ACSEEE

**Powersim** is a tool for modelling and simulation of dynamic systems. It can be used to study time continuous progress in a great number of areas, for example biology, economics, physics and ecology. The modelling is done by constructing a **Powersim** diagram.

Before going through the content do watch some videos you wil get idea of these softwares:

https://www.youtube.com/watch?v=LyS8ptNo4ww

## Why Choose PSIM for Power Electronics Simulation?

#### ****FAST****

PSIM is one of the fastest system-level simulators on the market. This means you can test hypotheses early and easily, and get from design to implementation quickly. PSIM is capable of simulating [*large and complex*](https://powersimtech.com/support/resources/literature-references/electrical-power-distribution-system-hv270dc-application-electric-aircraft/) power converter and control systems in a short time.

#### ****ACCURATE****

PSIM is used by some of the top universities and companies in the world, and is known for its ability to deliver accurate results without sacrificing simulation speed.

#### ****EASY TO USE****

PSIM has an intuitive user interface, easy implementation, and outstanding technical support. You’ll be up and running in no time.

#### ****CONTROL****

PSIM allows for mixed domain simulation. Design with analog s-domain, digital z-domain, or both in the same simulation.

**PSIM Software**

**PSIM** is an [Electronic circuit simulation](https://en.wikipedia.org/wiki/Electronic_circuit_simulation) software package, designed specifically for use in [power electronics](https://en.wikipedia.org/wiki/Power_electronics) and [motor drive simulations](https://en.wikipedia.org/wiki/Variable-frequency_drive) but can be used to simulate any [electronic circuit](https://en.wikipedia.org/wiki/Electronic_circuit). Developed by Powersim, PSIM uses [nodal analysis](https://en.wikipedia.org/wiki/Nodal_analysis) and the [trapezoidal rule](https://en.wikipedia.org/wiki/Trapezoidal_rule) integration as the basis of its simulation algorithm. PSIM provides a schematic capture interface and a waveform viewer Simview. PSIM has several modules that extend its functionality into specific areas of circuit simulation and design including: [control theory](https://en.wikipedia.org/wiki/Control_theory),[electric motors](https://en.wikipedia.org/wiki/Electric_motors), [photovoltaics](https://en.wikipedia.org/wiki/Photovoltaics%22%20%5Co%20%22Photovoltaics)and [wind turbines](https://en.wikipedia.org/wiki/Wind_turbines) PSIM is used by industry for research and product development and it is used by educational institutions for research and teaching.

## Modules

PSIM has various add on modules.There are modules that enable motor drive simulation, [digital control](https://en.wikipedia.org/wiki/Digital_control), and the calculation of thermal losses due to switching and conduction. There is a renewable energy module which allows for the simulation of photovoltaics (including temperature effects), batteries, [supercapacitor](https://en.wikipedia.org/wiki/Supercapacitor%22%20%5Co%20%22Supercapacitor), and wind turbines. Additionally there are several modules which allow co-simulation with other platforms to verify [VHDL](https://en.wikipedia.org/wiki/VHDL) or [Verilog](https://en.wikipedia.org/wiki/Verilog%22%20%5Co%20%22Verilog) code or to co simulate with an [FEA](https://en.wikipedia.org/wiki/Finite_element_method) program. The programs that PSIM currently co-simulates with are: [Simulink](https://en.wikipedia.org/wiki/Simulink%22%20%5Co%20%22Simulink), [JMAG](https://en.wikipedia.org/wiki/JMAG), and [ModelSim](https://en.wikipedia.org/wiki/Mentor_Graphics%22%20%5Co%20%22Mentor%20Graphics).

## Some modules are:

## 1)SPICE

With a vast library of SPICE models for industrial devices, SPICE provides the ability to analyze a particular device in detail, for example, the turn-on and turn-off transient of a semiconductor device.

Added in PSIM version 11.1 is the ability to make use of the LTspice\* engine to run models.

The SPICE Module provides access to:

* Dual PSIM/SPICE model definitions
* Running LTspice from PSIM
* Import of existing SPICE libraries and subcircuits
* Non-linear Capacitor model
* AC sweep of switchmode circuit in SPICE

The integration of SPICE into PSIM allows users to greatly reduce development time as a simulation does not need to be re-made in a new environment to study device level interactions.

We anticipate that users will use PSIM to study:

* Topology verification
* Control loop and stability
* Steady state junction temperatures and other thermal simulations
* Component sizing such as energy storage devices
* Code generation

 **2)**  **PSIM’s Motor Drive module**

* PSIM’s Motor Drive module gives you the building blocks for your next innovation.  Save design time and drive complex power electronics-based motor control systems with confidence.
* Analysis and design of a motor drive system is often a challenge because of the complexity in machine modeling and controller design. PSIM’s Motor Drive module offers an easy and effective way to model and simulate complex motor control algorithms and control systems.
* PSIM provides commonly used electric machine models, mechanical load models and control blocks (such as Maximum-Torque-Per-Ampere (MTPA) Control and Field Weakening Control blocks). You can create and use custom-built machine or load models for greater flexibility.

3) **PSIM’s Digital Control module-It**  is a flexible, time-saving solution to analyze systems in z-domain and convert from analog to digital control.

####  Achieve more analysis with less cost

 With higher performance and lower cost, microcontrollers/ DSPs have been increasingly used in converter control in power supply and motor drive applications,requiring control algorithms to be implemented in the discrete time z-domain.

Unlike analog control, there are unique issues in digital control loop design, such as:

* Sampling
* Quantization and resolution
* Delay resulting from processor computation

As a result, a controller that works in the analog s-domain may not work in the digital z-domain.

## Simulation and Power Conversion

**Test in Simulation – Not production**

PSIM’s Digital Control Module lets you implement the digital control algorithm with z-domain simulation blocks making it easy to check the performance and stability of a digital control loop. You can easily test the effects of quantization and observe the effects of digital delay on a control algorithm, allowing users to debug and troubleshoot in a simulation environment.

 3) SimCoupler fuses PSIM with Matlab/Simulink® by providing an interface for co-simulation. Part of a system can be implemented and simulated in PSIM and the rest in Matlab/Simulink. It’s straightforward, easy to set-up and requires minimal user input.

With SimCoupler, there’s no need to choose between tools. Use both and boost the simulation power for your innovation.

## The SimCoupler Module enables [Matlab/Simulink](http://www.mathworks.com/products/simulink/%22%20%5Ct%20%22_blank) users to implement and simulate power circuits in their original circuit form, thus greatly shortening the time to set up and simulate a system that includes electric circuits and motor drives.

**4)** PSIM’s **Renewable Energy module** gets you on your way to a real-world renewable energy power system. It simplifies and speeds up the development process – without sacrificing high-quality, trusted results.

The module offers a range of models for solar, wind and battery storage systems, all designed to enable you to simulate, model and analyze quickly and accurately.

Photovoltaics Models

PSIM provides two types of PV models.

Functional model – Simplified and easy to use.

Physical model – Takes into account the effects of light intensity and temperature. The physical model allows users to enter detailed parameters from solar cell’s datasheet. PSIM provides the Solar Module tool to facilitate the extraction of the model parameters from a manufacturers’ datasheet. This simplifies the process of modeling and analyzing a real-world photovoltaic power system.

Maximum-Power-Point-Tracking (MPPT) Blocks – Several sample MPPT blocks are provided based on the following methods:

Subsection of PVs – getting energy out of solar panel can be challenging, characteristics of panel change, the MPPT helps you track your operating conditions. Standard algorithms are:

First-order differential method

Incremental conductance method

Perturb & Observe method

**Wind Turbine, Battery & Ultracapacitor Model**

**Wind Turbine Models**The wind turbine model, together with the Motor Drive Module, can simulate wind power systems. In addition to the models, PSIM also includes pre-built examples for the following three most commonly used wind power system structures. These examples provide an excellent starting point for custom wind power system design and analysis.

Double-fed induction generator (DFIG) based

Permanent magnet synchronous generator based, and

Squirrel-cage induction generator based

***Battery Model***

In any renewable energy power system, battery storage is an essential part of the system.

PSIM’s battery model allows users to simulate battery charging and discharging process in an energy storage system. The battery model can be used to model various types of batteries. The battery model is also included in the HEV Design Suite.

***Ultracapacitor Model***

Added in Version 10, the ultracapacitor model provides an accurate representation of the charging, discharging, and hold characteristics of an ultracapacitor. Additionally, there is an added modeling utility for extracting the parameters from the manufacturers’ datasheet.

## Q: Is PSIM Windows 7, 8, and 10 compatible? Does PSIM run in a 64-bit environment?A: Yes, PSIM Versions 11.0, 10.0, 9.3, 9.2, 9.1, 9.0, 8.0, and 7.1.2 are all tested and fully functional in Windows 7 and 8. PSIM Versions 12.0, 11.1, 11.0, 10.0 and 9.3 are tested and fully functioning on Windows 10. Yes, PSIM Versions 12.0, 11.1, 11.0, 10.0, 9.3, 9.2, 9.1, 9.0, 8.0, and 7.1.2 are all tested and running on 64-bit machines. In addition, PSIM Version 9 and later versions are available as both 32- and 64-bit.

## Q: Can PSIM run on Linux?A: The License Manger for network licenses can be installed on Linux.  However, PSIM will run only on Windows OS.  A user may use a Windows emulator on Linux to run PSIM.

## : What is the minimum system requirement to run PSIM?A: Minimum windows configuration:

## -2GB of RAM

## -20 GB of free disk space

## -a CPU that ranks at least 4000 in  [www.cpubenchmark.net](http://www.cpubenchmark.net/%22%20%5Ct%20%22_blank)

## If you are going to use large schematic files or large graphs, we recommend:

## -16GB of RAM

## -100 GB of free disk space

## Comparison with SPICE

## PSIM has a much faster simulation speed than [SPICE](https://en.wikipedia.org/wiki/SPICE) based simulators based on its usage of the ideal switch. With the additional Digital and SimCoupler Modules almost any kind of logic algorithm can be simulated. Since PSIM uses ideal switches the simulated waveforms will reflect this, making PSIM more suited for system level studies rather than switching transition studies. Additionally, PSIM has a simplified interface compared to other simulators and as a result has a more intuitive interface.

[MOSFET](https://en.wikipedia.org/wiki/MOSFET) and [Diode](https://en.wikipedia.org/wiki/Diode) Level 2 models were added in the version 10 release. These models allow the simulation of the switch transition, reverse recovery effects, and gate drive circuitry. A comparison with a PSIM & SPICE model of the same device showed similar resulting waveforms with a comparable simulation speed given identical operating conditions.

PowerSim recently partnered with CoolCAD Electronics to add [CoolSPICE](https://en.wikipedia.org/wiki/CoolSPICE%22%20%5Co%20%22CoolSPICE), a [SPICE](https://en.wikipedia.org/wiki/SPICE) based integrated circuit modeling and design tool, as a bundle option for the PSIM software package. The advantage being that PSIM would then have the flexibility to be able to run [SPICE](https://en.wikipedia.org/wiki/SPICE) based models and net-lists.