Linear Regulator Series

Usage of SPICE Macromodel (for LDO)

This application note explains how to install SPICE macromodels (hereafter "SPICE models") and how the circuit symbols are configured.

Contents

1. Typical application circuit

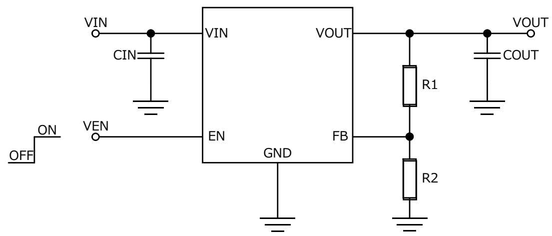

Figure 1. Basic circuit diagram of adjustable output type LDO

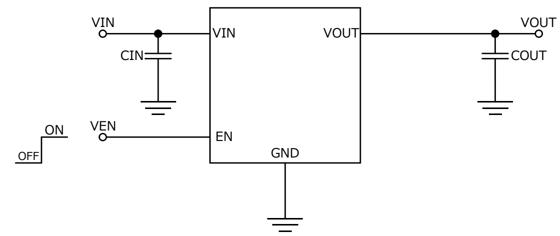

Figure 2. Basic circuit diagram of fixed output type LDO

Pin name	Function
VOUT	Output pin
FB	Output voltage setting pin
GND	Ground pin
EN	Enable pin
VIN	Input pin

2. What is a SPICE model for IC

When you use various types of SPICE simulators, the simulation time tends to increase with the number of elements (nodes) inside the IC. Naturally, a shorter simulation time is preferable for users. Therefore, it is necessary for a circuit to be simplified so that the number of nodes is reduced. A SPICE model is a simplified circuit described by a programing language (Figure 3).

However, since a single SPICE model may not be able to reproduce all the functions and operations of the actual equipment, multiple SPICE models are used to complement each other in such a case. Accordingly, it is necessary to use the SPICE models that are suitable for each application.

Examples shown below are the functions that are simplified when a circuit is converted into an equivalent circuit.

- Temperature characteristics
- Operations of parasitic elements
- Specific operations
- Characteristics of the terminal protection diodes
- Terminal impedance

Since other functions are also simplified, please contact us before you evaluate the characteristics that are not listed in the modeling reports attached to the SPICE models.

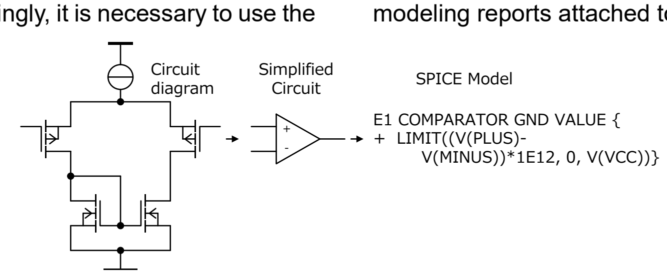

Figure 3. Steps for creating a SPICE model

No. 61AN027E Rev.001
JULY 2018

3. How to access SPICE models

3.1 How to download SPICE models from ROHM's website

Open ROHM's home page (Figure 4).

URL

https://www.rohm.com

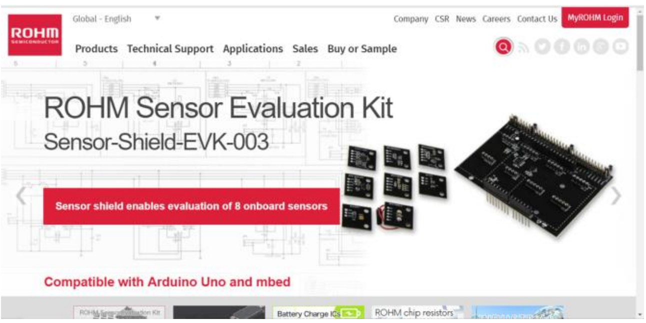

Figure 4. ROHM's home page

Next, click [Technical Support] -> [Design Support] -> [Design Simulation Models] in this order (Figure 5).

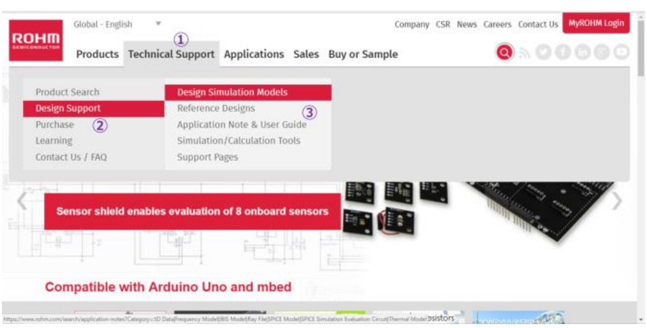

Figure 5. How to access SPICE models from the ROHM's website

The next window is displayed (Figure 6).

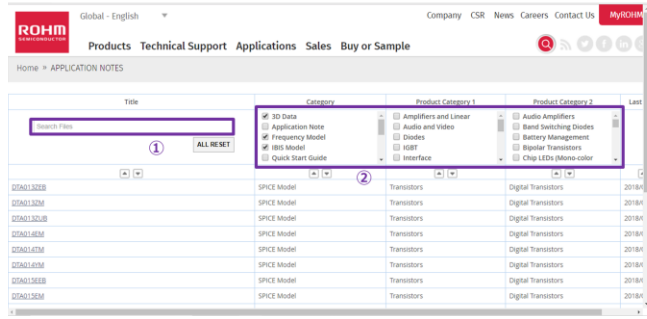

Figure 6. Page for the design simulation models

There are two ways to obtain the SPICE models on this page.

1. Enter the product name under [Title] to search.
2. Check the applicable check boxes in [Category], [Product Category 1], and [Product Category 2]. Select the product name from the displayed list.

3.2 How to download SPICE models from PSpice Community

Open the special page for ROHM in PSpice Community.

URL

http://www.pspice.com/models/rohm

1. Select [Power Management] from the menu on the left of the page.
2. Then, click [Linear Regulators] to display the list of the registered models.
3. Agree to the license and download the models.

4. Preparation for simulations

4.1 Registration of SPICE models

4.1.1 When using PSpice

This section introduces an example when PSpice (Ver. 17.2) is used.

In the menu bar, click [File] -> [New] -> [Project] in this order (Figure 7).

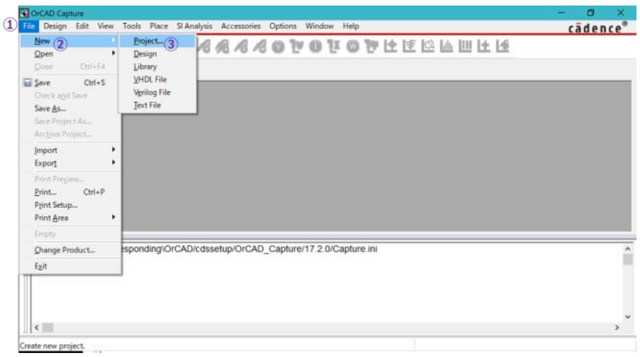

Figure 7. Creating a project

Check [PSpice Analog or Mixed A/D] in the displayed dialog box, enter a filename and a location to save the project, and press the [OK] button. In this example, “LDO” is entered as the filename and the LDO folder on the desktop is selected as the location to save the project. In the save folder, the SPICE models are saved as files with the extension .lib, and their symbols are saved as files with the extension .olb (Figure 8).

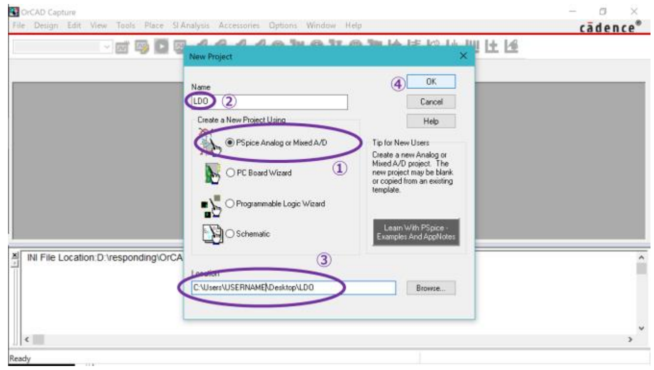

Figure 8. Selecting a project type

In the dialog box displayed next, check [Create a blank project] and press the [OK] button (Figure 9).

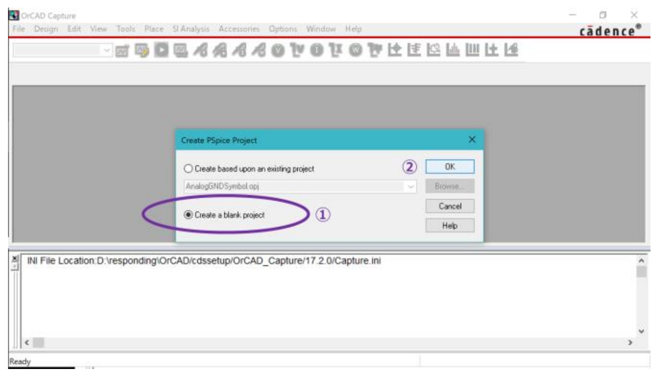

Figure 9. Creating a new project

A new project is created without any circuit entry (Figure 10).

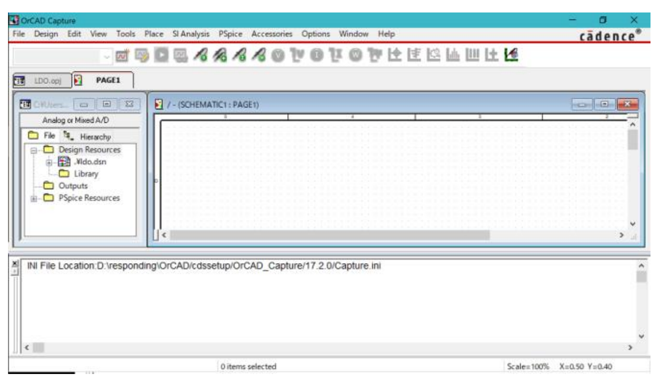

Figure 10. Display of project in the initial state

Add the SPICE models and the symbol groups. To add a symbol group, right-click [Library] on the left of the window and select [Add File] (Figure 11).

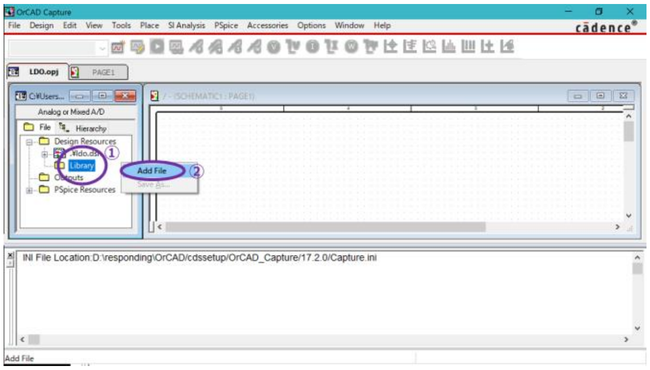

Figure 11. Adding a file to the library

In the dialog box, select a displayed symbol file (with the extension .olb) and press the [Open] button. Then, the symbol group is registered to the library (Figure 12).

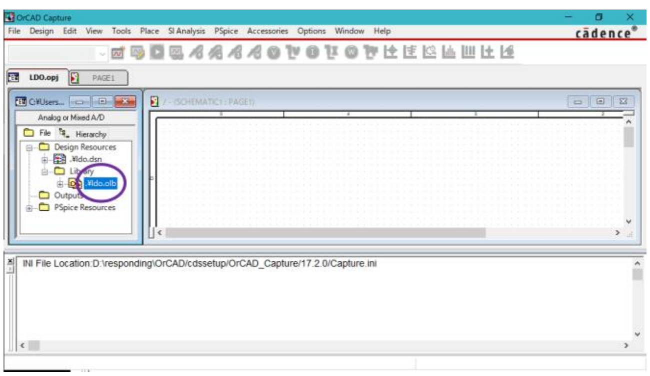

Figure 12. Window when the registration to the library is completed

When you have created a new symbol, associate a SPICE model with the symbol. Click [+] on the left of the registered symbol group in Figure 12, right-click on the symbol to be used, and click [Associate PSpice Model…] (Figure 13).

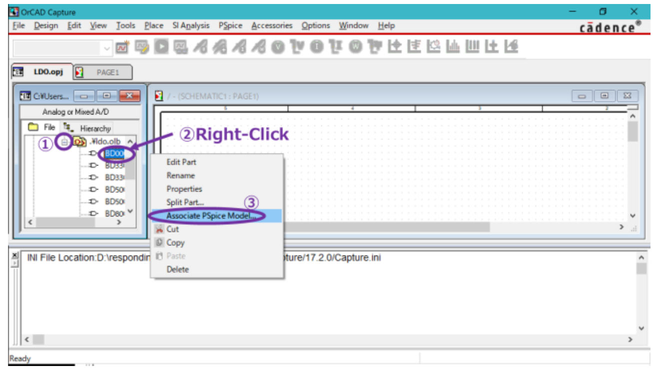

Figure 13. Associating a SPICE model with a symbol

If a previous symbol has already been used, a warning message is displayed indicating that the pin assignment has previously been made. To register a new symbol, press the [Yes] button. The model import dialog box is displayed (Figure 14).

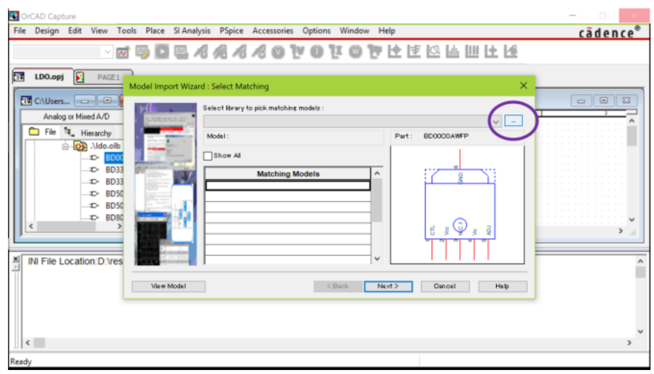

Figure 14. Model import dialog box

Select […] on the upper right of the dialog box. When a new dialog box is displayed, select a SPICE model file and press the [Open] button (Figures 14 and 15).

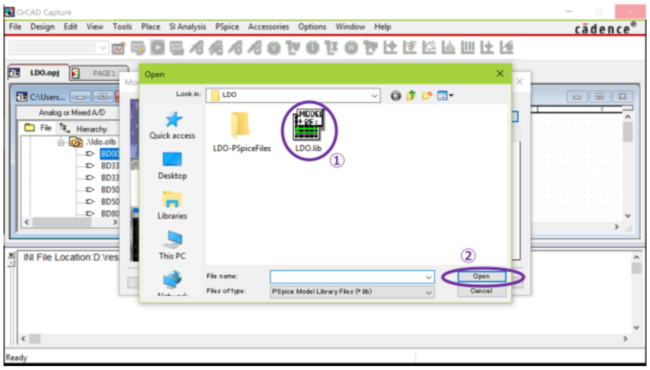

Figure 15. Window for model selection

From the list of available SPICE models, select a model to be used and press the [Next] button (Figure 16).

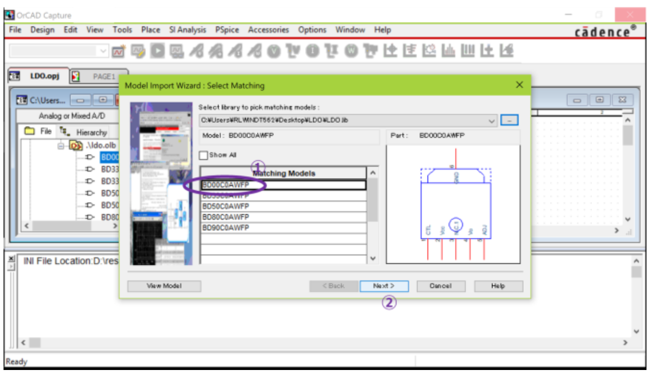

Figure 16. Selection of a model to be used

Make the pin assignment of the SPICE model and the symbol. Select from the list box under [Symbol Pin] to make the assignment. Assign the pin with the same name (Figure 17).

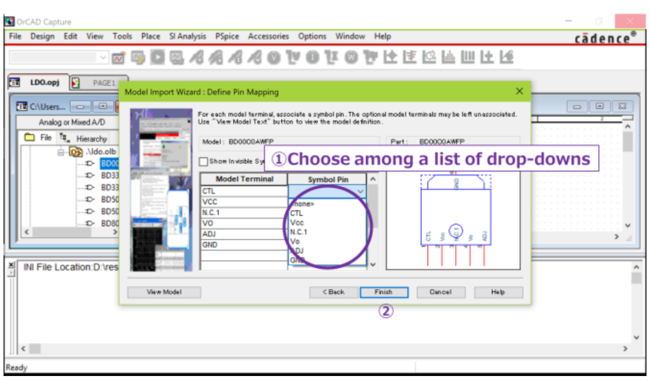

Figure 17. Pin assignment

After all pin assignments have been completed, press the [Finish] button and then press the [OK] button in the displayed dialog box (Figure 18).

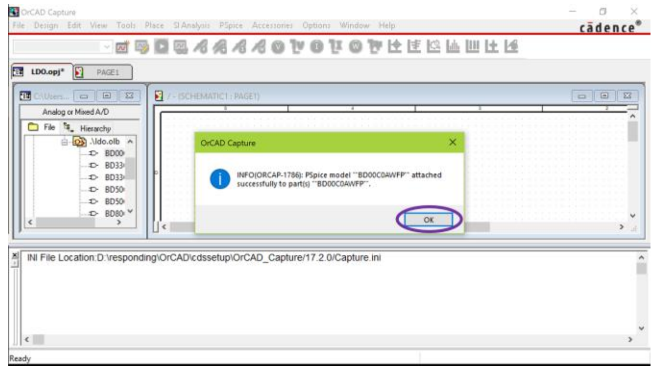

Figure 18. Notification of completed registration

4.1.2 When using other simulators

When considering SPICE models for use in other simulating environments, the SPICE models obtained as described in Chapter 3 cannot be used since they are encrypted for PSpice. Please contact us if you need general-purpose SPICE models.

4.2 Configuration of elements for simulations

To keep this chapter applicable to all the simulators, the methods for inserting or wiring elements that depend on individual simulators are not described in this chapter. In addition, the symbols used here are of general purpose.

Configure the IC and the passive and active elements to be used. Main elements required for the LDO simulations are the passive elements (resistor and capacitor) and the active elements (constant voltage source, PWL voltage source, constant current source, PWL current source, and ground).

A simple example is shown below (Figure 19).

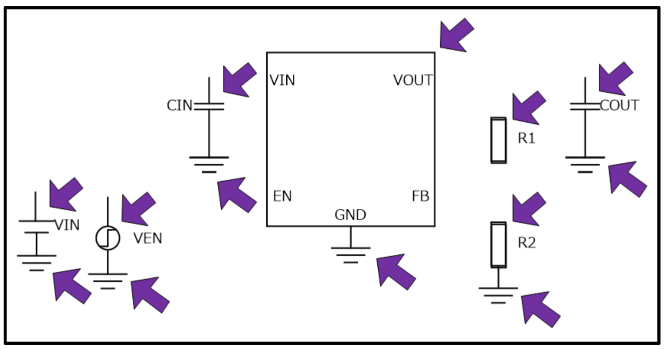

Figure 19. State where the IC and the passive and active elements have been configured

4.3 Connection of circuit diagram

Connect the IC and the elements once their configuration is completed. The IC and each element have pins, and every pin must be either connected or terminated. When a pin is open, it may cause an unintended operation.

The result of making the connections in Figure 19 is shown below (Figure 20).

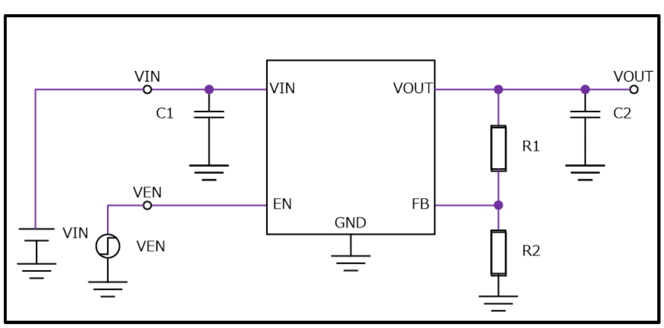

Figure 20. State where the connections have been made

4.4 Entry of various properties

For the transient analyses where rise signals are input into EN, connect a PWL voltage source that can create PWL signals on the EN pin of the IC. The voltage of PWL voltage source can be set for each point of time without restriction. Accordingly, when using a PWL voltage source, set the voltage required at a specified time, for example, 5 V at 10 µs. In the next example, the output of 5 V is produced from the input of 12 V. Capacitors of 10 µF are connected with the input and output in order to input the rise signals into EN (Figure 21).

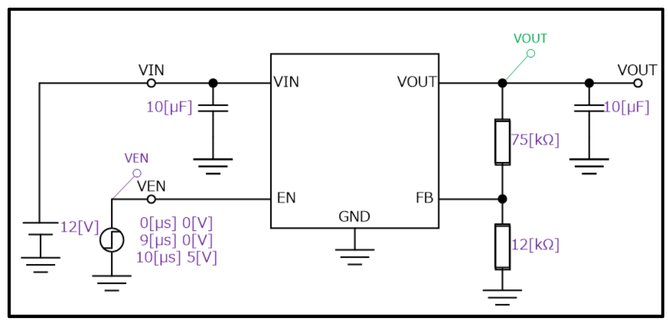

Figure 21. State where the property settings are completed

5. Simulation

Although methods vary with individual simulators, results are mainly obtained by executing (running) the transient analyses. If the configuration and connections have been properly made and the property setting values for each element are correct, the following results should be obtained. When VEN rises, VOUT starts rising generally with a delay and remains constant after reaching the set voltage (Figure 22).

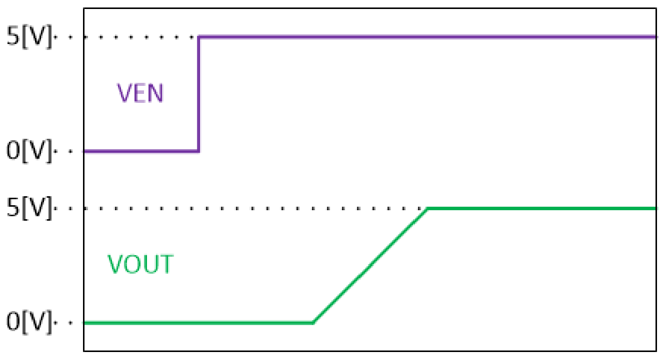

Figure 22. Example of a startup waveform by EN

For other characteristics, refer to the modeling reports that are enclosed with the SPICE models.

Notes

1) The information contained herein is subject to change without notice.

2) Before you use our Products, please contact our sales representative and verify the latest specifications :

3) Although ROHM is continuously working to improve product reliability and quality, semiconductors can break down and malfunction due to various factors.
Therefore, in order to prevent personal injury or fire arising from failure, please take safety measures such as complying with the derating characteristics, implementing redundant and fire prevention designs, and utilizing backups and fail-safe procedures. ROHM shall have no responsibility for any damages arising out of the use of our Poducts beyond the rating specified by ROHM.

4) Examples of application circuits, circuit constants and any other information contained herein are provided only to illustrate the standard usage and operations of the Products. The peripheral conditions must be taken into account when designing circuits for mass production.

5) The technical information specified herein is intended only to show the typical functions of and examples of application circuits for the Products. ROHM does not grant you, explicitly or implicitly, any license to use or exercise intellectual property or other rights held by ROHM or any other parties. ROHM shall have no responsibility whatsoever for any dispute arising out of the use of such technical information.

6) The Products specified in this document are not designed to be radiation tolerant.

7) For use of our Products in applications requiring a high degree of reliability (as exemplified below), please contact and consult with a ROHM representative : transportation equipment (i.e. cars, ships, trains), primary communication equipment, traffic lights, fire/crime prevention, safety equipment, medical systems, servers, solar cells, and power transmission systems.

8) Do not use our Products in applications requiring extremely high reliability, such as aerospace equipment, nuclear power control systems, and submarine repeaters.

9) ROHM shall have no responsibility for any damages or injury arising from non-compliance with the recommended usage conditions and specifications contained herein.

10) ROHM has used reasonable care to ensure the accuracy of the information contained in this document. However, ROHM does not warrants that such information is error-free, and ROHM shall have no responsibility for any damages arising from any inaccuracy or misprint of such information.

11) Please use the Products in accordance with any applicable environmental laws and regulations, such as the RoHS Directive. For more details, including RoHS compatibility, please contact a ROHM sales office. ROHM shall have no responsibility for any damages or losses resulting non-compliance with any applicable laws or regulations.

12) When providing our Products and technologies contained in this document to other countries, you must abide by the procedures and provisions stipulated in all applicable export laws and regulations, including without limitation the US Export Administration Regulations and the Foreign Exchange and Foreign Trade Act.

13) This document, in part or in whole, may not be reprinted or reproduced without prior consent of ROHM.

ROHM SEMICONDUCTOR

Thank you for your accessing to ROHM product informations.
More detail product informations and catalogs are available, please contact us.

ROHM Customer Support System

http://www.rohm.com/contact/