нстр мент   проектировани   и кальк л тор

писание

По вопросам продаж и поддержки обращайтесь:

Design Tools & Calculators

Analog Devices' Design Tools simplify your design and product selection process through ease of use and by simulating results that are optimized and tested for accuracy. Analog Devices Design tools are web based or downloadable but always free to use.

- Reduce your testing time and get to the finished product faster
- Overcome engineering challenges and simplify calculations
- Use industry leading products to create the best design
- Be confident in the results and your solution

SPICE

LTspice®

ADI Certified
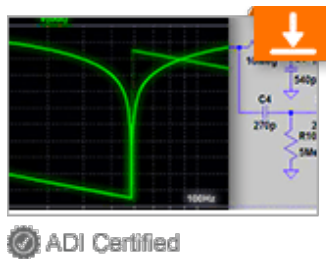
LTspice® is a high performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits. Included in the download of LTspice are macromodels for a majority of Analog Devices switching regulators, amplifiers, as well as a library of devices for general circuit simulation.

SPICE Models
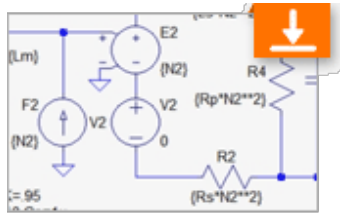
A collection of SPICE simulation models for Analog Devices' products

EE-Sim® Power Tools

EE-Sim® DC-DC Design Tool

ADI Certified
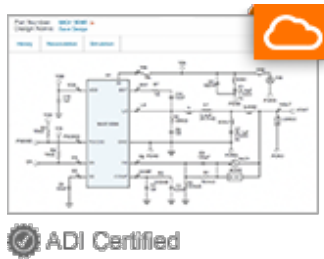
The EE-Sim® DC-to-DC Converter Tool uses your requirements to quickly create a complete power design including a schematic, BOM with commercially available parts, and time and frequency domain simulations.

EE-Sim OASIS Simulation Tool
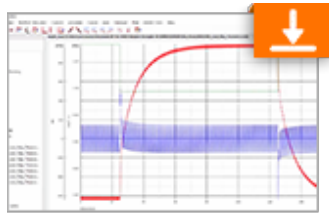
To enable engineers to quickly and accurately simulate their circuit designs, Analog Devices offers the EE-Sim OASIS simulation tool. It includes both the SIMPLIS simulation engine, which is the industry's leading simulator for switched-mode power ICs, and the SIMetrix SPICE simulation engine, known for its robust and speedy convergence.

EE-Sim Solution Finder

The EE-Sim Solution Finder is a cutting edge part selection tool that enables you to find the right switching power supply faster. Why is the solution finder better than a parametric search? The solution finder validates your design requirements to ensure the suitability of each recommended part. It also compares the efficiency, size, and cost of each recommended solution—taking into account the complete solution BOM, not just the IC.

Amplifier and Linear Tools

Signal Chain Designer - BETA

ADI Certified

Signal Chain Designer is a web-based tool designed to create and simulate complex precision signal chains. See your circuit's performance before you commit to your PCB: transfer function, noise, power consumption, input range, and DC error. Quickly experiment with different parts and architectures. Signal chains can be exported to LTspice for further analysis.

ADI Precision Studio

ADI Certified

A suite of web tools to help you design signal conditioning circuits faster: Analog Filter Wizard, Precision ADC Driver Tool, Photodiode Wizard, In Amp Diamond Plot, Direct Digital Synthesis Simulator, and Virtual Eval.

ADI DiffAmpCalc™
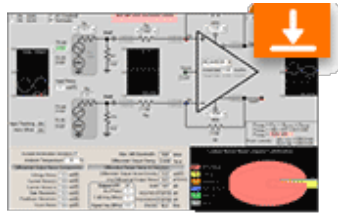
Download ADI DiffAmpCalc™ for designing differential amplifier circuits and reduce design time from hours to minutes! The tool is easy to use and features an interactive user interface to quickly get you up and running.

Analog Filter Wizard

ADI Certified

Use the Analog Filter Wizard to design low-pass, high-pass, or band-pass filters with actual op amps in minutes. As you progress through the design process, you can observe the characteristics of your filter design from ideal specifications to real world circuit behavior. Quickly evaluate the tradeoffs in op amp specifications - including gain-bandwidth, noise, and supply current – to determine the best filter design for your requirements.

Analog Photodiode Wizard

ADI Certified

Use Photodiode Wizard to design a transimpedance amplifier circuit to interface with a photodiode. Select a photodiode from the library included in the tool, or enter custom photodiode specifications. Quickly observe tradeoffs between Bandwidth, Peaking (Q), and ENOB/SNR. Modify circuit parameters, and immediately see results in plots for pulse response, frequency response, and noise gain.

Instrumentation Amplifier Diamond Plot Tool

ADI Certified

Instrumentation amplifier datasheets typically show a graph (or several variations) of the Output Swing vs Input Common-Mode Voltage, also known as the Diamond Plot, which is a comprehensive graph of all external and internal headroom limits. Given enough basic information about a circuit, the Diamond Plot can be adjusted for various Supply Voltages, Gains, and Reference Pin Voltages.

Additional Tools

- In-Amp Error Calculator
- OpAmp Error Budget Calculator
- Power Dissipation vs Die Temp

Clock and Timing Tools

TimerBlox® Designer

TimerBlox® parts are small, accurate and simple timing devices, designed for 5 basic operations: Voltage-Controlled Oscillation (VCO), Low Frequency Clocking, Pulse-Width Modulation (PWM), One-Shot Generation and Signal Delay.

To speed and simplify your design process, TimerBlox Designer is an Excel based selection and synthesis tool that allows you to choose and configure the TimerBlox part best suited for your application.

ADIsimCLK™
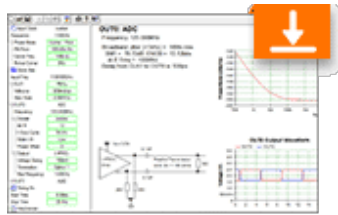
ADIsimCLK™ is the design tool developed specifically for Analog Devices' range of ultra-low jitter clock distribution and clock generation products. Whether your application is in wireless infrastructure, instrumentation, networking, broadband, ATE or other areas demanding predictable clock performance, ADIsimCLK will enable you to rapidly develop, evaluate and optimize your design. It is a highly successful tool for predicting phase noise and jitter for ADI clock products.

ADIsimDDS

ADI Certified

Use ADIsimDDS to select and evaluate the performance of a direct digital synthesis device, including the impact of an external reconstruction filter. View the frequency location and magnitude of resulting harmonic images, and determine the filter requirements needed to achieve target performance of resulting output signal.

Additional Tools

- OscCalc – BETA
- ADIsimPLL™
- Real Time Clock Calculator
- Battery Life Calculator

Data Converter Tools

Visual Analog™
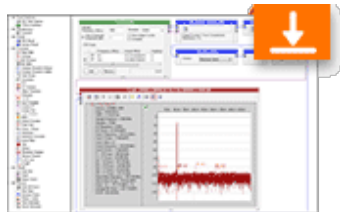
For designers who are selecting or evaluating high speed ADCs, VisualAnalog™ is a software package that combines a powerful set of simulation and data analysis tools with a user-friendly graphical interface. VisualAnalog allows designers to customize their input signal and data analysis.

Virtual Eval - BETA

ADI Certified

Virtual Eval is a web application to assist designers in product evaluation of ADCs and DACs. Using detailed models on Analog's servers, Virtual Eval simulates crucial part performance characteristics within seconds. Configure operating conditions such as input tones and external jitter, as well as device features like gain or digital down-conversion. Performance characteristics include noise, distortion, and resolution, FFTs, timing diagrams, frequency response plots, and more.

Precision ADC Driver Tool

Use the Precision ADC Driver Tool to simulate the performance of precision ADC and driver combinations. Potential issues with driver selection, kickback settling, and distortion are flagged, and design tradeoffs can be quickly evaluated. Simulations and calculations include system noise, distortion, and settling of the ADC input.

Precision DAC Error Budget Tool

ADI Certified

The Precision DAC Error Budget Tool is a web application that calculates the DC Accuracy of precision DAC signal chains. It shows how the static errors accumulate throughout your signal chain to quickly evaluate the design tradeoffs. Calculations include the DC errors introduced by Voltage References, Operation Amplifiers and Precision DACs.

Additional Tools

- DigiPtMeter (Linear-to-Log pot) (ZIP)
- Sigma Delta Calculator (XLS)
- RTD Configurator and Error Budget Calculator (EXE)
- Frequency Folding Tool
- Sigma-Delta ADC Tutorial
- SNR/THD/SINAD Calculator
- SPIController
- Data Conversion Calculator
- Jitter Calculator
- Coherent Sampling Calculator

Power Management Tools

LTpowerCAD® & LTpowerPlanner©

ADI Certified
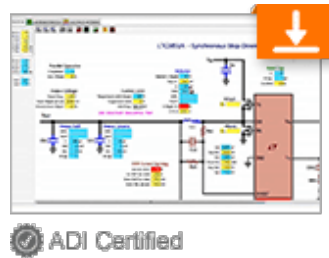
The LTpowerCAD® design tool is a complete power supply design tool program that can significantly ease the tasks of power supply design with µModule regulators and many other products, mostly monolithic buck step-down regulators. Unlike conventional simulation tools, this LTpowerCAD tool guides users throughout the whole supply design process: it searches suitable parts according to user's supply specifications; then it guides the user to select and optimize circuit component values with suggestions and warnings. The tool shows real-time results of feedback loop bode plot, as well as the power stage performance. The design can be exported to LTspice simulation circuit for users to further verify their designs. The tool also provides PCB layout examples. **Within the LTpowerCAD program, there is also a LTpowerPlanner tool for system-level "power tree" designs and optimizations.**

LTpowerPlay®

ADI Certified

LTpowerPlay® is a powerful development environment supporting the Power System Management (PSM) products of Analog Devices, including PMBus Power System Managers and DC/DC Power Converters with PSM. The software supports a variety of different tasks, including an offline mode (with no hardware present) in order to build a multi-chip configuration file that can be saved and reloaded at a later time.

ADI Power Studio®

ADI Certified

ADI Power Studio® is a powerful, Windows-based development environment supporting Analog Devices' Super Sequencer® products. ADI Power Studio supports a variety of different tasks, such as evaluating Analog Devices' ICs by connecting to a demo board system, and, in an offline mode (with no hardware present), building a multichip configuration file that can be saved and reloaded at a later time.

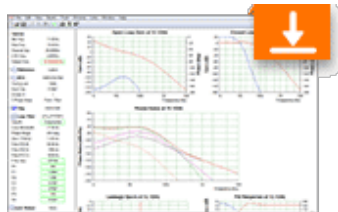**Additional Tools**

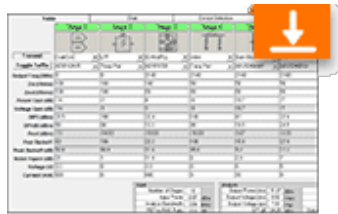- Variable Buck Resistor Calculator (XLSX)
- Power Dissipation vs Die Temp

RF and Synthesis Tools

ADIsimPLL™

The ADIsimPLL design tool is a comprehensive and easy-to-use PLL synthesizer design and simulation tool. All key non-linear effects that can impact PLL performance can be simulated, including phase noise, Fractional-N spurs, and anti-backlash pulse.

ADIsimRF™

ADIsimRF is an easy-to-use RF signal chain calculator. It calculates Cascaded Gain, Noise Figure, IP3, P1dB and Power Consumption. The number of stages can be varied up to a maximum of 20. Stages can be easily inserted, removed or temporarily muted.

Additional Tools

- Wireless Cascaded Gain and Noise Figure (ZIP)
- ISM-RF Crystal Calculator (XLS)
- Calculating Equivalent Parallel Impedance (ZIP)
- Calculating Transmitted Power Using Return Loss (ZIP)
- ISM-RF Baseband Calculator (XLS)
- Power Conversion Table (RF Power Conversion) (ZIP)
- ADIsimCLK
- ADIsimDDS
- ADIsimFrequency Planner Tool
- ADIsimSRD Design Studio
- RF Impedance Matching Calculator
- VRMS/dBm/dBu/dBV calculators

Cybersecurity

1-Wire Calculator

This tool aides in determining the proper pullup resistor size for a 1-Wire system by taking into consideration the system's supply voltage, tolerance, along with parameters for both the host and target.

DeepCover® Security Lab

The DeepCover Security Lab is an interactive tool to demonstrate how to successfully apply security protocols in your system or product. It is designed to help you understand the importance of security, learn cryptographic terminology and apply these principles using Analog's latest Secure Authenticator products.

JESD204 Receiver Transport Layer Design File Package

This command line executable tool generates a Verilog module which implements the JESD204 receive transport layer. The user specifies in a configuration file one or more modes to be supported by the transport layer module. These modes are defined as a set of JESD204 parameter values: L, M, F, S, N', and CF. The transport layer converts JESD204 lane data output from a JESD204 link layer IP to a data bus with a fixed width, containing interleaved virtual converter samples. Both JESD204B and JESD204C link layers are supported.

По вопросам продаж и поддержки обращайтесь:

Алматы (7273)495-231
Ангарск (3955)60-70-56
Архангельск (8182)63-90-72
Астрахань (8512)99-46-04
Барнаул (3852)73-04-60
Белгород (4722)40-23-64
Благовещенск (4162)22-76-07
Брянск (4832)59-03-52
Владивосток (423)249-28-31
Владикавказ (8672)28-90-48
Владимир (4922)49-43-18
Волгоград (844)278-03-48
Вологда (8172)26-41-59
Воронеж (473)204-51-73
Екатеринбург (343)384-55-89

Иваново (4932)77-34-06
Ижевск (3412)26-03-58
Иркутск (395)279-98-46
Казань (843)206-01-48
Калининград (4012)72-03-81
Калуга (4842)92-23-67
Кемерово (3842)65-04-62
Киров (8332)68-02-04
Коломна (4966)23-41-49
Кострома (4942)77-07-48
Краснодар (861)203-40-90
Красноярск (391)204-63-61
Курск (4712)77-13-04
Курган (3522)50-90-47
Липецк (4742)52-20-81

Магнитогорск (3519)55-03-13
Москва (495)268-04-70
Мурманск (8152)59-64-93
Набережные Челны (8552)20-53-41
Нижний Новгород (831)429-08-12
Новокузнецк (3843)20-46-81
Ноябрьск (3496)41-32-12
Новосибирск (383)227-86-73
Омск (3812)21-46-40
Орел (4862)44-53-42
Оренбург (3532)37-68-04
Пенза (8412)22-31-16
Петрозаводск (8142)55-98-37
Псков (8112)59-10-37
Пермь (342)205-81-47

Ростов-на-Дону (863)308-18-15
Рязань (4912)46-61-64
Самара (846)206-03-16
Санкт-Петербург (812)309-46-40
Саратов (845)249-38-78
Севастополь (8692)22-31-93
Саранск (8342)22-96-24
Симферополь (3652)67-13-56
Смоленск (4812)29-41-54
Сочи (862)225-72-31
Ставрополь (8652)20-65-13
Сургут (3462)77-98-35
Сыктывкар (8212)25-95-17
Тамбов (4752)50-40-97
Тверь (4822)63-31-35

Тольятти (8482)63-91-07
Томск (3822)98-41-53
Тула (4872)33-79-87
Тюмень (3452)66-21-18
Ульяновск (8422)24-23-59
Улан-Удэ (3012)59-97-51
Уфа (347)229-48-12
Хабаровск (4212)92-98-04
Чебоксары (8352)28-53-07
Челябинск (351)202-03-61
Череповец (8202)49-02-64
Чита (3022)38-34-83
Якутск (4112)23-90-97
Ярославль (4852)69-52-93

Россия +7(495)268-04-70   Казахстан +7(7172)727-132   Киргизия +996(312)96-26-47

эл.почта: agu@nt-rt.ru || сайт: https://analog.nt-rt.ru/