



WEBENCH[®]
Design Center

TINA-TI 9: A New Simulation Solution

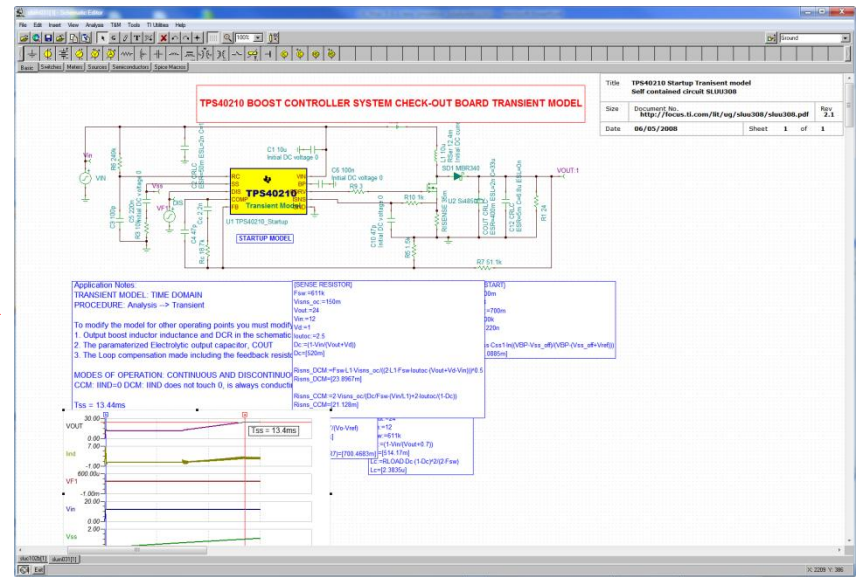
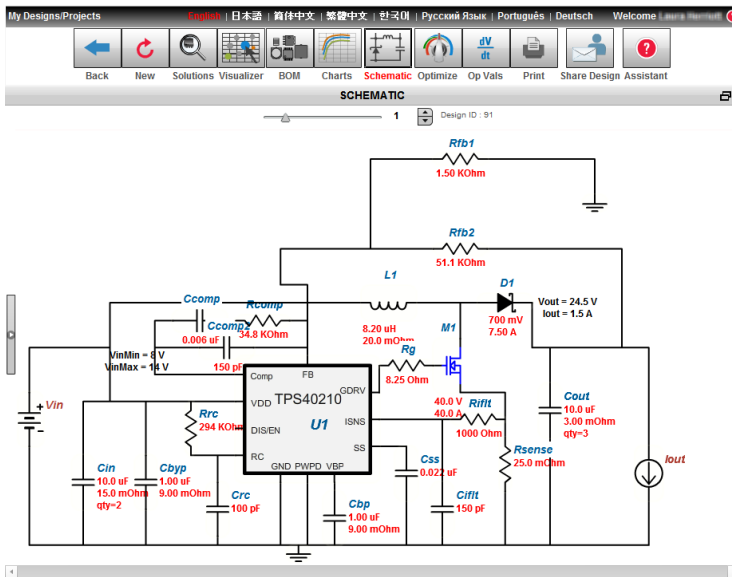


TINA-TI™ and the Pro™ Tools

The best way to begin a design is to start with the requirements and use one of the WEBENCH® tools to create a reference design. For example, for switching mode power supplies, you can use WEBENCH® Power Designer.

WEBENCH® Power Designer

TINA-TI™ Circuit Simulator





TINA-TI 9 Features and Enhancements (1/2)

- **FREE:** Available at ti.com in 5 languages
- **Schematic Symbol Editor:**
 - Develop custom schematic symbols
 - Usable with the Macro Wizard
- **No active components required for analysis**
 - Can simulate a circuit with just passives
- **Macros do not have to be from TI**
 - You can import models from other manufactures!
- **Multi-variable sweeping**
- **Block Wizard added to TINA-TI**



TINA-TI 9 Features and Enhancements (2/2)

- Multi-core processor support.
- Faster single core simulation.
- Available in English, Chinese, Japanese, and Russian.
- Initial Condition and Nodeset Components.
- Linear and nonlinear controlled sources:
 - VCVS, CCVS, VCCS, CCCS
 - Controlled Source Wizard
- WAV files as stimulus (signal source).
- Play calculated waveforms on PC's speakers.
 - Export calculated waveforms as a *.wav file.



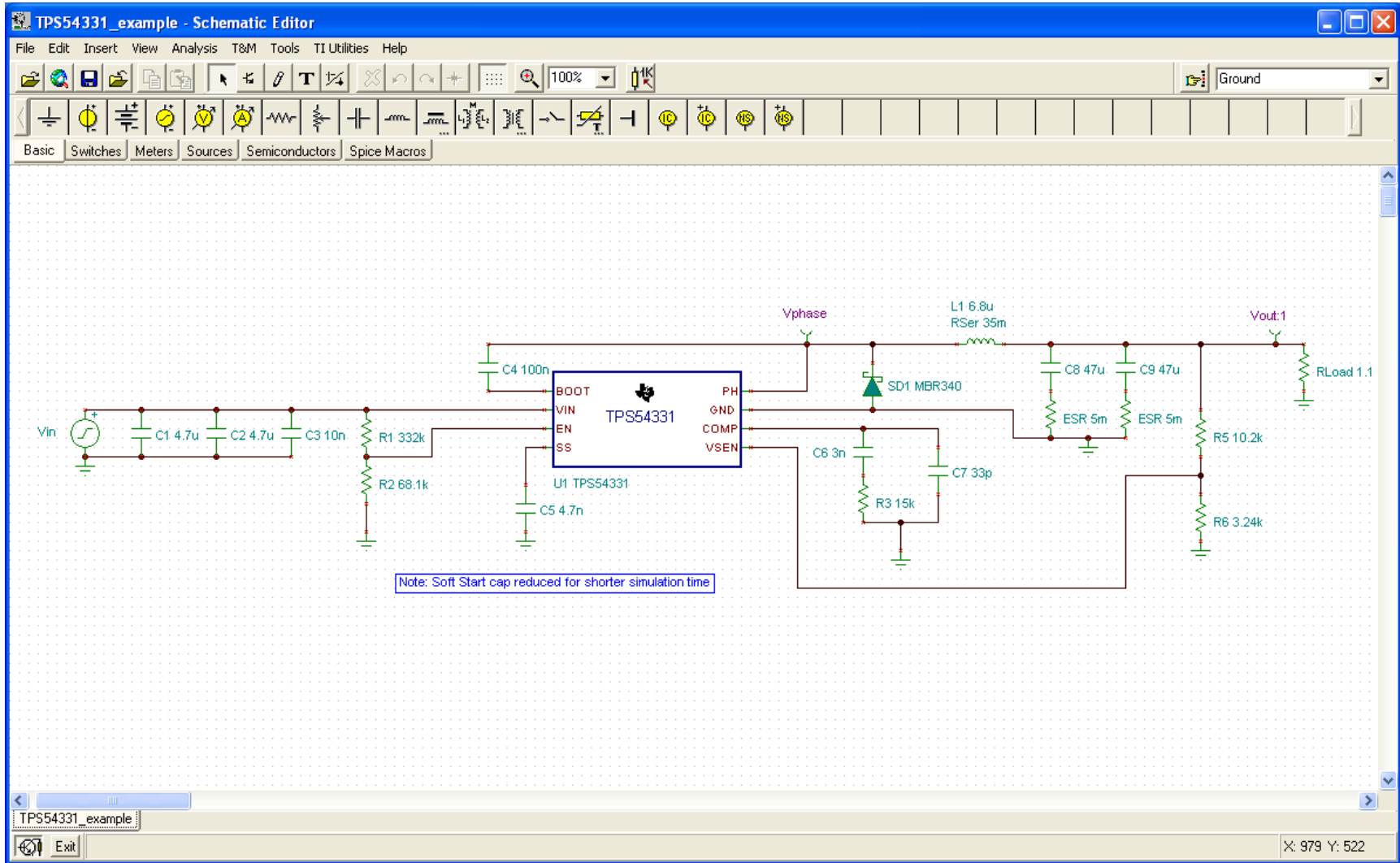
Models in TINA-TI 9

- TINA-TI 9 will ship with over 1000 models and reference designs.
 - Over 400 power models with reference designs
 - Over 600 signal chain models available
- New models and reference designs are added on an ongoing basis (below parts added in past quarter):

CSD17308Q3	LM22680	LM2853	LMH6629	OPA4180	TPA6211A1-Q1	TPS54418	VCA2615
DRV595	LM25011	LM3414	LMH6640	OPA627	TPS22986	TPS54478	VCA2617
INA206	LM25018	LM3481	LMH6643Q-Q1	THS4531A	TPS43060	TPS61175	VCA5807
LM22675	LM25019	LM48560	LMP91200	TL4242-Q1	TPS43061	TPS62141	VCA5807
LM22677	LM25575	LM4871	LMP91200	TLV2464	TPS51125	TPS65300-Q1	XTR300
LM22677	LM26001	LME49600	LMR12010	TLV62090	TPS51225	TPS65320-Q1	
LM22678	LM2830	LME49830	LMZ10500	TLV62130	TPS54231	TPS7A1601	
LM22679	LM2852	LMH6523	OPA1S2385	TLV803M	TPS54350	TPS81256	

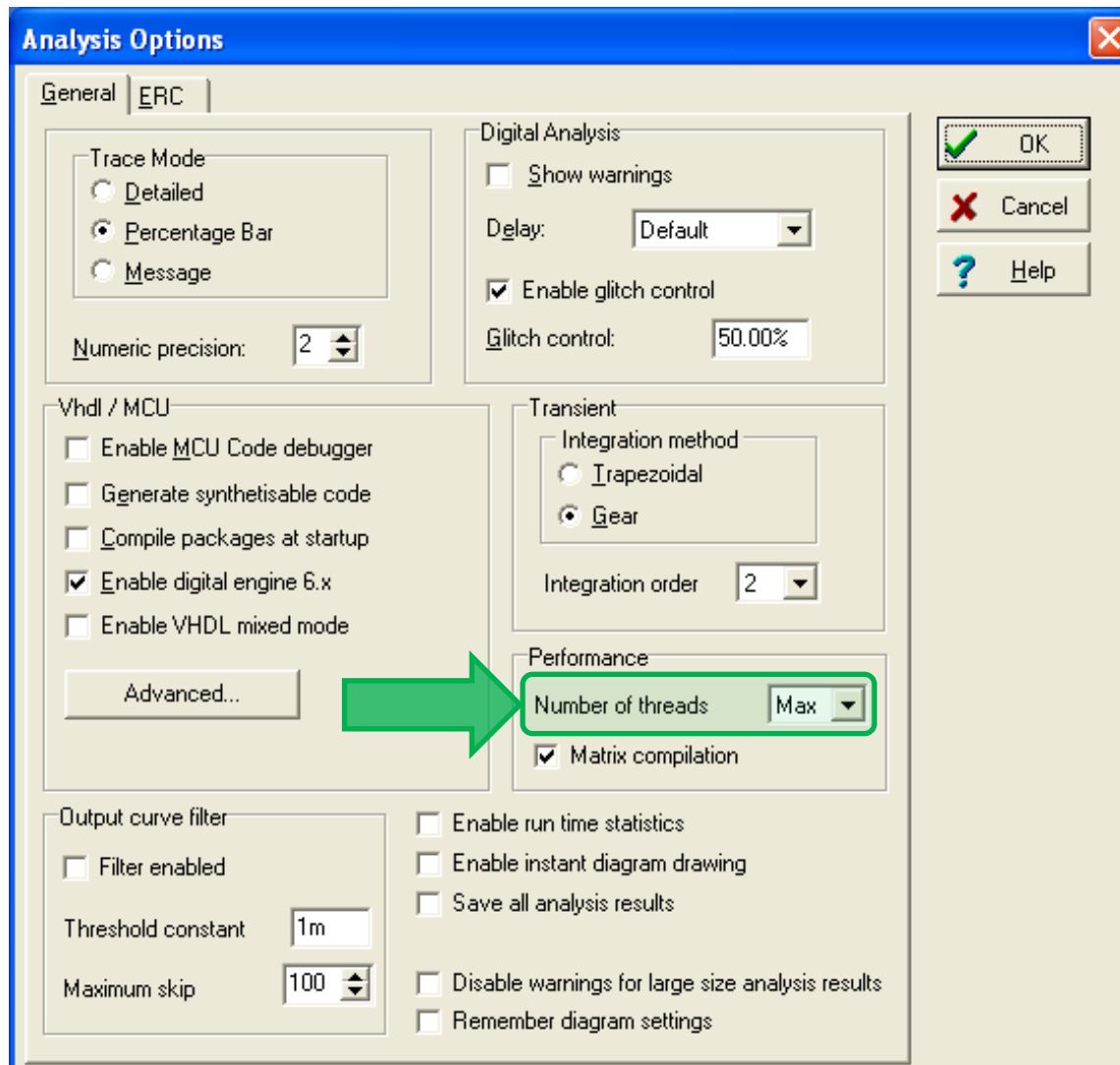


TINA-TI™ 9: Same Look and Feel





TINA-TI™ 9: Multi-Core Support





Simulation Speed: TINA-TI 7 vs TINA-TI 9

- The table shows the transient sim time.
- Hardware platform: 2.8GHz quad core.
- TINA9 is WinXP/Win7 compatible.

Model	TINA7 (min.)	TINA9 (min.)	Speed-up factor
TPS40180	28	14	2
UCCx809	19	5	3.8
UCC28C43	11	3	3.7
TPS62293	60	3	20



Installing TINA-TI 9



Downloading TINA-TI™

TEXAS INSTRUMENTS

Sample & Purchase Cart | English | 简体中文 | 日本語 | my.TI Login

Products Applications Tools & Software Support & Community Sample & Buy About TI Search



LVC MOS Buffers

TI Introduces Universal LVC MOS Buffers

- Industry Leading Phase-Noise Floor in crystal mode
- Ultra-Low Additive Jitter (25fs typ)
- 3.3/2.5/1.8/1.5V Output Supply
- Excellent PSRR performance -44 dB/Hz

Request a **TEXAS INSTRUMENTS**

Select "Tools & Software"

Sample & Purchase Cart | English | 简体中文 | 日本語 | my.TI Login

Products Applications Tools & Software Support & Community Sample & Buy About TI Search

Products Applications **Tools & Software** Support & Community Sample & Buy

Browse products

- Amplifiers & Linear
- Audio
- Broadband RF/IF & Digital Radio
- Clocks & Timers
- Data Converters
- DLP® & MEMS
- High-Reliability
- Interface
- Logic
- DLP® - TV, Projectors, & Cinema
- Power Management
- Processors
 - ARM® Processors
 - Digital Signal Processors (DSP)
 - Microcontrollers (MCU)
 - OMAP™ Mobile Processors
- Switches & Multiplexers
- Temperature Sensors & Control ICs
- Wireless Connectivity
- Calculators & Education Technology

View new products

Parametric Selection

Find the right part fast

WEBENCH® Designer

Enter your power supply requirements:

Vin	Min	14.0	V	Max	22.0	V
Output	Vout	3.3	V	Iout	2.0	A
Ambient Temp					30	°C

Multiple Loads **Power Architect**

Make Your MSP430™ LaunchPad ROCK

Audio Capacitive Touch BoosterPack

- C553x DSP controlled by MCU
- MP3 record & playback
- Achieves lowest power consumption

Buy Now

Products Applications **Tools & Software** Support & Community Sample & Buy About TI Search

Analog

SwitcherPro™ Software Tool

Find the right part fast

TINA-TI™ - Circuit Simulation

WEBENCH® Designer

My Designs

Power FPGA/µP Sensors LED

Enter your power supply requirements:

Vin	Min	14.0	V	Max	22.0	V
Output	Vout	3.3	V	Iout	2.0	A
Ambient Temp					30	°C

Multiple Loads **Power Architect**

Single Output **Start Design**

View all Analog tools & software

ARM Microprocessors & Digital Signal Processors (DSP)

Overview of Tools & Software by processor type

[IDEs including Code Composer Studio™ \(CCStudio™ v5\)](#)

- What's New:**
- Sitara ARM® Cortex™-A8, ARM9®
 - C6000™ Multicore DSP
 - C5000™ Ultra Low Power DSP

Find software and tools for your device

Select a device

Microcontrollers (MCU)

MSP430™ Ultra-Low Power 16-Bit Microcontroller
Tools | Software

Stellaris® ARM® Cortex™-M-based Microcontrollers
Tools | Software

32-Bit Real-time C2000™ Microcontrollers
Tools | Software

Hercules™ ARM® Safety MCUs
Tools & Software

Select Tina-TI Circuit Simulation

Texas Instruments (TI) is a global analog and digital semiconductor IC design and manufacturing company. In addition to analog technology and microcontroller (MCU) semiconductors, TI designs and manufactures semiconductor solutions for analog and digital embedded and application processing.

TI Worldwide | Contact Us | Website Feedback | my.TI Login | Site Map | m.ti.com (Mobile Version) **BSS**

© Copyright 1995-2012 Texas Instruments Incorporated. All rights reserved. Trademarks | Privacy Policy | Terms of Use

Texas Instruments (TI) is a global analog and digital semiconductor IC design and manufacturing company. In addition to analog technologies, digital signal processing (DSP) and microcontroller (MCU) semiconductors, TI designs and manufactures semiconductor solutions for analog and digital embedded and application processing.

TI Worldwide | Contact Us | Website Feedback | my.TI Login | Site Map | m.ti.com (Mobile Version) **BSS**

© Copyright 1995-2012 Texas Instruments Incorporated. All rights reserved. Trademarks | Privacy Policy | Terms of Use

Texas Instruments is a member of **ecia**



Downloading TINA-TI™

www.ti.com/webench



[Samples & Purchase Cart](#) | [Contact Us](#) | [TI Worldwide: United States](#) | [my.TI Login](#)

[Products](#)
[Applications](#)
[Design Support](#)
[Sample & Buy](#)
[All Searches](#)

[TI Home](#) > WEBENCH® Design Center

Links for TINA-TI v9 download

WEBENCH® Design Center

WEBENCH® Designer Tools & Eco-System

WEBENCH Designer tools are powerful software algorithms and visual interfaces that deliver complete power, lighting, and sensing applications in seconds. This enables the user to make value based comparisons at a system and supply chain level before a design is committed. This expert analysis is not possible anywhere else.

WEBENCH® Architect Tools

- > Power Architect (multi supply)
- > Processor Power Architect
- > FPGA Power Architect
- > LED Architect (enter lumens)
- > All WEBENCH Tools

WEBENCH® Designer Tools

- > Power (single supply)
- > LED (enter LED)
- > Sensor AFE & Sensor Interface
- > Active Filters | Amplifiers
- > EasyPLL
- > All WEBENCH Tools

Other Software

- > SwitcherPro™ Software Tool
- > ADCPro™ Evaluation Software
- > ClockPro™ Programming Software
- > FilterPro™ v3.1 Design Software
- > TI Gadgets and Widgets
- > Calculators and Other Utilities
- > Thermal Analysis

TINA-TI™ - Downloadable Circuit Simulation

- > Spice Simulation Tool
- > Free form Schematic Capture

Download Models

- > SpiceRack - A Complete list of SwitcherPro™, PSpice, Tina-TI™ Reference Designs and Spice Models
- > Complete SPICE Model Libraries
- > IBIS and BSDL Model Libraries

Supply Chain Partners:

- > LEDs
- > FPGA & uP
- > Passives
- > Sensors
- > Test Systems
- > Software
- > Distributors
- > All Partners

Related Resources

- > PowerLab™ Reference Design Library
- > Hardware Design Tools and Software
- > Application Notes/Technical Documents
- > Packaging Information
- > eLab™ Design Tools Brochure
- > Online Training
- > Analog eLab™ Videocasts
- > [History of Internet Innovation](#)



WEBENCH® Designer *MyDesigns*

Power **FPGA/μP** Sensors LED

Enter your power supply requirements:

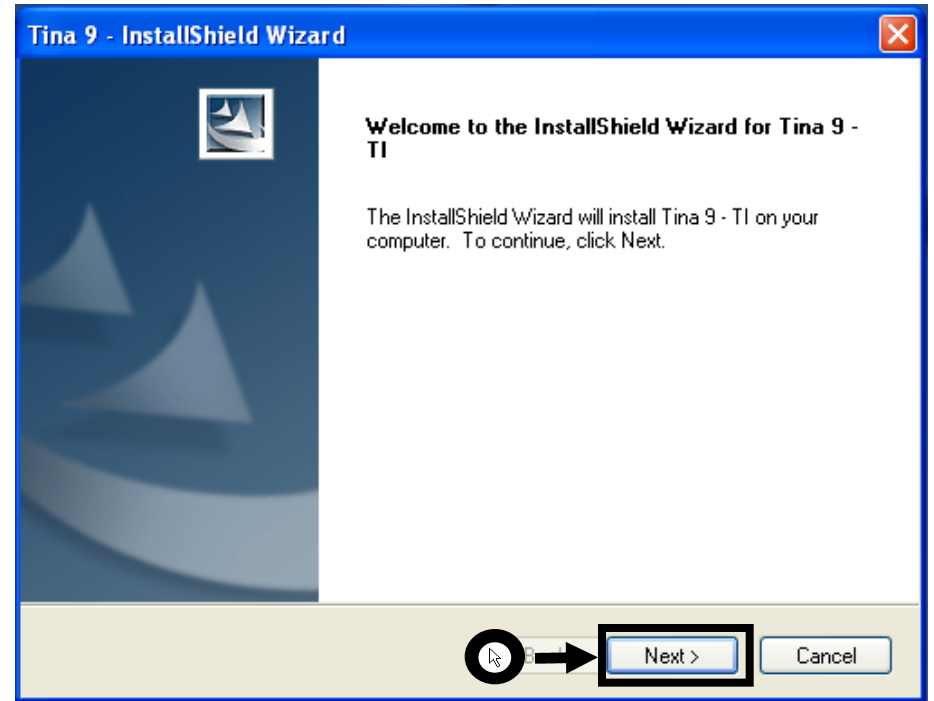
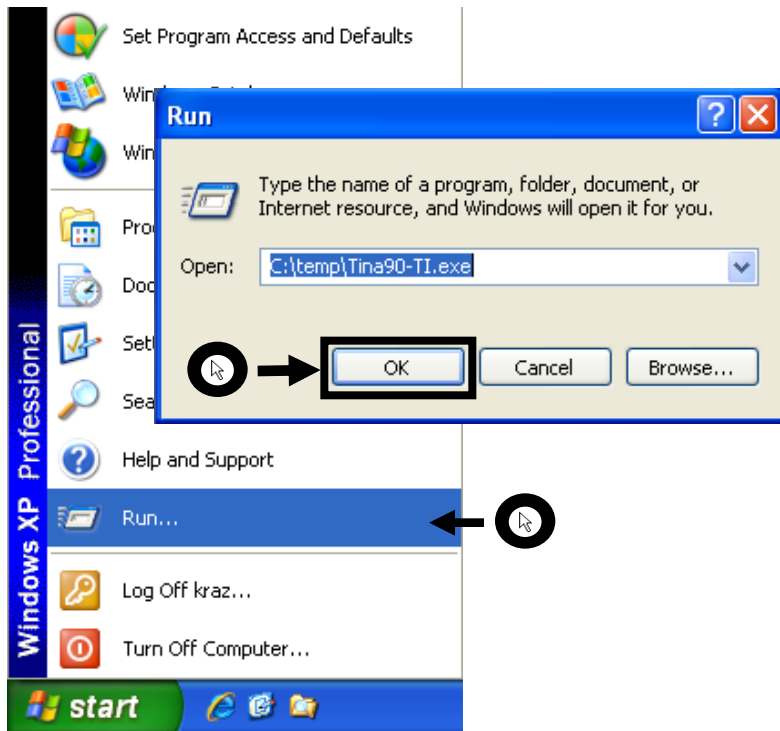
Vin V V
 Vout V A
 Ambient Temp °C

[? WEBENCH Help Page](#)

WEBENCH® Design Center on E2E Community



Installing TINA-TI 9





Installing TINA-TI 9

Tina 9 - InstallShield Wizard

License Agreement

Please read the following license agreement carefully.

Press the PAGE DOWN key to see the rest of the agreement.

TINA-TI
Special Complimentary Basic Edition
Distributed by Texas Instruments
© 2009 by DesignSoft - Texas Instruments

This complimentary, limited, version of TINA (TINA-TI) is distributed by Texas Instruments (TI) in cooperation with DesignSoft, supplier of TINA. It includes TI integrated circuit macromodels and application examples and is intended to demonstrate commercial circuit applications for TI's products.

Do you accept all the terms of the preceding License Agreement? If you select No, the setup will close. To install Tina 9 - TI, you must accept this agreement.

Print

InstallShield

Yes No

Tina 9 - InstallShield Wizard

Customer Information

Please enter your information.

Please enter your name and the name of the company for which you work.

User Name:
Michael Krasnicki

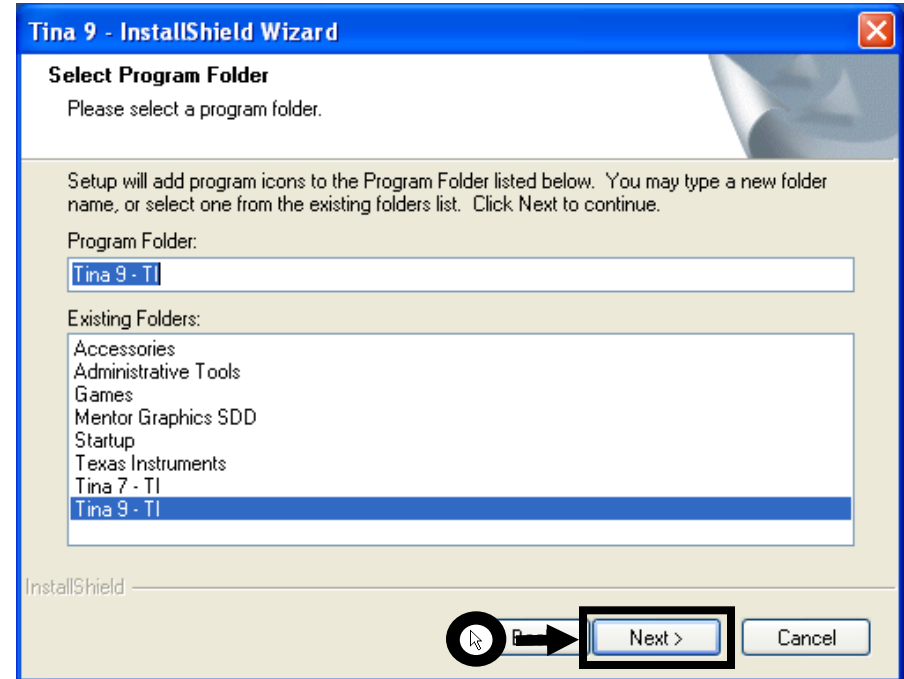
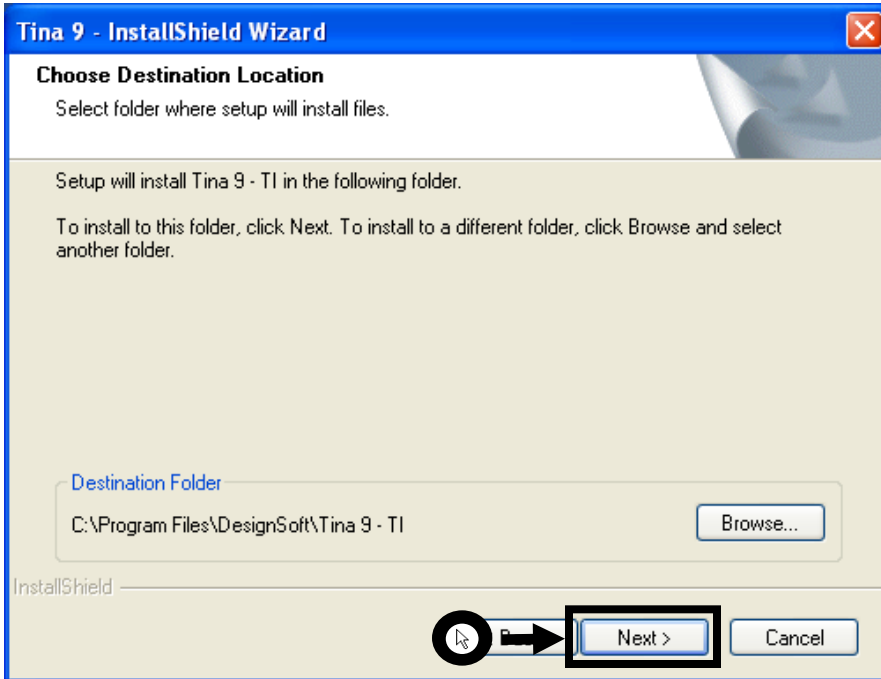
Company Name:
Texas Instruments

InstallShield

Next > Cancel

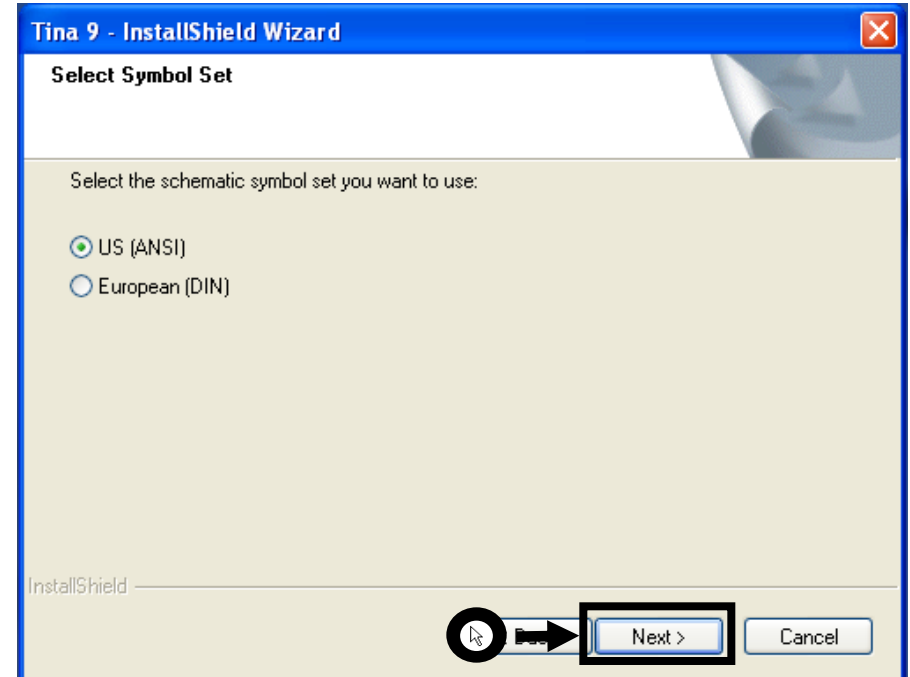
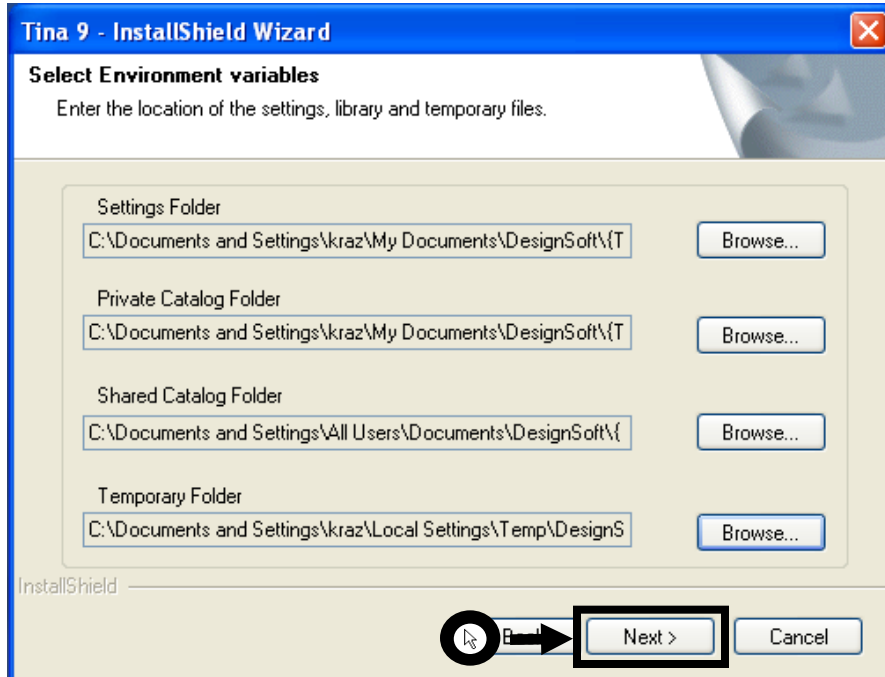


Installing TINA-TI 9



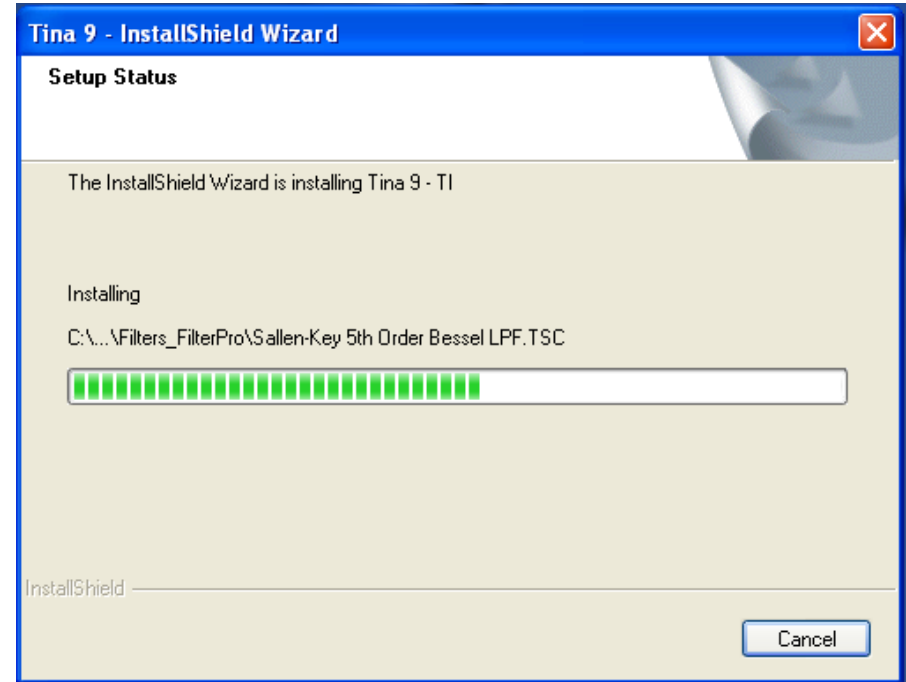
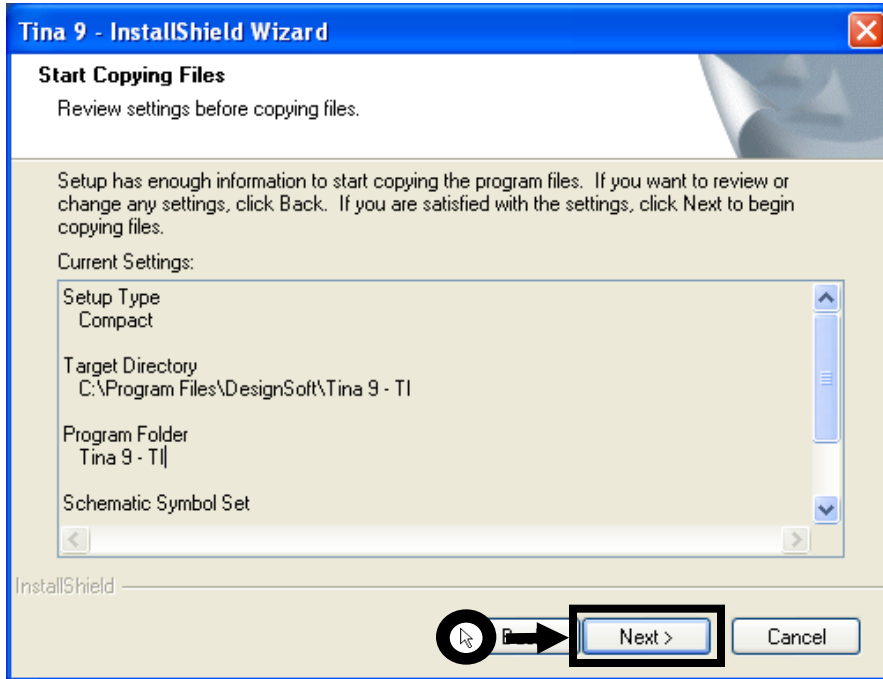


Installing TINA-TI 9



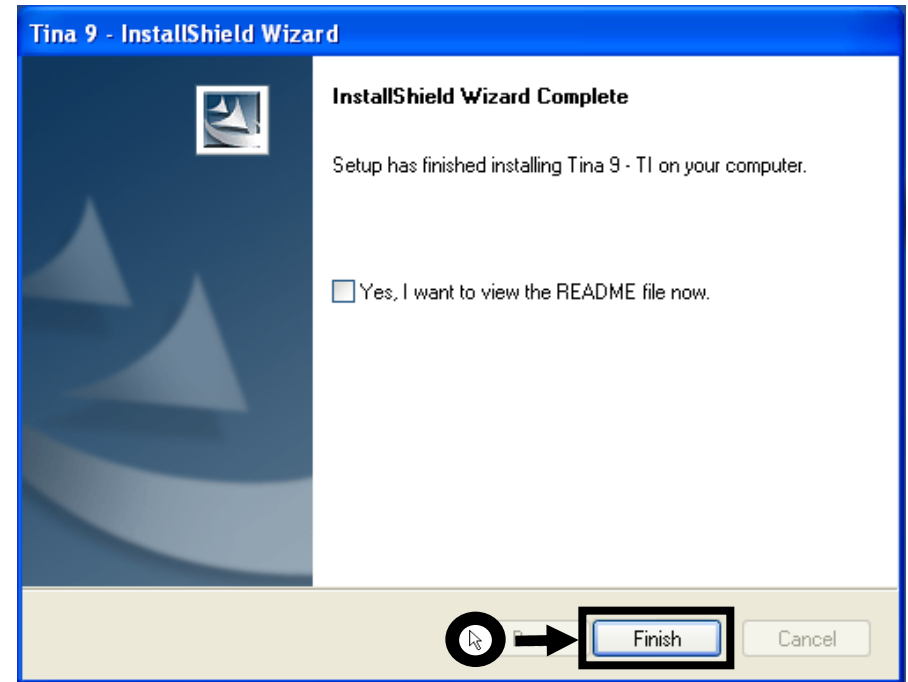
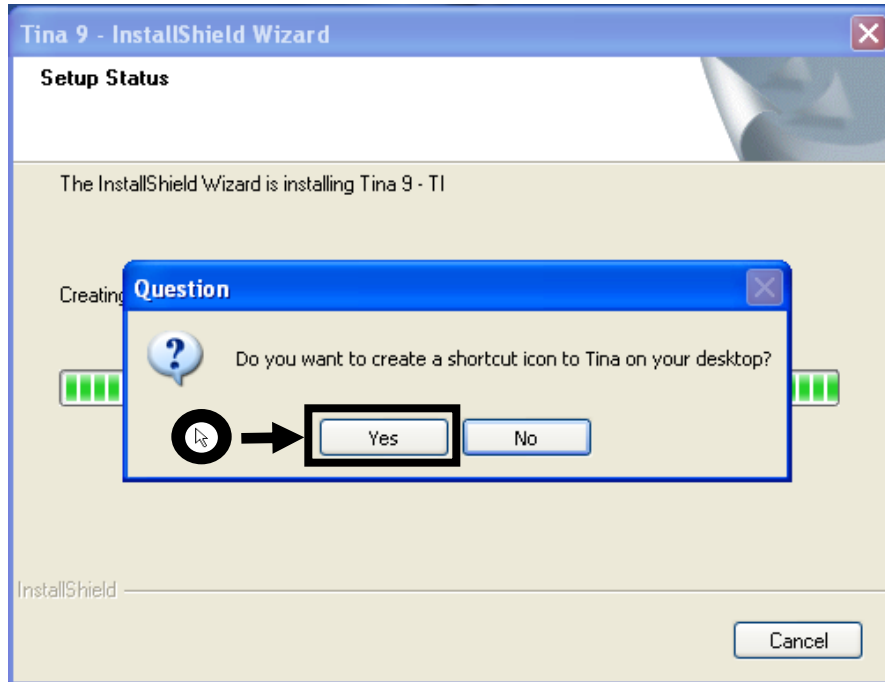


Installing TINA-TI 9





Installing TINA-TI 9





Downloading the Model Library

WEBENCH® Design Center

WEBENCH® Designer Tools & Eco-System

WEBENCH Designer tools are powerful software algorithms and visual interfaces that deliver complete power, lighting, and sensing applications in seconds. This enables the user to make value based comparisons at a system and supply chain level before a design is committed. This expert analysis is not possible anywhere else.

WEBENCH® Architect Tools

- > Power Architect (multi supply)
- > Processor Power Architect
- > FPGA Power Architect
- > LED Architect (enter lumens)
- > All WEBENCH Tools

WEBENCH® Designer Tools

- > Power (single supply)
- > LED (enter LED)
- > Sensor AFE & Sensor Interface
- > Active Filters | Amplifiers
- > EasyPLL
- > All WEBENCH Tools

Other Software

- > SwitcherPro™ Software Tool
- > ADCPro™ Evaluation Software
- > ClockPro™ Programming Software
- > FilterPro™ v3.1 Design Software
- > TI Gadgets and Widgets
- > Calculators and Other Utilities
- > Thermal Analysis

TINA-TI™ - Downloadable Circuit Simulation

- > Spice Simulation Tool
- > Free form Schematic Capture

Download Models

- > SpiceRack - A Complete list of SwitcherPro™, PSpice, Tina-TI™ Reference Designs and Spice Models
- > Complete SPICE Model Libraries
- > IBIS and BSDL Model Libraries

Supply Chain Partners:

- > LEDs
- > FPGA & uP
- > Passives
- > Sensors
- > Test Systems
- > Software
- > Distributors
- > All Partners

Related Resources

- > PowerLab™ Reference Design Library
- > Hardware Design Tools and Software
- > Application Notes/Technical Documents
- > Packaging Information
- > eLab™ Design Tools Brochure
- > Online Training
- > Analog eLab™ Videocasts
- > [History of Internet Innovation](#)



WEBENCH® Designer MyDesigns

Power FPGA/μP Sensors LED

Enter your power supply requirements:

Min Max
Vin 14.0 V 22.0 V

Vout Iout
Output 3.3 V 2.0 A

Ambient Temp 30 °C

Multiple Loads Single Output
Power Architect Start Design

? WEBENCH Help Page

WEBENCH® Design Center on E2E Community



Downloading the Model Library

WEBENCH® Design Center

SPICE Model Resources

[PSpice, TINA-TI Reference Designs and Spice Models](#)

[Complete Table with Links to Files](#)

SPICE Models Downloads

- HSpice Models and Reference Designs
 - HSpice Models Collection ([ti_hspice.zip](#))
 - HSpice Models Index File ([ti_hspice_index.txt](#))
- PSpice Models and Reference Designs
 - PSpice Models Collection ([ti_ospice_models.zip](#))
- TINA-TI (For more info, go to TINA-TI Folder, See FAQs)
 - TINA-TI Spice Models
 - TINA-TI Spice Models Collection ([ti_tina_ti_spice_models.zip](#))
 - TINA-TI Reference Designs
 - TINA-TI Reference Designs Collection([ti_tina_ti_ref_designs.zip](#))
- General SPICE Models (no specific simulator)
 - General SPICE Models Collection ([ti_spice_models.zip](#))
 - General SPICE Models Index File ([ti_spice_models_index.txt](#))



Introduction to TINA-TI



Starting TINA-TI



Noname - Schematic Editor

File Edit Insert View Analysis T&M Tools Utilities Help

- Contents
- Component Help
- Check for Updates...
- Upgrade
- DesignSoft on the Web
- About

About TINA

TINA
The Complete Electronics Lab

Version 9.3.50.40 SF-TI
Build date: Thursday, February 09, 2012, 6:31:40 PM

Copyright 1993-2012 DesignSoft, Inc. All rights reserved.

TINA-TI V9
Special Complementary Basic Edition
Distributed by Texas Instruments
© 2011 by DesignSoft - Texas Instruments

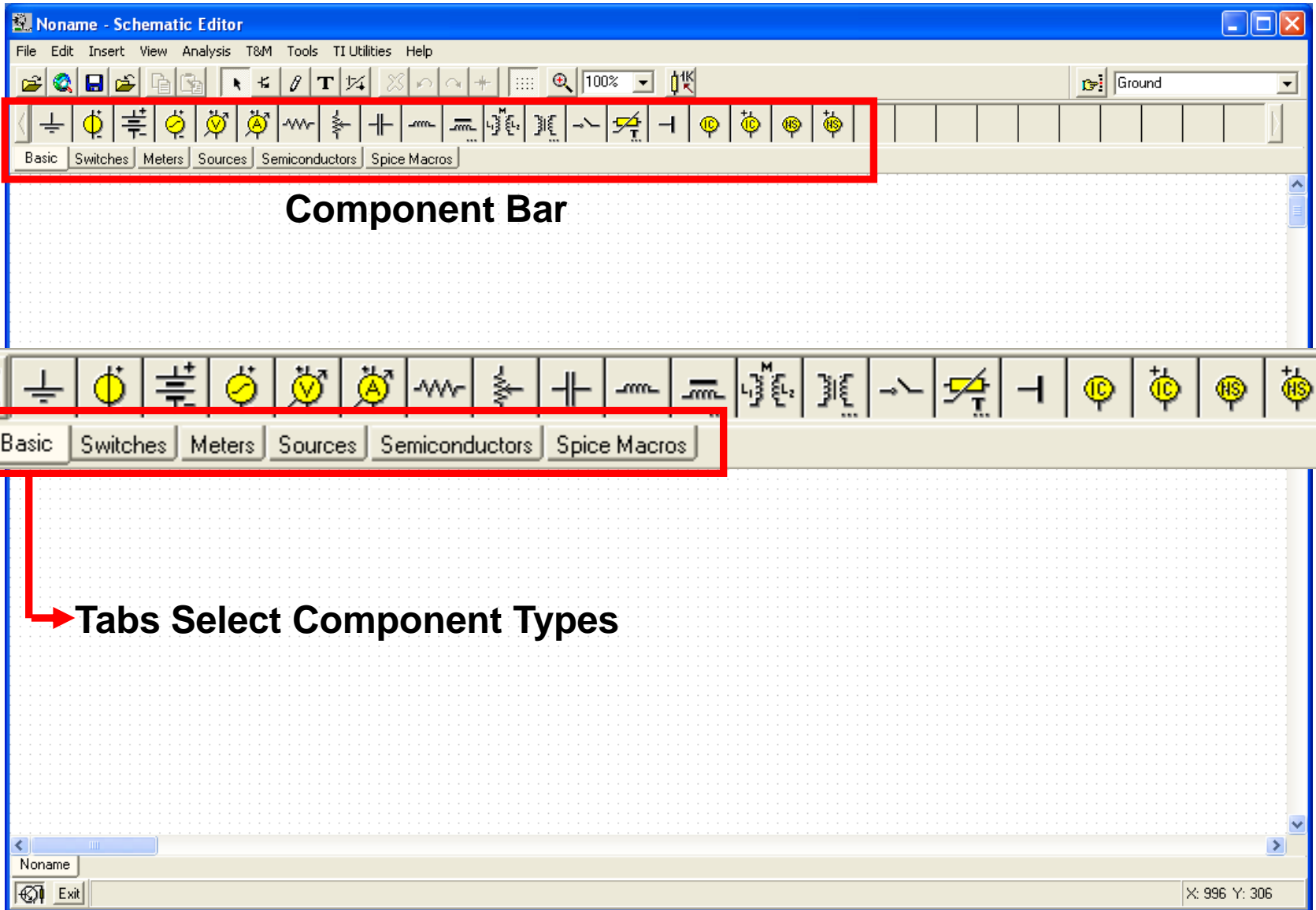
OK

sluc102b[1] slum031[1] Noname

Exit X: 797 Y: 12



TINA-TI Schematic Editor Tour



Component Bar

→ Tabs Select Component Types



TINA-TI Schematic Editor Tour

The screenshot shows the TINA-TI Schematic Editor window titled "Noname - Schematic Editor". The menu bar includes File, Edit, Insert, View, Analysis, T&M, Tools, TI Utilities, and Help. The main toolbar is divided into sections: File Functions, Edit/Select Tool, Text Tool, and Rotate/Mirror tools. A red box highlights the top toolbar, and another red box highlights a sub-section of the Edit/Select Tool. Red arrows point from text labels to specific icons in the toolbars.

Copy/Paste (points to the Copy and Paste icons in the top toolbar)

File Functions (points to the File Functions icons in the top toolbar)

Edit/Select Tool (points to the Edit/Select icon in the sub-toolbar)

Text Tool (points to the Text icon in the sub-toolbar)

Delete Object (points to the Delete icon in the sub-toolbar)

Rotate/Mirror tools (points to the Rotate and Mirror icons in the sub-toolbar)

Last Component Tool (points to the Last Component icon in the sub-toolbar)

Wire Tool (points to the Wire icon in the sub-toolbar)

Hide/Reconnect Tool (points to the Hide/Reconnect icon in the sub-toolbar)

At the bottom of the window, the status bar shows "Noname" and "Exit", with coordinates "X: 996 Y: 306".



TINA-TI Schematic Editor Tour

The screenshot displays the TINA-TI Schematic Editor window. The title bar reads "Noname - Schematic Editor". The menu bar includes "File", "Edit", "Insert", "View", "Analysis", "T&M", "Tools", "TI Utilities", and "Help". The "Analysis" menu is open, showing the following options: "ERC...", "Mode...", "Select Control Object", "Set Analysis Parameters...", "DC Analysis", "AC Analysis", "Transient...", "Steady State Solver...", "Fourier Analysis", "Noise Analysis...", and "Options...".

Red arrows point from the "Analysis" menu item in the menu bar to the "Analysis" menu. Further red arrows point from the "DC Analysis", "AC Analysis", and "Fourier Analysis" items to their respective sub-menus:

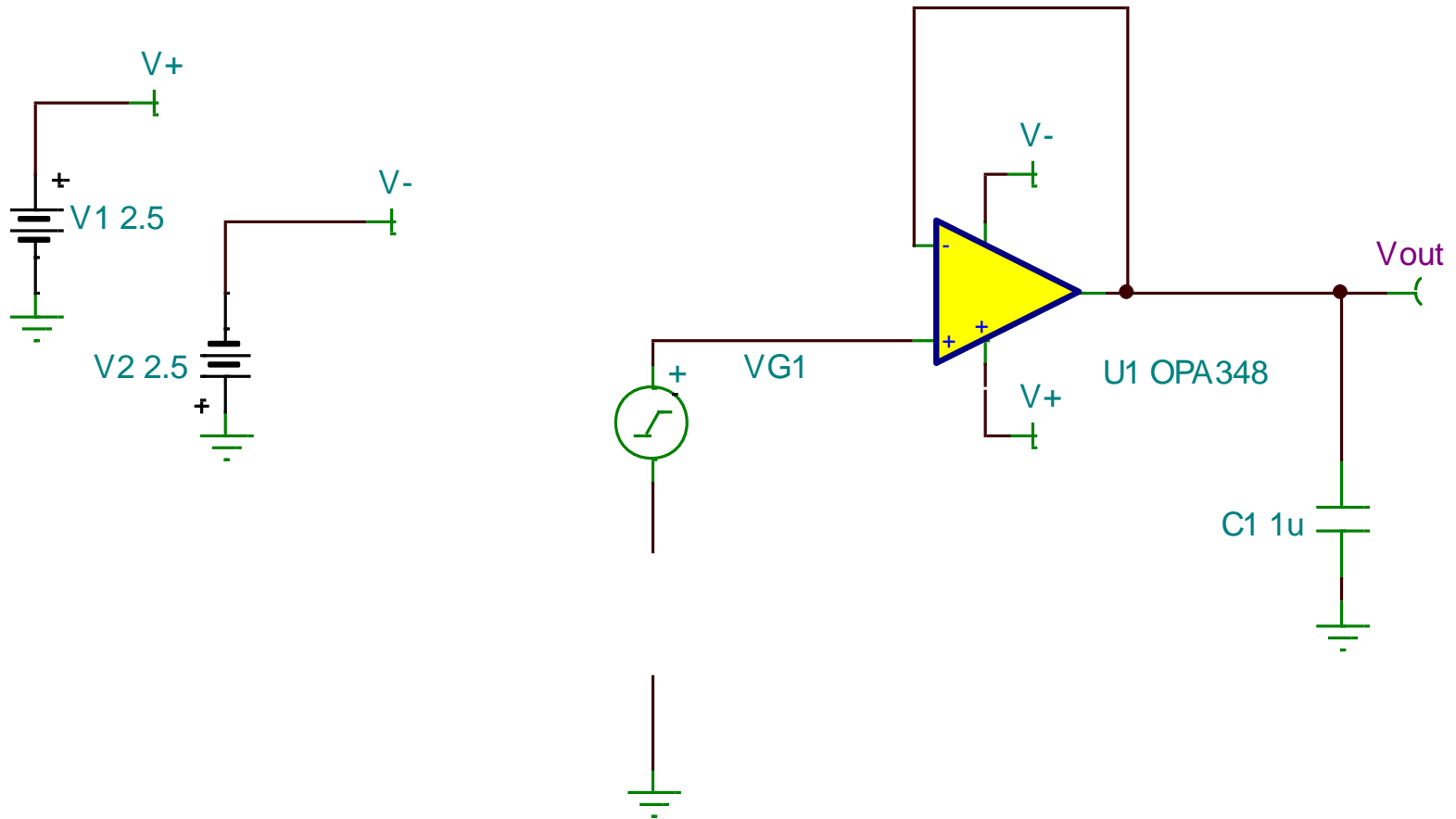
- DC Analysis sub-menu:** Calculate nodal voltages, Table of DC results, DC Transfer Characteristic..., Temperature Analysis...
- AC Analysis sub-menu:** Calculate nodal voltages, Table of AC results, AC Transfer Characteristic...
- Fourier Analysis sub-menu:** Fourier Series...

The main workspace is a grid with a dotted pattern. The status bar at the bottom shows "Noname" and "Exit" buttons, along with coordinates "X: 996 Y: 306".



OPA348 with a Capacitive Load

Let's simulate the OPA348 with a 1 uF load





Select an Operational Amplifier

2. Select op amp symbol – list opens

1. Select *Spice Macros*

3. Scroll list and select op amp - click OK

4. Click on symbol and position in workspace

Operational Amplifiers

- OPA344
- OPA345
- OPA347
- OPA348
- OPA349
- OPA350
- OPA353
- OPA354
- OPA355
- OPA357
- OPA358

Shape: (Auto) Auto-select

Manufacturer: All 50/305

Show all components

OK Cancel Help

U1 OPA348



Add Capacitor

1. Select the capacitor symbol to add capacitors.

2. Double click the capacitor to view the parameters

3. Type over the 1uF value and enter the new capacitor value

Parameter	Value
Label	C1
Parameters	(Parameters)
Capacitance [F]	1u
RPar [Ohm]	Infinite
Initial DC voltage [V]	0
Temperature	Relative
Temperature [C]	0
Linear temp. coef. [1/C]	0
Quadratic temp. coef. [1/C ²]	0
Maximum voltage [V]	100
Maximum ripple current [A]	1
Fault	None



Manipulate Components

The screenshot shows the 'Noname - Schematic Editor' window. The toolbar at the top contains various icons for component placement and manipulation. A red box highlights three icons: a curved arrow for rotate left, a curved arrow for rotate right, and a plus sign for mirror. A red arrow points from the text 'Use rotate and mirror functions, or right click to arrange components' to these icons. Below the toolbar, a yellow operational amplifier component labeled 'U1 OPA348' and a capacitor component labeled 'C1 990n' are placed on a grid. A context menu is open over the capacitor, listing options such as 'Cancel Mode', 'Last Component', 'Auto Repeat', 'Wire', 'Auto Wire', 'Delete', 'Rotate Left', 'Rotate Right', 'Mirror', 'Properties...', and 'Enter Macro'. A red arrow points from the text 'Or double right click to access menu' to the 'Rotate Left' option in the menu.

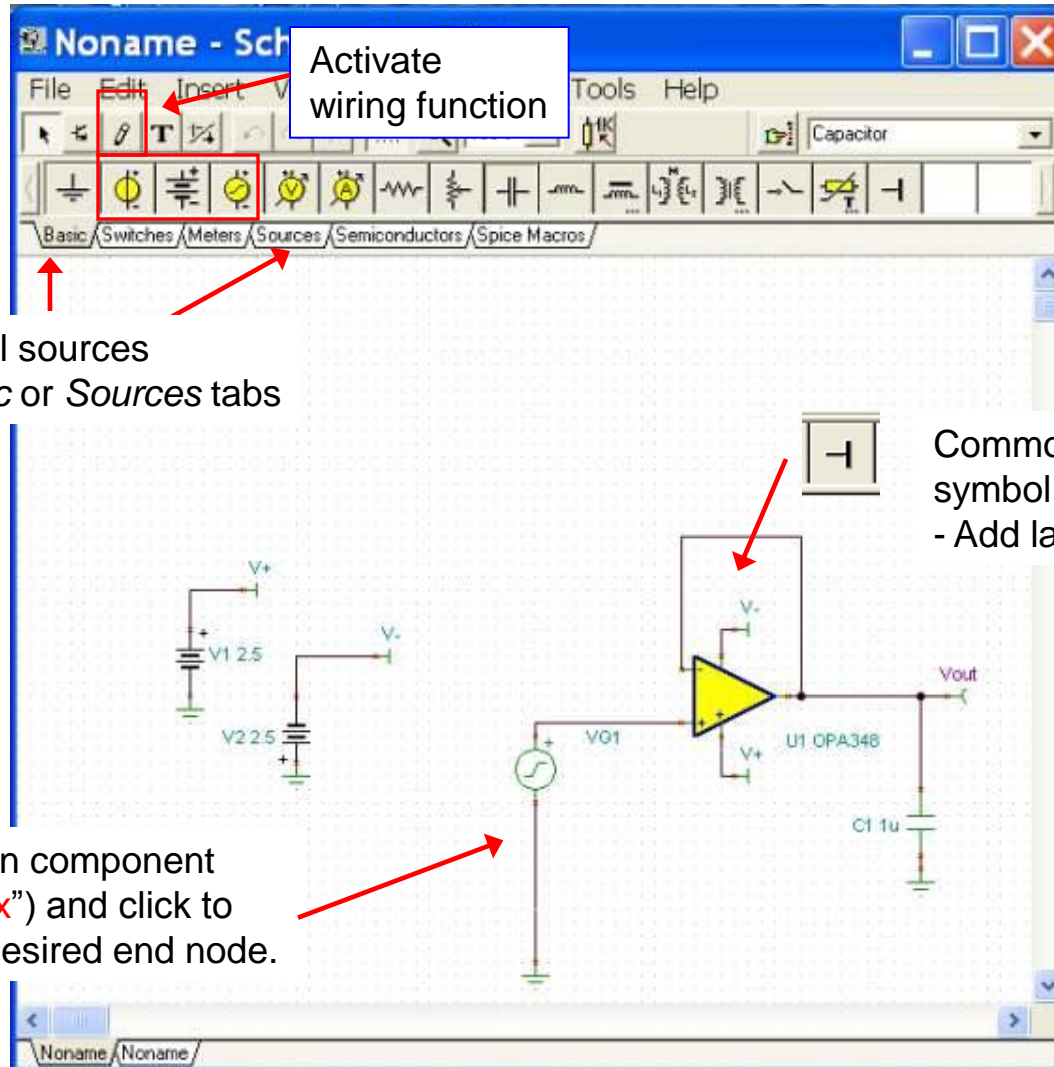
Complete component placement and ready the circuit for wiring

Use rotate and mirror functions, or right click to arrange components

Or double right click to access menu



Add Sources, Generators, and Wiring



Supplies and signal sources selected from *Basic* or *Sources* tabs

Common connection symbol in *Basic* tab - Add label

Wire - place pointer on component End node (small red "x") and click to start wire. Unclick at desired end node.



Function Generators

The image shows a screenshot of the Texas Instruments SPICE schematic editor. The main window is titled "Noname - Schematic Editor" and displays a circuit diagram with a voltage generator (VG1), two DC sources (V1 and V2), an operational amplifier (U1 OPA348), and a capacitor (C1 1u). The "VG1 - Voltage Generator" dialog box is open, showing a parameter table. The "Signal" parameter is set to "Sine wav" and has three adjacent dots to its right. The "Signal Editor" dialog box is also open, showing a waveform plot and a table of parameters: Amplitude [V] (A) is 100m, Frequency [Hz] (f) is 1k, and Rise/fall time [s] (tau) is 1n. The period of the waveform is indicated as $T = 1/f = 1m$.

Label	Value	Units
Label	VG1	
Parameters	(Parameters)	
DC Level [V]	0	
Signal	Sine wav	
Internal resistance [Ohm]	0	
IO state	Input	
Fault	None	

Parameter	Value	Units
Amplitude [V] (A)	100m	
Frequency [Hz] (f)	1k	
Rise/fall time [s] (tau)	1n	

Click on the Signal box, then click on the 3 adjacent dots that appear

The Signal Editor appears

Select signal type

Set the Amplitude, Frequency, Phase, etc.

**Double click on generator VG1
The parameter table appears**

Double click DC Sources to set up parameters



Performing Analysis with TINA-TI

- Running Electrical Rules Check (ECR)
- Running DC Analysis
- Running AC Analysis
- Running Transient Analysis
- Design a Stable OPA348 Buffer Circuit
- Power Modeling in TINA-TI



Running Electrical Rules Check (ERC)

opa348_w_1u - Schematic Editor

File Edit Insert View Analysis T&M Tools Help

ERC...

- Mode...
- Select Control Object
- Set Analysis Parameters...
- DC Analysis
- AC Analysis
- Transient...
- Steady State Solver...
- Fourier Analysis
- Noise Analysis...
- Options...

Select: *Analysis > ERC*

This wire intentionally incomplete

Note: performing any Analysis will automatically start an ERC

ERC Report

Electric Rules Check

Generating graph...
Graph generation is OK. Starting ERC...
Warning: Pin 4 of Subcircuit U2 is not connected.
Warning: Wire is not connected.
Done (0 errors; 2 warnings)

Click any of the errors/warnings above to highlight the questionable wires or components in the schematic editor.

Automatic ERC
 Show on Warnings

Re-check Close Help

Click on error line and the error is highlighted in the schematic



DC Analysis

Select: *Analysis > DC Analysis > Calculate nodal voltages*

The screenshot shows the TI Schematic Editor interface for a circuit named 'opa348_w_1u'. The 'Analysis' menu is open, and 'DC Analysis' is selected, which has opened a sub-menu where 'Calculate nodal voltages' is highlighted. A 'Voltages/Currents' dialog box is also open, displaying a table of results for various components in the circuit.

Component	Value
_R100[7,47]	-18.75uA
_R101[56,55]	-2.29nA
_R102[56,58]	-2.29nA
_R103[53,19]	-2.29nA
_R104[57,19]	-2.29nA
_R105[4,42]	-27.43uA
_R106[10,41]	27.43uA

The dialog box also includes a 'Show' section with the following options checked:

- Nodal Voltages
- Currents
- Other Voltages
- Outputs



AC Analysis

The screenshot shows the TI Schematic Editor interface. The 'Analysis' menu is open, with 'AC Analysis' selected. A sub-menu is visible, showing 'AC Transfer Characteristic...' as the selected option. The 'AC Transfer Characteristic' dialog box is open, showing the following settings:

- Start frequency: 10 [Hz]
- End frequency: 10M [Hz]
- Number of points: 100
- Sweep type: Logarithmic
- Diagram: Amplitude, Nyquist, Phase, Group Delay, Amplitude & Phase

The schematic diagram shows an OPA348 op-amp configured as a voltage follower. The input is connected to a 2.5V source (V2) through a 2.5V source (V1). The output is connected to a 1uF capacitor (C1). The op-amp is labeled U1 OPA348. The input is labeled VG1. The output is labeled V- and V+.

Select: *Analysis > AC Analysis > AC Transfer Characteristics*

Enter start & end frequencies
Select sweep type and plot diagrams



AC Analysis Transfer Characteristics

Gain-Phase, Bode plot for OPA348 w/1 uF Load

Double click on an axis to arrange the scale
Set Axis box appears

Set Axis

Label
Text: Gain (#dB) OK

Font ... Name: Arial Size: 8 Style: Normal Cancel

Numbers
Format: Engineering 1k Help

Divide by factor: 1 Precision: 2

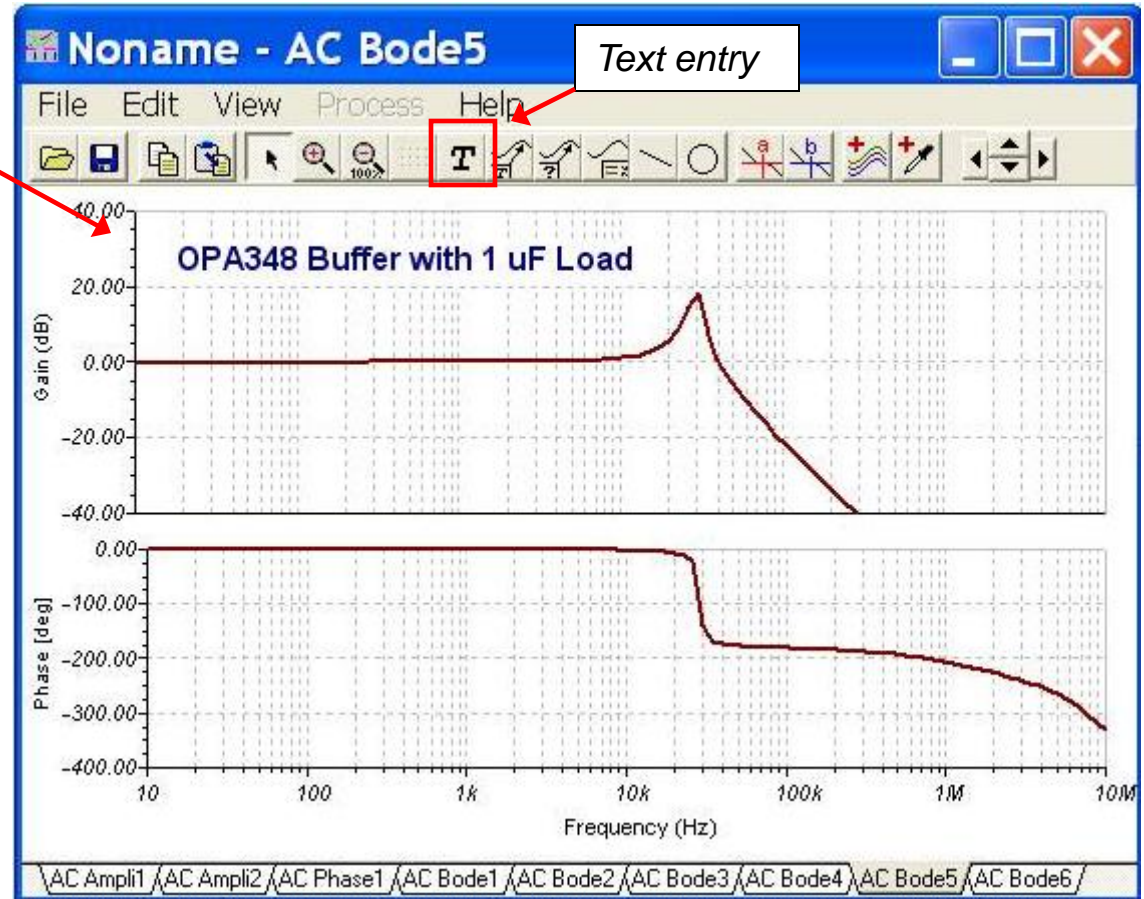
Font ... Name: Arial Size: 8 Style: Italic

Scale
Linear-dB Lower limit: -40

Ticks: 5 Upper limit: 40

Force size settings when resize window
 Round axis scale

Enter axis limits
and other settings





Transient Analysis

Select: *Analysis > Transient*

Signal Editor

VG1 set for
1V unit step
Start at 20us

opa348_w_1u - Schematic Editor

File Edit Insert View Analysis T&M Tools Help

- ERC...
- Mode...
- Select Control Object
- Set Analysis Parameters...
- DC Analysis
- AC Analysis
- Transient...**
- Steady State Solver...
- Fourier Analysis
- Noise Analysis...
- Options...

Transient Analysis

Start display 0 [s]

End display 3m [s]

Calculate operating point
 Use initial conditions
 Zero initial values

Draw excitation

Integration method
 Trapezoidal Gear

Integration order 2

OK Cancel

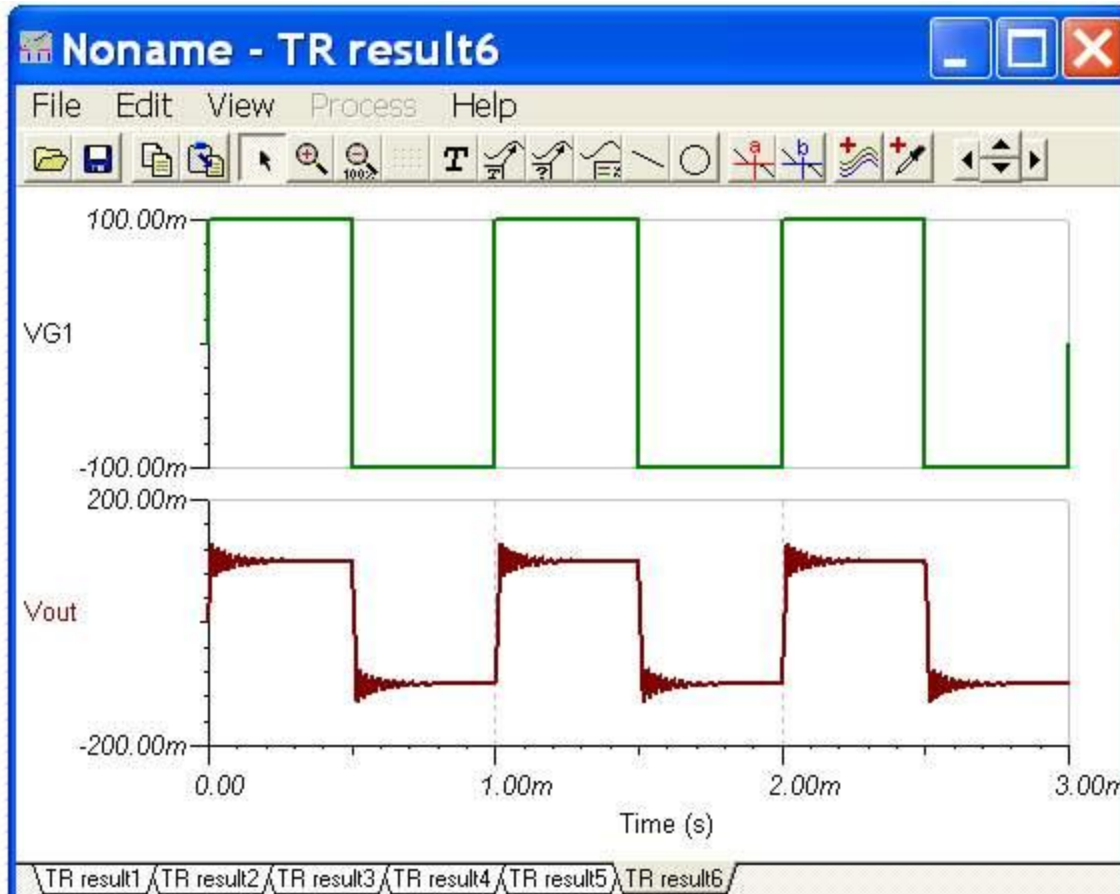
Enter display Start and End times

Select integration method and order

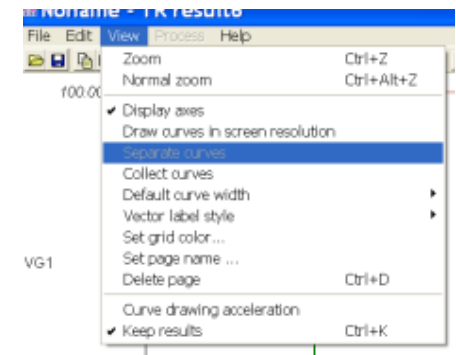


Time domain Graphs

Square Wave Response – OPA348 Buffer with 1 uF load

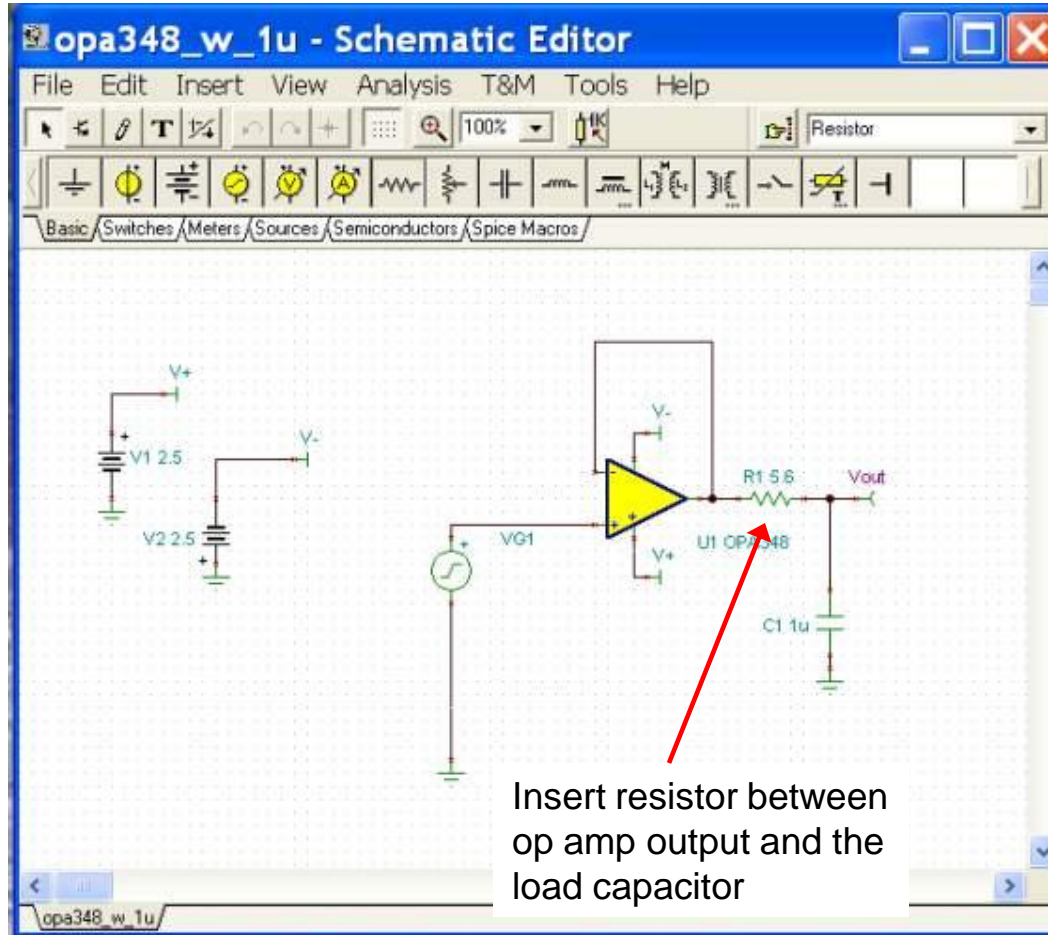


Separate Curves





Design a Stable OPA348 Buffer



$$\text{GBWP}_{\text{OPA348}} = 1 \text{ MHz}$$

$$C_L = 1 \text{ uF}$$

$$R_L \geq 2 \text{ ohms}$$

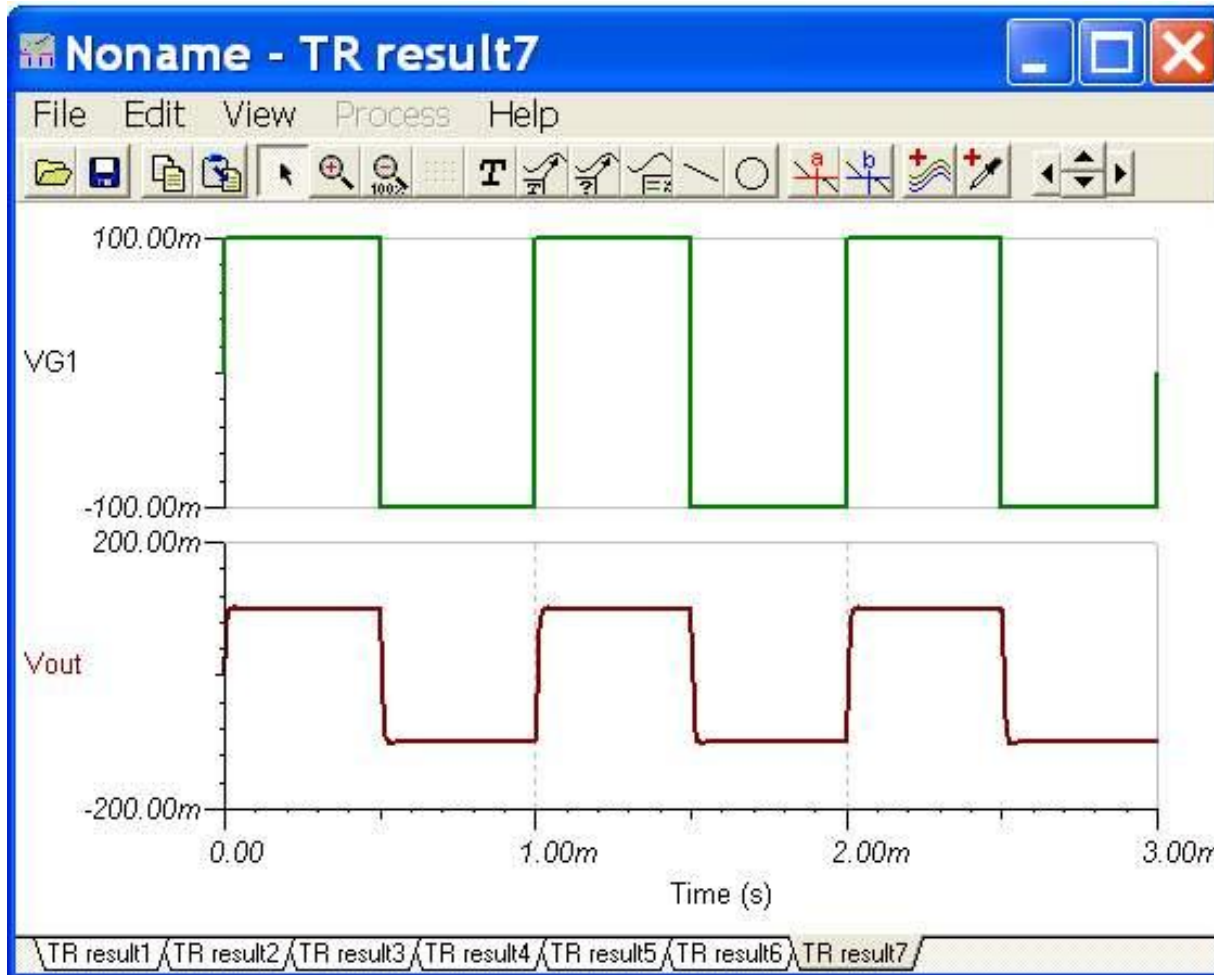
also

$$R_O \text{ of OPA348} \sim 50 \text{ ohms}$$

$$R_L \geq 5.6 \text{ ohms}$$



Stable System - Transient Signal Results



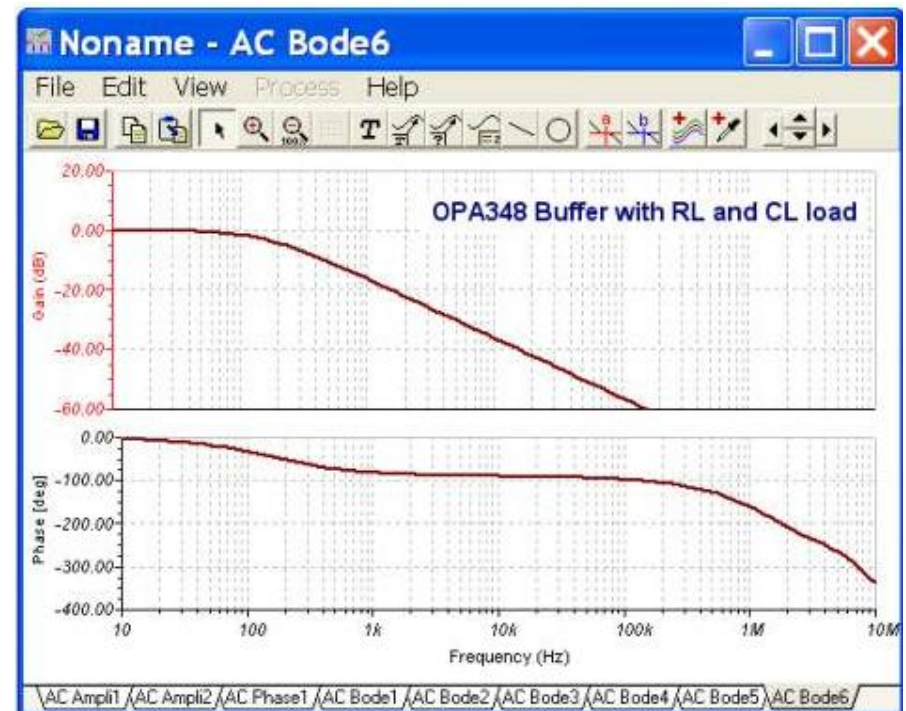
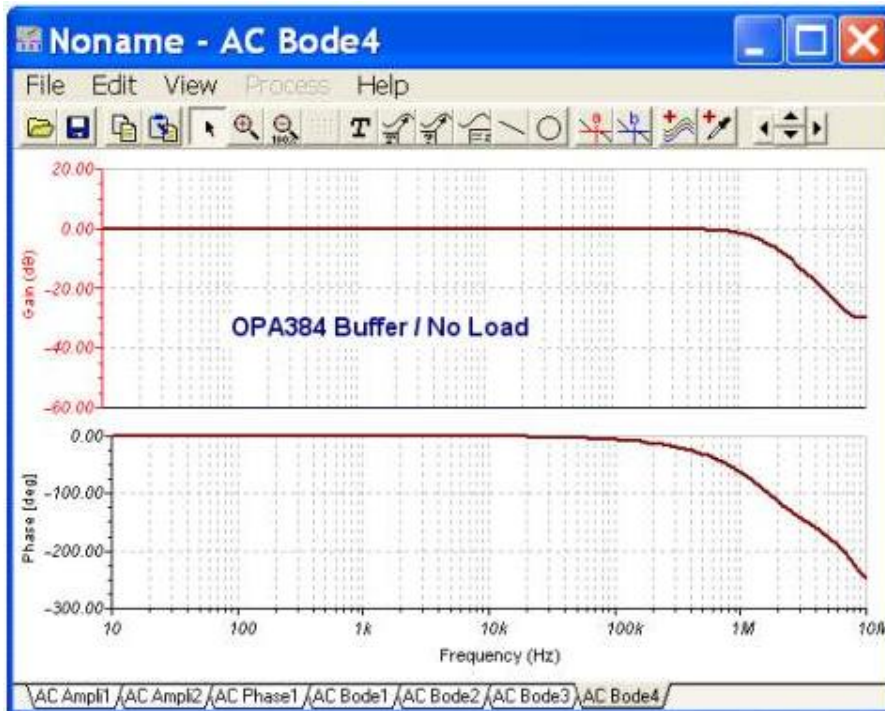
$$C_L = 1 \text{ uF}$$

$$R_L = 5.6 \text{ ohms}$$



Go Back to the AC Analysis

C_L has Modified the Open-loop gain of the OPA348 Considerably





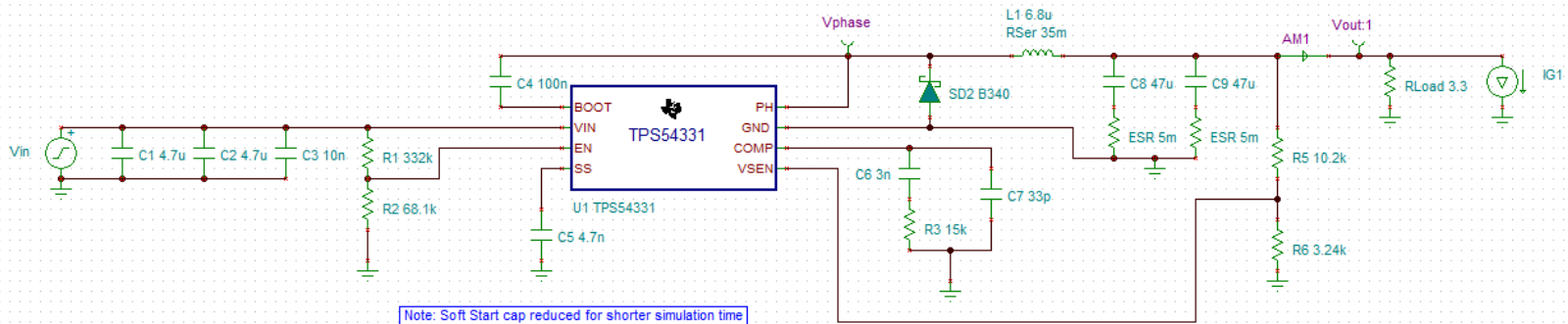
Power Modeling in TINA-TI v9

- TINA-TI v9 is much faster for power simulation
 - Improved convergence algorithms
 - Multi-core support
- TINA-TI v9 has more power model support
 - Over 200 power models built-in
 - Reference Designs for each power model



Power Example

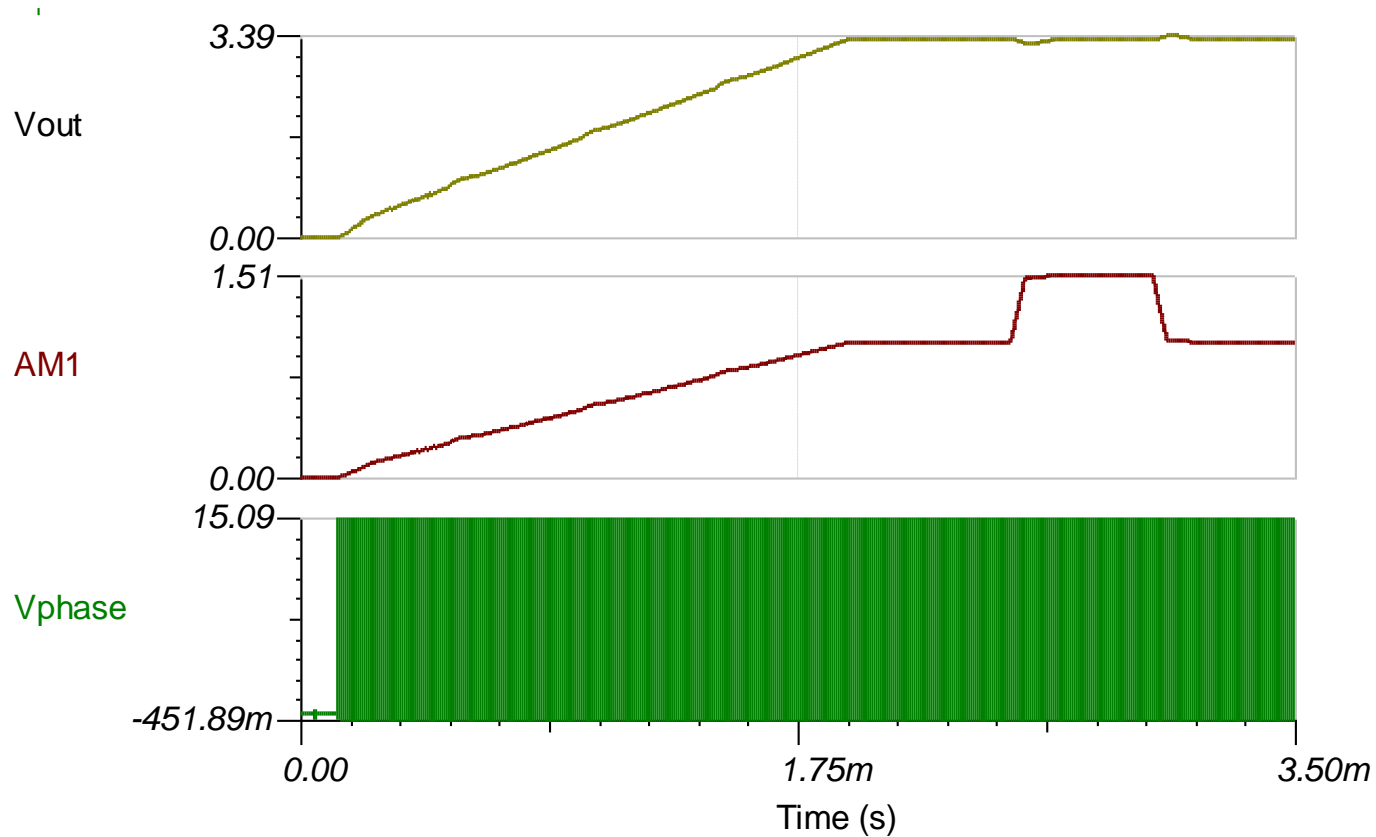
A Typical Load transient simulation for the TPS54331





Power Example

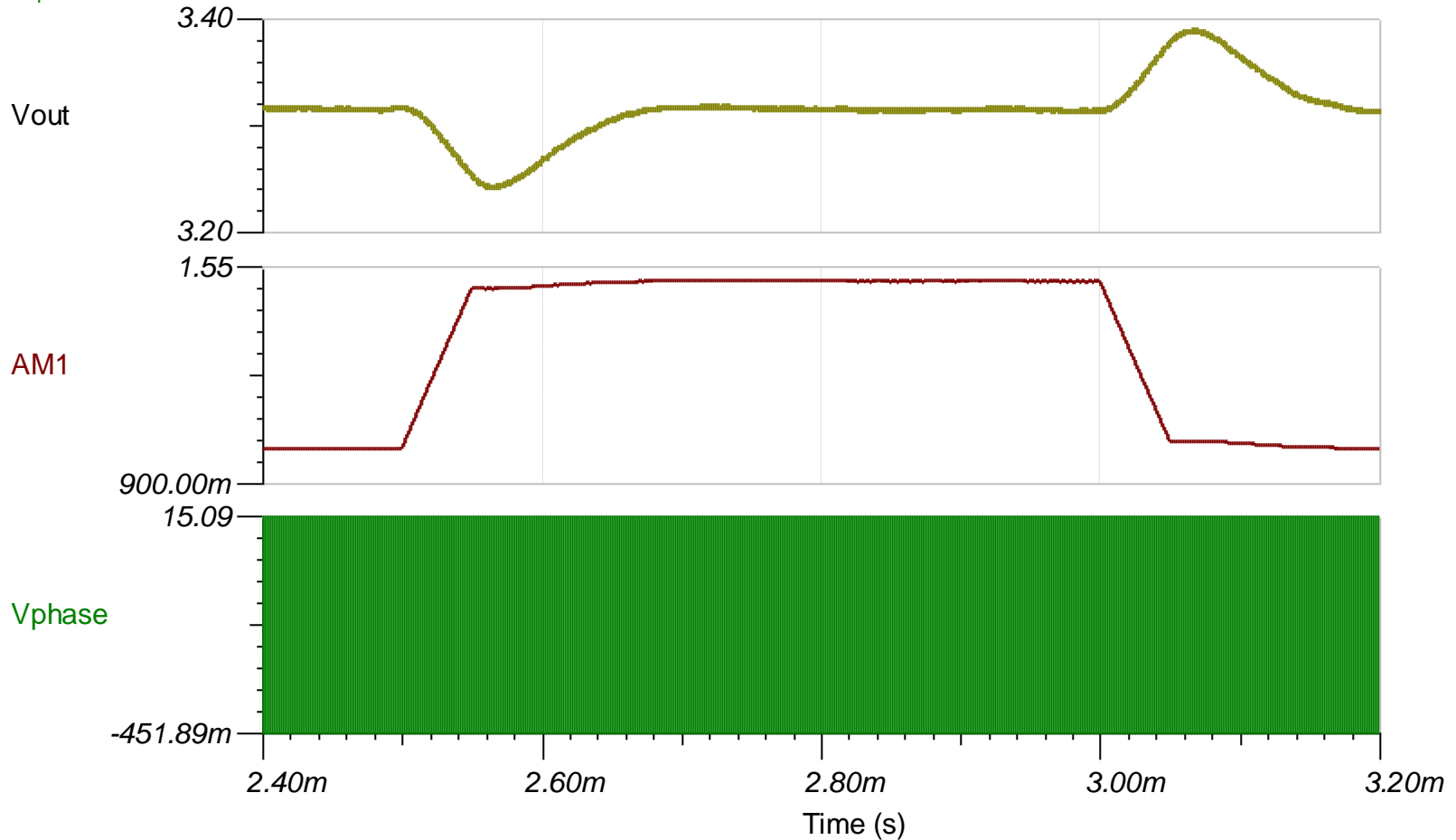
Results: 3.3V V_{out} @ 1A with a 500mA pulse





Power Example

Results: 3.3V V_{out} @ 1A with a 500mA pulse





Other TINA-TI Features

- Importing 3rd Party Models
- Noise Analysis
- Fourier Analysis – Distortion
- DC Sweep
- Post Processing
- Test and Measurement
- Parameter Stepping



Importing a 3rd Party Model into TINA-TI

```
b340a.cir - Notepad
File Edit Format View Help
*****
* Copyright:
* Thomatronik GmbH, Germany *
* info@thomatronik.de *
*****
* SPICE3
.subckt b340a 1 2
ddio 1 2 legd
dgr 1 2 grd
.model legd d is = 1.29079E-005 n = 1.30606 rs = 0.0238242
+ eg = 0.651876 xti = 1.88931 tnom = 27
+ cjo = 8.63505E-010 vj = 0.700074 m = 0.617102 fc = 0.5
+ tt = 1.4427E-009 bv = 44 ibv = 10 af = 1 kf = 0
.model grd d is = 1E-015 n = 0.780978 rs = 0.311944
+ eg = 1.00036 xti = 0.205739 tnom = 27
.ends
```





Importing a 3rd Party Model into TINA-TI

The image shows the TINA-TI Schematic Editor interface with two 'New Macro Wizard' dialog boxes open. The background window is titled 'Noname - Schematic Editor' and has a menu bar (File, Edit, Insert, View, Analysis, T&M, Tools, TI Utilities, Help) and a toolbar. A 'Tools' menu is open, with 'New Macro Wizard...' highlighted. A mouse cursor points to this option.

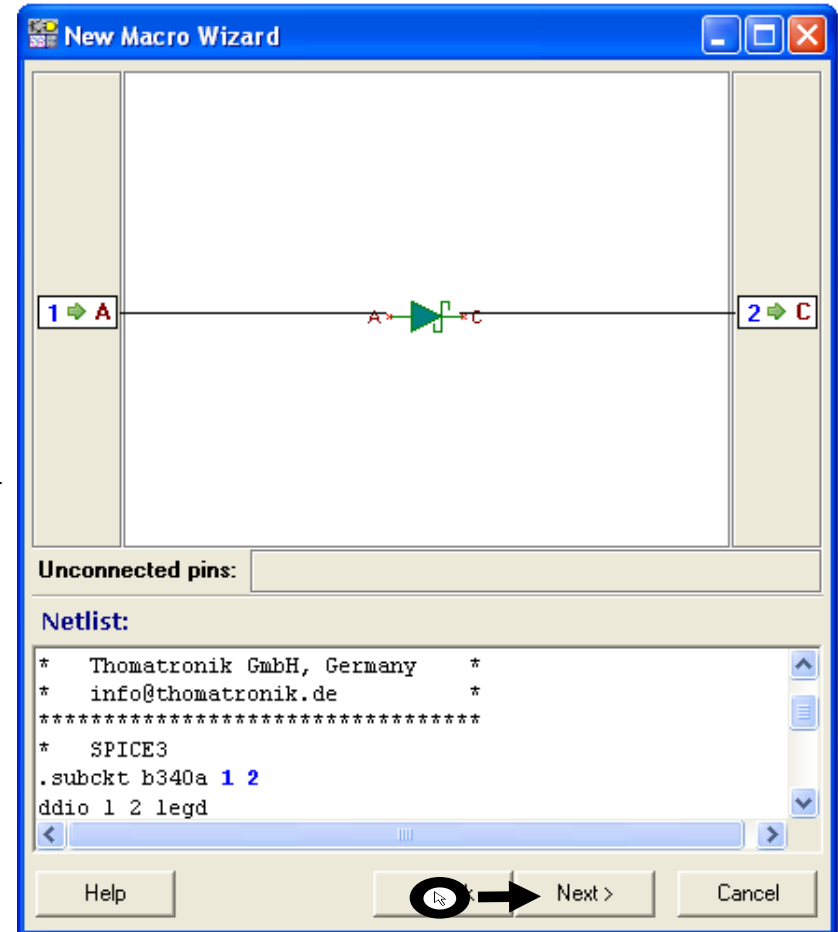
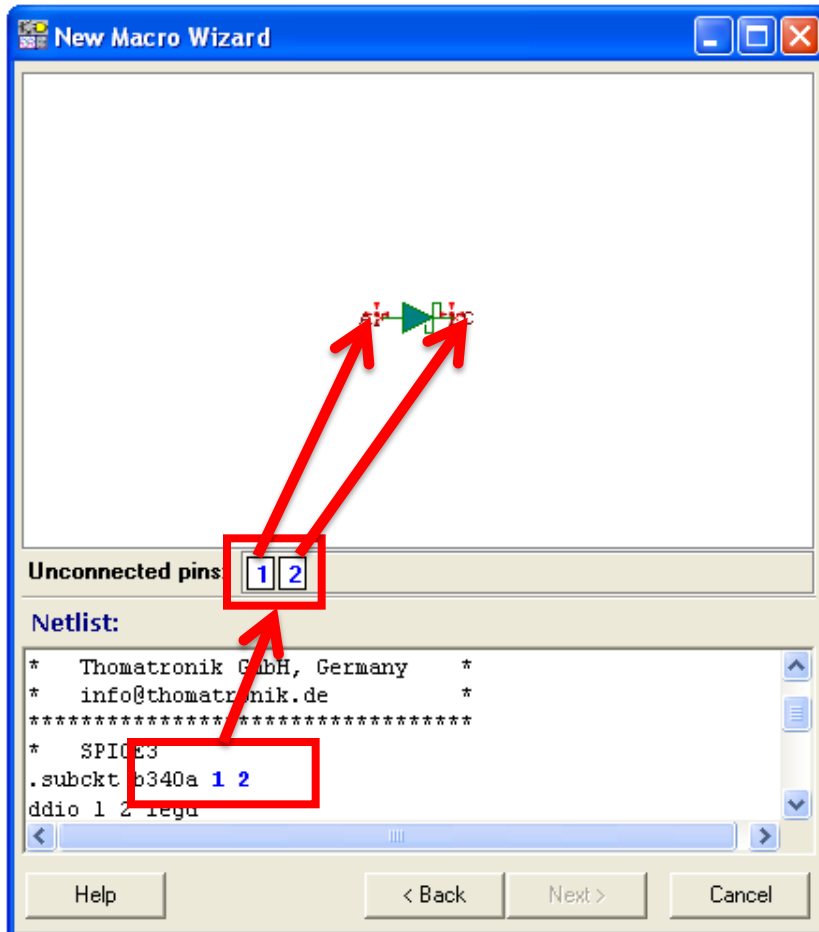
The first 'New Macro Wizard' dialog box is titled 'Select the source of the macro.' It has a 'Macro Name' field containing 'B340A Diode'. Below this are radio buttons for 'Empty circuit', 'Current circuit', 'From file' (selected), and 'From the Web'. The 'From file' option has a file path field containing 'C:\Documents and Settings\kraz\Desktop\b340a.cir' and a 'Browse' button. At the bottom, there are 'Help', 'Next >', and 'Cancel' buttons. A mouse cursor points to the 'Next >' button.

The second 'New Macro Wizard' dialog box is titled 'Select the shape you want to assign:'. It has two radio buttons: 'Auto generate shape' and 'Load shape from library' (selected). Below this is a list box containing 'All' and '64'. A preview window shows a diode symbol with pins labeled 'A' and 'C'. Below the preview is a 'Filter' section with a checkbox 'Show suggested shapes only.' and a note: '(Notice: If you can't find the shape you are looking for, uncheck this checkbox.)'. There is a 'Search:' text box. Below that are two dropdown menus: 'Number of pins' (set to '1-5') and 'Shape type' (set to 'Diodes'). At the bottom, there are 'Help', '< Next >', and 'Cancel' buttons. A mouse cursor points to the '< Next >' button.

Arrows indicate the flow of the process: from the 'Tools' menu to the first dialog, then to the second dialog, and finally to the 'Next >' button in the second dialog.

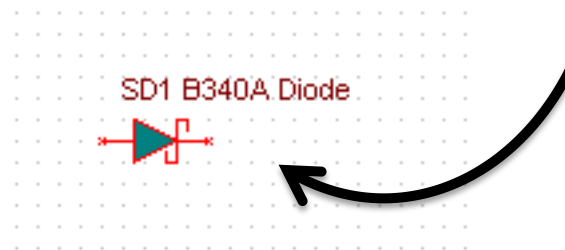
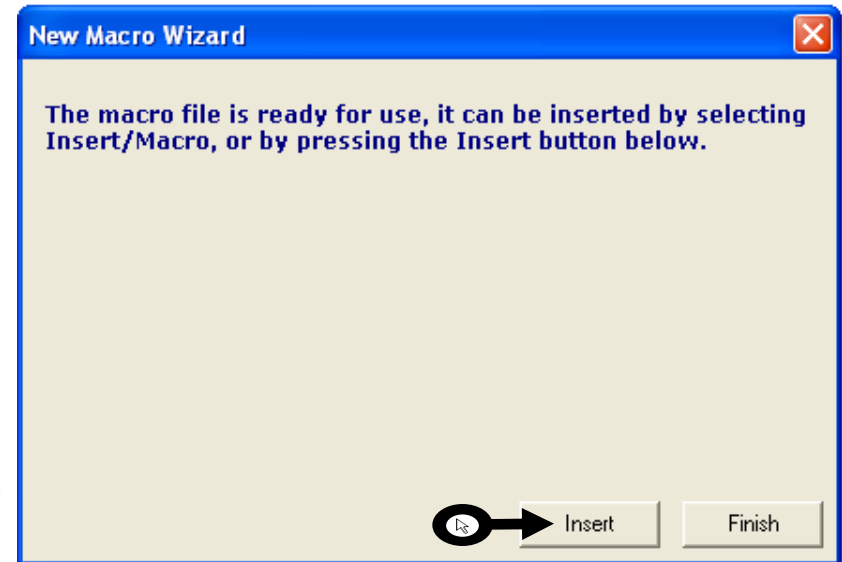
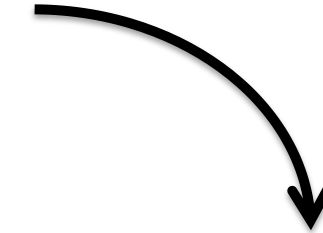
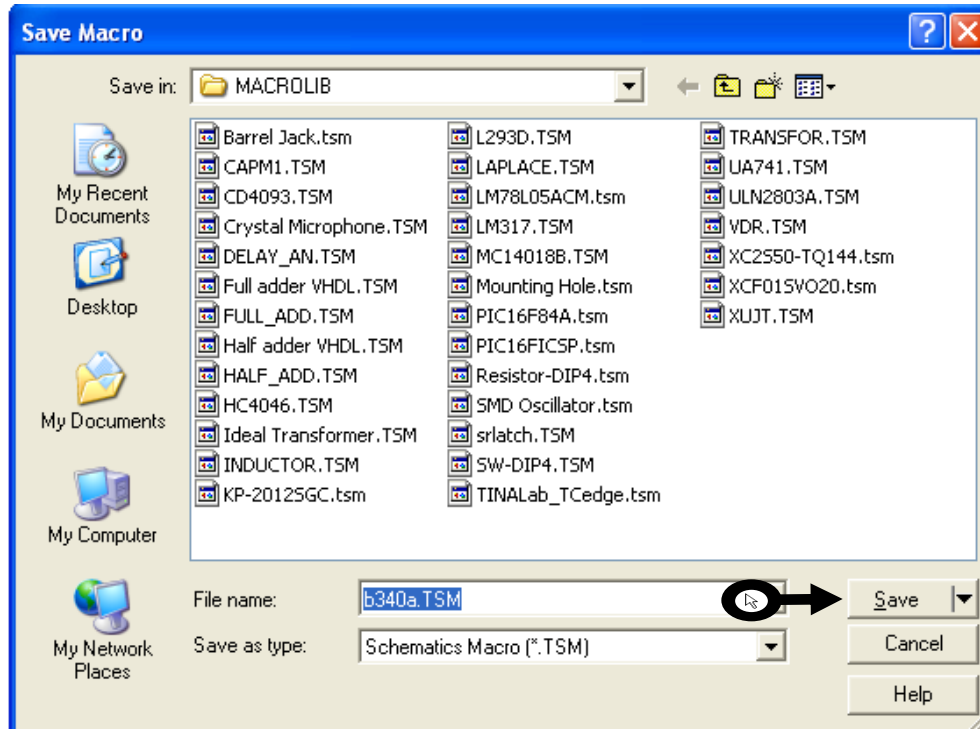


Importing a 3rd Party Model: Getting the Pinout Correct





Importing a 3rd Party Model

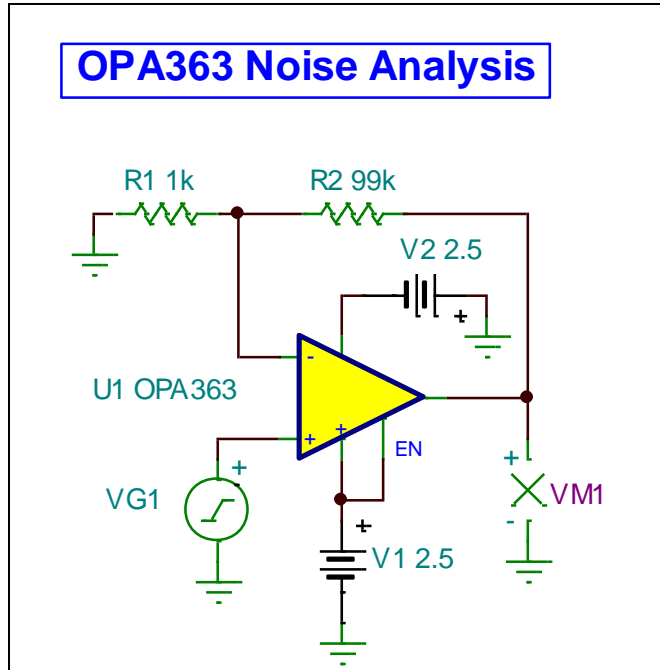




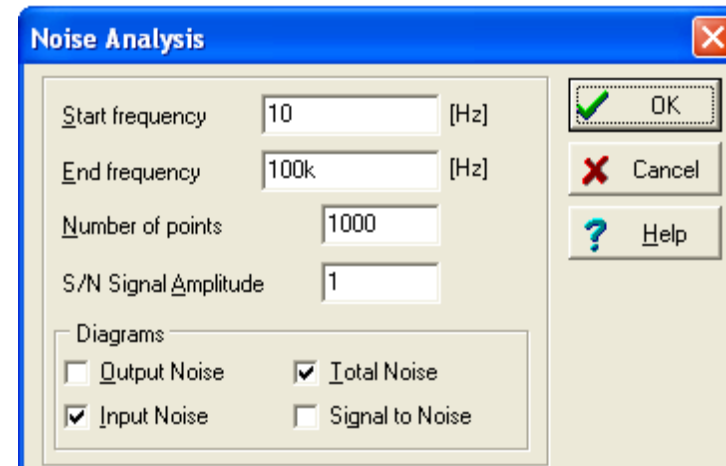
Noise Analysis

OPA363 + resistor noise

OPA363 Noise Analysis



Select: *Analysis > Noise Analysis*

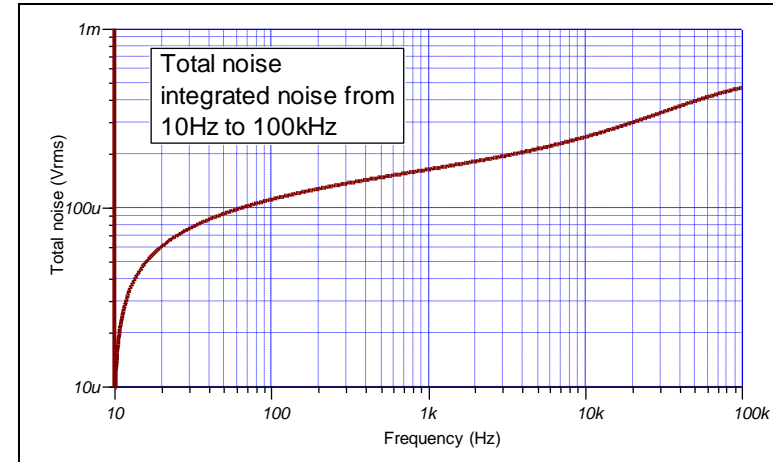
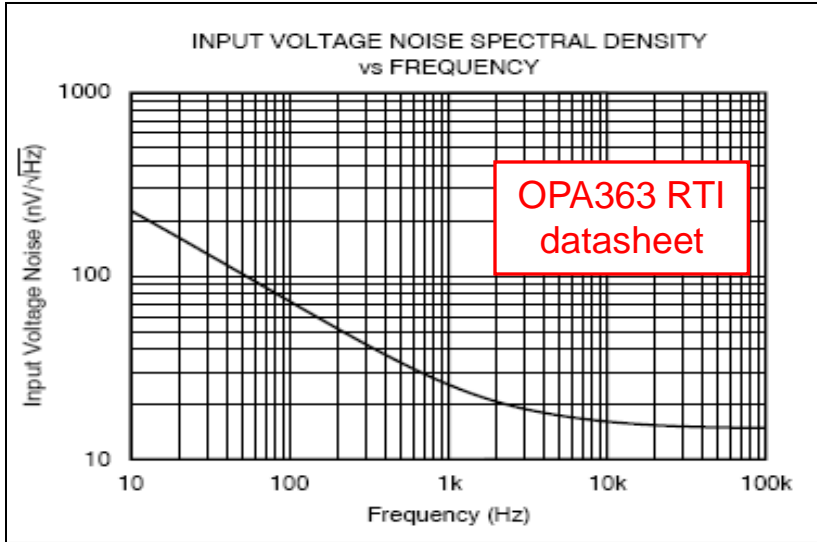


Input Noise and Total Noise responses are selected for simulation

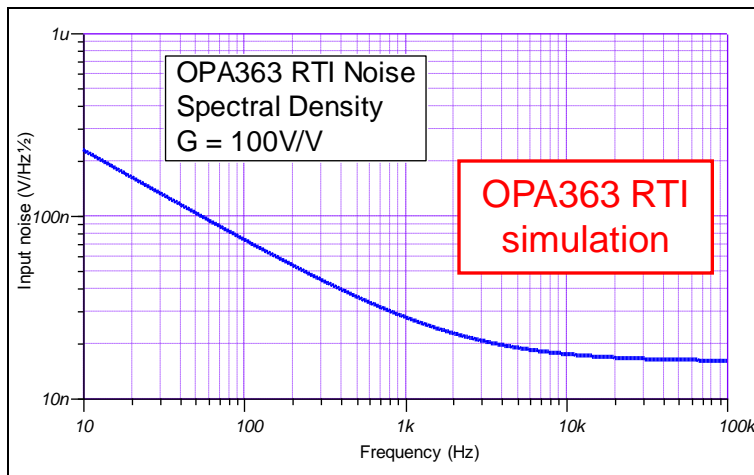


Datasheet vs. Simulation

Noise Analysis



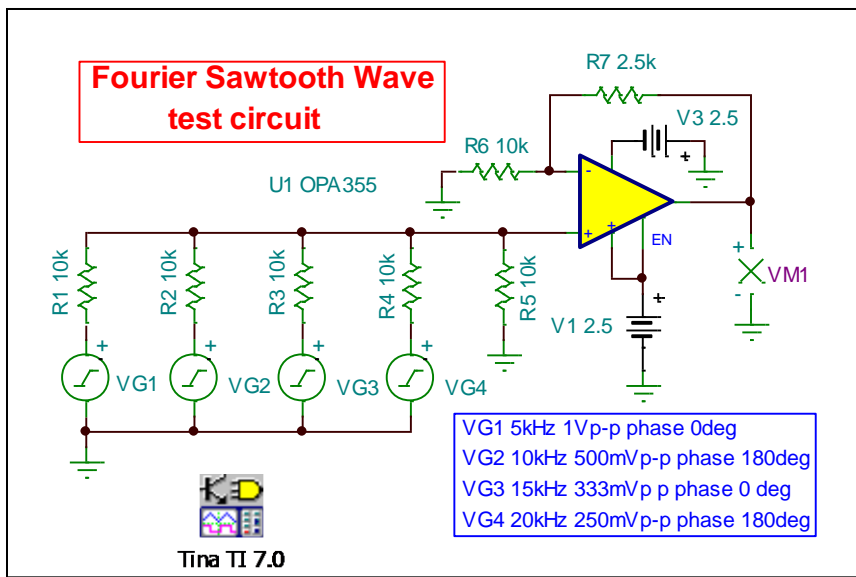
OPA363 Total noise simulation



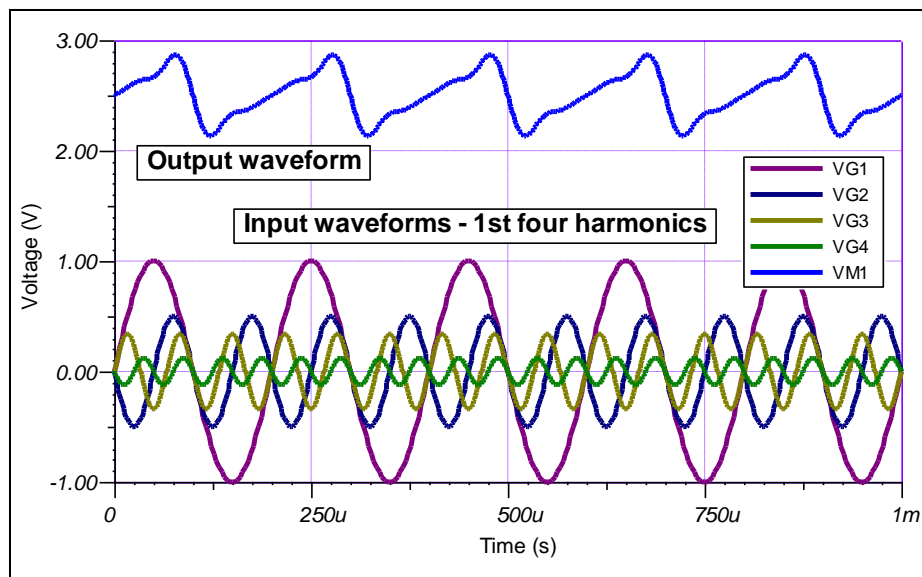


Fourier Analysis

Fourier Sawtooth Wave test circuit



First Step: perform a Transient Analysis



Result from TINA-TI Transient Analysis

Fourier Series for sawtooth wave

$$x(t) = \frac{2v}{\pi} \left(\sin \omega_0 t - \frac{1}{2} \sin 2\omega_0 t + \frac{1}{3} \sin 3\omega_0 t - \frac{1}{4} \sin 4\omega_0 t + \dots \right)$$

where: $\omega_0 = \frac{2\pi}{T}$ $T = \text{period}$ $v = V_{\text{peak}}$



Frequency, Amplitude and Phase Fourier Analysis

Fourier Analysis > Fourier Series

Fourier Series

Sampling start time: 400u
Base frequency: 5k
Number of samples: 16384
Number of harmonics: 7
Format: $C * \exp(j * (kwt + fi))$
Output: VM1

Transient initial condition
 Calculate operating point
 Use initial conditions
 Zero initial values

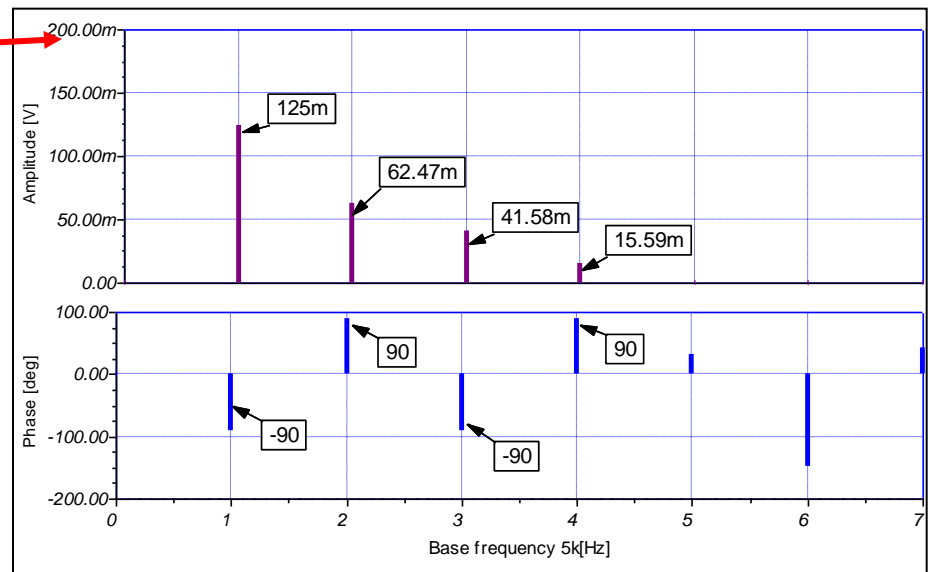
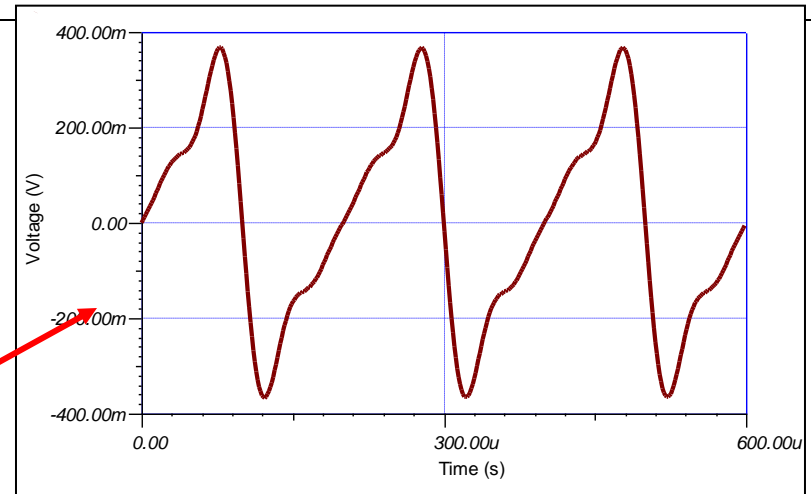
Buttons: Calculate, Cancel, Help, Draw

Fourier coefficients

k	Amplitude (C)	Phase (φ)
0.	250.74u	0
1.	125m	-90
2.	62.47m	90
3.	41.58m	-90
4.	15.59m	90
5.	20.39p	31.22
6.	9.48p	-148.32
7.	2.86p	43.07

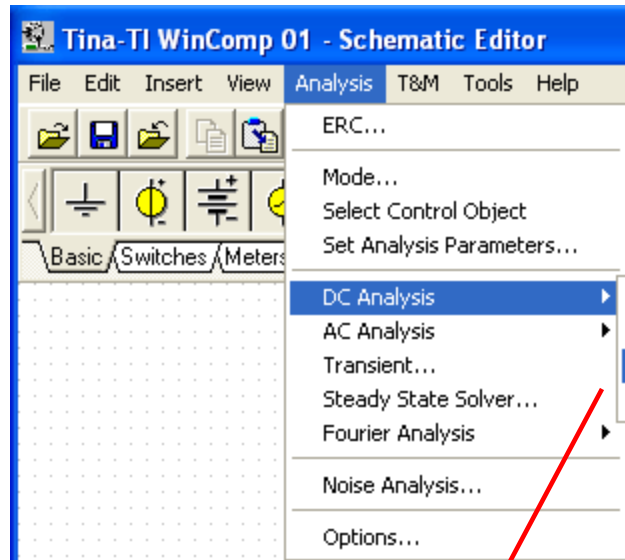
Harmonic distortion: 61.322%

- Press Calculate
- Then Draw





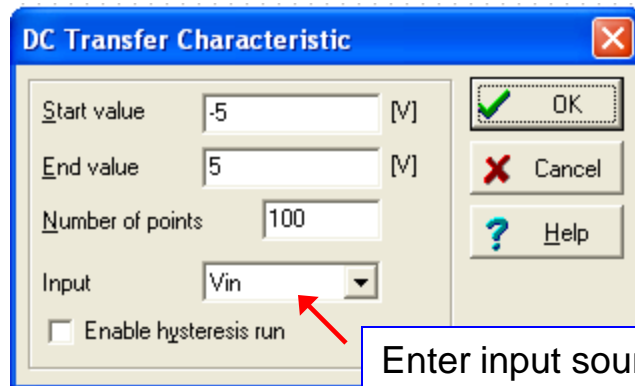
DC Analysis – DC Sweep With a Window Comparator



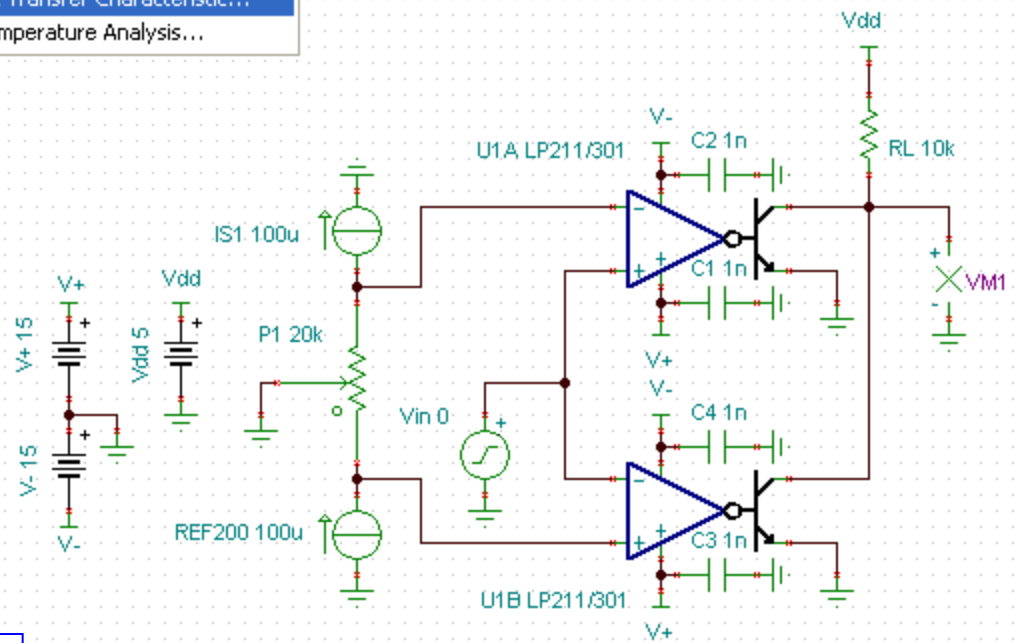
Select: *Analysis > DC Analysis > DC Transfer...*

TINA-TI window comparator example

Enter start and end sweep voltages



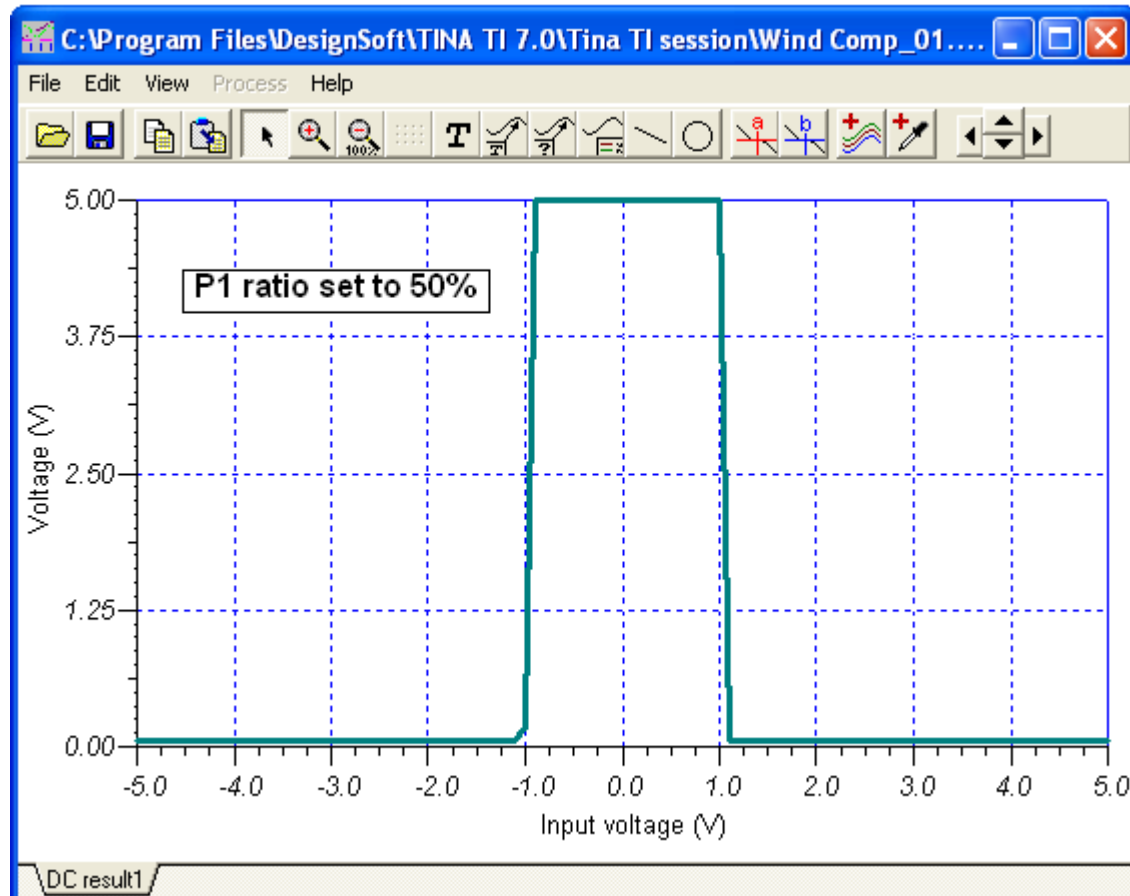
Enter input source





Comparator Output Response vs. Input Source

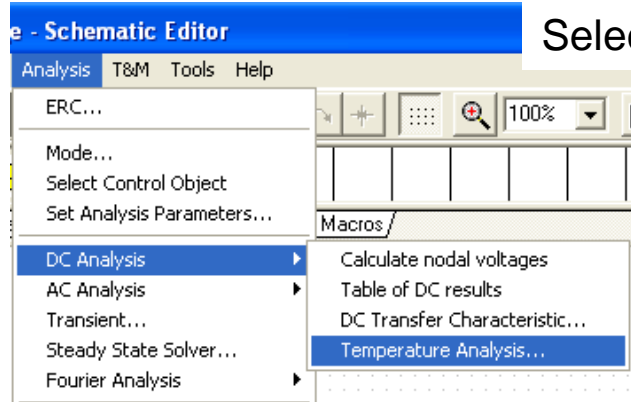
Vertical scale:
The output
response



Horizontal scale: The input source, V_{IN} , is swept from -5V to +5V

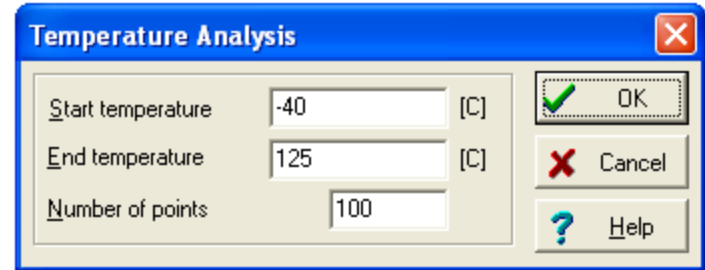


DC Analysis: Temperature Analysis

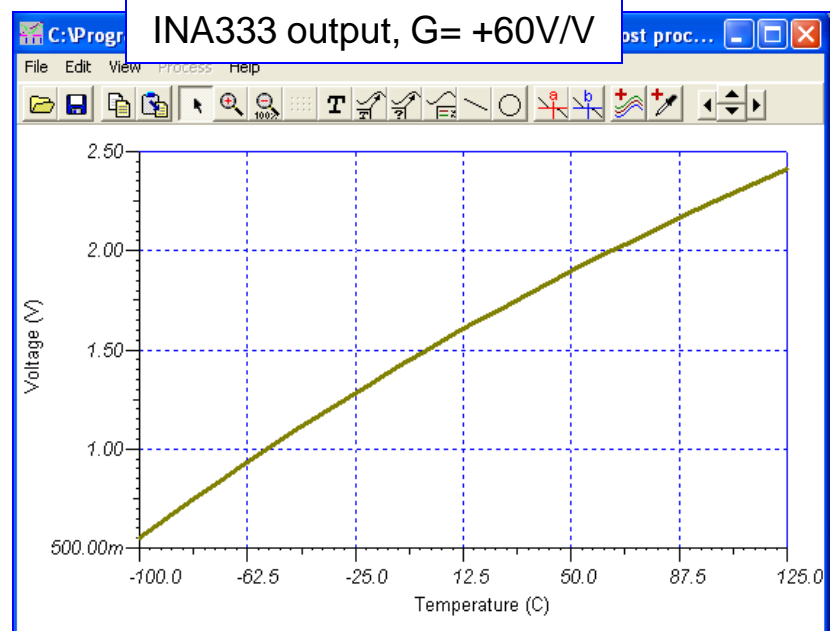
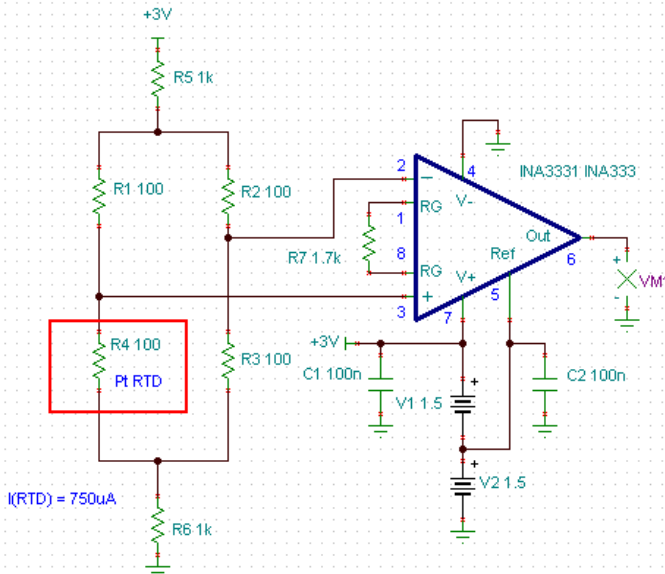


Select: *Analysis > DC Analysis > Temperature Analysis*

Enter *Start*
and *End*
temperatures



Pt100 RTD Temperature Sweep
INA333 chopper IA is drift free

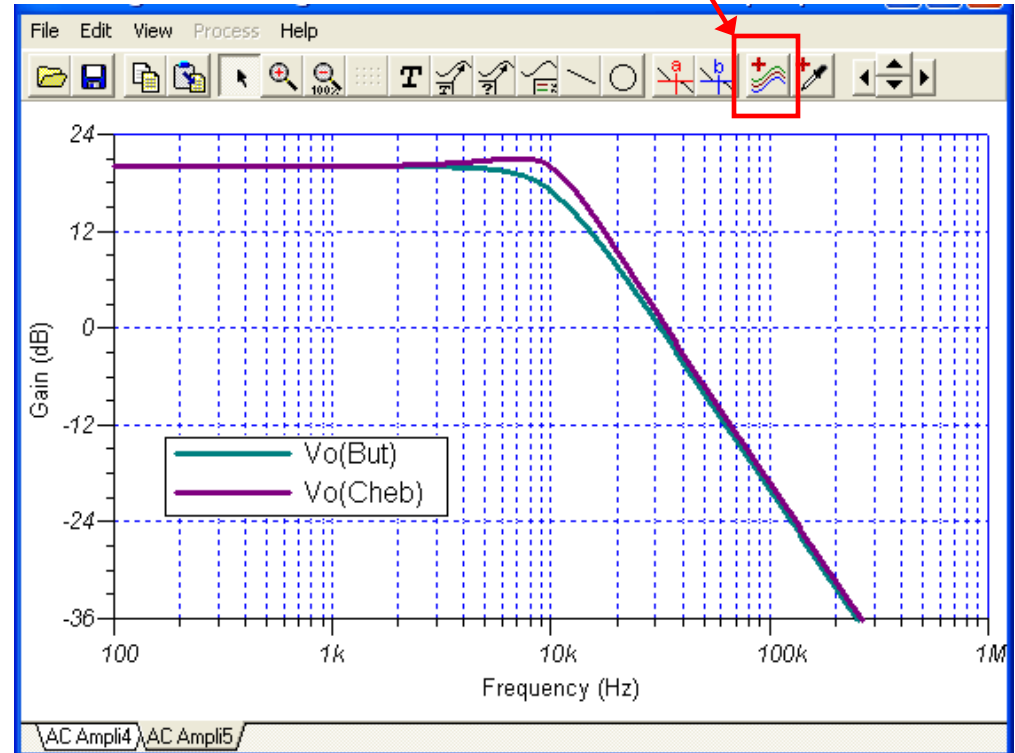
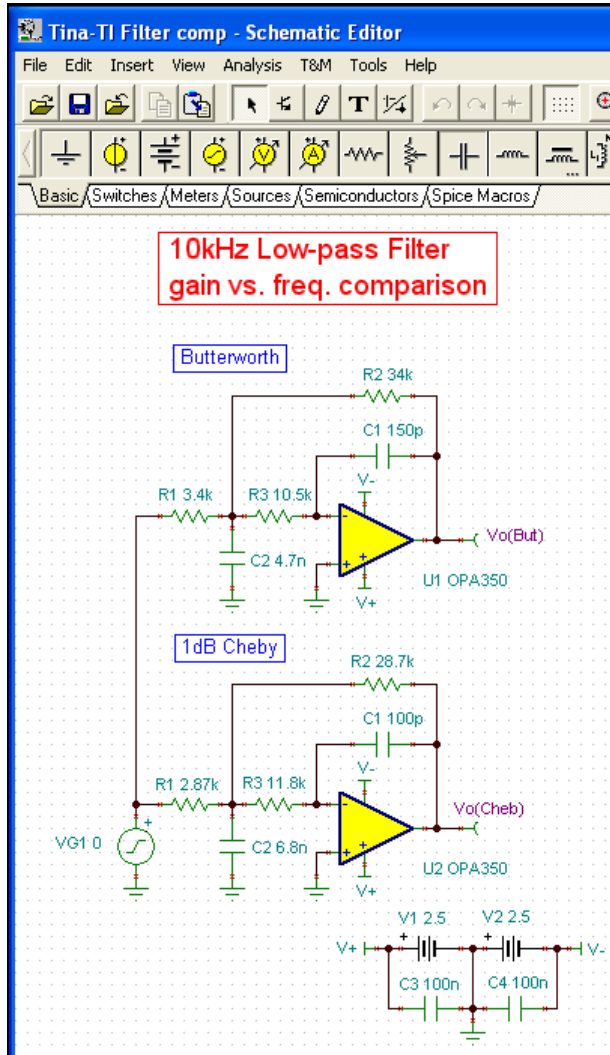




Post Processing Analysis

Goal: Find the difference in the two filter gains across frequency

Select the *Post Processing* function icon





Math Tool Applied after an Analysis

The image shows two sequential screenshots of the 'Postprocessing' dialog box. The first screenshot shows the 'Available curves' list with 'Vo(But)' and 'Vo(Cheb)' selected. The 'Show' section has 'Outputs' checked. The 'Line Edit' field contains 'Vo (But) (s)'. The second screenshot shows the 'Curves to insert' list with 'Vo(But)' and 'Vo(Cheb)' selected. The 'Show' section has 'Outputs' checked. The 'Line Edit' field contains 'Vo (But) (s) / Vo (Cheb) (s)'. The 'Built-in functions' dropdown is set to '/'. The 'New function name' field contains 'FilDif'. The 'Preview' button is highlighted.

Select a filter's output voltage

Select **Outputs**

Enter each voltage in the line editor using the down arrow

Add math function " / " (s);

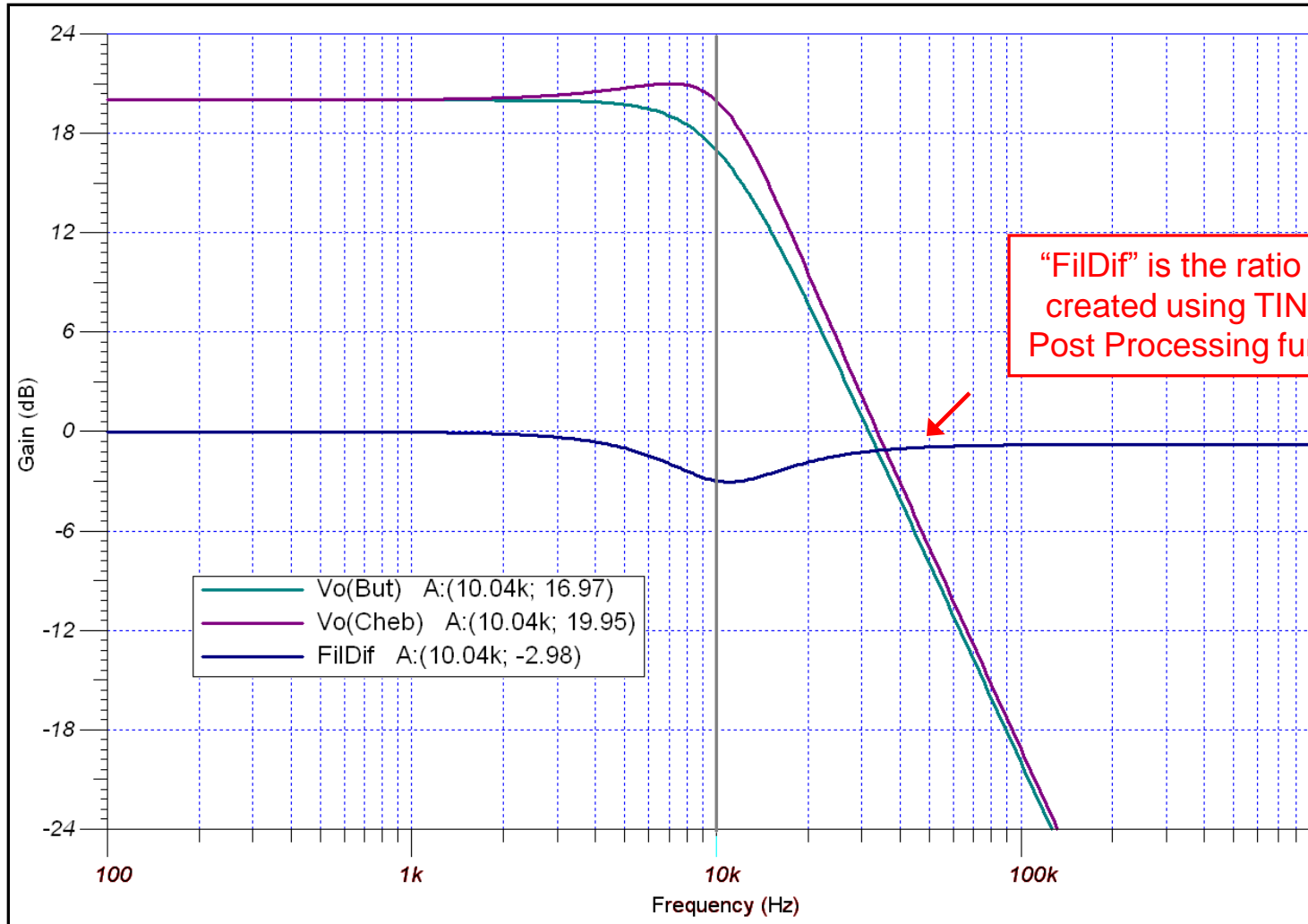
Select math operator from "Built-in functions"

Add using down arrow

Name new function then *Preview* and *Create*



Ratio of Two Curves



“FilDif” is the ratio curve created using TINA-TI’s Post Processing function



Test & Measurement: Using Virtual Instruments

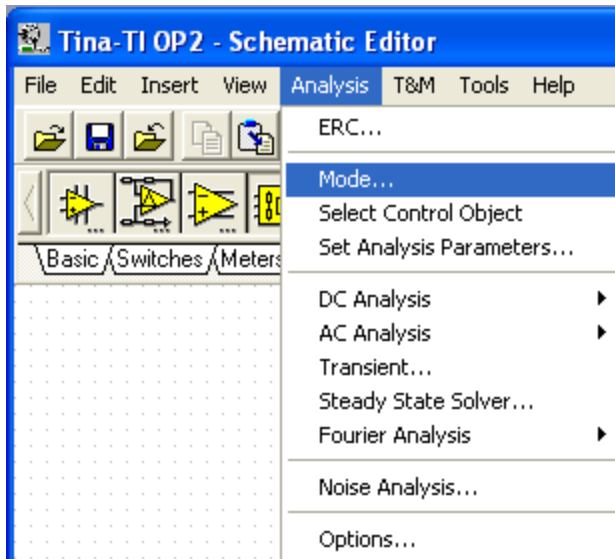
Select: *T&M > Oscilloscope (etc.)*

TLC555 Astable Oscillator
 $f = 10\text{kHz}$, 20% duty cycle

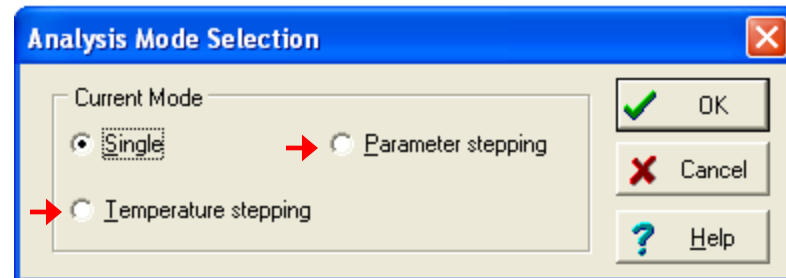
A circuit measurement point is selected by name



Temperature and Parameter Stepping



Select: *Analysis > Mode*

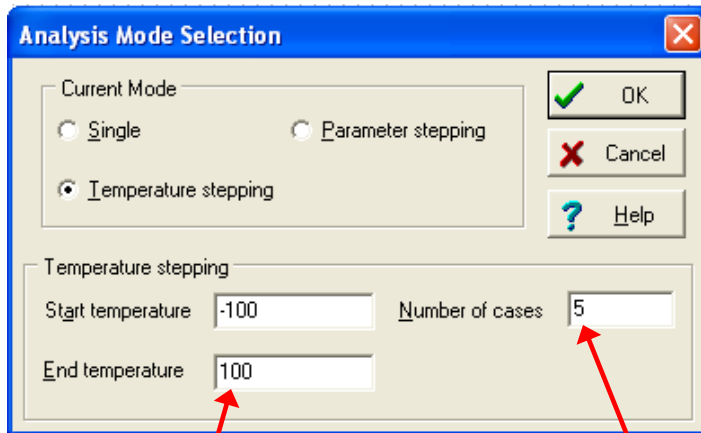


Select: *Temperature stepping
or Parameter stepping*



Temperature Stepping Applied to a DC Sweep

Temperature Stepping is selected



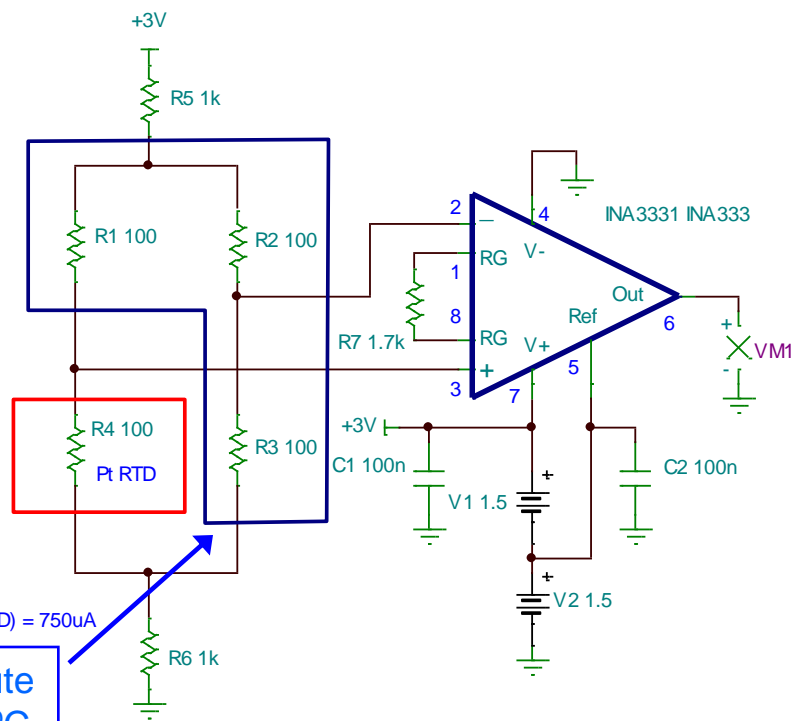
The temperature will be stepped across 5 discrete temperature from -100°C to +100°C

Set RTD Relative Temperature to 0°C

Set R1– R3 Absolute Temperature to 27°C

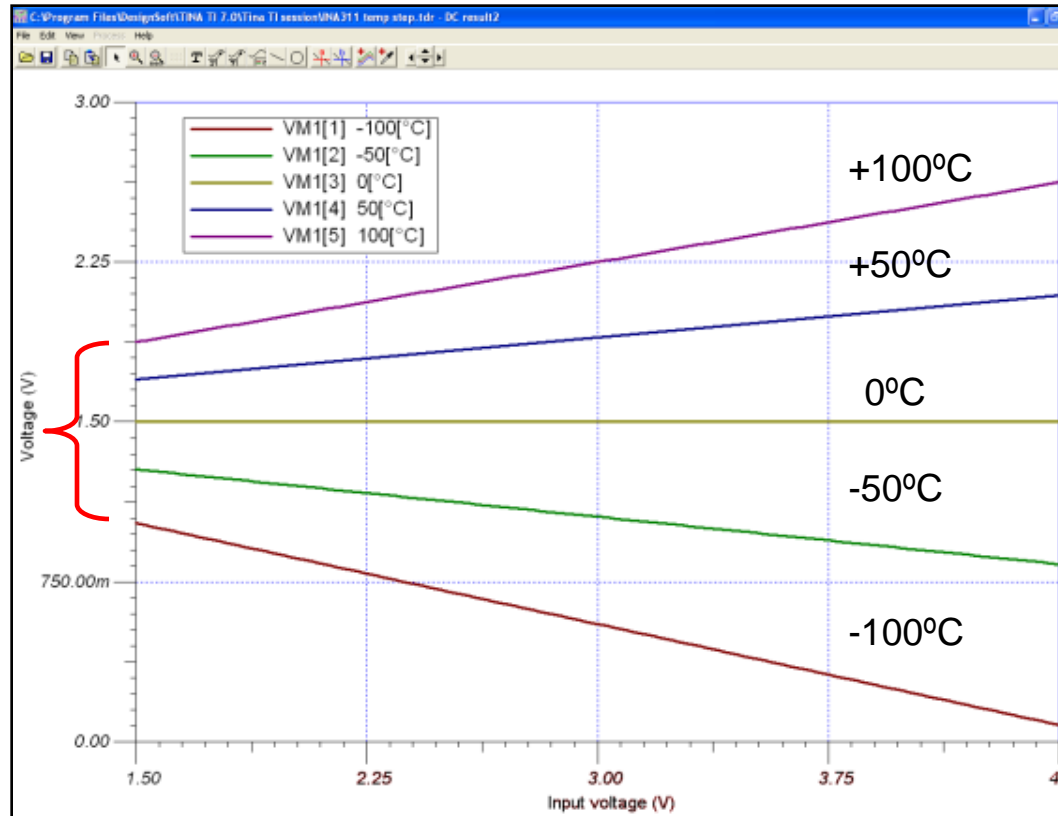
VS1 is swept using *DC Analysis* > *DC transfer* > *Characteristic* to derive the output span at stepped temperatures

Pt100 RTD Temperature Sweep
INA333 IA has negligible drift





Voltage Out vs. Temperature vs. Input Voltage



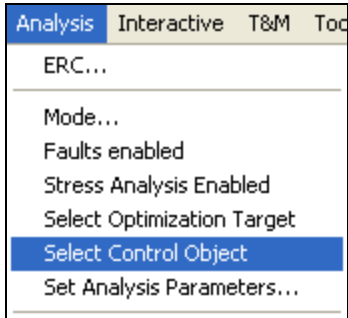
Vm1 output span with V_{S1} at 1.5V

Vm1 output span with V_{S1} at 4.5V



Parameter Stepping: DC and AC Example

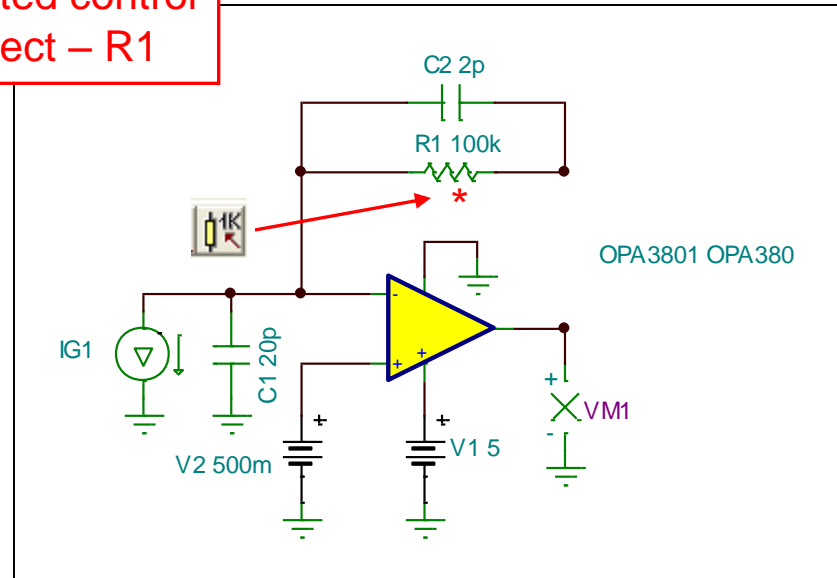
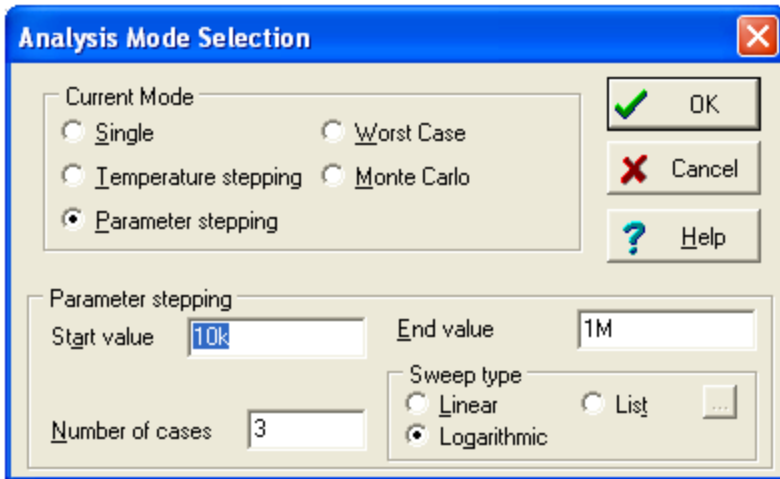
OPA380 Transimpedance Amplifier
R1 is Stepped to Affecting Gain and AC Response



or



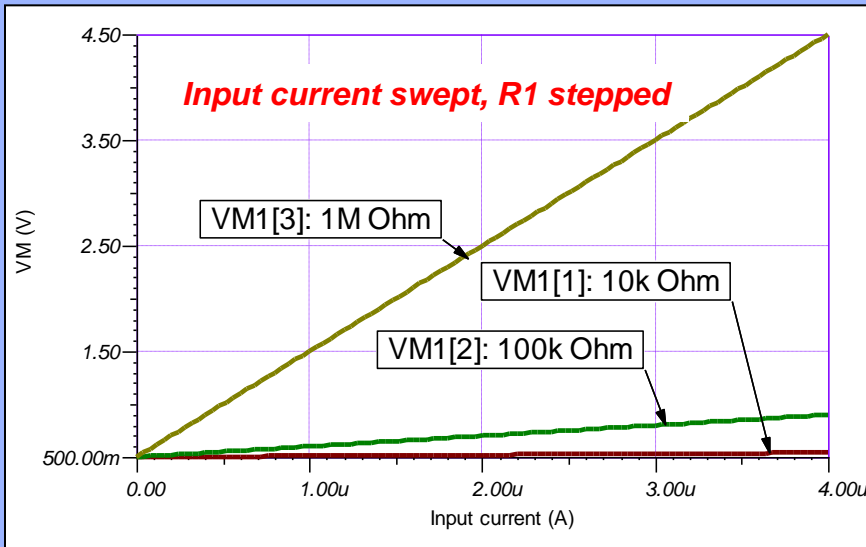
Selected control object – R1



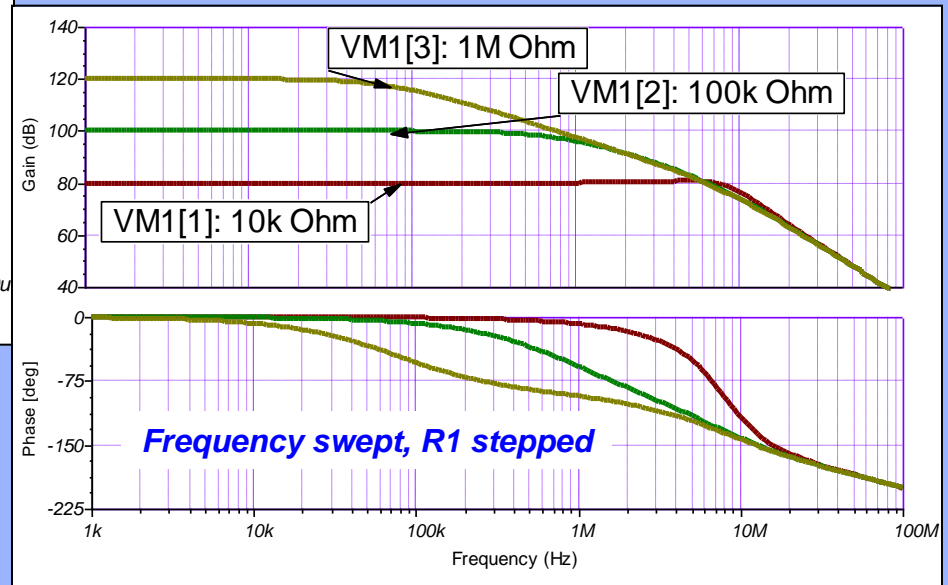


Parameter Stepping: DC and AC Example

OPA380 transimpedance amplifier example
R1 Log stepped 10k Ω , 100k Ω , 1M Ω



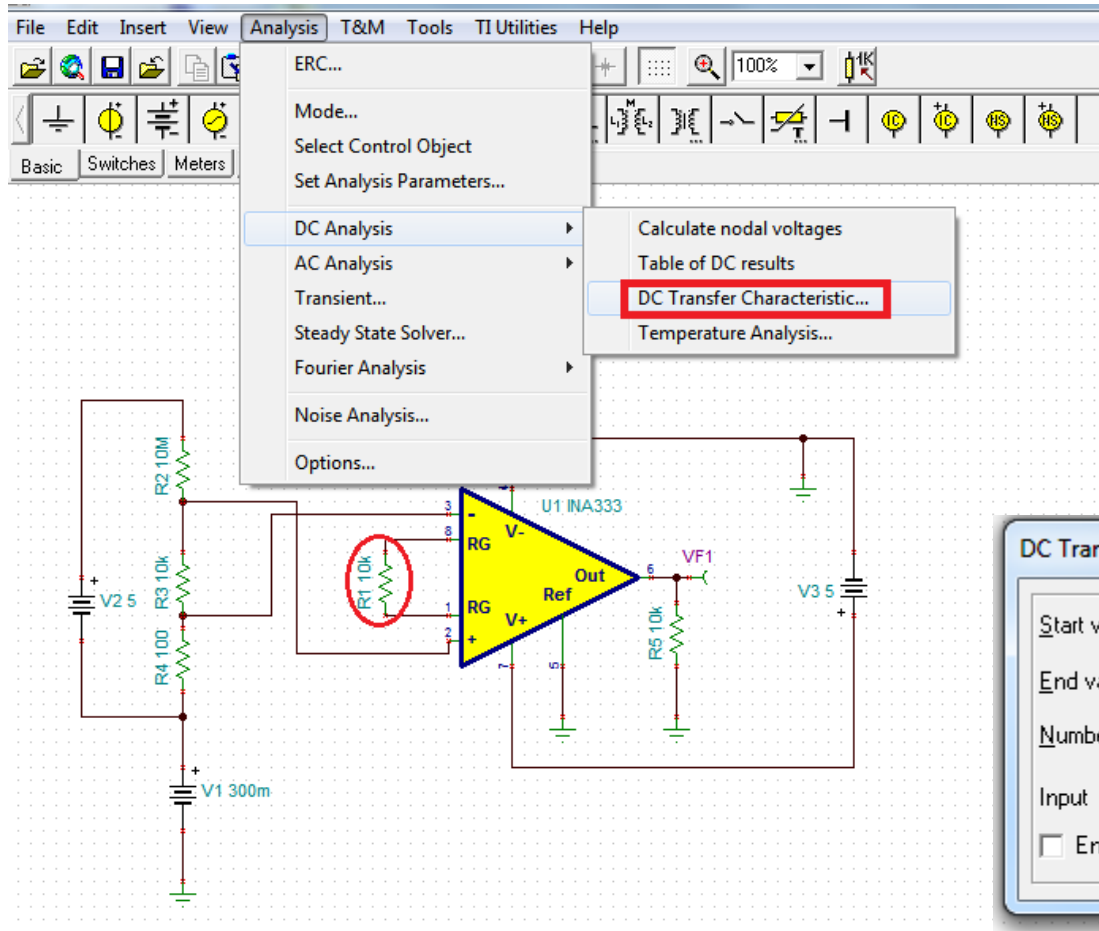
Select: *AC Analysis* >
AC transfer characteristic



Select: *DC Analysis* >
DC transfer characteristic



DC transfer for parameter stepping



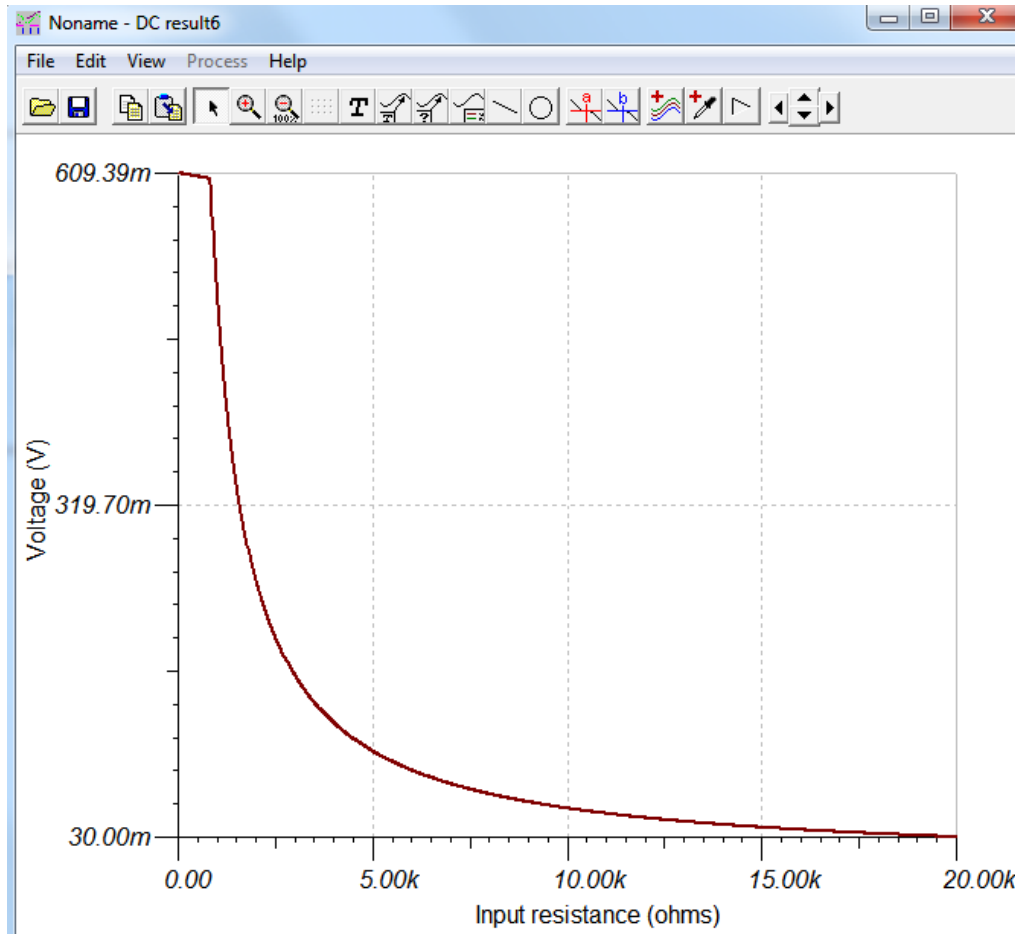
The 'DC Transfer Characteristic' dialog box is shown. It contains the following fields and options:

- Start value: 0 [ohm]
- End value: 20k [ohm]
- Number of points: 1000
- Input: R1 (selected from a dropdown menu)
- Enable hysteresis
- Buttons: OK, Cancel, Help



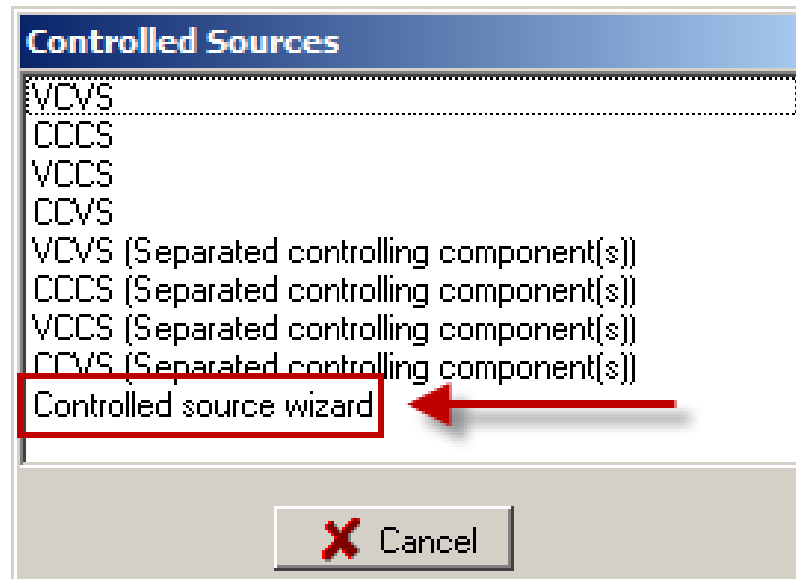
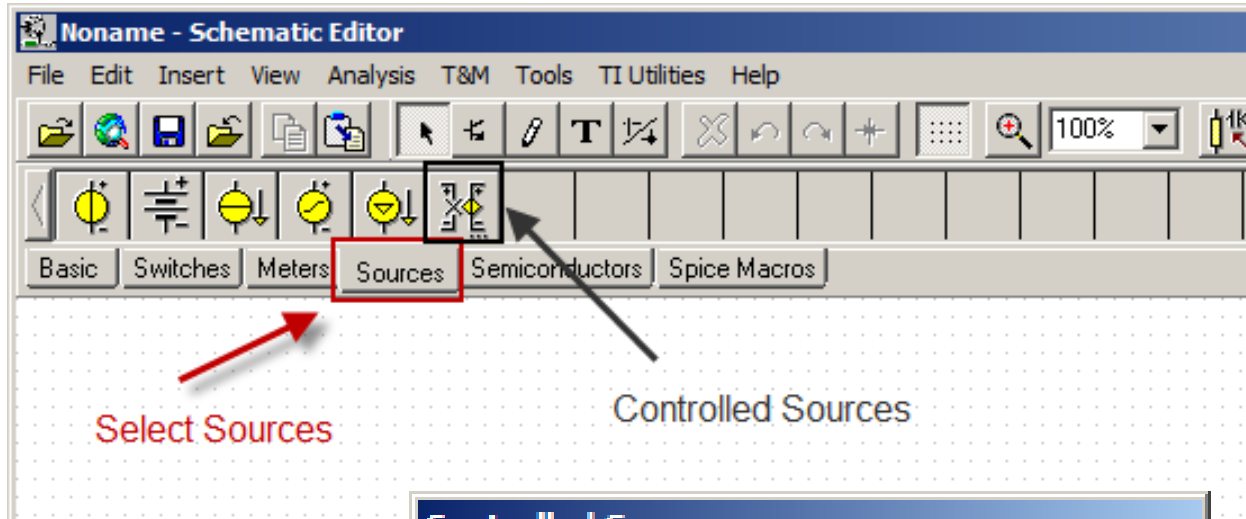
DC transfer for parameter stepping

The result of R1 stepping



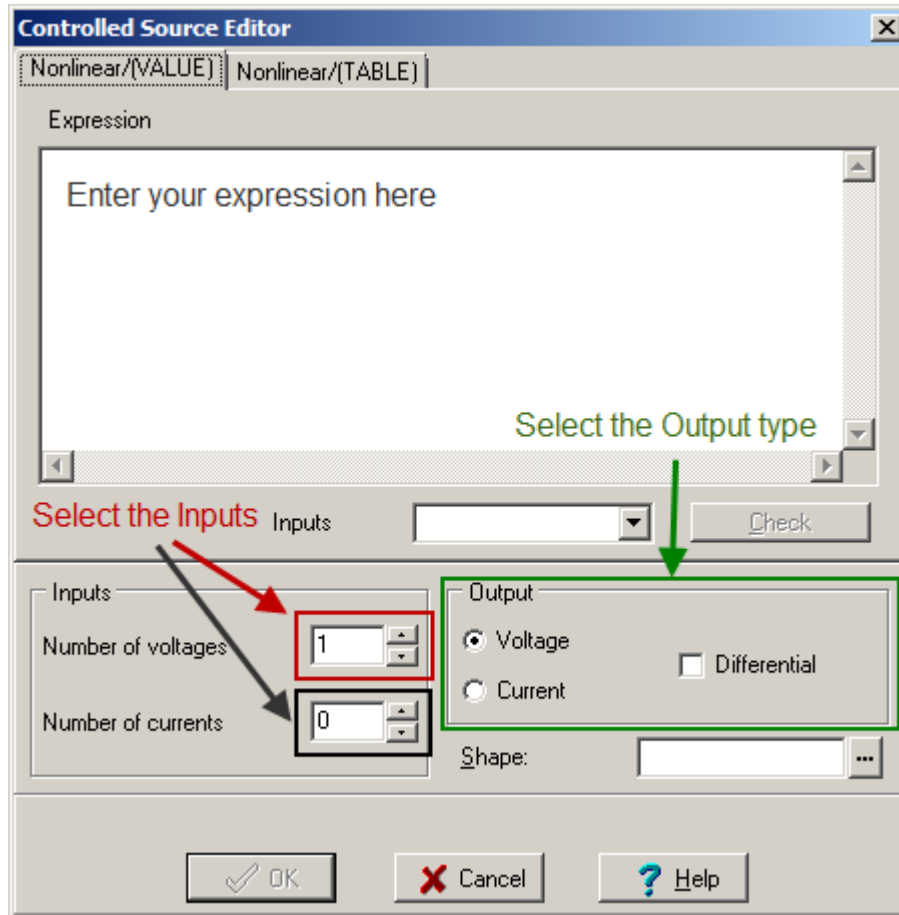


Controlled Source Wizard





Controlled Source Wizard



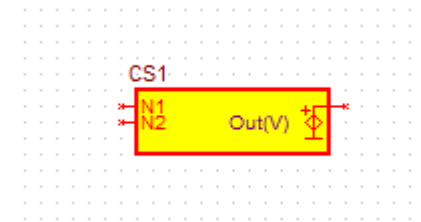
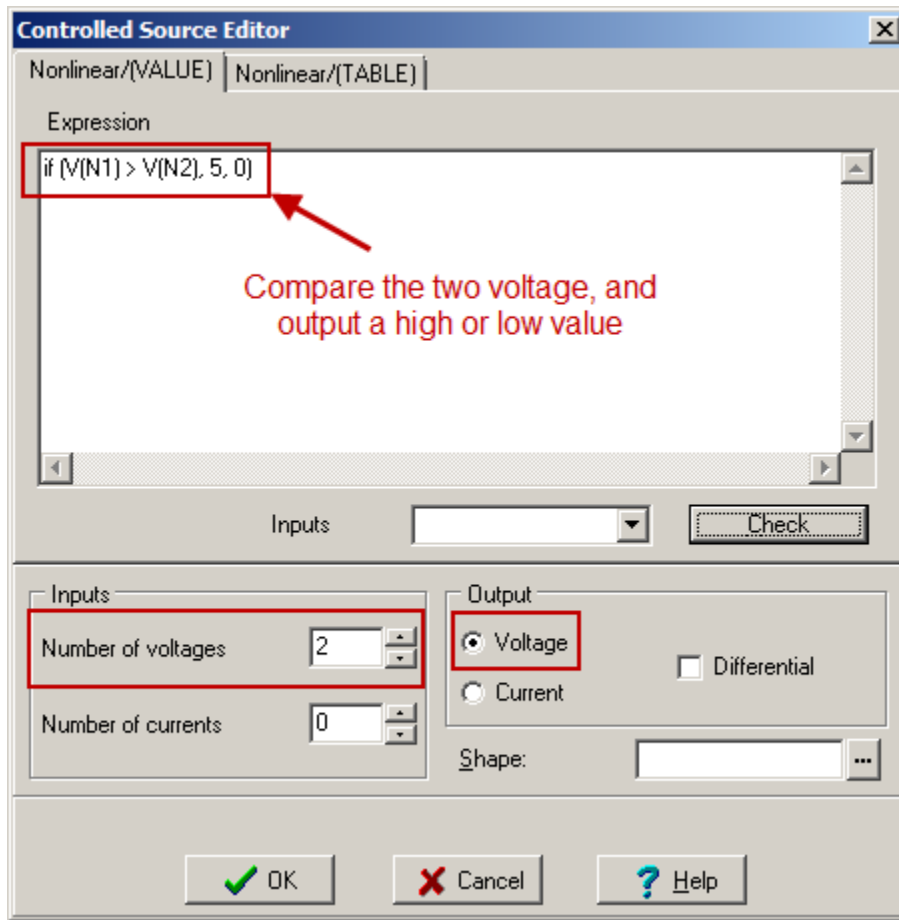
Expressions are entered in the dialog area. The expression will relate the inputs to the outputs.

You can use “if” statements, regular expressions, and functions in your expressions. Built in functions include $\sin(x)$, $\cos(x)$, $\text{sqr}(x)$, $\text{sqrt}(x)$, $\text{min}(x,y)$, $\text{max}(x,y)$ and others. A full list can be seen in the Help section for built-in functions.

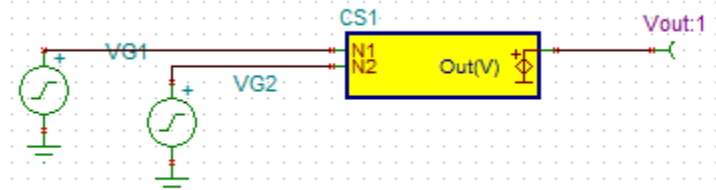
In the Table mode, the block uses input and output data pairs.



Controlled Source Wizard (Comparator)

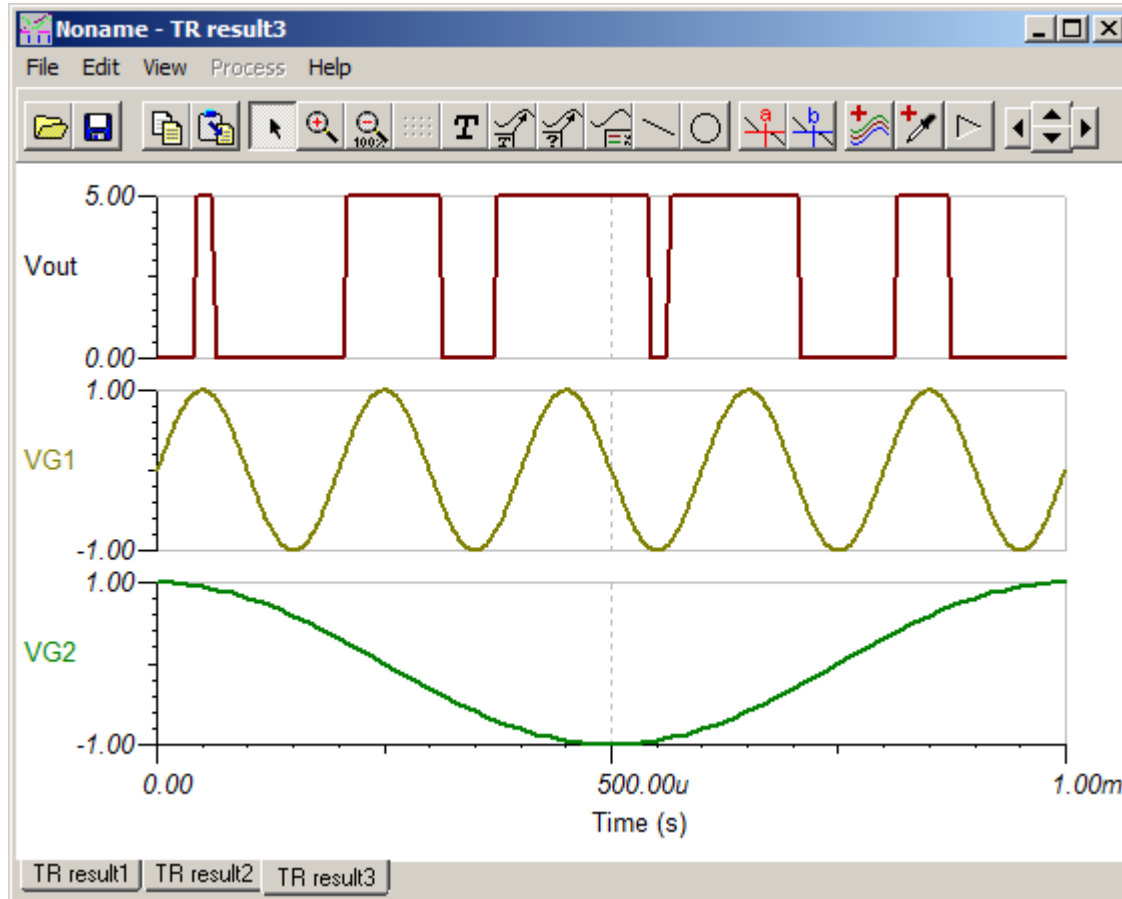


Place the controlled source and connect the two voltages and monitor the output.





Controlled Source Wizard (Comparator)

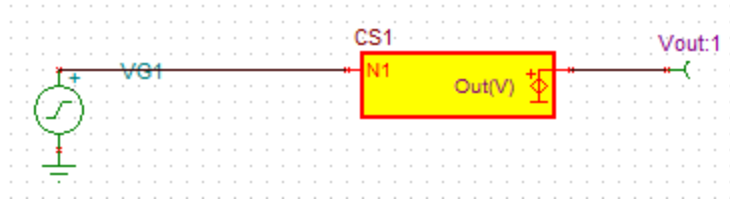
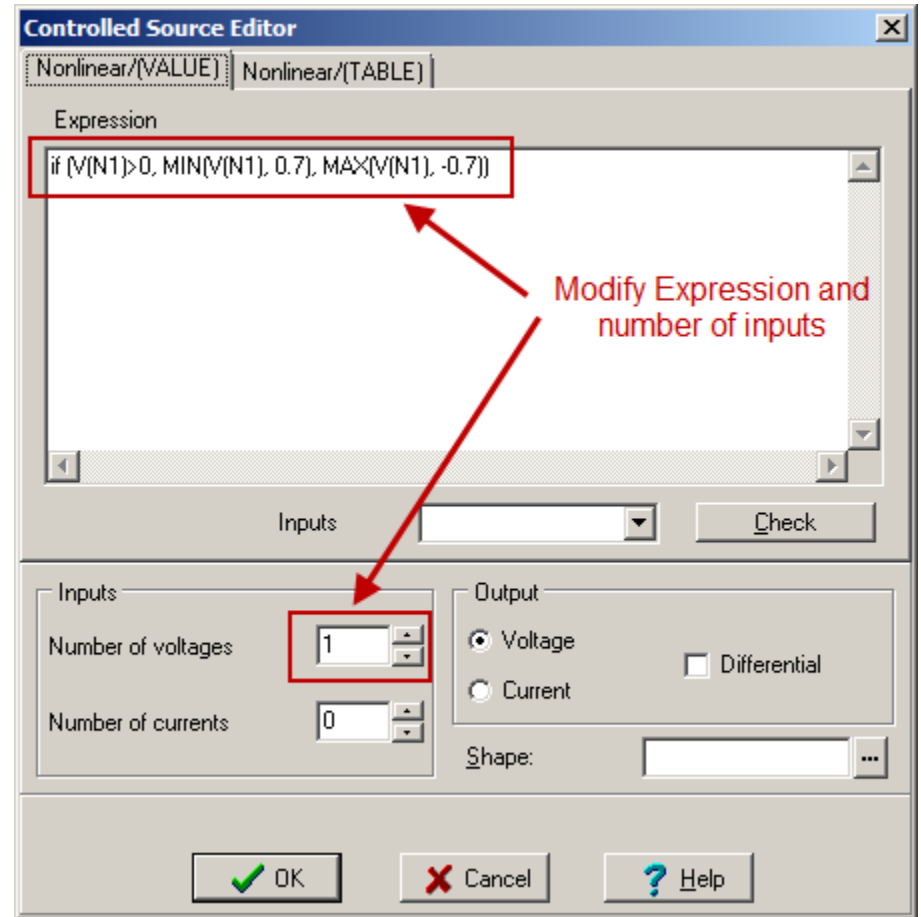
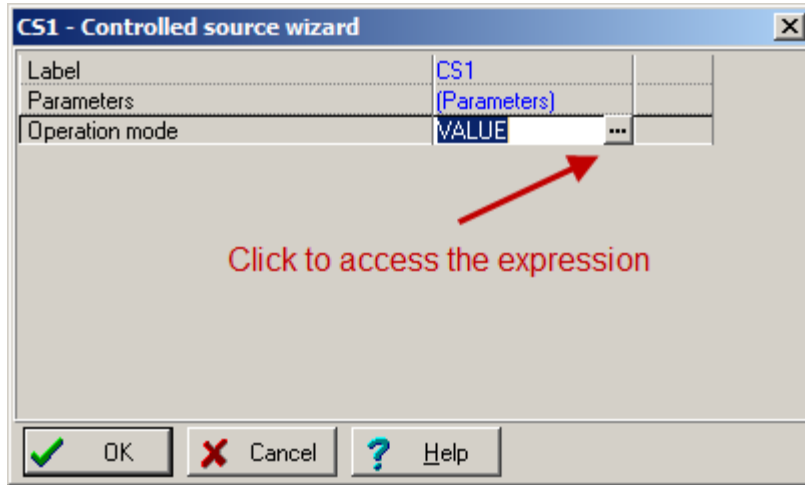


The output is the comparison of the two different frequency sine and cosine waves.

The comparator could be made with more than two inputs and you can operate on the inputs to customize the comparison further.

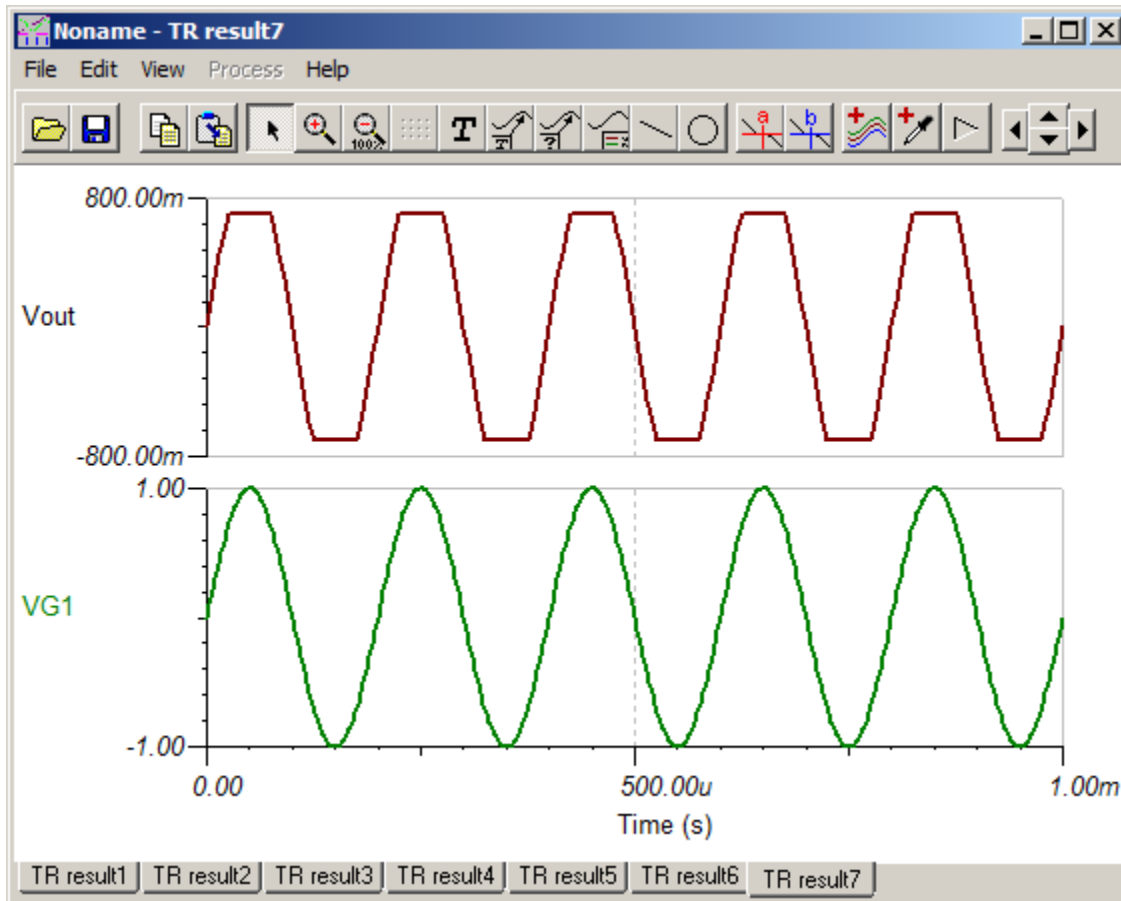


Controlled Source Wizard (Limiter)





Controlled Source Wizard (Limiter)

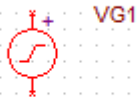


The voltage is now limited between 0.7 and -0.7.

Note that the voltage between the limits is not effected. It could be modified again by adding another controlling block or by use of a more complicated expression.



Piecewise Linear Source from file



VG1 - Voltage Generator

Label	VG1	
Parameters	(Parameters)	
DC Level [V]	0	<input type="checkbox"/>
Signal	Unit step	...
Internal resistance [Ohm]	0	<input type="checkbox"/>
ID state	Input	
Fault	None	

OK Cancel Help

Navigate to the file location and load it into the Signal Editor. This may take a few seconds for a large file. Click Test to see the waveform.

Signal Editor

Signal (t)

- New
- Open...
- Save
- Save As...
- Set Font...
- Test

Press the Test button to see the signal.

Line: 1 Col: 1

Test Menu

OK Cancel Help

Signal Editor

Signal (t)

Max: 15.066784

Min: -617.817085 | Max: 2.5m

3E-9	0.000286977831394169
5.47836994138525E-9	0.00056068355149972
9.47836994138525E-9	0.00106366301335871
1.14783699413852E-8	0.00133332972852408
1.34783699413852E-8	0.00161050369947947
1.54783699413852E-8	0.00189278556945435
1.94783699413853E-8	0.00246621073891576

Press the Test button to see the signal.

Line: 1 Col: 1

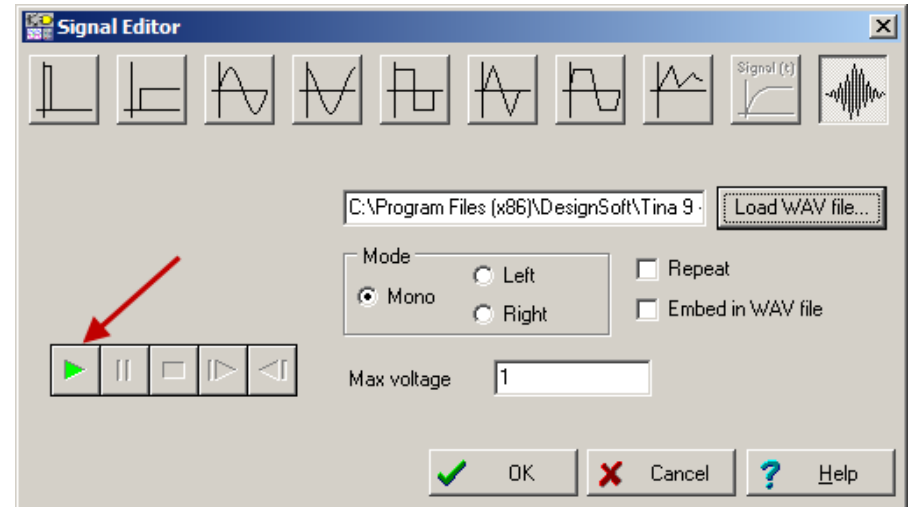
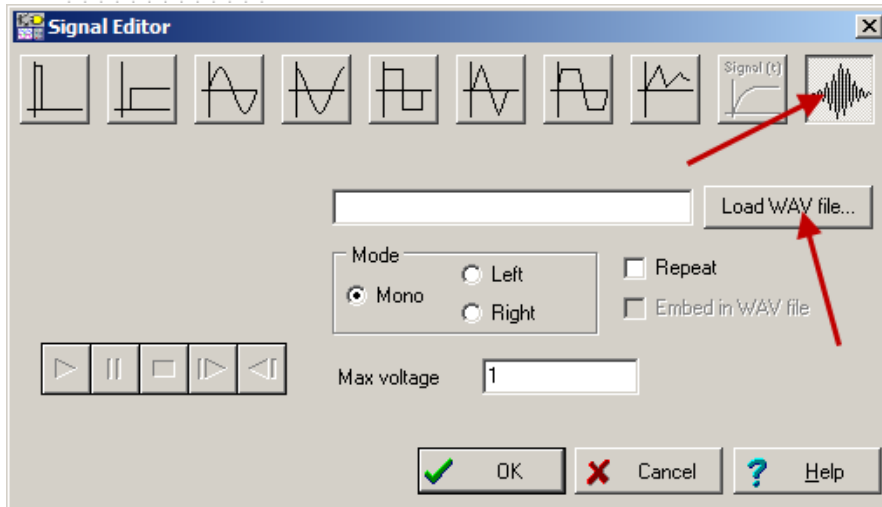
Test Menu

OK Cancel Help



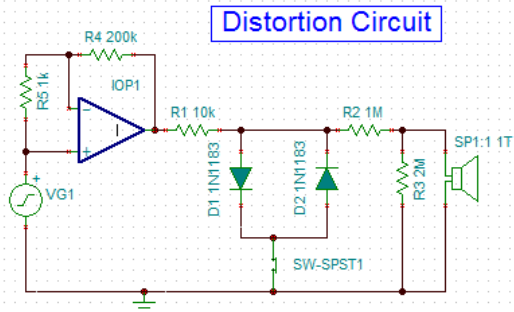
WAV file used as an input

Once the WAV file is selected, navigate to the location for the .wav file to be used. Once loaded, the play arrow will become green and you can test it if you wish.





WAV file used as an input - Example



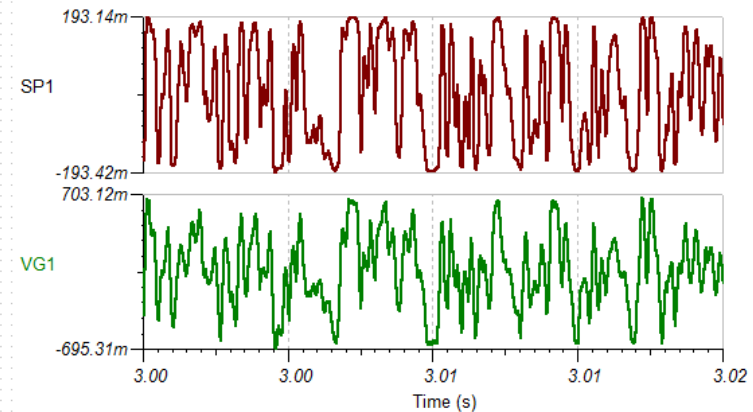
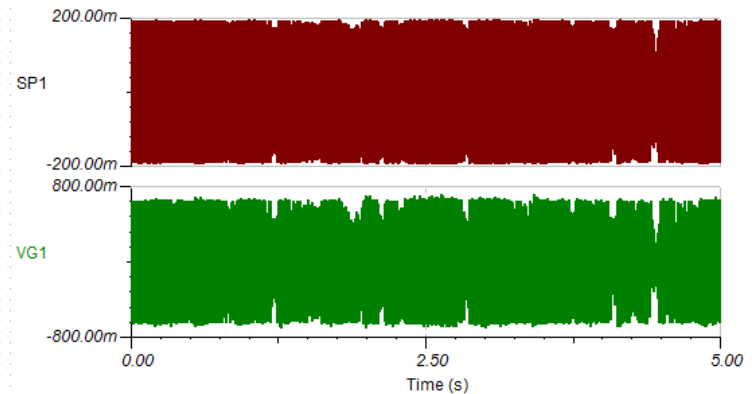
With the settings of this distortion circuit you can not only see the waveforms but also listen to the effect of distortion.

Press the TR button to start simulation and playback on the loudspeakers of your computer.

Press the A key on the keyboard or change the switch to disable or enable distortion.

You can also run a Transient analysis from the Analysis menu and display the waveforms. If you click on a curve you can also play it back by pressing the green arrow-shaped Play Sound button on the toolbar of the Diagram Window. This feature is especially useful when your computer is not fast enough for interactive playback.

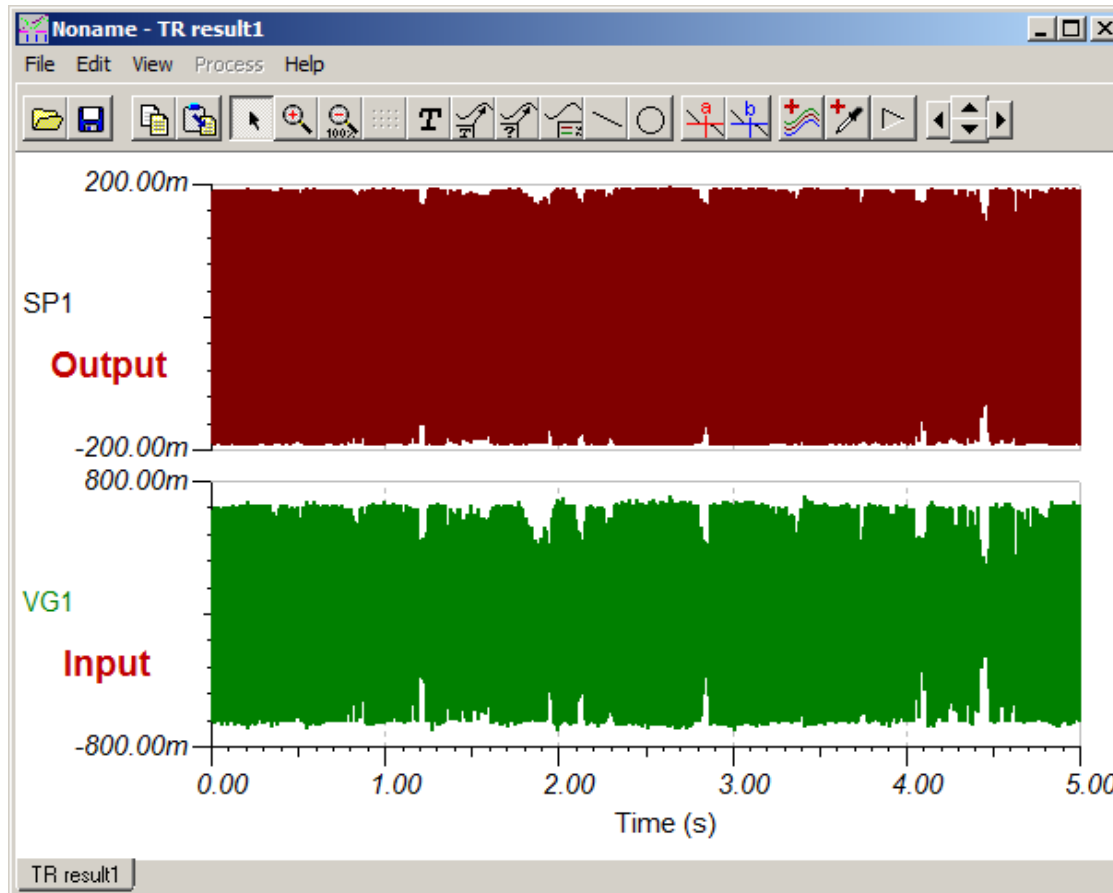
Run Analysis/Transient...





WAV file used as an input - Example

Select Analysis → Transient
Run the simulation for a few seconds.



Both the input and the output file can be played through the speakers built into the PC.

You can hear the changes from the circuit (in this case added distortion), make modifications, and listen to the results.



WAV file used as an input - Example

The WAV file is the time domain input and the circuit operates on the WAV file to produce the desired output.

This example can be found in the Examples directory under the sub directory WAV file examples.

Limitation: TINA-TI v9 cannot play the file in real time using the TR button or the keyboard button interfaces, as the instructions on the schematic describe.

NOTE: In order to play back the modified waveform, you must select it by right clicking on the waveform in the waveform viewer. The arrow for playback will become green and you can then hear the resulting waveform.



Where to find Models for TI Products



Where Can I find Models?

- TI Product Folders
 - Models that have been released are placed on TI's website in the product folders
- ESP
- TI Spice Rack
 - <http://www.ti.com/spicerack/>
 - Updated regularly with new model releases
- WEBENCH[®] Design Center home page (<http://www.ti.com/webench>)



Models in Product Folder

TPS40210

(ACTIVE) Wide Input Range Current Mode Boost Controller

★★★★★ 5 out of 5 2 reviews | Add your review and give us feedback

PitchPak

Alert me about changes

Description & Features

Sample & Buy

Technical Documents

Tools & Software

Support & Community

Circuit Design & Simulation

Models

Design Kits & Evaluation Modules

Reference Designs

Software & Development Tools

TI Design Network

Datasheet

PDF > 4.5-V to 52-V INPUT CURRENT-MODE BOOST CONTROLLER. (Rev. E)
(PDF, 1274 KB) 12 Oct 2011
View All Technical Documents

Description

The TPS40210 and TPS40211 are wide-input voltage (4.5 V to 52 V), non-synchronous boost controllers. They are suitable for topologies which require a grounded source N-channel FET including boost, flyback, SEPIC and various LED Driver applications. The device features include programmable soft start, overcurrent protection with automatic retry and programmable oscillator frequency. Current mode control provides improved transient response and simplified loop compensation. The main difference between the two parts is the reference voltage to which the error amplifier regulates the FB pin.

Show More

Features

Featured Tools and Software

- > TPS40210 PSpice Average and Transient Models (Rev. D) (Simulation Models)
- > Development Boards/EVMs (Development Boards/EVMs)
- > Isolated Flyback 30V@0.4A for Alarm Systems (Hardware Reference Designs)

View All

WEBENCH® TPS40210

	Min	Max	Range
Vin	8.00	14.00	V 4.50 – 52.00V
Vout	24.50	V	5.00 – 300.00V
Iout	1.50	A	0.2000A

Example product folder for the TPS40210. Clicking on “Circuit Design & Simulation” under “Tools & Software” takes you to the section in the product folder where the models are located.

Different model types and Reference Designs may be available.



Models in Product Folder

Tools & Software

Circuit Design & Simulation

WEBENCH® TPS40210

	Min	Max	Range
Vin	8.00 V	14.00 V	4.50 – 52.00V
Vout	24.50 V		5.00 – 300.00V
Iout	1.50 A		≤ 30.00A
Ambient Temp	30 °C		≤ 100°C

Lowest BOM Cost | Smallest Footprint | Highest Efficiency

Footprint: 659 | BOM Cost: \$6.04 | Efficiency: 91%

Open Design

659mm²

What are WEBENCH® tools?

Models (7)

Title	Category	Type	Size (KB)	Date	Views
TPS40210 PSpice Average and Transient Models (Rev. D)	PSpice Model	ZIP	216 KB	21 Aug 2009	454 views
TPS40210 TINA-TI Average Reference Design (Rev. B)	TINA-TI Reference Design	TSC	412 KB	21 Aug 2009	243 views
TPS40210 TINA-TI Average Spice Model (Rev. A)	TINA-TI Spice Model	ZIP	9 KB	21 Aug 2009	263 views
TPS40210 TINA-TI Transient Start-up Reference Design	TINA-TI Reference Design	(Multiple Files)		03 Dec 2008	235 views
TPS40210 TINA-TI Transient Start-up Spice Model	TINA-TI Spice Model	(Multiple Files)		03 Dec 2008	349 views
TPS40210 TINA-TI Transient Steady State Reference Design (Rev. B)	TINA-TI Reference Design	(Multiple Files)		03 Dec 2008	252 views
TPS40210 TINA-TI Transient Steady State Spice Model	TINA-TI Spice Model	(Multiple Files)		03 Dec 2008	317 views

Show Less

Design Kits & Evaluation Modules (2)

Name	Part#	Type
Development Boards/EVMs	TPS40210EVM	Development Boards/EVMs
Power Stage Designer of Most Commonly Used Switchmode Power Supplies	POWERSTAGE-DESIGNER	Analysis Software

Models (7)

Title	Category	Type
TPS40210 PSpice Average and Transient Models (Rev. D)	PSpice Model	ZIP
TPS40210 TINA-TI Average Reference Design (Rev. B)	TINA-TI Reference Design	TSC

Show More

Reference Designs (101)

Description	Part#	Company
100VAC-265VAC 100W High Bay LED Lighting (100V @ 1.2A)	PMP4862.1	Texas Instruments
10Vdc-100Vdc Input, 12V/1A SEPIC converter	PMP6545	Texas Instruments
120VAC Input to 5V/1.25W Output, Ultra Compact Isolated SEPIC	PMP6711	Texas Instruments

Show More

Design Kits & Evaluation Modules (2)

Name	Part#	Type
Development Boards/EVMs	TPS40210EVM	Development Boards/EVMs



Models at <http://www.ti.com/spicerack>

www.ti.com/spicerack Contains all available SPICE Models



[Samples & Purchase Cart](#) | [Contact Us](#) | [TI Worldwide: United States](#)

[Products](#)
[Applications](#)
[Design Support](#)
[Sample & Buy](#)

[All Searches](#)

TI Home > [WEBENCH® Design Center](#) > PSpice, TINA-TI Reference Designs and Spice Models

WEBENCH® Design Center

PSpice, TINA-TI Reference Designs and Spice Models

Part Number	Available in SwitcherPro™?	PSpice	TINA-TI Reference Design	TINA-TI Spice Model
ACF2101		PSpice Model		
ADS6128			TINA-TI Transient Reference Design	TINA-TI Transient Spice Model
ADS6129			TINA-TI Transient Reference Design	TINA-TI Transient Spice Model
ADS6148			TINA-TI Transient Reference Design	TINA-TI Transient Spice Model
TPS40192	In SwitcherPro	PSpice Average Model PSpice Transient Model	TINA-TI Average Reference Design (Rev. A)	TINA-TI Average Spice Model TINA-TI Transient Spice Model
TPS40193	In SwitcherPro	PSpice Model	TINA-TI Average Reference Design (Rev. A)	TINA-TI Transient Spice Model TINA-TI Average Spice Model
TPS40195	In SwitcherPro	PSpice Average and Transient Model (Rev. A)	TINA-TI Average Reference Design TINA-TI Transient Reference Design (Rev. B)	TINA-TI Transient Spice Model TINA-TI Average Spice Model
TPS40200	In SwitcherPro	PSpice Average Model (Rev. C) PSpice Transient Model	TINA-TI AC Analysis Reference Design (Rev. C) TINA-TI Transient -5.2V Buck/Boost Reference Design TINA-TI Transient Buck Reference Design	TINA-TI Average Spice Model (Rev. A) TINA-TI Transient Spice Model
TPS40210	In SwitcherPro	PSpice Average and Transient Models (Rev. D)	TINA-TI Transient Start-up Reference Design TINA-TI Average Reference Design (Rev. B) TINA-TI Transient Steady State Reference Design (Rev. B)	TINA-TI Transient Start-up Spice Model TINA-TI Transient Steady State Spice Model TINA-TI Average Spice Model (Rev. A)
TPS40211		PSpice Average Model (Rev. A)	TINA-TI Transient Start-up	TINA-TI Average Spice Model



Power Models Demystified

- All power models are encrypted with supported simulators.
- Supported Simulators:
 - PSPICE
 - Encrypted models require version 15.7 or newer.
 - Unencrypted models can be used in most of the older versions.
 - TINA-TI: All of the .tsm and .tsc files on the web are encrypted and can be run with TINA-TI 7/9 or TINA Industrial 7/8/9.



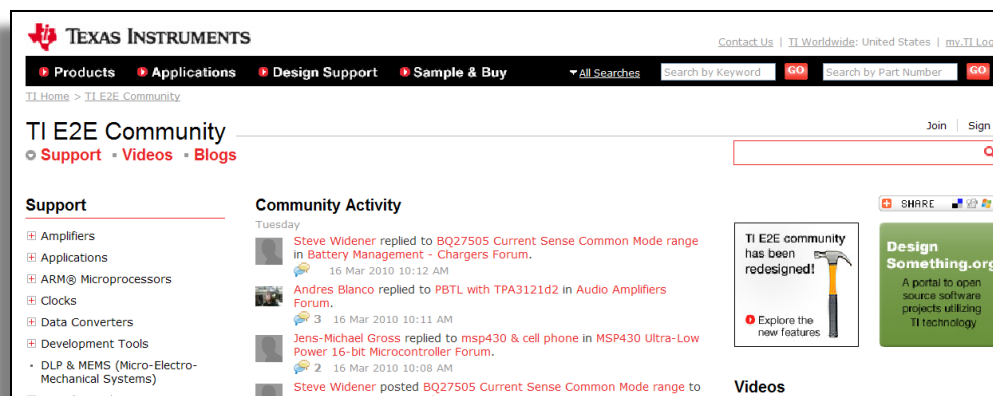
Additional Resources



TI E2E™ Community

Ask questions, share knowledge, explore ideas, and help solve problems with fellow engineers on the TI E2E™ Community

- Access to a rich library of technical content
- Facilitates connections with like-minded engineers
- Takes personalization to a new level
- Advanced search functionality
- Recognizes the involvement of active members



<http://e2e.ti.com>



TI E2E™ Community – Questions on Tina-TI

Products Applications Tools & Software **Support & Community** Sample & Buy About TI

TI E2E™ Support Community
engineer.to.engineer, solving problems

Contact Technical Support
Asia
Europe, Middle East, Africa

E2E Support Forums

- Amplifiers
- Applications
- ARM® & DSP Microprocessors
- Clocks & Timers
- Data Converters
- Development Tools**
- DLP® & MEMS
- Embedded Software
- Interface
- Logic
- Low Power RF & Wireless Connections
- Microcontrollers
- OMAP™ Mobile Processors
- Power Management
- RF & Digital Radios
- Other Analog & Mixed-Signal ICs

TEXAS INSTRUMENTS

Products Applications Design Support Sample & Buy All Searches

TI Home > TI E2E Community > Support Forums > Development Tools

TI E2E™ Community

Support Forums Videos Blogs Groups

WEBENCH® Design Center

Code Composer Studio
TI C/C++ Compiler
MAVRK Toolbox

Show All E2E Forums

WEBENCH® Design Center

Welcome to the WEBENCH® Design Center Section of the TI E2E Support Community. Ask questions, share knowledge, explore ideas, and help solve problems with fellow engineers. To post a question, click on the forum tab then "New Post".

Home Announcements Forums Files Design Notes **New Post**

Browse Forums

Forum	Posts	Last Post
Power and Lighting Tools Forum ☆ Support for Power and Lighting designs, using the WEBENCH Processor Power Architect, FPGA Power Architect, Power Designer (single supply), Power Architect (multi supply), LED Designer, LED Architect, and SwitcherPro™ Power Supply Designer tools.	241	Yesterday
Signal Chain/Signal Path Tools Forum ☆ This forum is for support of Signal Chain (analog signal processing, clocking, data communication and integrity, etc.) designs, using WEBENCH Sensor AFE, Sensor Designer, Active Filter Designer, Amplifier Designer, EasyPLL and FilterPro™ tools.	41	2012/4/18
Simulation and Models Forum ☆ This forum is for support on simulation, modeling, and TINA-TI. Learn more about TINA-TI at www.ti.com/tina-ti. To view PSpice and TINA-TI model availability, go to www.ti.com/spicereack	292	Today
Tools - Other Forum ☆ General questions and support for other tools in the WEBENCH® Design Center: selection tools (Power Quick Search, Interface Quick Search, etc.), legacy design tools (SWIFT-SW, TPS40K-SW, etc.) and other design products (Power Stage Designer, etc.).	29	2012/4/21

<http://e2e.ti.com>

Development Tools

WEBENCH Design Center



Links

- Links to WEBENCH[®] Design Center Products:
 - www.ti.com/webench
 - www.ti.com/tina-ti
 - www.ti.com/spicerack
 - [Sensor AFE and Sensor Interface](#)
 - [Active Filter Designers](#)
 - [Clock Tree Builder](#)



Questions?