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

ADIsimPE is a result of Analog Devices, Inc. teaming with SIMPLIS Technologies. ADIsimPE allows 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 non-switching circuits. It provides full Pspice compatibility for use with industry-standard SPICE models. SIMPLIS (SIMulation Piecewise-Linear System) 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, PLLs and ADC/DAC applications.

Details are available from SIMPLIS Technologies.

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

ADIsimPE offers:

- Full schematic capture
- SPICE or SIMPLIS simulation mode
- Waveform viewer and analysis
- More than 1,000 IC models and application circuits to get you started
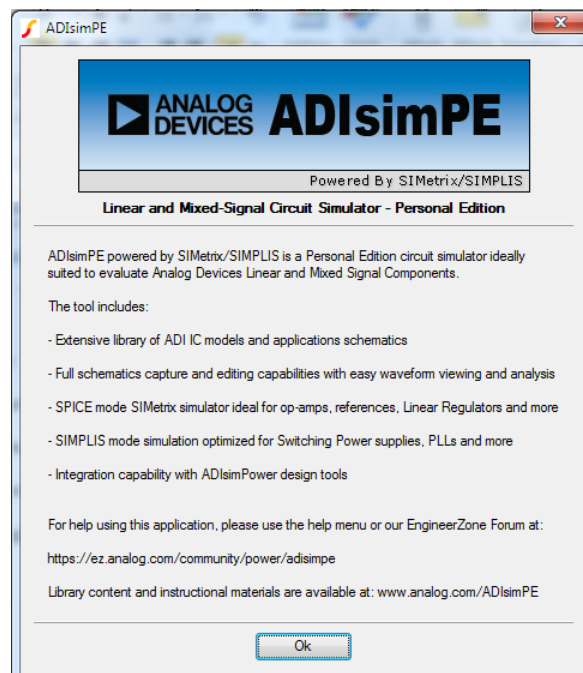- Support by EngineerZone

Figure 1. Splash Page from ADIsimPE

TABLE OF CONTENTS

REVISION HISTORY

6/14—Revision 0: Initial Version

GETTING STARTED

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

To download the file, you must be registered with myAnalog.

INSTALLATION PROCEDURES

To install ADIsimPE on your computer (Microsoft® Windows), use the following steps:

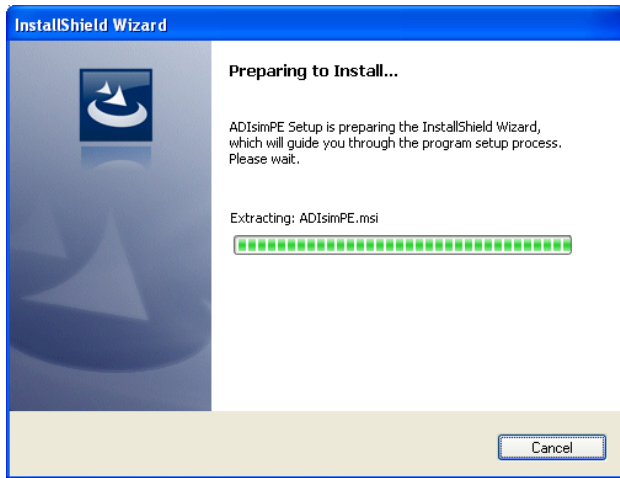1. Double click **adisimpe.exe**, which is a self-extracting executable that starts the InstallShield Wizard.

Figure 2. Preparing to Install

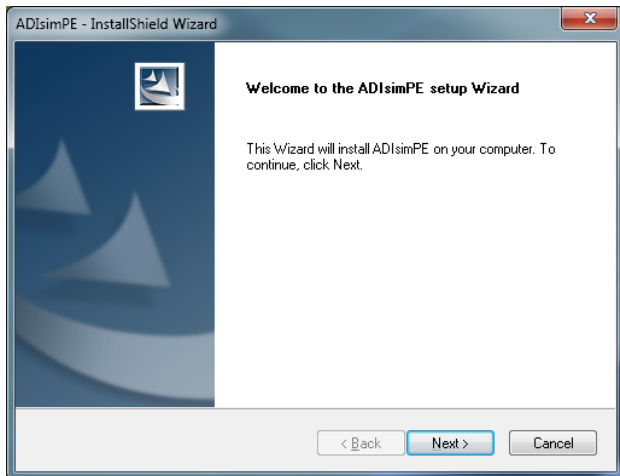2. Click **Next** from the initial InstallShield Wizard dialog box. The license agreement appears (see Figure 4).

Figure 3. Startup Dialog Box for the ADIsimPE Installation

3. Accept the license agreement by clicking **Next**. A destination directory appears. (If you do not accept the license agreement, you cannot proceed with the installation of ADIsimPE.)

Figure 4. ADIsimPE License Agreement

4. Select the destination directory.
   The InstallShield 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**.

5. After selecting the directory destination, click **Next** (see Figure 5). The **Ready to Install the Program** screen appears (see Figure 6).
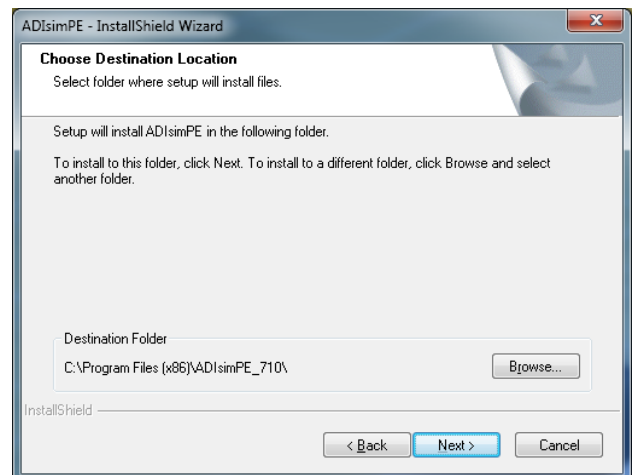

Figure 5. Destination Directory

6. Click **Install** to begin the installation, **Cancel** to quit, or **Back** to make installation changes.
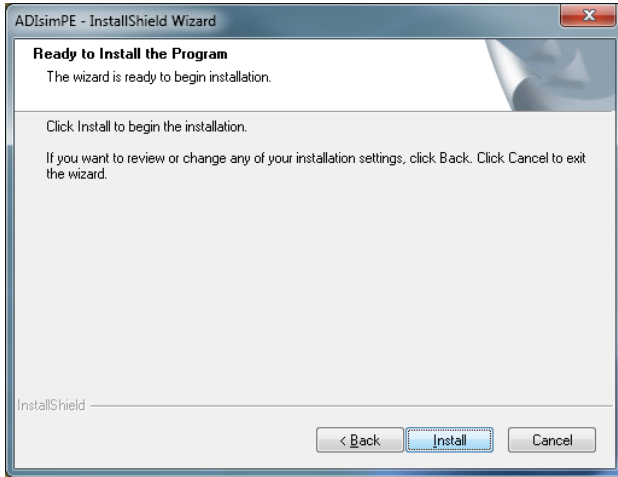

Figure 6. Ready to Install

7. The InstallShield 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 installation of ADIsimPE that may not work. (Use the **Cancel** button if the InstallShield is idle many minutes and does not progress.)
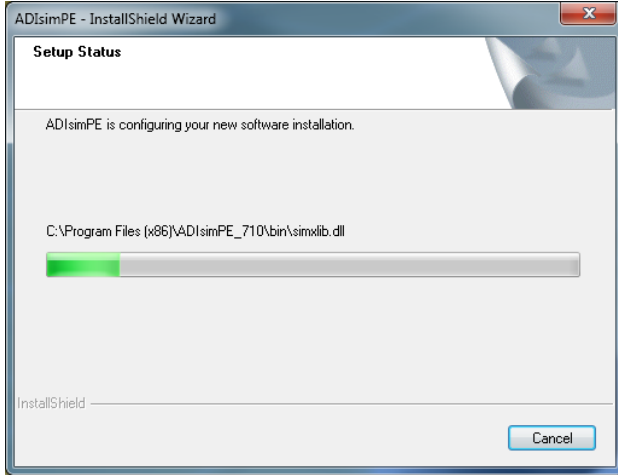

Figure 7. Status of the Installation of Files

8. Once the InstallShield has installed all of 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.
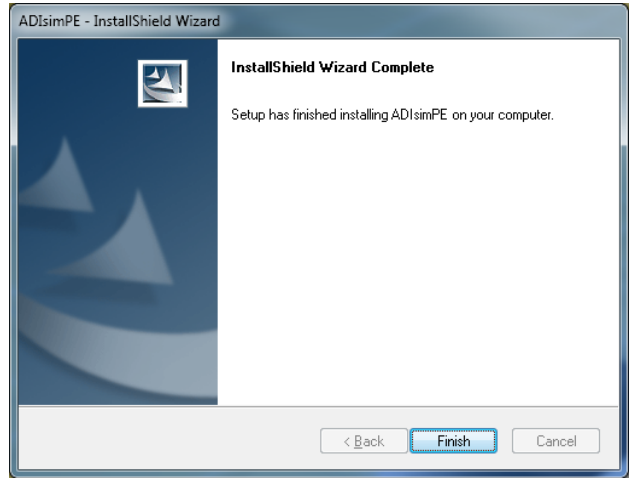

Figure 8. Installation Finished

HOW TO USE THE SOFTWARE

To open the software, use the following steps:

1. Click the **Start** menu.
2. Select **All Programs**.
3. Choose **ADIsimPE 7.20** (where 7.20 indicates your version number, which may vary). See Figure 9.
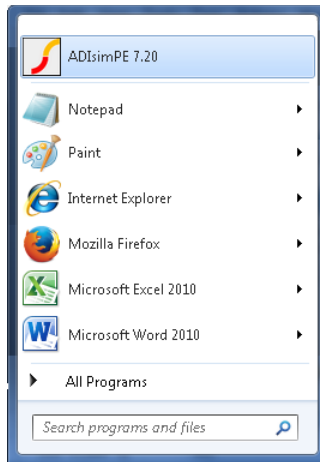

Figure 9. Choose ADIsimPE

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

The example files are 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 you started (see the Library of Analog Devices Schematics section).
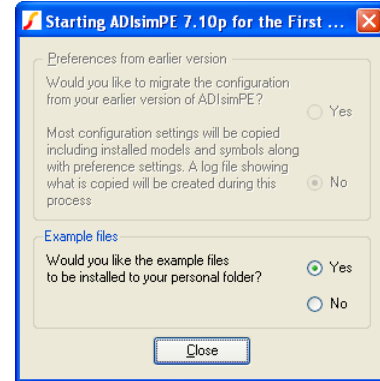

Figure 10. Starting Up ADIsimPE

4. After making your selections, click **Close**. The splash screen appears (see Figure 11).

Figure 11. ADIsimPE Splash Screen

5. Click **Ok**. The command shell opens (see Figure 12 and Figure 14).

When ADIsimPE is started from your Windows Start button, you may open a new (blank) schematic or an existing schematic using options from the **File** pull-down menu in the Command Shell. These operations then open the Schematic Window.
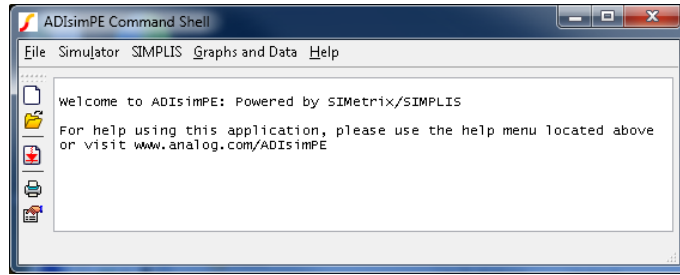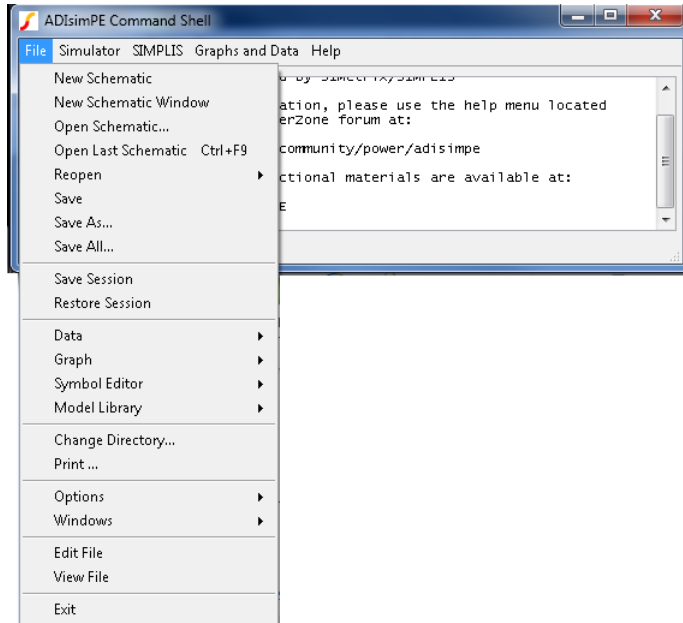
Figure 12. ADIsimPE Command Shell

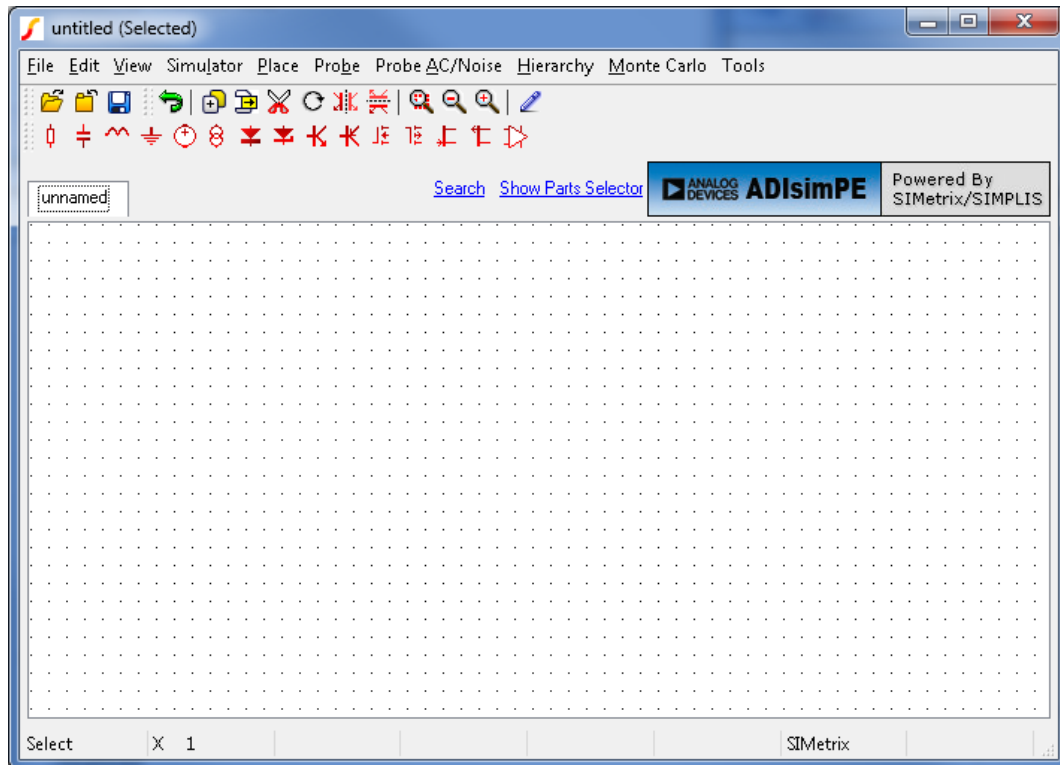*Figure 13. ADIsimPE Open New Schematic*

Figure 14. Schematic Window

UPDATING ADIsimPE

The library of schematics will continue to grow, particularly as new parts become available. ADIsimPE can be configured to regularly check for updates. To do so, from the ADIsimPE Command Shell window, click **Help** and select **Check for Updates**. The screen in Figure 15 appears. Analog Devices suggests checking for updates monthly.
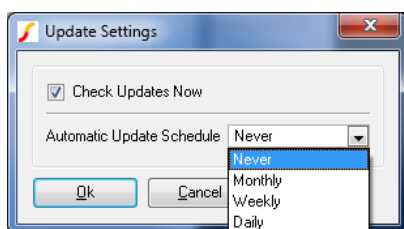

Figure 15. Update Settings

The check for updates 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 you opted to have the examples files installed, they are in a directory named in a manner that reflects its software version, such as:

C:\My Documents\ADIsimPE\Examples-72

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-72\SIMetrix\ Analog Devices**
- **C:\My Documents\ADIsimPE\Examples-72\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 your 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 ADIsimPE. Because the models are encrypted, they do not count towards the ADIsimPE limitations imposed upon schematic size for simulation.

Before inserting a model on a schematic, make sure ADIsimPE is in the proper simulation mode, because the available models are dependent upon what is being used. To change the simulation mode, in the schematics editing window, select **File > Select Simulator**. 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.
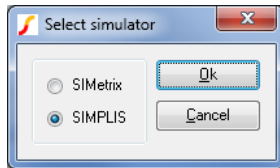

Figure 16. Select Simulator

ADIsimPE makes a 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 and click **From Analog Devices Library**.

- If you are in **SIMetrix** mode, the submenu from the pull-down will show **Operational Amplifiers**.
- If you are in **SIMPLIS** mode, the submenu from the pull-down will show **Switching Regulators** and **Other Components**.
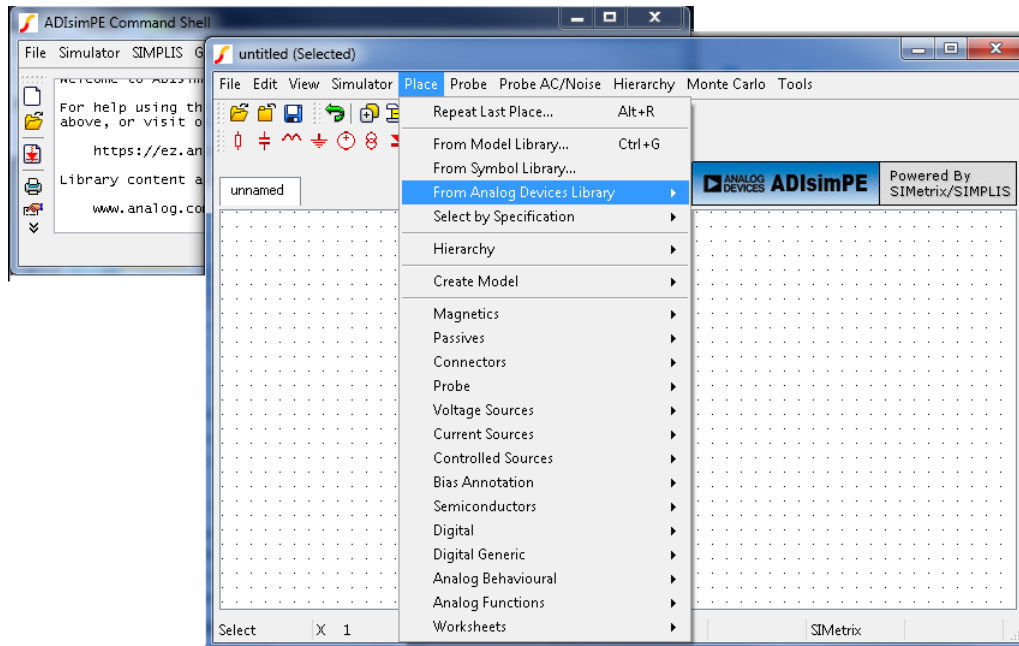

Figure 17. Analog Devices Library

Additionally, the **Show Parts Selector** link under the tool bars is a useful way to locate models for insertion into your schematic without having to go into the **Place** pull-down menu (see Figure 19).

The schematic editor allows you to insert any number of parts. However, the type of simulation (**SIMetrix** or **SIMPLIS**) restricts which models can be in the circuit for simulation. Incompatible models are highlighted in a different color (see Figure 18).
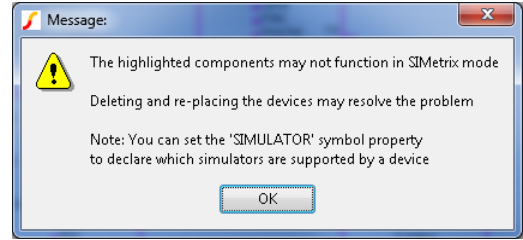

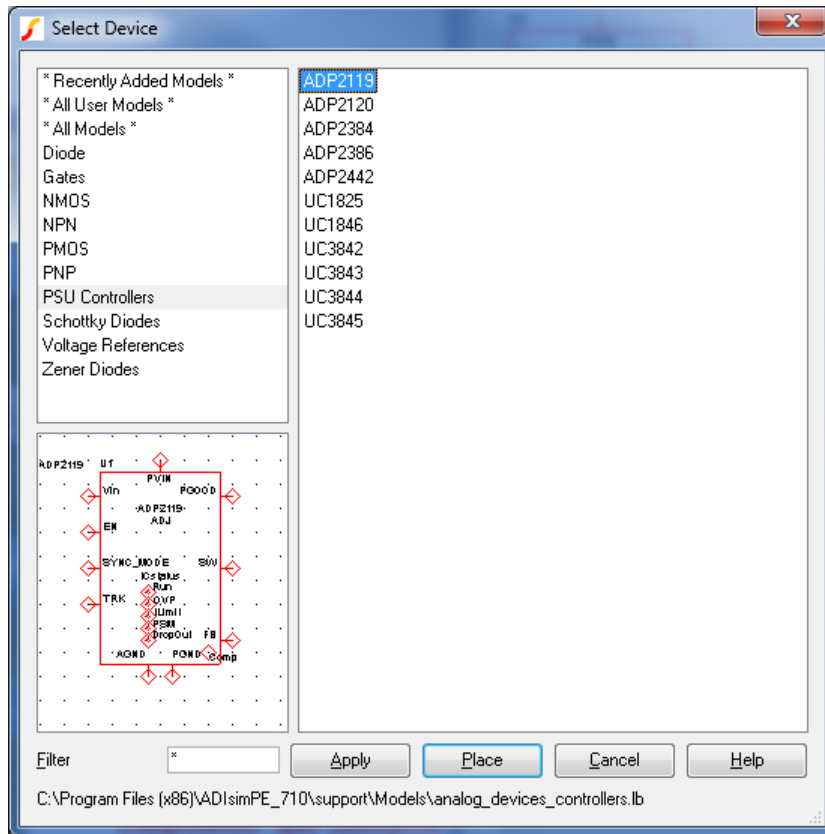*Figure 18. Error Message when Mixing Models*

Figure 19. Select Model

QUICK START STEPS FOR RUNNING SIMULATIONS

ADIsimPE offers SIMetrix/SIMPLIS tutorials. These are found in the **Command Shell** window in the **Help** menu (see Figure 20).
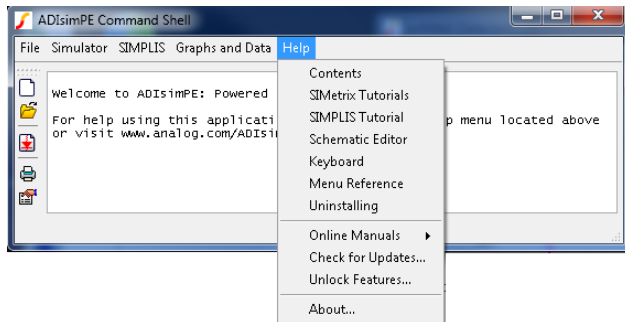

Figure 20. SIMetrix/SIMPLIS Tutorials

Another excellent path for understanding how to work with ADIsimPE is to explore the examples. The examples, if installed, can typically be found in the command shell by selecting **File** > **Open Schematic** or in the schematic window by clicking **File** > **Open**. The default location is the directory where the examples are installed.
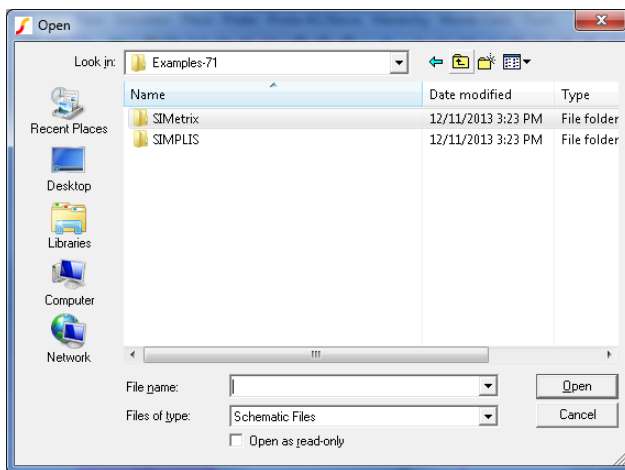

Figure 21. SIMetrix/SIMPLIS Examples

QUICK START STEPS FOR RUNNING POWER SIMULATIONS

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

1. Part and design tool selection.
2. Design and optimization.
3. Simulation.

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

ADIsimPower lets you download appropriate design tools for the selected part. These are Microsoft Excel-based.

When you run the respective ADIsimPower design tool on your local computer during Step 2 with your design criteria and settings for the part's features, the tool produces a schematic and bill of materials for your design that you can further optimize.

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

When ADIsimPE is launched from a design tool, it is ready to simulate your design immediately (press the **F9** key).

Thus, the preferred way of running SIMPLIS simulations on Analog Devices power parts is from its respective design tool, because then you inherit simulation values for:

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

The **Simulate with ADIsimPE/SIMPLIS** button from the design tool opens a dialog box to specify the desired simulation (see Figure 23).

*Figure 22. Button in ADIsimPower (Excel Tool): **Simulate with ADIsIMPE/Simplis***

The design tool's export process accesses the appropriate embedded SIMPLIS model and changes settings based on the current design state and desired simulation. The export process prompts for an appropriate directory and name for the exported file.

The default export directory is the location of the Excel design tool when started. If the design tool was 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 changed 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** and **Export & Run** buttons both save a SIMPLIS file for the user to access. The **Export & Run** button performs the additional step of starting ADIsimPE (or SIMPLIS) with the exported file. However, the **Export & Run** button is disabled if the design tool does not detect a computer registry entry for ADIsimPE (or SIMPLIS).

Simulation Options

The simulation options shown depend on (a) the design tool and (b) 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 option **Rail to Simulate** is not 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** and **Export & Run** buttons are disabled.
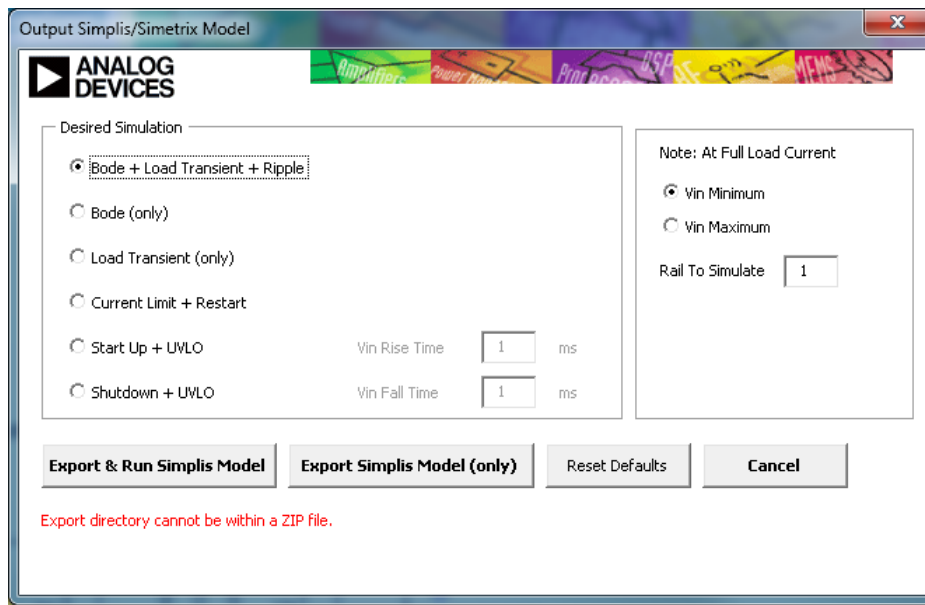

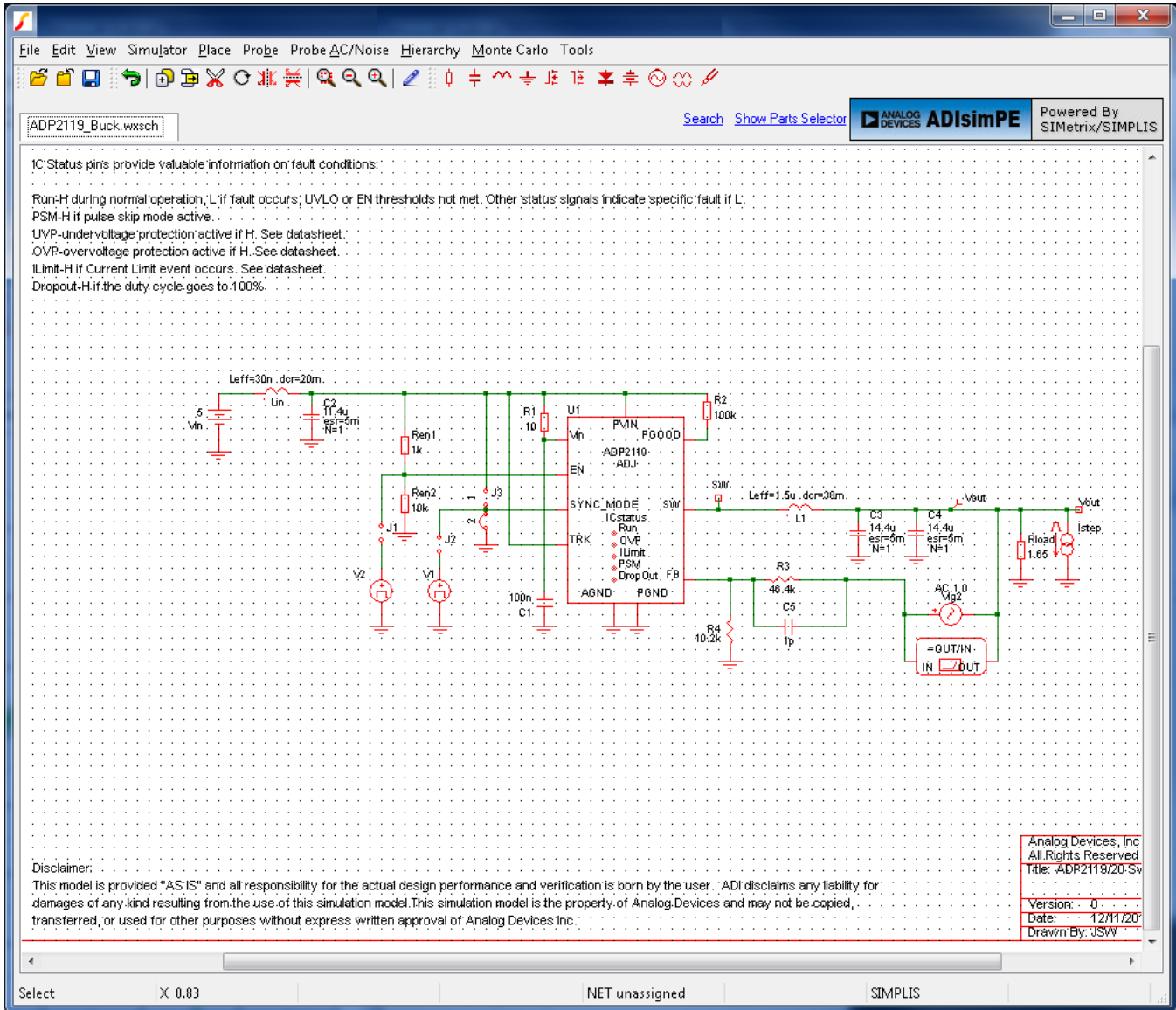Figure 23. Exporting from Analog Devices Design Tools

Figure 24. 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 you to adjust the simulation, generate results, and explore 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. You can add to the schematic load circuitry and other parts of your application and simulate how this would 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 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 an 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** item (or press the **F9** key).

Additional probes can be placed within the schematic to observe.

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's limits can be easily accessed. This dialog box is accessed when you double-click on the schematic component, or right-click and select **Edit Part** (see Figure 25).
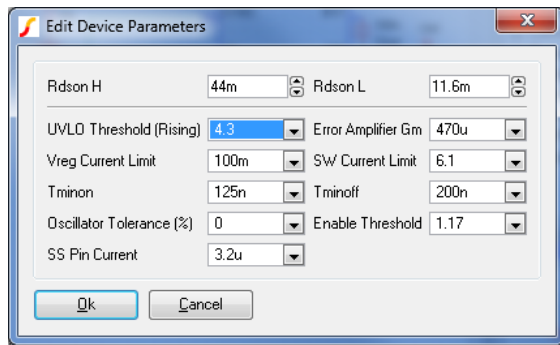

Figure 25. Edit Device Parameters: Switching Regulators

Another example is an additional dialog box associated with the parameters of the output capacitance (C_{OUT}). Often the optimum output capacitance can't 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 26).
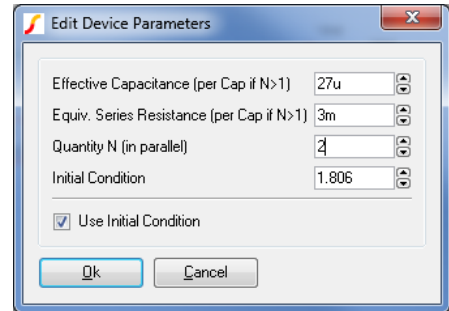

Figure 26. Edit Device Parameters: Capacitor

The **Edit Device Parameters** dialog box for this 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.

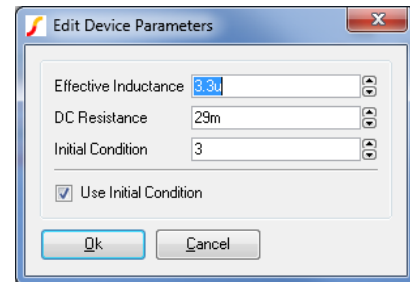The inductor is another component that has been enhanced.

Figure 27. 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 customer's larger application circuit 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 simulate the circuit. Also, 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 limits apply to what can be simulated. These include:

- A maximum of 15 additional state variables. A capacitor or an inductor each requires one state variable. Each time-varying or small-signal AC source requires 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 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. 100 topologies will be enough for simple switching circuits that use simple models only. 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