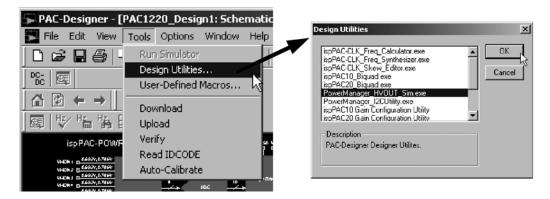**Lattice** ®
Semiconductor
Corporation

HVOUT Simulator Calculates The Actual Power Supply Ramp Rate

Several Power Manager devices from Lattice incorporate charge-pump gate-driver outputs known as HVOUT pins. The design utility "PowerManager_HVOUT_Sim.exe" (HVOUT_Sim) is included in PAC-Designer® to simulate the behavior of various MOSFET circuits when driven by HVOUT pins. This utility takes into consideration the following: programmable voltage and current settings of the HVOUT pins, the supply voltage, additional gate capacitance, and load value. It also uses ordinary data sheet parameters instead of a SPICE model for simulating the MOSFET. This application note describes in detail how to use the HVOUT_Sim utility to determine the ramp rate in the user's circuit.

The Design Utilities

PAC-Designer contains several design utilities for various devices to support customers with design related issues that are not directly addressed by the normal windows of PAC-Designer. To start the HVOUT_Sim, use the menu item **Tools -> Design Utilities...** shown in Figure 1 to open the **Design Utilities Tool Box** dialog.
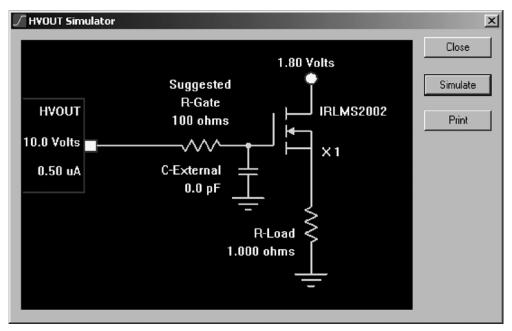
Figure 1. PAC-Designer Utilities Menu Opens Design Utilities Tool Box

From the Design Utilities Tool Box, Select the **PowerManager_HVOUT_Sim.exe** utility and click the **OK** button to start the program.

The HVOUT_Sim is a dialog-based application that represents a single HVOUT driving an N-Channel MOSFET that switches a supply voltage to a load as shown in Figure 2. A gate resistor value is suggested based on the MOSFET parameters.

Figure 2. HVOUT Simulator Interface

Making Changes to the Circuit

The HVOUT_Sim interface is similar to PAC-Designer in that the user can double-click on "hot-spots" to edit values in the schematic. The cursor changes to the arrow with brackets, as shown in Figure 3, when the mouse is over a "hot-spot." By double-clicking here, a supporting dialog box will open that can be used to modify the design.

Figure 3. Hot-Spot Cursor

HVOUT Settings

Double-clicking on the **Voltage** section of the HVOUT block in the schematic will open the **HVOUT Voltage** dialog box (see Figure 4) in which the voltage can be set from 6V to 12V. Likewise, double-clicking on the current section of the HVOUT block in the schematic will open the **HVOUT Current** dialog box (see Figure 5) in which the constant current setting can be set from 0.5µA up to 400µA.
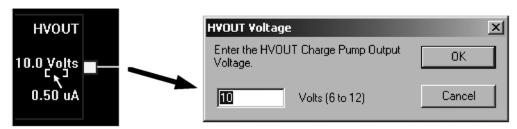
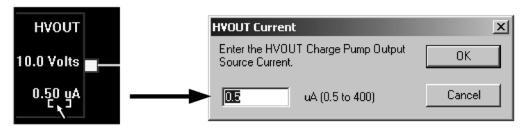*Figure 4. Changing the HVOUT Voltage Setting*

Figure 5. Changing the HVOUT Current Setting

Note that the HVOUT current setting can be set to values that exceed the maximum setting for a single Power Manager HVOUT pin. For example, the maximum setting for the HVOUT for an ispPAC-POWR1220AT8 device is only 100µA but the limit in the dialog is 400µA. This allows designers to simulate designs that tie multiple HVOUTS together to achieve faster ramp rates.
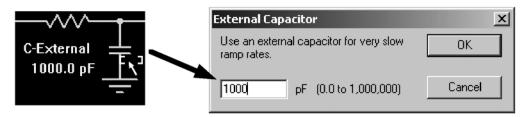
R-Gate Value

To prevent parasitic oscillations in the power MOSFET, it is a common practice to place a small resistor in series with the gate pin. By placing this resistor as physically close to the gate pin as possible the potential for source-to-gate feedback is greatly reduced. The recommended value for this series gate resistor a minimum that is calculated from timing values for the MOSFET. Larger values (up to double) can be used with no noticeable change in the turn on timing.

External Capacitor Value

The default value for the external capacitor is zero because it is not required for typical power supply control when using HVOUTs and MOSFETs. However, an external capacitor can be added to the circuit as shown to either slow down a ramp rate or to prevent ramp rate variations that may result from changes in MOSFET parameters, lots, or vendors. To set a value for the external cap, double-click on the symbol or text to obtain the **External Capacitor** dialog box shown in Figure 6. Enter a value in picofarads and click the **OK** button to set a value for the external capacitor.
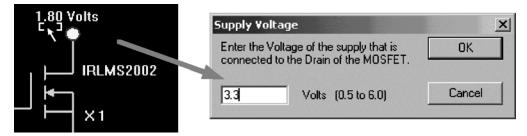
Figure 6. Changing the External Capacitor Value to 1000pF

Power Supply Voltage

The value of the power supply that is connected to the drain of the MOSFET should be selected to represent the supply being used in the circuit driving the load. Double-click on the supply value or symbol to open the **Supply Voltage** dialog box that is shown in Figure 7. Enter a voltage value and click the **OK** button to change the value in the schematic.

Figure 7. Changing the Power Supply Value to 3.3V

Load Resistor

The HVOUT_Sim uses a single resistor to represent the lumped load at the source of the MOSFET. The value of this resistor should be set to establish a realistic Drain-to-Source current. This can be either a "steady-state" value or an "impulse" value depending on the type of simulation that is desired. To change the value of the load resistor, double-click on the text or symbol to activate the **Load Resistance** dialog box that is shown in Figure 8. Enter a resistance value and click the **OK** button to set the value in the schematic.

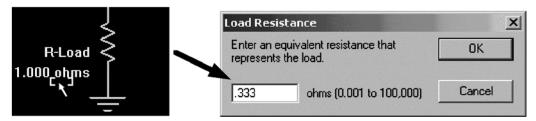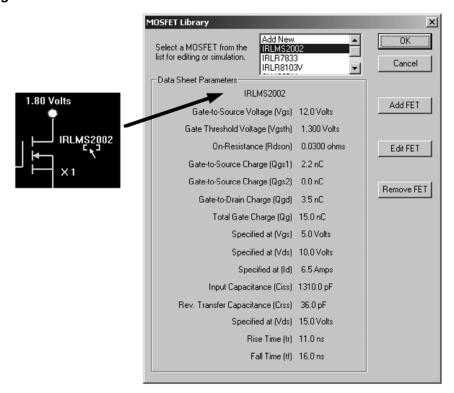*Figure 8. Changing the Load Value to 0.333 Ohms*
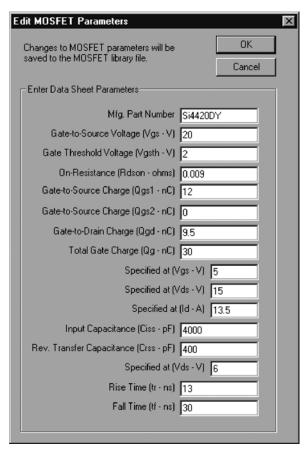

Figure 9. Selecting the MOSFET

Selecting the MOSFET

The MOSFET to be used in the simulation can be selected by double-clicking on the symbol or part number text to bring up the **MOSFET Library** dialog box that is shown in Figure 9. The **MOSFET Library** dialog can be used to simply change the MOSFET that will be used in the current simulation, or the dialog can be used to maintain the MOSFET Library. To select the MOSFET, highlight the desired MOSFET from the list and click the OK button. To assist in the selection of the MOSFET, the data sheet parameters are updated to reflect the selected device.

Adding, Deleting and Modifying MOSFETs in the MOSFET Library

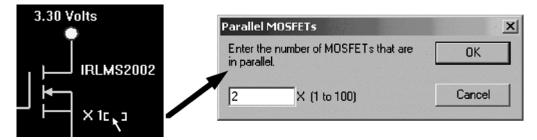*Figure 10. Editing the MOSFET Data Sheet Parameters*

The **MOSFET Library** dialog can also be used to add a new MOSFET, edit an existing MOSFET, or to delete an unused MOSFET. In this way a designer can customize the MOSFET Library to suit their specific needs. Click the **Add FET** button to add a MOSFET to the list and click the **Edit FET** button to modify a specific parameter of an existing MOSFET. In either case, the **Edit MOSFET Parameters** dialog, shown in Figure 10, will be displayed.

The HVOUT_Sim utility is a simulator that uses common line items from a MOSFET data sheet in order to simulate the circuit instead of a SPICE model. Thus, the accuracy of the simulation will be dependant on correctly entering the values from the data sheet. Note that some MOSFET data sheets do not have separate entries for Qgs1 and Qgs2. In this case enter the Qgs value for Qgs1 and zero for Qgs2 (as shown in Figure 10). Also note that the gate-charge parameters (Qgs1, Qgs2, Qgd and Qg) are specified at a specific Vgs, Vds, and Id values. Likewise the capacitance values (Ciss and Crss) are specified at a specific Vds value. That is because both the gate-charge and input capacitance parameters vary as a function of applied voltage and load. HVOUT_Sim fine-tunes these parameters based on the design for accurate simulations.

Multiple MOSFETs

In order to reduce on-resistance of the MOSFET, a common practice is to parallel multiple MOSFETs. By default the HVOUT_Sim Utility uses only a single MOSFET to do the supply switching. To place additional MOSFETs in parallel, double-click on the **X1** in the schematic and change the number in the **Parallel MOSFETs** dialog box as shown in Figure 11.

Figure 11. Adding a Second MOSFET in Parallel

Running the Simulation

After all the circuit parameters have been set, running the simulation is as simple as clicking the **Simulate** button. The results are displayed in a "Virtual Scope". The actual ramp rate voltage is shown along with the gate voltage waveform, (see Figure 12).

Figure 12. Simulation Results are Displayed in a Virtual Oscilloscope

In this window, one can enable two cursors to measure time and voltage. One cursor is driven from the left mouse button, and the other from the right mouse button. The difference between the two is computed and displayed in the lower right corner. There is also a time scale adjustment to zoom in for more detail. The trigger reference is time-zero when the Power Manager enables the HVOUT and the ground reference is fixed at one division up from the bottom.

Both the gate voltage and the source voltage are plotted in the output window as a function of time. Note that in a real design a normal oscilloscope probe cannot be placed on the gate or HVOUT pin without significantly loading the node down. This is certainly true when the HVOUT current setting is less than 10 microamps.

Buttons are provided to either print or export the results of the simulation. The export button brings up a typical file navigation dialog so the results can be placed into a folder of choice. The format of the exported file is comma separated variables (CSV) and can be imported into any number of spreadsheet programs.

Summary

The HVOUT_Sim software utility is very useful for simulating ramp rates for designs based around Power Manager HVOUT output pins and power MOSFET transistors. The user interface is easy to learn and use, yet flexible enough to cover a wide range of designs. The simulation is based upon standard MOSFET data sheet parameters rather than a SPICE model so that designers can evaluate designs much quicker and without requiring a SPICE simulator. The results are computed quickly and presented in a virtual oscilloscope for analysis. Print and export functions are provided to document and compare results. This utility is another of many that Lattice offers in an effort to provide valuable support tools for its sophisticated programmable devices.

Related Literature

- ispPAC-POWR1208 Data Sheet
- ispPAC-POWR1208P1 Data Sheet
- ispPAC-POWR1220AT8 Data Sheet
- AN6043 Using the ispPAC-POWR1208 MOSFET Driver Outputs
- AN6048 Using Power MOSFETs with the ispPAC-POWR1208

Technical Support Assistance

Hotline: 1-800-LATTICE (North America)
          +1-503-268-8001 (Outside North America)

e-mail: isppacs@latticesemi.com

Internet: www.latticesemi.com