SPICE MODULE

PSIM

A winning combination of device simulation and system simulation

PSIM's SPICE Module, powered by CoolCAD Electronics' CoolSPICE engine, provides the capability to run SPICE simulation in the PSIM environment.

While PSIM is excellent in system-level and control simulation, SPICE excels in device simulation. 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.

With the PSIM engine and SPICE engine in one integrated environment, users will be able to switch easily between running PSIM simulation and SPICE simulation.

Such a combination gives users the capability to have the proof of concept quickly in PSIM, and then zoom in to study a circuit in detail in SPICE.

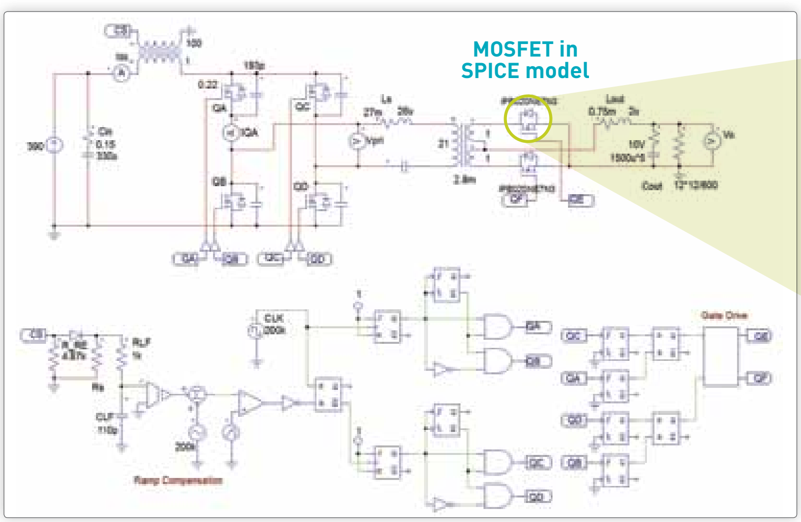
Functions are provided such that SPICE netlists from other SPICE software can be easily imported and simulated in PSIM.

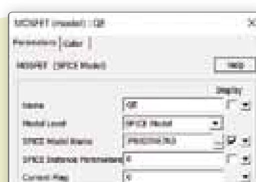
In addition, CoolCAD Electronics is recognized as the industrial leader in wide band gap device modeling. The SPICE Module will provide the capability to simulate power converters using SiC and GaN devices.

The combination of PSIM and SPICE provides the ultimate environment for all your design needs.

FEATURES & BENEFITS

- Supports SPICE models and simulation
- Seamless transition between PSIM simulation and SPICE simulation
- Supports standard SPICE netlist from other software





```
**********************************
*  MOSFET: Infineon IPB020NE7N3  75V 120A 2.0mohm 108nF
**********************************
.model IPB020NE7N3 VDMOS (Rg=2.7 Vto=4.2 Rd=740u Rs=657u Rb=987u
+ Kp=482.9 Lambda=0.03 Cgdmin=68p Cgdmax=3.44n
+ A=0.2 Cgs=10.64n Cjo=7.77n M=0.27 Is=207.3p VJ=0.9 N=1.16
+ TT=40n mfg=Infineon Vds=75 Ron=2.0m Qg=155n)
```

Left: **Phase-shifted** full-bridge DC-DC power supply
Above: **MOSFET** dialog box
Below: **MOSFET** SPICE model