

# **Full-Bridge AC-DC Power Supply Basic Simulation Circuit Reference Guide**

**RD153-RGUIDE-01**

---

**TOSHIBA ELECTRONIC DEVICES & STORAGE CORPORATION**

## Table of Contents

<b>1. Introduction</b> .....	<b>3</b>
<b>2. Outline of Full-Bridge AC-DC Power Supply (DC-DC Converter Section)</b> .....	<b>7</b>
2.1. Power Supply Specifications .....	7
2.2. Circuit Configuration .....	8
<b>3. Simulation Result</b> .....	<b>11</b>
<b>4. Product Overview</b> .....	<b>18</b>
4.1. TK20N60W .....	18
4.2. TPH5200FNH .....	18
<b>5. Using the Simulation Circuit</b> .....	<b>19</b>

## 1. Introduction

Most electrical equipments, such as 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 so it is necessary to convert AC voltage to DC voltage in order to these equipment's.

The AC-DC power supply 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: One is transformer-base and the other is switch-base.

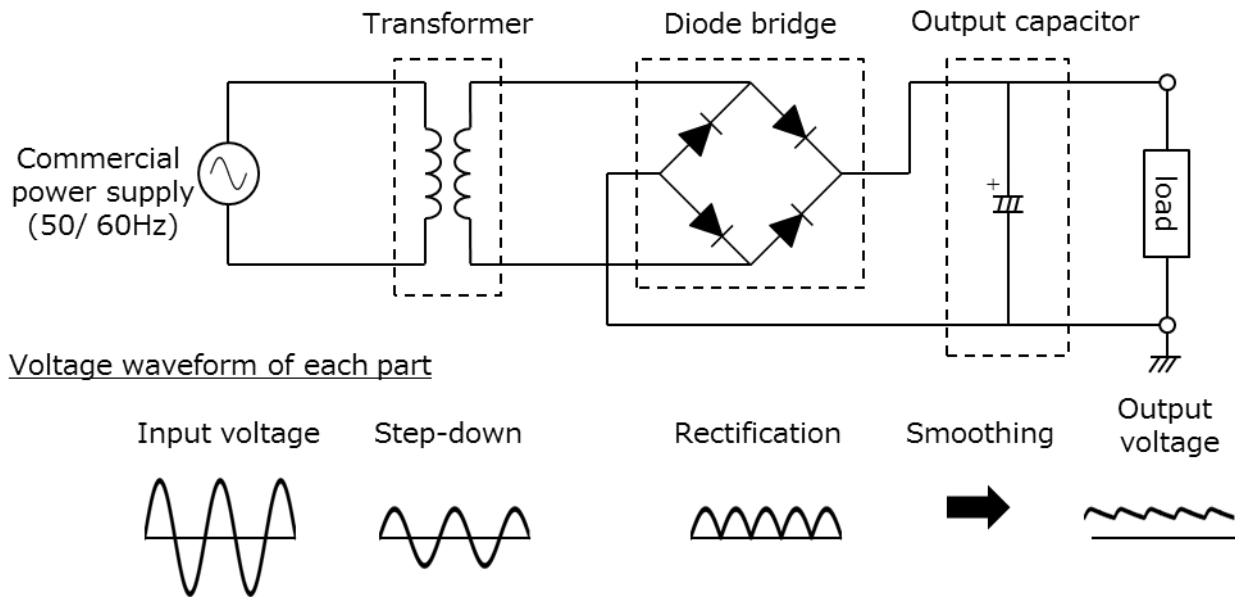
In the transformer-base, the AC voltage is stepped down by a transformer at a commercial frequency, the negative voltage portion of the AC voltage is converted to a positive voltage by a diode bridge (rectifier bridge), and the voltage is smoothed by using a capacitor, then generating producing a DC 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 switch-base, 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 generated by rectifying the AC voltage transmitted to the secondary side with a diode and smoothing it with an output capacitor.

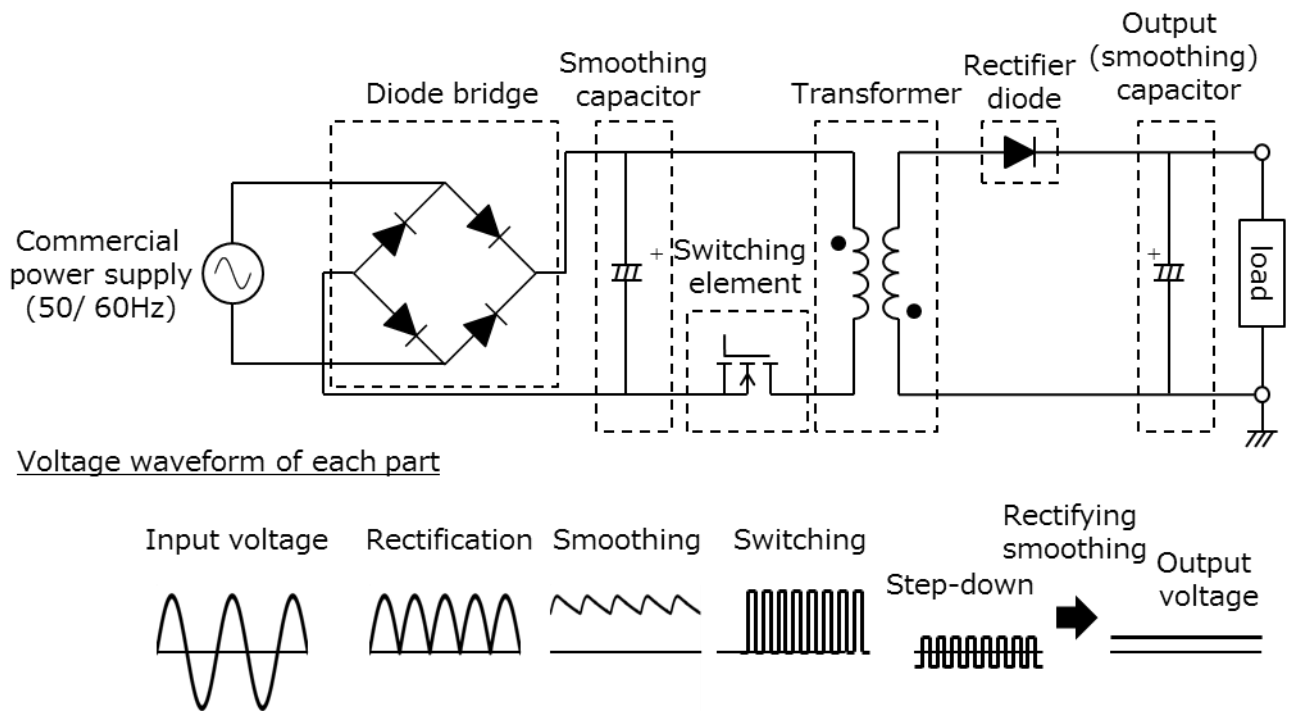
The switch-base can reduce the amount of energy stored per switching cycle by increasing the switching frequency, so that small and light transformers and capacitors can be used. Therefore, the AC-DC power supply can be reduced in size and weight.

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 switch-base power supplies are the mainstream of AC-DC power supplies.

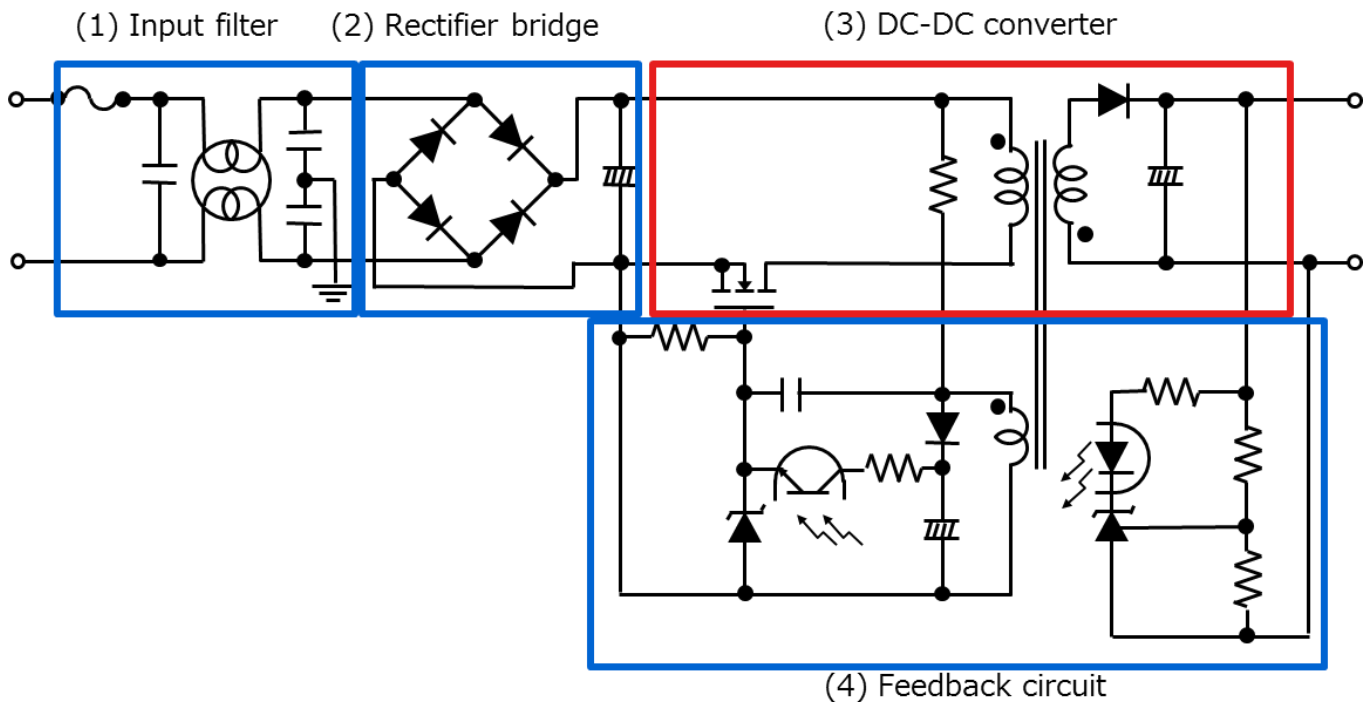


**Fig. 1.1 Transformer-Base AC-DC Power Supply Configuration**



**Fig. 1.2 Switch-Base AC-DC Power Supply Configuration**

Fig. 1.3 shows block diagram of a switch-base AC-DC power supply. It consists of four blocks: (1) Input filters, (2) Rectifying bridges, (3) DC-DC converter, and (4) Feedback circuit. The function of each block is shown below.



**Fig. 1.3 Block Diagram of Switching Type AC-DC Power Supply**

(1) Input filter

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

(2) Rectifier bridge

The AC voltage is rectified, and the DC voltage is transmitted to the DC-DC converter.

As shown in the figure, in the configuration with only a rectifier bridge and a capacitor, the power factor deteriorates.

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

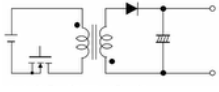
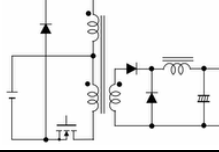
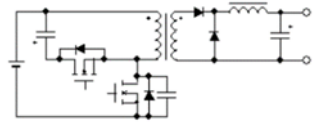
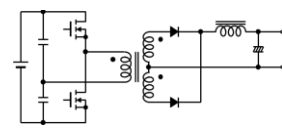
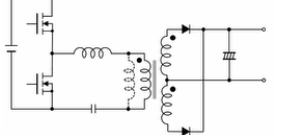
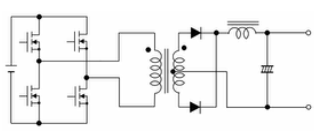
Converts the voltage rectified in (2) to an arbitrary DC voltage.

(4) Feedback circuit

The switching MOSFET is controlled so that the output voltage becomes a desired value.

Switch-base 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 1.1 Typical Topologies and Characteristics of DC-DC Converters**

Circuitry of the DC-DC converter section	Power level	Advantages	Disadvantages
Flyback 	< 200 W	<ul style="list-style-type: none"> <li>• Small part count</li> </ul>	<ul style="list-style-type: none"> <li>• Reduction in efficiency at high power</li> <li>• Large transformer</li> </ul>
Forward 	50 W~500 W	<ul style="list-style-type: none"> <li>• Higher efficiency than a flyback circuit</li> </ul>	<ul style="list-style-type: none"> <li>• Transformer reset circuit required</li> </ul>
Active Clamp Forward 	50 W~500 W	<ul style="list-style-type: none"> <li>• Higher efficiency than a forward</li> </ul>	<ul style="list-style-type: none"> <li>• There are many parts.</li> <li>• Be difficult to control</li> </ul>
Half-Bridge 	100 W~1 kW	<ul style="list-style-type: none"> <li>• High efficiency</li> <li>• Low noise</li> </ul>	<ul style="list-style-type: none"> <li>• Specially designed transformer is required.</li> <li>• Be difficult to control</li> </ul>
Resonant Half-Bridge (LLC Resonance) 	100 W~1 kW	<ul style="list-style-type: none"> <li>• Higher efficiency than a half bridge</li> <li>• Low noise</li> </ul>	<ul style="list-style-type: none"> <li>• Specially designed transformer is required.</li> <li>• Be difficult to control</li> </ul>
Full-bridge 	> 200 W	<ul style="list-style-type: none"> <li>• High efficiency</li> <li>• Capable of increasing the power capacity</li> </ul>	<ul style="list-style-type: none"> <li>• There are many parts.</li> <li>• Be difficult to control</li> </ul>

Regarding the basic operation of the full-bridge method, we are distributing it on our web. Please also refer to the video below.

The video of the full-bridge DC-DC converter are shown here →

[Click Here](#)

The full-bridge method described in this document is widely used in power supplies requiring high efficiency and high power density, because the power supply input voltage applies the primary side of the transformer directly due to cross coupled two switching elements of four switching elements located on the primary side of the transformer turn on/off at the same timing.

We provide basic simulation circuits (RD153-SPICE-01) on our web to understand full-bridge DC-DC converter circuit operation in switch-base AC-DC power supplies.

This document provides an overview of this basic simulation circuit and explains how to use it. The Cadence's Capture and PSpice<sup>®</sup> A/D tools are required to operate the simulator circuits from OrCAD. Simulation circuits and documentation have been prepared in accordance with OrCAD 17.2.

## 2. Outline of Full-Bridge AC-DC Power Supply (DC-DC Converter Section)

The Basic Simulation Circuit (RD153-SPICE-01) is a full-bridge AC-DC power supply with 1 kW output, and assumes a circuit (DC-DC converter circuit) after AC voltage input is converted to DC voltage through rectifier bridges and PFC circuits, etc. For the PFC circuit, the basic simulation circuit is published together with the reference design. Please refer to the following.

The basic simulator including the PFC power supply is →

[Click Here](#)

### 2.1. Power Supply Specifications

The full-bridge DC-DC power supply specifications described in this document are as follows:

- Input voltage ( $V_{in}$ ): 380 V
- Output voltage ( $V_{out}$ ): 48 V
- Output current ( $I_{out}$ ): 0 to 21 A
- Secondary MOSFET operating frequency: 200 kHz (twice the primary operating frequency)
- Winding ratio:  $n_1:n_2:n_3 = 4:1:1$
- Allowable ripple current width on the secondary side ( $\Delta I_{ripple}$ ): 20 %

### 2.2. Circuit Configuration

Fig. 2.1 shows the simulation circuit for OrCAD<sup>®</sup>. This is a full-bridge type DC-DC converter power supply, which mainly consists of a power section (full-bridge) and a PWM controller section. The secondary side of the power section is a synchronous rectification circuit using MOSFETs. The PWM controller section is a general-purpose controller with a built-in MOSFET gate driver, which is provided to realize PWM circuit. The switching MOSFETs are "TK20N60W" and "TPH5200FNH" as example.

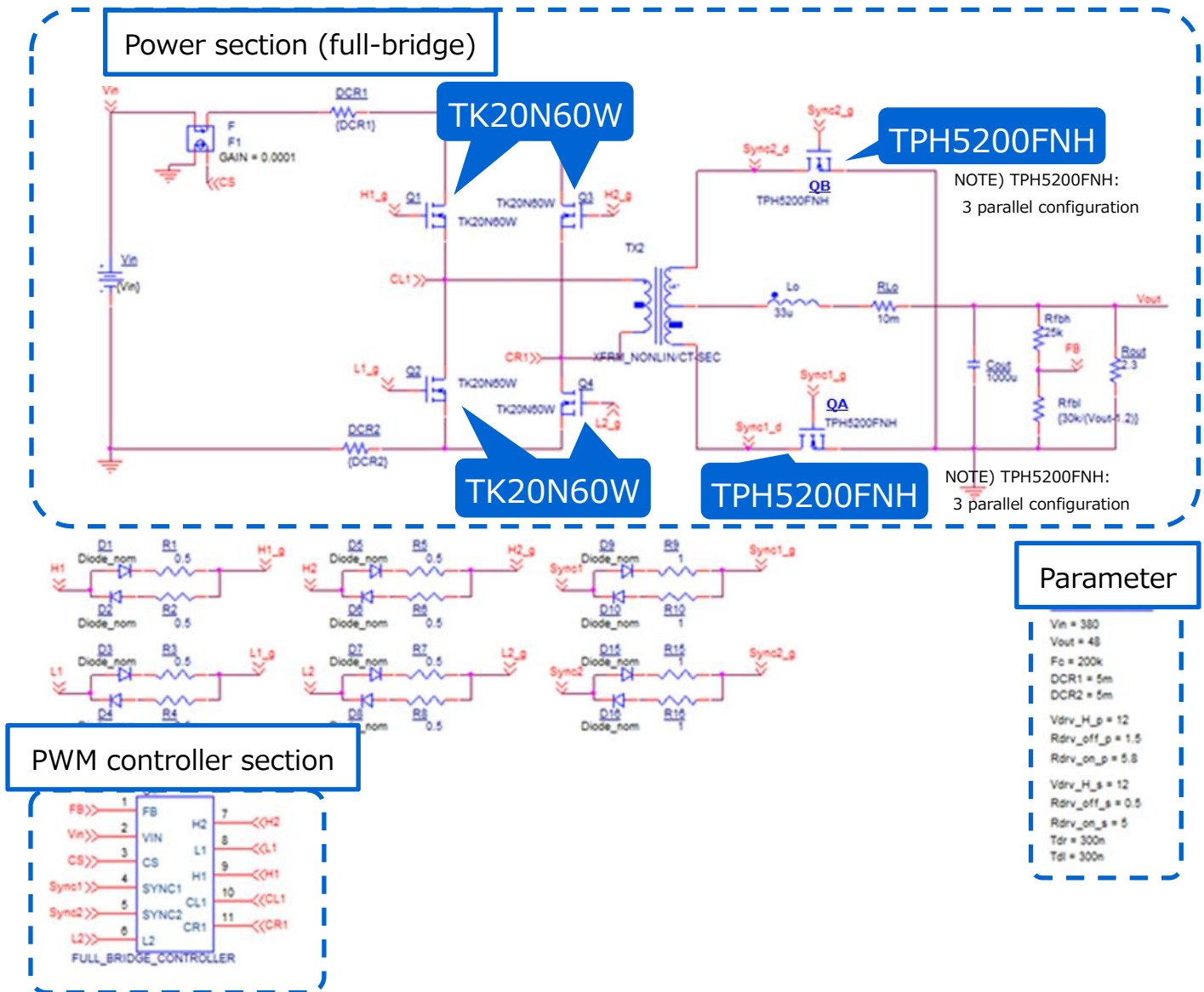


Fig. 2.1 Simulation Circuit of 1 kW Full-Bridge (DC-DC Converter) Power Supply



### Selection of primary side MOSFET

Primary side MOSFET (TK20N60W:  $V_{DSS}=600$  V,  $I_D=20$  A) is selected from the following viewpoints.

(1) Device breakdown-voltage

The voltage applied to the device at static 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

When the input current is maximum, it I at the maximum output power. If the conversion efficiency at the maximum output power (=1 kW) is 90 %, the maximum average input current is 2.9 A. Select a MOSFET with a current rating of 6 A or more.

### Selection of secondary side MOSFET

Secondary side MOSFET (TPH5200FNH:  $V_{DSS}=250$  V,  $I_D=26$  A) is selected from the following viewpoints.

(1) Device breakdown-voltage

Since the winding ratio is 4:1, the voltage between the middle and both ends of the secondary winding at static state is 95 V, which is 1/4 of the input voltage, and 190 V, which is twice the voltage, is applied to each MOSFET on the secondary side.

A surge voltage is generated when the transformer current is switched. Select a device with a breakdown-voltage of 250 V or more.

(2) Current rating

When the output current is maximum, it is at the maximum output power. Maximum output current of 21 A at maximum output power (=1 kW) is shared by the MOSFET of the two current paths on the secondary side, a current of 10.5 A is applied per path. Since the current is large and conduction losses in the MOSFET becomes large, when three elements are arranged in parallel per phase, the average current per element is about 3.5 A. Therefore, a MOSFET with a current rating of 7 A or more is required. In addition, select a MOSFET with as low an on-resistance as possible with emphasis on suppressing 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) = 380
- Transformer winding ratio:  $n_2/n_1 = 1/4$
- Output Voltage:  $V_{out}$  (V) = 48
- Twice the switching frequency of the MOSFET:  $F_c$  (Hz) = 200k
- Max. output current:  $I_{out\_max}$  (A) = 21
- Allowable ripple current range:  $\Delta I_{ripple}$  (%) = 20

The inductance value ( $L_o$ ) of the output inductor is calculated by the following formula.

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

The inductance value ( $L_o$ ) of the output inductor is calculated as 28.3  $\mu\text{H}$  from the above equation, and 33  $\mu\text{H}$  is selected as the setting 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

The operation simulation waveforms of each part in the simulation circuit are shown by the points in Figure 3.1 ((1) to (4)).

- (1) Full-bridge basic operation (Primary MOSFET drain-source voltage)
- (2) Secondary side synchronous rectification operation (Secondary side MOSFET drain-source voltage and current")
- (3) Output inductor voltage and current at both ends
- (4) Output voltage and current as a power supply

When actually using the circuit model, it is possible to display the waveform at any point other than those shown in Fig. 3.1. The waveform display method is described in Chapter 5.

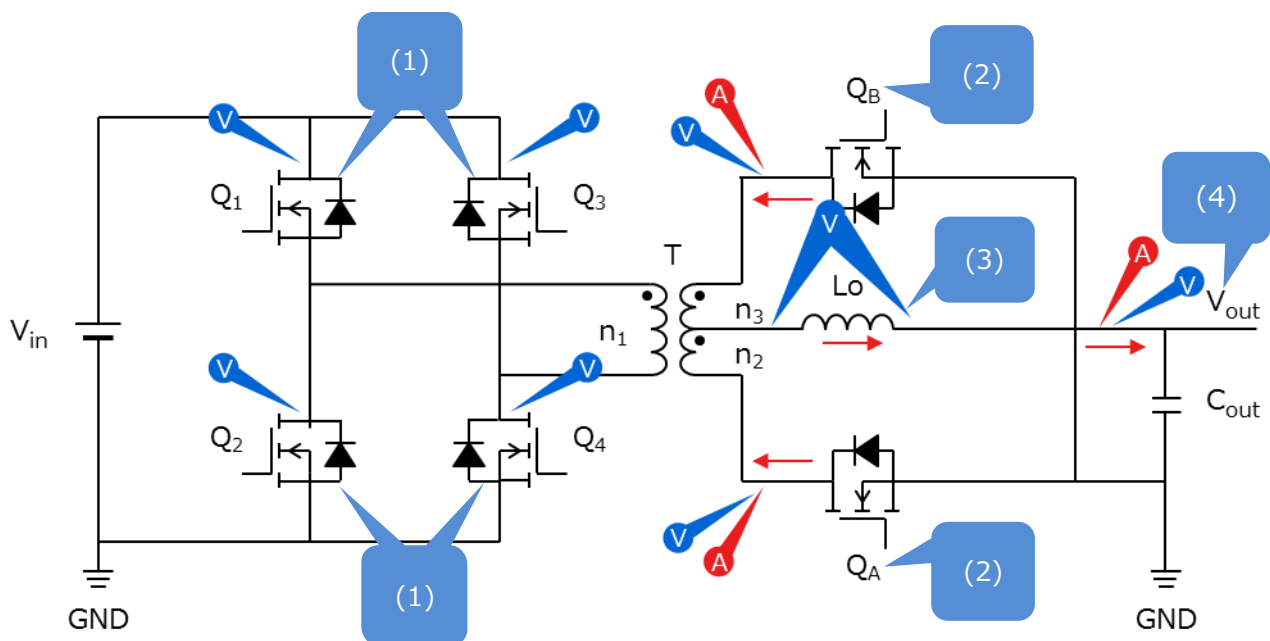
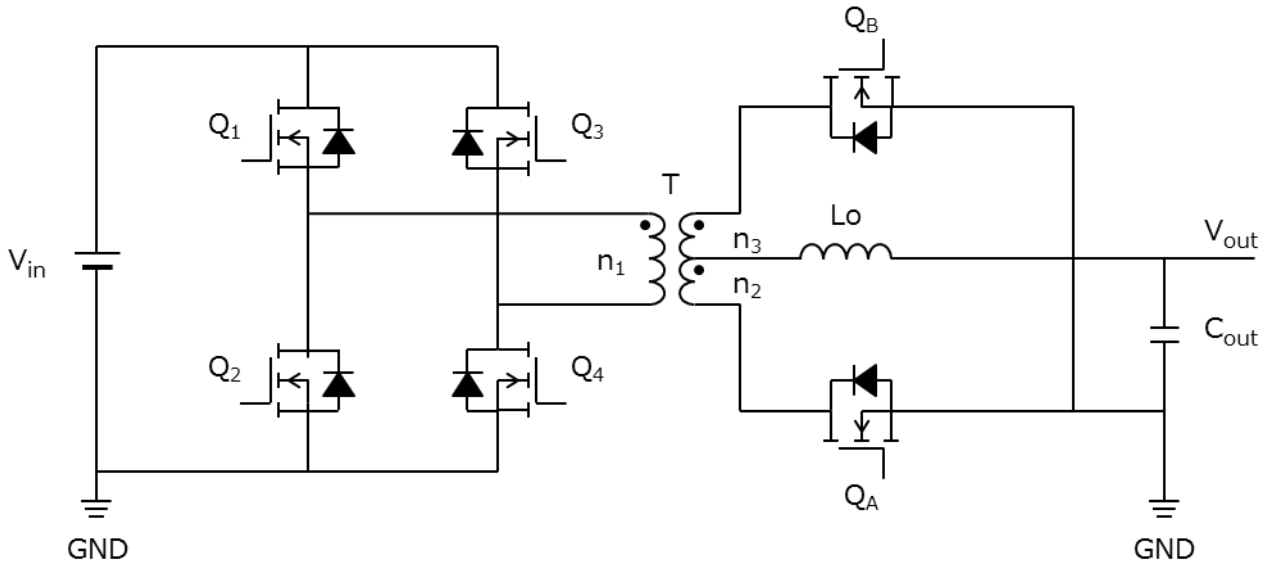


Fig. 3.1 Simulation Waveform Measurement Point List

### (1) Full-bridge basic operation

The basic operation of the full-bridge method is described in Figure 3.2, which shows the full-bridge circuit.



Q<sub>1</sub>~Q<sub>4</sub>: Primary MOSFET  
 Q<sub>A</sub>, Q<sub>B</sub>: Secondary MOSFET  
 L<sub>O</sub>: Output inductor

**Fig. 3.2 Full-Bridge Circuit**

In the full-bridge method, Q<sub>1</sub> and Q<sub>4</sub>, Q<sub>3</sub> and Q<sub>2</sub> are alternately turned on and off in pairs respectively, and the output voltage is controlled using PWM (pulse-width modulation) control.

For the output-voltage V<sub>out</sub>, it can be calculated by:

$$V_{out} = 2 \times \frac{n_2}{n_1} \times V_{in} \times \frac{T_{on}}{T}$$

T: Cycle of the primary side  
 T<sub>on</sub>: On-time

The following description describes the parallel MOSFET on the secondary side as Q<sub>A</sub> and Q<sub>B</sub>.

A. Q<sub>1</sub> and Q<sub>4</sub> are on, Q<sub>3</sub> and Q<sub>2</sub> are off

This is the period during power is transferred from the primary side to the secondary side. The primary winding voltage at this time is the input voltage (V<sub>in</sub>). For the secondary winding, the voltages corresponding to the winding ratios are given with the polarity symbol of the n<sub>2</sub> as positive.

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

The voltage is applied to the  $L_O$  via the  $Q_A$ ,

$$\Delta i_{L_O(Q1, Q4\_on)} = \frac{1}{L_O} \times \left( \frac{n_2}{n_1} \times V_{in} - V_{out} \right) \times T_{on}$$

The current determined by the equation increases linearly to charge the  $C_{out}$  and provide the output current  $I_{out}$ . At this time, magnetic energy is stored in the  $L_O$ .

B.  $Q_1$ ,  $Q_2$ ,  $Q_3$  and  $Q_4$  are off

The energy stored in the  $L_O$  flows into the  $Q_A$  and  $Q_B$ . To the  $L_O$  at this time, the current decrease lineally as follows.

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

C.  $Q_1$  and  $Q_4$  are off,  $Q_2$  and  $Q_3$  are on

This is the period during power is transferred from the primary side to the secondary side. During this period, the primary winding voltage becomes  $-V_{in}$ . For the secondary winding, the voltage corresponding to the winding ratio is applied with the polarity symbol side of the  $n_3$  taken as a minus.

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

This voltage is applied to the  $L_O$  via the  $Q_B$ , the current flows as follow:

$$\Delta i_{L_O(Q3, Q2\_on)} = \frac{1}{L_O} \times \left( \frac{n_3}{n_1} \times V_{in} - V_{out} \right) \times T_{on}$$

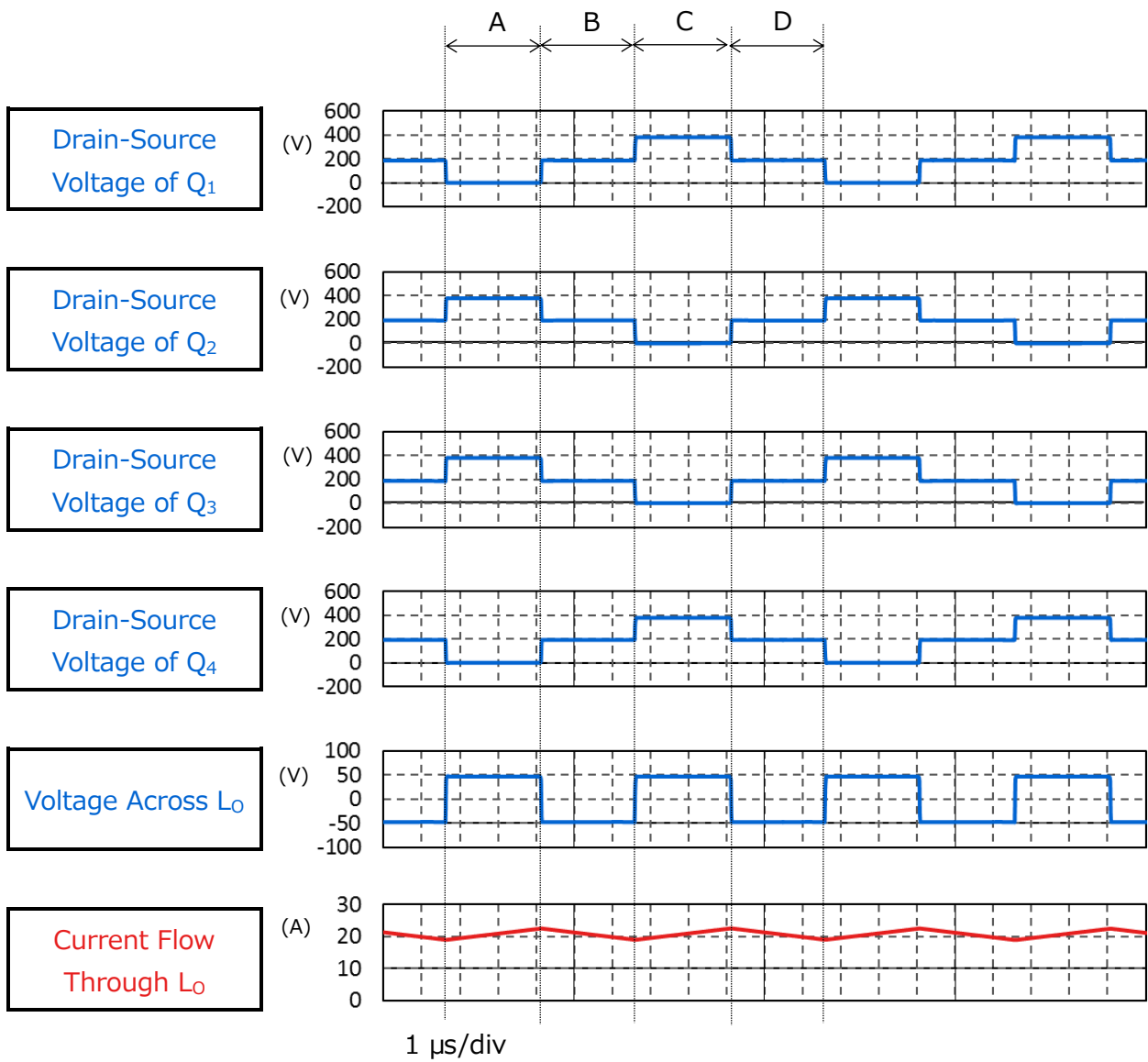
The current flows.

D.  $Q_1$ ,  $Q_2$ ,  $Q_3$  and  $Q_4$  are off

The energy stored in the  $L_O$  flows into  $Q_A$  and  $Q_B$ , the current flows as follow:

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

Figure 3.3 shows the primary-side MOSFET drain-source voltage and output inductor voltage/current waveforms.



**Fig. 3.3 Primary MOSFET Drain-Source Voltage, Output Inductor Voltage/Current**

## (2) Secondary side synchronous rectification operation

This simulation model uses a synchronous rectifier circuit that uses MOSFET instead of diodes for the secondary rectifier. Generally, the conduction loss due to the on-resistance of the MOSFET is smaller than that of the diode so the synchronous rectifier circuit can reduce the conduction loss. The larger output current, the greater loss reduction effect of the synchronous rectifier circuit, and is often used in applications where high efficiency and large capacity are required.

The operation of the secondary MOSFET in the respective periods are as follows.

- a.  $Q_A$  is off,  $Q_B$  is on

Voltage corresponding to the winding ratio with the polarity symbol side of the secondary side winding  $n_2$  as plus voltage,

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

is applied and current flows through the  $Q_A$  to the  $L_O$ .

- b.  $Q_A$  is on,  $Q_B$  is on

Electric power stored in the  $L_O$  recirculates through the  $Q_A$  and  $Q_B$ .

- c.  $Q_A$  is on,  $Q_B$  is off

Voltage corresponding to the winding ratio with the polarity symbol side of the secondary side winding  $n_3$  as a minus voltage

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

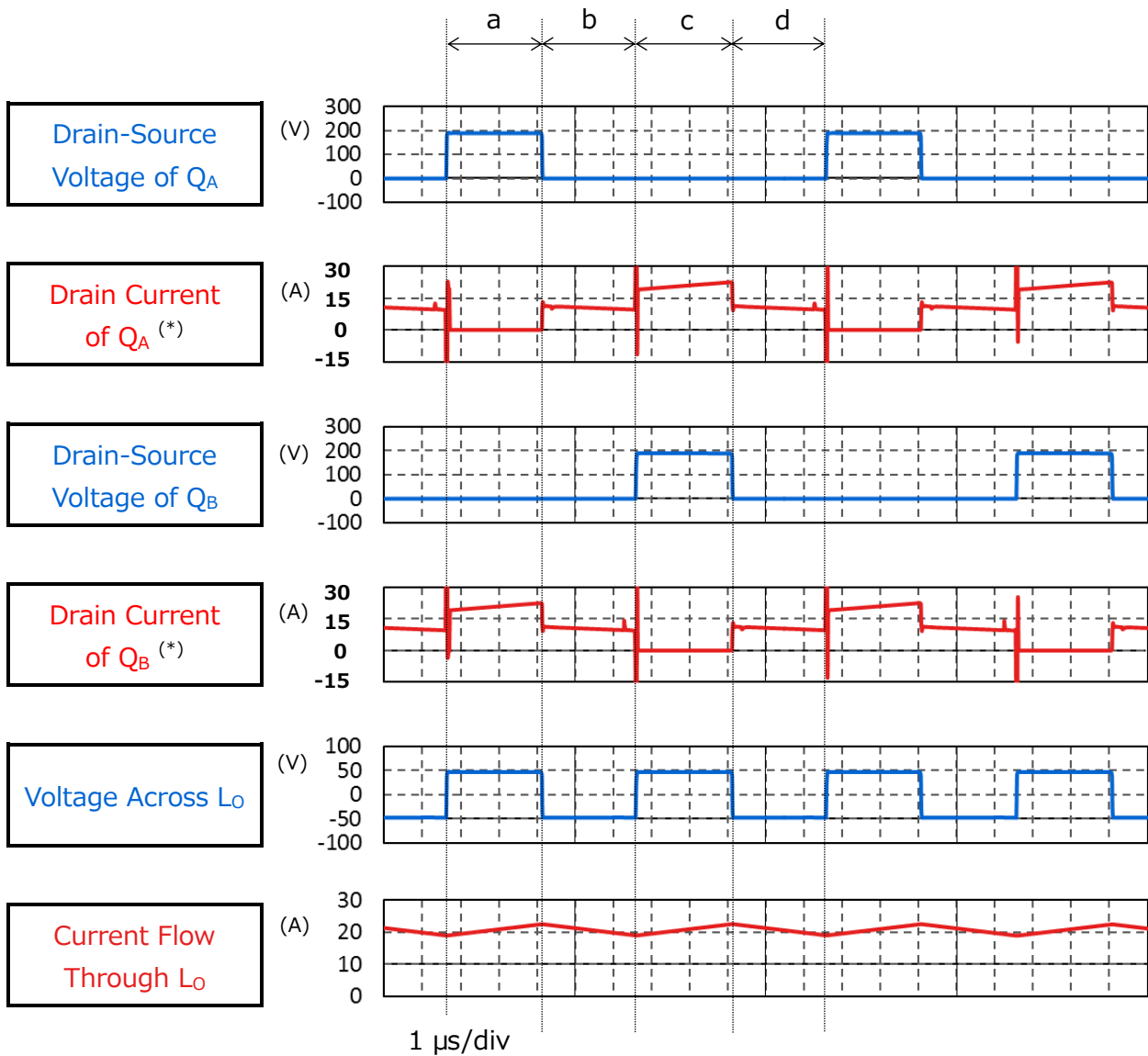
is applied and current flows through the  $Q_B$  to the  $L_O$ .

- d.  $Q_A$  is on,  $Q_B$  is on

Electric power stored in the  $L_O$  recirculates through the  $Q_A$  and  $Q_B$ .

### (3) Output inductor voltage and current at both ends

Figure 3.4 shows the output inductor voltage and current waveforms in conjunction with the secondary-side synchronous rectification operation (secondary-side MOSFET drain-source voltage and current) described in (2).



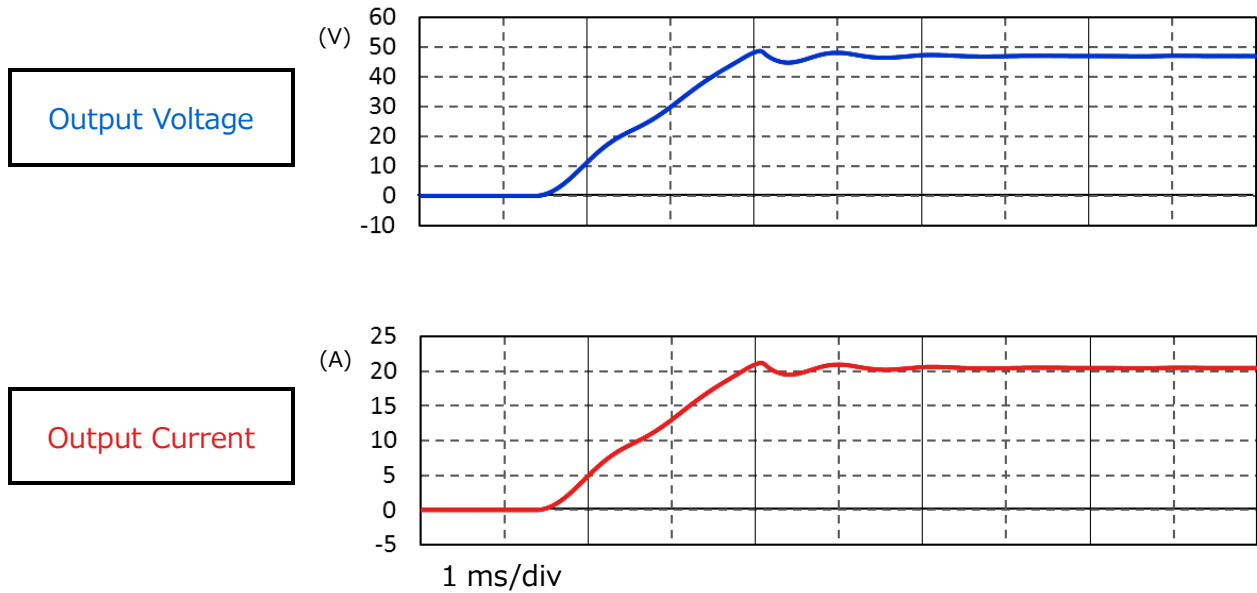
\* : Drain current is positively directed from the MOSFET to the transformer secondary winding.  
The drain current shows the waveform per MOSFET.

**Fig. 3.4 Secondary-Side MOSFET Drains/Source Voltage/Current Output Inductor Voltage/Current Waveforms**



## (4) Output voltage and current as a power supply

Figure 3.5 shows the output voltage and current waveforms of this power supply circuit. It can be seen that it is stable at the set voltage and current.



**Fig. 3.5 Output Voltage and Current Waveforms**

### 4. Product Overview

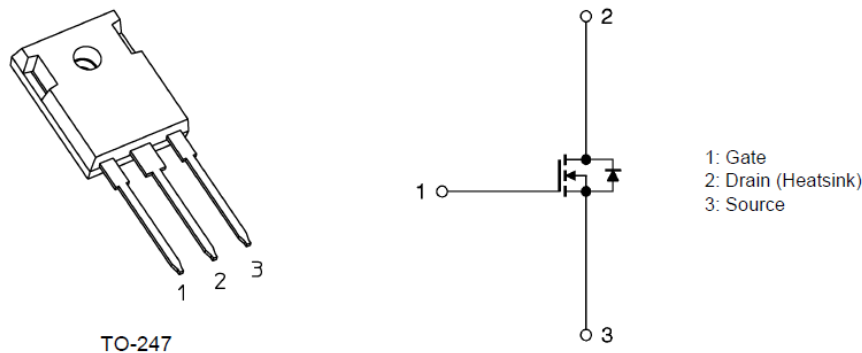
This section introduces the outline of our products that have been tested by incorporating PSpice® models into these circuits.

#### 4.1. TK20N60W

##### Features

- $V_{DSS}=600\text{ V}$ ,  $I_D=20\text{ A}$
- Low on-resistance by adopting super-junction structural DTMOS:  $R_{DS(ON)}=0.13\ \Omega$  (Typ.)
- Optimization of gate switching speed
- Easy-to-handle enhancement type:  $V_{th} = 2.7\text{ to }3.7\text{ V}$  ( $V_{DS}=10\text{ V}$ ,  $I_D=1\text{ mA}$ )

##### Appearance and terminal arrangement

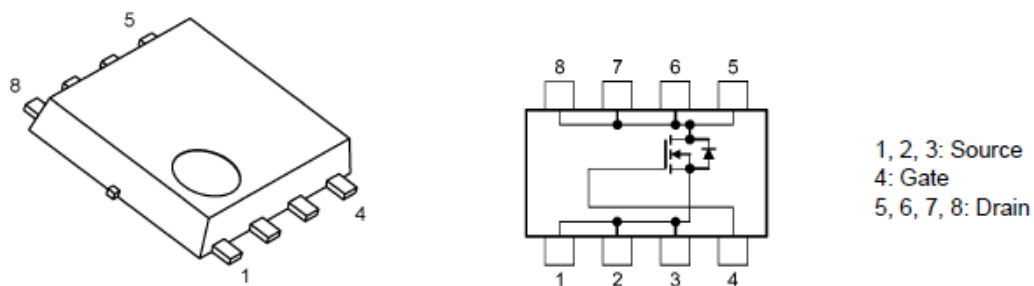


#### 4.2. TPH5200FNH

##### Features

- $V_{DSS}=250\text{ V}$ ,  $I_D=26\text{ A}$
- High speed switching
- Small gate-input charge:  $Q_{SW} = 8.2\text{ nC}$  (Typ.)
- Low on-resistance:  $R_{DS(ON)}=44\text{ m}\Omega$  (Typ.) ( $V_{GS}=10\text{ V}$ )
- Lower leakage current:  $I_{DSS} = 10\ \mu\text{A}$  (Max.) ( $V_{DS}=250\text{ V}$ )
- Easy-to-handle enhancement type:  $V_{th} = 2.0\text{ to }4.0\text{ V}$  ( $V_{DS}=10\text{ V}$ ,  $I_D=1.0\text{ mA}$ )

##### Appearance and terminal arrangement



SOP Advance

Width 5.0 × Length 6.0 × Height 0.95 (mm)

### 5. Using the Simulation Circuit

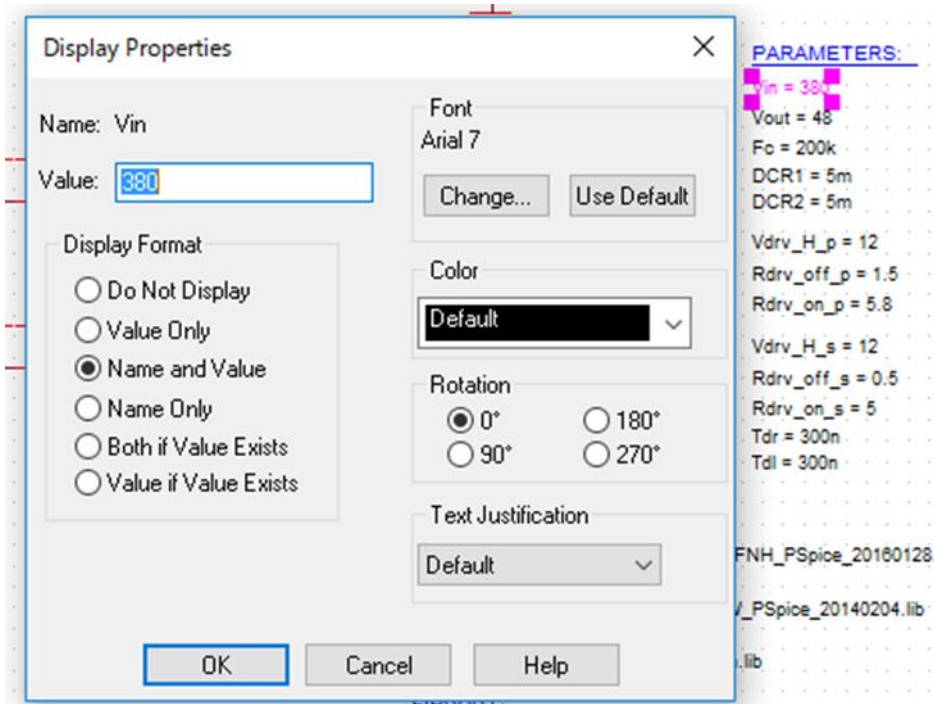
You can freely change various parameters with OrCAD® Capture to verify the circuit operation according to the actual power supply specifications and evaluate how these parameters affect the circuit operation. This section shows how to set simulation parameters and verify the circuit operation.

#### 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**

Variable name	Unit	Description
Vin	V	Input voltage
Vout	V	Output voltage
Fc	Hz	Switching frequencies of the secondary MOSFET
DCR1	Ω	Primary power plane parasitic resistance
DCR2	Ω	GND plane parasitic resistance value on the primary side
Vdrv_H_p	V	Power supply voltage of the primary gate driver
Rdrv_off_p	Ω	Primary MOSFET Internal resistance of gate driver (off side)
Rdrv_on_p	Ω	Primary MOSFET Internal resistance of gate driver (on side)
Vdrv_H_s	V	Power supply voltage of the secondary gate driver
Rdrv_off_s	Ω	Secondary MOSFET Internal resistance of gate driver (off side)
Rdrv_on_s	Ω	Secondary MOSFET Internal resistance of gate driver (on side)
Tdr	Sec	Dead time in Q3 and Q4
Tdl	Sec	Dead time of the legs of Q1 and Q2

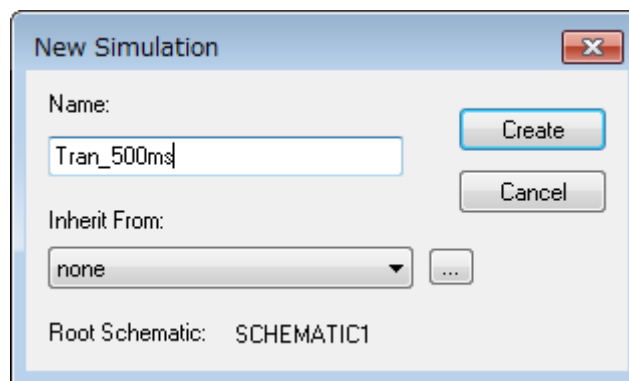


**Fig. 5.1 Parameter Setting Screen**

**Setting analysis parameters**

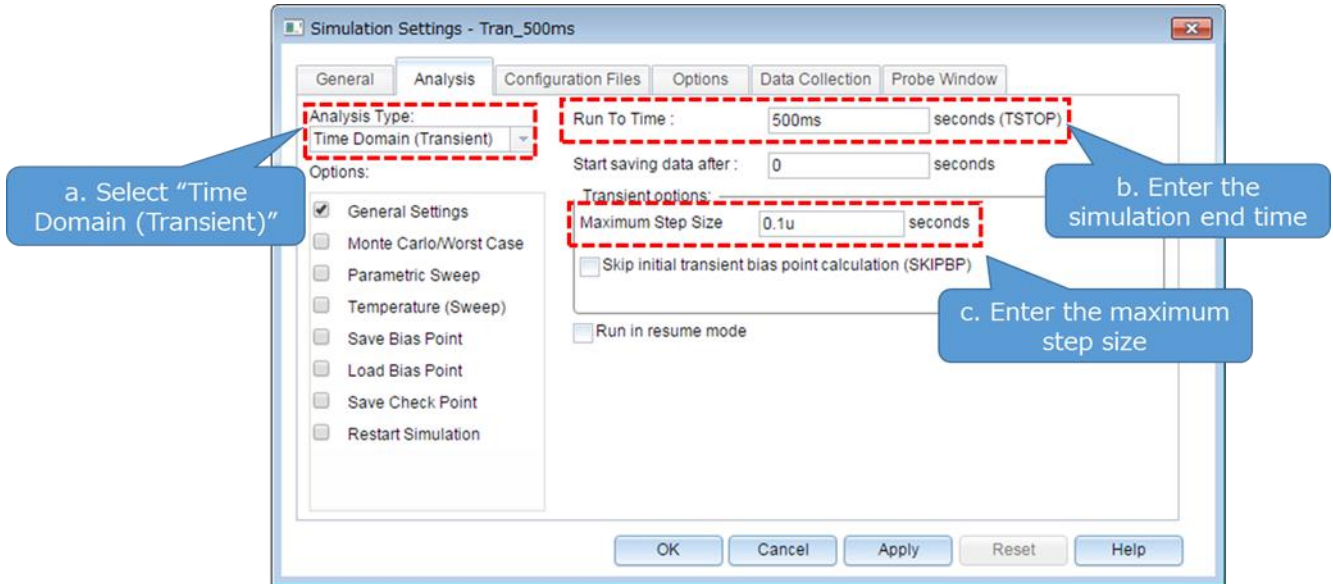
The following describes how to run a simulation on the simulation circuit.

- (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**.



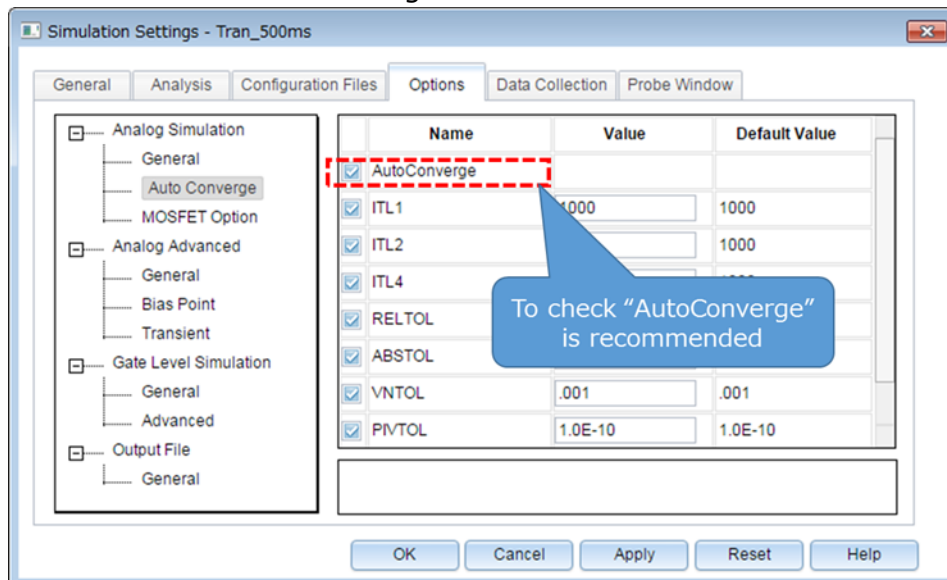
**Fig. 5.2 New Simulation Dialog Box**

- (2) 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 Window**

- (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.

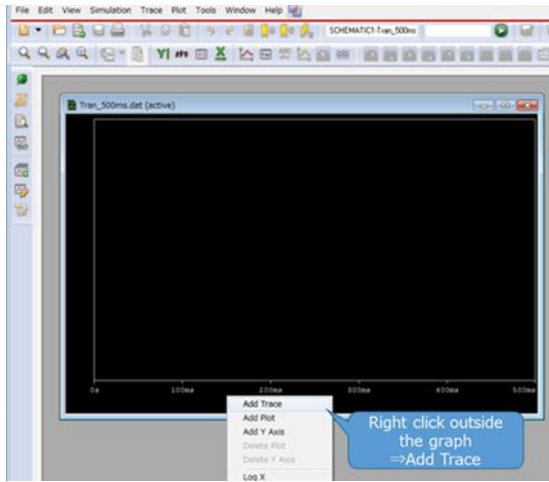


Fig. 5.5 Graph Window

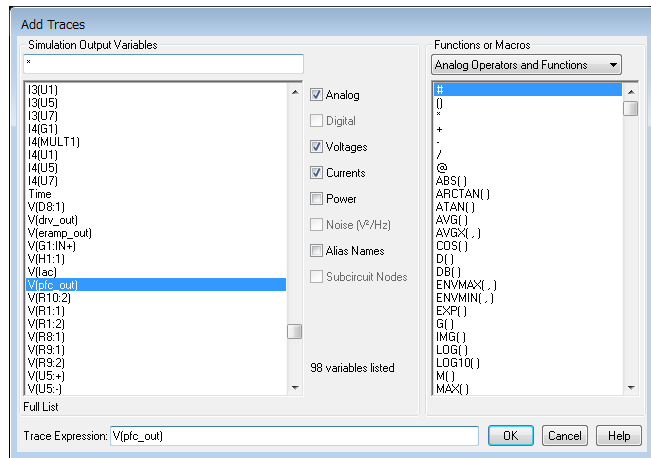


Fig. 5.6 "Add Traces" Window

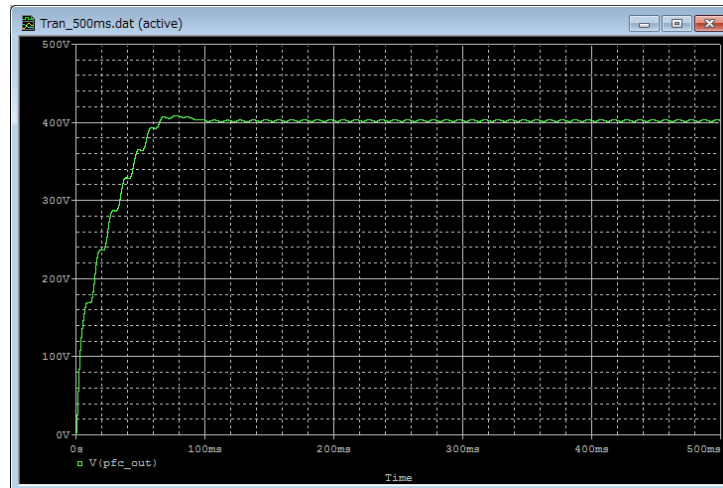
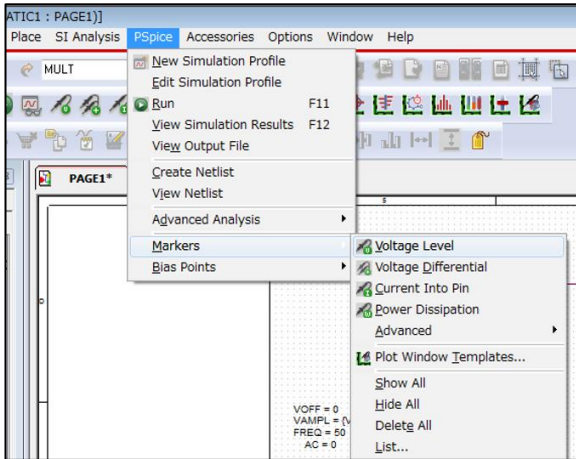


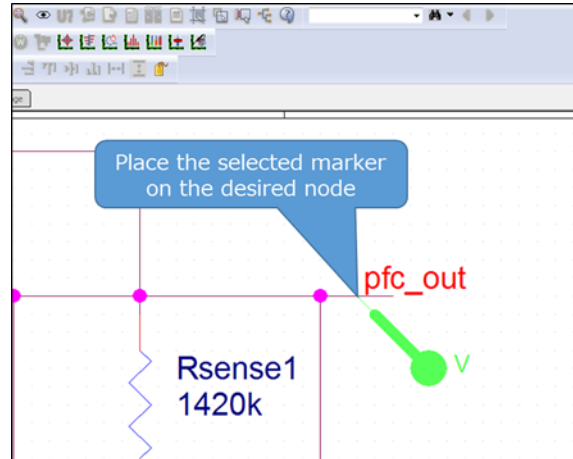
Fig. 5.7 Result Waveform Display (Example: Output Voltage Waveform)

### Method 2: Adding markers

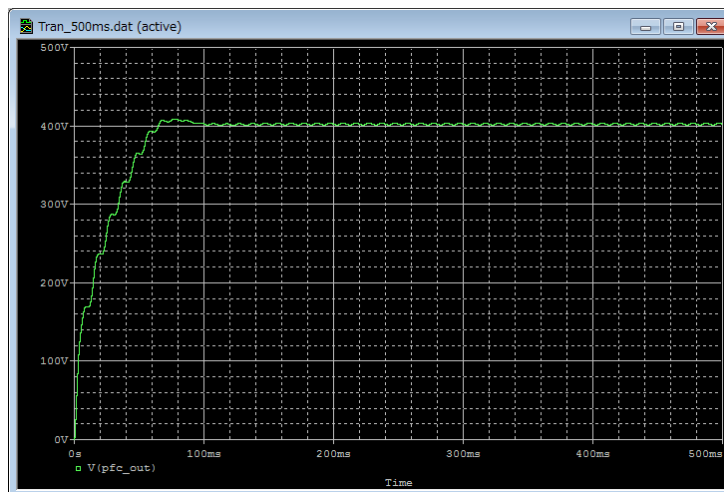
1. From the menu bar of OrCAD® Capture, select **PSpice - Markers** and then a type of marker as shown in Fig. 5.8.
2. Place the selected marker on the desired node in the simulation circuit as shown in Fig. 5.9.
3. Then, its waveform appears in the graph window of PSpice A/D as shown in Fig. 5.10.



**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.