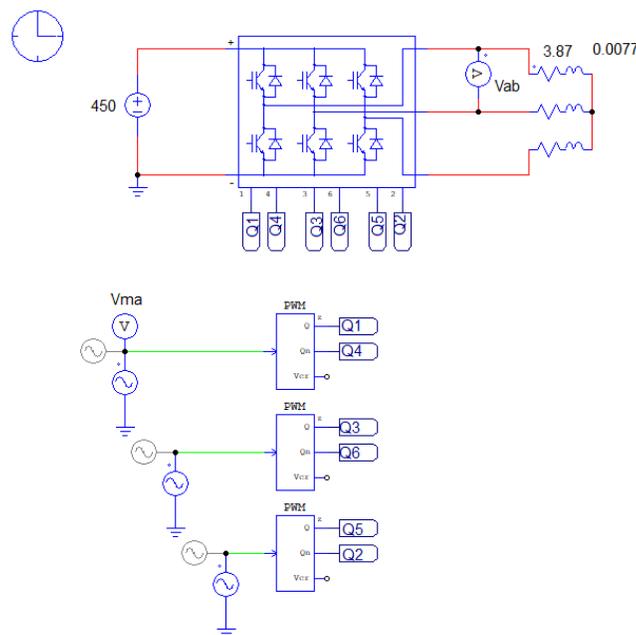# DSIM

TUTORIAL
Getting Started with DSIM

November 2019

1. Overview

DSIM is a simulation software specifically designed for power electronics. With a ground breaking simulation engine and innovative modelling approach, it achieves an unprecedented and unparallel performance. It is several orders of magnitude faster than any existing simulation software. Moreover, its ability to simulate large converter systems and at the same time switch transients is unique, and it makes it ideally suited for large scale power converter systems, high power converter systems, microgrid, and any systems that are computation intensive.

This tutorial describes how to get started with DSIM. For the purpose of illustration, we will use the 3-phase voltage source inverter example shown below. This example can be found in the folder "examples\DSIM\3-phase inverter".

2. Identifying DSIM Elements

Show DSIM Library Elements Only:

DSIM shares the same graphic user interface (GUI) as the PSIM software. The library elements in the **Elements** menu are also shared. Some elements are common to PSIM and DSIM. But some elements can be used in PSIM only, or in DSIM only. To find out which elements can be used in DSIM only, go to **Options** >> **Settings** >> **Advanced**, and check the DSIM box only. Then the Elements menu will show only the elements that are supported by DSIM. The dialog window is shown below:
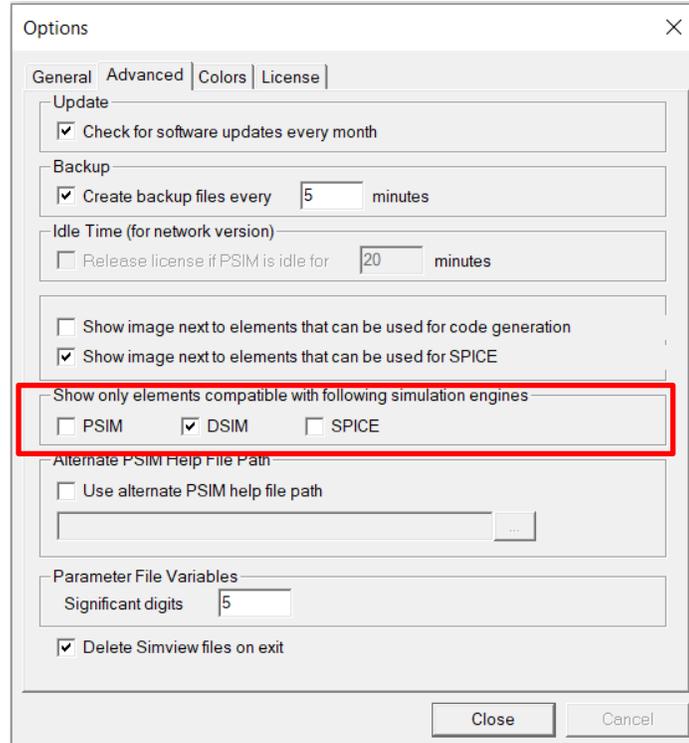
Identify Elements Not Supported by DSIM:

DSIM supports digital control in z-domain only. It does not support analog control. To find out elements that are not supported by DSIM, go to **View** >> **Check Elements**, and from the drop-down menu, select **Show only elements not supported by DSIM simulation**. Elements not supported by DSIM will be highlighted.

3. Creating a Circuit for DSIM Simulation

To create a circuit for DSIM, place elements from the Elements menu on the schematic, and connect the elements with the wire tool.

Please note that to build a power converter, one must use the switch module blocks under **Elements** >> **Power** >> **Switches** >> **Switch Modules**. One cannot use discrete switch elements, such as a single diode, IGBT, or MOSFET, to build the converter. DSIM does support bi-directional switches. But the bi-directional switch should be used for a one-time event (such as load change, open-circuit, or short-circuit). It should not be used for PWM operation as the DSIM engine is not optimized to handle it.

Also, to control the converter, one cannot use a comparator to generate the PWM signal, and an on-off switch controller to control the converter. Instead, one must use the PWM controllers in **Elements** >> **Other** >> **Switch Controllers**.

For the 3-phase inverter example, select **3-ph Inverter** from the **Switch Modules** menu as shown below.
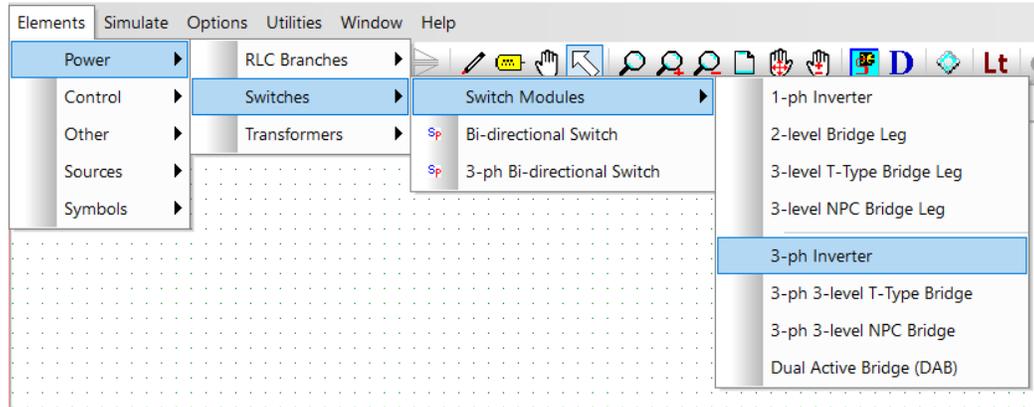
To control the inverter, select **Carrier PWM Controller (z-domain)** from the **Switch Controllers** menu, as shown below.
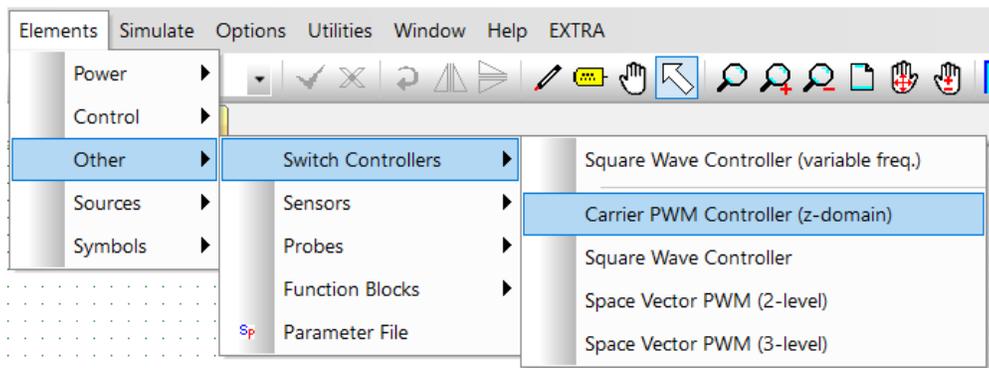

Note that the dead time in the carrier PWM controller is not supported in DSIM at the moment.

Select the 3-phase RL load from **Elements** >> **Power** >> **RLC Branches**, and dc and ac voltage sources from **Elements** >> **Sources** >> **Voltage**. Note that for the 3 sinusoidal modulation sources, either a single-ended source or a sine source with a ground can be used.

To display voltages and currents in the circuit, use the voltage probes and current probe in **Elements** >> **Other** >> **Probes**. Alternatively, for switch modules and individual RLC branches, one can set the voltage flag and current flag to 1 to display them.

Use labels to connect two nodes and to simplify schematic layout. To place a label, go to **Edit** >> **Place Label** or click on the *Place Label* icon in the toolbar.

After the circuit is completed, select **Simulate** >> **Simulation Control**, and go to the DSIM tab to define the simulation end time, maximum time step, and the relative error, as shown below.
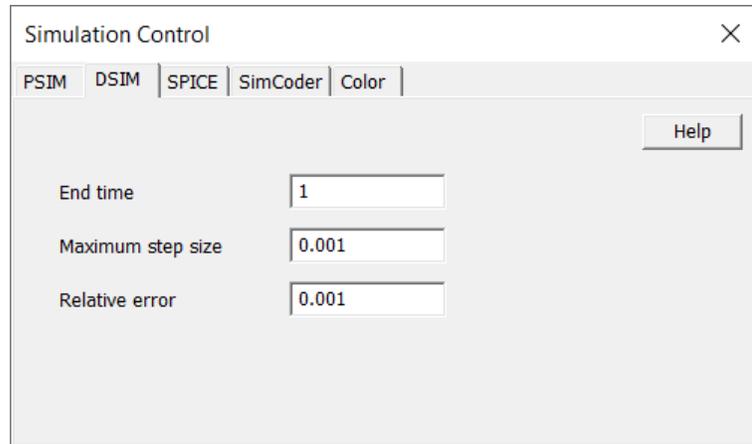
4. Running DSIM Simulation

To run DSIM simulation, select **Simulate** >> **Run DSIM Simulation** or click on the *Run DSIM Simulation* icon in the toolbar. Once the simulation is completed, the waveform display program will be launched, and a dialog window will pop up to select waveforms for display.

Below is a screenshot of the waveform display program.
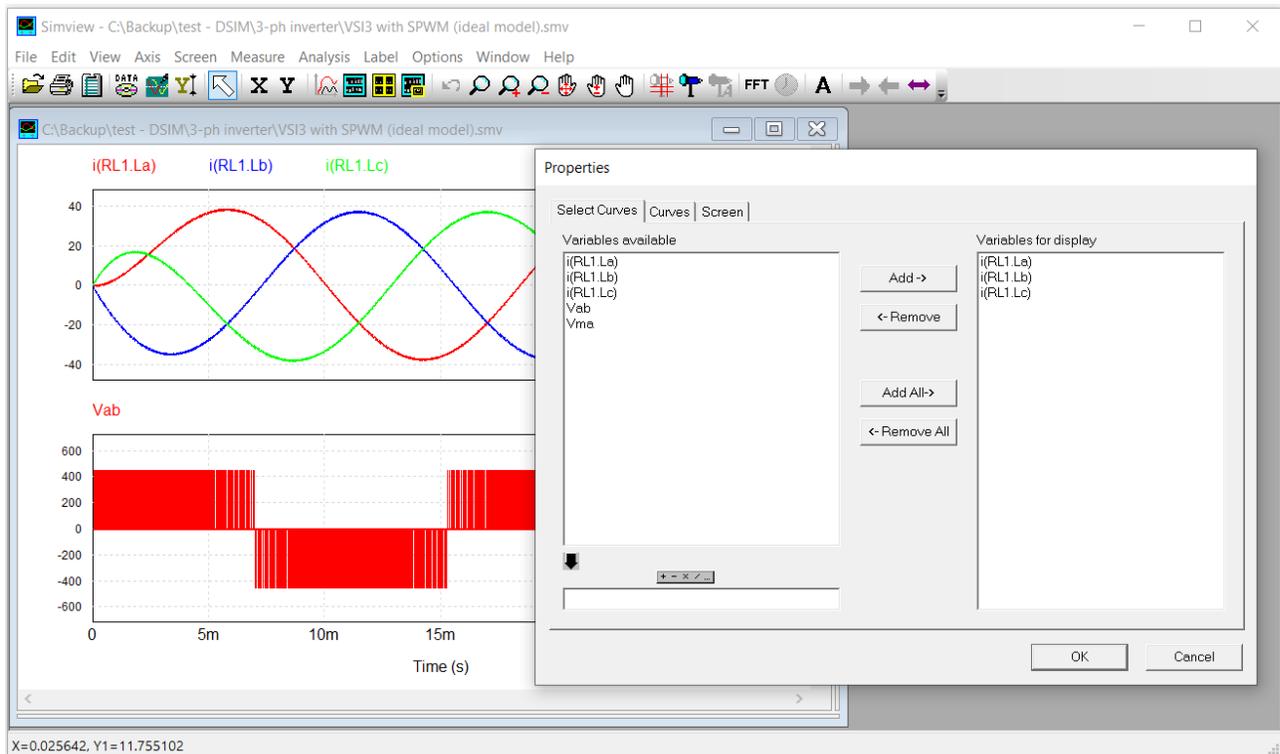
A number of examples are provided in the folder "examples\DSIM". Instructional videos are also available from the Powersim's YouTube channel at https://www.youtube.com/user/PowersimInc.