Half bridge (HB) DC-DC Power Supply Basic Simulation Circuit

Reference Guide

RD174-RGUIDE-01

TOSHIBA ELECTRONIC DEVICES & STORAGE CORPORATION

Contents

1. Introduction

Most electrical equipment, including information and communications equipment and home appliances, operate with DC voltage. Therefore, it is not possible to operate directly with commercial power supplied by AC voltage, and it is necessary to convert AC voltage to DC voltage and supply it.

The AC voltage is converted from AC voltage to DC voltage by the AC-DC power supply. Depending on the power supply specifications of the loads in the equipment, the DC voltage supplied from the AC-DC power supply must be further converted to DC voltage according to the specifications.

Fig. 1.1 shows an example of the configuration of the power supply lines in the equipment. There are several power supply lines depending on the load, and the load may be connected to the output voltage supplied from the AC-DC power supply as it is, or the output voltage may be further converted to a different DC voltage via the DC-DC power supply. The DC-DC power supply converts the DC voltage into a different DC voltage.

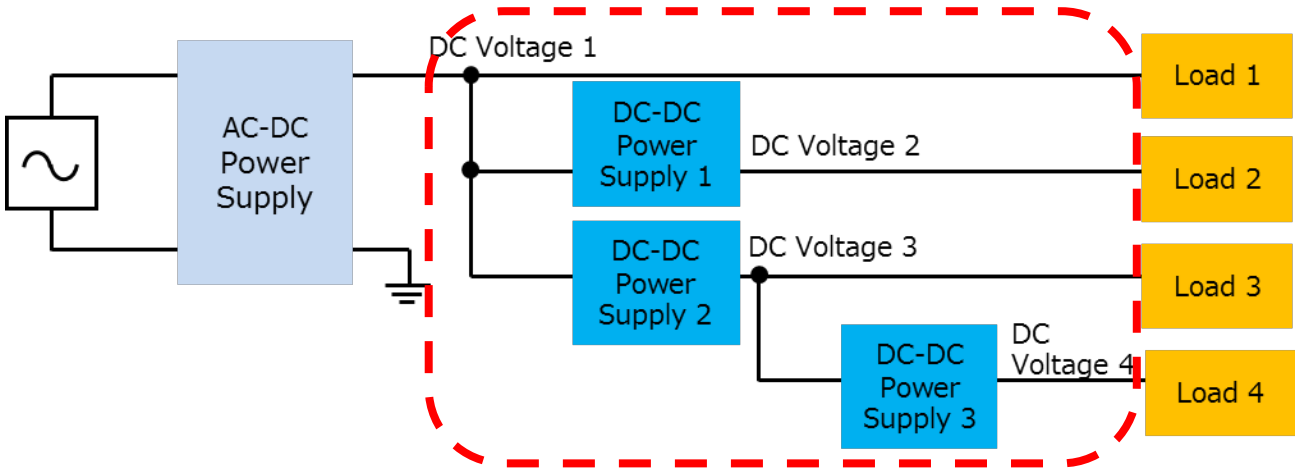


Fig. 1.1 Example of power supply line configuration in the equipment.

There are two main types of methods for converting DC voltage into DC voltage: the series regulation method and the switching regulation method (hereafter, the series method and the switching method). This document deals with switching schemes that are mainstream in DC-DC power supplies. The switched-based adjusts on/off of the switching element to generate a desired voltage, which makes the control circuit more complicated than the series method, but it generally reduces the loss compared to the series method.

The switching system includes an isolated DC-DC power supply in which the input side and the output side are isolated through a transformer, and a non-isolated DC-DC power supply in which the input side and the output side are not isolated. This document deals with isolated DC-DC power supplies.

The isolated DC-DC power supply switches the input DC voltage at frequencies of several 10-200 kHz with switching elements, transfers power to the secondary side via transformers, then rectifies, smooths, and outputs the DC voltage. The DC voltage of the final stage is controlled by adjusting the on/off period of the switching element.

Fig. 1.2 shows exemplary circuit blocks of an isolated switched-based DC-DC power supply. It consists of two blocks: (1) DC-DC converters and (2) feedback circuits. The function of each block is shown below.

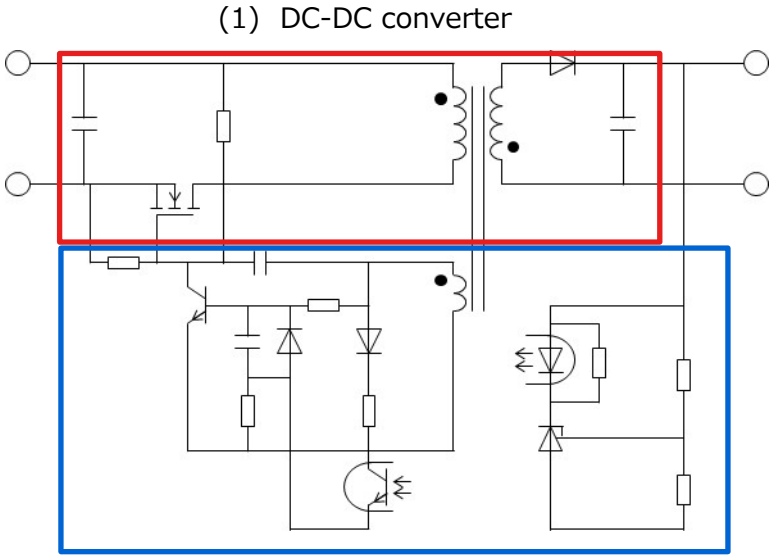


Fig. 1.2 Examples of circuit blocks for isolated switched DC-DC power supply

(1) DC-DC converter

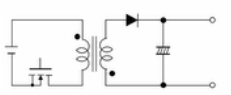
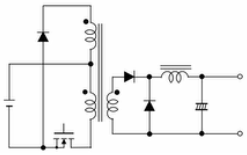
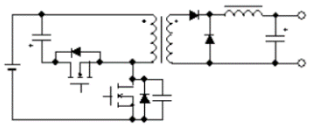
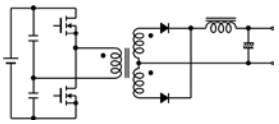
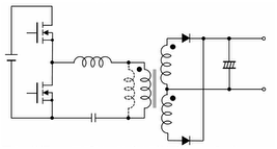
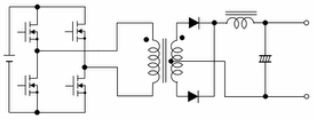
Converts the input DC voltage to an arbitrary DC voltage.

(2) Feedback circuit

The on/off of the switching element is controlled so that the output voltage becomes a desired value.

There are a variety of topologies for isolated-switching DC-DC converters that use transformers. Table 1.1 shows typical topologies and characteristics of DC-DC converters.

Table 1　Commonly used DC-DC converter topologies and their characteristics

Circuitry system for DC-DC converter		Power level	Advantages	Disadvantages
Flyback		< 120 W	* Small part count	* Decrease in efficiency at high power * Large transformer
Forward		100 W~500 W	* Higher efficiency than a flyback Circuit	* Requires a transformer reset circuit
ACF (Active clamp Forward		100 W~1 kW	* Higher efficiency than a forward circuit	* Large part count * Difficult to control
Half-bridge		100 W~ 1.6 kW	* High efficiency * Low noise	* Requires a custom -designed transformer * Difficult to control
Resonant Half-bridge (LLC resonance)		100 W~ 1.6 kW	* Higher efficiency than a half bridge * Low noise	* Specially designed transformers are required. * Difficult to control
Full-bridge		> 1 kW	* High efficiency * Capable of increasing the power capacity	* Large part count * Difficult to control

The Half-bridge method used in this document is widely adopted for power supplies that require high efficiency and high power density by switching two switching elements that are highly efficient and have a relatively low breakdown-voltage of the voltage applied to the primary side of the transformer (1/2 of the V_{in}) , which is a relatively low voltage MOSFET). The basic simulation circuit (RD174-SPICE-01) is provided on our website in order to understand the operation of the Half-bridge-type DC-DC converter.

This document describes the outline and usage of this simulation circuit.

Cadence's OrCAD®Capture and PSpice®A D tools are required to operate the simulator.

The simulator circuits and documentation are based on OrCAD® 17.2.

2. Outline of power supply for HB type DC-DC converter

The basic simulation circuit (RD174-SPICE-01) is a 100 W HB DC-DC power supply.

2.1. Power specification

The specifications of the HB type DC-DC power supply described in this document are as follows.

- Input voltage: 48 V
- Output voltage: 5 V
- Output current: 0 to 20 A
- Operating frequency of primary MOSFET: 150 kHz
- Winding ratio: n_1:n_2:n_3 = 3:1:1
- Secondary Allowable Ripple Current Width: 30 %

2.2. Circuit configuration

Fig. 2.1 shows the simulation circuit for OrCAD®. This is a HB type DC-DC converter power supply, which mainly consists of a power unit (Half-bridge) and a PWM-controller unit. The transformer secondary side of the power section is a synchronous rectification circuit using MOSFET. The PWM controller is a general-purpose controller with a built-in MOSFET gate driver, which is provided to realize the PWM circuit. The circuit use the TPN13008NH and TPH2R003PL as switching MOSFETs.

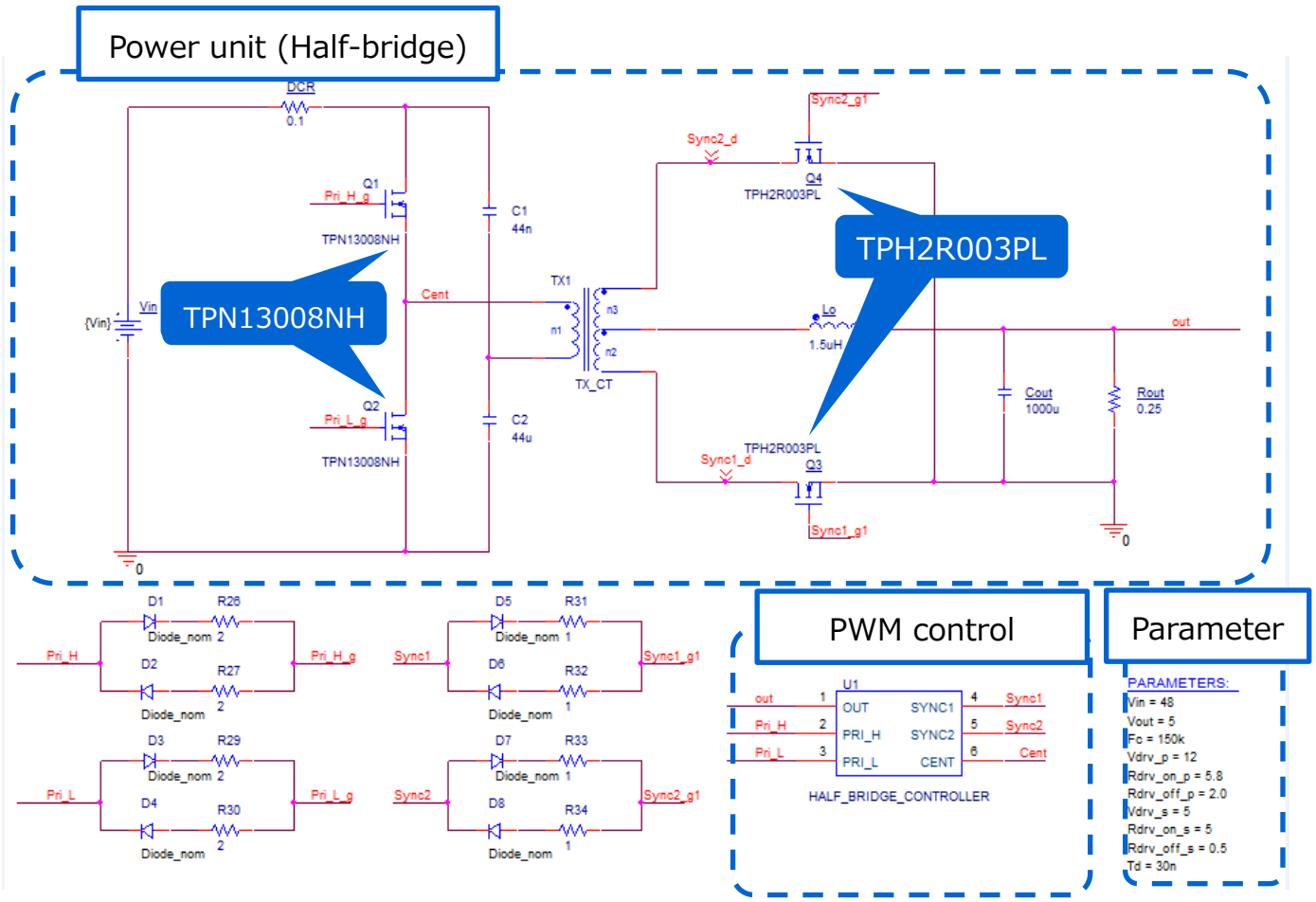


Fig. 2.1 Simulation circuit of a 100 W HB (DC-DC Converter) Power supply

Selection of primary MOSFET

Primary MOSFET (TPN13008NH: V_{DSS} = 80 V, I_D = 18 A) is selected from the following perspectives.

(1) Device breakdown-voltage

The voltage applied to the device at steady state is the input voltage (=48 V). Select a device with a breakdown-voltage of 80 V or higher, taking into account the surge voltage at the time of switching and other factors.

(2) Current rating

The HB DC-DC power supply has the maximum input current when the output power has the maximum value. If the conversion efficiency at maximum output power =100 W is 85 %, the maximum average input current will be 2.45 A. Select a device with a current rating of 5 A or higher.

Selection of secondary MOSFET

Secondary MOSFET (TPH2R003PL: V_{DSS} = 30 V, I_D = 100 A) is selected from the following perspectives.

(1) Device breakdown-voltage

The primary winding voltage at steady state is 24 V of $V_{in}/2$. Since the winding ratio is 3:1, the voltage between the mid-point and both ends of the secondary winding is 8 V, 1/6 of the input voltage, and 16 V, 2 times the voltage, is applied to the secondary MOSFET. Select a device with a breakdown-voltage of 30 V or higher because a surge-voltage is generated when the transformer current is switched.

(2) Current rating

The HB DC-DC power supply has the maximum input current when the output power has the maximum value. Maximum output current of 20 A at maximum output power =100 W is shared by the MOSFET of the two current paths on the secondary side. Therefore, a current of 10 A flows per path. Therefore, we emphasize suppression of conduction loss at a current rating of 20 A or higher and select a device with as low an on-resistance as possible.

Selection of output inductor

This section explains how to select the output inductor on the secondary side. The inductance value of the output inductor in this simulation circuit can be calculated using the following items, which are power supply specifications.

- Input voltage: V_{in} (V)
- Transformer winding ratio: n_2/n_1
- Output voltage: V_{out} (V)
- Power-conversion efficiency
- Two times the switching frequency of the primary MOSFET: F_c (Hz)
- Maximum output current: I_{out_max} (A)
- Allowable ripple current width: ΔI_{ripple} (%)

The inductance value (L_O) of the output inductor is calculated by the following equation.

$$L_o = \frac{\left(\frac{n_2}{n_1} \times \frac{V_{in}}{2} - V_{out}\right) \times V_{out}}{\left(\frac{n_2}{n_1} \times \frac{V_{in}}{2} \times F_c\right) \times I_{out\_max} \times \Delta I_{ripple} \times 0.01}$$

Based on the power supply specifications, suppose that the input voltage (V_{in}) is 48 V, the transformer winding ratio (n_2/n_1) is 1/3, the output voltage (V_{out}) is 5 V, the switching frequency (F_c) is 300 kHz, the maximum output current (I_{out_max}) is 20 A, and the allowable ripple current width (ΔI_{ripple}) is 30 %.

From the above equation, the inductance value (L_O) of the output inductor is calculated to be 1.04 μH. Select 1.5 μH as the set value.

In the actual design, the inductance value of the inductor changes due to the DC superposition characteristic. Select a component that can secure the calculated value in a state where the inductance value is lowered due to the DC superposition characteristic.

3. Simulation result

Here, the operation simulation waveforms of the respective parts in the simulation circuit are shown by the respective points in Fig. 3.1 ((1) to (4)).

(1) HB primary-side basic operation ("primary-side MOSFET drain-source voltages and currents")

(2) Secondary side synchronous rectification operation ("secondary side MOSFET drain-source voltage/current")

(3) Voltage and current across the output inductor

(4) "Output voltage/current" as a power supply

When the circuit model is actually used, waveforms can be displayed for parts other than those shown in Fig. 3.1. The method of displaying waveforms is described in Chapter 5.

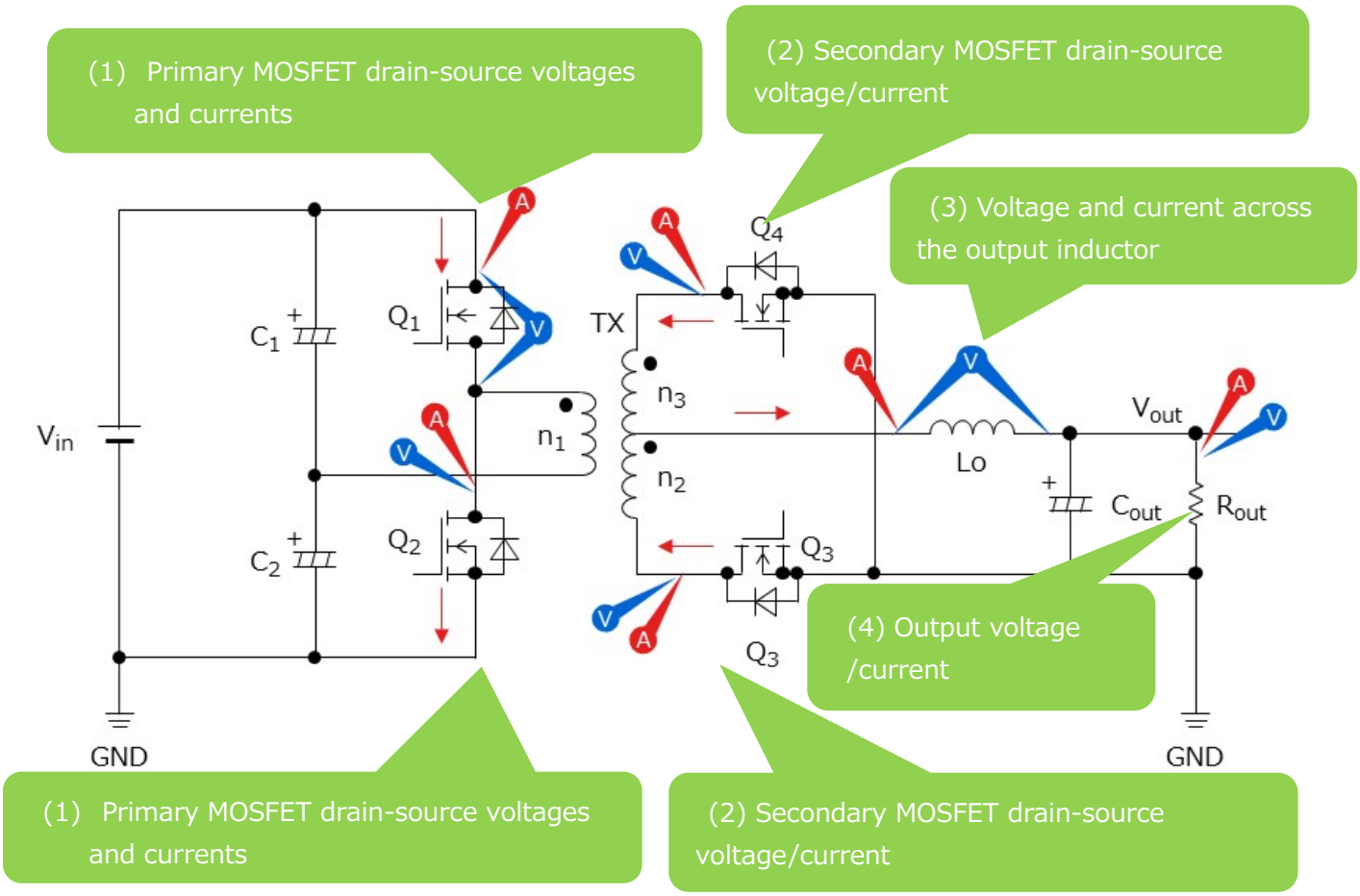


Fig. 3.1 Points at which simulation waveforms are measured

(1) Basic operation of HB system

Fig. 3.2, which shows the DC-DC converter, explains the basic operation of the HB method.

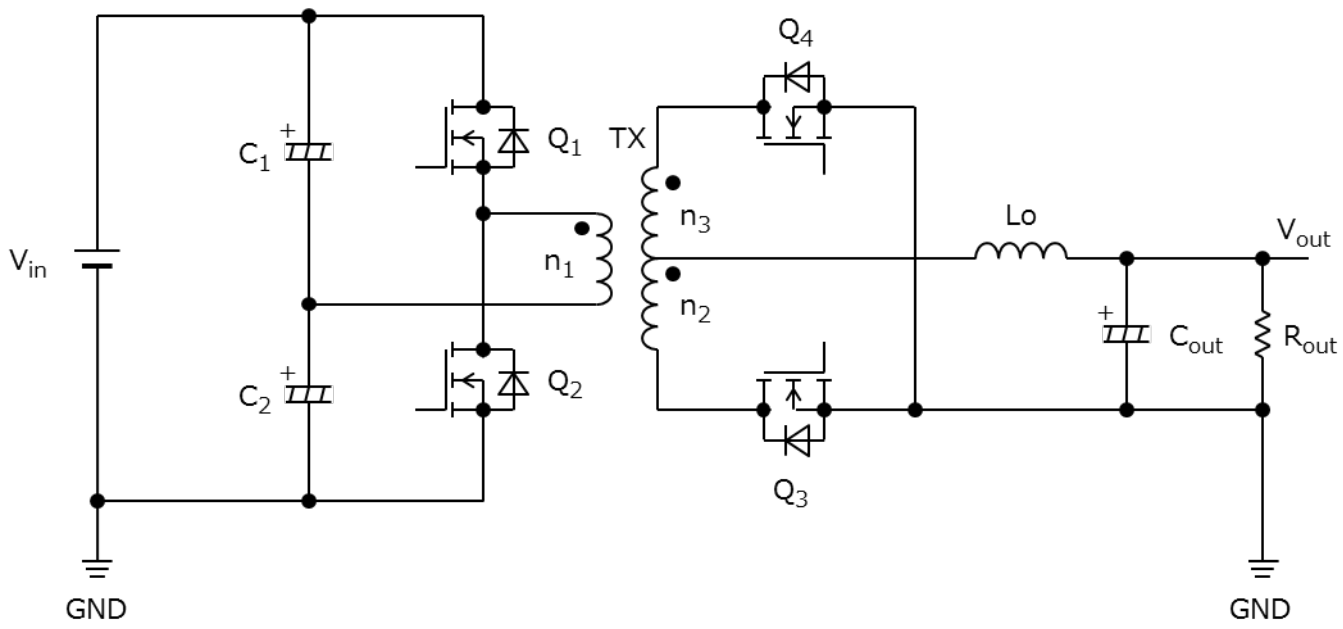


Fig. 3.2 HB Circuit

Let the primary MOSFET be Q1 and Q2, the secondary MOSFET be Q 3 and Q 4, and the power inductor be LO. In the HB method, Q1 and Q2 are alternately turned on and off at the same on-time, and PWM (Pulse Width Modulation) control is used to control the output voltage.

The output voltage V_{out} is as follows.

$$V_{out} = \frac{n_2}{n_1} \times V_{in} \times \frac{T_{on}}{T}$$

(T: primary cycle)

(T_{on}: On-time)

The operation of the HB is divided into four sections a to d according to the operation of the Q_1 and Q_2.

a. Q_1 ON, Q_2 OFF

The period during which power is transferred from the primary side to the secondary side.

Theprimary current is supplied through the V_{in}→Q_1→n_1→C_2 and C_1→Q_1→n_1→C_2 paths.

At this time, the primary winding voltage is 1/2 of the input voltage (V_{in}). Voltages corresponding to the winding ratio are applied to the secondary side winding with the polarity symbol side of the n_2 being positive.

$$\frac{n_2}{n_1} \times \frac{V_{in}}{2}$$

Is applied to the L_O through the Q_3,

$$\Delta i_{Lo(Q1\_on)} = \frac{1}{L_O} \times \left(\frac{n_2}{n_1} \times \frac{V_{in}}{2} - V_{out}\right) \times T_{on}$$

And the current determined by {circumflex over (x)} increases linearly and flows to charge the C_{out} and provide the output current I_{out}. At this time, L_O stores magnetic energy.

b. Q_1 OFF, Q_2 OFF

The MOSFET between the Q1 and the Q2 is turned off, and the energy stored in the L_O is circulation through the Q3 and the Q6. To the LO at this time

$$\Delta i_{Lo(all_{off})} = \frac{-V_{out}}{L_O} \times \left(\frac{T}{2} - T_{on}\right)$$

The current flowing decreases by Δi_{LO} (all $_{off}$) and flows.

c. Q_1 OFF, Q_2 ON

The period during which power is transferred from the primary side to the secondary side.

The primary current is supplied through the $V_{in} \rightarrow C_1 \rightarrow n_1 \rightarrow Q_2$ and $C_2 \rightarrow n_1 \rightarrow Q_2 \rightarrow C_2$ paths. During this period, the primary winding voltage is-V_{in} /2, and the secondary winding voltage depends on the winding ratio with the polarity symbol side of the n_3 being minus.

$$\frac{n_3}{n_1} \times \frac{V_{in}}{2}$$

To the L_O through the Q_4 in the same manner as in a Current flows.

$$\Delta i_{Lo(Q2\_on)} = \frac{1}{L_O} \times \left(\frac{n_2}{n_1} \times \frac{V_{in}}{2} - V_{out}\right) \times T_{on}$$

d. Q_1 OFF, Q_2 OFF

Like b, the energy stored in the L_O passes through Q_3 and Q_4 Current flows.

$$\Delta i_{Lo(all_{off})} = \frac{-V_{out}}{L_O} \times \left(\frac{T}{2} - T_{on}\right)$$

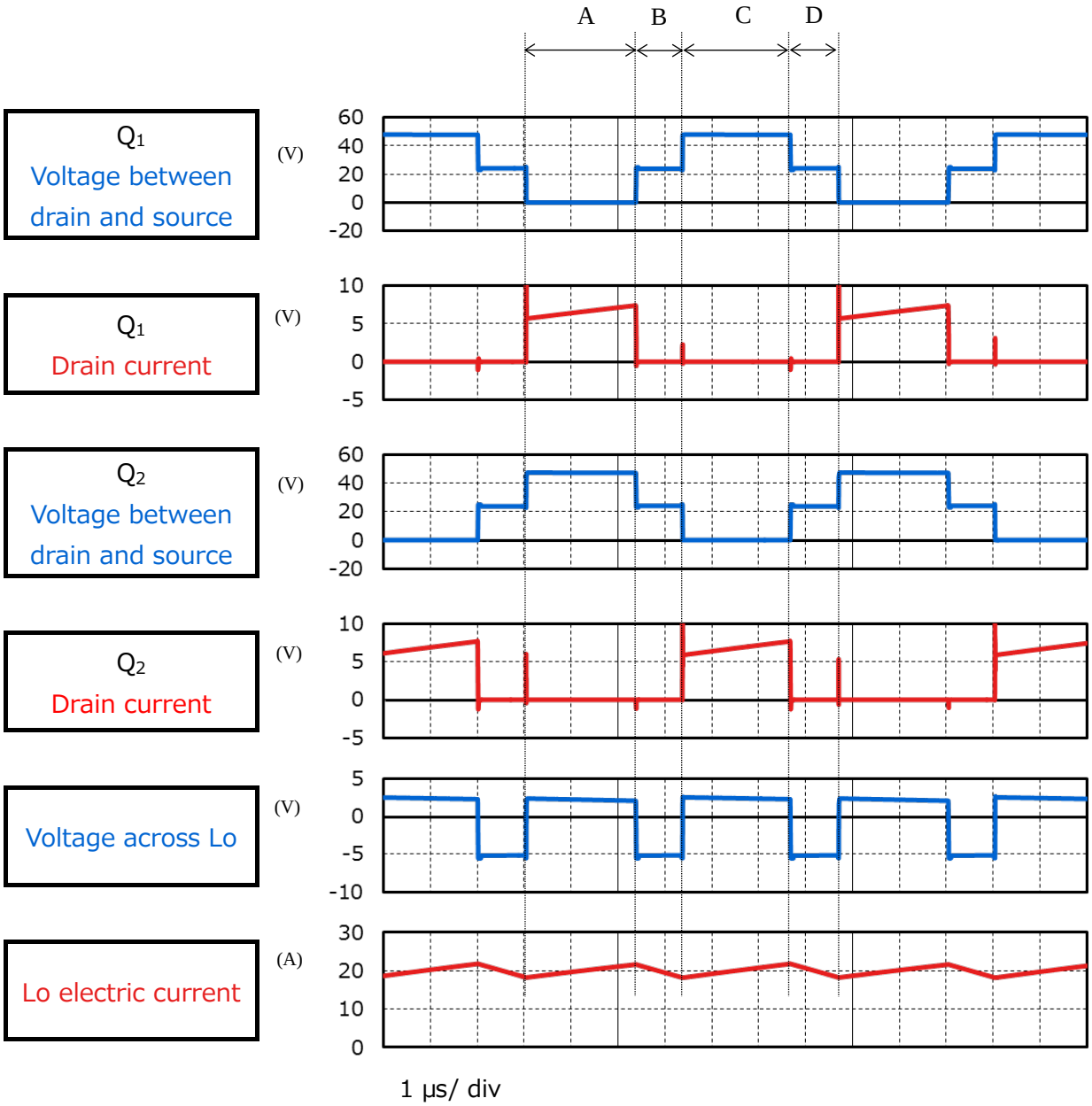


Fig. 3.3 Waveforms of the drain-source voltage of the primary-side MOSFETs and the voltage and current waveforms of the output inductor

(2) Synchronous rectification operation on the secondary side

This simulation circuit employs a synchronous rectifier circuit using MOSFET instead of diodes as the secondary rectifier. Since the conduction loss due to the on-resistance of the MOSFET is generally smaller than the conduction loss due to the forward voltages of the diodes, synchronous rectifiers can reduce the loss. The larger the output current is, the greater the loss reduction effect of the synchronous rectifier circuit is, which is often adopted in applications requiring high efficiency and large capacity.

The operation of the secondary MOSFET during the periods is as follows.

a. Q_3 ON and Q_4 OFF

Voltages corresponding to winding ratios with the polarity symbol side of the secondary winding n_2 as a plus side

$$\frac{n_2}{n_1} \times \frac{V_{in}}{2}$$

The current flows through the Q_3 to the L_O.

b. Q_3 ON, Q_4 ON

Energy stored in the L_O is recycled through the Q_3 and Q_4.

c. Q_3 OFF, Q_4 ON

Voltages corresponding to winding ratios with the polarity symbol side of the secondary side winding n_3 set to minus

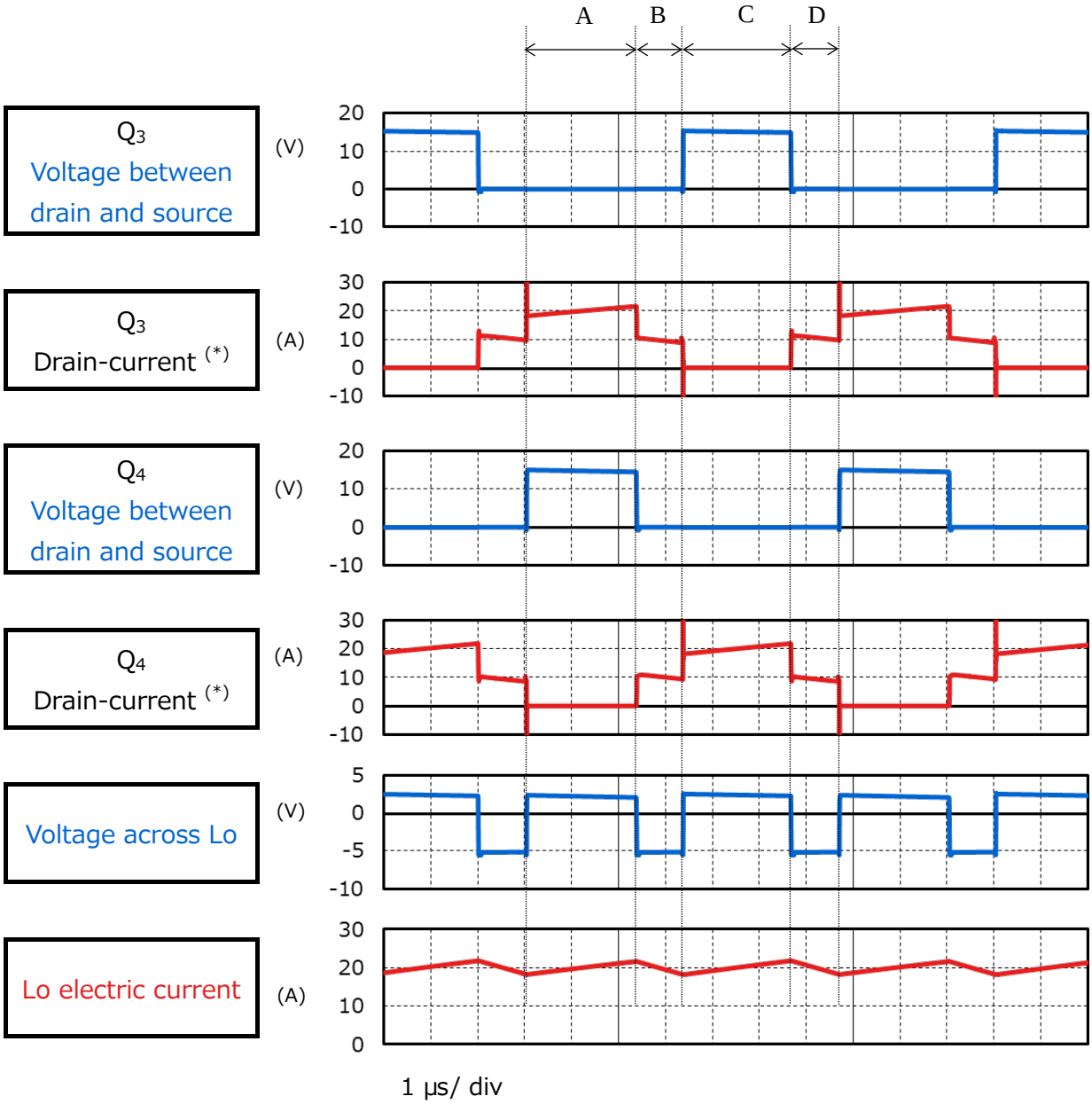
$$\frac{n_3}{n_1} \times \frac{V_{in}}{2}$$

The current flows through the Q_3 to the L_O.

d. Q_3 ON, Q_4 ON

The electric power stored in the L_O is recycled through the Q_3 and the Q_4.

(3) Voltage and current across the output inductor



* : The drain current flows from the MOSFET to the secondary winding of the transformer in the positive direction.

Fig. 3.4 Voltage between Drain-Source of Secondary MOSFET, Current Output Inductor Voltage and Current Waveforms

(4) “Output voltage / current” as a power supply

Fig. 3.5 shows the output voltage and current waveforms of this power supply circuit. After about 1ms, it can be seen that the setting voltage and current are stable.

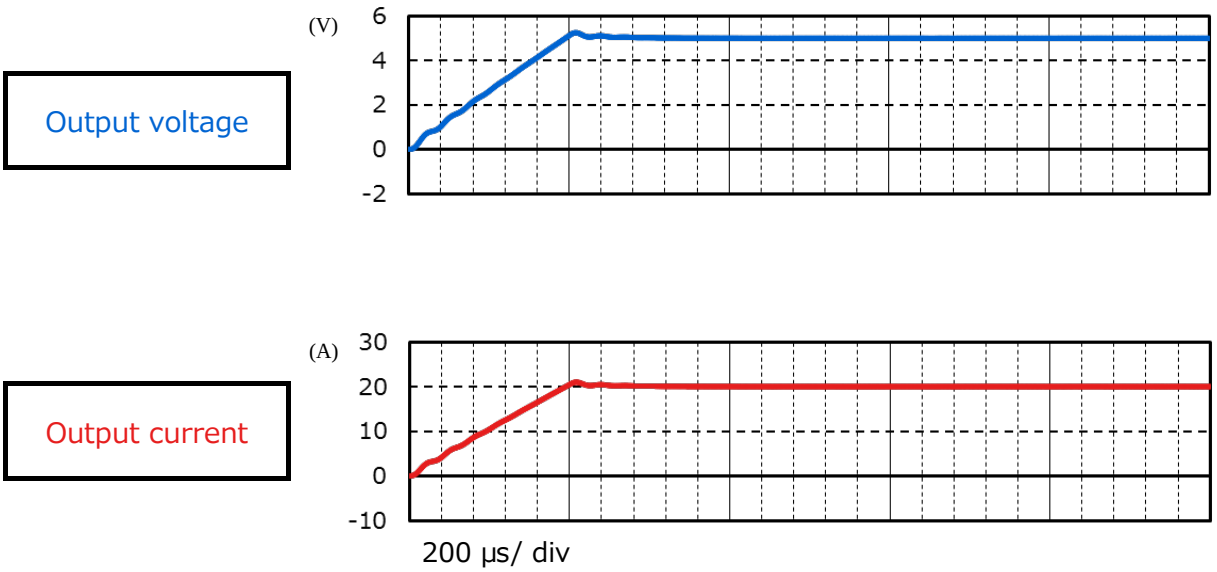


Fig. 3.5 Output Voltage/Current Waveform

4. Product Overview

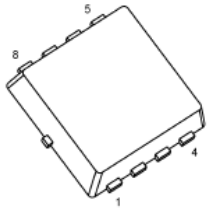
This section introduces the outline of our products that have been tested by incorporating PSpice® models into these circuits.

4.1. TPN13008NH

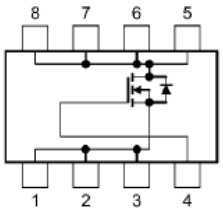
Feature

- V_{DSS}=80 V, I_D=18 A
- High-speed switching
- Low-gate input-charge: Q_{SW} = 6.7 nC (normal)
- Low on-resistance: $R_{DS(ON)}$= 10.8 mΩ (normal) (V_{GS}=10 V)
- Low leakage current: I_{DSS} = 10 μA (max) (V_{DS}=80 V)
- Easy-to-handle enhancement type: V_{th} = 2.0 to 4.0 V (V_{DS} = 10 V, I_D = 0.2 mA)

Appearance and Terminal Arrangement



TSON Advance



1, 2, 3: Source
4: Gate
5, 6, 7, 8: Drain

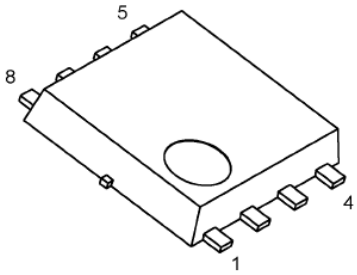
Width 3.3 × length 3.3 × height 0.85 (mm)

4.2. TPH2R003PL

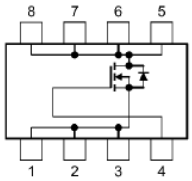
Feature

- V_{DSS}=30 V, I_D=100 A
- High-speed switching
- Low-gate input-charge: Q_{SW} = 22 nC (normal)
- Low on-resistance: $R_{DS(ON)}$=1.3 mΩ (normal) (V_{GS}=10 V)
- Low leakage current: I_{DSS} = 10 μA (max) (V_{DS}=30 V)
- Easy-to-handle enhancement type: V_{th} = 1.1 to 2.1 V (V_{DS} = 10 V, I_D = 0.5 mA)

Appearance and Terminal Arrangement



SOP Advance



1, 2, 3: Source
4: Gate
5, 6, 7, 8: Drain

Width 5.0 × length 6.0 × height 0.95 (mm)

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

Variable name	Unit	Description
Vin	V	Input voltage
Vout	V	Output voltage
DCR1	Ω	Power plane parasitic resistance value on the primary side
DCR2	Ω	Primary GND plane parasitic resistance
Fc	Hz	Switching frequencies of the secondary MOSFET
Rdrv_on_p	Ω	Primary MOSFET Gate driver internal resistance (on side)
Rdrv_off_p	Ω	Primary MOSFET Gate driver internal resistance (off side)
Rdrv_on_s	Ω	Secondary MOSFET Gate driver internal resistance (on side)
Rdrv_off_s	Ω	Secondary MOSFET Gate driver internal resistance (off side)
Vdrv_H_p	V	Power supply voltage of the primary side gate driver
Vdrv_H_s	V	Power supply voltage of the secondary side gate driver

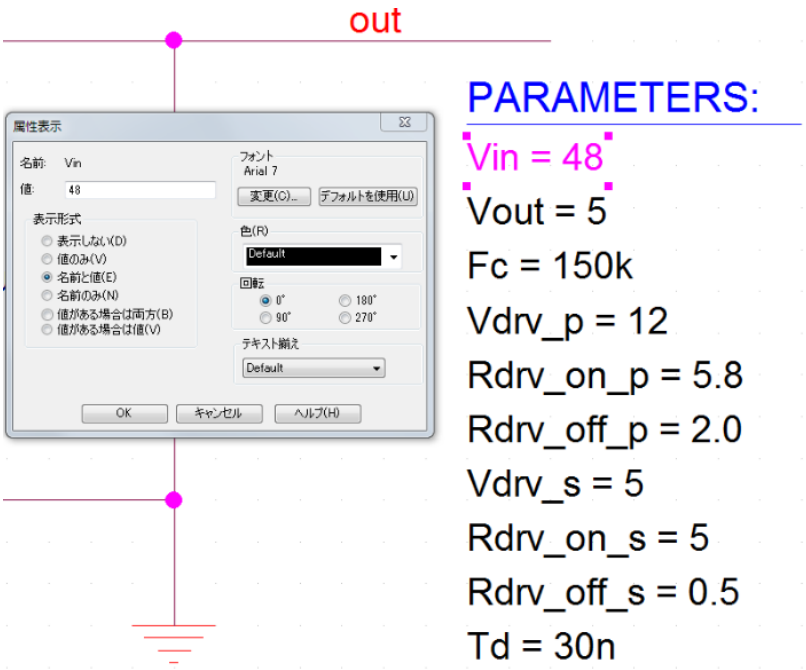


Fig. 5.1 Parameter Setting Screen

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**.

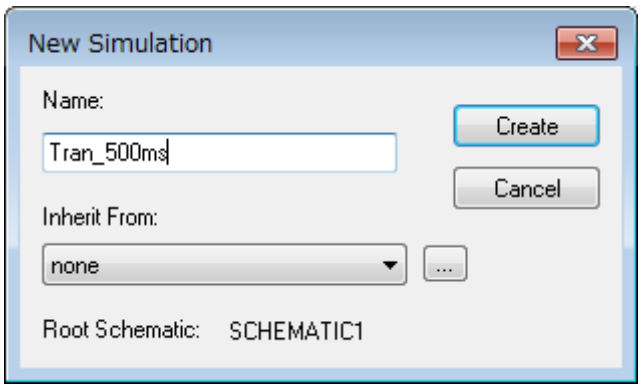


Fig. 5.2 New Simulation dialog box

(2) Then, the Simulation Settings dialog box shown in Fig. 5.3 appears. In this dialog box, you can set parameters for various types of analysis. First, click the **Analysis** tab. Select **Time Domain (Transient)** from the **Analysis Type** drop-down list. Enter the simulation end time in the **Run To Time** field and the maximum step size in the **Maximum Step Size** field.

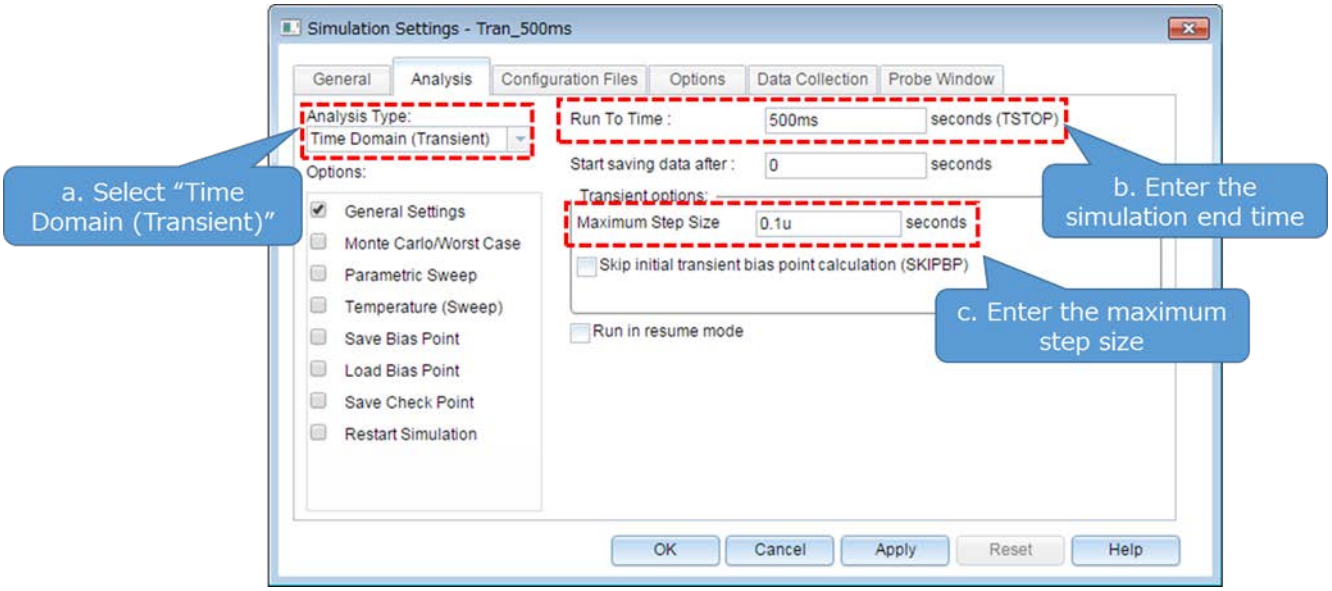


Fig. 5.3 Simulation Settings - Analysis dialog box

(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.

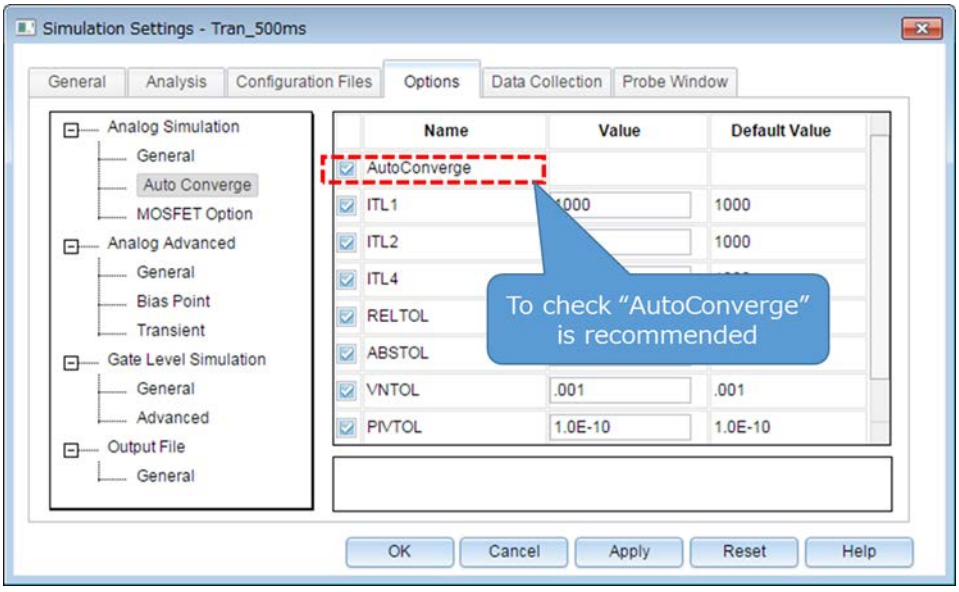


Fig. 5.4 Simulation Settings - Options window

(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.

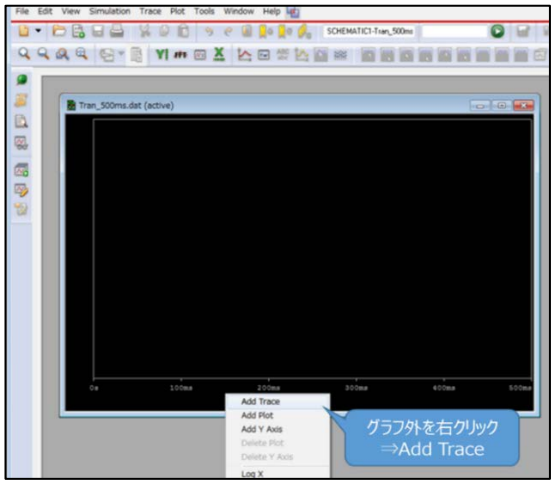


Fig. 5.5 Graph Window

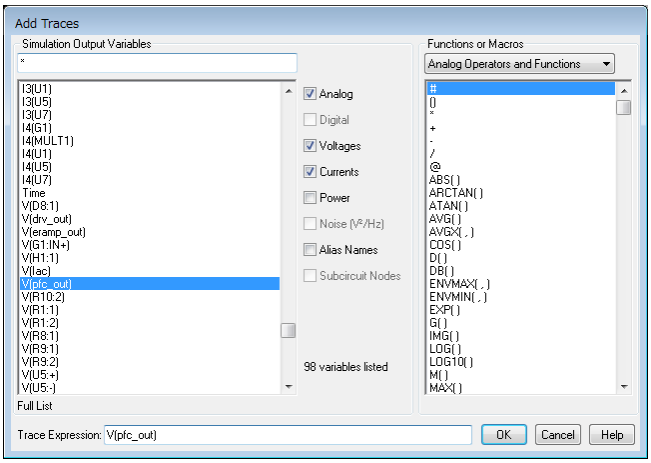


Fig. 5.6 "Add Traces" window

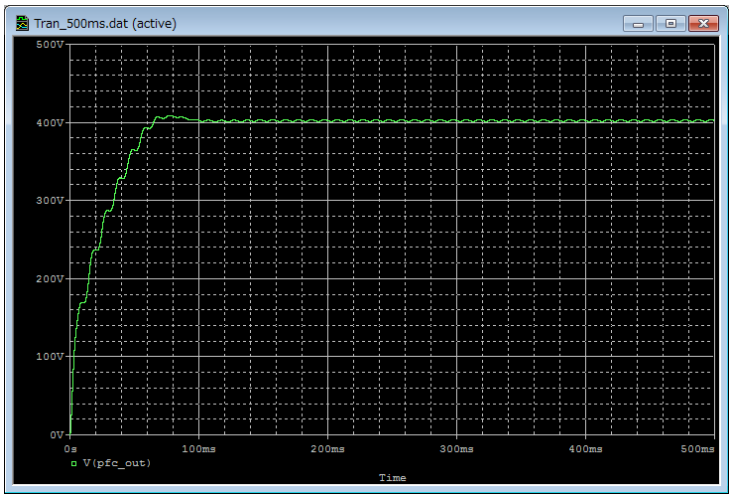


Fig. 5.7 Simulation waveform view (Example: Output voltage waveform)

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.

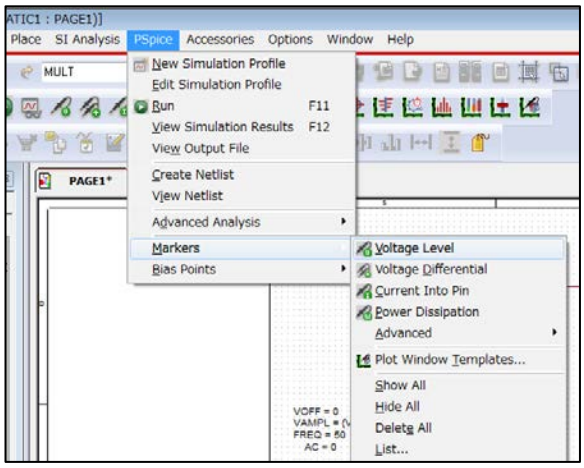


Fig. 5.8 Selecting a marker type

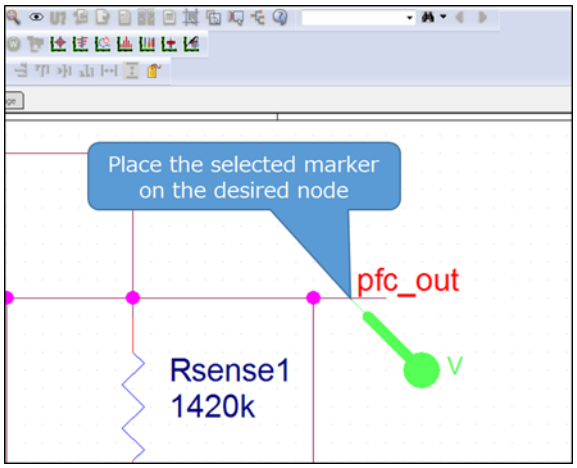


Fig. 5.9 Placing a marker in the circuit

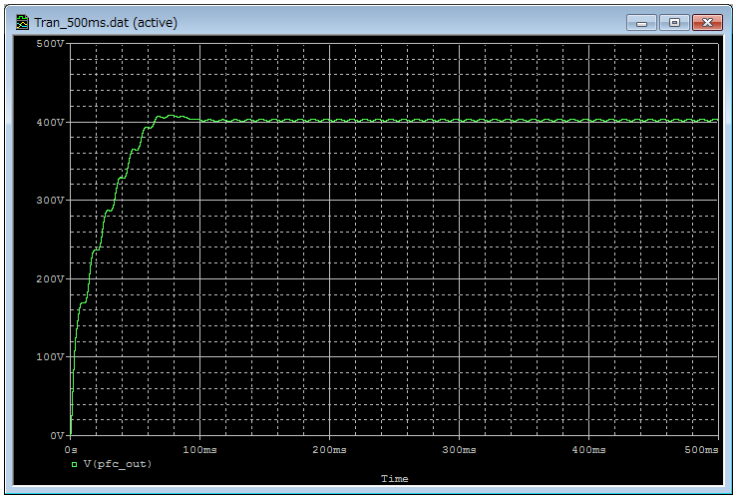


Fig. 5.10 Simulation waveform view (Example: Output voltage waveform)

※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.

Term of service

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.