Half Bridge (HB) AC-DC Power Supply Basic Simulation Circuit

Reference Guide

RD173-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-DC power supply that converts AC voltage to DC voltage. The AC-DC power supply may be installed outside the equipment, or it may be installed inside the equipment and its presence may not be known from outside.

There are two main types of AC-DC power supply: the transformer-based and the switched-based.

In the transformer-based, the AC voltage is stepped down by a transformer at a commercial frequency, the negative voltage portion of the AC voltage is converted to a positive voltage by a diode bridge (rectifier bridge), and the voltage is smoothed by using a capacitor, thereby producing a DC voltage. Because this method transfers energy at a commercial frequency (50 Hz or 60 Hz), the energy stored per cycle is large, a very large transformer and capacitor is needed, and the entire AC-DC power supply is large and heavy.

On the other hand, in the switched-based, a full-wave rectification is performed by a diode bridge without stepping down a commercial AC voltage, and then the DC voltage is smoothed by a capacitor. This smoothed DC voltage is switched by the switching element at a frequency of several tens to several hundreds kHz, which is much higher than the commercial frequency, and is applied to the primary side of the transformer. A voltage in proportion to the winding ratio is transmitted to the secondary side of the transformer. The final DC voltage is produced by rectifying the AC voltage transmitted to the secondary side with a diode and smoothing it with an output capacitor. The switched-based can reduce the amount of energy stored per switching cycle by increasing the switching frequency, so that small and light transformers and capacitors can be used. Therefore, the AC-DC power supply can be reduced in size and weight.

In addition, the DC voltage of the final stage can be adjusted to a constant voltage by controlling the on/off time in the cycle in which the switching element is switching.

For these reasons, today's switched-based power supplies are the mainstream of AC-DC power supplies.
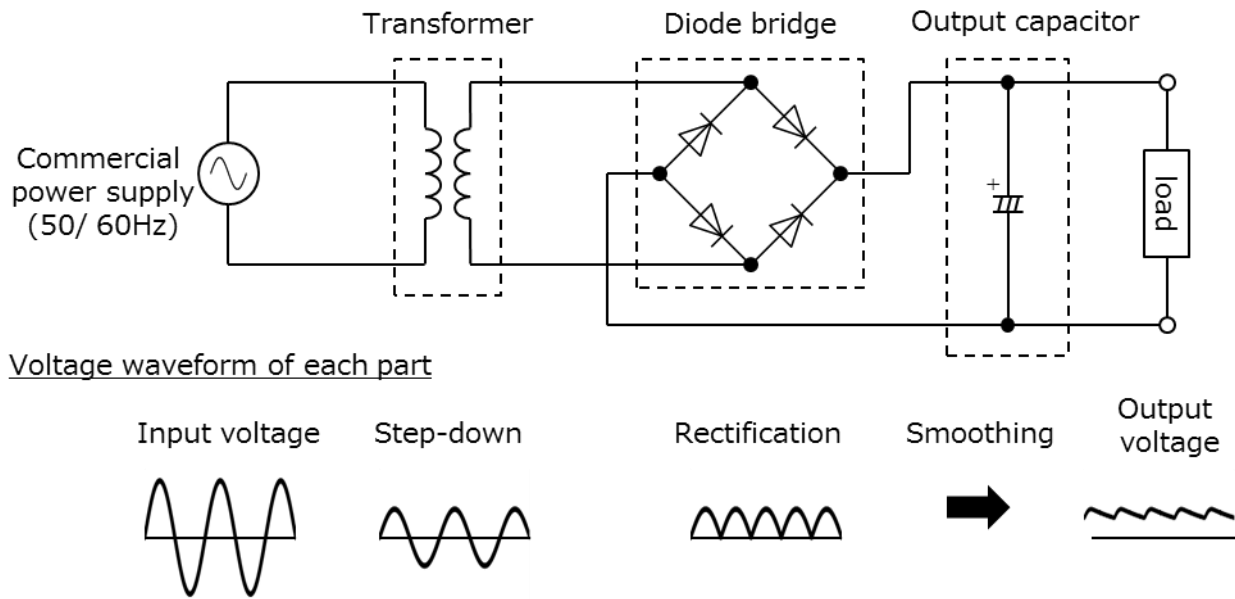
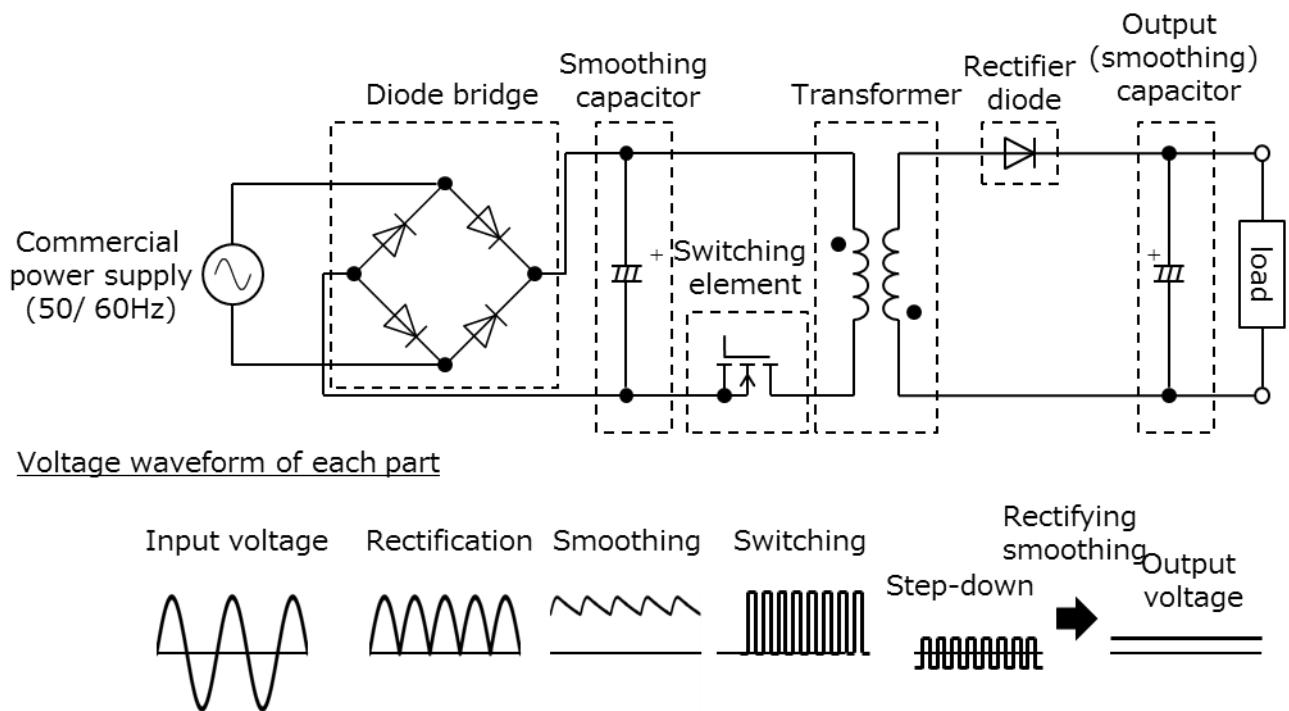**Fig. 1.1 Transformer-based AC-DC Power Supply Configuration**

Fig. 1.2   Switched-based AC-DC Power Supply Configuration

Fig. 1.3 shows exemplary circuit blocks of a switched-based AC-DC power supply. It consists of four blocks: (1) input filters, (2) rectifying bridges, (3) DC-DC converters, and (4) feedback circuits. The function of each block is shown below.
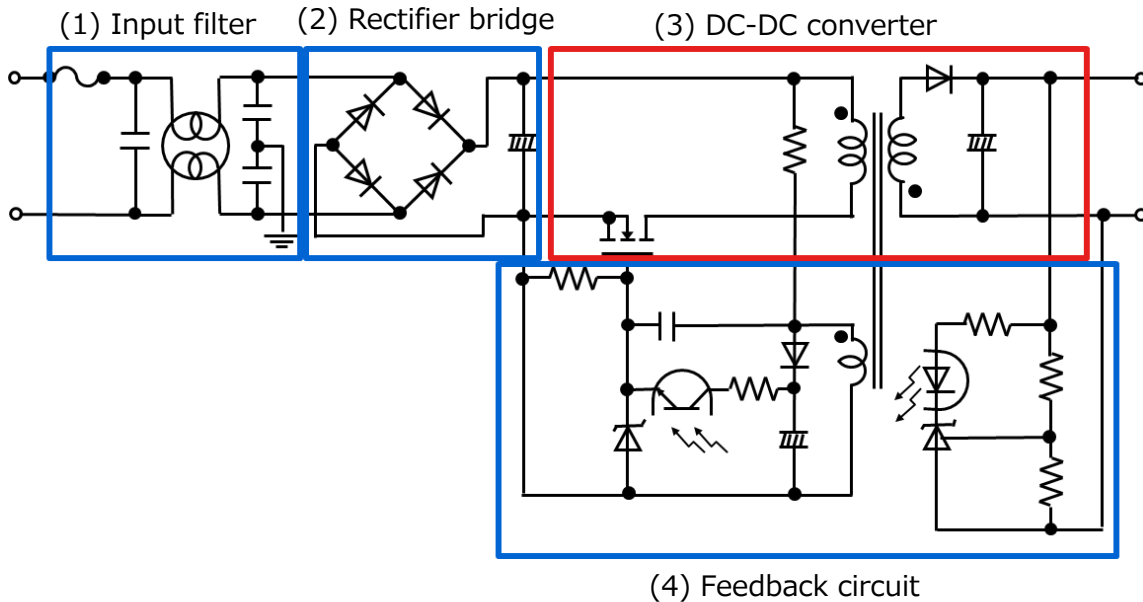
Fig. 1.3   Examples of Switched-based AC-DC Power Supply Circuit Blocks

(1) Input filter

   This prevents noise generated by the switching power supply from propagating to the commercial line.

(2) Rectifier bridge

   The AC voltage is rectified, and the DC voltage is transmitted to the DC-DC converter at the subsequent stage. The configuration consisting of a rectifier bridge and a capacitor as shown in Fig. 1.1 degrades the power factor.

   In recent years, the method of converting to DC voltage through a circuit capable of power factor correction (PFC: Power Factor Correction) is the mainstream.
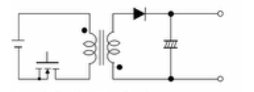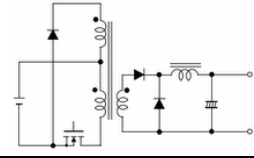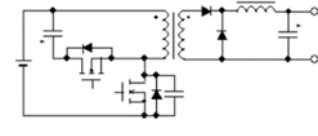
(3) DC-DC converter
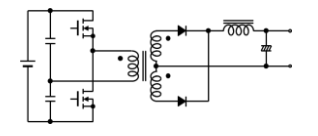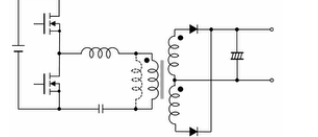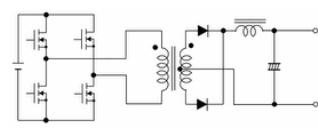
   Converts the voltage rectified in (2) to an arbitrary DC voltage.

(4) Feedback circuit

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

   Switched-mode AC-DC power supplies rectify an inputted AC voltage and convert the rectified AC voltage to an arbitrary DC voltage via a DC-DC converter. There are a variety of topologies for DC-DC converters. 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	Disadvantage
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	* 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 HB 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 (RD173-SPICE-01) is provided on our website in order to understand the operation of the HB DC-DC converter in the switching AC-DC power supply.

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 AC-DC (DC-DC) converter

The AC-DC power supply contains the DC-DC converter circuit and rectifying bridges or PFCs. The basic simulated circuit (RD173-SPICE-01) is assumed to be a DC-DC converter circuit after the PFC circuit. PFC circuits are disclosed together with the reference design. PFC circuits are referred to below.

The basic simulation circuit including the PFC power supply is as follows. Click Here

2.1. Power specification

The specifications of the HB DC-DC power supply described in this document are as follows.
- Input voltage: 380 V
- Output voltage: 24 V
- Output current: 0 to 12.5 A
- Operating frequency of primar MOSFET: 100 kHz
- Winding ratio: $n_1:n_2:n_3 = 6:1:1$
- Secondary Allowable Ripple Current Width: 30%

2.2. Circuit configuration

Fig. 2.1 shows the simulation circuit for OrCAD®. This is a HB DC-DC converter power supply, which mainly consists of a power unit (HB) 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 uses the TK10A60W and TPH2900ENH as switching MOSFETs.
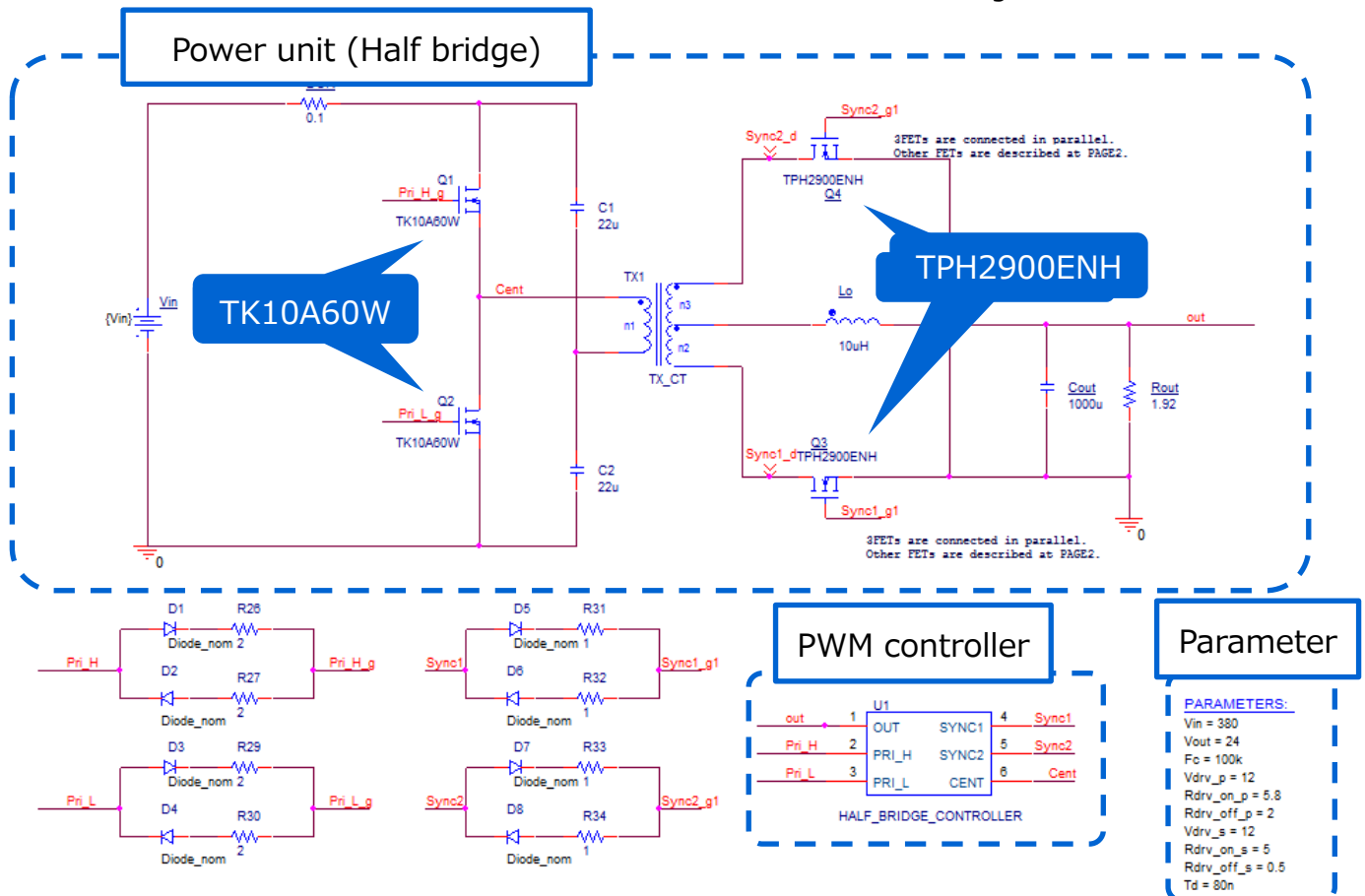

Fig. 2.1 Simulation circuit of a 300 W HB (DC-DC Converter) Power supply

Selection of primary MOSFET

Primary MOSFET (TK10A60W: V_{DSS} = 600 V, I_D = 9.7 A) is selected from the following perspectives.

  (1) Device breakdown-voltage

    The voltage applied to the device at steady state is the input voltage (=380 V). Select a device with a breakdown-voltage of 600 V or higher, considering the surge voltage at the time of switching and other factors.

  (2) Current rating

    The HB AC-DC power supply has the maximum input current when the output power has the maximum value. If the conversion efficiency at maximum output power =300 W is 85%, the maximum average input current will be 0.93 A. Select a device with a current rating of 2 A or higher.

Selection of secondary MOSFET

Secondary MOSFET (TPH2900ENH: V_{DSS} = 200 V, I_D = 33 A) is selected from the following perspectives.

  (1) Device breakdown-voltage

    The primary winding voltage at steady state is 190 V as $V_{in}/2$. The voltage between the middle point and both ends of the secondary winding is 31.7 V, 1/12 of the input voltage, because the winding ratio is 6:1, and 63.3 V, 2 times the voltage, is applied to the secondary MOSFET. Select a device with a breakdown-voltage of 200 V or higher because the surge voltage is generated when the transformer current is switched.

  (2) Current rating

    The HB AC-DC power supply has the maximum input current when the output power has the maximum value. Maximum output current of 12.5 A at maximum output power =300 W is shared by the MOSFET of the two current paths on the secondary side. Therefore, a current of 6.3 A flows per path. Since the current is large and the conduction loss of the MOSFET becomes large, when three elements are arranged in parallel per phase, the mean current per element will be about 2.1 A. Therefore, select a device with a current rating of 4 A or more and a low on-resistance as much as possible with an emphasis on suppressing the conduction loss.

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: η (%)
- ・ Two times the switching frequency of the primary MOSFET: F_c (Hz)
- ・ Maximum output current: I_{out_max} (A)
- ・ Allowable peak-to-peak ripple current: ΔI_{ripple} (%)

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

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

where, the input voltage (V_{in}) =380 V, transformer winding ratio (n_2/n_1) = 1/6, output voltage (V_{out}) =24 V), switching frequency (F_c) =200 kHz, maximum output current (I_{out_max}) =12.5 A, and the allowable peak-to-peak ripple current (ΔI_{ripple}) = 30% according to the power supply specifications of this model, the inductance value (L_O) of the output inductor is calculated as 7.7 μH from the above equation, and therefore, 10 μH is selected 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 and 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.
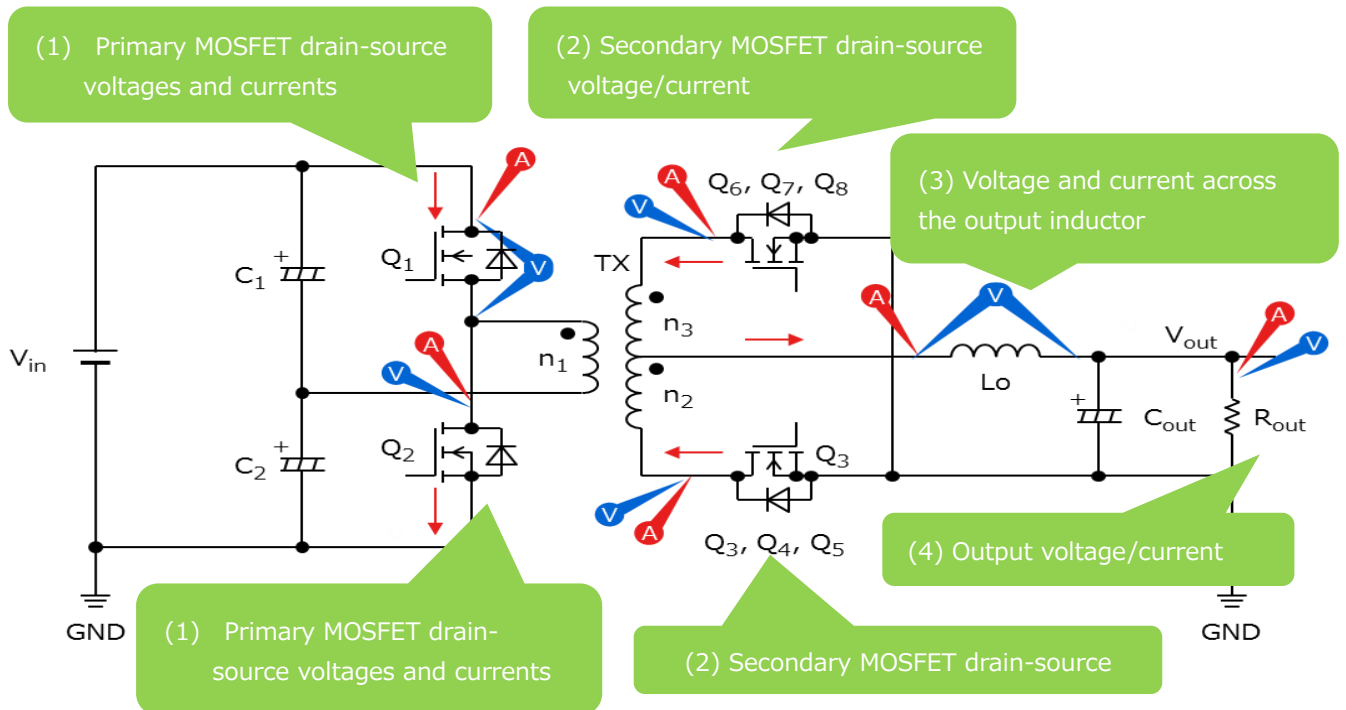

Fig. 3.1   Points at which simulation waveforms are measured

(1) Basic operation of HB system

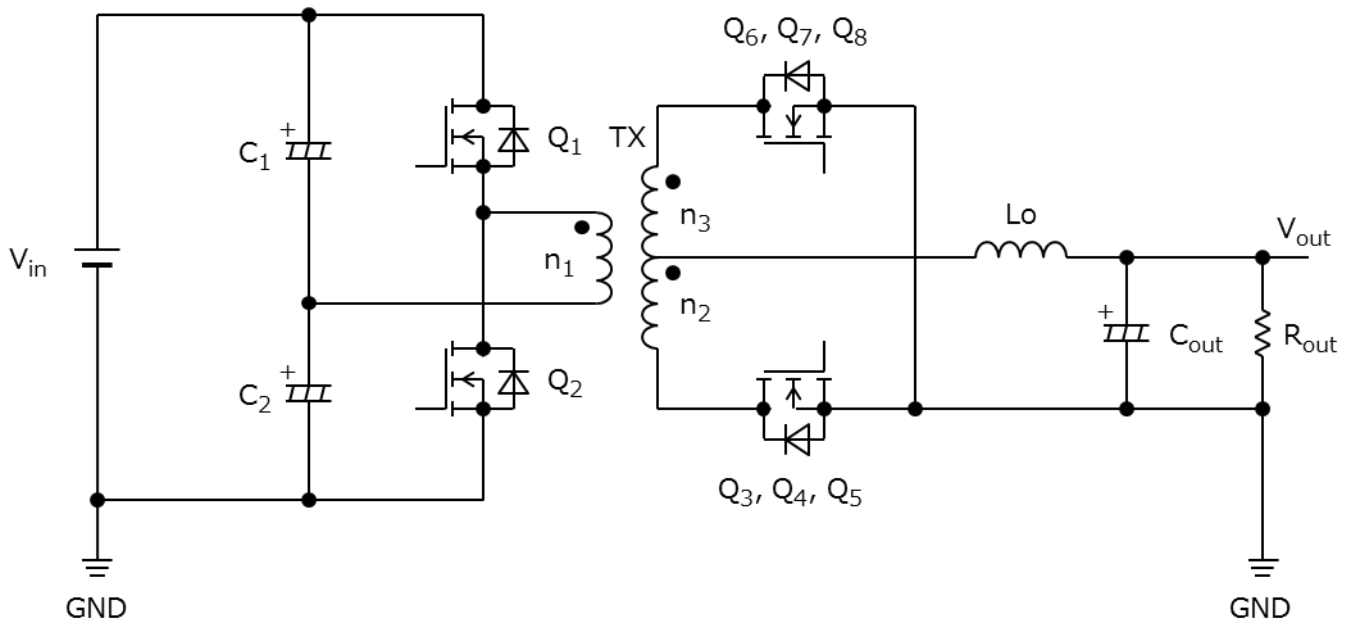Fig. 3.2, which shows the DC-DC converter, explains the basic operation of the HB method.

Fig. 3.2   HB Circuit

Let the primary MOSFET be Q_1 and Q_2, the secondary MOSFET be Q_3 to Q_8, and the power inductor be L_O. In the HB method, Q_1 and Q_2 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.

In the following description, three MOSFETs connected in parallel on the secondary side are described as Q3 and Q6 for simplicity.

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 $V_{in} \rightarrow Q_1 \rightarrow n_1 \rightarrow C_2$ and $C_1 \rightarrow Q_1 \rightarrow n_1 \rightarrow C_2$ 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, LO stores magnetic energy.

b. Q_1 OFF, Q_2 OFF

The MOSFET between the Q_1 and the Q_2 is turned off, and the energy stored in the L_O is circulation through the Q_3 and the Q_6. To the L_O 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 $V_{in} \to C_1 \to n_1 \to Q_2$ and $C_2 \to n_1 \to Q_2 \to C_2$ 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.

$$\Delta i_{Lo(Q2\_on)} = \frac{1}{L_O} \times \left(\frac{n_3}{n_1} \times \frac{V_{in}}{2} - V_{out} \right) \times T_{on}$$

d. Q_1 OFF, Q_2 OFF

Like b, the energy stored in the L_O passes through Q_3 and Q_6 current flows.

$$\Delta i_{Lo(all_{off})} = \frac{-V_{out}}{L_O} \times \left(\frac{T}{2} - T_{on} \right)$$
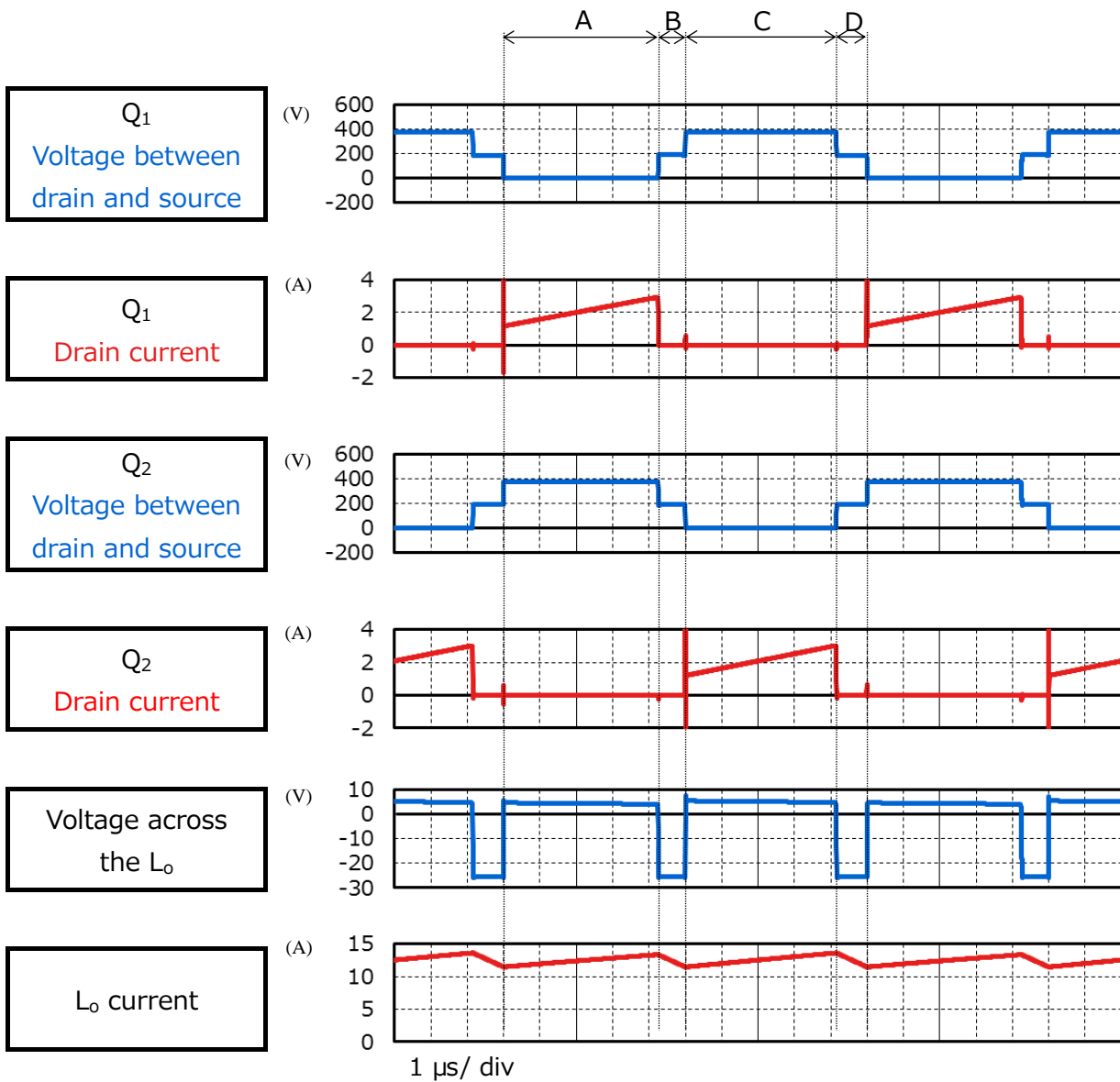
Fig. 3.3 Waveforms of the drain-source voltage of the primary-side MOSFETs and the voltage and current waveforms of the output inductor

(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, the more effective it is in reducing conduction loss. Synchronous rectification circuits are commonly used for applications that require high efficiency and output power.

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

a. Q_3 ON and Q_6 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_6 ON

Energy stored in the L_O is recycled through the Q_3 and Q_6.

c. Q_3 OFF, Q_6 ON

Voltages corresponding to winding ratios with the polarity symbol side of the secondary side winding n_3 set to minus
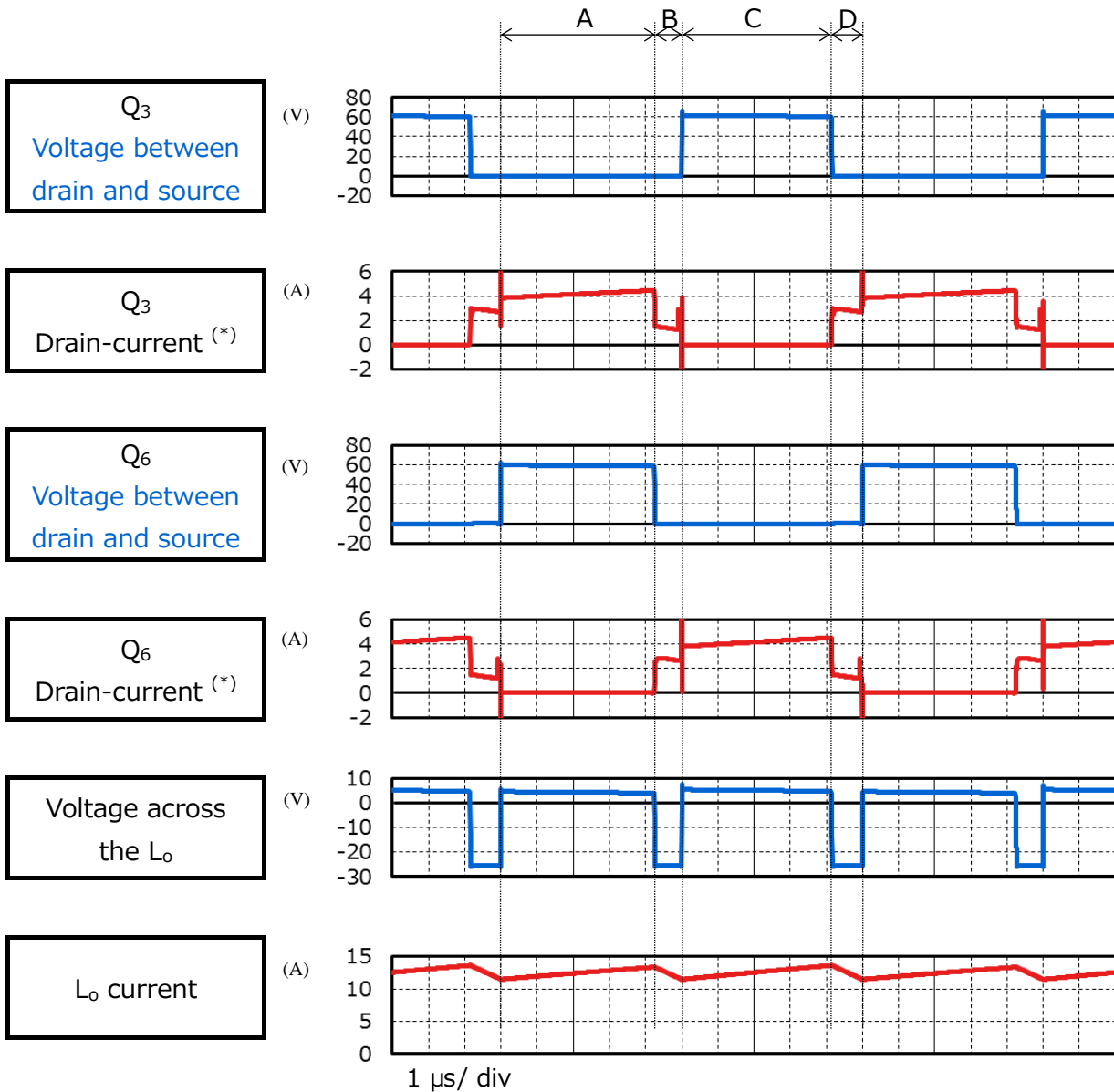
$$\frac{n_3}{n_1} \times \frac{V_{in}}{2}$$

The current flows through the Q_6 to the L_O.

d. Q_3 ON, Q_6 ON

The electric power stored in the L_O is recycled through the Q_3 and the Q_6.

© 2019

(3) Voltage and current across the output inductor

Q_3 Voltage between drain and source	(V)	
Q_3 Drain-current [*]	(A)	
Q_6 Voltage between drain and source	(V)	
Q_6 Drain-current [*]	(A)	
Voltage across the L_o	(V)	
L_o current	(A)	

1 μs/ div

* : The drain current flows from the MOSFET to the secondary winding of the transformer in the positive direction. Three in parallel
Drain current waveforms for one device of the MOSFET are shown.

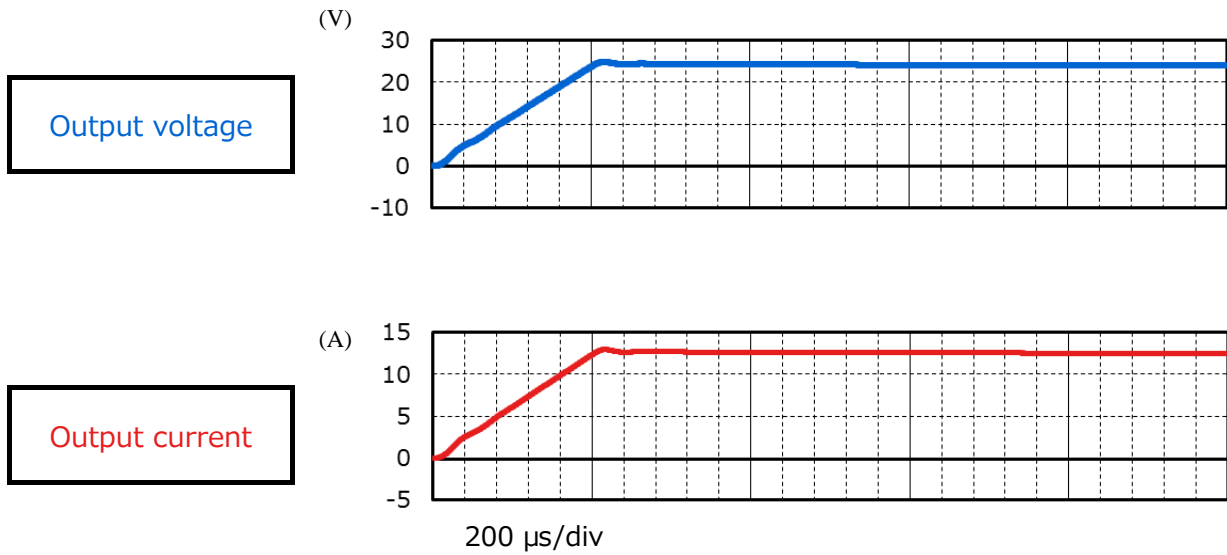**Fig. 3.4   Voltage between Drain-Source of Secondary MOSFET, Current Output Inductor Voltage and Current Waveforms**

4. Product Overview

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

4.1. TK10A60W

Feature

- V_{DSS}=600 V, I_D=9.7 A
- Low on-resistance by adopting super junction structural DTMOS: $R_{DS(ON)}$=0.327 Ω (normal)
- Optimized gate switching speed
- Easy-to-handle enhancement type: V_{th} = 2.7 to 3.7 V (V_{DS}=10 V, I_D=0.5 mA)

### Appearance and Terminal Arrangement

1: Gate
2: Drain
3: Source

TO-220SIS
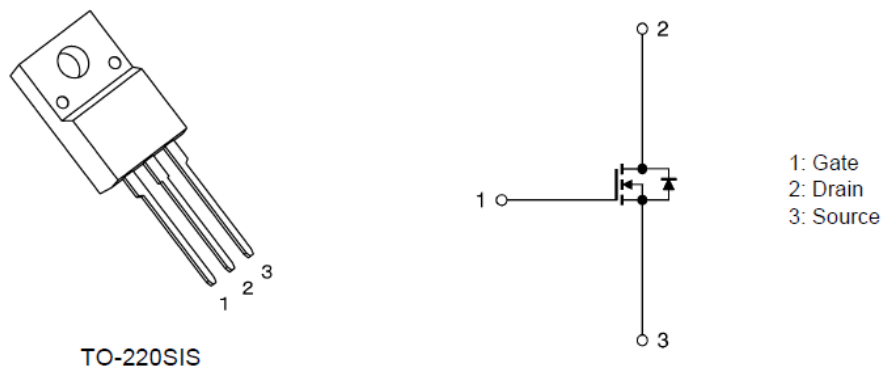
4.2. TPH2900ENH

Feature

- V_{DSS}=200 V, I_D=33 A
- High-speed switching
- Low-gate input-charge: Q_{SW} = 8.2 nC (normal)
- Low on-resistance: $R_{DS(ON)}$=24 mΩ (normal) (V_{GS}=10 V)
- Low leakage current: I_{DSS} = 10 µA (max) (V_{DS}=200 V)
- Easy-to-handle enhancement type: V_{th} = 2.0 to 4.0 V (V_{DS}=10 V, I_D=1.0 mA)

Appearance and Terminal Arrangement

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

SOP Advance
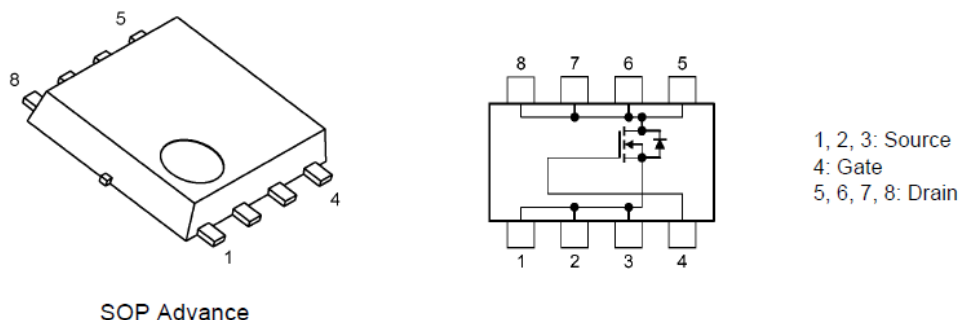
Width 5.0 × length 6.0 × height 0.95

(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

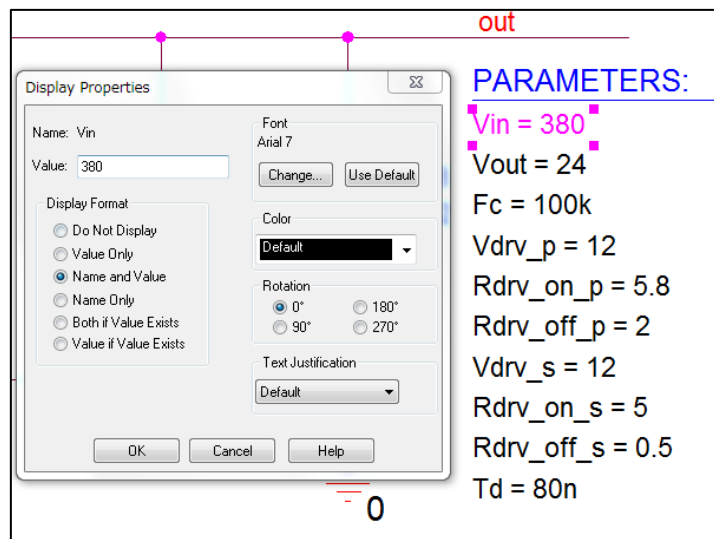| Parameter name	Unit	Description
Vin	V	Input voltage
Vout	V	Output voltage
Fc	Hz	Switching frequencies of the primary MOSFET
Rdrv_on_p	Ω	Primary MOSFET Gate driver internal resistance (on side)
Rdrv_off_p	Ω	Primary MOSFET Gate driver internal resistance (off side)
Rdrv_on_s	Ω	Secondary MOSFET Gate driver 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
Td	Ns	Dead time setting of the secondary MOSFET

Fig. 5.1 Parameter Setting Screen

PARAMETERS:
Vin = 380
Vout = 24
Fc = 100k
Vdrv_p = 12
Rdrv_on_p = 5.8
Rdrv_off_p = 2
Vdrv_s = 12
Rdrv_on_s = 5
Rdrv_off_s = 0.5
Td = 80n

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**.
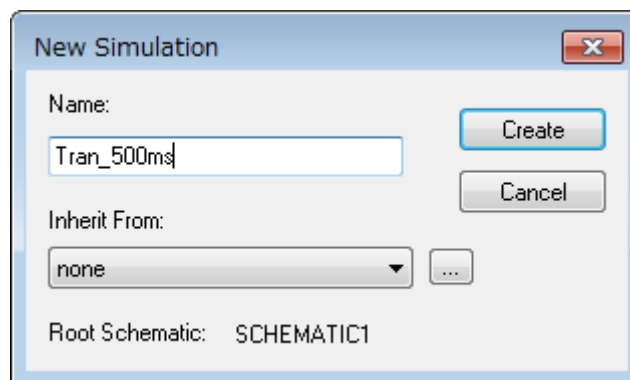

Fig. 5.2 New Simulation dialog box

(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.
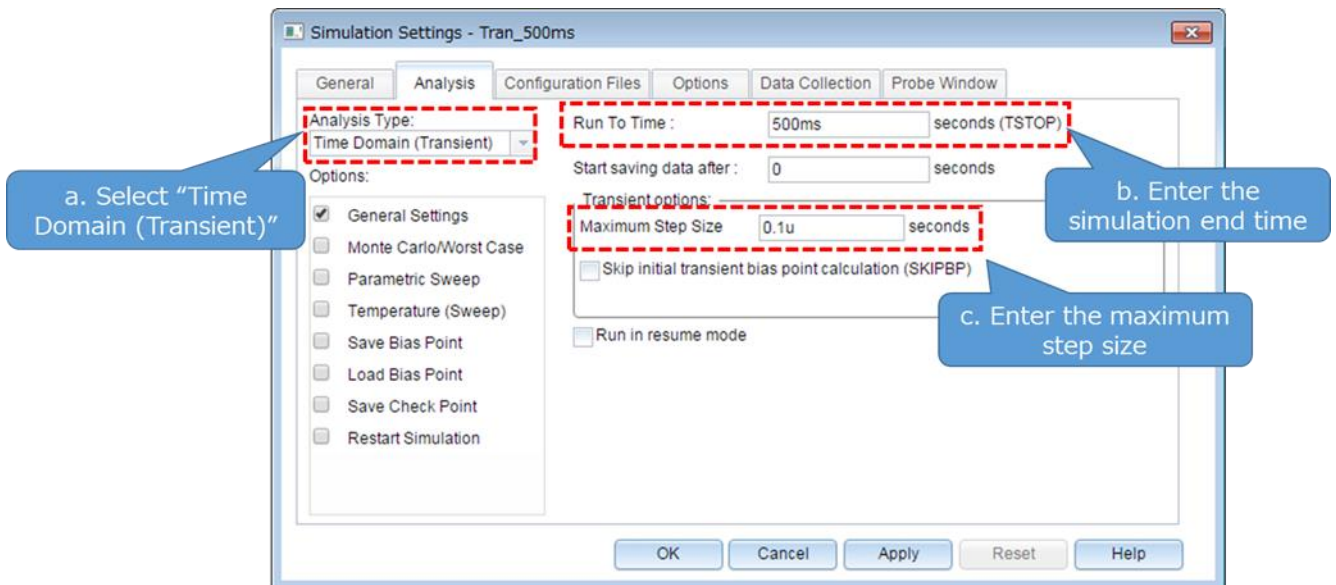

Fig. 5.3 Simulation Settings - Analysis dialog box

(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.
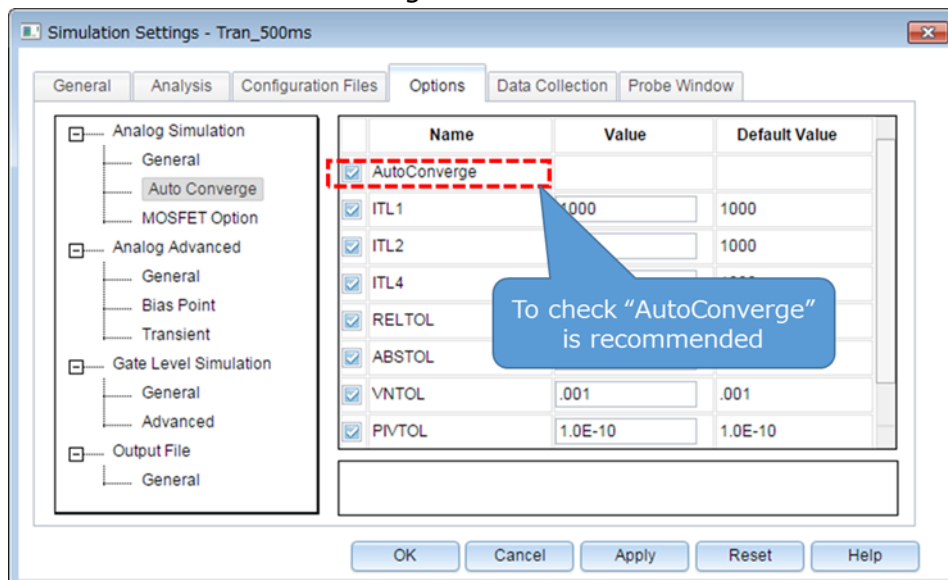

Fig. 5.4 Simulation Settings - Options window

(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.

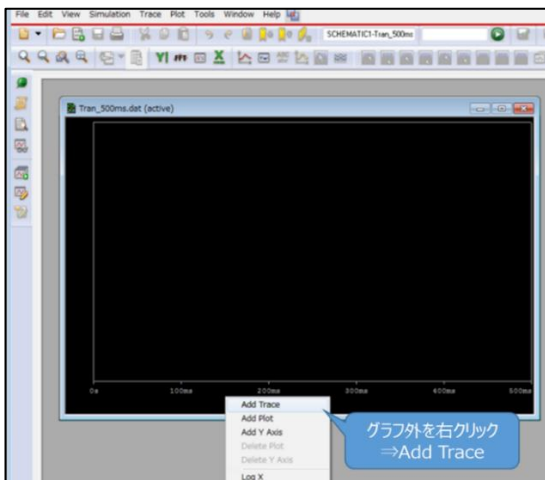(3) Click **OK**. Then, the selected waveform appears as shown in Fig. 5.7.
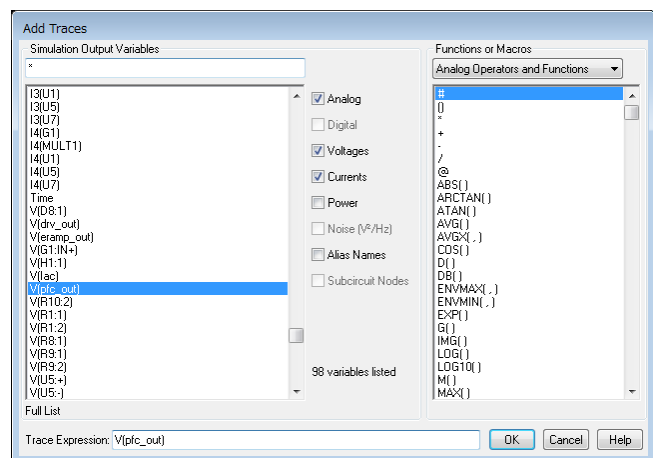

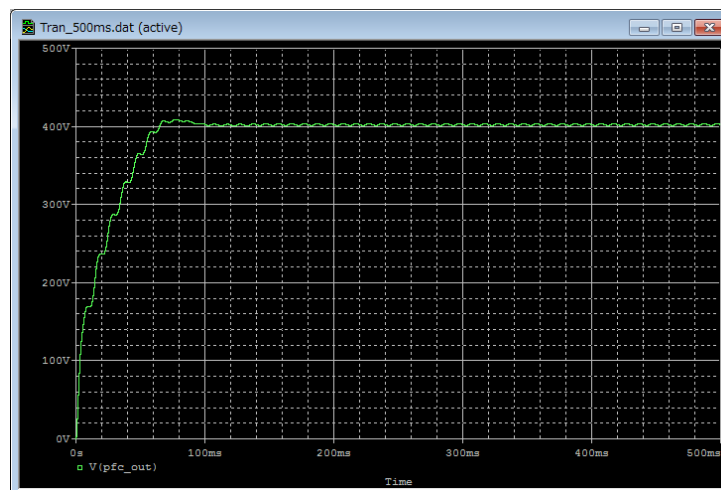**Fig. 5.5 Graph Window**

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.

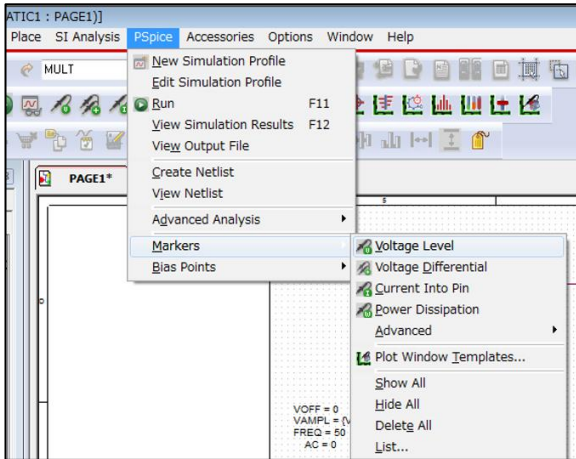(3) Then, its waveform appears in the graph window of PSpice® A/D as shown in Fig. 5.10.

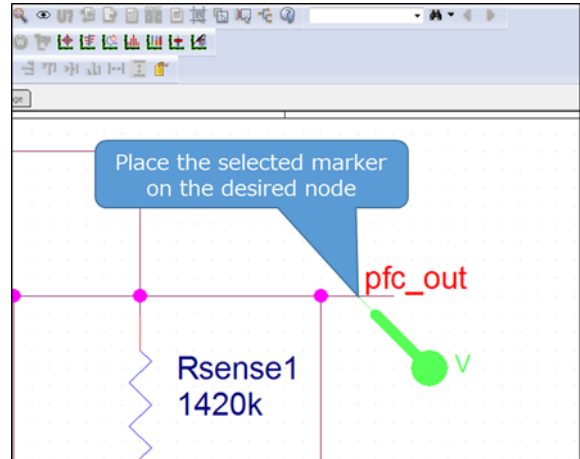**Fig. 5.8 Selecting a marker type**

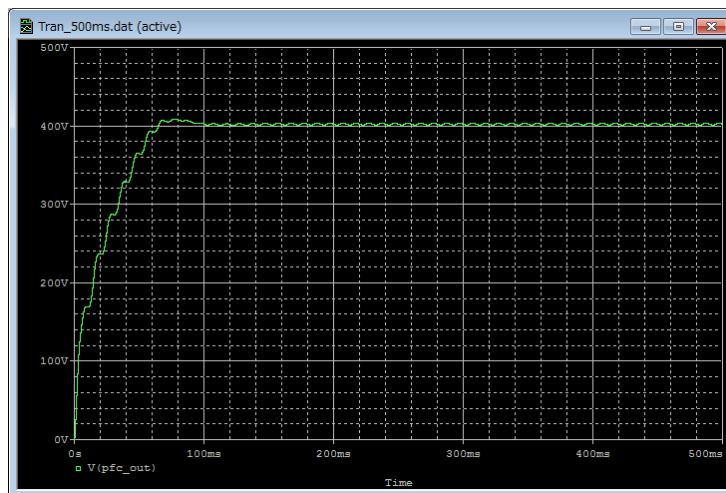**Fig. 5.9 Placing a marker in the circuit**

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.

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.