# PSIM

**PSIM** has been a leading power electronics and motor drive simulation and design software for over 25 years. With an intuitive, user-friendly interface and a robust simulation engine, PSIM is the all-in-one solution that meets users' simulation and design needs.

PSIM can handle quick power converter loss calculations, motor drive efficiency calculations, conducted EMI analysis, and analog/digital control. PSIM also offers automatic embedded code generation for rapid control prototyping. Additionally, with PSIM's various Design Suites, users can design power supplies, electromagnetic interference (EMI) filters, and motor drive systems quickly and conveniently.

PSIM also integrates seamlessly with other Altair products (Twin Activate, Embed, Flux/FluxMotor, MotionSolve) and with third-party software, and offers an integrated solution for multi-domain, multi-physics systems.

**Here are some Key Features of "PSIM"**

Active Switch Simulation

Simulate high-speed switching without the convergence issues typically seen in other tools due to large derivatives in current and voltage waveforms.

Quick Power Loss Simulations

Calculate the switching and conduction losses of switching devices (diodes, MOSFET, IGBT, SiC/GaN devices) as well as the core and winding losses of inductors – without slowing down simulation speeds.

Embedded Code Generation

Let your code be a self-starter and generate high-quality, consistent C code from a PSIM control schematic to boost designs' speed and reduce development costs and time-to-market.

Easy Design Verification

PSIM has intuitive design verification tools, including Monte Carlo, Sensitivity, and Fault Analysis to assist Design Failure Mode and Effect Analysis (DFMEA) of power converter designs.

C-block Capabilities

Enter custom C code directly into the c-block without a compiler. PSIM will automatically interpret and execute at runtime by a built-in C interpreter.

SPICE Link

Link to SPICE software and study parasitic interactions and gate drive circuits in the PSIM environment. The library of SPICE models lets users analyze particular devices in detail.