Key New Features in Version 12.0

PSIM v12.0 represents a major release with many new functions and improvements. Key new features are:

- Dual time step simulation
- Auto code generation for TI F2837x DSP
- Support of PE-Expert4’s FPGA board for multi-level and multi-modular converters
- AC analysis of switchmode circuits in LTspice
- Single-phase/3-phase conventional and enhanced Phase-Lock Loop blocks
- PMSM with spatial harmonics effect
- 6-phase PMSM
- Improved nonlinear Switched Reluctance Motor model
- Improvement in the Thermal Module
- Major improvements in SIMVIEW
- Link with RidleyWorks

Basic PSIM Improvement

Dual Time Step Simulation

PSIM uses fixed time step in simulation in the previous versions. When there is a narrow pulse in a circuit (such as in a dead time circuit), the width of the pulse limits the time step. In v12.0, this issue is addressed by having dual time steps: one larger step for normal use and another smaller step for a narrow pulse and at the moment of switching. This helps to increase accuracy and speed up simulation.

The figure below shows the waveforms when dual-step simulation is used. The simulation points in the yellow circles are with the smaller time step.
Single-phase/3-phase Conventional and Improved Phase-Lock Loops (PLL)

Built-in blocks are provided for single-phase/3-phase conventional as well as improved PLL blocks. In addition, guidelines are provided to design and tune the PLL parameters.

For a single-phase circuit, for example, one major advantage of the improved PLL block is that it does not have the double frequency ripple issue. The bottom panel of the waveforms on the left shows the input and estimated frequencies of the conventional PLL block, whereas the middle panel of the waveforms on the right shows the input and estimated frequencies of the improved PLL block.

Single-phase and 3-phase inverter examples are provided in the example folder to illustrate the use of the PLL blocks. For example, the figure on the left shows a single-phase grid-connected inverter with the enhanced PLL, and the figure on the right shows a 3-phase grid-connected inverter.

Built-in Switch Converter Modules

To facilitate quick and easy implementation of large multi-level multi-modular converter systems, built-in converter modules are provided, as shown below.
With these blocks, a large converter system can be quickly assembled.

Other New Elements

Several new elements are added to the PSIM library, notably,

- 10-winding inductor
- 3-phase ac cable
- 3-phase PQ-controlled load
- Level-2 NPN and PNP transistors
- Time delay block for logic signals
- 3-D lookup table
- AC sweep block with arbitrary excitation profile

Motor Drive Module

PMSM Model with Spatial Harmonic Effect

A PMSM model that takes into account the spatial harmonic effect is provided. In a linear PMSM model, the back EMF is assumed to be ideal, producing a constant torque at the steady state. But in reality, the back EMF contains harmonics due to slots, resulting torque ripples.

The waveform above on the left is the back EMF from a linear PMSM model, and the one on the right is from the PMSM model with spatial harmonics effect.

One big advantage of the spatial harmonics model is that it is very easy to define. Beside the traditional linear PMSM model parameters, it needs only the information of the open-circuit voltage harmonics and the open-circuit test conditions.

The result of this model is compared with the result from JMAG, a finite element analysis software, through a motor drive example shown below.
What’s New in V12.0

The results from the PSIM model are very close to those of the JMAG model, as shown below.

6-Phase PMSM Model
A 6-phase PMSM model is provided. It consists of two sets of 3-phase windings.

Improved Nonlinear Switched Reluctance Motor (SRM) Model
Nonlinear SRM models are improved in v12.0. The new model uses the flux table and the torque to give more accurate current and torque results.

High-frequency Induction Machine Model
A squirrel-cage induction machine model that takes into account the high-frequency effect is provided. With this model, it is possible to study the effect of a long ac cable on the voltage spike. The figure below, for example, shows an inverter motor drive system with an ac cable.
By taking into account inverter stray capacitances, the ac cable, and the high-frequency parameters, one can study how the high dv/dt of the inverter impacts the drive system and how cable parameters can be properly selected to mitigate the effect.

**Digital Control Module**

**Variable Sampling of ZOH**

Previously the sampling position of an input in a digital control system is always at the beginning of a sampling period. In v12.0, a new zero-order hold (ZOH) block is provided so that the sampling position can be defined and adjusted on-the-fly.

**Thermal Module**

Improvement is made to the way the device thermal equivalent circuit is handled. Previously the thermal equivalent circuits of a device need to be built manually outside the device. Now it is included as part of the device package, as shown below.
What’s New in V12.0

This greatly simplifies the schematic layout, and makes it easier to display device losses and junction temperatures.

**SPICE Module**

**AC Analysis of Switchmode Circuits**

In general, ac analysis cannot be performed directly to a switchmode circuits in SPICE-based software. One can perform such analysis through post-processing and multiple simulation runs, but the process is complicated and cumbersome. PSIM v12.0 provides an easy and convenient way of performing ac analysis of switchmode circuits in LTspice simulation. This provides a powerful way for SPICE users to find out the frequency response of a switchmode circuit easily.

The figure below shows a buck converter in switchmode, and the frequency response of the output voltage Vacr versus the modulation signal Vacs in LTspice simulation from the PSIM environment.

![Buck Converter and Frequency Response](image)

**Nonlinear Capacitor Model**

A nonlinear capacitor model is provided for LTspice simulation where the capacitance is a function of the capacitor voltage, as shown below.

![Capacitance vs Voltage Graph](image)

**Defining Node Names for SPICE Modeling**

PSIM v12.0 provides the capability to define a name for a node so that the voltage of this node can be used in a math expression. This is often needed in SPICE modelling. For example, the figure below shows that the two nodes of the resistor R1 are defined as n1 and n2, and the voltage V(n1,n2) is used elsewhere in the circuit.

![Node Names](image)
SimCoder Module

F2837x Hardware Target

A new F2837x Hardware Target is provided to support TI F2837x DSP for auto code generation. All major DSP functions, such as ADC, PWM (including high-resolution PWM), digital input/output, SCI, SPI, CAN, counter, encoder, etc., can be easily implemented with the built-in blocks.
Support of PE-Expert4’s FPGA Board

V12.0 supports FPGA boards of the DSP development platform PE-Expert4 from Myway. The FPGA boards can generate up to 144 PWM outputs, and are ideally suited for multi-level and multi-modular converter applications.

With the PE-Expert4 Target and the new support to the FPGA boards, PSIM can generate the required hardware code automatically. For example, the figure below shows a 7-level inverter with 36 switches.
What’s New in V12.0

The code generated by PSIM is ready to run on PE-Expert4 as it is, and there is no need for manual coding for either DSP and FPGA. This greatly accelerates the development process.

SIMVIEW:

The waveform processing software SIMVIEW is completely redesigned with significant improvements and many new functionalities. Below are some of SIMVIEW’s new functionalities.

Multiple Y Axes
Link from RidleyWorks:

RidleyWorks is a design software from Ridley Engineering. After one completes the design in RidleyWorks, it is now possible to export the design, including the design parameters and schematic that is ready to simulate, into PSIM, as shown below.
With the speed and many more functionalities in PSIM, for example, multiple simulation runs and parameter sweep through script, one can greatly speed up the development process.