eCADSTAR™

DATASHEET

SPICE CONTROLLER

Many PCBs include analog circuits in addition to, for example, digital control and signal processing. Analog signals represent continuous quantities, so it's very important to optimize them.

LTspice® is an industry-standard version of SPICE that is free to download and use, from Analog Devices .

It equips you to perform many kinds of analysis, including DC, AC, transient, and Monte Carlo. Combining LTspice with SPICE controller in Schematic Editor makes analog simulation a safer, more repeatable process.

WITH SPICE CONTROLLER IN SCHEMATIC EDITOR, YOU CAN …

- Set and retrieve simulation settings in Schematic Editor
- Map to generic LTspice ® transistor models in both your schematic and eCADSTAR library
- Launch LTspice simulation from Schematic Edtor
- Determine optimal operating ranges for your analog circuits
- Optimize and share analog circuits within and beyond your engineering team

www.ecadstar.com

THE BENEFITS OF LTspice® FOR THE eCADSTAR SCHEMATIC EDITOR

LTspice® is an industry-standard, trusted simulator, so why not just write a SPICE netlist and use it independently?

The difference is that Spice Controller brings analog simulation directly into your design flow, preserving your settings with your schematic design data. That makes reviews, enhancements and engineering changes much safer, more controlled processes. In the example above, transistor model mappings, passive component sweep settings and amplitude sweep settings all remain with the schematic design.

SPICE Controller does not replace any of the trusted, key functions of LTspice®. Rather, it complements them, tying schematic design data tightly to simulation. You can reload settings directly from your schematic, making analog circuit optimization and re-use integral parts of your design process.

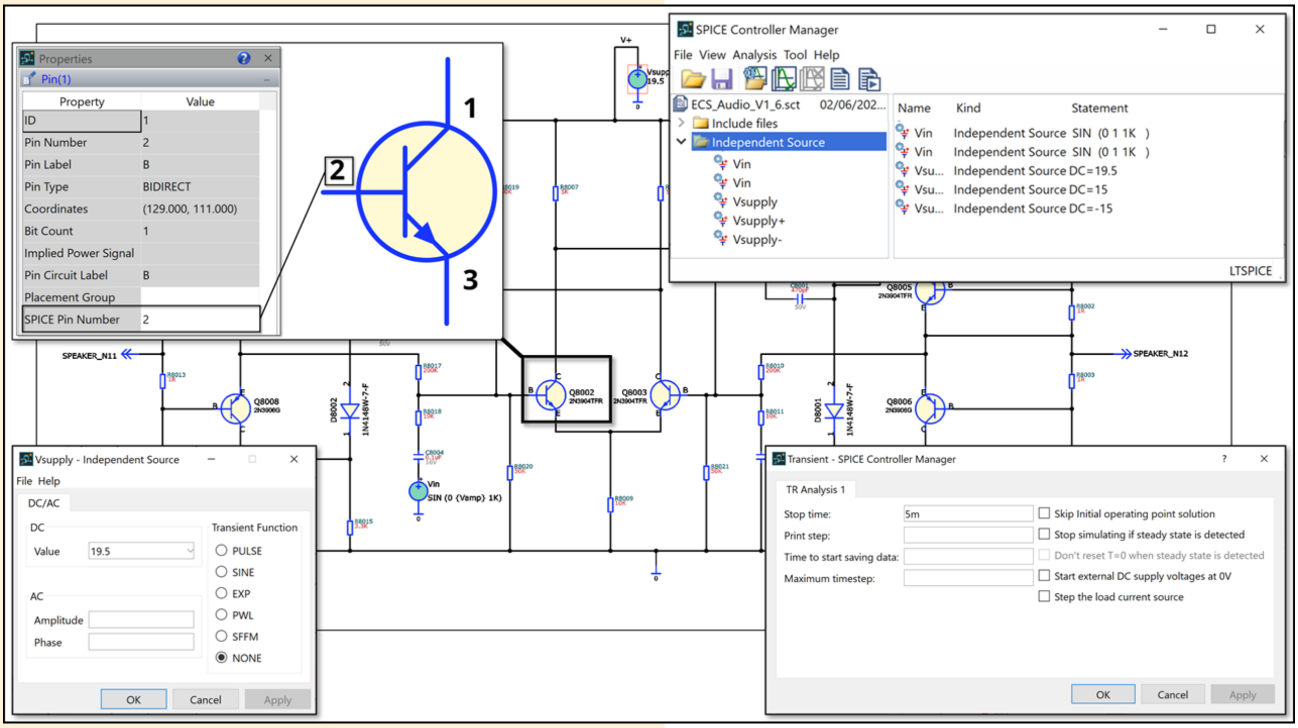

FIGURE 1: *Without leaving Schematic Editor, you can map symbols to generic LTspice® transistor models and set simulation parameters, in this case including input sine-wave stimulus and supply voltage. You can even map to generic transistor models in your eCADSTAR library, so mappings are automatically present for all your designs.*

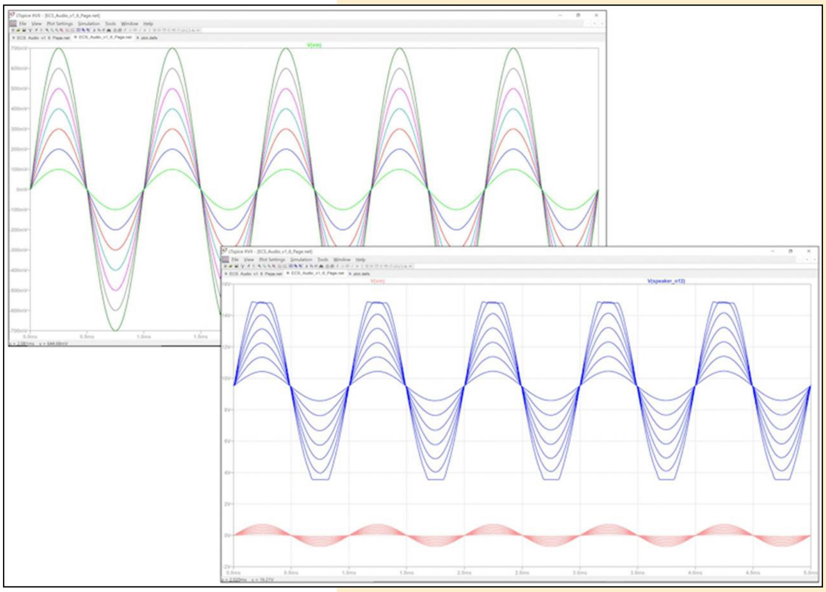

FIGURE 2: By sweeping a range of input signal amplitudes, we can find out where the signal begins to be distorted by clipping at the output

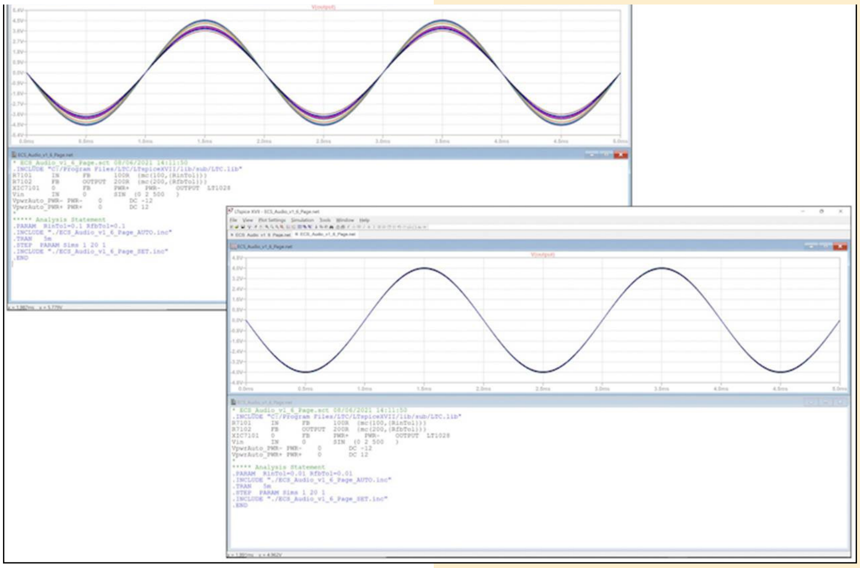

FIGURE 3: Sweeping passive component values shows the difference in output variation between ten percent and one percent tolerance components, allowing engineers to balance cost and performance

TYPICAL ANALOG SIMULATION CASE

This power amplifier circuit could be used in multiple schematic designs. If the input signal amplitude gets too high, the audio amplifier will clip the output. If the input signal amplitude gets too low, the amplifier may not produce an output signal at all. This circuit can be optimized for different conditions in different designs, even if the basic circuit remains the same.

The same methods and user interface are used throughout eCADSTAR.

To discover the range of input amplitude that yields unclipped output, we can set up simulation parameters directly in Schematic Editor. This setup stays with the schematic design and so it is easy to share and re-use without separating it from design data.

HOW CAN I GET THE eCADSTAR SPICE Controller?

SPICE Controller is included with eCADSTAR Advanced HS, Advanced 3D and Ultimate product bundles. Alternatively, it is available as an optional extra with eCADSTAR Base or standalone eCADSTAR Schematic Editor.

TO SUMMARISE...

•Spice Controller brings analog simulation directly into your design flow making reviews, enhancements and engineering changes a much safer, more controlled processes.

•Map symbols to generic LTspice® transistor models and set simulation parameters, without leaving the Schematic Editor.

• Lastly, the SPICE Controller is included with eCADSTAR Advanced HS, Advanced 3D and Ultimate product bundles.