

## Using Analog Devices Hot Swap Controller Simulation Models

### FEATURES

SIMetrix-/SIMPLIS-compatible schematic files  
 SPICE-based simulator behavior model

### APPLICATIONS

Simulate board level design  
 Verify integrity of circuit designs  
 Predict circuit behavior  
 Current-limit control  
 Sense voltage adjustment  
 TIMER pin  
 Soft star  
 Current foldback  
 Severe overcurrent protection  
 Start-up current limit

### SOFTWARE REQUIREMENTS

Windows XP, Windows Vista, Windows 7  
 SIMetrix/SIMPLIS (free version), advanced software suite to efficiently design power electronics circuits

### ONLINE RESOURCES

#### Documents

[ADM1275 data sheet](#)  
[ADM1276 data sheet](#)  
[ADM1075 data sheet](#)  
[ADM1170 data sheet](#)  
[ADM1171 data sheet](#)  
[ADM1172 data sheet](#)

#### Software

Analog Devices hot swap controller simulation models  
 (download from the product Web page)

### GENERAL DESCRIPTION

Analog Devices, Inc., hot swap controllers allow circuit boards to be inserted onto a live backplane in a controlled fashion without interrupting system operation.

Hot swap controller simulation models, in the form of SIMetrix-/SIMPLIS-compatible schematic files, are available for download on applicable product Web pages. These simulation models assist with the board level design of Analog Devices hot swap controllers, allowing you to check the integrity of the circuit and to predict circuit behavior. The behavior model is based on the Simulation Program with Integrated Circuit Emphasis (SPICE) simulator. The model reproduces most of the important analog behavior of the hot swap controllers, including current-limit control, sense voltage adjustment, TIMER pin, soft start, current foldback, severe overcurrent protection, and start-up current limit.

This user guide provides information about using a simulation model to simulate your design. For detailed information about each hot swap controller, refer to the appropriate device data sheet, available on [www.analog.com](http://www.analog.com). Although this guide uses the ADM1275/ADM1276 hot swap controller simulation model as an example, the techniques for installing and using the simulation software described in this guide apply to all Analog Devices hot swap controller simulation models. Other simulation models can be downloaded from the applicable product Web page.

### MAIN SCHEMATIC OF THE ADM1275/ADM1276 HOT SWAP CONTROLLER SIMULATION MODEL

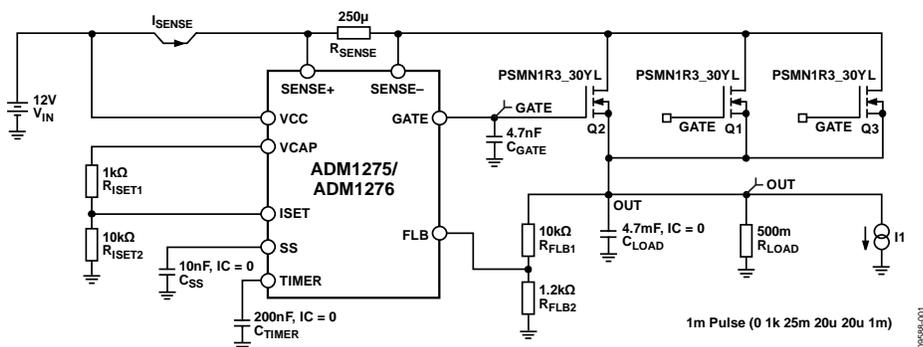


Figure 1.

**TABLE OF CONTENTS**

Features .....	1	Getting Started.....	3
Applications.....	1	Software Installation Procedures.....	3
Software Requirements .....	1	Hot Swap Controller Circuitry .....	4
Online Resources.....	1	How to Use the Software .....	5
General Description .....	1	Overview of the Graphical User Interface .....	5
Main Schematic of the ADM1275/ADM1276 Hot Swap Controller Simulation Model.....	1	Editing the Circuit.....	6
Revision History .....	2	Importing Models .....	6
		Running a Simulation .....	7

**REVISION HISTORY**

**5/13—Rev. 0 to Rev. A**

Changed ADM1275/ADM1276 Simulation Model to Serve as an Example of How All Analog Devices Hot Swap Controller Simulation Models Work .....	Universal
Reorganized Layout.....	Universal
Added Features, Applications, Software Requirements, and Online Resources Sections; Changes to Title and General Description.....	1
Changes to Installing the SIMetrix/SIMPLIS Software Suite Section .....	3
Changes to Hot Swap Controller Circuitry Section.....	4
Changes to Editing the Circuit Section .....	6
Changes to Importing Models Section .....	6
Changes to Disclaimer .....	8

**4/11—Revision 0: Initial Version**

## GETTING STARTED

### SOFTWARE INSTALLATION PROCEDURES

#### ***Installing the SIMetrix/SIMPLIS Software Suite***

Before using an Analog Devices hot swap controller simulation model, you must download the SIMetrix/SIMPLIS software suite. You can download a free version of the tool from the SIMetrix Technologies website.

SIMetrix/SIMPLIS is an advanced software suite that enables efficient design of power electronics circuits.

SIMetrix is a mixed-mode circuit simulation package designed for electronics engineers. It includes an enhanced SPICE simulator, schematic editor, and waveform viewer in a unified environment.

SIMPLIS (Simulation for Piecewise Linear System) is a circuit simulator designed for rapid modeling of switching power systems. SIMPLIS is bundled with the premium version of SIMetrix.

This user guide provides the basic knowledge required to run an Analog Devices hot swap controller simulation model. For more information about the SIMetrix/SIMPLIS tools, see the user manual and reference on the SIMetrix Technologies website.

#### ***Installing the Analog Devices Hot Swap Controller Simulation Model Schematic File***

Analog Devices hot swap controller simulation models can be downloaded from the product Web page.

After the SIMetrix/SIMPLIS software is installed, you can open the simulation model schematic file using the software. This file has the suffix **.sxsch** (see Figure 4).

## HOT SWAP CONTROLLER CIRCUITRY

Analog Devices hot swap controllers allow circuit boards to be inserted onto a live backplane in a controlled fashion without interrupting system operation.

The load current is measured using an internal current sense amplifier that measures the voltage across a sense resistor that is placed in the power path between two current sense pins. The hot swap controller starts to regulate the current after the voltage drop across the sense resistor reaches the circuit breaker voltage. You can adjust the hot swap current limit by adjusting the sense resistor value and/or the circuit breaker voltage.

An Analog Devices hot swap controller limits the current through the sense resistor via the GATE pin by controlling the gate voltage of an external N-channel FET that is placed in the power path. The sense voltage—and, therefore, the load current—is maintained below the preset current limit. The hot swap controller protects the external FET by limiting the time that the FET remains on

while the current is at its maximum value. This current-limit time is set by the choice of capacitor connected to the TIMER pin.

In addition, a foldback resistor network can be used to actively lower the current limit as the voltage across the FET is increased. This helps to maintain constant power in the FET and facilitates maintaining operation within the safe operating area (SOA).

The simulation models for the Analog Devices hot swap controllers are designed for maximum visual simplicity. You can easily identify and edit the external components, making it very straightforward to build and simulate the design. See the Quick Instructions in the **Schematic Editor** window for information about each component (see Figure 4).

The internal circuitry of the downloaded simulation model is included in the bottom section of the **Schematic Editor** window. Do not modify the components in this section unless you understand them fully.

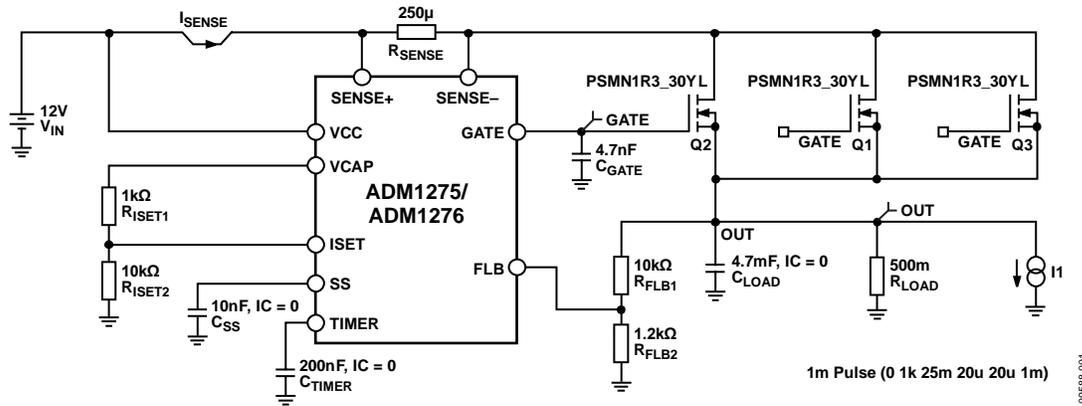


Figure 2. ADM1275/ADM1276 Simulation Model Schematic

09588-001



## EDITING THE CIRCUIT

In the **Schematic Editor** window, you can easily edit various components to form a circuit (see Figure 4).

To add a component, click **Place** in the menu bar and then choose the component from the menu.

To edit passive components (resistors, capacitors, and so on), double-click the appropriate symbol in the schematic. For more complex semiconductor components (for example, transistors and FETs), choose an existing model from the tool library as follows:

1. Choose **Place > From Model Library**, or double-click the appropriate symbol.
2. Choose a model from the library (see Figure 6).

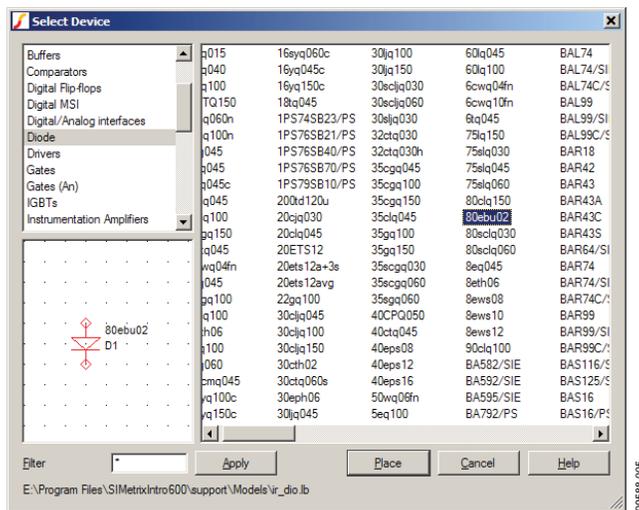


Figure 6. Model Library

If the desired model is not in the library, you can generally find the SPICE model of the component on the manufacturer's website and import it into SIMetrix (see the Importing Models section).

You can also import a customized model using the .MODEL statement. This very powerful statement allows you to change model parameters and create your own models.

### Example of Adding a Diode

For example, to include a diode with a reverse breakdown voltage of 22 V and a breakdown current of 3 mA, follow these steps:

1. Place a diode symbol in the schematic and change its name to a name of your choice, for example, **mydiode**. (To change a symbol name, right-click the symbol and choose **Edit/Add properties**).
2. In the **Schematic Editor** window, press **F11** to open the **Simulator Command** window (also known as the F11 window) and type this line:

```
.Model mydiode D(BV=22 IBV=0.003)
```

For more information about SPICE command syntax, see the simulator reference manual on the SIMetrix Technologies website.

To view or edit the netlist of the circuit, open the **Command Shell** window and click **Simulator** in the menu bar (see Figure 3).

## IMPORTING MODELS

SIMetrix has an extensive range of component models in its libraries, but, if desired, you can add an external model to your simulation.

To add an external model to your simulation, download the SPICE models (preferably Level 3 models) from the manufacturer's website. Depending on the manufacturer, the model file can have different extensions, for example, .lib, .mod, and .cir. If you open the model file in a text editor, you can see that the file contains one of the following:

- Single .MODEL statement (for example, see the statement in Step 2 of the Example of Adding a Diode section within the Editing the Circuit section). This type of model file is common for semiprimitive components such as diodes.
- A .SUBCKT statement followed by several .MODEL statements. This type of file creates a subcircuit within the model, which consists of different submodels. This type of model file is common for complex components such as FETs.

To import a model into the software, do one of the following:

- Open the **Command Shell** window (see Figure 3). From the menu bar, choose **File > Model Library > Add/Remove Libraries** and select the model file from the file explorer.
- Drag the model file into the **Schematic Editor** window.

In either case, the newly added model can be found in the **Schematic Editor** window by choosing **Place > From Model Library**. The model should be in the **Recently Added Models** list (see Figure 7).

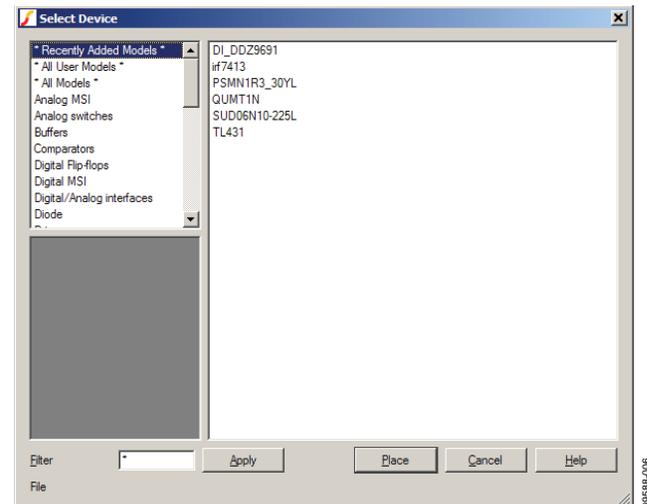


Figure 7. List of Recently Added Models

Occasionally, the library file provided by the manufacturer contains models of multiple components. In this case, a newly imported model does not appear in the **Recently Added Models** list, and you must add the model manually.

To add a model manually, follow these steps:

1. Place a component symbol in the schematic. For example, to add an N-channel FET, place an NMOS symbol in the schematic.
2. Change the name of the new symbol to the model name defined in the library file. (Right-click the symbol and choose **Edit/Add properties**.)
  - If the file contains only a single .MODEL statement, the component name is the word directly following the .MODEL statement.
  - If the file contains a .SUBCKT statement, the model name is the word directly following the .SUBCKT statement.

Some manufacturers, notably Infineon, have multiple level modeling for the same component, providing different modeling complexity. One example is the two extra thermal nodes (Tj and Tcase) for the MOSFET Level 3 models to simulate temperature effects.

For example, to import the model for the Infineon FET IPB065N15N3, follow these steps:

1. Download the **OptiMOS3\_150V\_200V\_250V.lib** file from the Infineon website.
2. Import this file into the tool.
3. Open the library file using a text editor.

In the library file, you can see two .SUBCKT statements for the IPB065N15N3:

```
.SUBCKT IPB065N15N3 drain gate source Tj
Tcase PARAMS: ...
```

```
.SUBCKT IPB065N15N3_L1 drain gate source
PARAMS: ...
```

To import the standard 3-pin MOSFET model,

1. Place a standard MOSFET symbol in the **Schematic Editor** window.
2. Rename the symbol using the name of the Level 1 model, IPB065N15N3\_L1 (see Figure 8).



Figure 8. 3-Pin and 5-Pin N-Channel FET Symbols

To import the 5-pin MOSFET model, place a 5-pin N-channel FET symbol in the **Schematic Editor** window as follows:

1. Choose **Place > From Symbol Library**.
2. In the symbol tree, choose **Semiconductors > MOSFETs (Discrete) > N-channel - thermal - 5 terminal** (see Figure 9).
3. After placing the symbol, rename it using the name of the Level 3 model, IPB065N15N3 (see Figure 8).

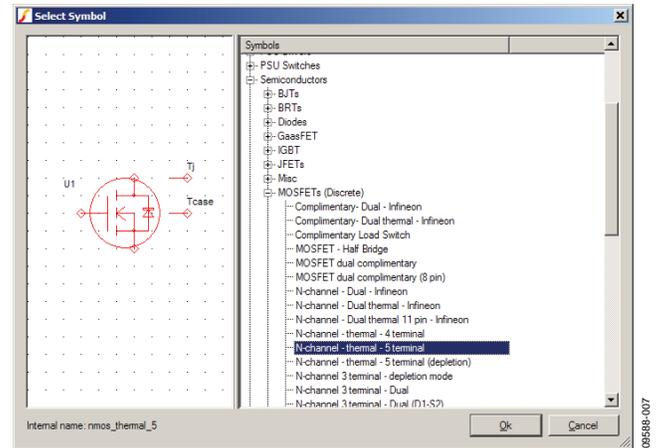


Figure 9. Symbol Library

For information about how to use the two thermal nodes, Tj and Tcase, see the documentation supplied with the Infineon models.

## RUNNING A SIMULATION

Before you run a simulation, open the **Schematic Editor** window and choose **Simulator > Choose Analysis** to set up the simulation parameters. For Analog Devices hot swap controller models, the transient analysis is most pertinent; therefore, in the **Choose Analysis** dialog box, set the parameters on the **Transient** tab (see Figure 10). On the **Transient** tab, you can set the duration of the transient simulation, the time to start the output of the data, the size of the time step, and other parameters.

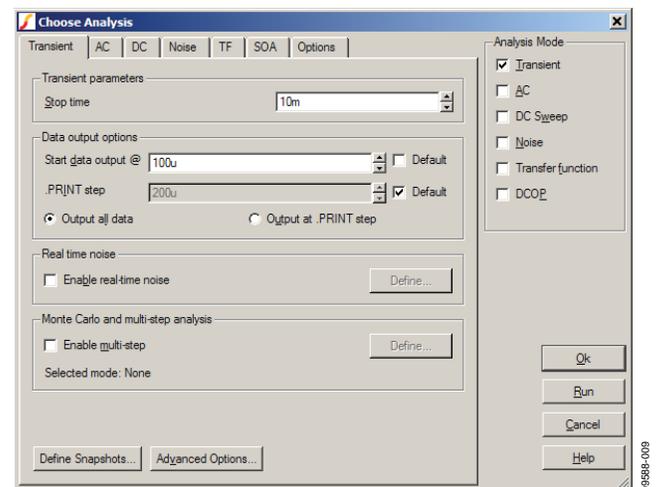


Figure 10. Choose Analysis Dialog Box, Transient Tab

After the appropriate parameters are set, run the simulation from the **Schematic Editor** window by choosing **Simulator > Run** or by pressing the **F9** key.

When the simulation finishes, you can view the simulation results by probing the voltage and current on the schematic (click **Probe** in the menu bar, select a probe option, and click the point of interest in the schematic). The resulting waveform should be visible in the **Waveform Viewer** window (see Figure 5).

Alternatively, you can place a fixed probe in the circuit by choosing **Place > Probe**. The results from the fixed probe are

automatically displayed in the waveform viewer window while the simulation is running.

To configure the plot settings of a fixed probe, double-click the probe. The **Edit Probe** dialog box opens (see Figure 11).

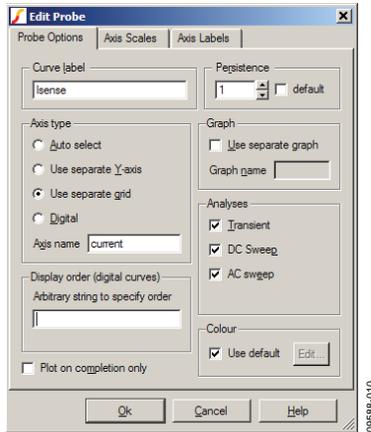


Figure 11. Edit Probe Dialog Box

In the **Edit Probe** dialog box, you can configure a fixed probe as follows (see Figure 11):

1. Rename the curve.
2. Specify the type of axes generated.
3. Set the number of saved waveforms from previous runs.
4. Specify whether to use separate grids or graphs.

#### DISCLAIMER

ALTHOUGH ANALOG DEVICES HAS MADE SUBSTANTIAL EFFORT TO ENSURE THESE SIMULATION MODELS PROVIDE ACCURATE RESULTS, THE COMPLEXITY, ESTIMATES USED, COMPONENT TOLERANCES, PCB LAYOUT VARIANCES, AND NONLINEARITIES PRECLUDE ABSOLUTE ACCURACY GUARANTEES. THE SIMULATION MODELS ARE PROVIDED "AS IS" AND ANALOG DEVICES MAKES NO WARRANTIES OR REPRESENTATIONS OF ANY KIND WITH RESPECT TO THEM. ANALOG DEVICES SPECIFICALLY DISCLAIMS ANY REPRESENTATIONS, ENDORSEMENTS, GUARANTEES, OR WARRANTIES, EXPRESS OR IMPLIED, RELATED TO THE SIMULATION MODELS INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTY OF MERCHANTABILITY, TITLE, FITNESS FOR A PARTICULAR PURPOSE OR NONINFRINGEMENT OF INTELLECTUAL PROPERTY RIGHTS. IN NO EVENT WILL ANALOG DEVICES BE LIABLE FOR ANY INCIDENTAL, SPECIAL, INDIRECT, OR CONSEQUENTIAL DAMAGES RESULTING FROM OR RELATED TO YOUR USE OF THE SIMULATION MODELS. IN NO EVENT SHALL ANALOG DEVICES' CUMULATIVE LIABILITY FOR ANY AND ALL CAUSES WHATSOEVER, REGARDLESS OF THE FORM OF ANY CLAIMS OR ACTIONS, EXCEED THE GREATER OF (1) ONE HUNDRED U.S. DOLLARS (\$100.00 U.S.).

These simulation models are the property of Analog Devices. They are design aids for use with Analog Devices' hot swap controllers. They may not be copied, transferred, or used for other purposes without express written approval of Analog Devices.

©2011–2013 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners.  
UG09588-0-5/13(A)



www.analog.com