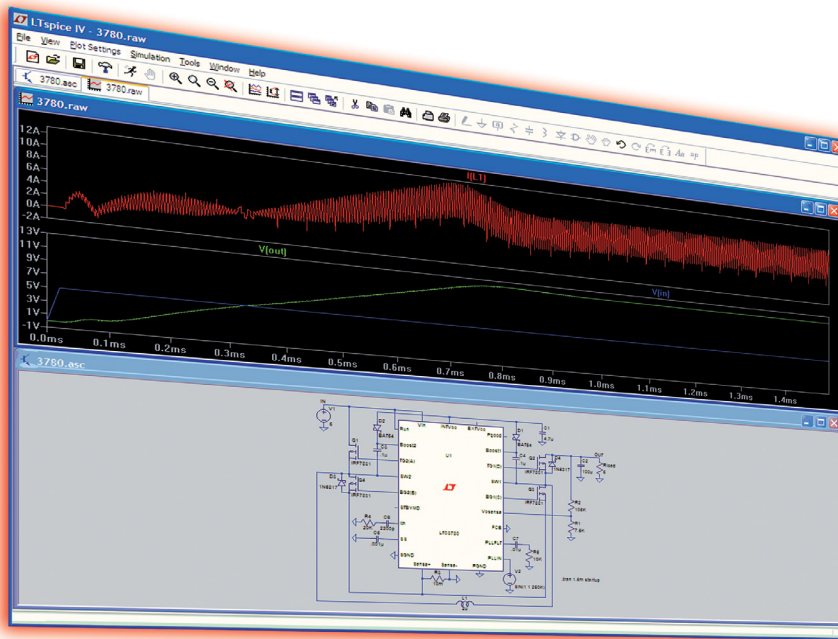


# LTspice IV



- Free Analog Circuit Simulator
- Unlimited Nodes/Nets
- Fast Simulations

LTspice® is a high performance SPICE simulator, schematic capture and waveform viewer designed to speed the process of power supply design. LTspice adds enhancements and models to SPICE, significantly reducing simulation time compared to typical SPICE simulators, allowing one to view waveforms for most switching regulators in minutes compared to hours for other SPICE simulators.



[www.twitter.com/ltspice](http://www.twitter.com/ltspice)

Follow @ LTspice on Twitter for up-to-date information on models, demo circuits, events and user tips.



[www.linear.com/ltspice](http://www.linear.com/ltspice)

Included in the download is a complete and fully functional SPICE program, help files, macro models for Linear Technology's power products, over 200 op amp models, as well as models for resistors, transistors and MOSFETs.



[video.linear.com/ltspice](http://video.linear.com/ltspice)

View Instructional Videos.

Sign up for the Linear Insider email newsletter to receive LTspice updates @ [www.linear.com/mylinear](http://www.linear.com/mylinear)



See Demo



LT, LT, LTC, LTM, LTspice, Linear Technology and the Linear logo are registered trademarks of Linear Technology Corporation. All other trademarks are the property of their respective owners.

