Active Clamp Forward AC-DC Power Supply Basic Simulation Circuit

Reference Guide

RD157-RGUIDE-01

TOSHIBA ELECTRONIC DEVICES & STORAGE CORPORATION

Table of Contents

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.

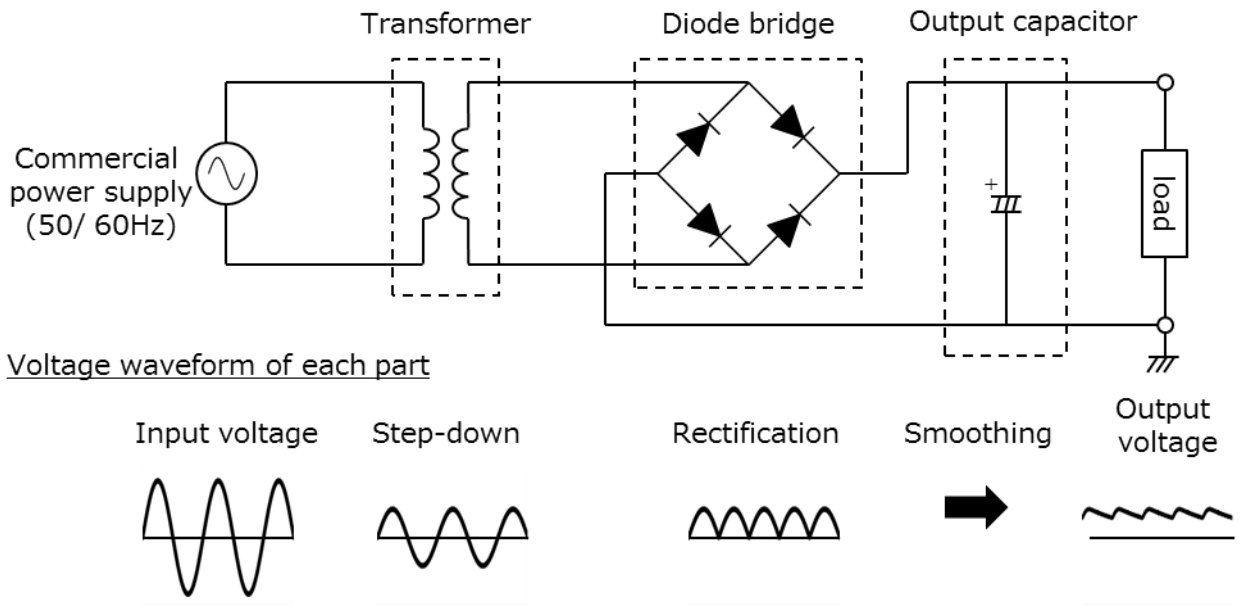


Fig.1.1 Transformer-Base AC-DC Power Supply Configuration

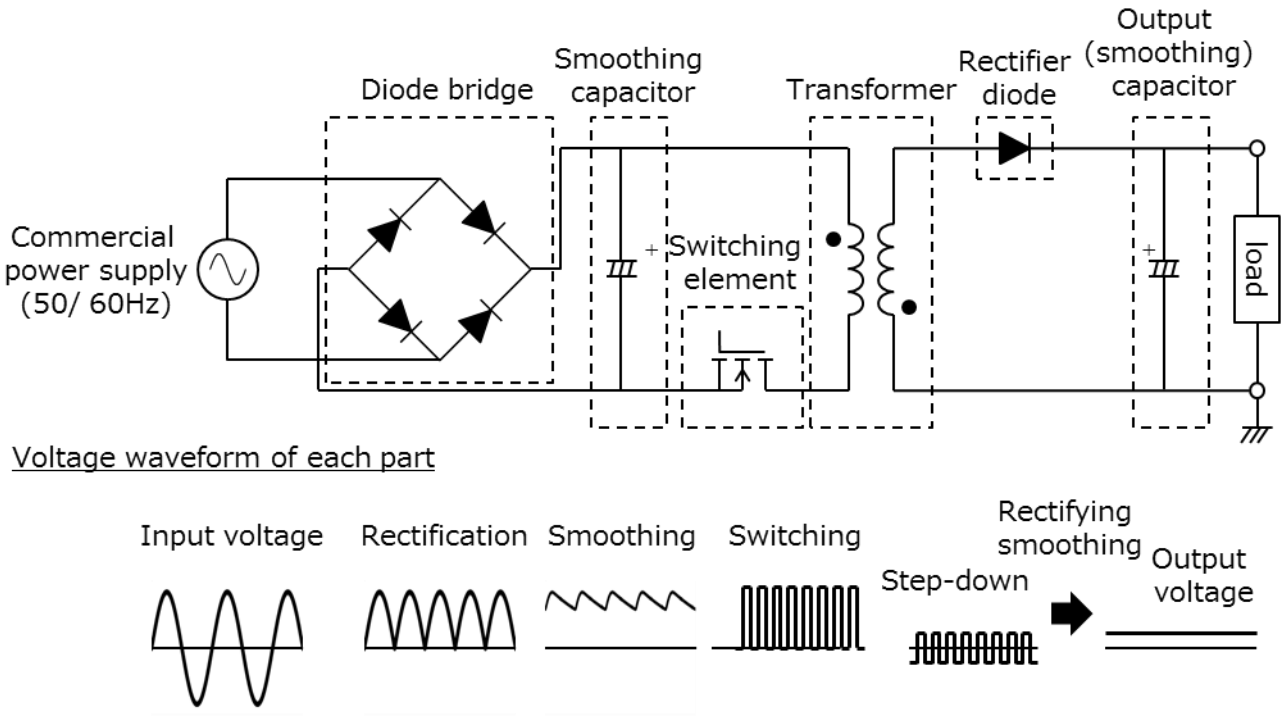


Fig. 1.2 Switch-Base AC-DC Power Supply Configuration

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.

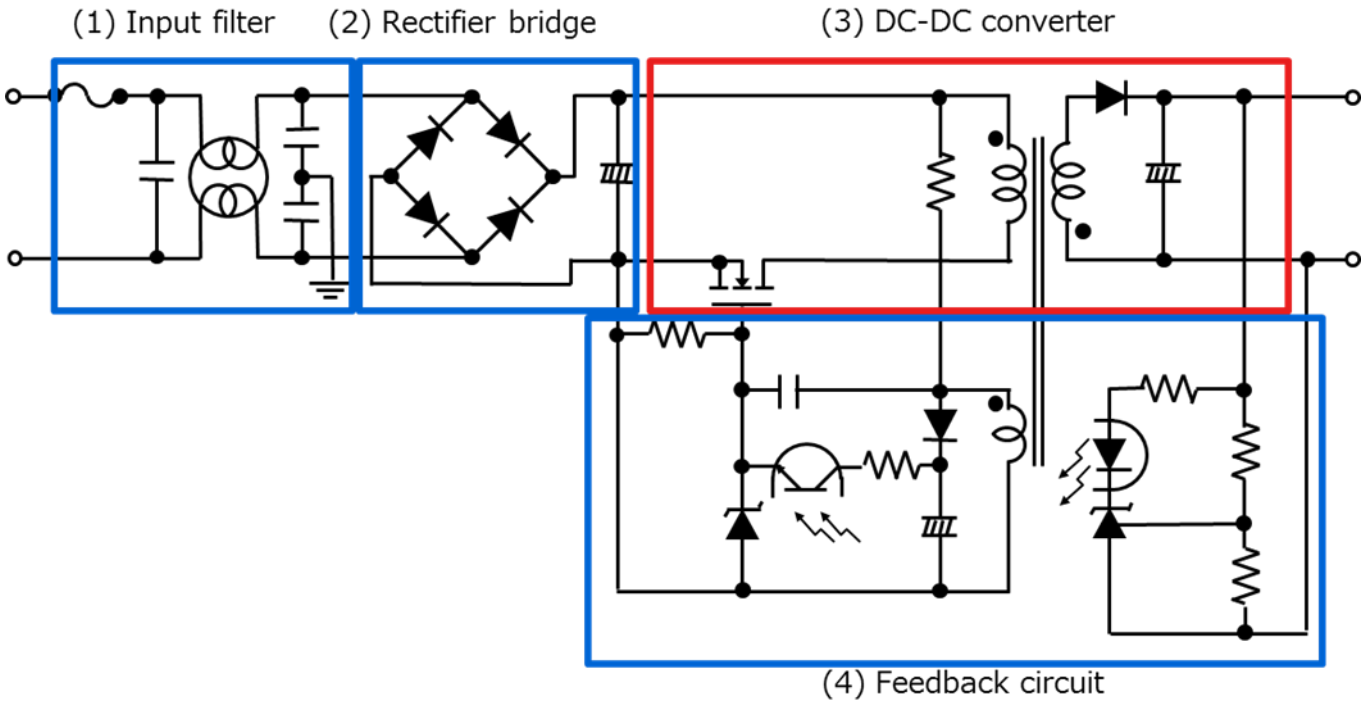


Fig. 1.3 Block Diagram of Switching Type AC-DC Power Supply

(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

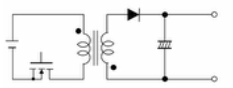
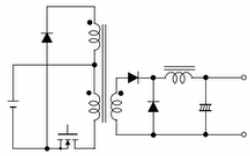
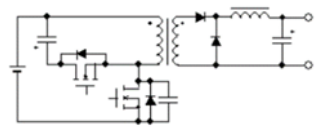
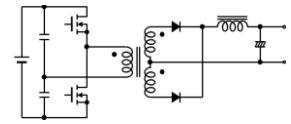
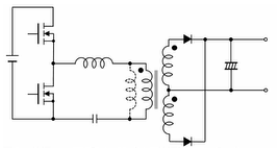
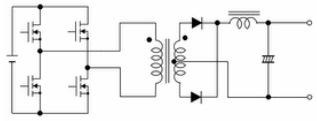
Converts the voltage rectified in (2) to an arbitrary DC voltage.

(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 DC-DC converter section		Power level	Advantages	Disadvantages
Flyback		< 200 W	・ Small part count	・ Reduction in efficiency at high power ・ Large transformer
Forward		50 W～500 W	・ Higher efficiency than a flyback	・ Transformer reset circuit required
Active clamp forward		50 W～500 W	・ Higher efficiency than a forward	・ There are many parts. ・ Be difficult to control
Half bridge		100 W～1 kW	・ High efficiency ・ Low noise	・ Specially designed transformer is required. ・ Be difficult to control
Resonant half bridge (LLC resonance)		100 W～1 kW	・ High efficiency than a half bridge ・ Low noise	・ Specially designed transformer is required. ・ Be difficult to control
Full bridge		> 200 W	・ High efficiency ・ Capable of increasing the power capacity	・ There are many parts. ・ Be difficult to control

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

The Video of the DC-DC converter are shown here → Click Here

The active clamp forward method described in this document is widely used for power supplies requiring high efficiency because the active clamp operation when resetting the transformer allows products with low tolerant voltage to be used for the MOSFET for the main switch, and the MOSFET for the main switch and active clamp operate at zero volt switching (ZVS) to achieve low switching losses. In order to understand the operation of the active clamp forward method DC-DC converter in a switching method AC-DC power supply, Toshiba provide basic simulation circuits (RD157-SPICE-01) on our website.

This document provides an overview of this basic simulation circuit and explains how to use it. The Cadence's Capture and PSpice ® 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 Active Clamp-Forward AC-DC Power Supply (DC-DC Converter Section)

The basic simulation circuit (RD157-SPICE-01) is an active clamp-forward AC-DC power supply with 100 W 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.

Active clamp forward AC-DC power basic simulation circuit: → Click Here

2.1. Power Supply Specifications

The active clamp-forward AC-DC power supply (DC-DC converter circuit) specifications described in this document are as follows:

- Input voltage (V_{in}) : 200 V
- Output voltage (V_{out}) : 5 V
- Output current (I_{out}) : 0 to 20 A
- Primary MOSFET Operating Frequency (F_C) : 150 kHz
- Line ratio: n_1:n_2 = 10:1
- Secondary allowable ripple current width (ΔI_{ripple}) : 50 %

2.2. Circuit Configuration

Fig. 2.1 shows the simulation circuit for OrCAD®. This is the active clamp forward type DC-DC converter power supply, which mainly consists of a power section (active clamp forward) 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 main switching MOSFET is "TK10A60W" and the secondary synchronous rectifier MOSFETs are "TPH2R003PL" as examples.

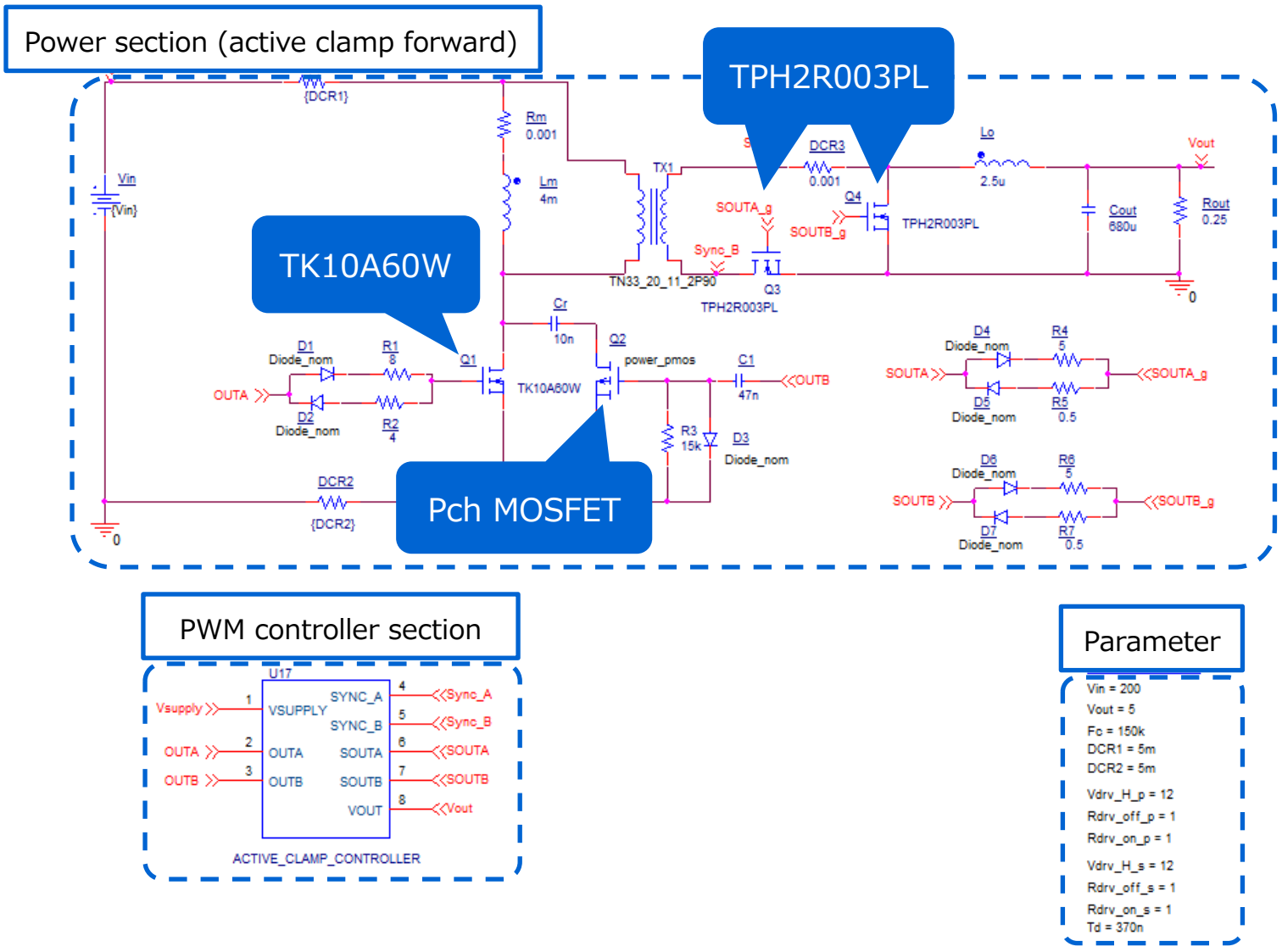


Fig. 2.1 Simulation Circuit of 100 W Active Clamp Forward (DC-DC converter) Power Supply

Selection of primary MOSFET

Primary side MOSFET (TK10A60W: V_{DSS}=600 V, I_D for the primary main switch = 9.7 A) is selected from the following viewpoints.

(1) Breakdown-voltage

The voltage applied to the MOSFET at static state is the input voltage (=200 V). Select a MOSFET with a breakdown-voltage of 400 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 is at the maximum output power. If the conversion efficiency at the maximum output power (=100W) is 90%, the maximum average input current is 0.56 A. Select a MOSFET with a current rating of 2 A or more.

MOSFET for active clamping on the primary side requires a device with a withstand voltage of -400 V or less, considering the surge voltage and other factors during switching as well as MOSFET for the main switch. Current flows in this MOSFET with an exciting current (I_{mag}) up to -0.08 A determined by the equation below. However, OrCAD® included general-purpose elements are used because our lineup does not have an element with a current of-0.08 A or less at a withstand voltage of-400 V or less.

$$I_{mag} = \frac{V_{in} \times D_{uty}}{F_c \times L_{mag}} \div 2$$

V_{in}: 200 V, D_{uty}: 0.5, F_c: 150 kHz, L_{mag} (excitation inductance) : 4 mH

Selection of secondary MOSFET

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

(1) Breakdown-voltage

Since the winding ratio is 10:1, the voltage between the middle point and both ends of the secondary winding at static state is 12 V, which is 1/10 of the input voltage, and 12 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 MOSFET with a breakdown-voltage of 30 V or more.

(2) Current rating

The maximum output current is at the maximum output power. Since the maximum output current when the maximum output power (=100 W) is 20 A, an element with a current rating of 40 A or more is required. In addition, select an element 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) = 200
- Transformer winding ratio: n_2/n_1 = 1/10
- Output voltage: V_{out} (V) = 5
- MOSFET switching frequency: F_c (Hz) = 150k
- Max. output current: I_{out_max} (A) = 20
- Ripple Current Width Permitted: Delta I_{ripple} (%) = 50

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

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

The inductance value (L_o) of the output inductor is calculated as 2.5 μH from the above equation, and 2.5 μH is selected as the setting value from the power supply specifications of this model. In the actual design, the inductance value of the inductor varies 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 Results

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

(1) Active clamp forward basic operation ("Primary MOSFET Drain-Source Voltage/Current")

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

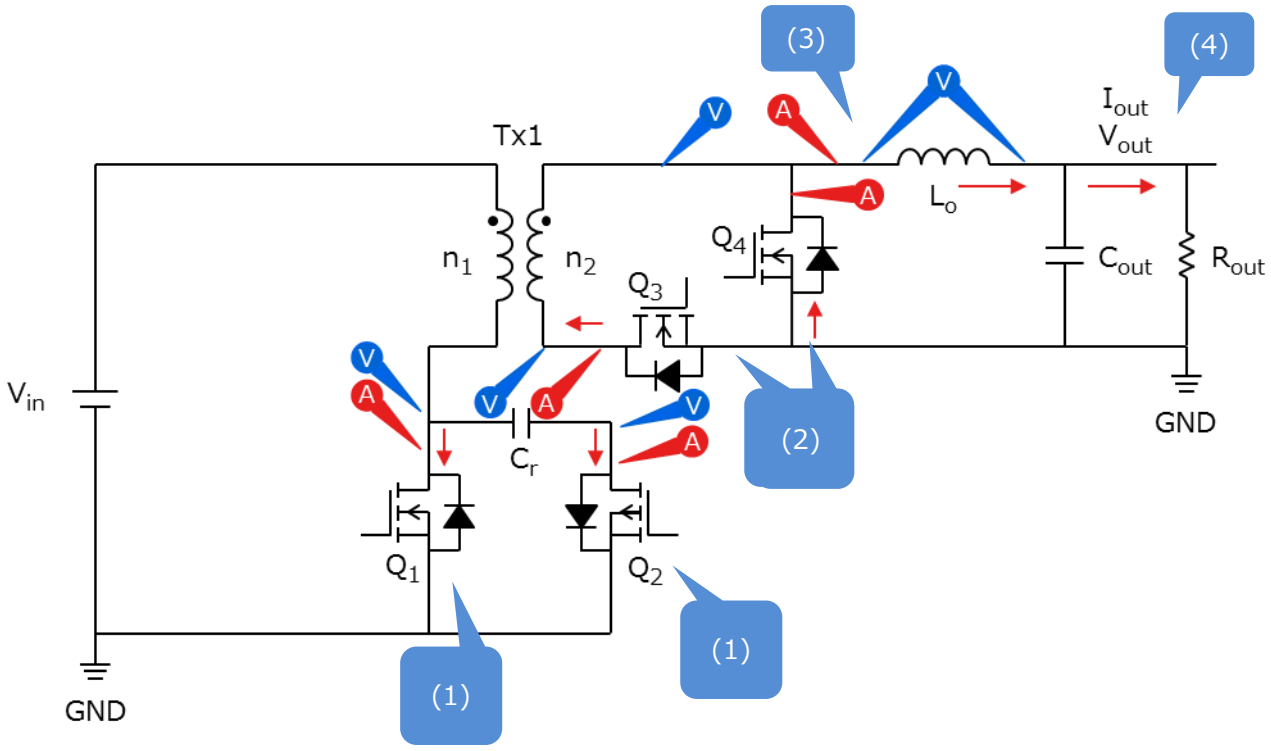
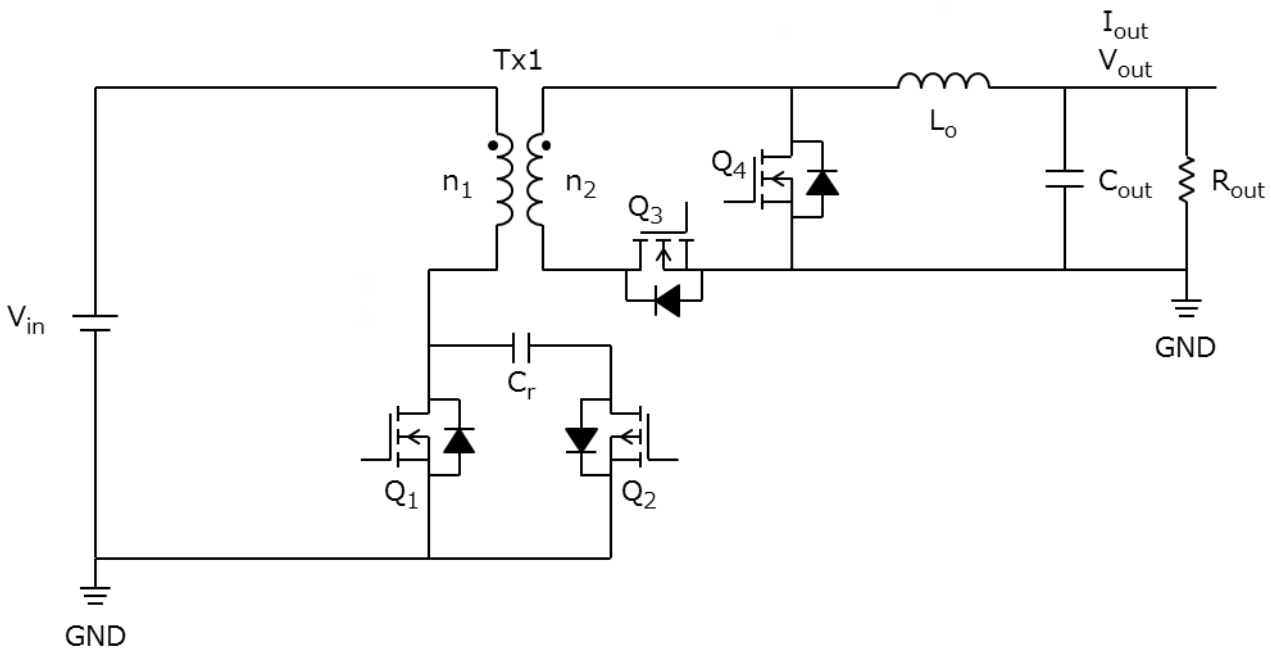


Fig. 3.1 Simulation Waveform Measurement Point List

(1) Active clamp forward basic operation

The basic operation of the active clamp forward method is described in Fig. 3.2, which shows the active clamp forward circuit.



Q_1～Q_2: Primary MOSFET
Q_3～Q_4: Secondary MOSFET
L_o: Output inductor
C_r: Resonance capacitor

Fig. 3.2 Active Clamp Forward Circuit

In the active clamp forward method, the output voltage is stabilized by controlling Q_1 in PWM (Pulse Width Modulation).

For the output-voltage V_{out}, it can be calculated by:

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

T: Period of the primary side
T_{on}: On time

A. Q_1 is on, Q_2 is off

This is the period during which power is transferred from the primary side to the secondary side. The primary winding voltage at this time is the input voltage (V_{in}). For the secondary winding, the voltages corresponding to the winding ratios are given with the polarity symbol of the n_2 as positive.

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

The voltage is applied to the L_O via the Q_3,

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

The current determined by is linearly increased 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 is off, Q_2 is off

The Q_1 off charges the Q_1 drain-source capacitance $C_{DS(Q1)}$ and increases $V_{DS(Q1)}$. When the $V_{DS(Q1)}$ exceeds the V_{in}, the voltage on the primary side of the transformer (Tx1) is reversed, the Q_3 is turned off, and the energy stored in the L_O is circulated through the Q_4. To the L_O at this time, the current decrease linearly as follows.

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

C. Q_1 is off, Q_2 is on

When the $V_{DS(Q1)}$ increases and exceeds the resonant capacitor voltage (V_{Cr}), the body diode $D_{(Q2)}$ of the Q_2 conducts and the exciting current of the Tx1 flows to the C_r. At this time, the $V_{DS(Q2)}$ of the Q_2 is nearly 0 V, so it is turned on with ZVS (zero-volt switching). The Tx1 resonates under the condition determined by the primary inductance and chromium. When the $V_{DS(Q1)}$ reaches the peak, the excitation current flows from the C_r to the V_{in}, and the direction of the Q_2 drain current is negative.

D. Q_1 is off, Q_2 is off

The Tx1's exciting current continues to flow in V_{in} directions, discharging the charge stored in the Q_1's drain-to-source capacitance ($C_{DS(Q1)}$). After discharging the $C_{DS(Q1)}$, the exciting current flows through the body diode $D_{(Q1)}$ of the Q_1. After this, the Q_1 turns on, but the $V_{DS(Q1)}$ turns on at ZVS since it is almost 0 V.

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

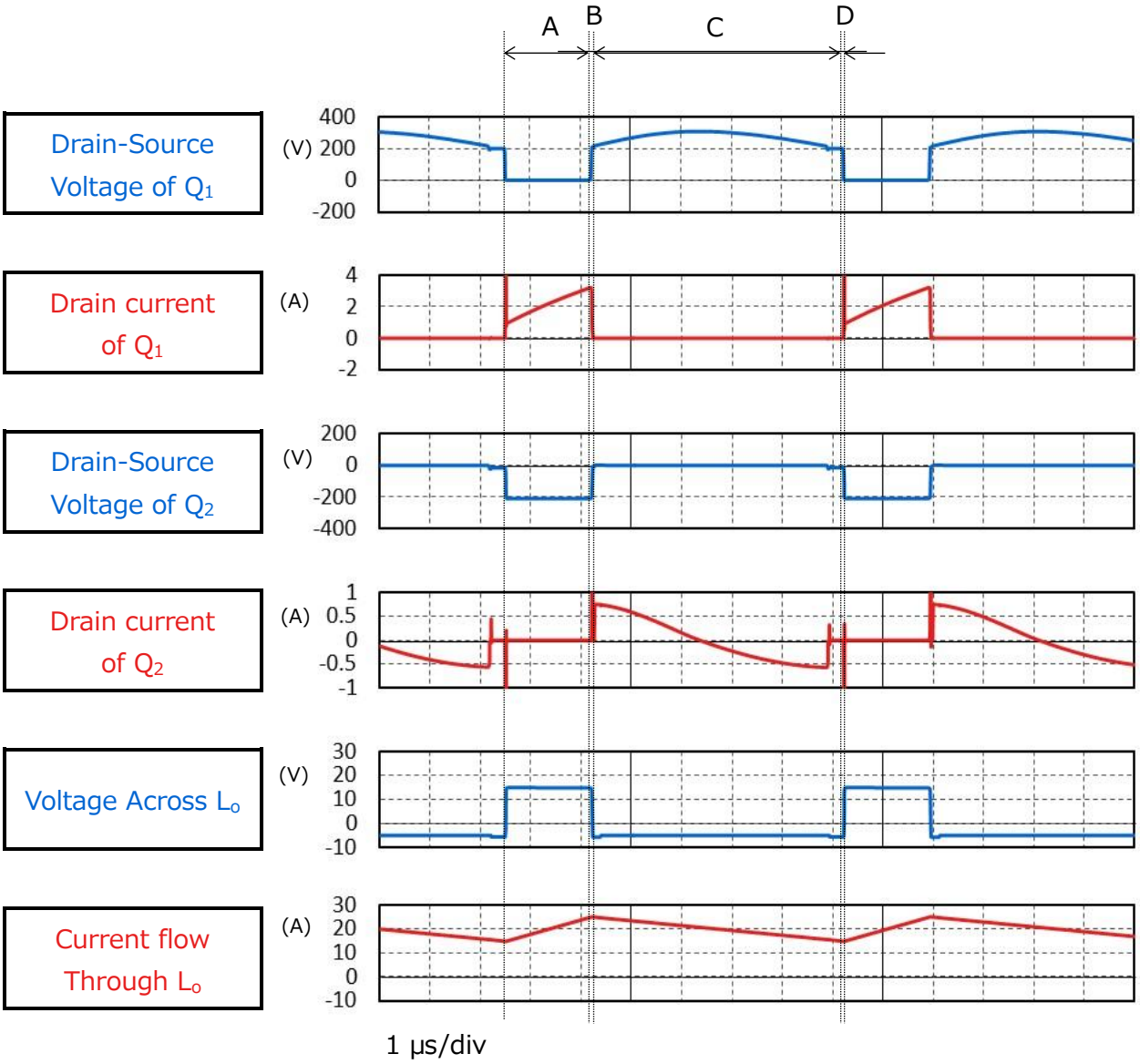


Fig. 3.3 Primary MOSFET Drain-Source Voltage/Current, Output Inductor Voltage/Current

(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 is as follows.

a. Q_3 is on, Q_4 is off

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, current flows through the Q_3 to the L_O and charges the C_{out} while supplying the output current (I_{out}).

b. Q_3 is off, Q_4 is off

The energy stored in the L_O passes through the Q_4 body diodes, causing an annular current to flow.

c. Q_3 is off, Q_4 is on

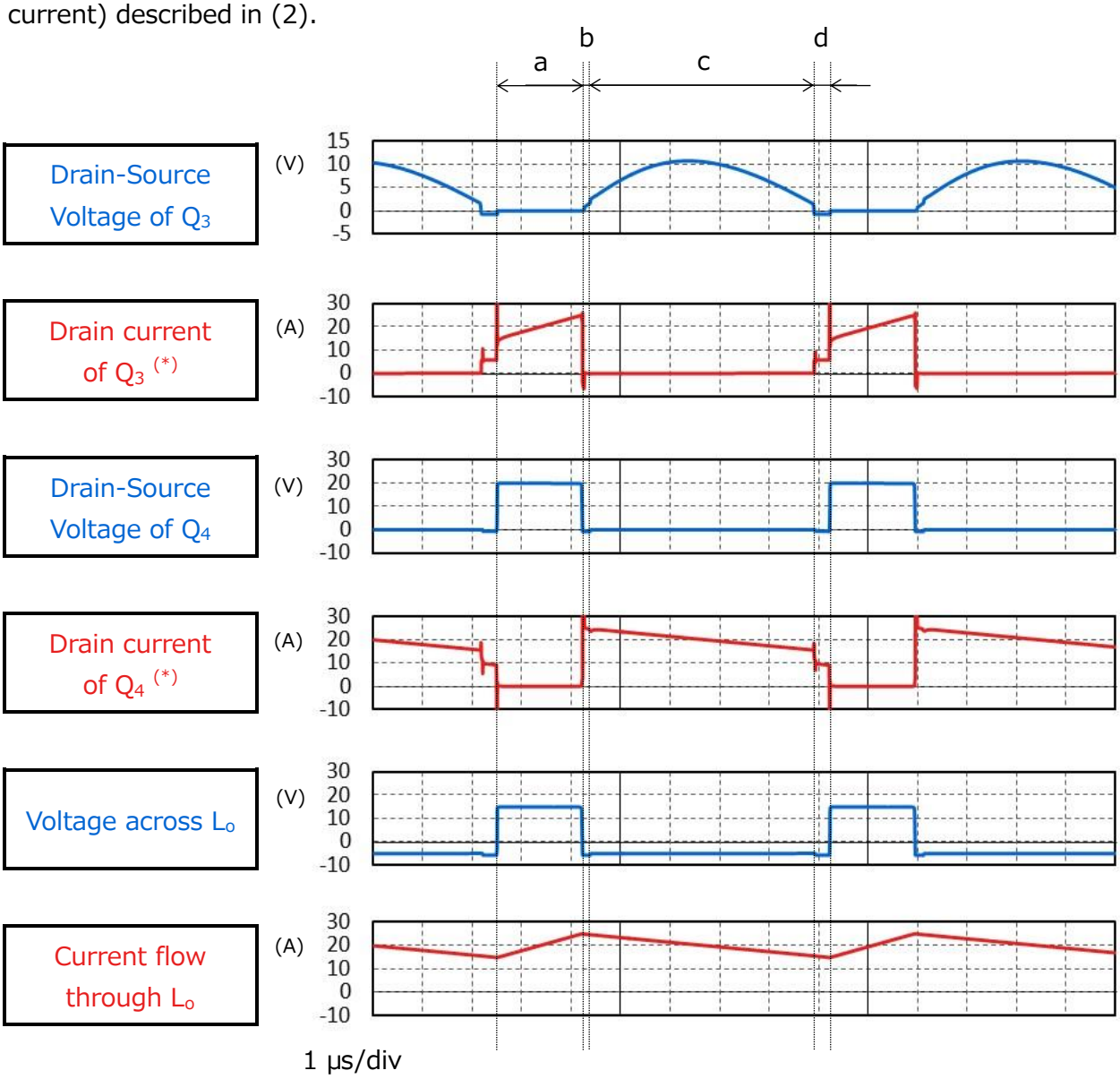
The energy stored in the L_O passes through the MOSFET part of the Q_4, and an annular current flows.

d. Q_3 is off, Q_4 is off

The energy stored in the L_O flows through the body diodes of the Q_3 and Q_4, causing an annular current to flow.

(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 source to the drain of the MOSFET.

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

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

Fig. 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.

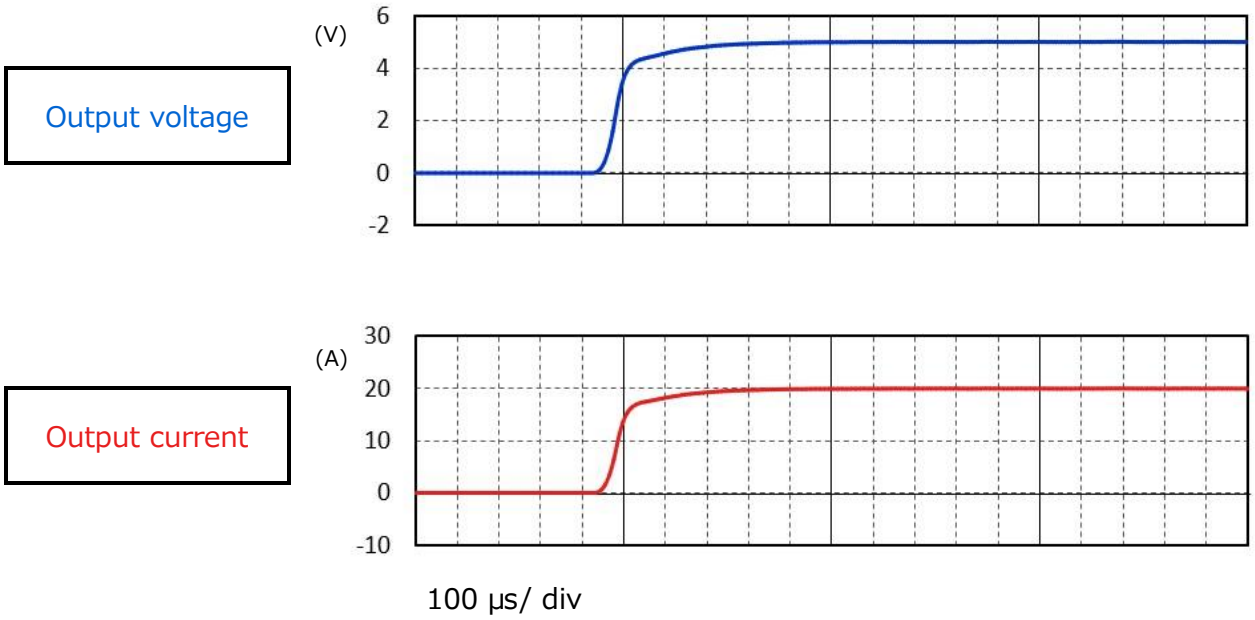


Fig. 3.5 Output Voltage and Current Waveforms

4. Product Overview

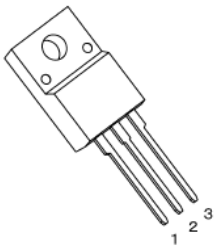
This section provides an overview of our products that have been tested by incorporating PSpice® models into the circuits.

4.1. TK10A60W

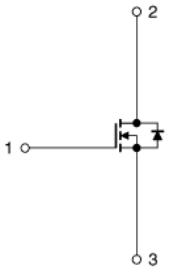
Features

- V_{DSS}=600 V, I_D=9.7 A
- Low on-resistance : $R_{DS(ON)}$=0.327 Ω (Typ.) (V_{GS}=10 V)
- Easy to control Gate switching
- Enhancement mode : V_{th} = 2.7 to 3.7 V (V_{DS}=10 V, I_D=0.5 mA)

Appearance and terminal arrangement



TO-220SIS



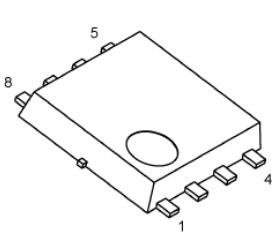
1: Gate
2: Drain
3: Source

4.2. TPH2R003PL

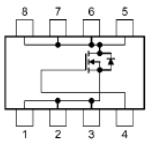
Features

- V_{DSS}=30 V, I_D=100 A
- High speed switching
- Small gate-input charge : Q_{SW} = 22 nC (Typ.)
- Small power charge : Qoss = 41 nC (Typ.)
- Low on-resistance : $R_{DS(ON)}$=1.3 mΩ (Typ.) (V_{GS}=10 V)
- Lower leakage current : I_{DSS} = 10 μA (Max.) (V_{DS}=30 V)
- Enhancement mode : 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

In order to verify the operation according to actual specifications and the changes according to circuit constants, the simulator can freely change various parameters and analyze the operation of the simulator on OrCAD's Capture. This section describes how to set parameters and how to analyze the operation when actually performing a simulation.

How to set parameters

Table 5.1 lists the parameters that can be set by the simulation circuit. Double-clicking a variable in the parameter setting section displays the "Display Properties" window shown in Fig. 5.1. Change the "Value" value in that window.

Table 5.1 List of Variables Settable in Parameter 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)
Td	Sec	Q_1 and Q_2 dead times

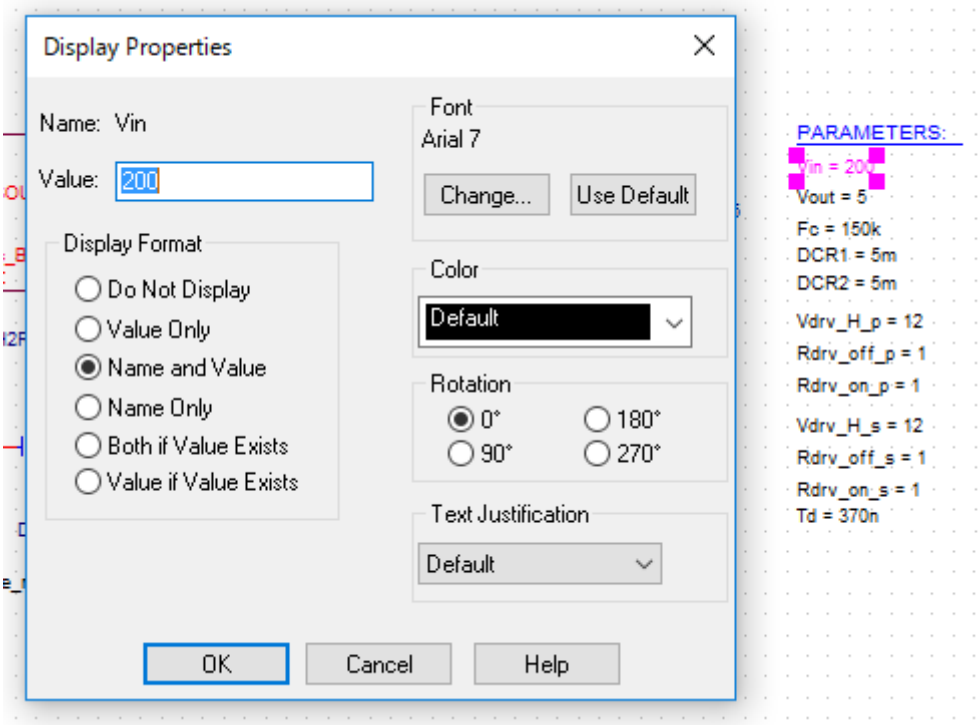


Fig. 5.1 Parameter Setting Screen

How to set analysis

The procedure for executing the simulation of this simulation circuit is described below.

(1) The "New Simulation" window shown in Fig. 5.2 is displayed by clicking "PSpice"-"New Simulation Profile" on OrCAD's Capture menu bar. Specify any profile names and click "Create".

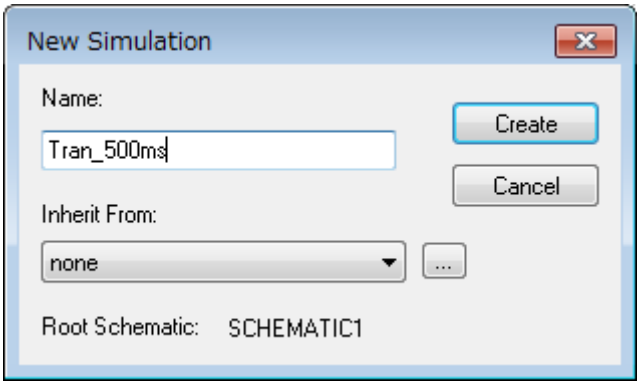


Fig. 5.2 New Simulation window

(2) After the steps in 1., the "Simulation Settings" window shown in Fig. 5.3 is displayed, allowing various analysis settings to be made. First, set the analysis method in Analysis tab. Specify "Time Domain(Transient" for "Analysis Type". Specify the duration of the analysis in "Run To Time" and the largest step length in the analysis in "Maximum Step Size".

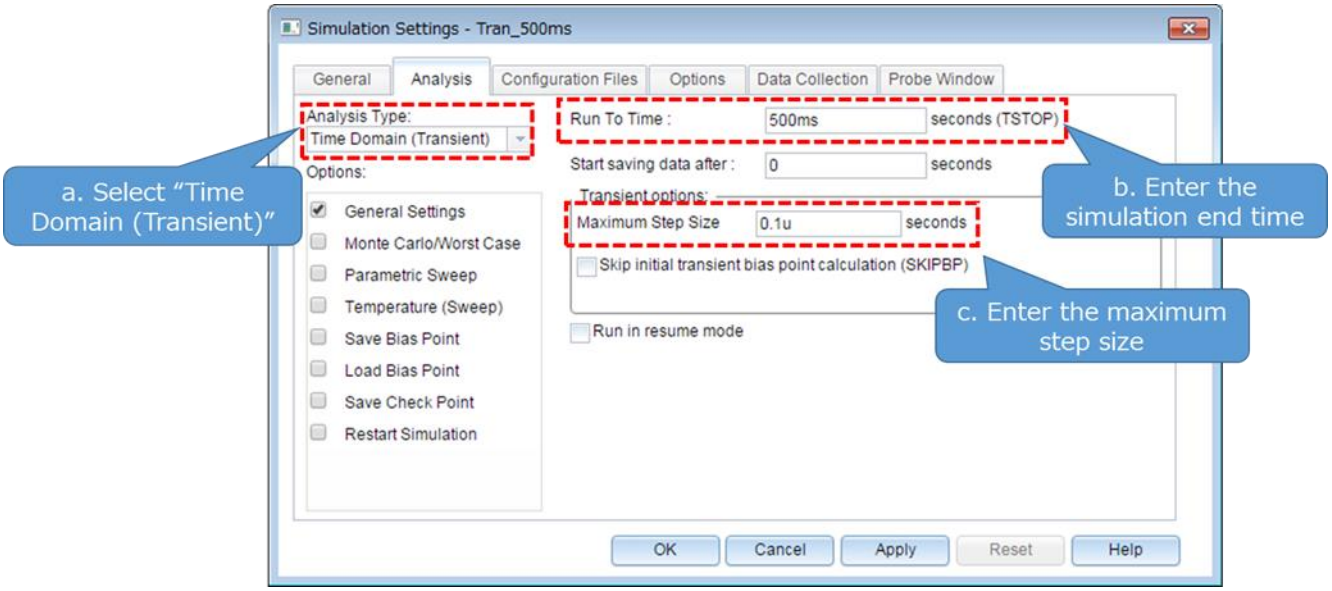


Fig. 5.3 Simulation Settings"-"Analysis window

(3) Set analysis options in Options tabs It is recommended to activate the auto-convergence function by checking the "Analog Simulation"-"Auto Converge"-"AutoConverge" checkbox as shown in Fig. 5.4.

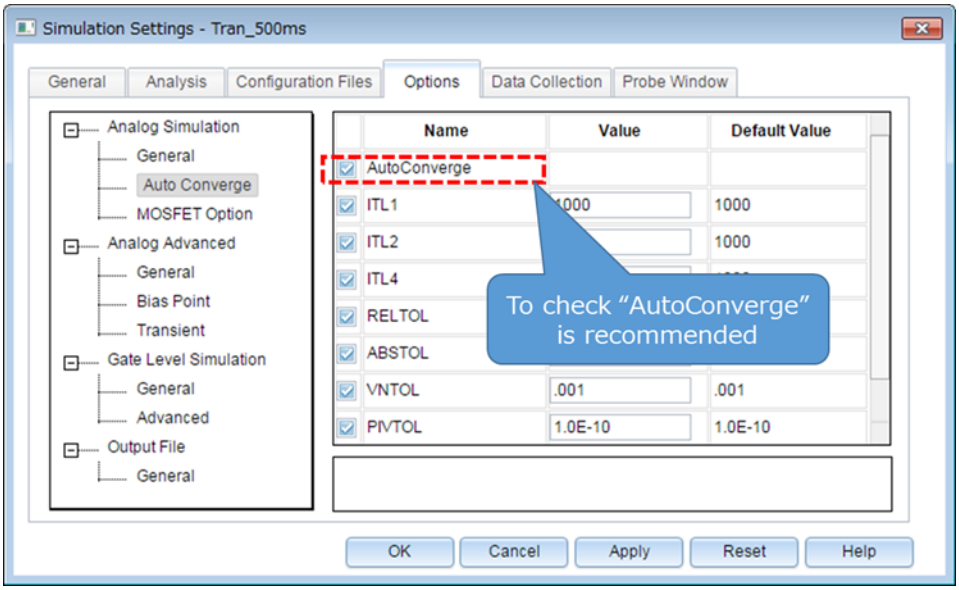


Fig. 5.4 "Simulation Settings"-"Options" window

(4) When the above settings are completed, click "OK" and close the "Simulation Settings" window.

(5) Execute simulations at "PSpice"-"Run" on OrCAD "Capture" menu bar. PSpice A/D starts automatically and the simulations are executed.

How to check the results

This section describes how to check the results after completion of simulation. There are two methods for displaying the result waveforms on PSpice A/D window. The procedures for each method are described below.

Method 1. Results display with net name specified

(1) Right-click outside the graph frame in the graph window, and then select "Add Trace". (Fig. 5.5)

(2) Select the waveforms to be displayed from the "Add Traces" window. Select V (net name) for the voltage waveform and I (element name) for the current waveform. (Fig. 5.6)

(3) After selecting, click "OK" to display the result waveform. (Fig. 5.7)

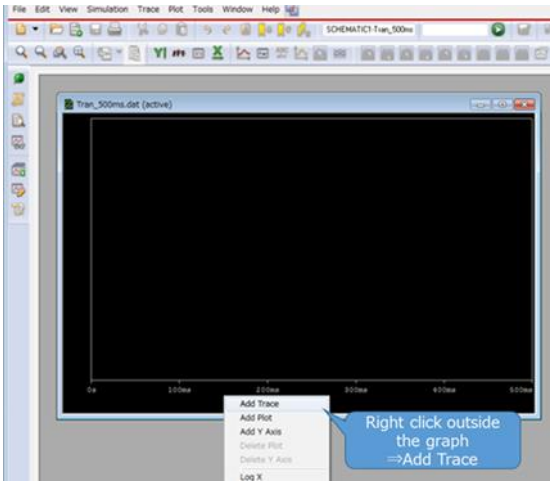


Fig. 5.5 Graph Window

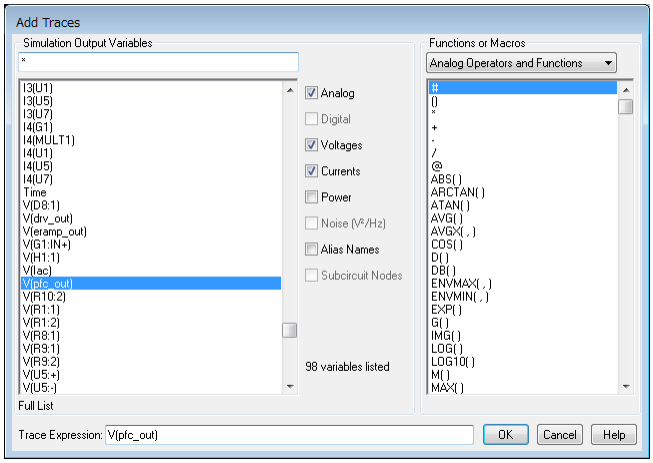


Fig. 5.6 Add Traces Window

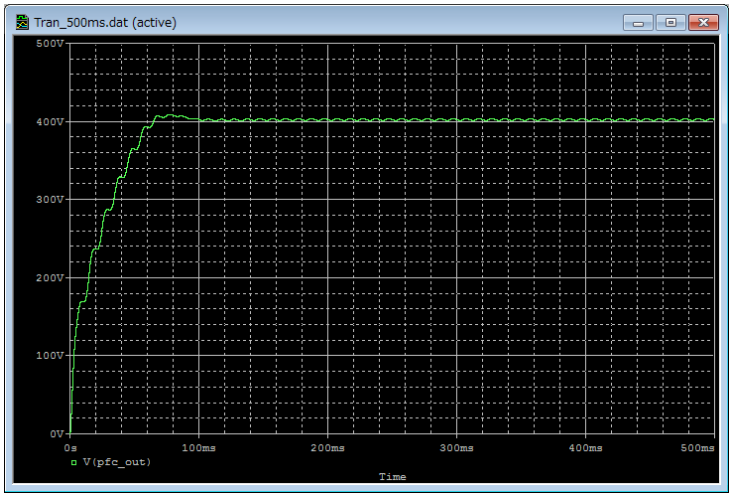


Fig. 5.7 Result Waveform Display (Example: Output Voltage Waveform)

Method 2. Displaying the result using Marker function

(1) Select Marker type according to the waveform to be displayed from "PSpice"-"Markers" on OrCAD's Capture menu bar (Fig. 5.8)

(2) Place Marker at the waveform measuring points on the simulator. (Fig 5.9)

(3) The result waveforms are displayed in the graph window of PSpice A/D. (Fig. 5.10)

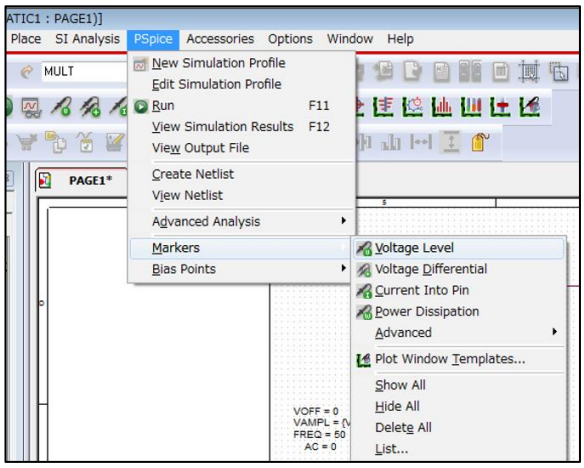


Fig. 5.8 Selecting Marker Type

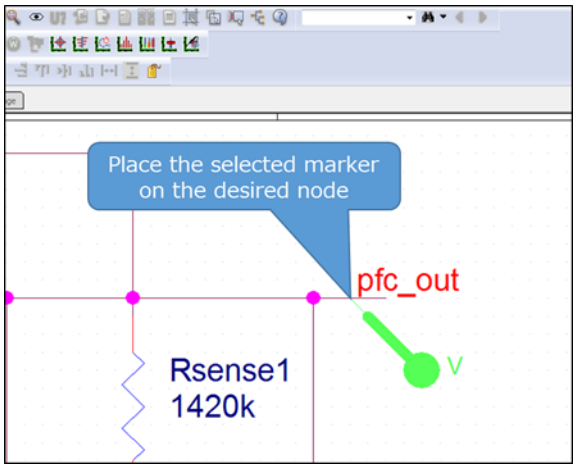


Fig. 5.9 Placing a Marker in the Circuits

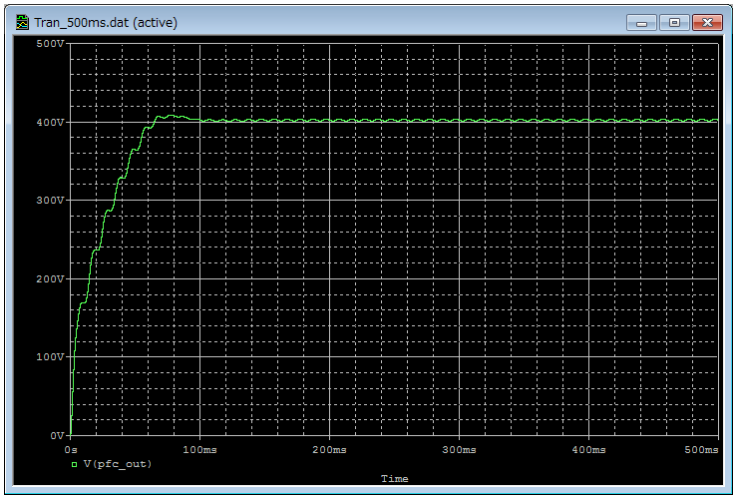


Fig 5.10 Simulation Waveform View (Example: Output Voltage Waveform)

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.