One Technology Way • P.O. Box 9106 • Norwood, MA 02062-9106, U.S.A. • Tel: 781.329.4700 • Fax: 781.461.3113 • www.analog.com

ADIsimPE Quick Start Guide

OVERVIEW

The ADIsimPE simulator is a result of Analog Devices, Inc., teaming with SIMPLIS Technologies. ADIsimPE allows the full simulation of Analog Devices products in the library while still offering the entire functionality of the SIMetrix/SIMPLIS Intro version. In addition, ADIsimPE integrates with ADIsimPower to quickly verify circuit performance.

ADIsimPE, which is powered by SIMetrix/SIMPLIS, is a circuit simulation suite optimized for the design and development of analog and mixed signal circuits. SIMetrix mode is ideal for the simulation of general nonswitching circuits. It provides full Pspice compatibility for use with industry-standard SPICE models. SIMulation piecewise linear system (SIMPLIS) mode simulates the operation of switching circuits with vastly improved robustness, speed, and accuracy compared to standard SPICE. It is particularly useful for switching power supply, phase-locked loops (PLLs) and analog-to-digital converter (ADC)/digital-to-analog converter (DAC) applications.

Additional details are available on the SIMPLIS Technologies website.

ADIsimPE is intended for Analog Devices customers who do not have a full license for the SIMetrix/SIMPLIS program. This tool can simulate Analog Devices encrypted schematics as well as nonencrypted Intro version compatible ones. The full license SIMetrix/SIMPLIS version can simulate all encrypted schematics from any partnered company.

ADIsimPE offers the following:

- Full schematic capture
- SPICE or SIMPLIS simulation mode
- Waveform viewer and analysis
- More than 1000 IC models and application circuits to get users started with
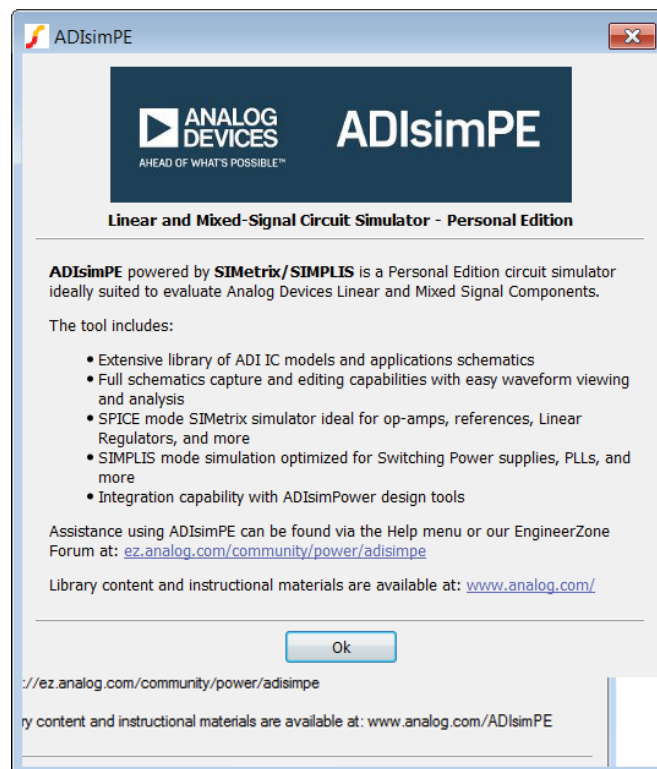- Support from the EngineerZone

Figure 1. *ADIsimPE Linear and Mixed-Signal Circuit Simulator*

TABLE OF CONTENTS

REVISION HISTORY

GETTING STARTED

Download the ADIsimPE installation file from www.analog.com/ADIsimPE.

Note that you must have a myAnalog account to download the ADIsimPE installation file from www.analog.com/ADIsimPE.

INSTALLATION PROCEDURES

Take the following steps to install ADIsimPE on a Microsoft® Windows®-based PC:

1. Double-click **adisimpe.exe**. This self extracting executable then initiates the **ADIsimPE InstallShield Wizard**.
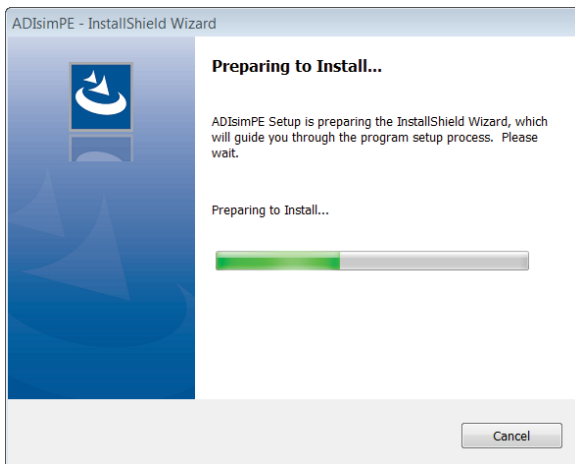

Figure 2. Preparing to Install the **ADIsimPE** *InstallShield Wizard*

Click **Next** within the **ADIsimPE** InstallShield Wizard dialog box (see Figure 3).
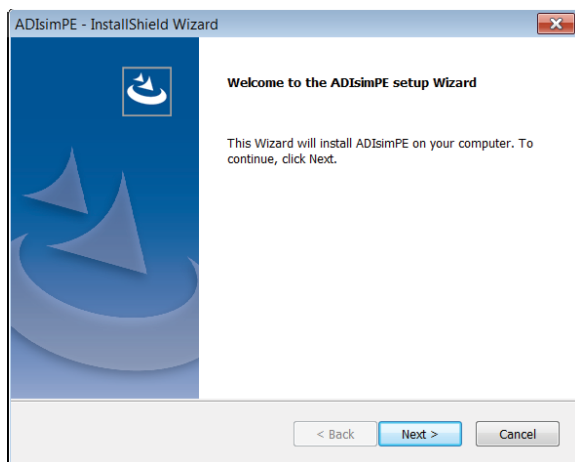
Figure 3. **ADIsimPE** *InstallShield Wizard Setup*

The license agreement then appears (see Figure 4).

2. Click **Next** to accept the license agreement with **I accept the terms of the license agreement** selected. The destination directory then appears. (If you do not accept the license agreement, you cannot proceed with the installation of the ADIsimPE.)
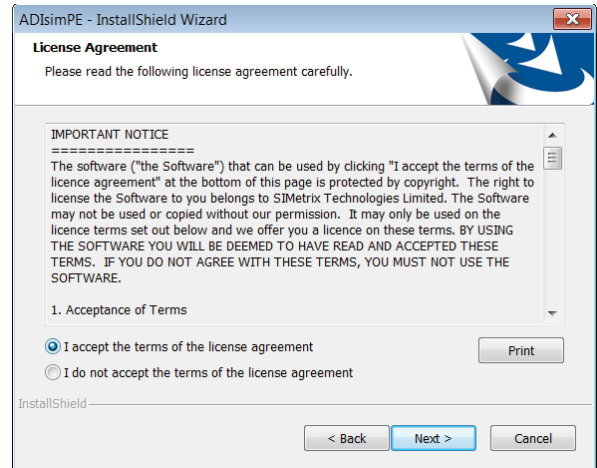
Figure 4. ADIsimPE **License Agreement**

3. Select the destination location. The **ADIsimPE InstallShield Wizard** default directory for ADIsimPE is **C:\Program Files (x86)** and reflects the ADIsimPE version number. (Your version number may vary from the one depicted in Figure 5.) To change the installation directory, click **Browse**.

4. After selecting the destination location, click **Next** (see Figure 5).

Figure 5. **Choose Destination Location** *Screen*

The **Ready to Install the Program** screen then appears (see Figure 6).

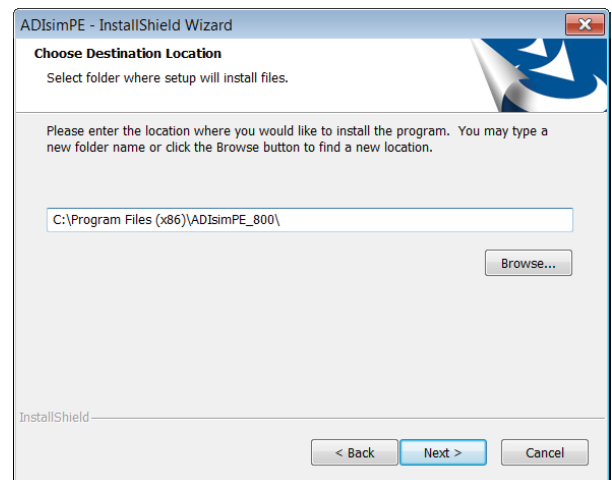5. Click **Install** to begin the installation, **Cancel** to quit, or **Back** to make installation changes (see Figure 6).
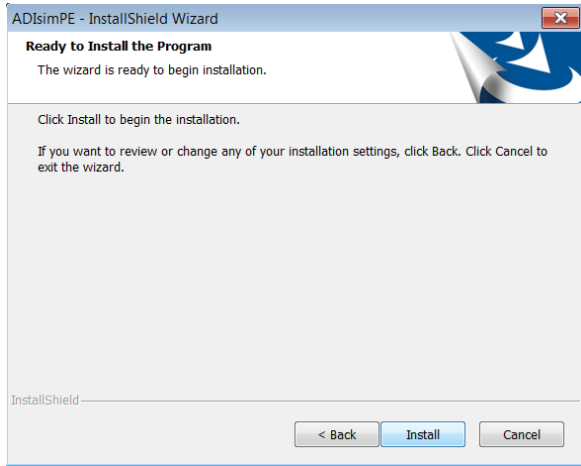

Figure 6. Ready to Install the Program

6. The **ADIsimPE** InstallShield Wizard provides a status of the installation of files and approximately where it is in process (see Figure 7). If you click **Cancel** at this point, you will interrupt the installation of files and be left with an incomplete ADIsimPE installation that may not work. (Use the **Cancel** button if the **ADIsimPE** InstallShield Wizard is idle many minutes and does not progress.)
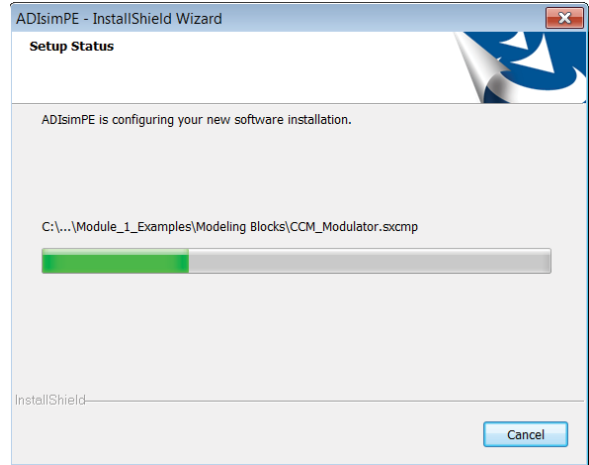
*Figure 7. **Setup Status** of the Installation of Files*

7. Once the **ADIsimPE** InstallShield Wizard has installed its files and configured itself in the Windows registry, the **InstallShield Wizard Complete** dialog box appears (see Figure 8). Click **Finish** to close the dialog box (see Figure 8).
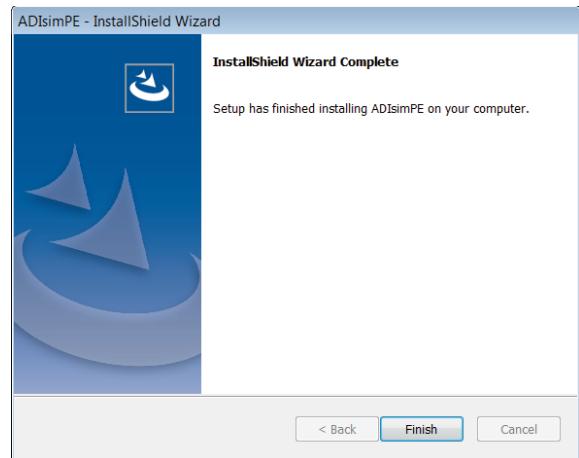
Figure 8. Installation Finished

Once installation is completed, it is recommended that users see the Updating ADIsimPE section to avoid future licensing issues.

HOW TO USE THE SOFTWARE

To open the software, take the following steps:

1. Click the **Start** menu.
2. Select **All Programs/ADIsimPE 8.00**, where 8.00 indicates your version number, which may vary (see Figure 9).

Figure 9. Choose ADIsimPE

The screen in Figure 10 then appears with the option of migrating your old configuration to the new version (if applicable) and the option to add example files.
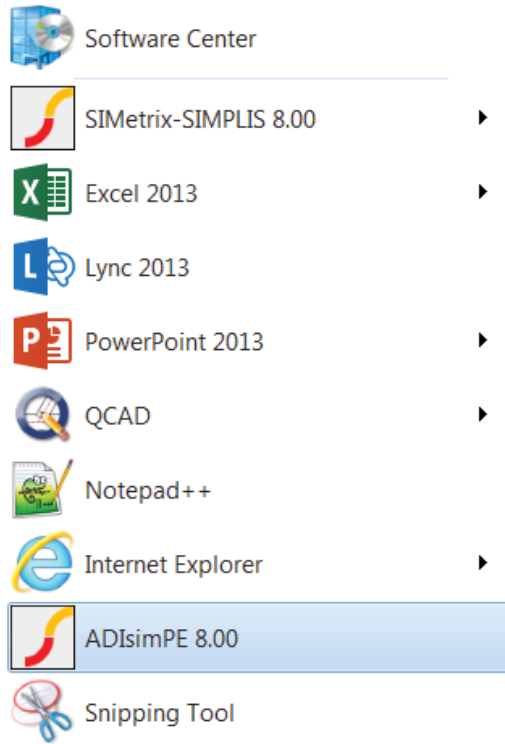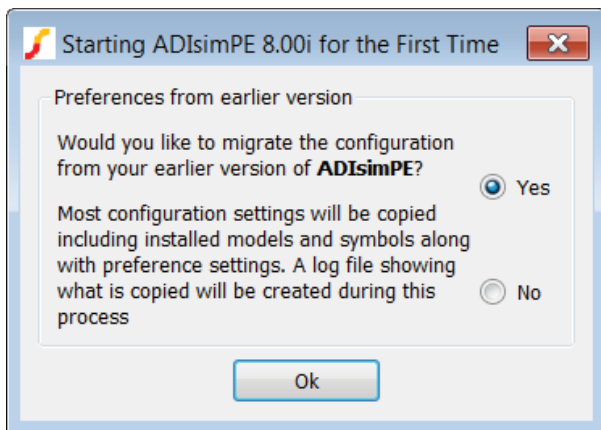

*Figure 10. **Starting ADIsimPE 8.00i for the First Time** Screen*

The example files are from the Analog Devices library, which includes all Analog Devices encrypted schematics and models from op amps to switching regulators, as well as a good selection of application circuits to get started with (see the Library of Analog Devices Schematics section).

3. After making the selections, click **Close**. The splash screen shown in Figure 11 then appears.
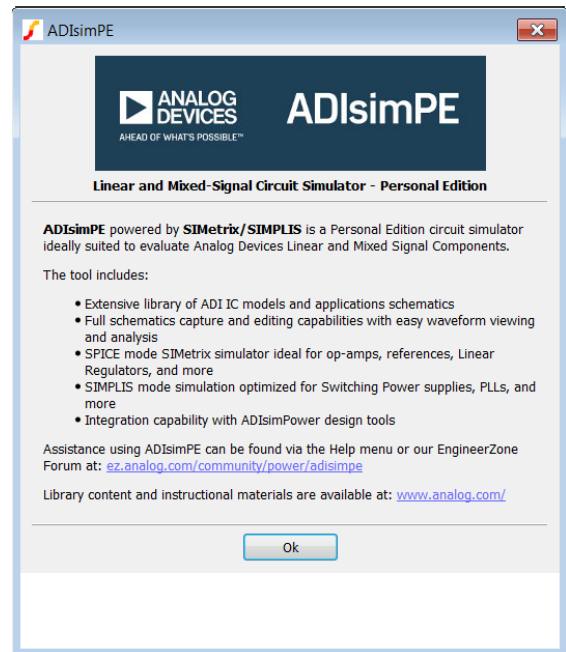
Figure 11. ADIsimPE Splash Screen

4. Click **Ok**. The command shell opens (see Figure 12 to Figure 14).

When ADIsimPE is started from the Windows **Start** button, users can open a new (blank) schematic or an existing schematic using the options found under the **File** pull-down menu in the **Command Shell** (see Figure 12). These options then open the **Schematic** window (see Figure 14).

Note that the default simulator is **SIMetrix**, and parts available vary depending on which default simulator is chosen.

Figure 12. *ADIsimPE* Command Shell Window

Figure 13. *ADIsimPE* Open New Schematic Window

Figure 14. *ADIsimPE Schematic Window*

UPDATING ADIsimPE

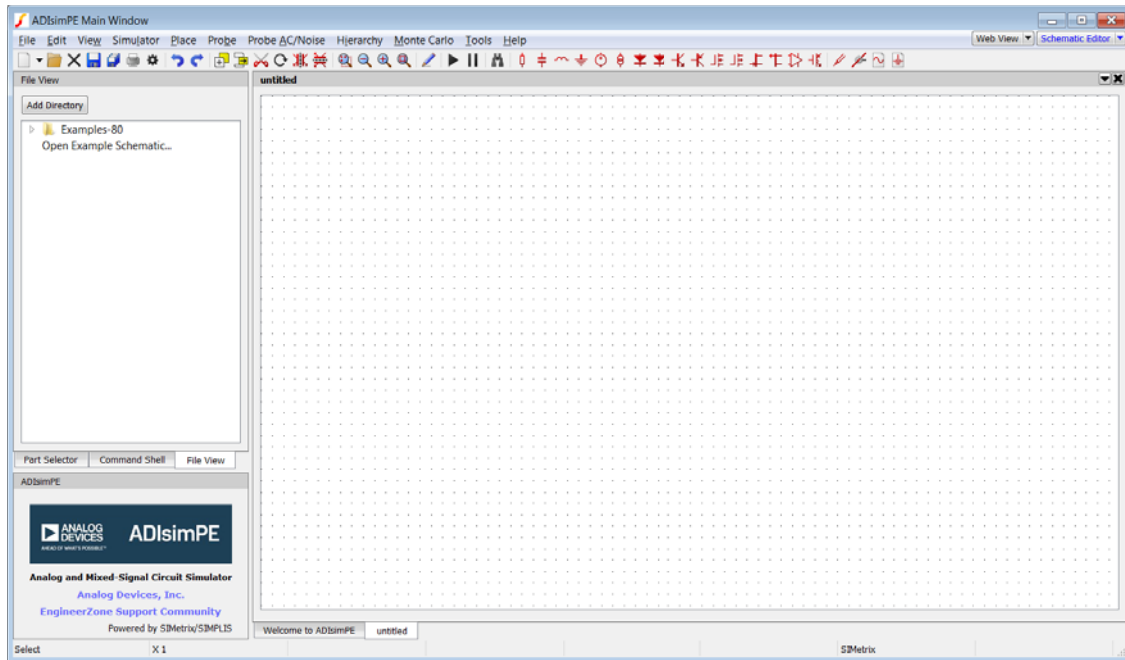The library of schematics continues to grow, particularly as new devices become available. Click **Help** in the ADIsimPE **Command Shell** window and check off the **Check Updates Now** box to check for updates regularly (see Figure 15). Analog Devices recommends checking updates monthly.
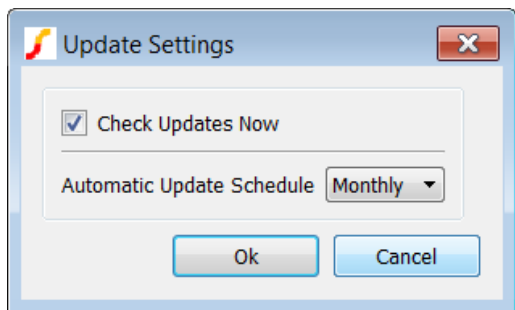

Figure 15. *Update Settings*

The **Check Updates Now** feature periodically checks (via the Internet) if there is a new release of ADIsimPE. New releases may contain updated model library content from Analog Devices and/or enhanced features for the application.

LIBRARY OF ANALOG DEVICES SCHEMATICS

If users opt to have examples files installed, these files are in a directory named in a manner that reflects its software version, such as: **C:\My Documents\ADIsimPE\Examples-80**.

This installation directory has two subdirectories, one for SIMetrix schematics and another for SIMPLIS schematics.

The schematics encrypted with Analog Devices products are in respectively named subdirectories:

- **C:\My Documents\ADIsimPE\Examples-80\SIMetrix\ Analog Devices**
- **C:\My Documents\ADIsimPE\Examples-80\SIMPLIS\ Analog Devices**

The schematics for general SPICE applications and linear circuits are in the SIMetrix subdirectory. The models for nonlinear and switching circuits are in the SIMPLIS subdirectory.

Many of the SIMPLIS schematics from Analog Devices for switching regulators are also available within their respective ADIsimPower design tools (such as, Microsoft Excel). When the schematic is exported from an ADIsimPower design tool, ADIsimPE also obtains settings such as V_{IN}, V_{OUT}, I_{OUT}, as well as component selection from the current design in the tool.

For more information about the integration between ADIsimPower and ADIsimPE, see the Quick Start Steps for Running Power Simulations section.

LIBRARY OF ANALOG DEVICES MODELS

Encrypted models of Analog Devices parts are available from within the ADIsimPE tool. Because the models are encrypted, they do not count toward the ADIsimPE limitations imposed upon schematic size for simulation.

Before inserting a model into a schematic, ensure that ADIsimPE is in the proper simulation mode because the available models are dependent upon which mode is used. To change the simulation mode in the **Schematics Edit** window, select **Simulator/Switch to SIMPLIS Mode** or **Simulator/Switch to SIMetrix Mode**. The resulting dialog box has radio buttons to choose either SIMetrix for linear applications or SIMPLIS for nonlinear or power switching applications (see Figure 16).
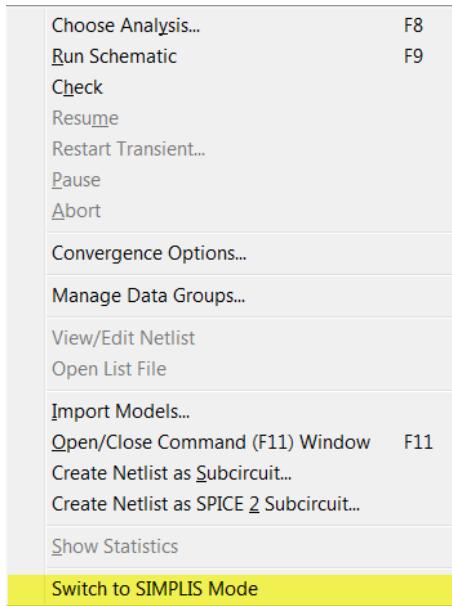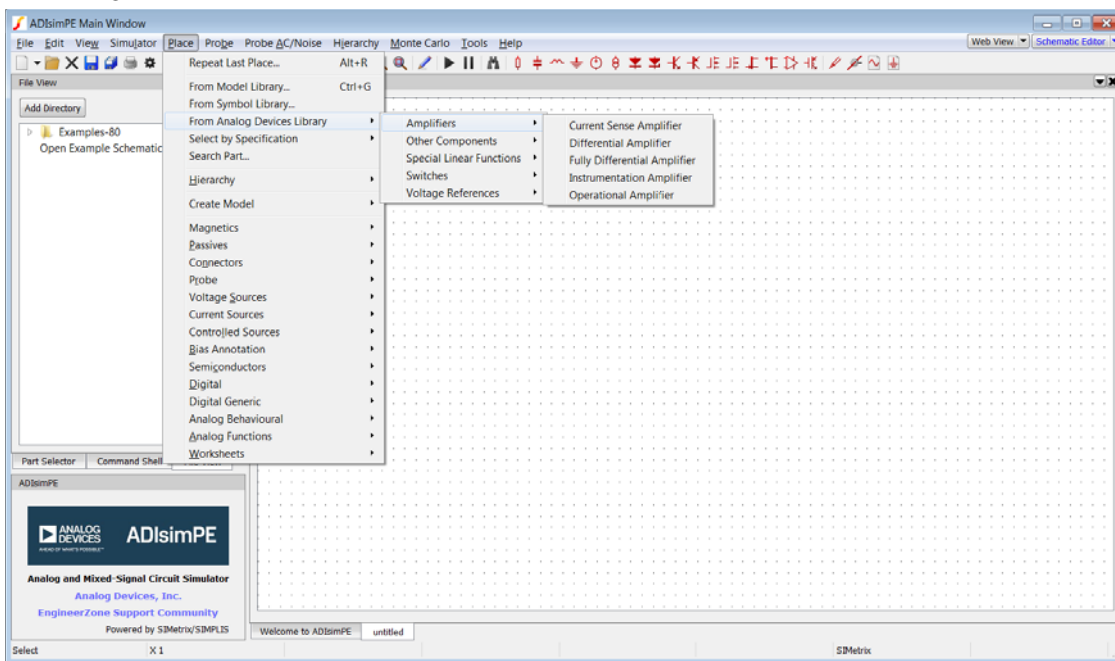

Figure 16. Select Simulator

ADIsimPE makes the following distinction between symbols and models:

- A symbol is what is graphically depicted within the schematic and shows how the circuit element connects with other elements.
- A model is a netlist description of the circuit element. The same schematic symbol may be used for many different models (or parts). The properties on a symbol specify which model is used for simulation. Inserting a symbol does not guarantee that it has an underlying model or the desired model.

To insert an existing model into your schematic, select from the **Place** pull-down menu and click **From Analog Devices Library**.

- When in SIMetrix mode, the submenu from the pull-down shows **Amplifiers, Switches**, and various other components (see Figure 17).
- When in SIMPLIS mode, the submenu from the pull-down shows **Switching Regulators** and other components.

Use the **Schematic Editor** to insert any number of devices (see Figure 17).

Figure 17. Analog Devices Library

However, the type of simulation (**SIMetrix** or **SIMPLIS**) restricts which models can be in the circuit for simulation. Incompatible models will highlight in a different color (see Figure 18).

Figure 18. Error Message when Mixing Models

In addition, the **Part Selector** tab, shown under the toolbars in Figure 19, is a useful way to locate models for insertion into schematics without having to go into the **Place/Select Device** pull-down menu (see Figure 20).
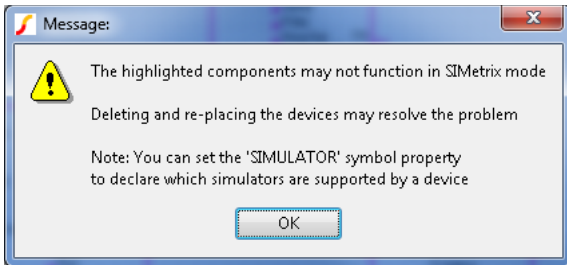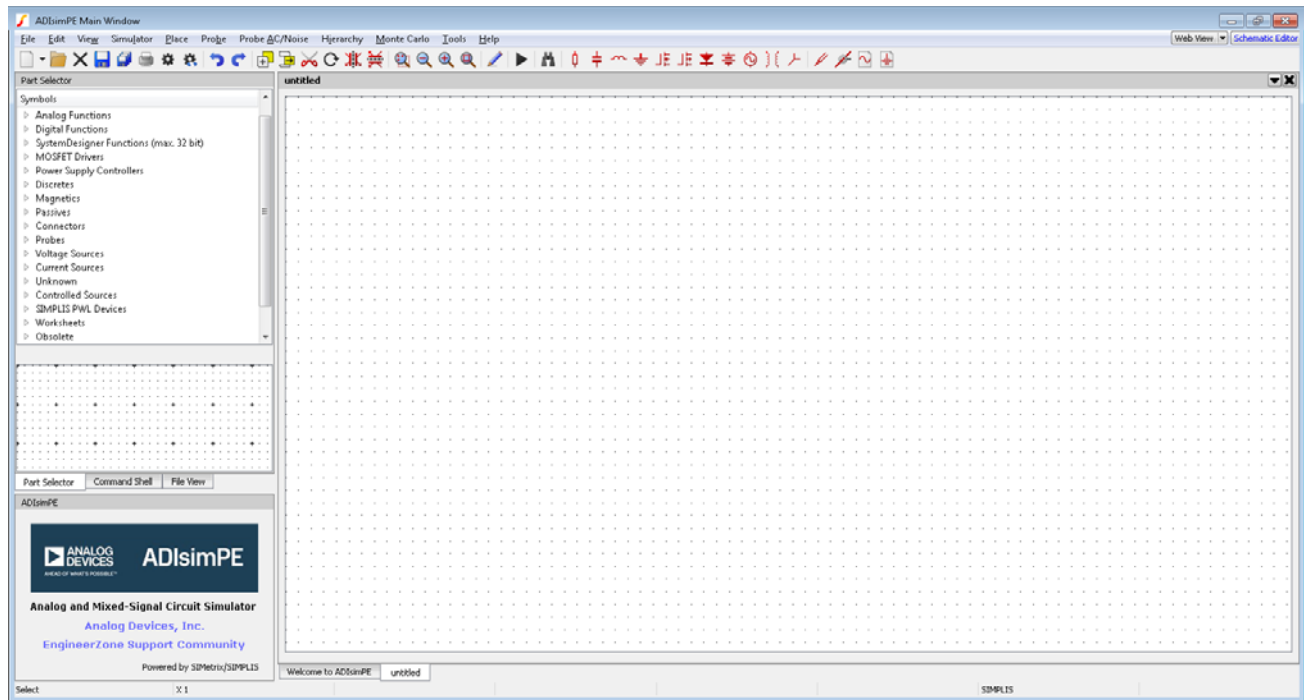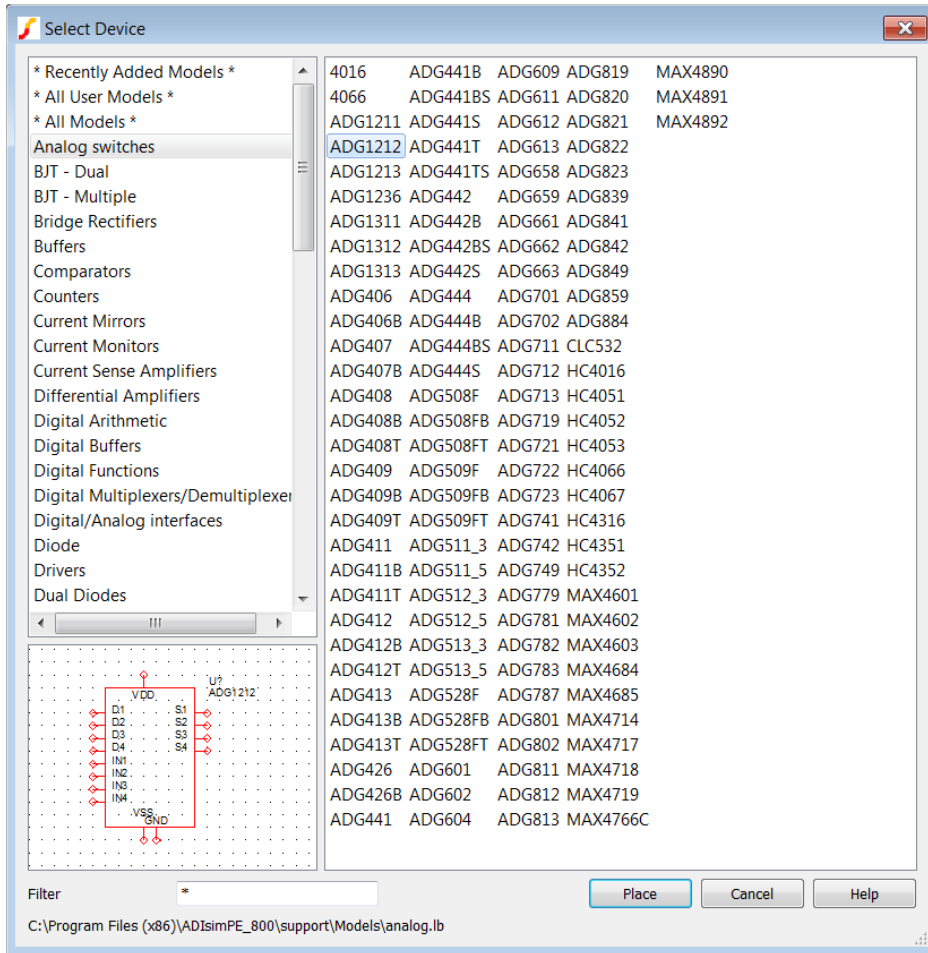

Figure 19. Part Selector Tab

Figure 20. Select Model

QUICK START STEPS FOR RUNNING SIMULATIONS

ADIsimPE features SIMetrix and SIMPLIS tutorials. Go to the **Command Shell** window **Help** menu to find these (see Figure 21).
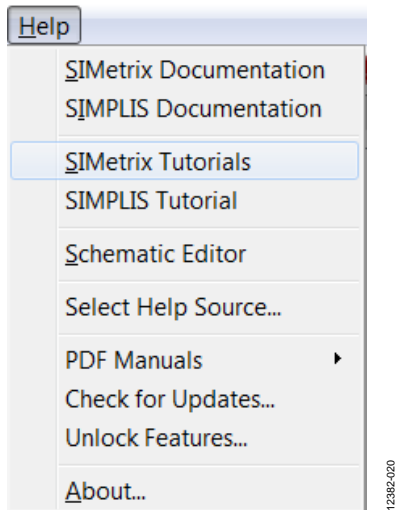

Figure 21. SIMetrix Tutorials or SIMPLIS Tutorials

Another excellent path for understanding how to work with ADIsimPE is to explore the examples. The examples, if installed, are typically found in the **Command Shell** window by selecting **File/Open Schematic** or in the **Schematic** window by clicking **File/Open** (see Figure 22). The default location is the directory where the examples are installed.
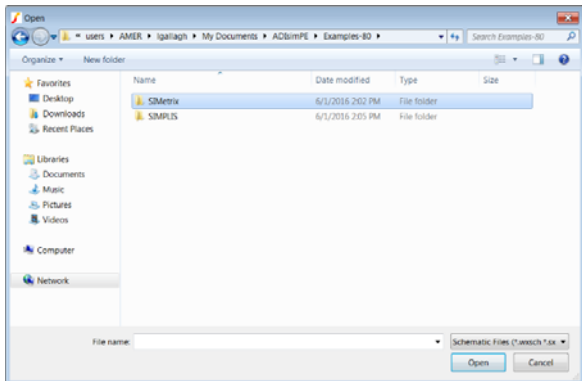

Figure 22. SIMetrix or SIMPLIS Examples

QUICK START STEPS FOR RUNNING POWER SIMULATIONS

Analog Devices power management incorporates the simulation abilities of the ADIsimPE tool into its recommended development process. To design power circuits, take the following steps:

1. Select the part and the design tool.
2. Design and optimization.
3. Simulate.

After entering the design criteria into ADIsimPower for Step 1, ADIsimPower recommends appropriate parts and topologies: www.analog.com/adisimpower.

ADIsimPower lets users download the appropriate design tools for the selected devices. Note that these tools are Microsoft Excel based.

When users run the respective ADIsimPower design tool on their local PC during Step 2 with their design criteria and settings for the features of the device, the tool produces a schematic and bill of materials for their design that they can further optimize.

Once Step 2 results in an optimized design, some ADIsimPower tools assist users with simulating (Step 3) by having a SIMPLIS schematic embedded in the tool for export. All settings relevant to their design and their desired simulation are exported to ADIsimPE with the model.

When the ADIsimPE simulator is launched from a design tool, it can immediately simulate users' designs by pressing **F9**.

Therefore, the recommended way of running SIMPLIS simulations on Analog Devices power devices is from their respective design tools because users can then inherit simulation values for the following:

- V_{IN}, V_{OUT}, and I_{OUT} operating conditions.
- All bill of material components needed for their designs.
- All advanced settings and jumpers.
- The settings of the desired simulation such as start, stop, and number of data points.

To specify the desired simulation, click **Simulate with ADIsimPE/Simplis** from the design tool (see Figure 23).

*Figure 23. Button in ADIsimPower (Excel Tool): **Simulate with ADIsimPE/Simplis***

The export process of the design tool will then access the appropriate embedded SIMPLIS model and change the settings based on the current design state and desired simulation. The export process will prompt for an appropriate directory and name for the exported file.

The default export directory is the location of the Excel design tool when it was started. If the design tool is started from within a ZIP file, the export directory must be changed. The export process cannot write into a ZIP file.

The default filename for export is the name of the embedded SIMPLIS model. Renaming of the export file is recommended to avoid confusion because settings internal to the file will change based on the current state of the design tool and the desired simulation. If a file in the export directory already has the desired name, the user is prompted to overwrite the existing file or cancel the export operation.

The **Export Simplis Model** and **Export & Run Simplis Model** buttons both save a SIMPLIS file for the user to access (see Figure 24). Clicking **Export & Run Simplis Model** performs the additional step of starting ADIsimPE (or SIMPLIS) with the exported file. However, the **Export & Run Simplis Model** button disables if the design tool does not detect a computer registry entry for ADIsimPE (or SIMPLIS).

Simulation Options

The simulation options shown depend on the design tool and the recommended solution from the design tool. Some design tools may restrict which simulation options are available. If the solution only has one rail, the **Rail to Simulate** option will not be present. In meeting customer design criteria to recommend a solution, the design tool may recommend solutions for which it does not have a SIMPLIS model. In these cases, the **Export Simplis Model** and **Export & Run Simplis Model** buttons disable.
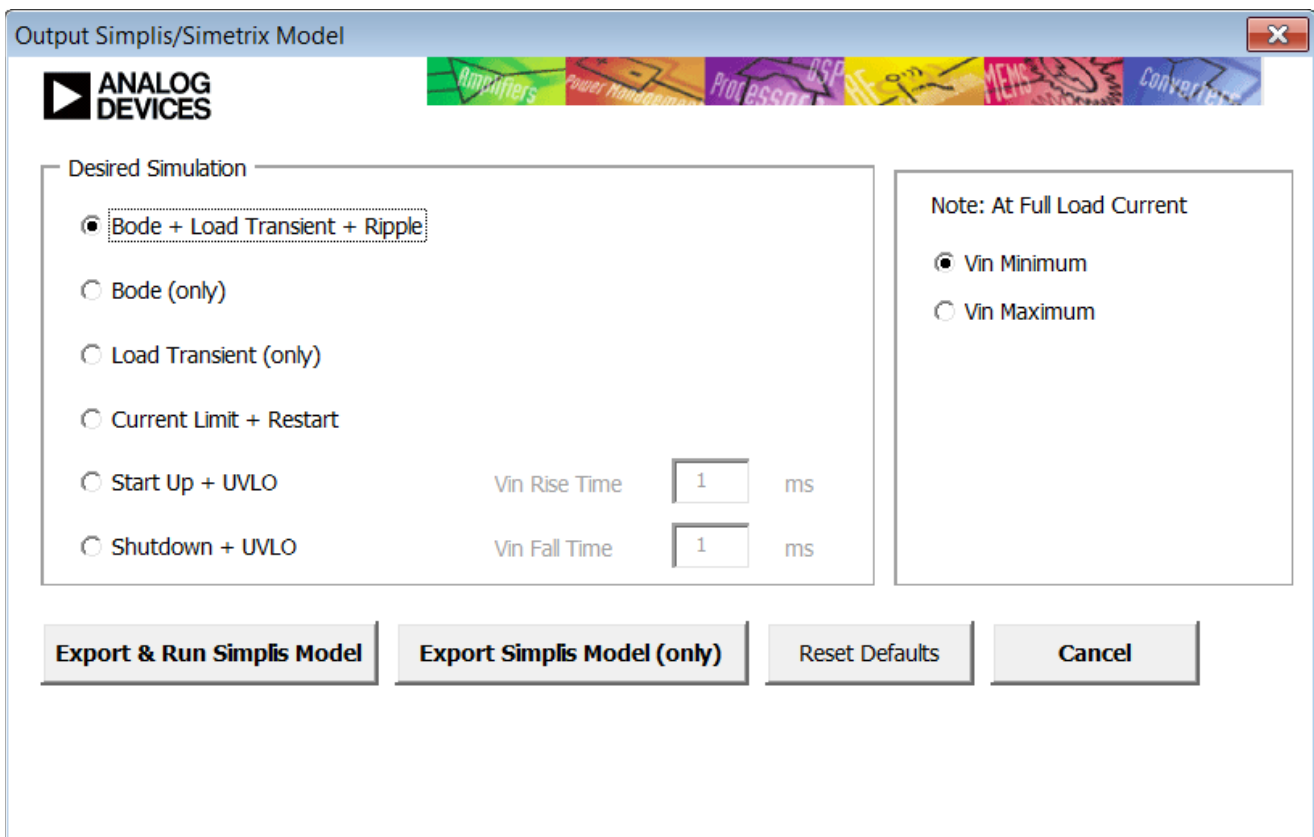

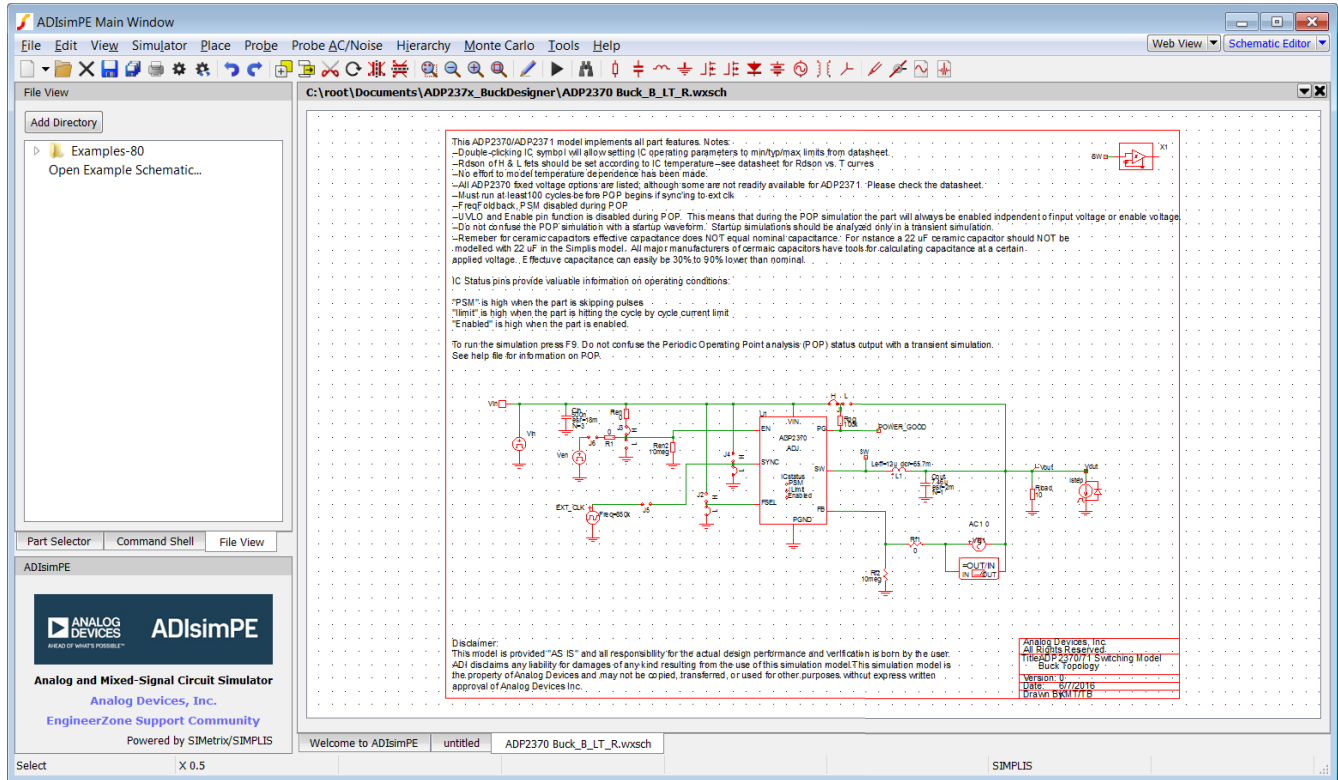*Figure 24. Exporting from Analog Devices Design Tools*

Figure 25. Example Schematic Exported from a Design Tool into ADIsimPE

It is not possible to import the settings from an exported ADIsimPE (SIMPLIS) file back into an ADIsimPower design tool.

ADIsimPE permits adjustments to the simulation, generates results, and explores the full capabilities of the SIMPLIS engine.

The benefits of using ADIsimPE (or the full SIMetrix/SIMPLIS version) go beyond the simulation/validation of the power management portion of a design. Users can add to the schematic load circuitry and other parts of their applications and simulate how this will work. Whereas the Analog Devices models are encrypted to protect Analog Devices' intellectual property (IP), ADIsimPE allows the simulation of much larger circuits than would otherwise be permitted with the free SIMPLIS distribution.

SIMetrix/SIMPLIS is a feature rich software application even when distributed as ADIsimPE. Its description and operation are beyond the scope of this document. For more information, refer to SIMPLIS Technologies documentation.

RUNNING SIMULATIONS FROM ANALOG DEVICES POWER MANAGEMENT SCHEMATICS

Assuming that the ADIsimPower design tool produced a valid design, when the SIMPLIS schematic is exported, it is ready for simulation within ADIsimPE. To run a simulation, select **Simulator/Run** (or press **F9**).

Users can place additional probes within the schematic for observation.

FEATURES OF ANALOG DEVICES POWER MANAGEMENT SCHEMATICS

Many schematics for power management provided by Analog Devices have enhancements that improve the simulation experience. The main purpose of these enhancements is so that the data sheet limits may be simulated with easy to understand pull-down menus.

One example is an additional dialog box associated with the parameters of switching regulators so that everything important for simulation of the IC and the IC limits is easily accessible. The **Edit Device Parameters** dialog box is accessed when a user double-clicks on the schematic component, or right-clicks and selects **Edit Part** (see Figure 26).
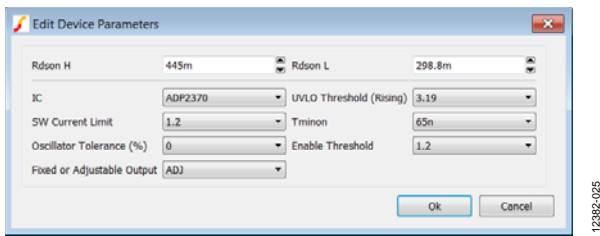
*Figure 26. **Edit Device Parameters**: Switching Regulators*

Another example is an additional dialog box associated with the parameters of the output capacitance (C_{OUT}). Often the optimum output capacitance cannot be achieved by a single, real-world capacitor. Instead, it is achieved by more than one capacitor in parallel. Although the buck schematic may only show the symbol for a single capacitor, the real-world design may call for more than one placed in parallel (see Figure 27).
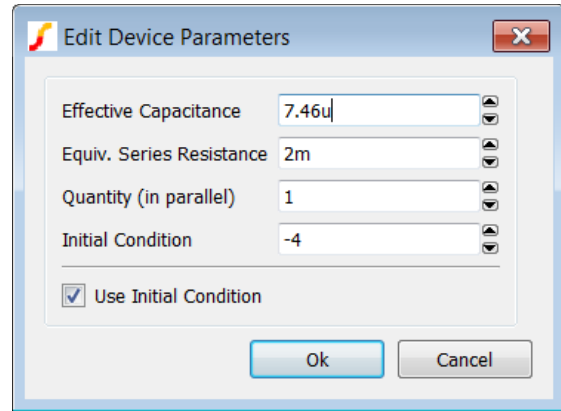
*Figure 27. **Edit Device Parameters**: Capacitor*

The **Edit Device Parameters** dialog box for this capacitor specifies how many capacitors are in parallel, their effective capacitance, and their effective series resistance. This information is critical for simulation and is important for the design engineer to know.

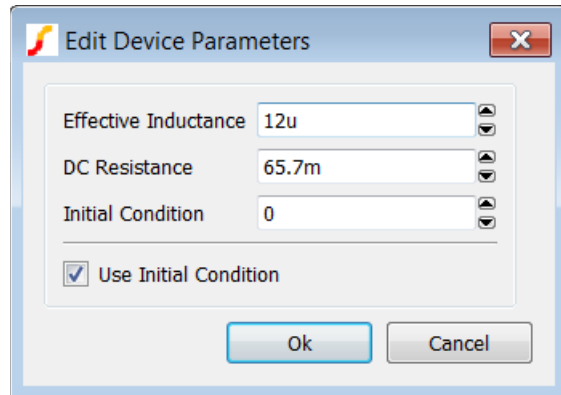The inductor is another component that has been enhanced (see Figure 28).

*Figure 28. **Edit Device Parameters**: Inductors*

ADIsimPE LIMITATIONS

ADIsimPE supports Analog Devices encrypted content as well as additional circuits. The additional circuits are bound by the limits of the SIMetrix/SIMPLIS Intro version:

- 120 internal analog nodes
- 36 digital nodes
- 72 digital ports
- 24 digital components
- 36 digital outputs

The internal analog node limit does not apply to the encrypted content. However, the node limit does apply to nodes from a larger application circuit of the customer including nodes inside elements, such as nonencrypted op amps. In practice, the Analog Devices application schematics already include the minimum necessary external components to define and to simulate the circuit. In addition, when an encrypted model is placed into a schematic, its internal representation does not count against the limits.

ADIsimPE cannot simulate encrypted schematics from other companies.

ADIsimPE has no limits on the size of the schematics that can be drawn. The only limits apply to what can be simulated. These limits include the following:

- A maximum of 15 additional state variables. A capacitor or an inductor each requires one state variable. Each time varying or small signal ac sources require one state variable, with the exception of sinusoidal or cosinusoidal sources, which require two state variables per source.
- A maximum of 10 additional capacitors or inductors combined.
- A maximum of six additional switches, simple or transistor.
- A maximum of six additional logic gates.
- A maximum of 26 states. Each piece wise linear (PWL) element requires one state. Each switch requires one state. Each time varying source requires one state. Each logic gate requires one state.
- A maximum of 100 new topologies because 100 topologies are enough for simple switching circuits that use only simple models. More complex circuits or circuits that have more complicated models may exceed this limit. Encrypted models that typically require multiple topologies can greatly expand this limitation.

The limits are not applicable to the full licensed version of SIMetrix/SIMPLIS.

www.analog.com