

ADIsimPE Quick Start Guide UG-706

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

2382-001

• Support from the EngineerZone

🖌 ADIsimPE
ANALOG DEVICES AHEAD OF WHAT'S POSSIBLE"
Linear and Mixed-Signal Circuit Simulator - Personal Edition
ADIsimPE powered by SIMetrix/SIMPLIS is a Personal Edition circuit simulator ideally suited to evaluate Analog Devices Linear and Mixed Signal Components. The tool includes:
 Extensive library of ADI IC models and applications schematics Full schematics capture and editing capabilities with easy waveform viewing and analysis SPICE mode SIMetrix simulator ideal for op-amps, references, Linear Regulators, and more SIMPLIS mode simulation optimized for Switching Power supplies, PLLs, and more Integration capability with ADIsimPower design tools
Assistance using ADIsimPE can be found via the Help menu or our EngineerZone Forum at: <u>ez.analog.com/community/power/adisimpe</u>
Library content and instructional materials are available at: www.analog.com/
Ok
://ez.analog.com/community/power/adisimpe
ry content and instructional materials are available at: www.analog.com/ADIsimPE

Figure 1. ADIsimPE Linear and Mixed-Signal Circuit Simulator

TABLE OF CONTENTS

Overview1	L
Revision History 2	2
Getting Started	3
Installation Procedures	3
How to Use the Software5	5
Updating ADIsimPE 7	7
Library of Analog Devices Schematics	7
Library of Analog Devices Models	3

ADIsimPE Quick Start Guide

Quick Start Steps for Running Simulations
Quick Start Steps for Running Power Simulations11
Running Simulations from Analog Devices Power Management Schematics13
Features of Analog Devices Power Management Schematics
ADIsimPE Limitations15

REVISION HISTORY

8/2016—Rev 0 to Rev. A	
Changes to Figure 1	1
Changes to Figure 2 to Figure 5	3
Changes to Figure 6 to Figure 8	4
Changes to Figure 9 to Figure 11	5
Changes to Figure 12 and Figure 13	6
Changes to Figure 14 and Figure 15	7
Changes to Library of Analog Devices Models Section,	
Figure 16, and Figure 17	8

Changes to Figure 18	9
Added Figure 19; Renumbered Sequentially	9
Changes to Figure 20	10
Changes to Figure 21 and Figure 22	
Changes to Figure 24	
Changes to Figure 25	13
Changes to Figure 26 to Figure 28	14

6/2014—Revision 0: Initial Version

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.

ADIsimPE - InstallShield Wiz	zard
2	Preparing to Install
	ADIsimPE Setup is preparing the InstallShield Wizard, which will guide you through the program setup process. Please wait.
	Preparing to Install
	Cancel

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).

Click Next to accept the license agreement with I accept 2. 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

- Select the destination location. The ADIsimPE 3. 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).

ADIsimPE - InstallShield Wizard	×
Choose Destination Location Select folder where setup will install files.	
Please enter the location where you would like to install the program. You may type a new folder name or click the Browse button to find a new location.	
C:\Program Files (x86)\ADIsimPE_800\	
Browse	
InstallShield - Cancel	

Figure 5. Choose Destination Location Screen

2382-005

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

UG-706

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

ADIsimPE - InstallShield Wizard	
Ready to Install the Program The wizard is ready to begin installation.	
Click Install to begin the installation.	
If you want to review or change any of you exit the wizard.	ur installation settings, click Back. Click Cancel to
InstallShield	
	< Back Install Cancel



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.)

ADIsimPE Quick Start Guide

ADIsimPE - InstallShield Wizard	×
Setup Status	
ADIsimPE is configuring your new software installation.	
C:\\Module_1_Examples\Modeling Blocks\CCM_Modulator.sxcmp	
InstallShield-	
	Cancel

Figure 7. Setup Status of the Installation of Files

 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).

ADIsimPE - InstallShield Wiz	ard
え	InstallShield Wizard Complete Setup has finished installing ADIsimPE on your computer.
	< Back Finish Cancel

Figure 8. Installation Finished

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

12382-011

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).



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.

🖌 Starting ADIsimPE 8.00i for the First Time 🛛 💌
Preferences from earlier version
Would you like to migrate the configuration from your earlier version of ADIsimPE ?
Most configuration settings will be copied including installed models and symbols along with preference settings. A log file showing what is copied will be created during this No
process
Ok

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.

ADIsimPE
ANALOG DEVICES ALEAT OF WHAT'S POSSIBLE"
Linear and Mixed-Signal Circuit Simulator - Personal Edition
ADIsimPE powered by SIMetrix/SIMPLIS is a Personal Edition circuit simulator ideally suited to evaluate Analog Devices Linear and Mixed Signal Components.
The tool includes:
Extensive norm of Not to induces and opplications successful and the set of the set
Assistance using ADIsimPE can be found via the Help menu or our EngineerZone Forum at: <u>ez.analog.com/community/power/adisimpe</u>
Library content and instructional materials are available at: www.analog.com/
Ok
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.

2382-010



Figure 12. ADIsimPE Command Shell Window

AHEAD OF WHAT'S POSSIBLE*	OlsimPE
Recent Schematics	ADIsimPE
ADP1741 PSRR vs Headroom 1.3Vout 100mA.sxsch	Example Circuits
ADP1741 1.75Vin 0.75Vout PSRR vs Iout 1Vheadroom.sxsch	🚱 Analog Devices Home Page
🧭 Open	🚱 Amplifier and Linear Design Tools
	SPICE Models for Download
	🚱 ADIsimPE Home Page
Create New Schematic Symbol	
Putting the Future First Analog Devices is proud to support FIRST® Robotics Competition. Learn More	
Default Simulator © SIMetrix © SIMPLIS	

12382-013

12382-012

Figure 13. ADIsimPE Open New Schematic Window

ADIsimPE Quick Start Guide

ADIsimPE Main Window		- • •
le Edit View Simulator Place Probe	Probe <u>A</u> C/Noise Hierarchy <u>Monte Carlo Tools Help</u>	Web View View Schematic Editor
) - 📄 🗙 🖬 🖨 📾 🏘 🏷 🥐 🗗	╡╳Ѻ╨美 ᡚQQQQ ∥ ▶ ┣│≬≑∽ᅷ♡╡ᆍᆃҠ⊀非非上午於兆 〃ᄽ№ –	
le View	untitled	- X
Add Directory		
European and		
Examples-80		
Open Example Schematic		
Colorter Command Chall The Manual		
Selector Command Shell File View		
imPE		
	1	
► DEVICES ADISIMPE		
NEAD OF WHIE'S POSSIBLE*		
alog and Mixed-Signal Circuit Simulator		
Analog Devices, Inc.		
Eastern Connect Connection		
Engineerzone support Community		
Powered by SIMetrix/SIMPLIS	Welcome to ADI3mPE untitled	
t X1	SEMetrix	

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.

🖌 Update Settings
Check Updates Now
Automatic Update Schedule Monthly -
Ok Cancel
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.

UG-706

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).

Choose Analysis	F8
Run Schematic	F9
C <u>h</u> eck	
Resu <u>m</u> e	
Restart Transient	
Pause	
Abort	
Convergence Options	
Manage Data Groups	
View/Edit Netlist	
Open List File	
Import Models	
Open/Close Command (F11) Window	F11
Create Netlist as <u>S</u> ubcircuit	
Create Netlist as SPICE 2 Subcircuit	
Show Statistics	

Figure 16. Select Simulator

Switch to SIMPLIS Mode

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).

ADIsimPE Main Window					
ile Edit View Simulator	Place Probe Probe AC/Noise Hierarchy	Monte Carlo Tools Help			Web View Schematic Editor
) 🗕 📷 X 🔚 🎒 🖮 🌣	Repeat Last Place Alt+R	Q. 🖉 🕨 II 🖄 0. ≑ ·	~ ÷ © 0 ≇ ≇ ⊀ ⊀ .	lE LE 🗜 🕆 🎲 HK 🖌 🖊 🔂 🕀	
File View	From Model Library Ctrl+G				₹×
Add Directory	From Symbol Library				
And Directory	From Analog Devices Library	Amplifiers +	Current Sense Amplifier		
Examples-80	Select by Specification	Other Components	Current Sense Ampliner		
Open Example Schematic	Search Part	Special Linear Functions	Eville Differential Amplifier		
		Switches	Fully Differential Amplifier		
	Hierarchy	Voltage Defenses	Instrumentation Amplifier		
	Create Model	voltage References	Operational Amplifier		
	Magnetics				
	Passives				
	Connectors				
	Probe				
	Voltage Sources				
	Current Sources				
	Controlled Sources				
	Bias Annotation				
	Semiconductors +				
	Digital				
	Digital Generic				
	Analog Behavioural				
	Analog Functions				
	Worksheets				
art Selector Command Shell	Tennenten				
DIsimPE					
	imPF				
ACCO IT MARTS FORMAT"					
Analog and Mixed-Signal Circu	uit Simulator				
Analog Devices, In	ic.				
EngineerZone Support Co	mmunity				
Powered by SIM	Metrix/SIMPLIS Welcome to ADIsimPE u	ntitled			
lect X1				SIMetrix	

2382-010

Figure 17. Analog Devices Library

Rev. A | Page 8 of 15

12382-119

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).



2382-018

Figure 19. Part Selector Tab

🖌 Select Device		×
* Recently Added Models *	4016 ADG441B ADG609 ADG819	MAX4890
* All User Models *	4066 ADG441BS ADG611 ADG820	MAX4891
* All Models *	ADG1211 ADG441S ADG612 ADG821	MAX4892
Analog switches	ADG1212 ADG441T ADG613 ADG822	
BJT - Dual 🗏	ADG1213 ADG441TS ADG658 ADG823	
BJT - Multiple	ADG1236 ADG442 ADG659 ADG839	
Bridge Rectifiers	ADG1311 ADG442B ADG661 ADG841	
Buffers	ADG1312 ADG442BS ADG662 ADG842	
Comparators	ADG1313 ADG442S ADG663 ADG849	
Counters	ADG406 ADG444 ADG701 ADG859	
Current Mirrors	ADG406B ADG444B ADG702 ADG884	
Current Monitors	ADG407 ADG444BS ADG711 CLC532	
Current Sense Amplifiers	ADG407B ADG444S ADG712 HC4016	
Differential Amplifiers	ADG408 ADG508F ADG713 HC4051	
Digital Arithmetic	ADG408B ADG508FB ADG719 HC4052	
Digital Buffers	ADG408T ADG508FT ADG721 HC4053	
Digital Functions	ADG409 ADG509F ADG722 HC4066	
Digital Multiplexers/Demultiplexer	ADG409B ADG509FB ADG723 HC4067	
Digital/Analog interfaces	ADG409T ADG509FT ADG741 HC4316	
Diode	ADG411 ADG511_3 ADG742 HC4351	
Drivers	ADG411B ADG511_5 ADG749 HC4352	
Dual Diodes 👻	ADG411T ADG512_3 ADG779 MAX4601	
4 III >	ADG412 ADG512_5 ADG781 MAX4602	
	ADG412B ADG513_3 ADG782 MAX4603	
· · · · · · · · · · · · · · · · · · ·	ADG412T ADG513_5 ADG783 MAX4684	
VDD ADG1212	ADG413 ADG528F ADG787 MAX4685	
	ADG413B ADG528FB ADG801 MAX4714	
D4	ADG413T ADG528FT ADG802 MAX4717	
INZ	ADG426 ADG601 ADG811 MAX4718	
IN4	ADG426B ADG602 ADG812 MAX4719	
diversity of the second	ADG441 ADG604 ADG813 MAX4766C	
• • • • • • • • • • • • • • • • • • • •		
rik		
Filter	Plac	e Cancei Help
C:\Program Files (x86)\ADIsimPE_800\suppor	t\Models\analog.lb	

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).

<u>H</u> el	р	
	SIMetrix Documentation	
	SIMPLIS Documentation	
	SIMetrix Tutorials	
	SIMPLIS Tutorial	
	Schematic Editor	
	Select Help Source	
	PDF Manuals	
	Check for Updates	
	Unlock Features	
	<u>A</u> bout	12382-020

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.

Organize • New folder				· · ·	0
★ Favorites	Name	Date modified	Type	Size	
Desktop	🗼 SIMetrix	6/1/2016 2:02 PM	File folder		
Downloads Recent Places	I. SIMPLIS	6/1/2016 2:05 PM	File folder		
Ubraries Documents Music Pictures Videos					
S Computer					
K Network					
File name				hematic Files (1 www.h.t	

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).

Simulate with ADIsimPE/Simplis

2382-022

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.

UG-706

2382-023

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.

Bode + Load Transient + Ripp O Bode (only)	ple			Note: A	t Full Load Current Iinimum
C Load Transient (only)				⊖ Vin ≀	Maximum
Current Limit + Restart	Vin Rise Time	1	ms		
C Shutdown + UVLO	Vin Fall Time	1	ms		
cport & Run Simplis Model	Export Simplis Mode	l (only)	Reset D	efaults	Cancel

ADIsimPE Quick Start Guide



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.

UG-706

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).

Rdson H	445m		Rdson L	298.8m	
ic	ADP2370	•	UVLO Threshold (Rising)	3.19	•
SW Current Limit	1.2	+	Tminon	65n	*
Oscillator Tolerance (%)	0	-	Enable Threshold	1.2	-
Fixed or Adjustable Output	ADJ	•			

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).

Edit Device Parameters X Effective Capacitance 7.46u Equiv. Series Resistance 2m Quantity (in parallel) 1 Initial Condition -4 Use Initial Condition Ok Cancel 2382-026

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).

📕 Edit Device Param	neters	×			
Effective Inductance	12u				
DC Resistance	65.7m				
Initial Condition	0				
Use Initial Condition					
Ok Cancel					

Figure 28. Edit Device Parameters: Inductors

ADIsimPE Quick Start Guide

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.

Legal Terms and Conditions

Information furnished by Analog Devices is believed to be accurate and reliable. However, no responsibility is assumed by Analog Devices for its use, nor for any infringements of patents or other rights of third parties that may result from its use. No license is granted by implication or otherwise under any patent or patent rights of Analog Devices. Trademarks and registered trademarks are the property of their respective owners. Information contained within this document is subject to change without notice. Software or hardware provided by Analog Devices may not be disassembled, decompiled or reverse engineered. Analog Devices' standard terms and conditions for products purchased from Analog Devices can be found at: http://www.analog.com/en/content/analog_devices_terms_and_conditions/fca.html.

©2014–2016 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners. UG12382-0-8/16(A)



www.analog.com