

# Accept No Simulations

## PSpice 16.6—The Best Just Keeps Getting Better

1984 marked the beginning of a new era for analog and mixed signal circuit simulation. Cadence® PSpice® was introduced and electrical engineers were able to simulate circuits on their PCs with speed and accuracy never thought possible. In the 29 years since PSpice was first released, many competitors have attempted to duplicate its capabilities and success. None have achieved that goal, and PSpice remains the standard by which all others are judged.

While entry level simulators abound, PSpice continues to be the innovator providing the most advanced circuit simulation available. Features like multi-core processing, 64 BIT precision, IBIS 5.0 support, and user customization mean the standard is now higher than ever. Why settle for anything less?

To learn more about PSpice 16.6:  
[www.ema-eda.com/PSpice](http://www.ema-eda.com/PSpice)

**cādence**®

**EMA** | Design Automation™  
[ema-eda.com](http://ema-eda.com)

©2013 Cadence Design Systems, Inc. All rights reserved.