selection of the high-side MOSFET

The high-side MOSFET (TPH6R003NL: $V_{DSS}=30$ V, $I_D=38$ A) was selected, taking the following into consideration:

1. Withstand voltage

Since the input voltage of the DC-DC power supply is 12 V, a MOSFET with a withstand voltage of 30 V or higher was selected for the simulation circuit, considering voltage surge that occurs during switching.

2. Current rating

When the input current is maximum, it is at the maximum output power. Suppose that the DC-DC power supply has a conversion efficiency of 85 % at the maximum output power of 40 W. Then, the maximum input power is calculated to be 47.1 A. The average current applied to the high-side MOSFET is roughly 3.9 A when the output power is at the maximum. Therefore, a MOSFET with a current rating of 8 A or higher and a peak current rating of 16 A or higher was selected for the high-side MOSFET.

Selection of the low-side MOSFET

The low-side MOSFET (TPH2R903PL: $V_{DSS}=30$ V, $I_D=70$ A) was selected, taking the following into consideration:

1. Withstand voltage

Since the input voltage of the DC-DC power supply is 12 V, a MOSFET with a withstand voltage of 30 V or higher was selected for the simulation circuit, considering voltage surge that occurs during switching.

2. Current rating

When the input current is maximum, it is at the maximum output power. The maximum output power of 40 W. Since the on-time of the low-side MOSFET is roughly 7/12 of the switching cycle, the average current applied to the low-side MOSFET is roughly 4.7 A when the output power is at the maximum. Therefore, a MOSFET with a current rating of 9 A or higher and a peak current rating of 18 A or higher was selected for the low-side MOSFET.

Selection of the inductor

The following paragraphs describe how to select an inductor. The value of the inductor to be used in the simulation circuit can be calculated from the following power supply parameters:

- Input voltage: V_{in} (V)
- Output voltage: V_{out} (V)
- Switching frequency: F_c (Hz)
- Output current: I_{out} (A)
- Inductor value: L (H)
- Inductor current ripple ratio: ΔI_L (%)

The ripple fluctuation rate (ΔI_L) of the inductor current is expressed as follows.

$$\Delta I_L = \left(\frac{V_{in} - V_{out}}{L} \times \frac{V_{out}}{V_{in} \times F_c} \right) \div I_{out} \times 100$$

The DC-DC power supply has the maximum inductor ripple current when the output current has the maximum value. From the power supply specifications, substituting 12 V for V_{in} (input voltage), 5 V for V_{out} (output voltage), 300 kHz for F_c (switching frequency), 8 A for I_{out} (output current), and 40 % for ΔI_L (inductor current ripple ratio), the inductor value (L) is calculated to be 3.04 μ H. Therefore, a 3.3 μ H inductor was selected for the simulation circuit.

In practice, the inductor value varies because of DC bias characteristics. Select an inductor that exhibits inductance greater than the above result even when inductance decreases because of DC bias characteristics.

3. Simulation results

This section shows the simulation waveforms at the points (1) to (2) shown in Fig. 3.1.

- (1) A non-isolated DC-DC power supply (buck converter) basic operation (Drain-source voltage and drain current of the high-side and low-side MOSFETs, and output inductor current)
- (2) Output voltage and current from the DC-DC power supply

The simulation circuit model also allows you to view other waveforms. See Section 5 for how to view waveforms.

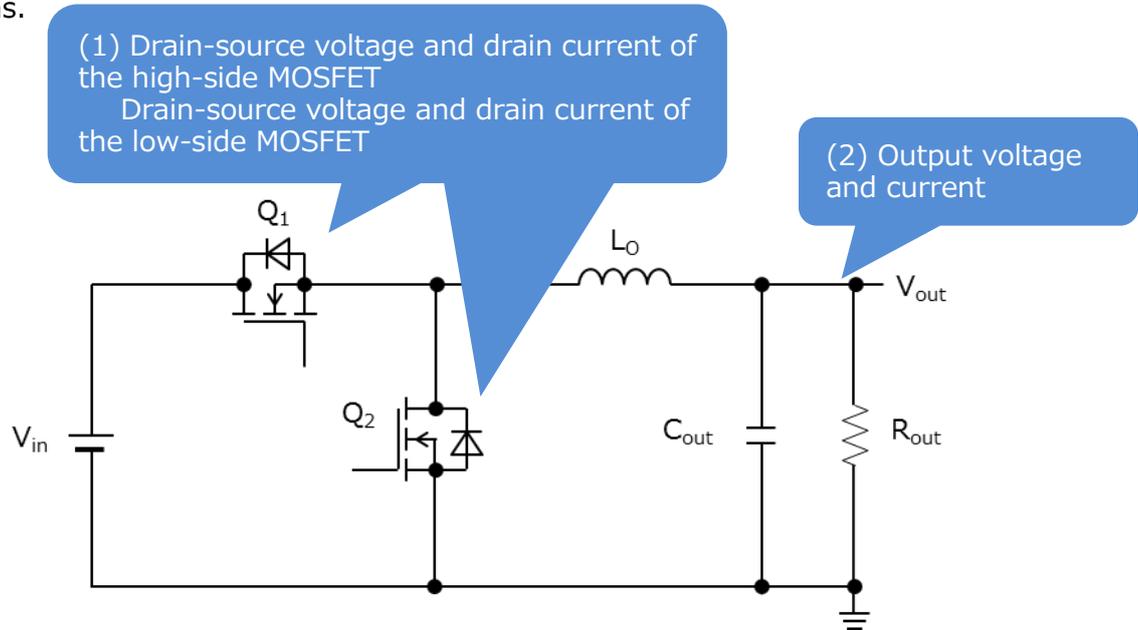


Fig. 3.1 Points at which simulation waveforms are measured

(1) Principle of operation of a buck converter

The following describes the basic operation of the buck DC-DC converter using the circuit shown in Fig. 3.2.

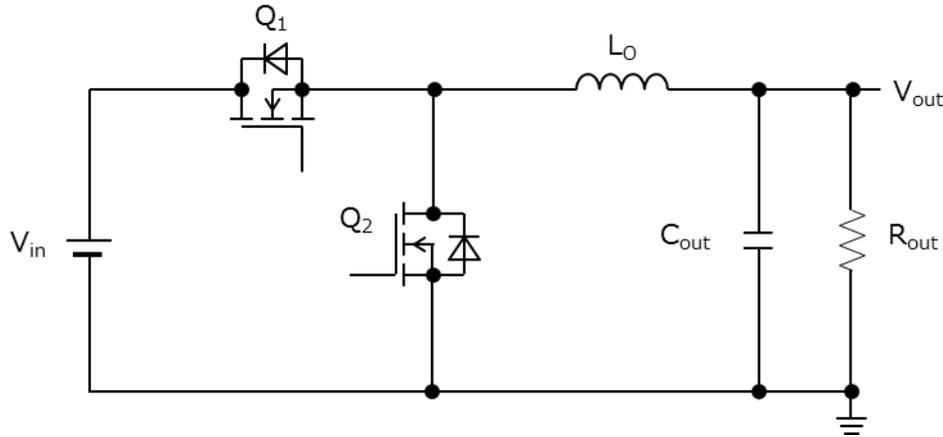


Fig. 3.2 Buck converter

Q_1 is the high-side MOSFET, Q_2 is the low-side MOSFET, and L_O is the output inductor. D (Q_2) is the drain-source body diode in Q_2 . Generally, a buck DC-DC converter is controlled with pulse-width modulation (PWM) by switching Q_1 at a constant frequency while varying its duty cycle ($D=T_{on}/T$). Its output voltage is expressed as:

$$V_{out} = \frac{T_{on}}{T} \times V_{in}$$

T : Cycle period (s)

T_{on} : On-time of the high-side MOSFET (s)

The operation of this buck converter during each on-off period is outlined below.

a. Q_1 ON, Q_2 OFF

When Q_1 turns on, $V_{Lo} = V_{in} - V_{out}$ is applied to the output inductor (L_O).

Therefore, the inductor current increases linearly according to the following equation, providing output current (I_{out}) while charging C_{out} .

$$i_{Lo(on)} = \frac{V_{in} - V_{out}}{L_O} \times T_{on}$$

During this period, magnetic energy is stored in L_O .

When Q_1 turns on, a pulsed current flows instantaneously, charging the output capacitance (C_{oss}) of Q_2 .

b. Q_1 OFF, Q_2 OFF.

This is a dead time during which both Q_1 and Q_2 are set to off in order to prevent a shoot-through current due to cross conduction. During this period, the energy stored in L_0 freewheels through $D(Q_2)$, supplying I_{out} . (v)

c. Q_1 OFF, Q_2 ON

Q_2 turns on, causing the $D(Q_2)$ freewheeling current to flow from the source to the drain of Q_2 , supplying I_{out} . The conduction loss that occurs while Q_2 is on is $I_D^2 \times R_{DS(ON)}$, which is smaller than diode conduction loss ($V_F \times I_F$). During this period, the current flowing through the inductor (L_0) decreases linearly according to the following equation:

$$i_{L_0(off)} = \frac{-V_{out}}{L_0} \times T_{off}$$

d. Q_1 OFF, Q_2 OFF

This is a dead time inserted to prevent shoot-through current in the same manner as b. During this period, a freewheeling current flows through $D(Q_2)$.

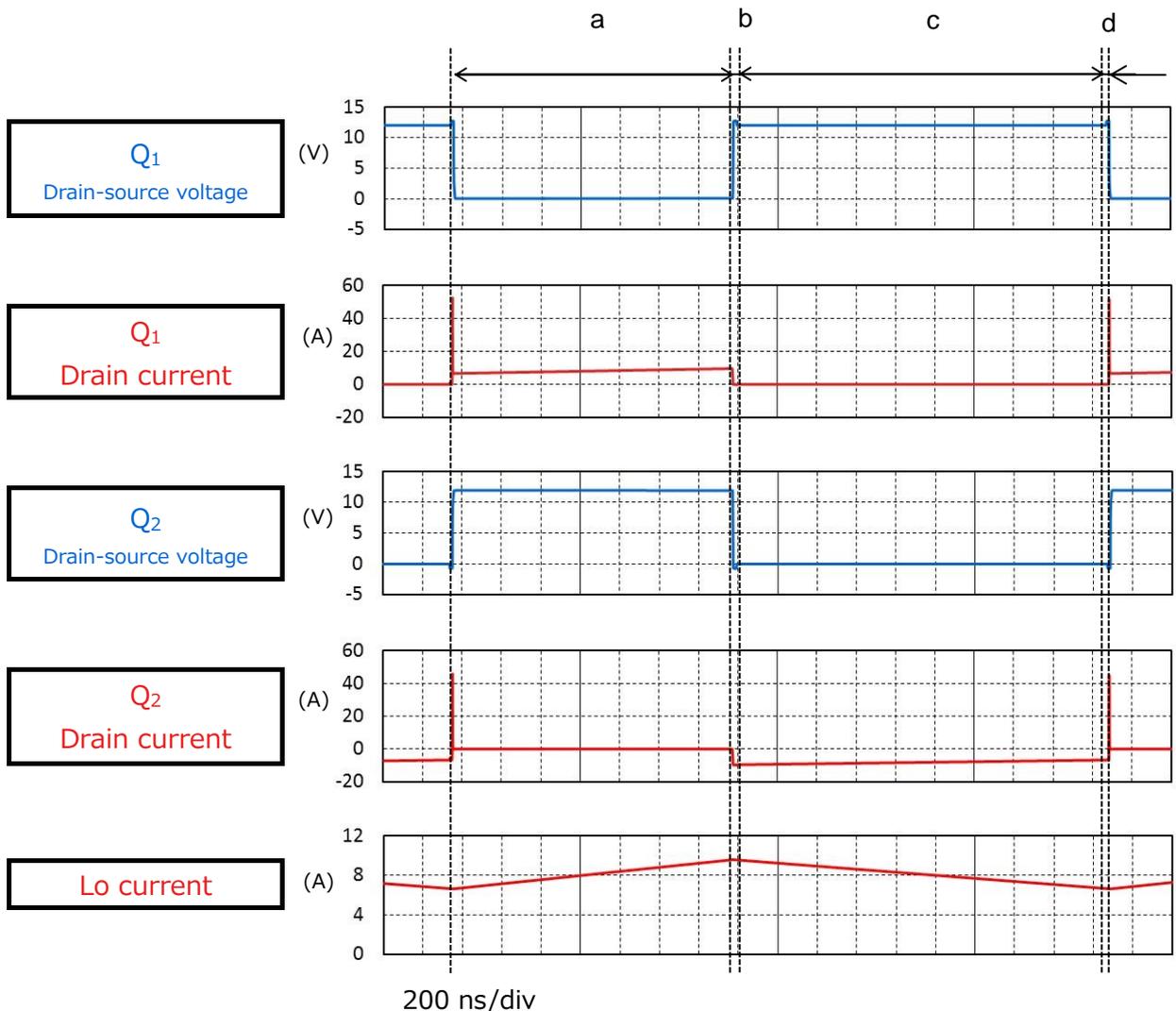


Fig. 3.3 Drain-source voltage and drain current of the high-side and low-side MOSFETs, and output inductor current

(2) Output voltage and current from the DC-DC power supply

Fig. 3.4 shows the waveforms of the output voltage and current from the DC-DC power supply. It shows that the DC-DC power supply achieves proper output voltage and current regulation following a soft-start period of 1 ms.

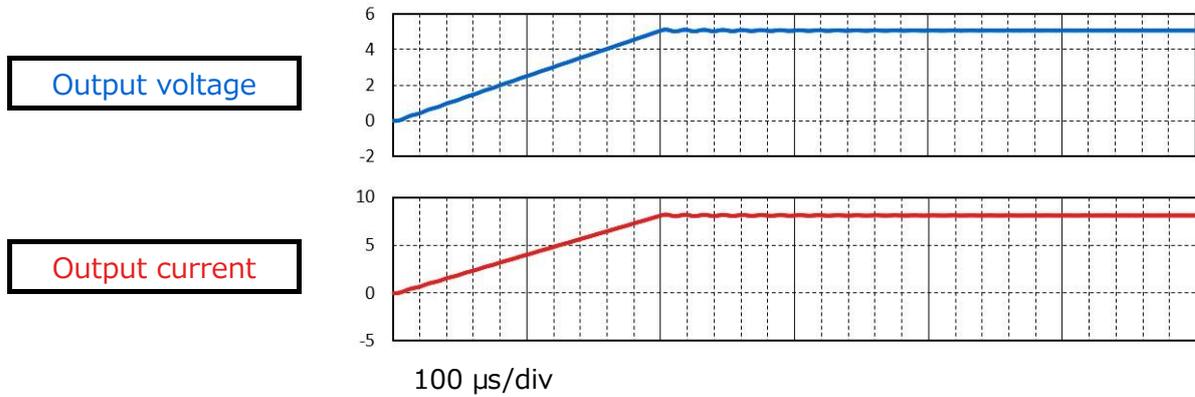


Fig. 3.4 Output voltage and current waveforms

4. Product Overview

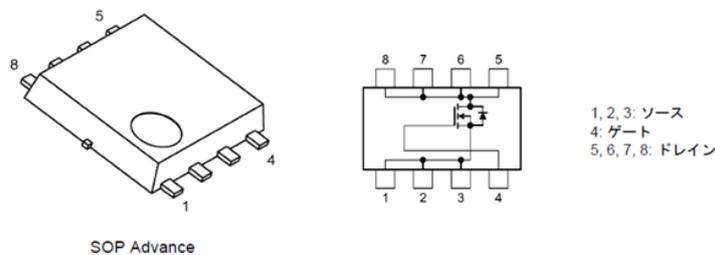
This section provides an overview of Toshiba's devices used as PSpice® models in the simulation circuit.

4.1. TPH6R003NL

Characteristics

- $V_{DSS}=30\text{ V}$, $I_D=38\text{ A}$
- Fast switching
- Low input gate charge: $Q_{SW}=4.3\text{ nC}$ (typ.)
- Low on-resistance: $R_{DS(ON)}=6.8\text{ m}\Omega$ (typ.) ($V_{GS}=4.5\text{ V}$)
- Low leakage current: $I_{DSS}=10\text{ }\mu\text{A}$ (max) ($V_{DS}=30\text{ V}$)
- Easy-to-use enhanced-mode MOSFET: $V_{th}=1.3\text{ to }2.3\text{ V}$ ($V_{DS}=10\text{ V}$, $I_D=0.2\text{ mA}$)

External view and pin assignment



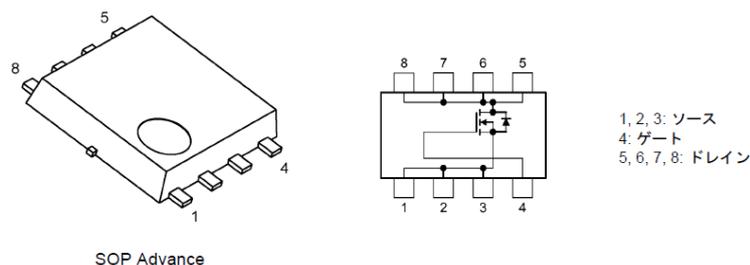
6.0mm (W) × 5.0mm (L) × 0.95mm (H)

4.2. TPH2R903PL

Characteristics

- $V_{DSS}=30\text{ V}$, $I_D=70\text{ A}$
- Fast switching
- Low input gate charge: $Q_{SW}=5.6\text{ nC}$ (typ.)
- Low output charge: $Q_{oss}=17\text{ nC}$ (typ.)
- Low on-resistance: $R_{DS(ON)}=2.1\text{ m}\Omega$ (typ.) ($V_{GS}=10\text{ V}$)
- Low leakage current: $I_{DSS}=10\text{ }\mu\text{A}$ (max) ($V_{DS}=30\text{ V}$)
- Easy-to-use enhanced-mode MOSFET: $V_{th}=1.1\text{ to }2.1\text{ V}$ ($V_{DS}=10\text{ V}$, $I_D=0.2\text{ mA}$)

External view and pin assignment



6.0mm (W) × 5.0mm (L) × 0.95mm (H)

5. Using the simulation circuit

You can freely change various parameters with OrCAD® Capture to verify the circuit operation according to the actual power supply specifications and evaluate how these parameters affect the circuit operation. This section shows how to set simulation parameters and verify the circuit operation.

Parameter settings

Table 5.1 shows the parameters you can set for the simulation circuit. Double-click a parameter name in the PARAMETERS section, then the Display Properties dialog box appears as shown in Fig. 5.1. Change the value in the Value field.

Table 5.1 Parameters that can be modified in the Parameters section

| Parameter name | Unit | Description |
|----------------|----------|--|
| Vin | V | Input voltage |
| Vout | V | Output voltage |
| L | H | Inductor value |
| Cin | F | Input capacitor |
| Cout | F | Output capacitor |
| Fc | Hz | Switching frequency |
| Vdrv_H | V | MOSFET drive voltage |
| Rdrv_off_h | Ω | Internal resistance of the turn-off gate driver for the high-side MOSFET |
| Rdrv_on_h | Ω | Internal resistance of the turn-on gate driver for the high-side MOSFET |
| Rdrv_off_l | Ω | Internal resistance of the turn-off gate driver for the low-side MOSFET |
| Rdrv_on_l | Ω | Internal resistance of the turn-on gate driver for the low-side MOSFET |

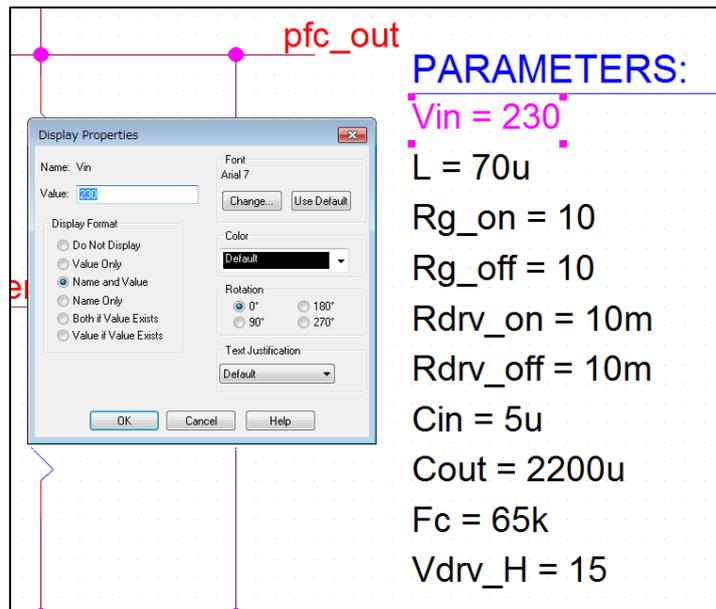


Fig. 5.1 Display Properties dialog box

Setting analysis parameters

The following describes how to run a simulation on the simulation circuit.

1. From the menu bar of OrCAD® Capture, select **PSpice - New Simulation Profile**. Then, the New Simulation dialog box shown in Fig. 5.2 appears. Enter an arbitrary profile name and click **Create**.

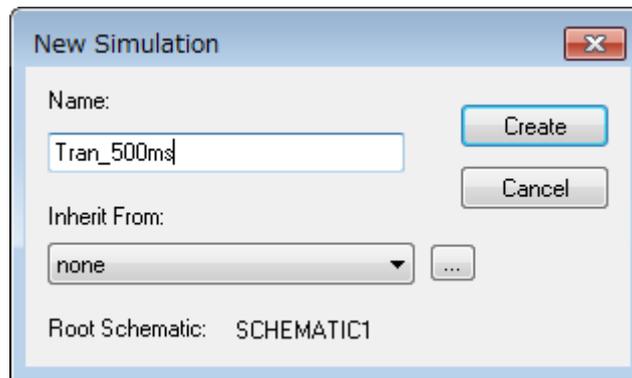


Fig. 5.2 New Simulation dialog box

- (2) The Simulation Settings dialog shown in Fig. 5.3 appears, which allows you to set various analysis parameters. Select the **Analysis** tab first. Select **Time Domain (Transient)** from the **Analysis Type** drop-down list. Enter an analysis end time in the **Run To Time** field and the maximum step size in the **Maximum Step Size** field.

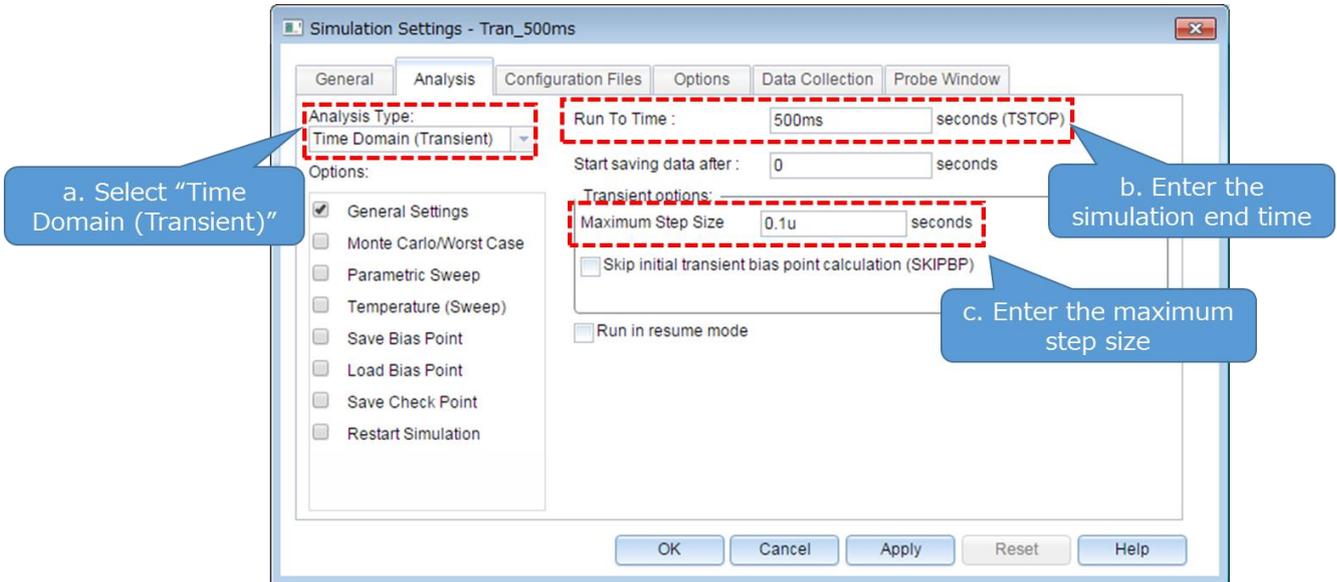


Fig. 5.3 Simulation Settings - Analysis dialog box

3. Click the **Options** tab to choose analysis options. For the simulation of our model, it is recommended to check **Analog Simulation - Auto Converge - AutoConverge** as shown in Fig. 5.4 to enable the automatic convergence feature.

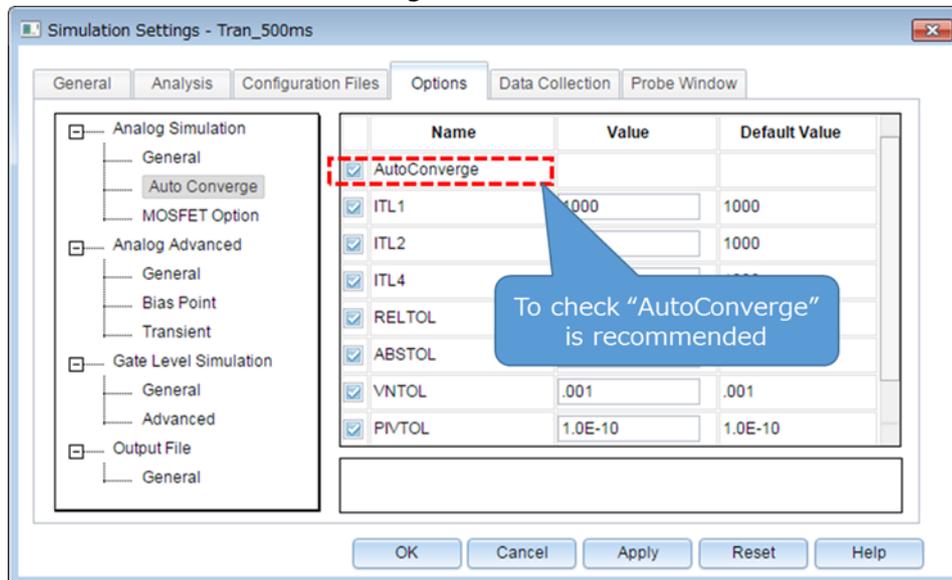


Fig. 5.4 Simulation Settings - Options dialog box

4. Click **OK** to close the Simulation Settings dialog box.
5. To run a simulation, select **PSpice - Run** from the menu bar of OrCAD® Capture. Then, PSpice A/D starts automatically and runs a simulation.

Viewing simulation results

The following describes how to view the simulation results. You can display the waveforms of the simulation results in two ways.

Method 1: Selecting traces

1. Right-click outside the graph area and select **Add Trace** as shown in Fig. 5.5.
2. Then, the Add Traces dialog box shown in Fig. 5.6 appears. Select traces to be added to a selected plot. To view a voltage waveform, select V(trace_name). To view a current waveform, select I(device_name). See Fig. 5.6.
3. Click **OK**. Then, the selected waveform appears as shown in Fig. 5.7.

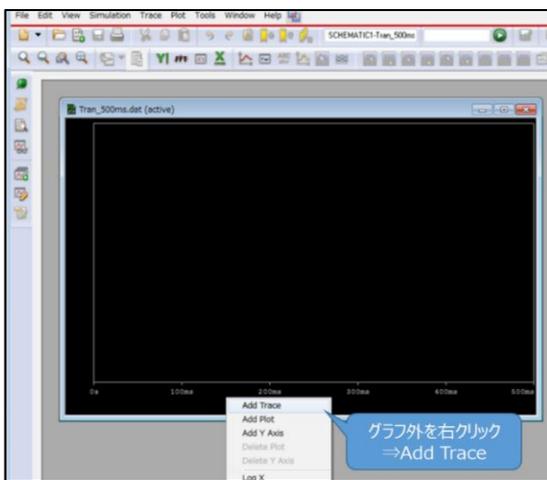


Fig. 5.5 Graph window

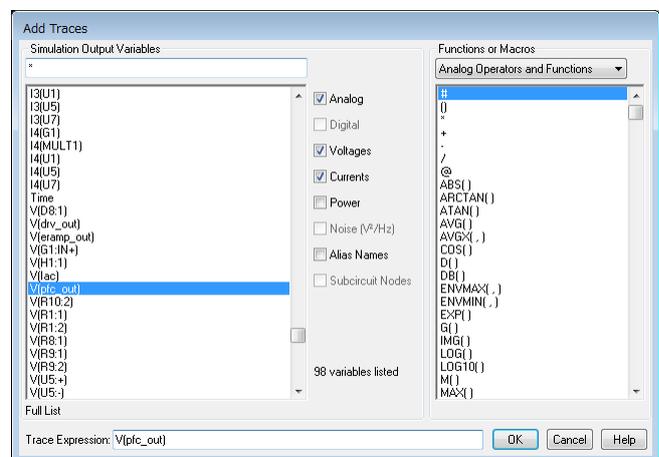


Fig. 5.6 Add Traces dialog box

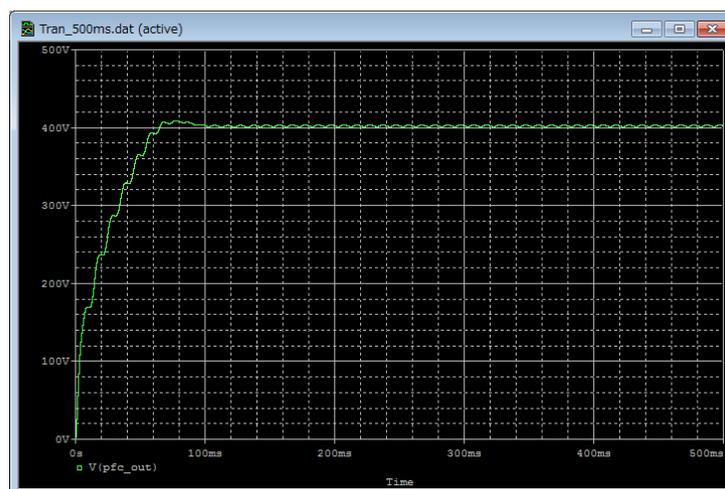


Fig. 5.7 Simulation waveform view (Example: PFC output voltage waveform)

Method 2: Adding markers

1. From the menu bar of OrCAD® Capture, select **PSpice - Markers** and then a type of marker as shown in Fig. 5.8.
2. Place the selected marker on the desired node in the simulation circuit as shown in Fig. 5.9.
3. Then, its waveform appears in the graph window of PSpice A/D as shown in Fig. 5.10.

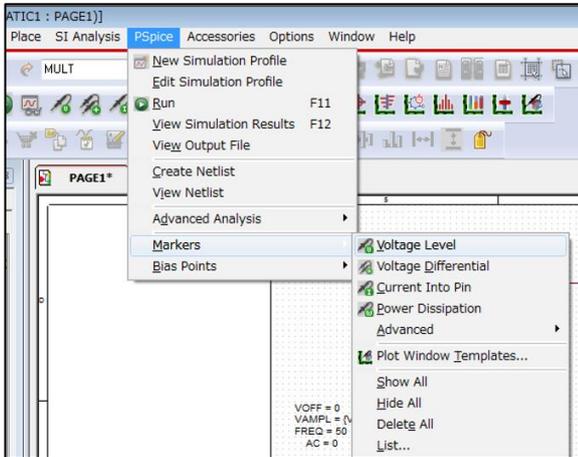


Fig. 5.8 Selecting a marker type

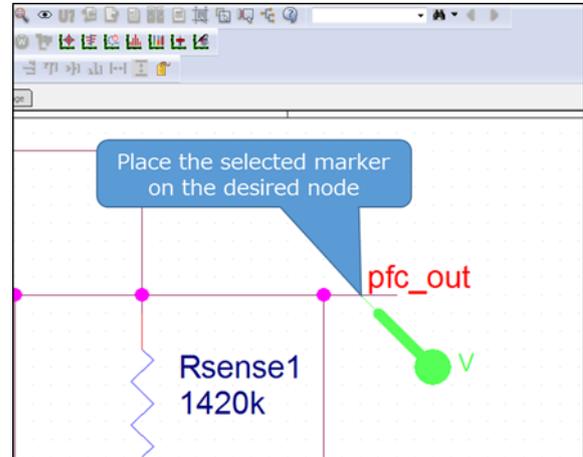


Fig. 5.9 Placing a marker in the circuit

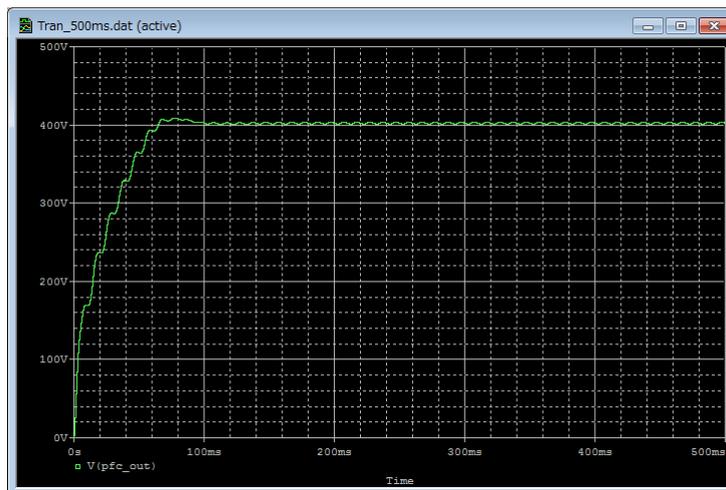


Fig. 5.10 Simulation waveform view (Example: Output voltage waveform)

※Cadence, Cadence logo, OrCAD, PSpice and OrCAD logo are trademarks or registered trademarks of Cadence Design Systems, Inc. in the U.S. and other countries.

Terms of Use

This terms of use is made between Toshiba Electronic Devices and Storage Corporation ("We") and customers who use documents and data that are consulted to design electronics applications on which our semiconductor devices are mounted ("this Reference Design"). Customers shall comply with this terms of use. Please note that it is assumed that customers agree to any and all this terms of use if customers download this Reference Design. We may, at its sole and exclusive discretion, change, alter, modify, add, and/or remove any part of this terms of use at any time without any prior notice. We may terminate this terms of use at any time and for any reason. Upon termination of this terms of use, customers shall destroy this Reference Design. In the event of any breach thereof by customers, customers shall destroy this Reference Design, and furnish us a written confirmation to prove such destruction.

1. Restrictions on usage

1. This Reference Design is provided solely as reference data for designing electronics applications. Customers shall not use this Reference Design for any other purpose, including without limitation, verification of reliability.
2. This Reference Design is for customer's own use and not for sale, lease or other transfer.
3. Customers shall not use this Reference Design for evaluation in high or low temperature, high humidity, or high electromagnetic environments.
4. This Reference Design shall not be used for or incorporated into any products or systems whose manufacture, use, or sale is prohibited under any applicable laws or regulations.

2. Limitations

1. We reserve the right to make changes to this Reference Design without notice.
2. This Reference Design should be treated as a reference only. We are not responsible for any incorrect or incomplete data and information.
3. Semiconductor devices can malfunction or fail. When designing electronics applications by referring to this Reference Design, customers are responsible for complying with safety standards and for providing adequate designs and safeguards for their hardware, software and systems which minimize risk and avoid situations in which a malfunction or failure of semiconductor devices could cause loss of human life, bodily injury or damage to property, including data loss or corruption. Customers must also refer to and comply with the latest versions of all relevant our information, including without limitation, specifications, data sheets and application notes for semiconductor devices, as well as the precautions and conditions set forth in the "Semiconductor Reliability Handbook".
4. When designing electronics applications by referring to this Reference Design, customers must evaluate the whole system adequately. Customers are solely responsible for all aspects of their own product design or applications. **WE ASSUME NO LIABILITY FOR CUSTOMERS' PRODUCT DESIGN OR APPLICATIONS.**
5. No responsibility is assumed by us for any infringement of patents or any other intellectual property rights of third parties that may result from the use of this Reference Design. No license to any intellectual property right is granted by this terms of use, whether express or implied, by estoppel or otherwise.
6. **THIS REFERENCE DESIGN IS PROVIDED "AS IS". WE (a) ASSUME NO LIABILITY WHATSOEVER, INCLUDING WITHOUT LIMITATION, INDIRECT, CONSEQUENTIAL, SPECIAL, OR INCIDENTAL DAMAGES OR LOSS, INCLUDING WITHOUT LIMITATION, LOSS OF PROFITS, LOSS OF OPPORTUNITIES, BUSINESS INTERRUPTION AND LOSS OF DATA, AND (b) DISCLAIM ANY AND ALL EXPRESS OR IMPLIED WARRANTIES AND CONDITIONS RELATED TO THIS REFERENCE DESIGN, INCLUDING WARRANTIES OR CONDITIONS OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, ACCURACY OF INFORMATION, OR NONINFRINGEMENT.**

3. Export Control

Customers shall not use or otherwise make available this Reference Design for any military purposes, including without limitation, for the design, development, use, stockpiling or manufacturing of nuclear, chemical, or biological weapons or missile technology products (mass destruction weapons). This Reference Design may be controlled under the applicable export laws and regulations including, without limitation, the Japanese Foreign Exchange and Foreign Trade Law and the U.S. Export Administration Regulations. Export and re-export of this Reference Design are strictly prohibited except in compliance with all applicable export laws and regulations.

4. Governing Laws

This terms of use shall be governed and construed by laws of Japan.