PSIM Version 6.0

for Windows 98/NT/2000/XP

1. Introduction

PSIM is a computer simulation package specially designed for the analysis and design of power converter circuits and systems. With efficient PSIM simulation environment, a wide range of power converter circuits and control techniques can be easily implemented and analyzed.

This document describes how to set up and simulate a circuit, and present trouble-shooting techniques. For installation instruction, please refer to Installation Guide.

2. Simulating a Circuit

To simulate the sample circuit "chop-1q.sch":

- Start PSIM.
- Load the schematic file into PSIM.
- Click on the **Run Simulation** icon in the toolbar, or choose **Run Simulation** from the **Simulate** menu, to start the simulation.
- Click on the **Run SIMVIEW** icon in the toolbar, or choose **Run SIMVIEW** from the **Simulate** menu, to start SIMVIEW.
- In SIMVIEW, select the curves available for display.
- You may modify the circuit in the PSIM schematic program, run PSIM again, and reload the new simulation results into SIMVIEW by simply clicking on the **Run SIMVIEW** icon in PSIM.

3. Troubleshooting

The following is a list of problems that one may encounter when running PSIM, and possible solutions.

Problem: Can not start PSIM or SIMVIEW

Error message: Sorry: no Hardlock with the correct ID found!

Possible causes of the problem and the solutions:

- **Cause:** The hardlock is not connected to the PC parallel port.
- **Solution:** Connect the hardlock directly to the PC parallel port.

- **Cause:** The hardlock for Version 6.0 is different from the hardlocks for the earlier versions, and the hardlock you are using is not the right one.
- **Solution:** Make sure that you are using the right hardlock.

- **Cause:** The hardlock driver is not installed.
- **Solution:** Run "hldrv32.exe" in the PSIM directory to install the hardlock driver.

- **Cause:** You are using the PSIM network version, but the hardlock driver is not properly installed. This can happen if you are not the system administrator but just an ordinary user. Even if you did run "hldrv32.exe", but since you do not have the writing permission to the Windows system directory, the hardlock driver files are not copied to the appropriate directory.
- **Solution:** Ask your system administrator to do the installation, including the hardlock driver installation.

- **Cause:** You are using the PSIM network version. The hardlock is connected and the hardlock driver is installed, but the hardlock server is not running.
- **Solution:** Ask your system administrator to run the hardlock server.

- **Cause:** You are using the PSIM network version. The hardlock is connected, the hardlock driver is installed, and the hardlock server is running. But under certain network settings and configurations, the hardlock program can not find the key over the network. In this case, you need to set two environment variables HL_SEARCH and HLS_IPADDR.
- **Solution:** Set the two environment variables HL_SEARCH and HLS_IPADDR as follows:

  HL_SEARCH = 378p,3BCp,IP

  HLS_IPADDR = xxx.xxx.xxx.xxx [IP address of the PC that runs the hardlock server]

  For example, to set the environment variable HL_SEARCH in Windows 2000:
  - Go to **Control Panel** and double click on **System**.
  - Click on **Advanced** tab. On this page, click on **Environment Variables…**.
  - Under **System variables**, click on **New…**.
  - Set **Variable Name** as: HL_SEARCH, and **Variable Value** as: 378p,3BCp,IP.

Problem: You purchased PSIM network version for, say, 3 users, but only 1 or 2 persons can use PSIM simultaneously.

Error message: Error: To many users on the network!

Possible causes of the problem and the solutions:

- **Cause:** You are running PSIM or SIMVIEW that are for 1 or 2 users.
- **Solution:** Make sure that you are running the correct PSIM and SIMVIEW files. Go to "About" in the "Help" menu, and you should see PSIM or SIMVIEW for 3 users (in this case). The number of users shown here should match the number of users that you purchased. If it does not, you are running a wrong file. Go to the PSIM directory on the network drive, and run PSIM and SIMVIEW directly from this directory.

4. Creating a New Circuit in PSIM

The following shows a step-by-step procedure to create the sample circuit "chop-1q.sch".

- Start PSIM. Click on the icon **New** in the toolbar (or select **New** from the **File** menu).
- From the **Elements** menu, go to **Sources**/**Voltage** and select **DC** for the dc voltage source. The cursor will change to the image of a dc voltage source.
- Place the source on the screen by left clicking the mouse. Now the cursor changes to another dc voltage source image. You can either click on the **Select** icon in the toolbar to get rid of the voltage source image, or simply ignore it and proceed to get the second element from the element menu.
- From the **Elements** menu, go to **Power**/**Switches** and select **IGBT**. An upright IGBT image will appear. Right click the mouse to rotate the image. In this case, you need to click 3 times to gett the orientation as required
- Repeat the same process, get diode (**DIODE**) from **Power**/**Switches**, inductor (**L**), capacitor (**C**), and resistor (**R**) from **Power**/**RLC Branches**.
- From **Other**/**Probes**, get single-ended voltage probe (**Voltage Probe**), double-ended voltage probe (**Voltage Probe (node-to-node)**), and current probe (**Current Probe**). The single-ended voltage probe is to measure the node-to-ground voltage, and the double-ended voltage probe is to measure the voltage between two nodes, and the current probe is to measure the current through a branch (in this case, the current through the inductor). Instead of using the current probe, you can also view the inductor current by simply setting the inductor current flag to 1.
- From **Power**/**Switches**, get the switch gating block **Gating Block**. There are two ways a switch is controlled. One is through a gating block. The gating block output is connected to the gate node of the switch. Refer to the on-line help (by double clicking on the gating block image and then clicking on "Help") for its usage.
- From **Other**, get the ground element **Ground**. Note that for all the elements above, you can also get them by simply clicking on the corresponding images in the element bar at the bottom of the window.
- Click on the **Wire** icon in the toolbar (or select **Wire** in the **Edit** menu). The cursor will change to the image of a pen. Left click the mouse and drag the mouse to draw a wire.
- To create the circuit on the right, you can either build the circuit as before, or copy the circuit you already built and make modifications. To copy the circuit on the left, click on the **Select** icon, left click the mouse and drag the mouse to define the region. After the region of the circuit is selected, a double line frame will appear. Click on the **Copy**, and then **Paste** icon.
- The circuit on the right uses the second way of controlling a switch, that is, with a switch controller. The input of the switch controller comes from a control circuit. The on-off switch controller used in this circuit can be obtained by selecting **On-Off Controller** from **Other**/**Switch Controllers**. The on-off switch controller behaviors like a switch base driver circuit. Unlike the real base driver circuit, however, a value of 0 will turn the transistor off, and a value of 1 will turn the switch on. The gating signal can be obtained by comparing a dc voltage source with a

triangular voltage source. The comparator can be obtained by selecting **Comparator** from **Control**, and the triangular voltage source **Triangular** from **Sources**/**Voltage**.

- Once the connection is complete, double click on the image of each element to set the parameters. For example, to set the inductor parameters, double click on the inductor image and type in the values.
- To set the simulation control parameters, click on **Simulation Control** from the **Simulate** menu and an image of a clock will appear. By double clicking on the clock image, you can specify the simulation time step, total study time, etc.