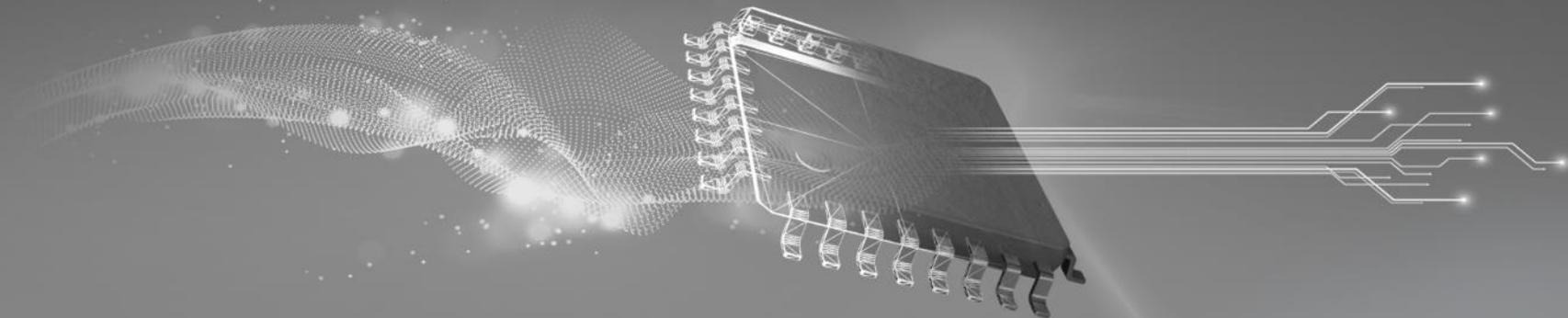


TI TECH DAYS



Getting started with the new PSpice[®] for TI design and simulation tool

Ian Williams

APP – LP – LDO

About me – Ian Williams

Business lead, Low-Voltage LDOs

- **Career**

- BSEE University of Texas at Dallas, 2009
- TI since 2009, LDO since July 2020



- **Expertise**

- 11 years working with various types of amplifiers
- Co-creator of GWL amplifier SPICE model architecture
- Co-creator of TI Precision Labs – Op Amps

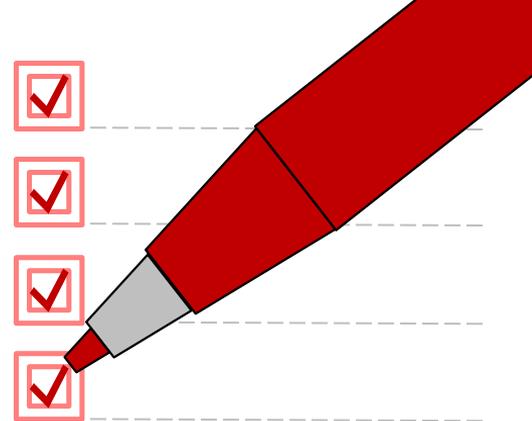
- **Fun fact**

- Big music guy – have performed at festivals, DJ'd at clubs and on FM radio, and even met my wife at Coachella 2013



Agenda

- TI simulation tools overview – **10 min.**
- PSpice® for TI deep dive – **10 min.**
 - Features and limitations
 - Built-in model library
- Setup and simulation examples – **25 min.**
 - Operational amplifier: OPA211
 - Power supply: TPS7A52
 - Modeling Application: Power MOSFET
- Additional resources



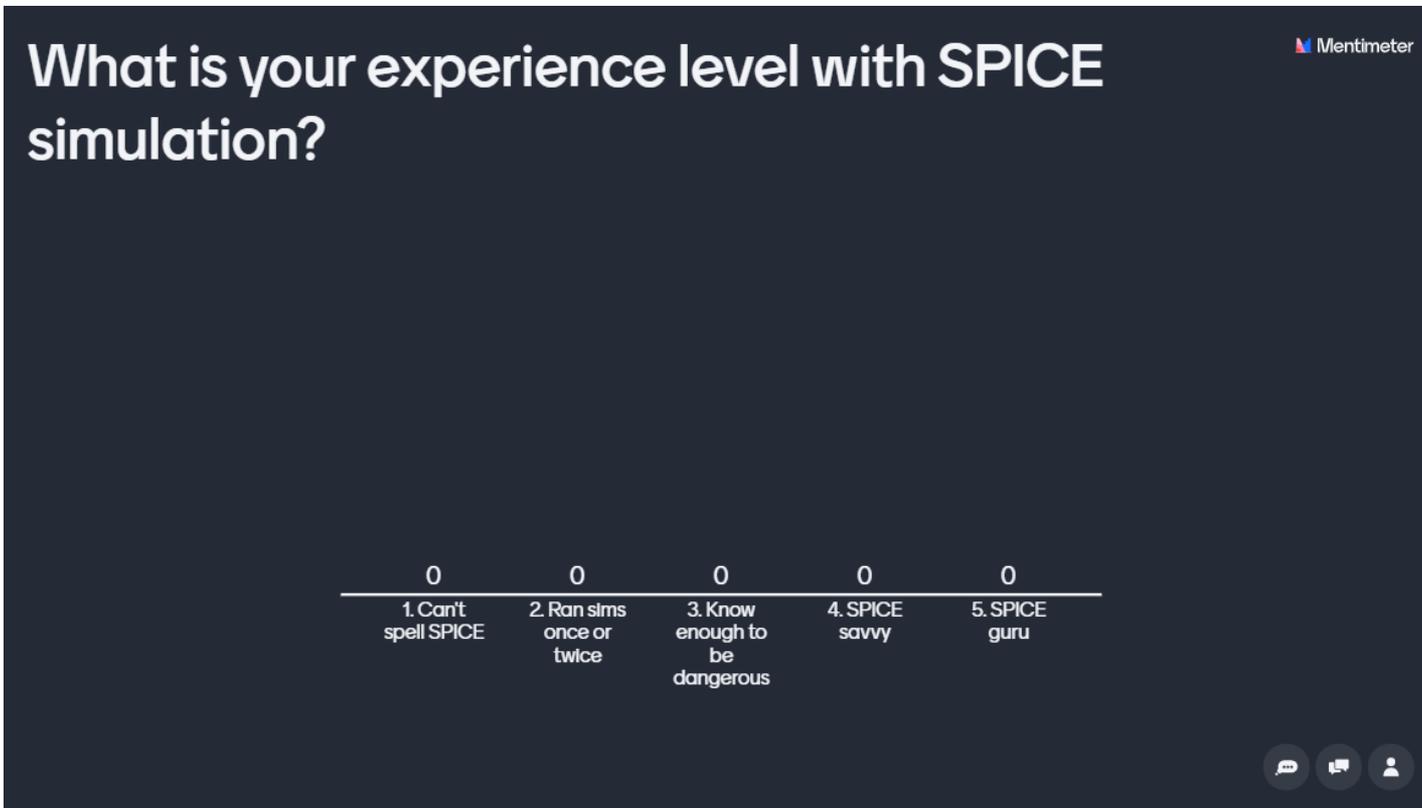
*Please ask your questions
in the chat!*

Part 1

TI simulation tools overview

Tip: SPICE stands for “Simulation Program with Integrated Circuit Emphasis”

Time for some audience participation...



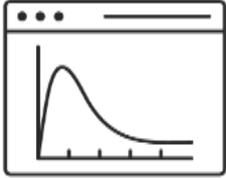
Introducing PSpice® for TI



PSpice for TI will help engineers speed time to market and reduce development costs, delivering:

- **Full-featured simulation of entire systems.**
 - Advanced capabilities, including Monte Carlo and worst-case analysis.
 - Synchronized library of >5,700 models and counting.
 - No design size limitations.
 - Easy transition to layout and prototype.
- **Integrated design resources.**
 - Quick access to TI product information.
 - No need to manually upload new TI models.

Why is TI partnering with Cadence?



Growing demand

There is an increased need for simulation software to test new design concepts, accelerate product development and demonstrate regulatory compliance.

Source: [ADI Research](#)

Short design timelines

Today's design engineers must produce accurate designs on tight deadlines — in many cases, reducing the prototyping and evaluation phases of their designs.



Desire for more advanced simulation

Existing simulation tools in the market lack advanced analysis capabilities, model portability and flexibility, and easy library synchronization.



“Tools that are intuitive and include system-level simulation capabilities can cut the development time and speed time to market.” – Kevin Anderson, Omdia

Is PSpice for TI replacing other TI tools?



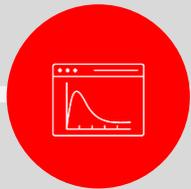
Fundamentals & skill-building

- Educational e-books
- [Technical articles](#)
- [TI Precision Labs](#)
- [Power Supply Design Seminars](#) (PSDS)
- Additional videos at [training.ti.com](#)



Investigation & brainstorming

- Easy part selection on TI.com
- [Reference designs](#) for specific applications
- Application notes and technical white papers



Design & simulation

- Evaluation modules
- [WEBENCH® Power Designer](#)
- [Filter Design Tool](#)
- [Analog Engineer's Calculator, Circuit Cookbooks & Pocket Reference](#)
- [TINA-TI™](#)



PSpice® for TI



Design support

- [e2e.ti.com](#)
- Forums for expert answers to technical questions

PSpice for TI vs. TINA-TI

	PSpice for TI	TINA-TI
Analysis / simulation		
AC, Noise, BIAS point, DC sweep, Transient, Fourier	Yes	Yes
Variable sweeps	Temperature, component, parametric	Temperature, component
Monte Carlo analysis	Yes	No
Worst case analysis	Yes	No
Libraries / components		
Internal libraries	~5700	~1300
Automatic library updates	Yes	No
Built-in modeling application	Yes	No
Schematics		
Create hierarchical schematic	Yes	No
Multipage schematics	Yes	No

Part 2

PSpice for TI deep dive

Tip: PSpice for TI runs offline!

Software features and limitations FAQ

Q. Does PSpice for TI work offline?

A. Yes, an internet connection is not required to run.

Q. Is there a maximum number of nodes?

A. No, there are no design size limitations. You can also use multi-page schematics and hierarchical blocks.

Q. Are there any other limits to be aware of?

A. Yes. The tool is designed primarily as a SPICE simulation environment for use with the built-in TI models. If third-party models are imported, then only three nodes can be probed simultaneously.

Built-in TI model library

INA293A1 PSpice model based on Green-Williams-Lis architecture
Tested using Cadence 17.4
Test date: 05/28/2020

DC = 12.5
IOFF = 12.5
IAMPL = 1
FREQ = 10k

PSpice Part Search

Hide Categories

Categories Library

- ▶ Texas Instruments
 - ▶ Amplifiers
 - ▶ Audio
 - ▶ Clocks & timing
 - ▶ Data converters
 - ▶ Interface
 - ▶ Isolation
 - ▶ Logic
 - ▶ Microcontrollers (MCU)
 - ▶ Motor drivers
 - ▶ Power management
 - ▶ RF & microwave
 - ▶ Sensors
 - ▶ Switches & multiplexers

Search All Categories

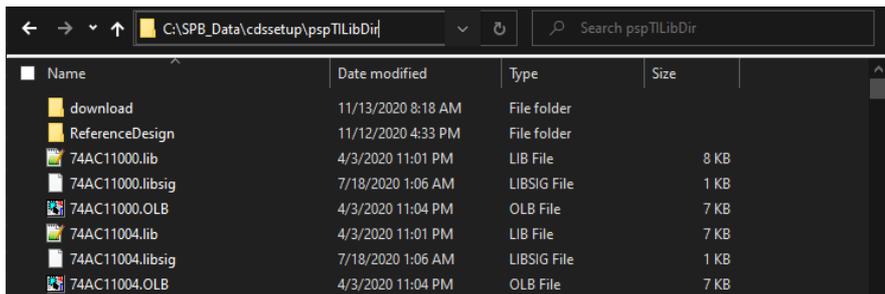
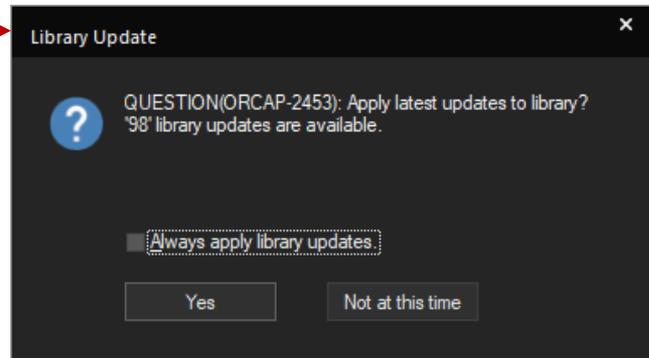
PART NAME	DESCRIPTION
"INA293A1"	"-4 to 110V, 1MHz, ultra-precise current sense amplifier Simulation Model"

Matches the product tree on [TI.com](https://www.ti.com)

Includes device-specific test benches
To accelerate your development

How do I update the TI model library?

- Updates to the model library are automatically detected and performed at software startup.
 - The user can choose not to update.
- The model library is installed locally on the user's hard drive. Models can be copied and imported into other tools if desired.
 - Location: **C:\SPB_Data\cdssetup\pspTILibDir**



Are models editable?

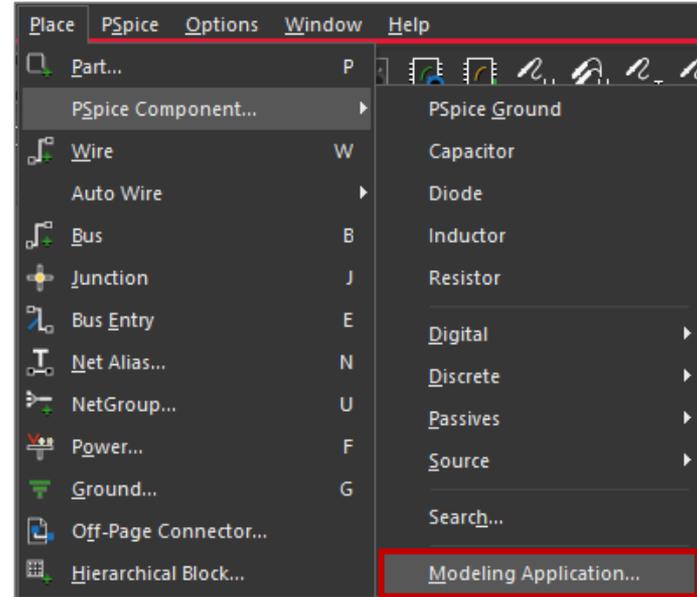
- Models are text files and may be edited with a text editor (I recommend [Notepad++](#)) from the library directory
 - **Note:** editing TI models breaks their signature, causing the tool to treat them as 3rd-party
- Some models have user-editable parameters. Editing these **does not** break signature.
- If you edit a model and save it in the same location, it will get over-written during the next library update.

```
37 .SUBCKT 74AC11000 Y A B VCC AGND
38 XU1 Y A B VCC AGND LOGIC_GATE_2PIN_OD_LVC_2i_NAND_PP_CMOS_74AC11000
39 .ENDS
40 .SUBCKT LOGIC_GATE_2PIN_OD_LVC_2i_NAND_PP_CMOS_74AC11000 OUT A B VCC GND
41 .PARAM VCC_ABS_MAX = 7
42 .PARAM VCC_MAX = 5.5
43 .PARAM RA = 220000000
44 .PARAM RB = 220000000
45 .PARAM CA = 3.5e-12
46 .PARAM CB = 3.5e-12
47 .PARAM ROEZ = 104.999999999999999
48 .PARAM COEZ = 3.5e-12
49 RA A GND {RA}
50 RB B GND {RB}
51 CA A GND {CA}
52 CB B GND {CB}
53 XUA NA A VCC GND LOGIC_INPUT_LVC_2i_NAND_PP_CMOS_74AC11000
54 XUB NB B VCC GND LOGIC_INPUT_LVC_2i_NAND_PP_CMOS_74AC11000
55 XUG NA NB NOUTG VCC GND LOGIC_FUNCTION_2_LVC_2i_NAND_PP_CMOS_74AC11000
56 XOUTPD NOUTG NOUTTPD VCC GND TPD_LVC_2i_NAND_PP_CMOS_74AC11000
57 XUOUT NOUTTPD NOUT_INT VCC GND LOGIC_PP_OUTPUT_LVC_2i_NAND_PP_CMOS_74AC11000
58 XICC VCC GND NVIOUT LOGIC_ICC_LVC_2i_NAND_PP_CMOS_74AC11000
59 SICC VCC GND VCC GND SW1
```

Model file example

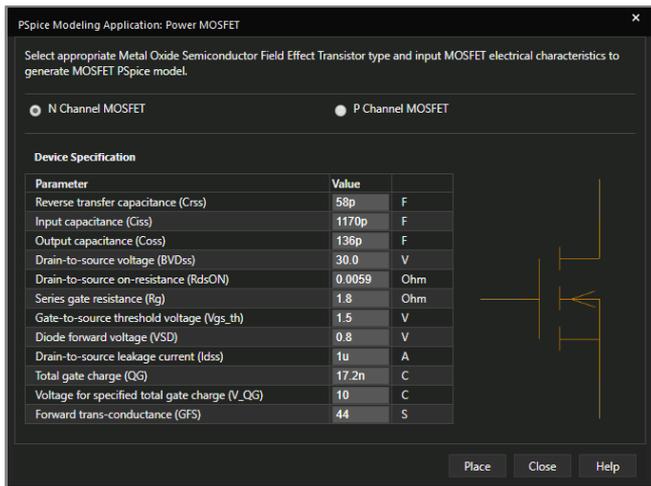
Modeling application

- Used to add customizable, parameterized components to your design:
 - Power MOSFETs
 - Power diodes
 - Passives with parasitics
 - Independent sources
 - Switches
 - Transformers
- Click *Place* → *PSpice Component...* → *Modeling Application* in the top menu bar
- **Note:** these components *do not* trigger the probe limit

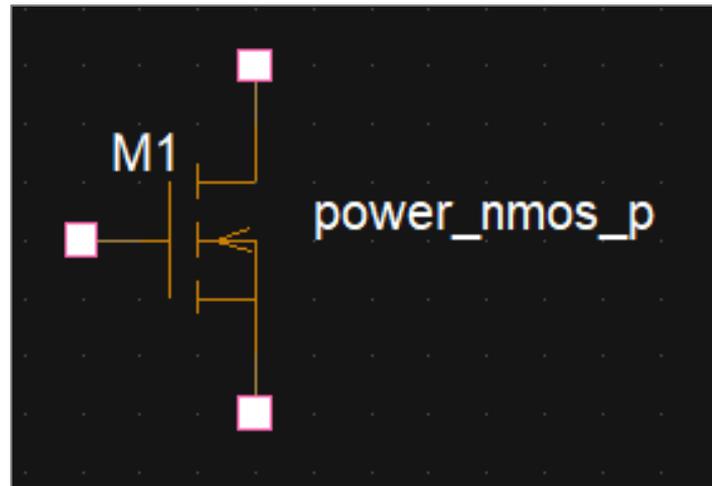


Modeling application, cont.

- A simple UI opens for each type of component with editable fields
- Customize each parameter to your liking, then click *Place* to drop in schematic
 - **Note:** Device parameters can still be edited from their properties once in the schematic



Power MOSFET window



Power NMOS in schematic

Additional included model libraries

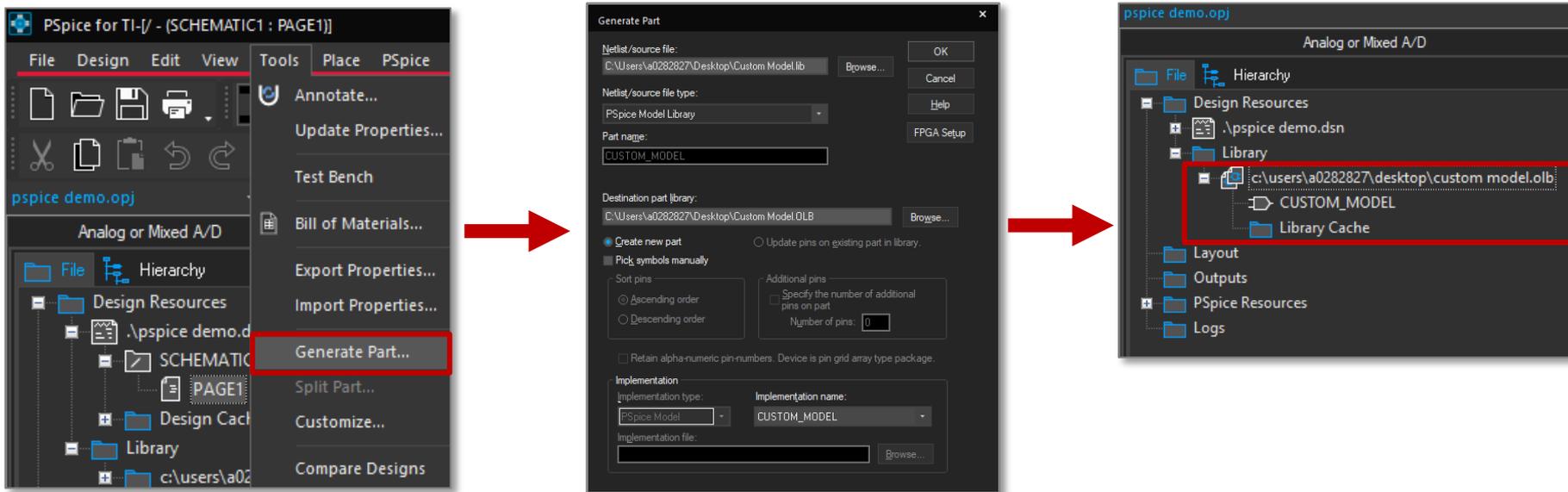
- These standard PSpice libraries are also included:

Library	Description
ABM	Analog behavioral models for various math functions
ANALOG	Passives, dependent sources, switches, transmission lines
BREAKOUT	Customizable versions of many device types
DIG_MISC	Digital timing control, pull-up / pull-down resistors
DIG_PRIM	Logic gates, flip-flops
SOURCE	Independent voltage and current sources
SPECIAL	Parameters, simulation control, library management, utilities

- **Note:** components from these libraries **do not** trigger the probe limit

Importing third-party or custom models

- With your **project** (.opj) selected, click **Tools** → **Generate Part**
- Browse to your model file in the new window, make your selections, and click **OK**
- The new model appears in your project's library



Types of models on TI.com

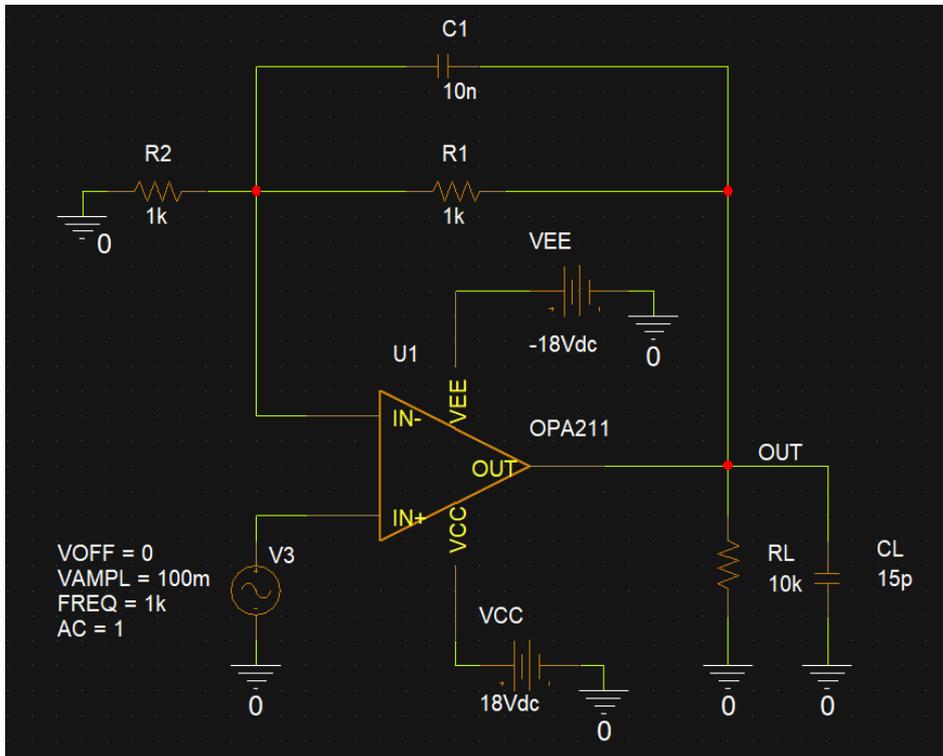
Type	Description
PSpice	Analog / mixed-signal model for use in PSpice-based simulators. Packaged in a way to be easy to import and use in PSpice / Cadence / OrCAD.
TINA-TI	Analog / mixed-signal model for use in PSpice-based simulators. Packaged in a way to be easy to import and use in TINA-TI.
HSPICE (uncommon)	Analog / mixed-signal model for use in HSPICE-based simulators. HSPICE is a branch of SPICE similar to PSpice, but models are not directly compatible.
SIMPLIS	Switch-mode power supply model for use in SIMPLIS .
IBIS	I/O Buffer Information Specification model, typically used for digital pin timing analysis. Compatible with a broad range of industry simulators.

Part 3

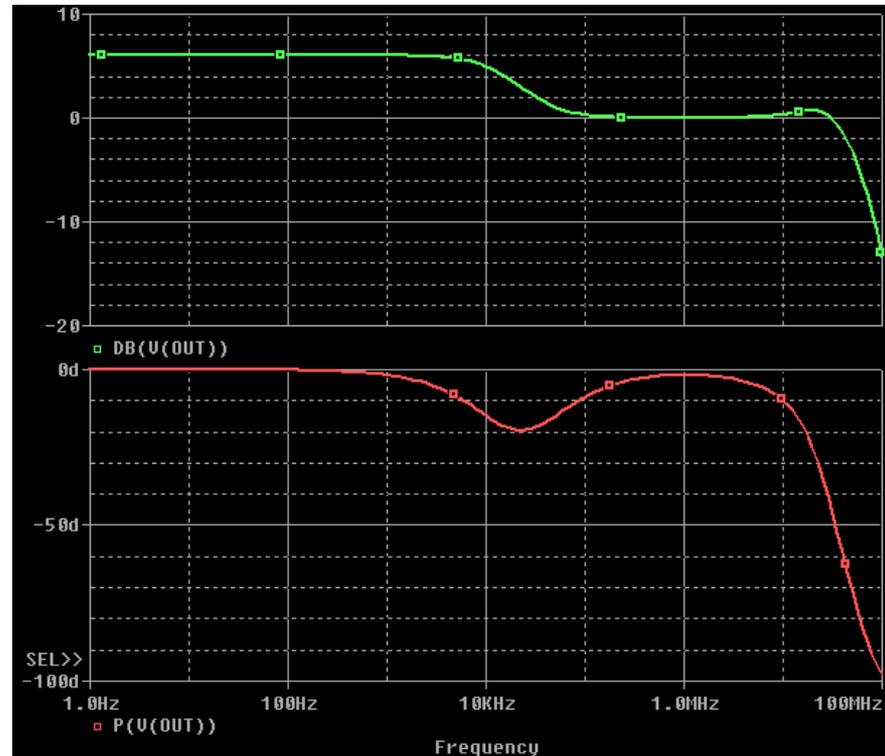
Setup and simulation examples

Tip: Enabling AutoConverge in your sim profile can fix a broad range of convergence issues.

Operational amplifier example – OPA211

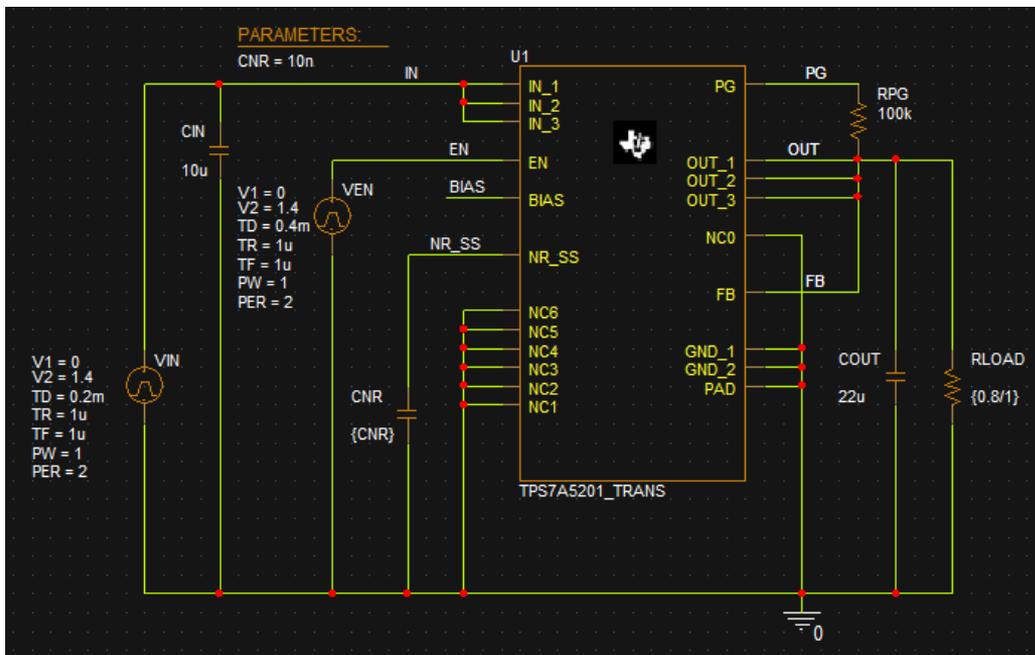


Schematic capture

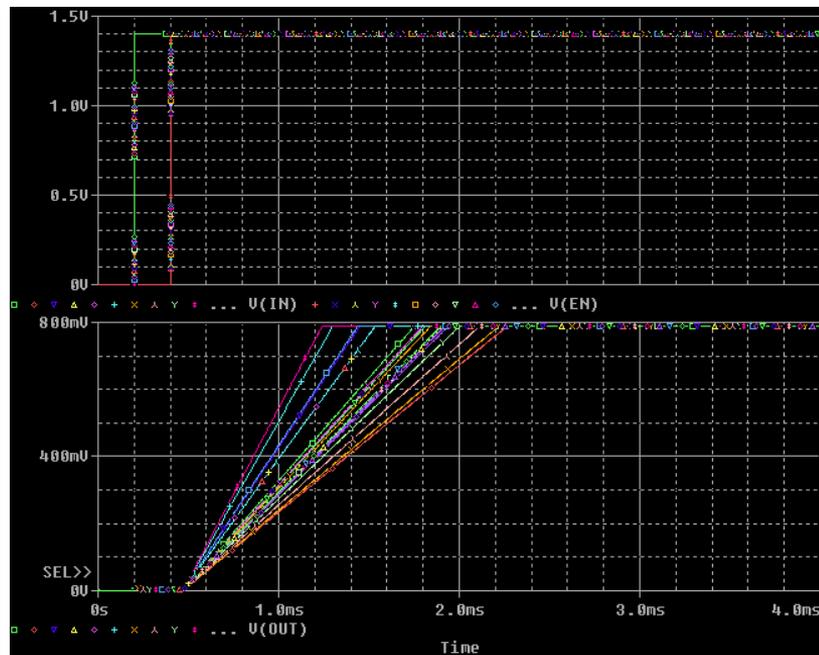


AC simulation result

Power supply example – TPS7A52

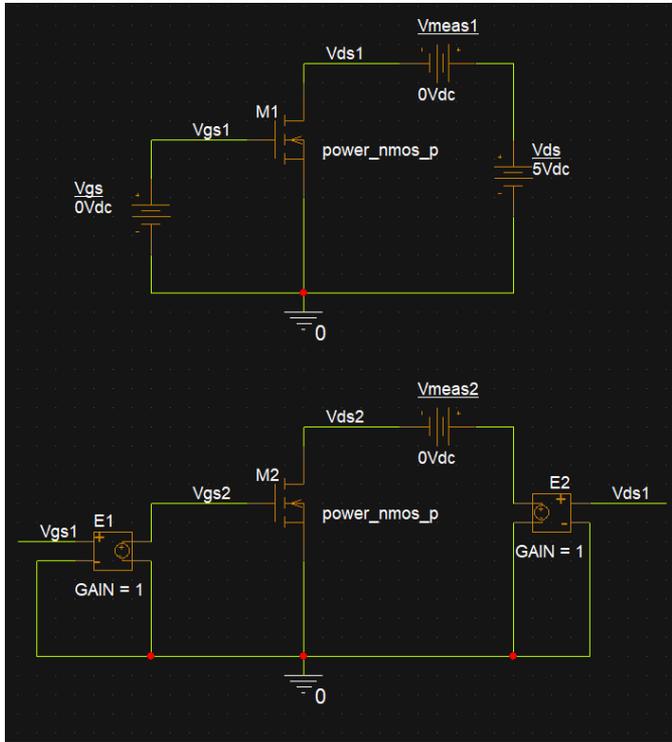


Schematic capture

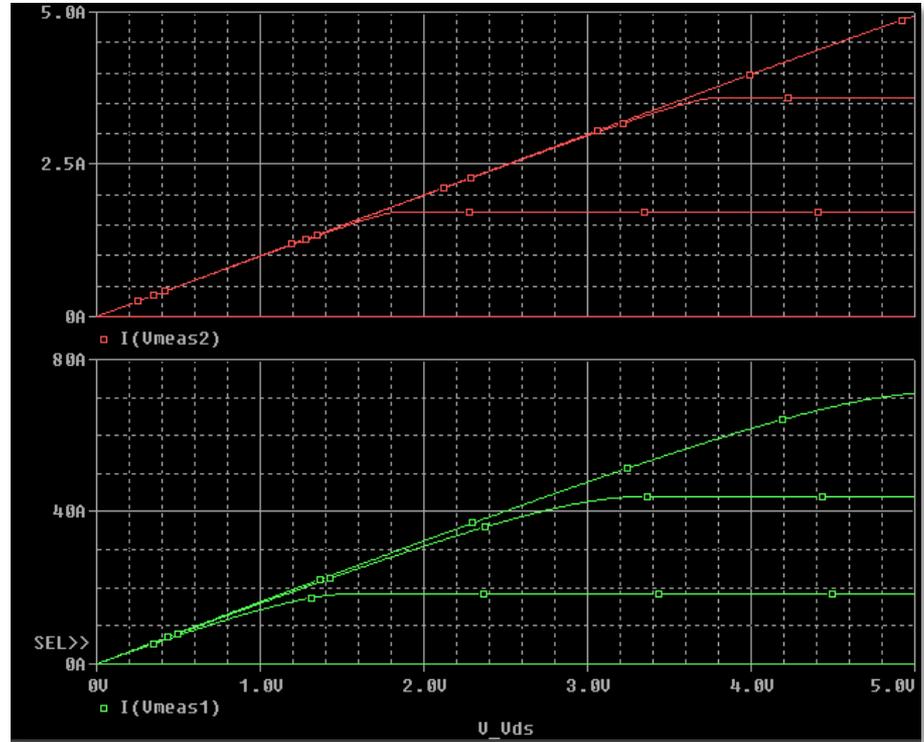


Transient simulation result

Modeling Application – Power MOSFET



Schematic capture



DC simulation result

Part 4

Additional Resources

Tip: Choose the “Last Plot” setting in your sim profile to preserve results display settings between runs.

Additional resources

Hands-on training manual

- Self-guided, step-by-step tutorial that walks the user through the entire tool workflow
 - Includes basic and more advanced content
 - Includes debugging and troubleshooting
- Available from the PSpice for TI start page:
 - Click *Training Course*

The screenshot shows a navigation menu with three main sections: Quick Start, Industry News & Trends, and Support. A red arrow points to the 'Quick Start' section, which includes a 'Quick Start Tutorial Video' and a 'Training Course' link. The 'Training Course' link is highlighted with a red box. Below the 'Quick Start' section, there are three article thumbnails: 'Transfer Function Gain and Relative Stability', 'AC Peak Voltage vs. Peak-to-Peak Voltage vs. RMS Voltage', and 'The Importance of Capacitor Impedance in AC Circuit Analysis...'. The 'Support' section includes links for 'PSpice for TI Capabilities', 'PSpice User Manual', 'Frequently Asked Questions (FAQs)', 'PSpice for TI Tool Forum', and 'TI E2E Support Center', along with a 'Learn More' button and an 'Upgrade' button.

The cover of the 'Mixed-Signal Simulation with TI-PSpice 2020 Simulation Labs' manual features the Cadence logo at the top left. The title is prominently displayed in white text on a red background. Below the title, there is a photograph of a green PCB with a circuit diagram overlaid. To the right of the PCB, a text box explains that the manual uses hands-on exercises for both new and moderately experienced users to maximize performance. Below this, a section titled 'Who will this benefit?' lists various engineering and design applications. The bottom half of the cover is divided into four quadrants, each with a red circular icon and a title: 'Quickly Create Schematic for PSpice Simulation', 'Import 3rd party models in your design', 'Learn Different Types of Analysis Techniques', and 'Quickly Solving Convergence errors'. Each quadrant contains a brief description of the topic. At the bottom, the 'Software Product Version 17.4 - 2019' is noted, and the Cadence logo is in the bottom right corner.

cadence

Mixed-Signal Simulation with TI-PSpice 2020®

Simulation Labs

This lab manual uses hands-on exercises for both new and moderately experienced users of Cadence® PSpice® to maximize the performance of their circuits.

Who will this benefit?
"Engineers and managers dealing with PCB designs, hardware, SPICE models, systems design, FPGA designs, RF circuits, analog/mixed or digital simulations, EM /EMC analyses, or electronics in general."

Quickly Create Schematic for PSpice Simulation
Learn how to quickly design your circuit idea in the OrCAD workspace & get it ready for PSpice simulation to see how it will behave in real world

Import 3rd party models in your design
Take advantage of the ability to bring in any third-party vendor model from internet into PSpice

Learn Different Types of Analysis Techniques
Learn when to use and how to configure and run the following analyses:

- Transient, AC Sweep, DC Sweep Analysis
- Parametric & Performance Analysis
- Temperature
- Noise

Quickly Solving Convergence errors
Learn how to resolve the most common Spice circuit convergence errors with PSpice and use **Auto Convergence** to converge a simulation automatically

Software Product Version 17.4 - 2019

cadence

11 Page

Additional resources, cont.

Overview content

How to simulate complex analog power and signal-chain circuits with PSpice for TI



Watch: [How to simulate complex analog power and signal-chain circuits with PSpice for TI](#)

Hardware engineers are often expected to deliver results within tight project timelines. Circuit and system designers must use all of the tools at their disposal to create accurate, robust designs that work well the first time. These demands, coupled with today's rapidly evolving work from home situation, mean that those that you can use at home or remotely for circuit simulation and verification are more valuable than ever before.

Using circuit design and simulation software tools are hard to install in many hardware engineers' homes. Here at TI, we offer remote design tools to help you design with almost all of our device families, from single-chip based calculator tools to those that provide graphical representations of complex performance. **PSpice for TI** is a powerful, free online power design tool that lets you design and verify your applications and quickly generate a full schematic and BOM of your design. The tool includes a basic PSpice simulation platform that is sufficient for general power design, but also offers **QSPICE** software, a more flexible PSpice based simulation platform, which supports a broader range of components and operates at link user interface sufficient for more analog designs.

Engineers are reducing the prototyping and evaluation phase of designs. In some cases, they are moving right to a final printed circuit board (PCB) - our engineers want to reduce the cost of circuit errors. To do this, we identified a growing need for high performance, full featured analog simulation platform. In together with Cadence Design Systems, TI has launched PSpice for TI, a full featured version of the industry standard OrCAD/PSpice environment, which allows users to simulate and debug designs to support evaluation and verification.

How to simulate complex analog power and signal-chain circuits with PSpice® for TI

[\[Link\]](#)



PSpice for TI overview video

[\[Link\]](#)

Technical training videos

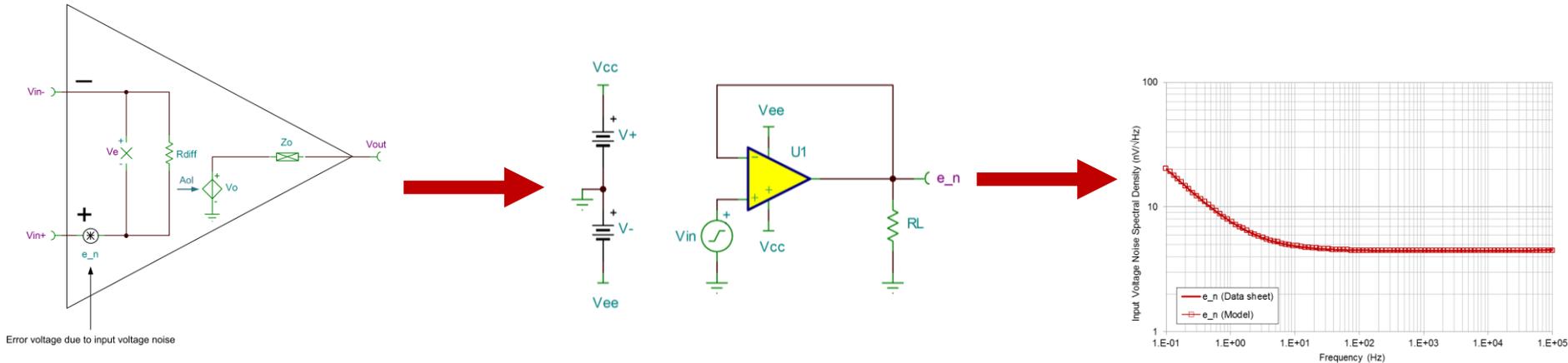


[\[Link to videos\]](#)

[TI.com/pspice-for-ti](https://www.ti.com/pspice-for-ti)

“Trust, but verify” SPICE models

- Series of articles that covers verifying parameters of amplifier models:
 - Part 1: [Output impedance](#)
 - Part 2: [Small-signal bandwidth](#)
 - Part 3: [Input-referred errors](#)
 - Part 4: [Noise](#)



Direct support from TI

- For tool-related support: [Simulation, hardware & system design tools forum](#)
- For specific model or product support: post to that product's forum
 - i.e. For amplifier support, post to the [Amplifiers forum](#)

TEXAS INSTRUMENTS

Search through millions of questions and answers

E2E™ support forums Forums

Technical articles TI training Getting started IMDS More

Amplifiers
Logic
Site support

Audio
Microcontrollers
Switches & multiplexers

Clock & timing
Motor drivers
Tools

Data converters
Power management
Wireless connectivity

DLP® products
Processors
Archived forums

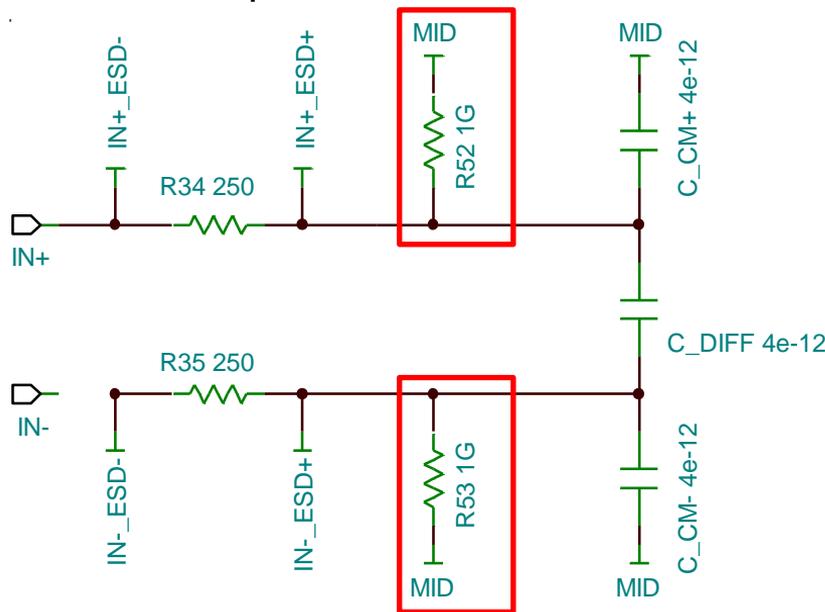
Note: these forums are all supported by TI applications engineers who are graded on responsiveness and quality of support. You should get an initial reply in 24h.

SPICE tips – analysis parameters

Option	Default	Relaxed	Effect
AutoConverge	Off	On	Relaxes multiple parameters if needed to enable convergence
ABSTOL	1e-12	1e-10	Sets the absolute tolerance of nodal currents between DC iterations
RELTOL	1e-3	3e-3	Sets the relative tolerance of the nodal voltages at each DC iteration compared to the first
GMIN	1e-12	1e-10	Adds conductance parallel to every p-n junction
CSHUNT	0	1e-15	Adds capacitance from every node to ground

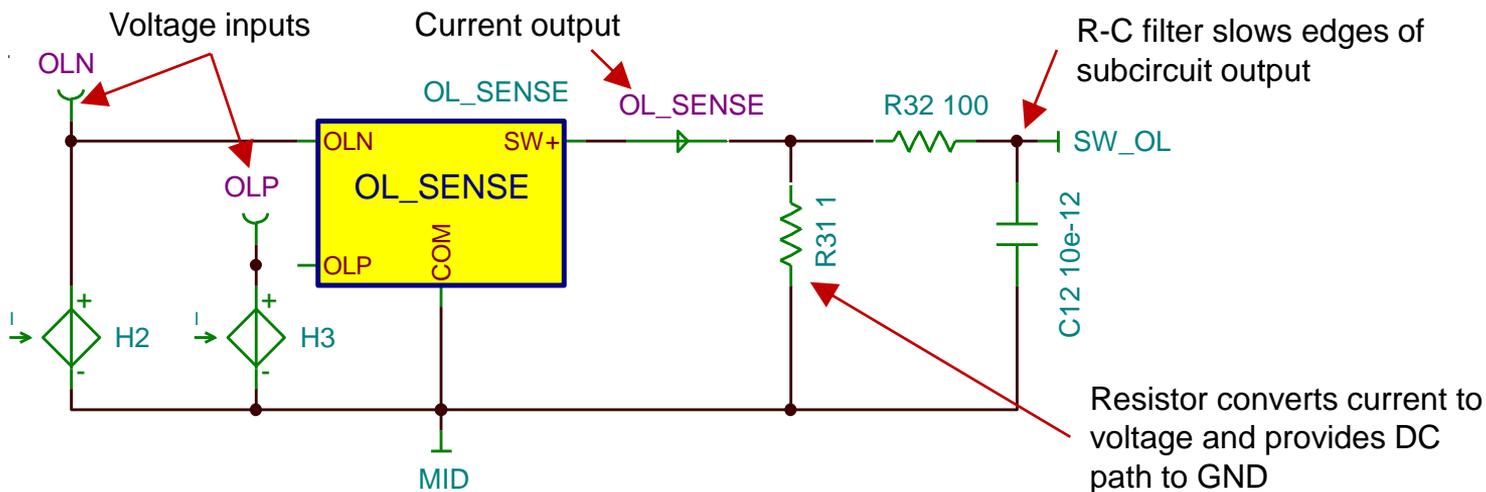
SPICE tips – DC path to GND

- Ensure DC path to ground at every node
 - Can force a path with large resistors (1T, etc.) that don't affect electrical performance
 - Try to use the smallest value possible



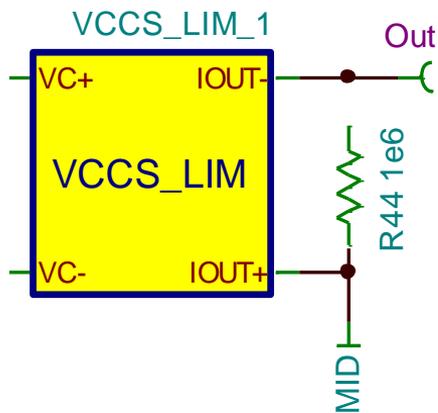
SPICE tips – linear circuits

- Design functional blocks with as linear behavior as possible
 - Sharp transitions or discontinuities cause issues with convergence checks
 - Use R-C filter networks to reduce bandwidth of subcircuits for smooth transitions
 - Use voltage inputs and current outputs wherever possible



SPICE tips – bounded matrix

- Keep matrix equations as tightly bounded as possible
 - Place limits on gain and buffer stages
 - Use only as much gain as required
 - Scale resistances to keep node voltages and currents in similar ranges

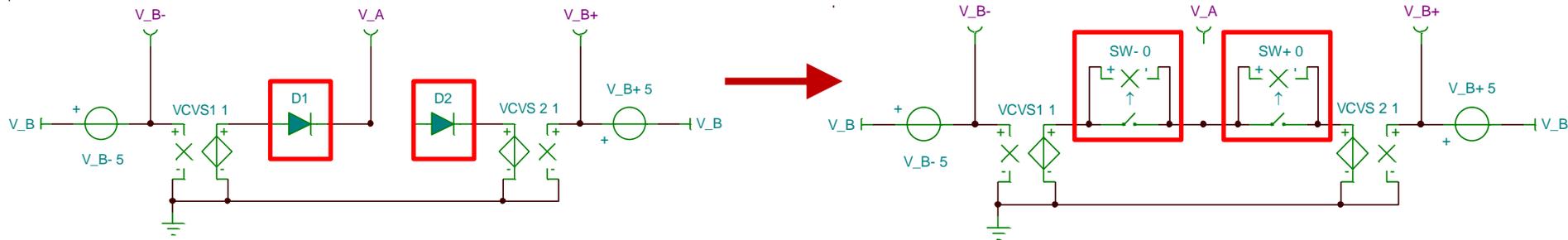


```
<Op Amp Template (OPA172):VCCS_LIM_1 [MACRO]> - Netlist Viewer
File Edit Analysis Help
* Voltage-controlled current source with limits - Aol first stage
.subckt VCCS_LIM_1 VC+ VC- IOU+ IOU-
.param Gain = 1e-4
.param Ipos = .5
.param Ineg = -.5
G1 IOU+ IOU- VALUE={LIMIT(Gain*V(VC+,VC-),Ineg,Ipos)}
.ends
Line: 8 Col: 1
Op Amp Template (OPA172) VCCS_LIM_1
SPICE Exit X: 1775 Y: 504
```

SPICE tips – simplified components

- Replace complex components with simple approximations if exact component modeling isn't necessary
 - **Example:** ideal diode → voltage-controlled switch

Diode equation: $i_D = I_S \left(e^{\frac{qV_D}{kT}} - 1 \right)$



Thank you!



©2020 Texas Instruments Incorporated. All rights reserved.

The material is provided strictly "as-is" for informational purposes only and without any warranty.
Use of this material is subject to TI's **Terms of Use**, viewable at [TI.com](https://www.ti.com)

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to TI's Terms of Sale (www.ti.com/legal/termsofsale.html) or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2020, Texas Instruments Incorporated