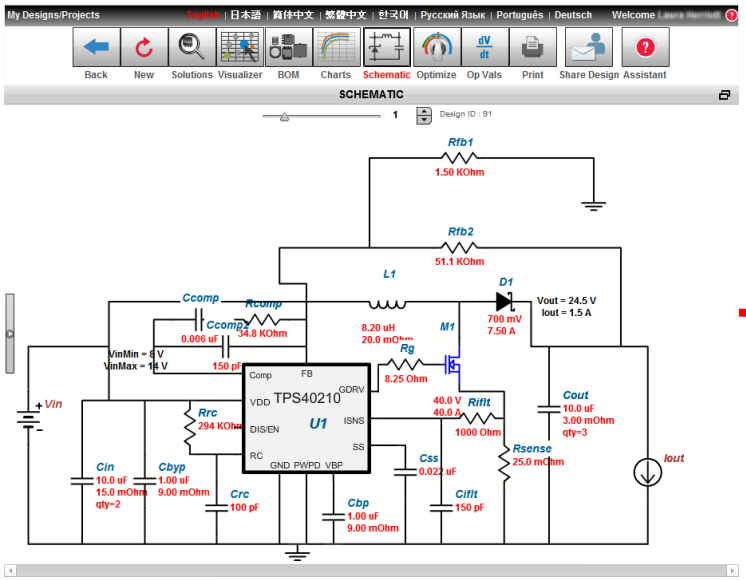
WEBENCH® Design Center

TINA-TI 9: A New Simulation Solution

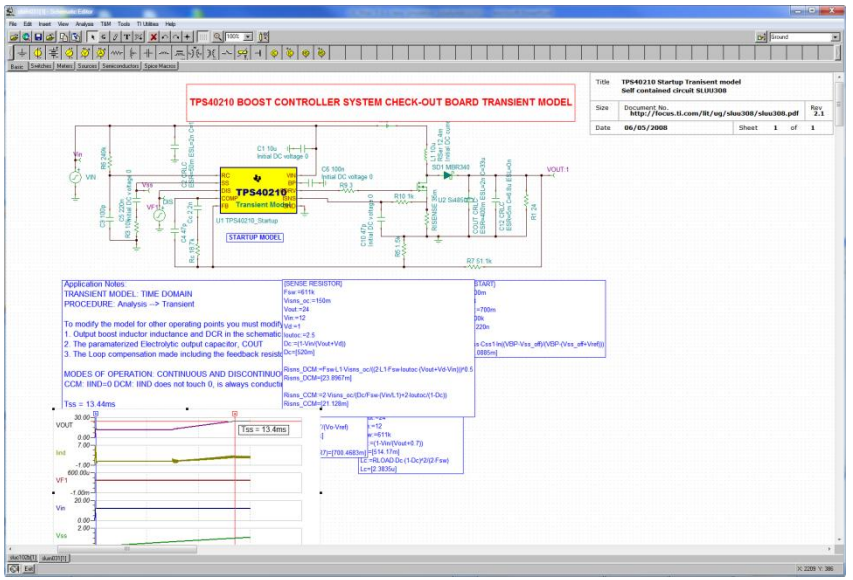
TINA-TI™ and the Pro™ Tools

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)

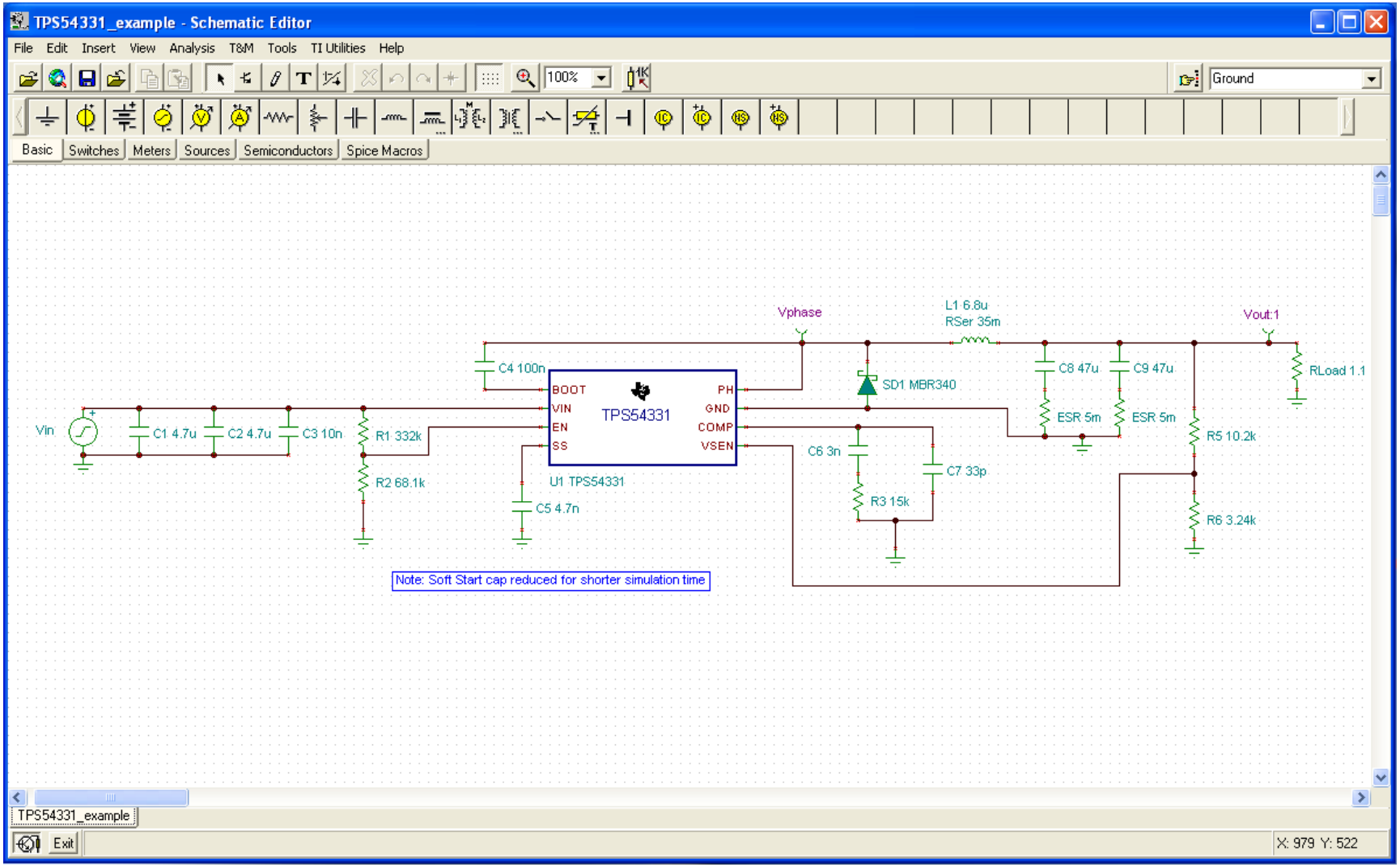
- Multi-core processor support.
- Faster single core simulation.
- Available in English, Chinese, Japanese, and Russian.
- Initial Condition and Nodeset Components.
- Linear and nonlinear controlled sources:
  - VCVS, CCVS, VCCS, CCCS
  - Controlled Source Wizard
- WAV files as stimulus (signal source).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

Models in TINA-TI 9

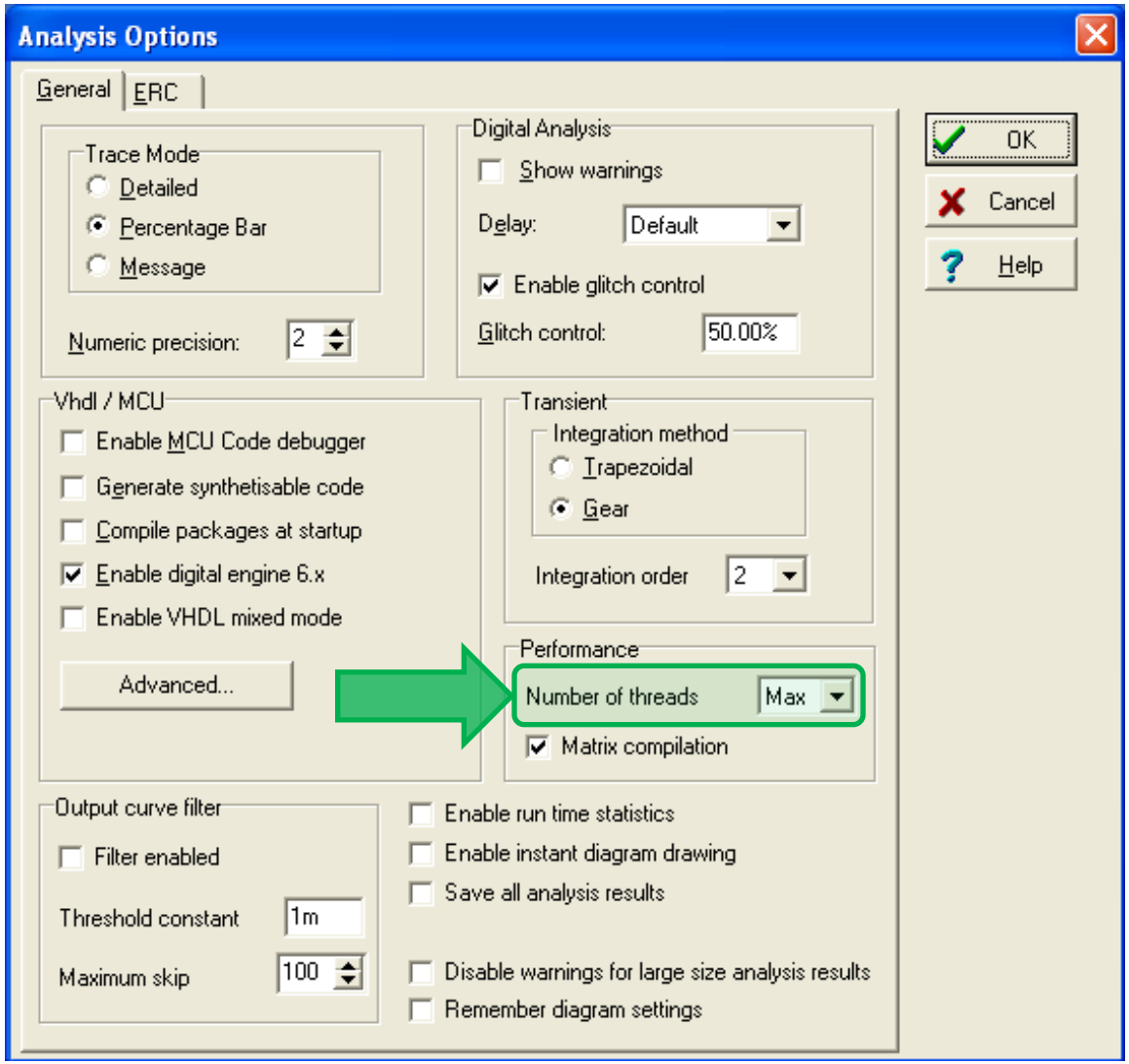
- TINA-TI 9 will ship with over 1000 models and reference designs.
  - Over 400 power models with reference designs
  - Over 600 signal chain models available
- New models and reference designs are added on an ongoing basis (below parts added in past quarter):

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



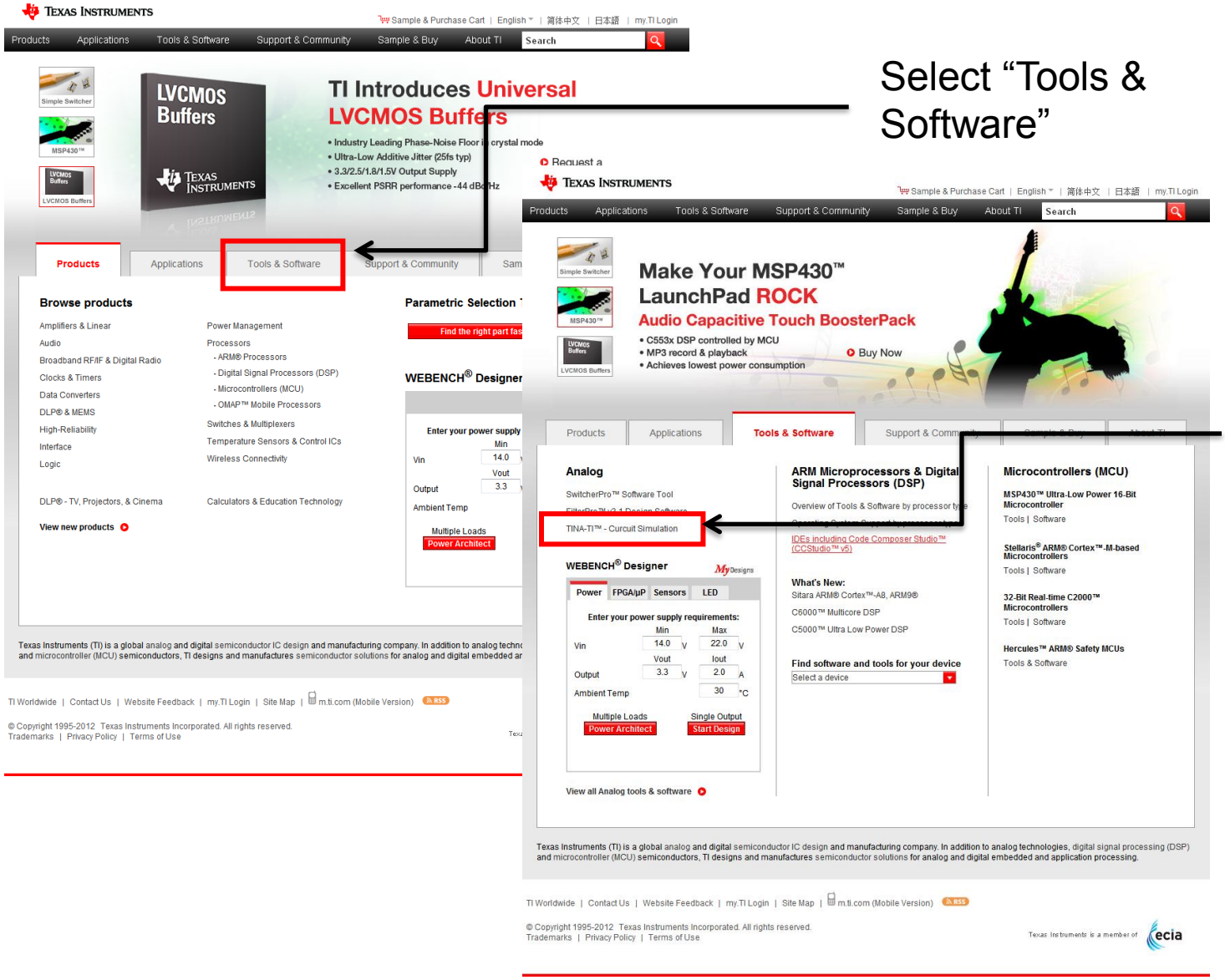
Simulation Speed: TINA-TI 7 vs TINA-TI 9

- 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

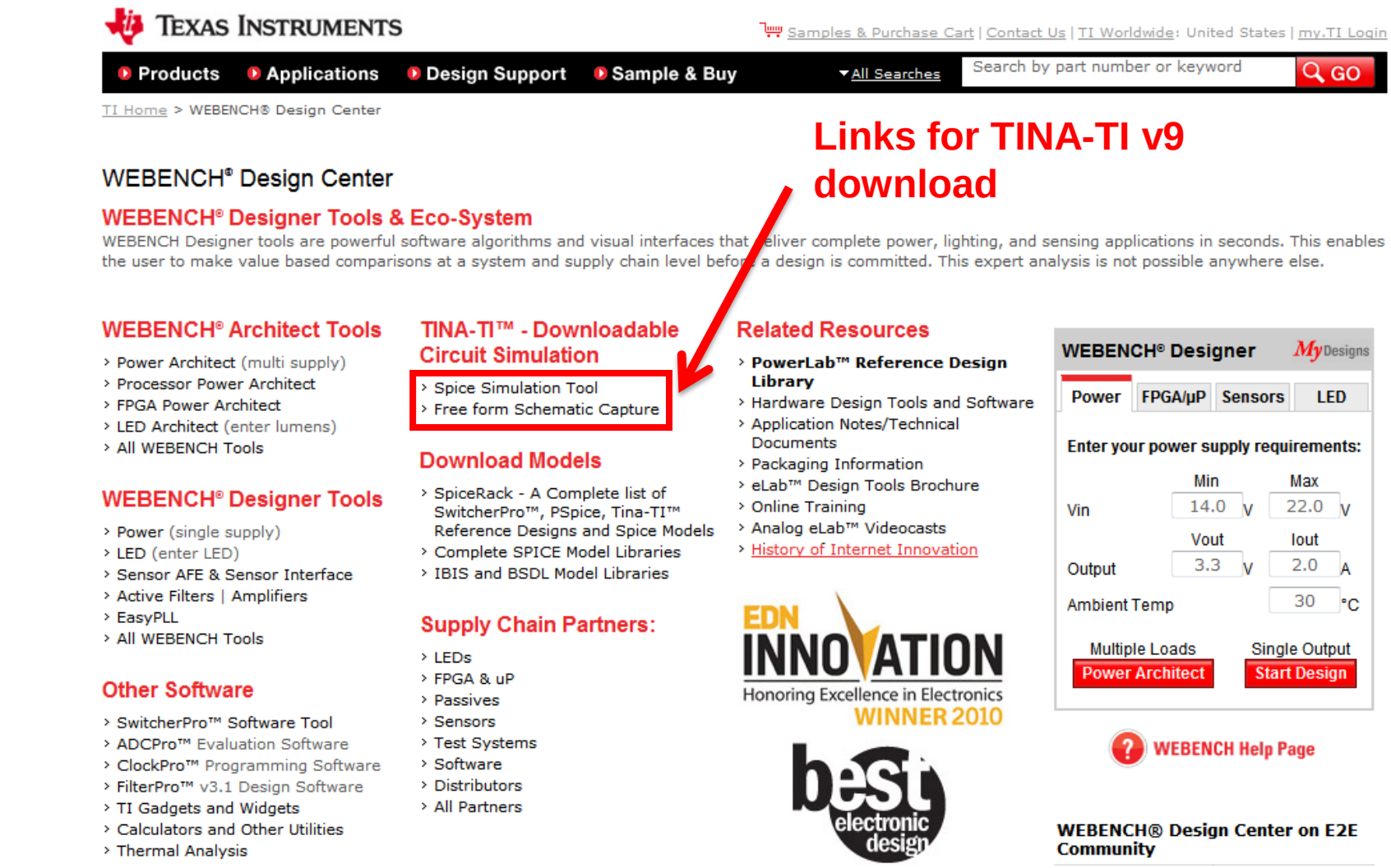
Installing TINA-TI 9

Downloading TINA-TI™

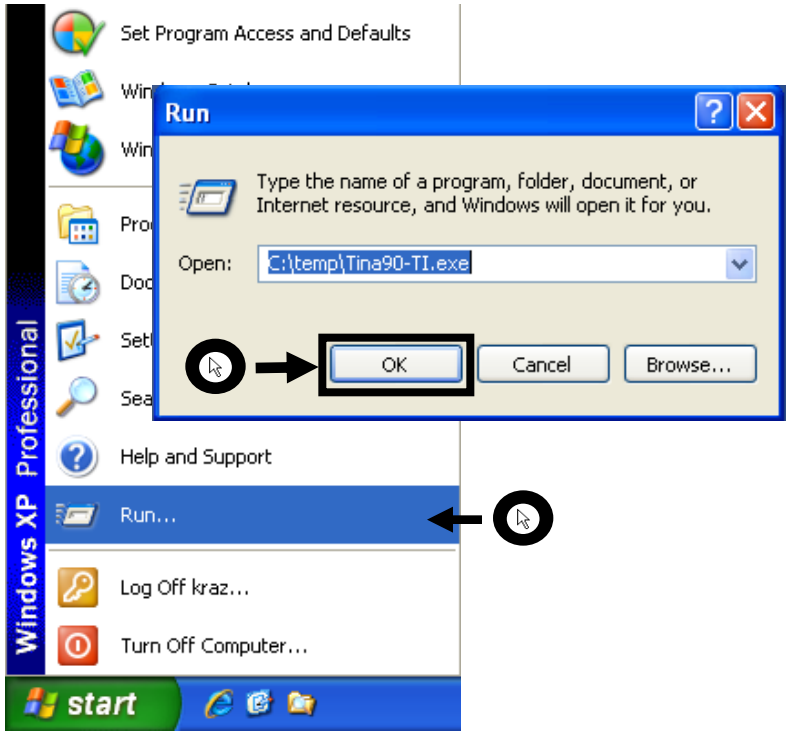


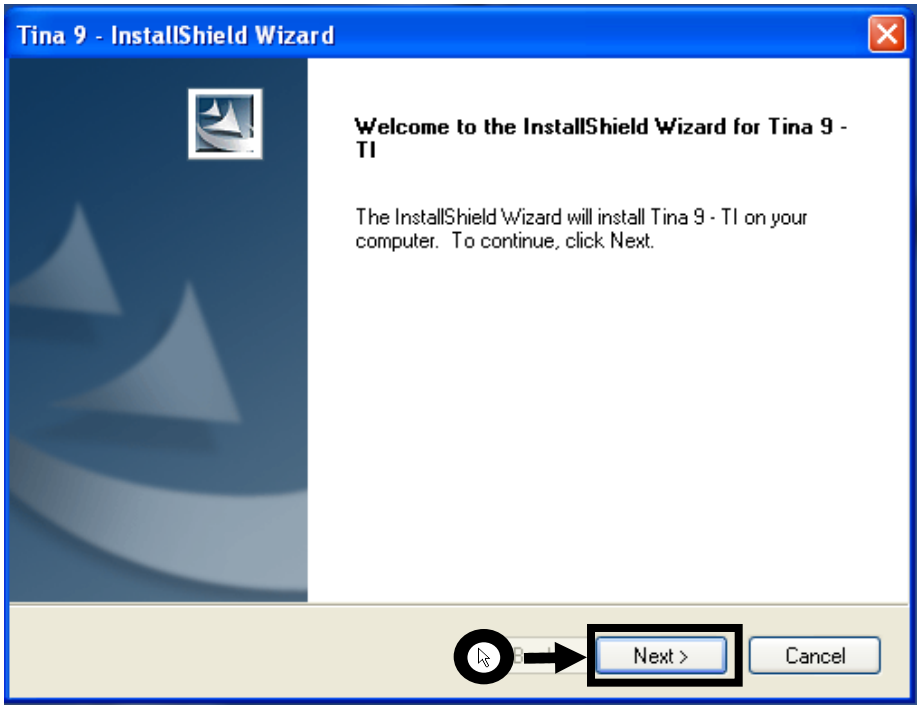
Downloading TINA-TI™

www.ti.com/webench

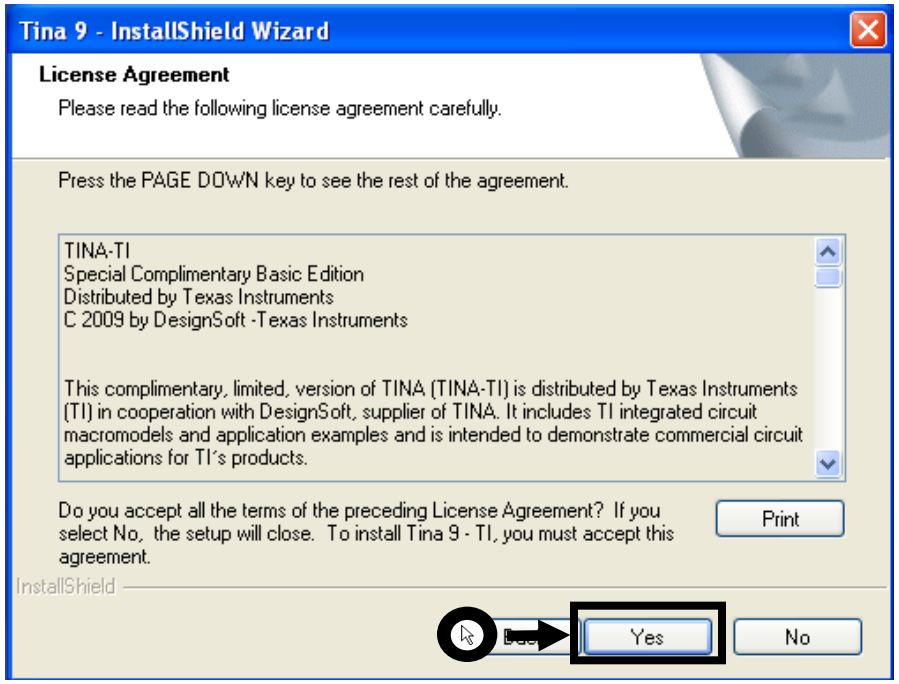


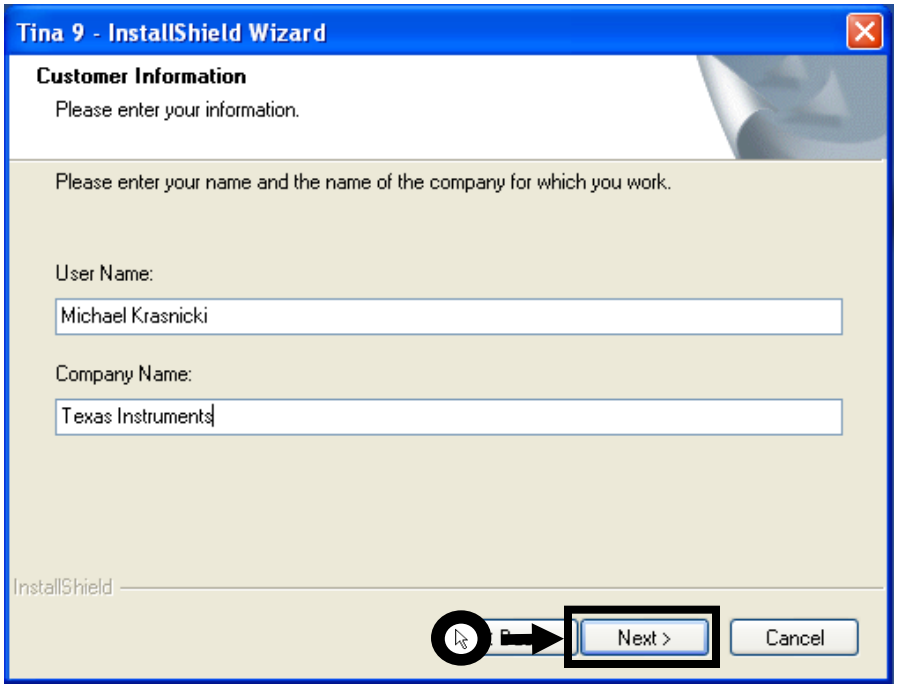
Installing TINA-TI 9



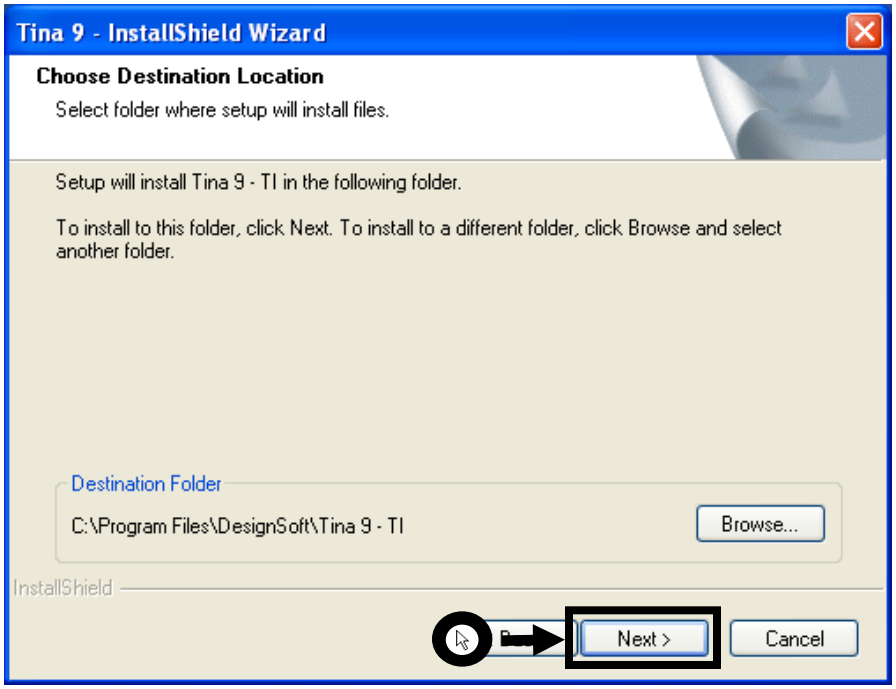


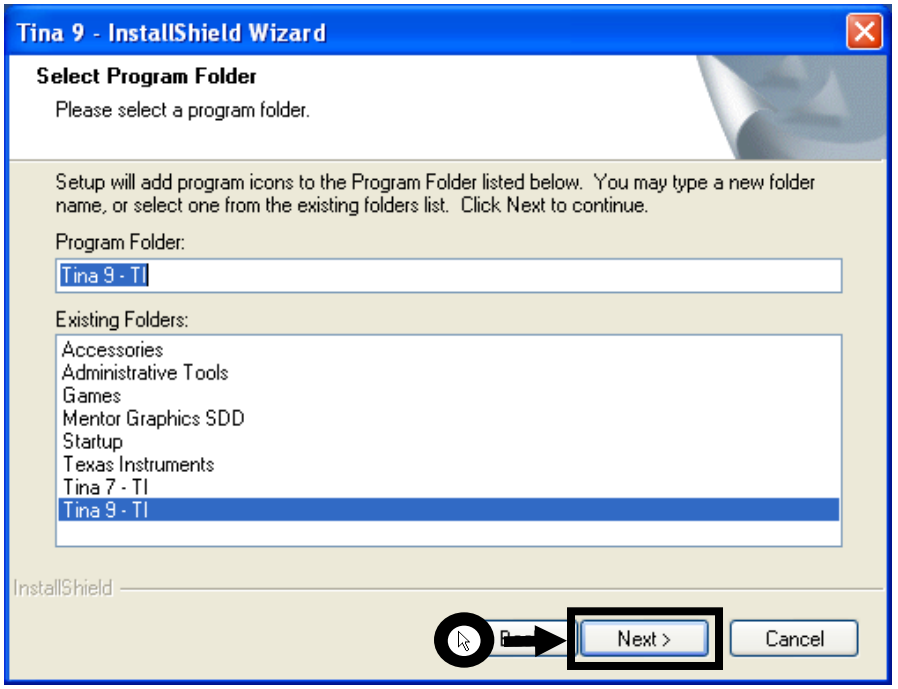
Installing TINA-TI 9



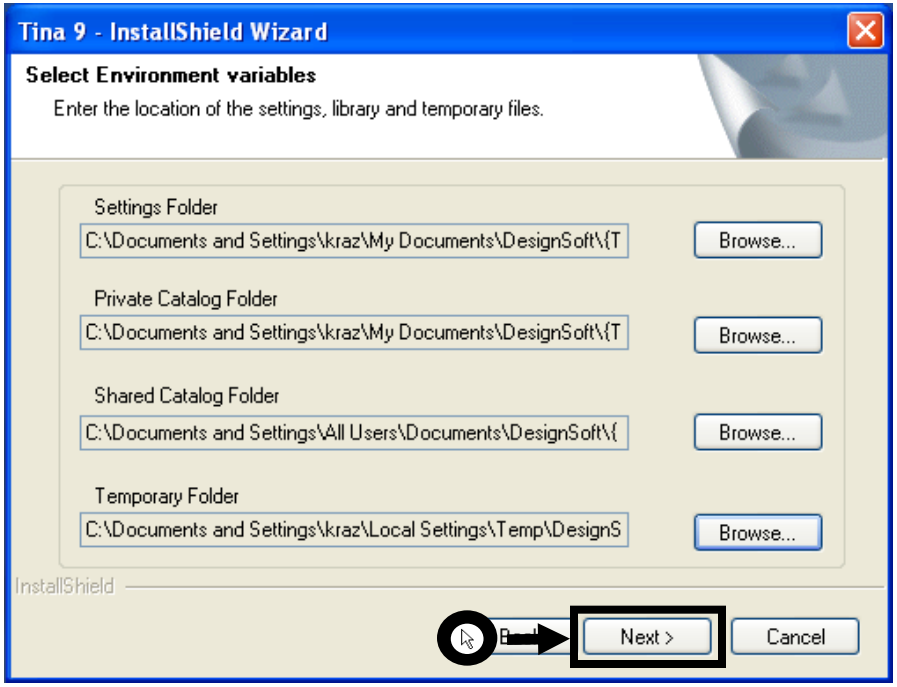


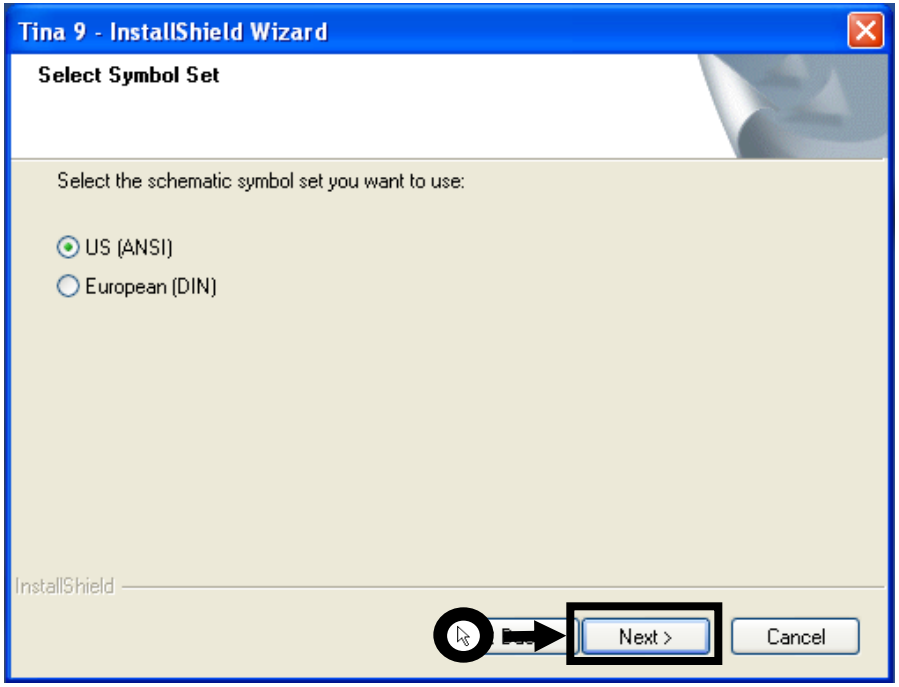
Installing TINA-TI 9



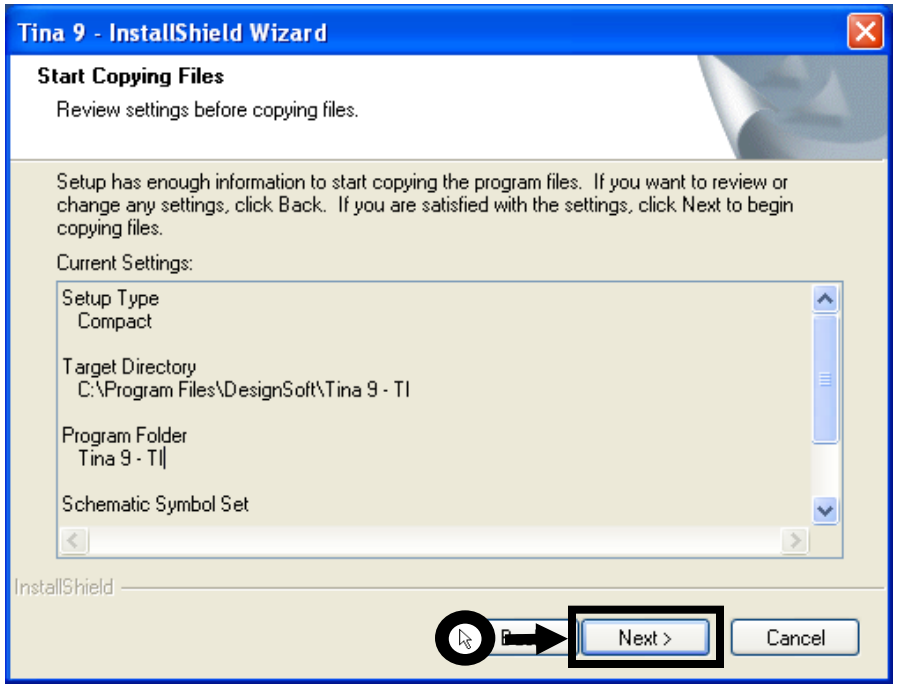


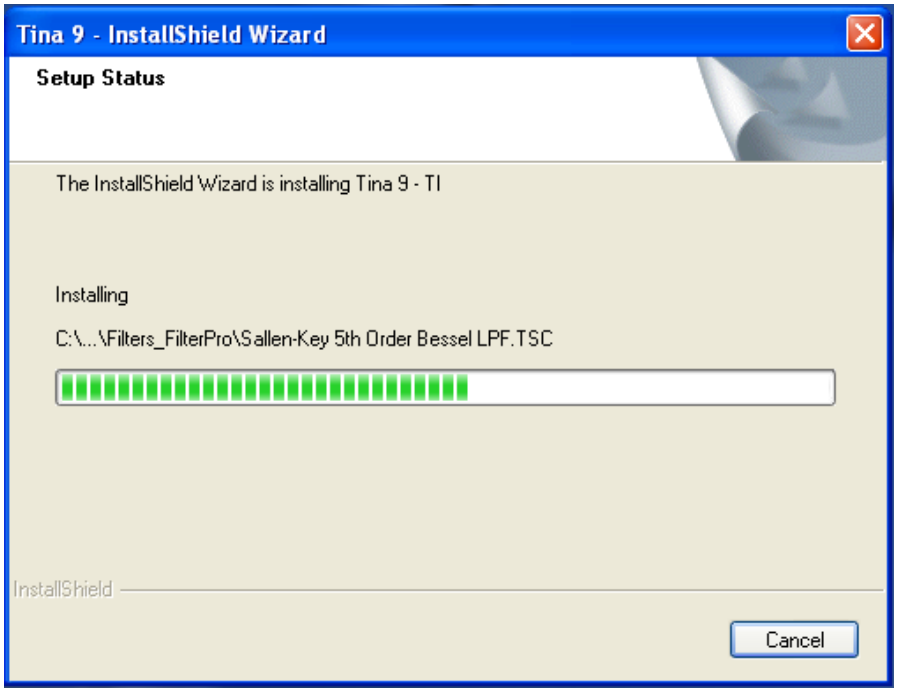
Installing TINA-TI 9



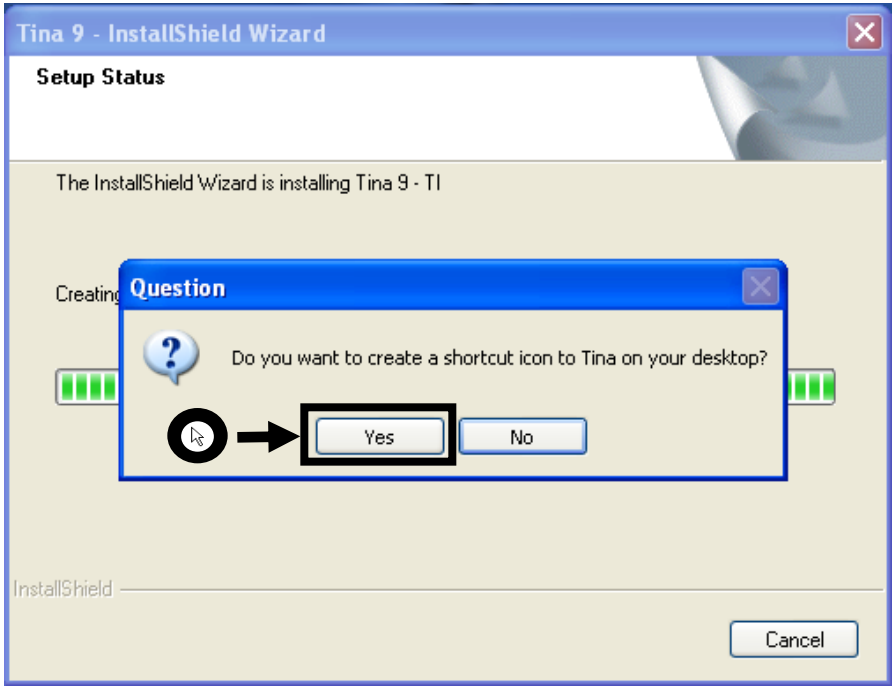


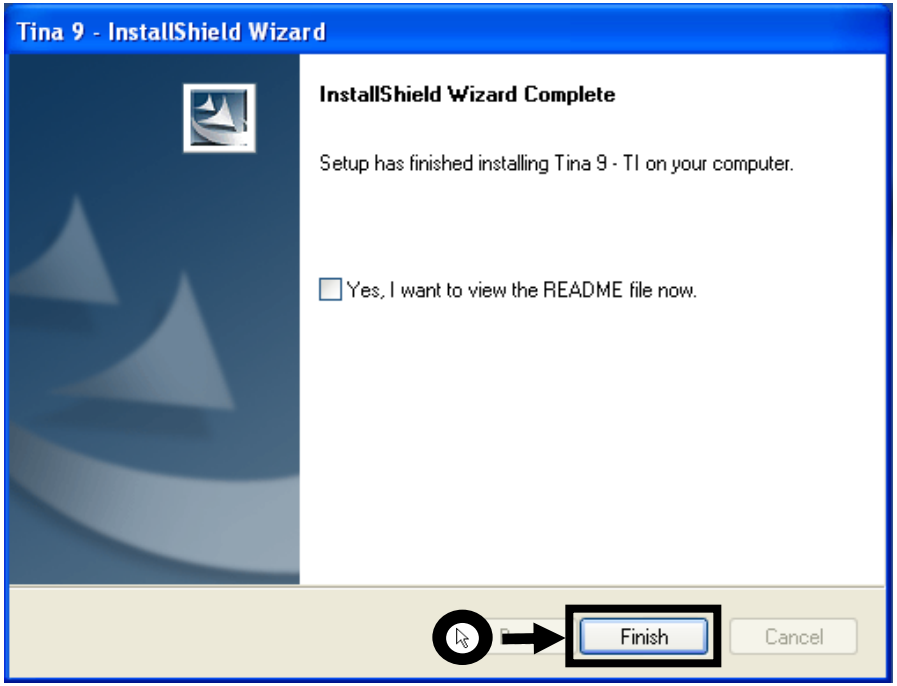
Installing TINA-TI 9



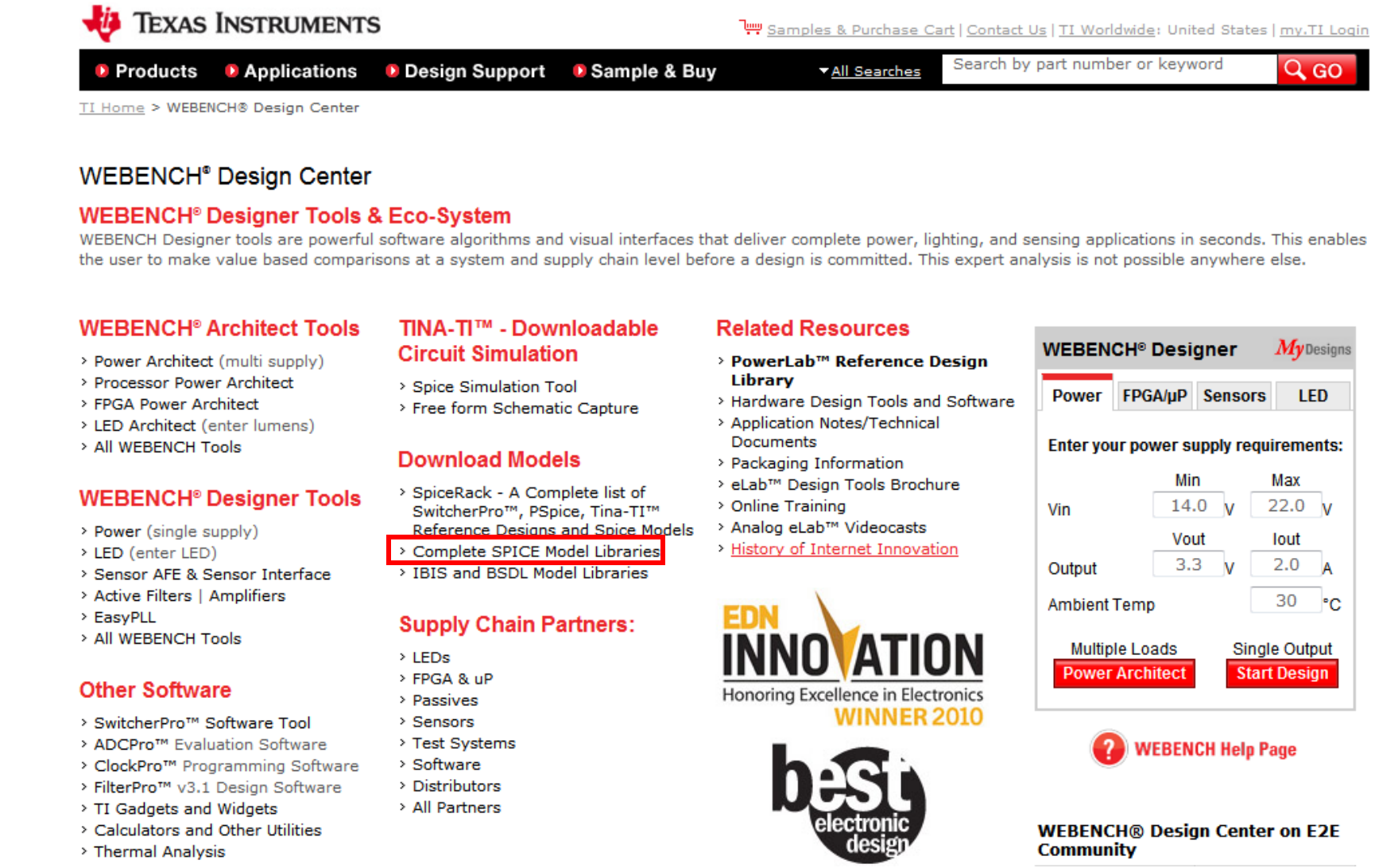


Installing TINA-TI 9

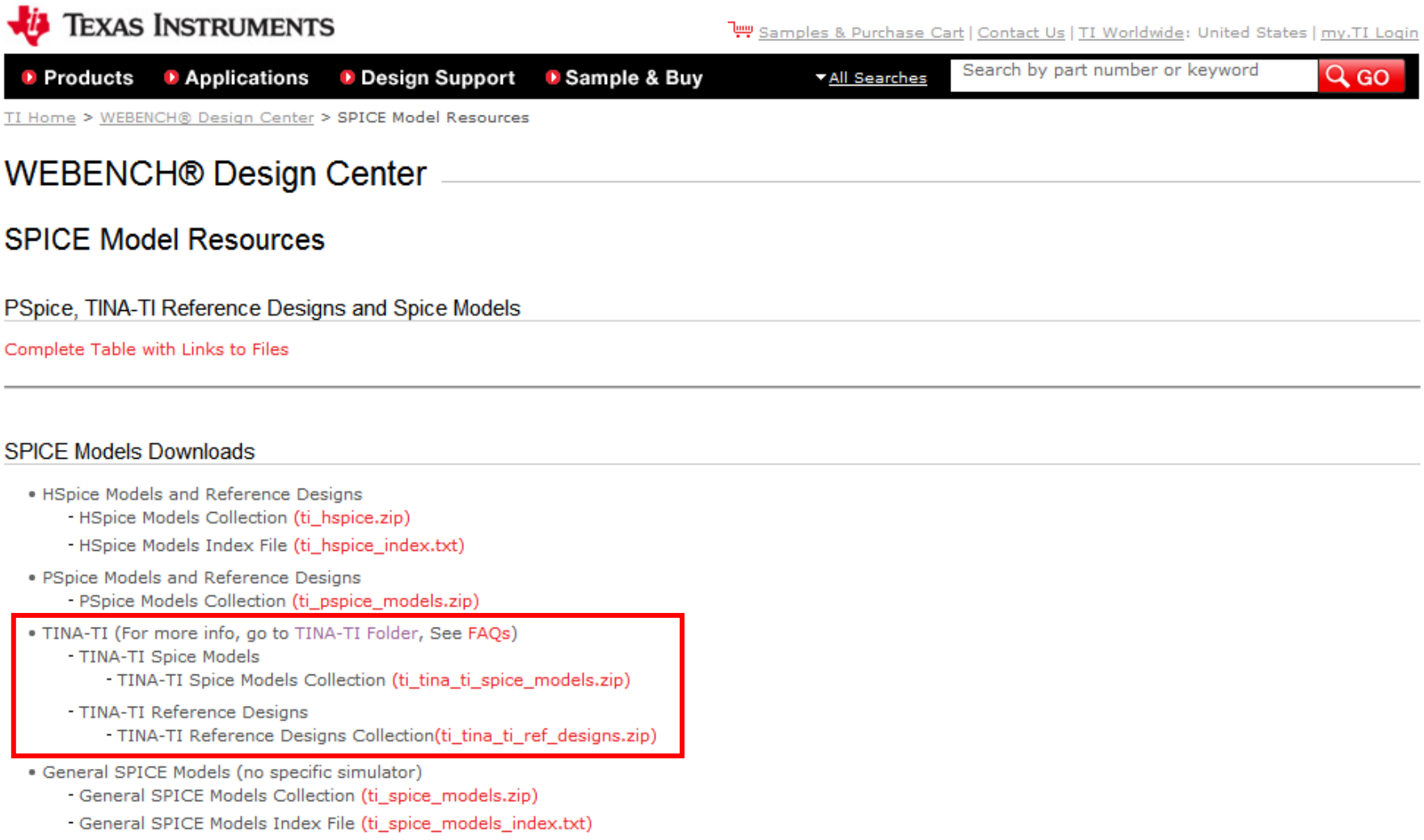




Downloading the Model Library



Downloading the Model Library



Texas Instruments

Samples & Purchase Cart | Contact Us | TI Worldwide: United States | my.TI Login

Products | Applications | Design Support | Sample & Buy | All Searches | Search by part number or keyword | GO

TI Home > WEBENCH® Design Center > SPICE Model Resources

WEBENCH® Design Center

SPICE Model Resources

PSpice, TINA-TI Reference Designs and Spice Models

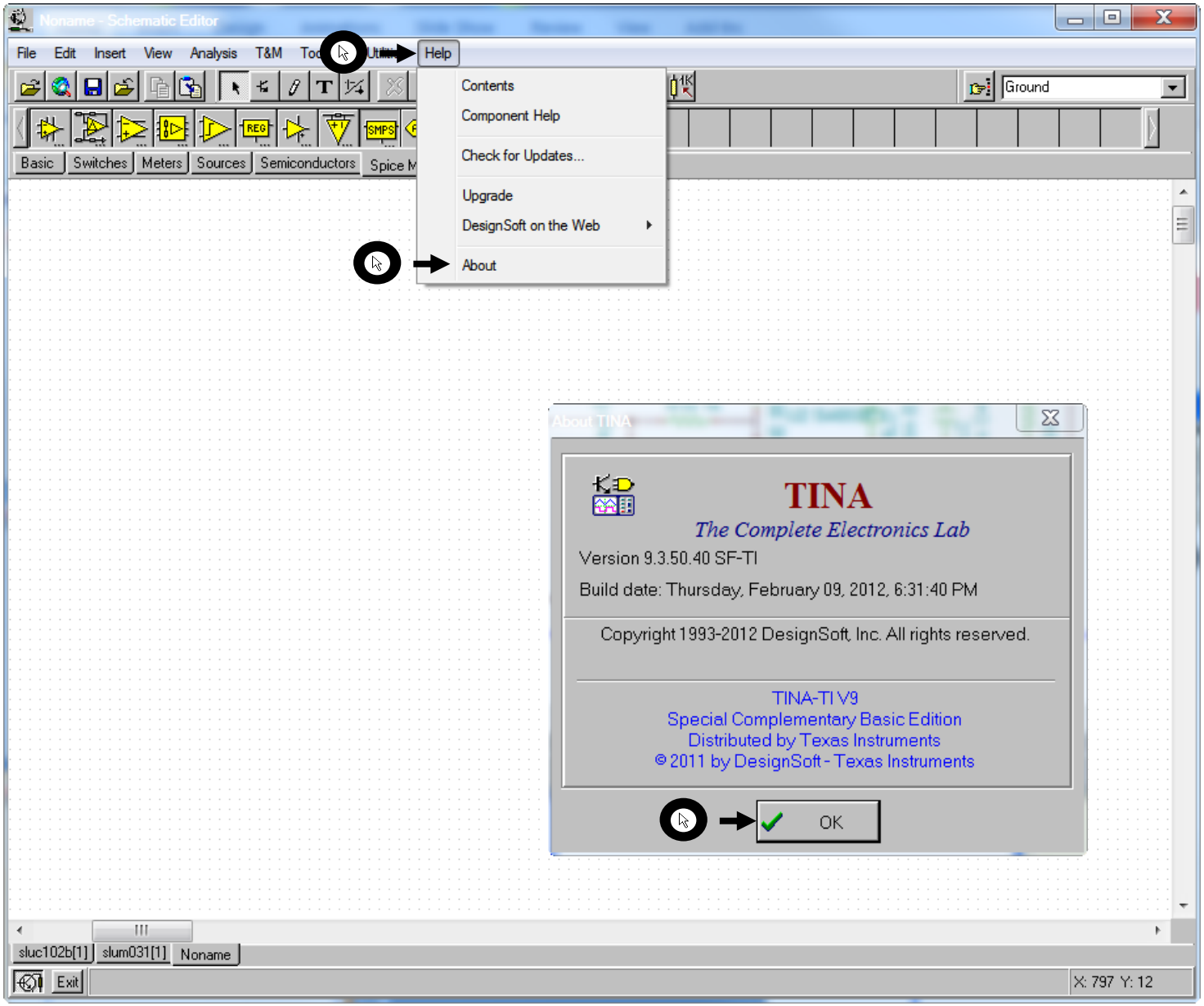
Complete Table with Links to Files

SPICE Models Downloads

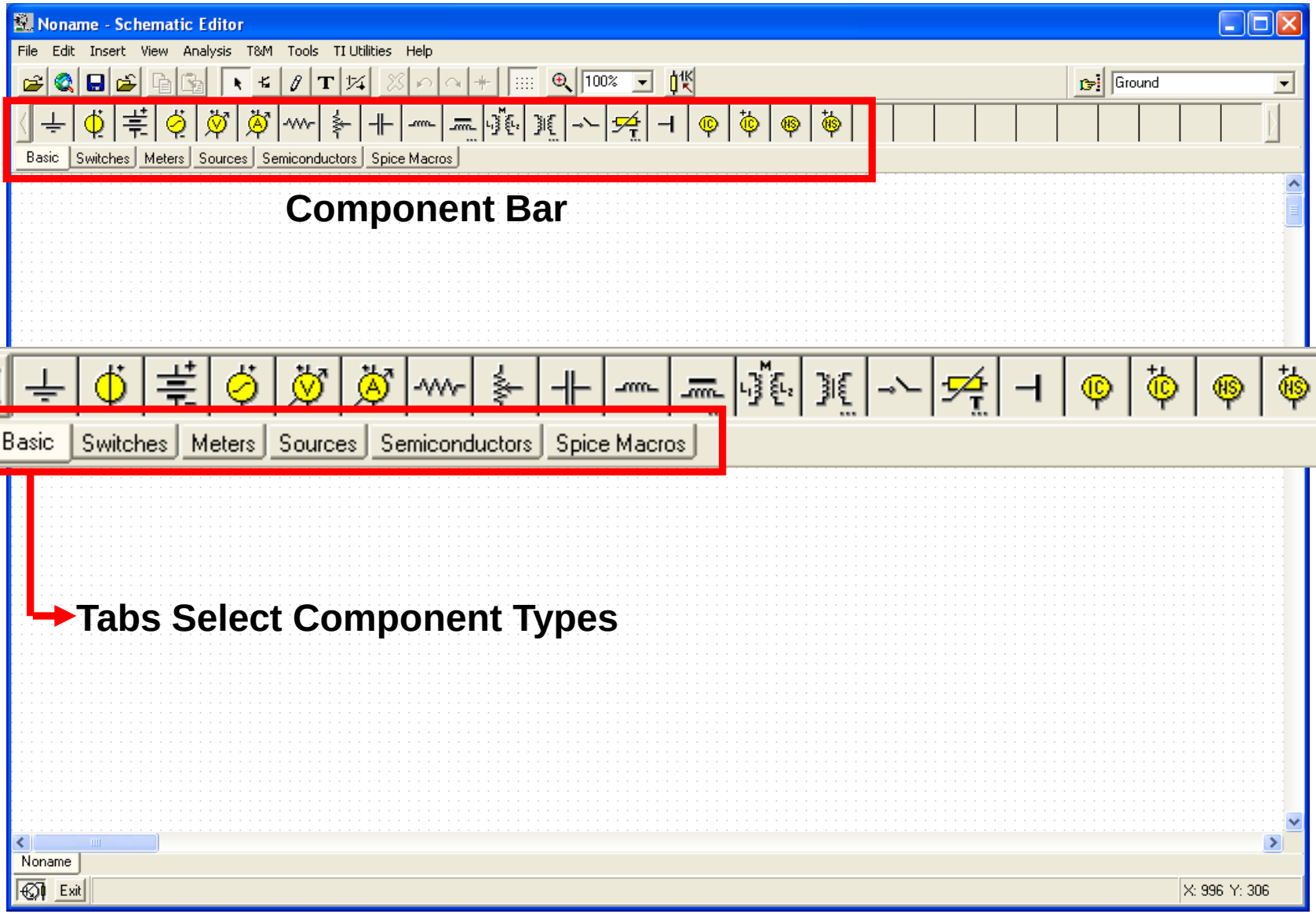
- HSpice Models and Reference Designs
  - HSpice Models Collection (ti_hspice.zip)
  - HSpice Models Index File (ti_hspice_index.txt)
- PSpice Models and Reference Designs
  - PSpice Models Collection (ti_pspice_models.zip)
- TINA-TI (For more info, go to TINA-TI Folder, See FAQs)
  - TINA-TI Spice Models
    - 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

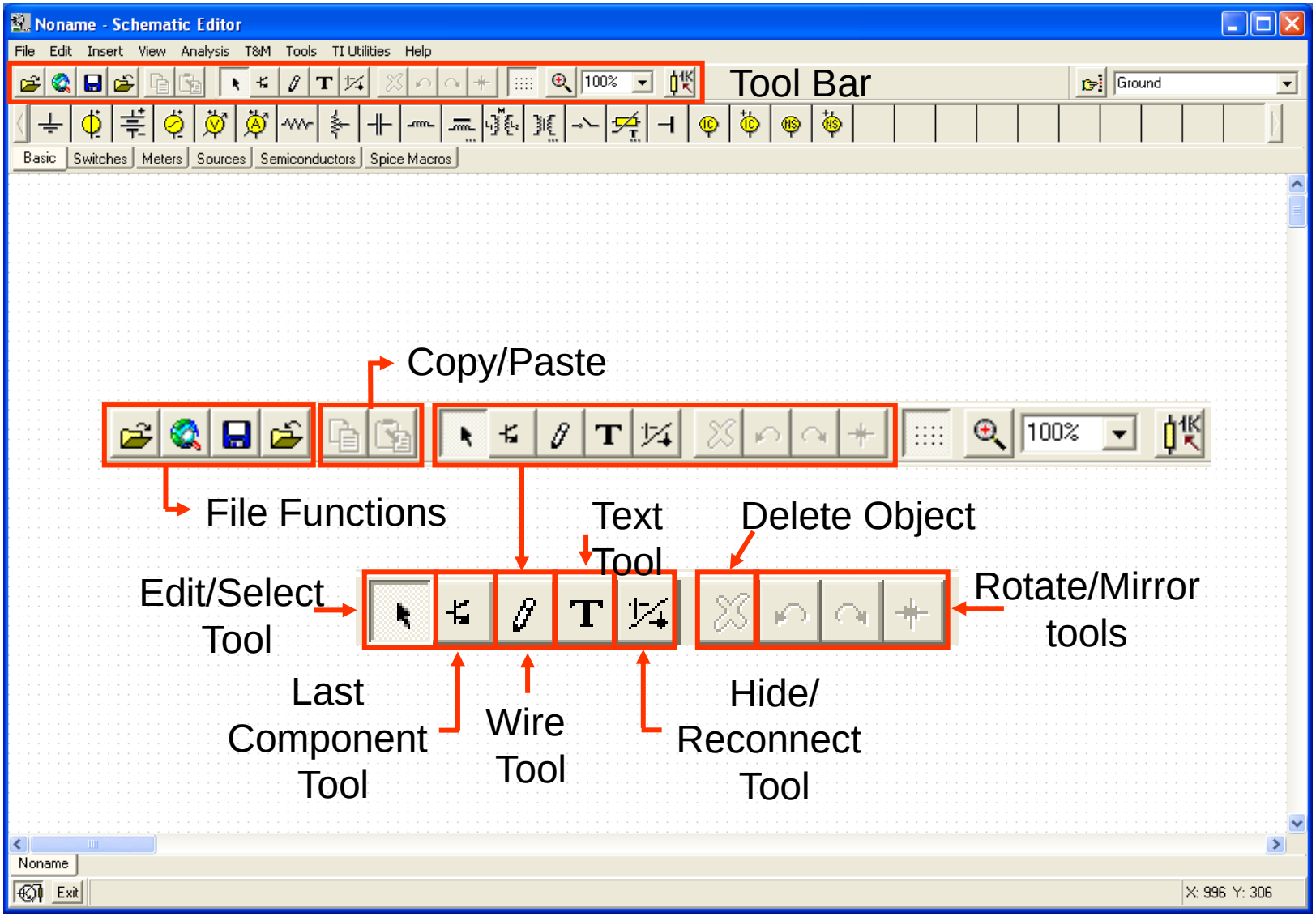
Starting TINA-TI



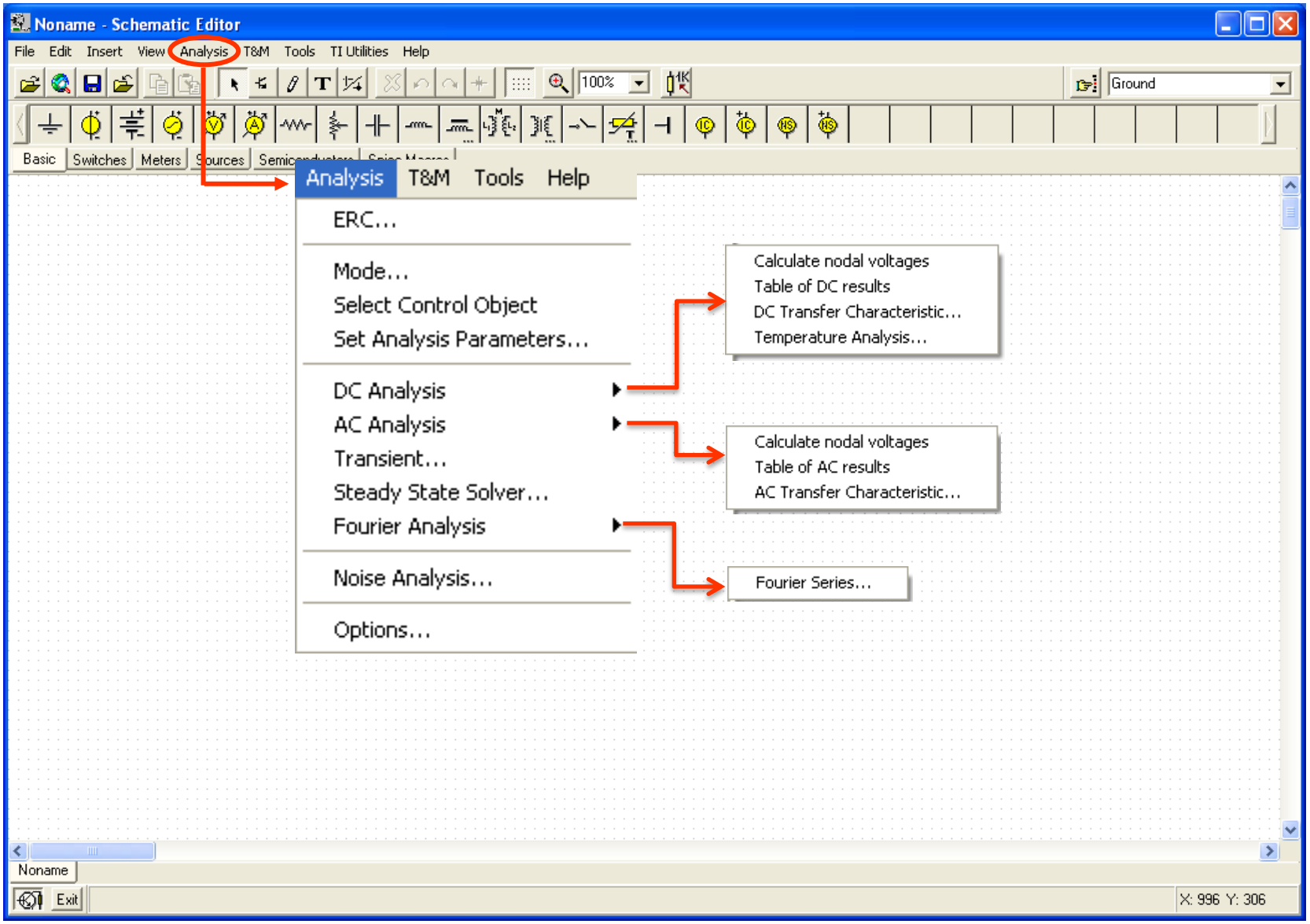
TINA-TI Schematic Editor Tour



TINA-TI Schematic Editor Tour

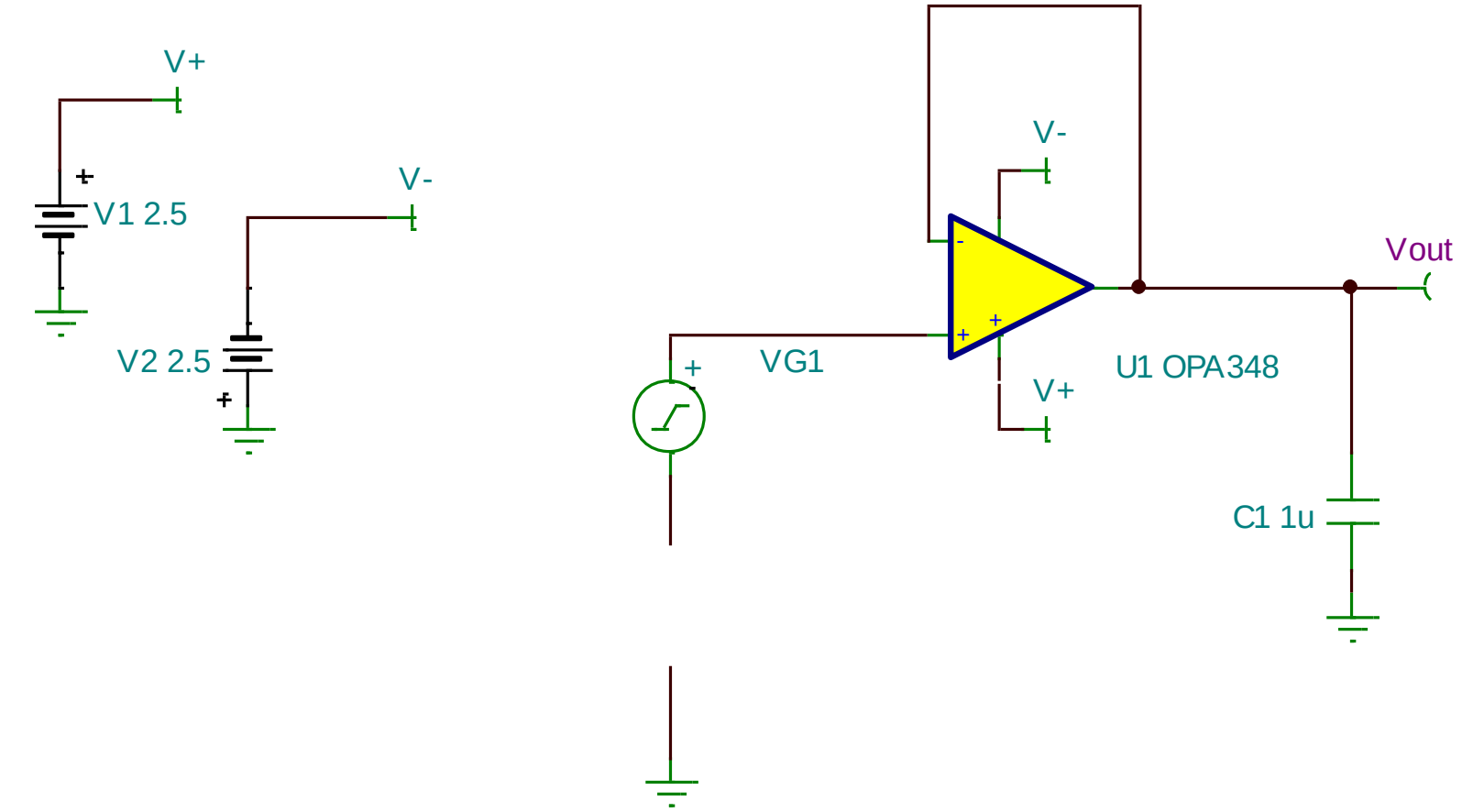


TINA-TI Schematic Editor Tour



OPA348 with a Capacitive Load

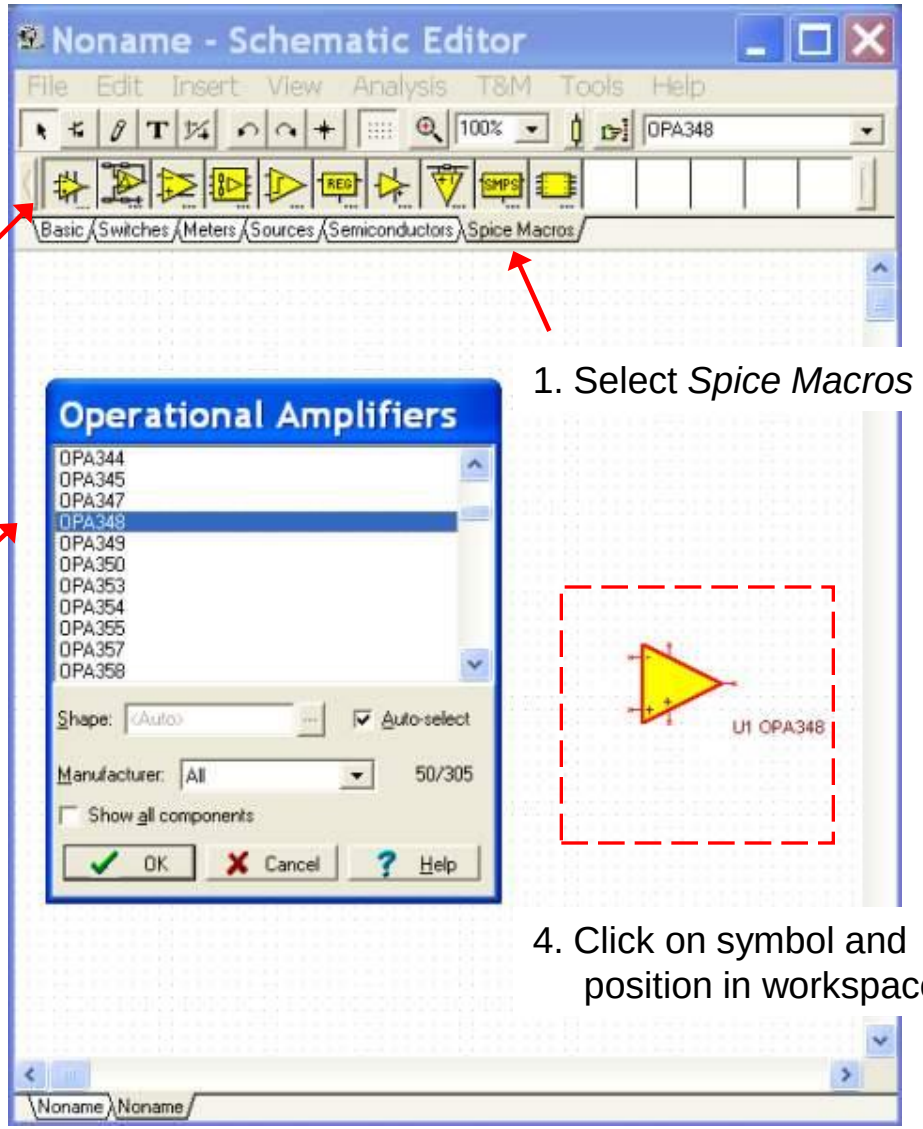
Let's simulate the OPA348 with a 1 uF load



Select an Operational Amplifier

2. Select op amp symbol – list opens

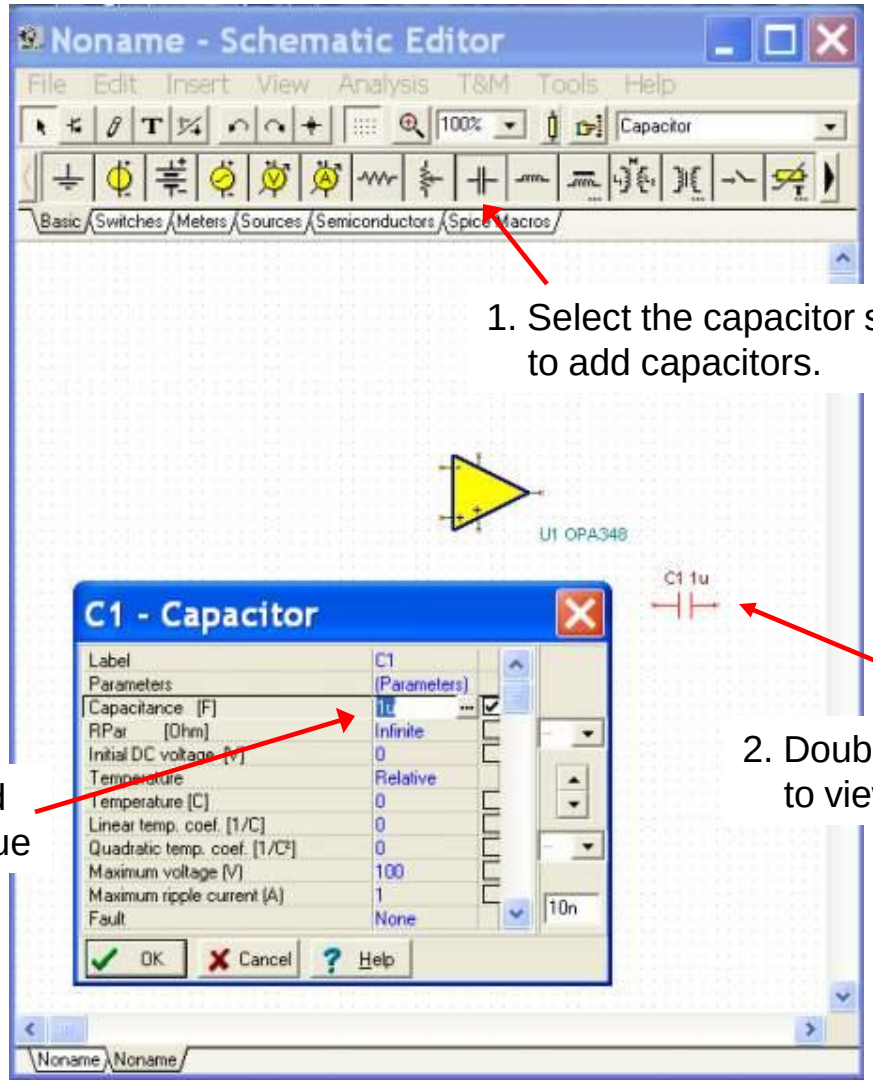
3. Scroll list and select op amp - click OK



1. Select *Spice Macros*

4. Click on symbol and position in workspace

Add Capacitor

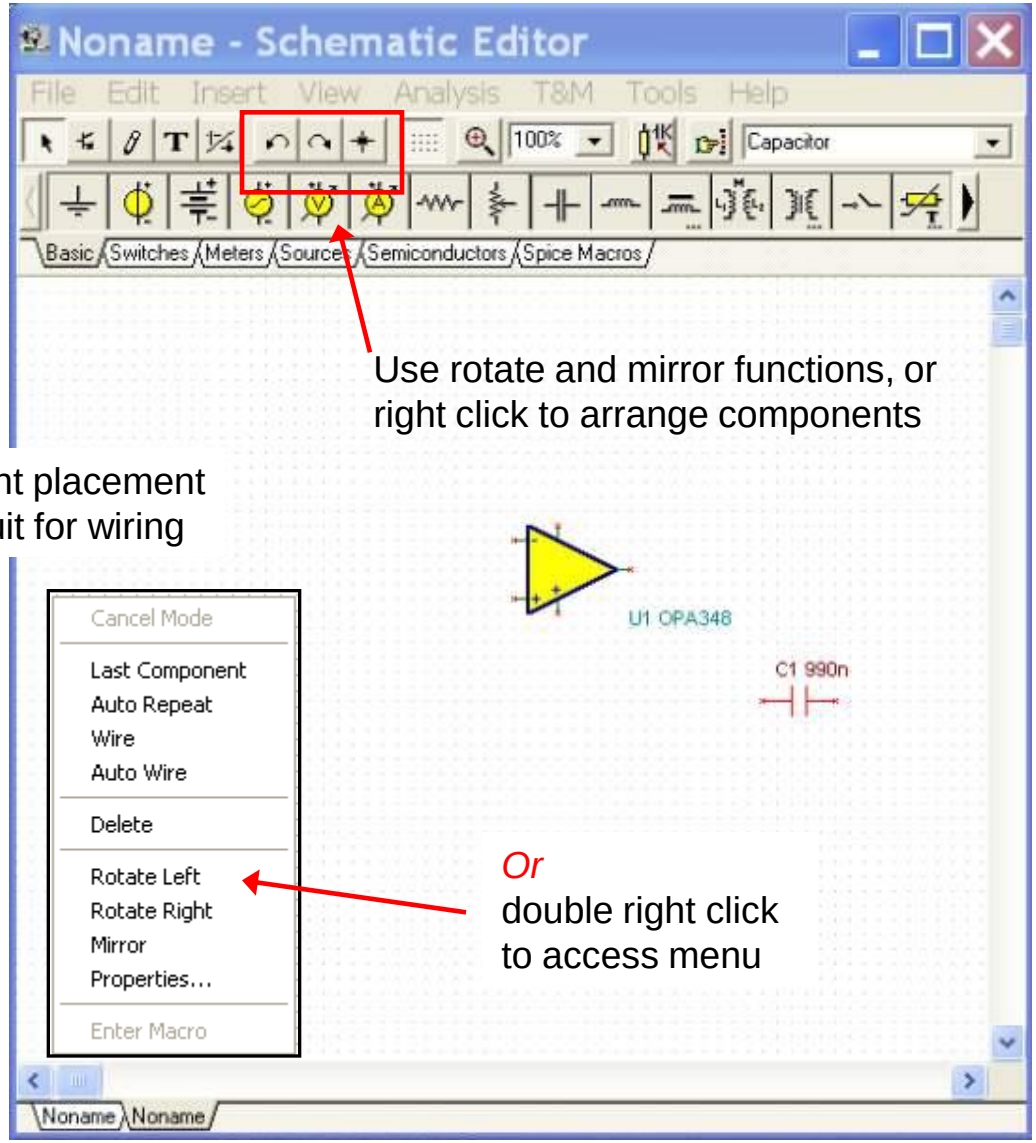


1. Select the capacitor symbol to add capacitors.

2. Double click the capacitor to view the parameters

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

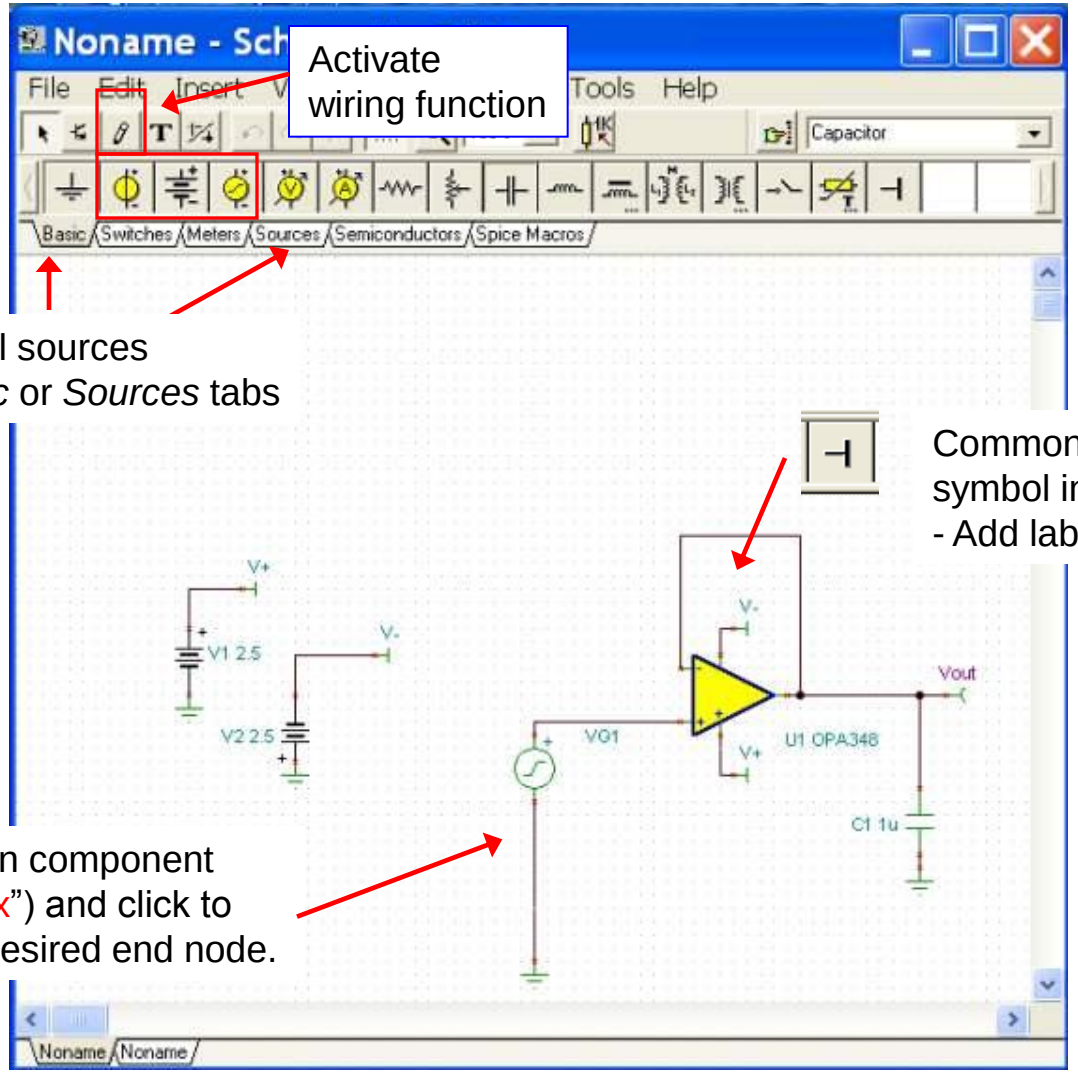
Manipulate Components



Complete component placement and ready the circuit for wiring

Or
double right click to access menu

Add Sources, Generators, and Wiring

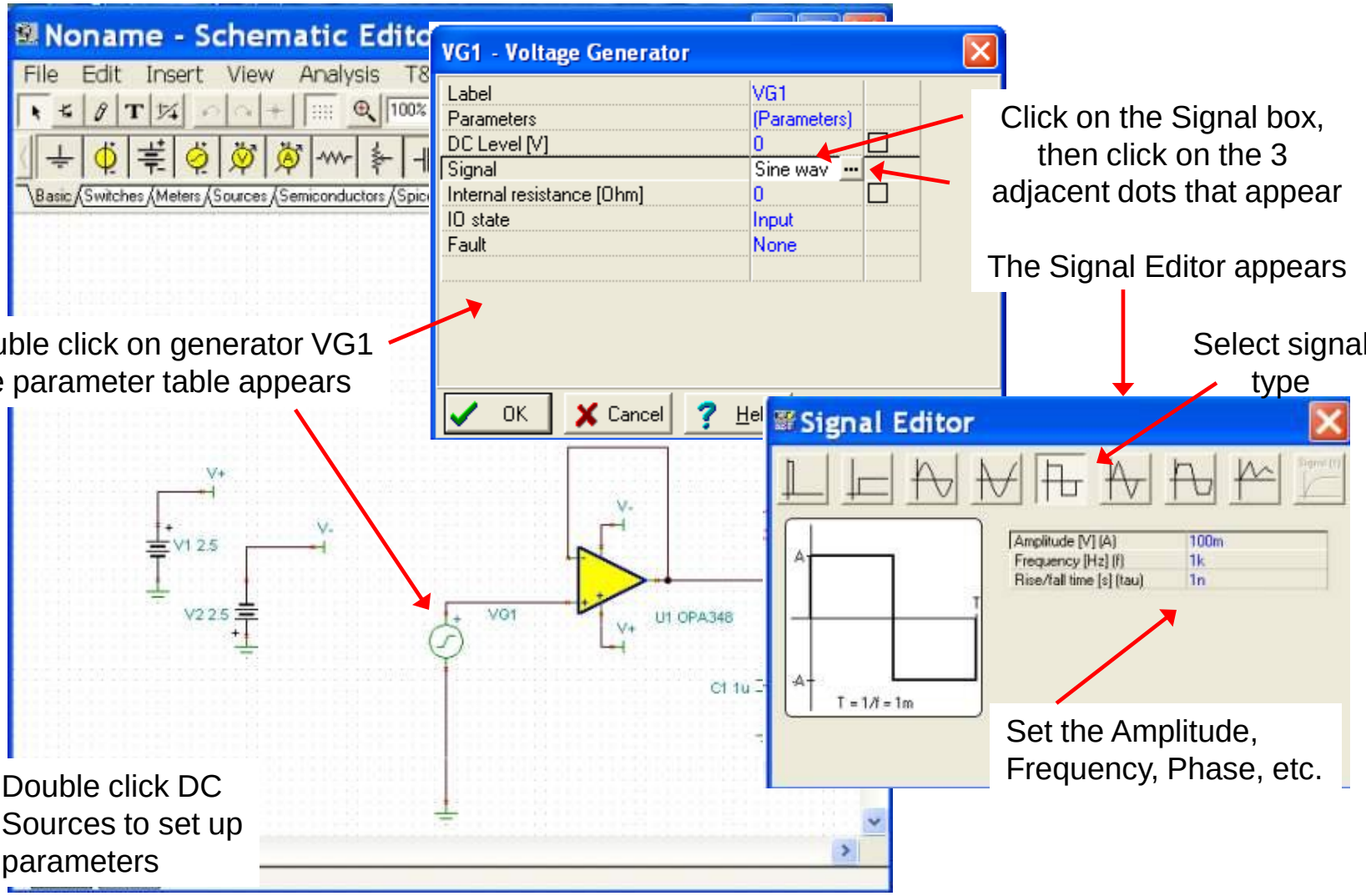


Supplies and signal sources selected from *Basic* or *Sources* tabs

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.

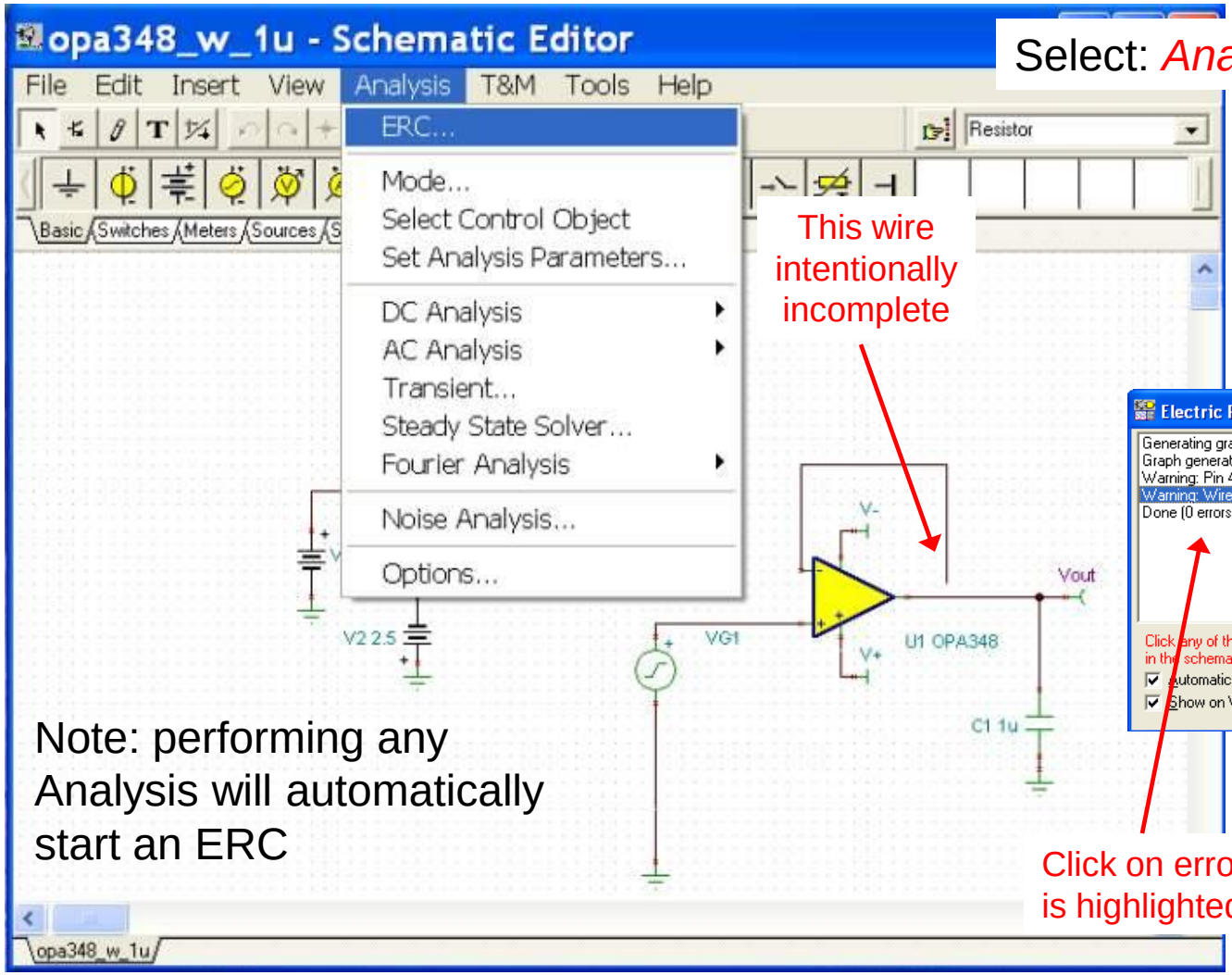
Function Generators



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

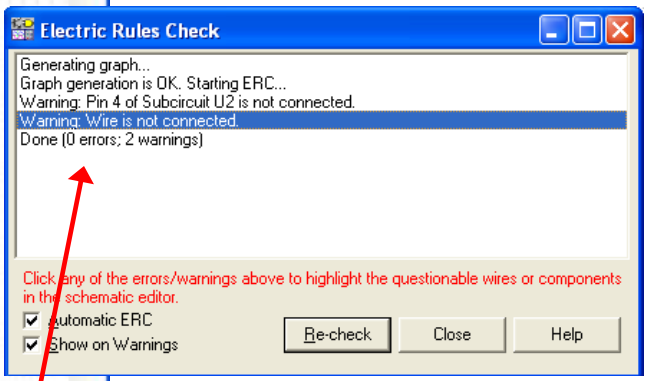
Running Electrical Rules Check (ERC)



Select: *Analysis > ERC*

This wire intentionally incomplete

ERC Report

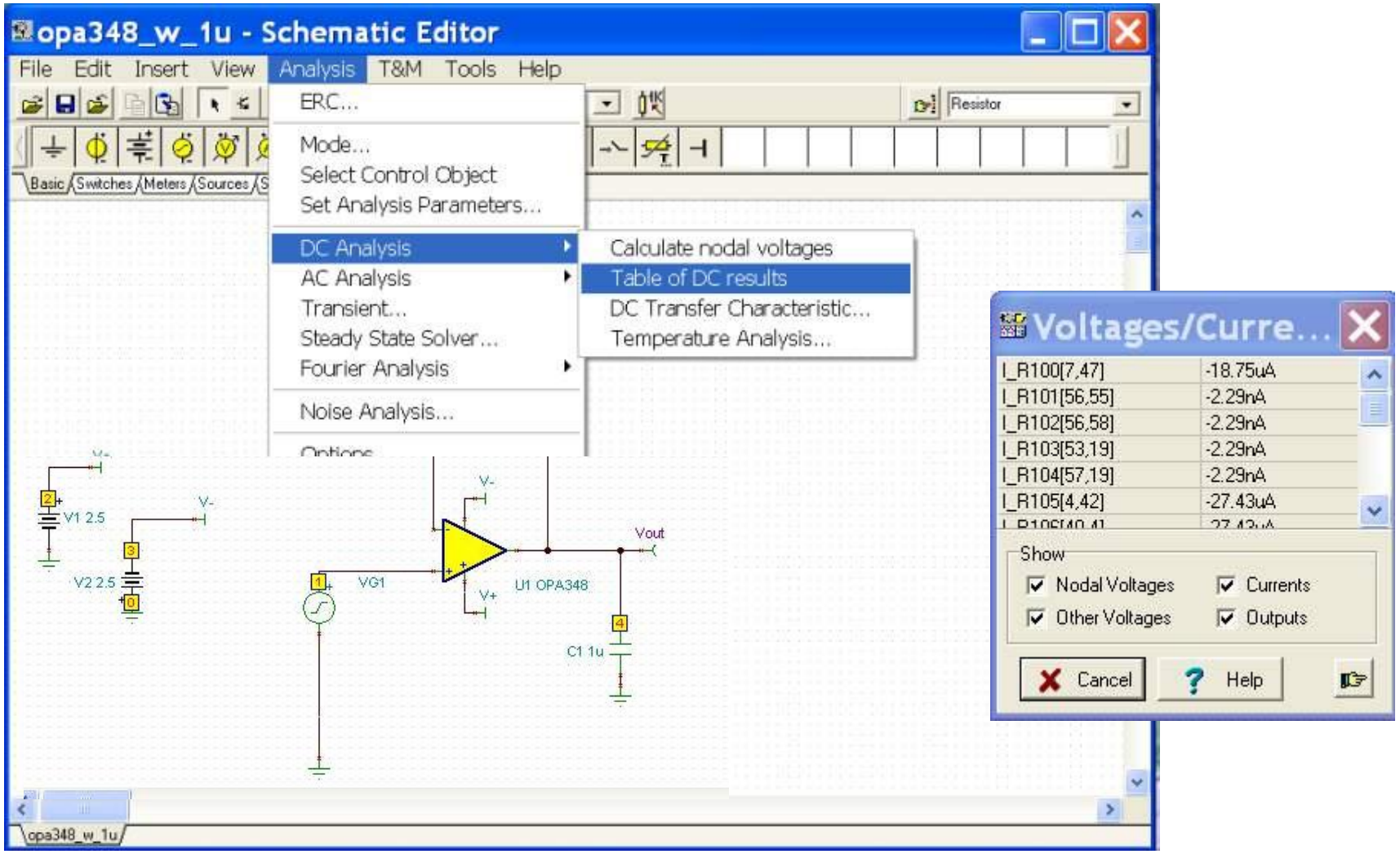


Note: performing any Analysis will automatically start an ERC

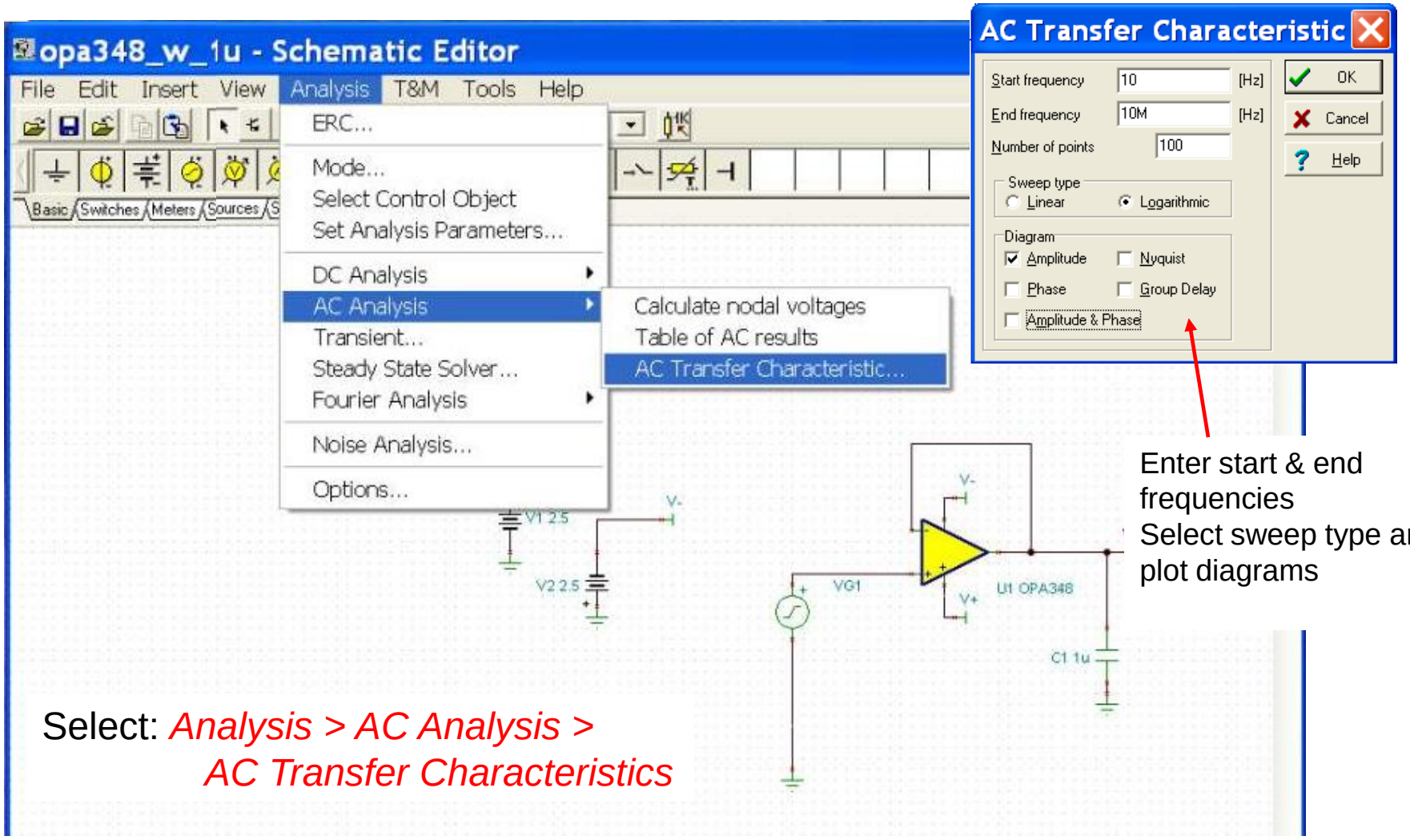
Click on error line and the error is highlighted in the schematic

DC Analysis

Select: Analysis > DC Analysis > Calculate nodal voltages



AC Analysis



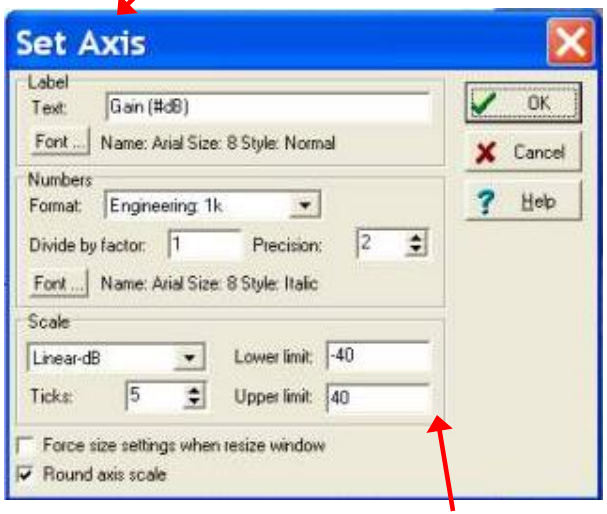
Enter start & end frequencies
Select sweep type and plot diagrams

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

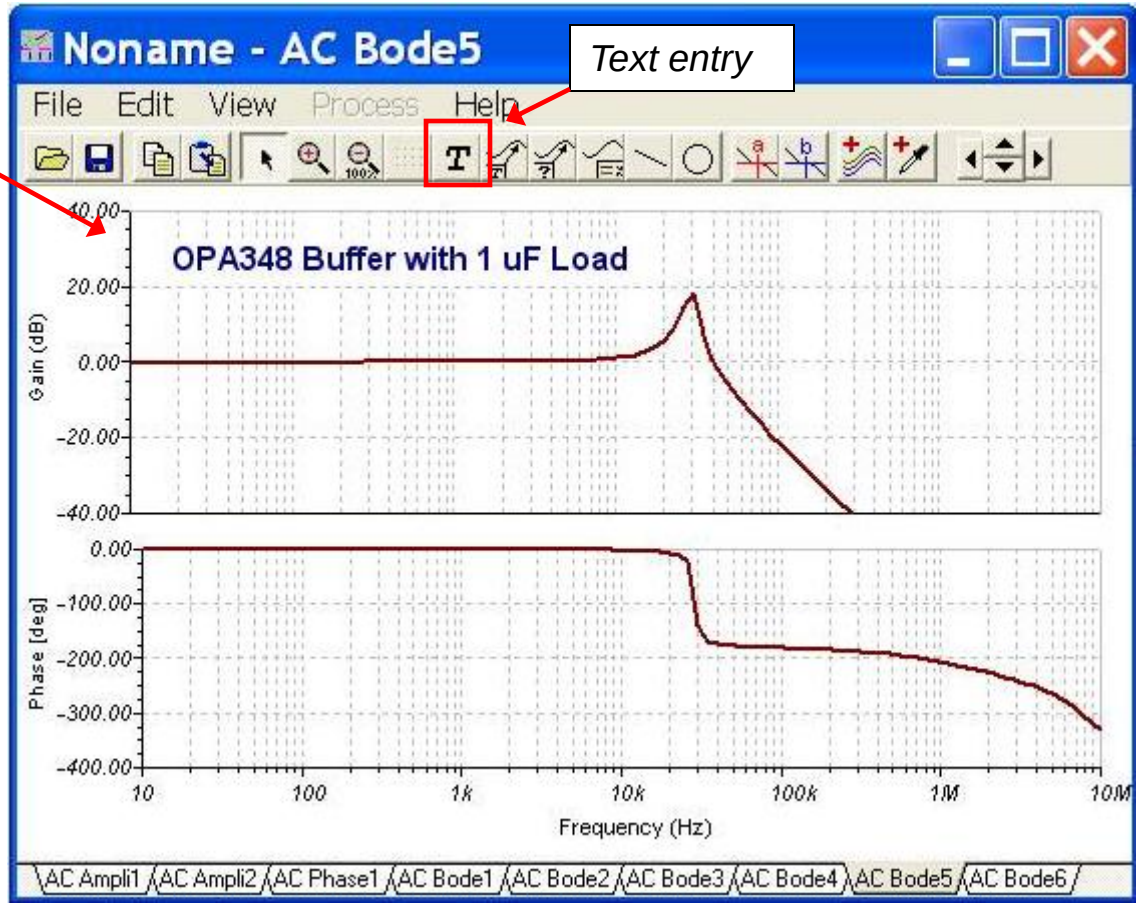
AC Analysis Transfer Characteristics

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

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

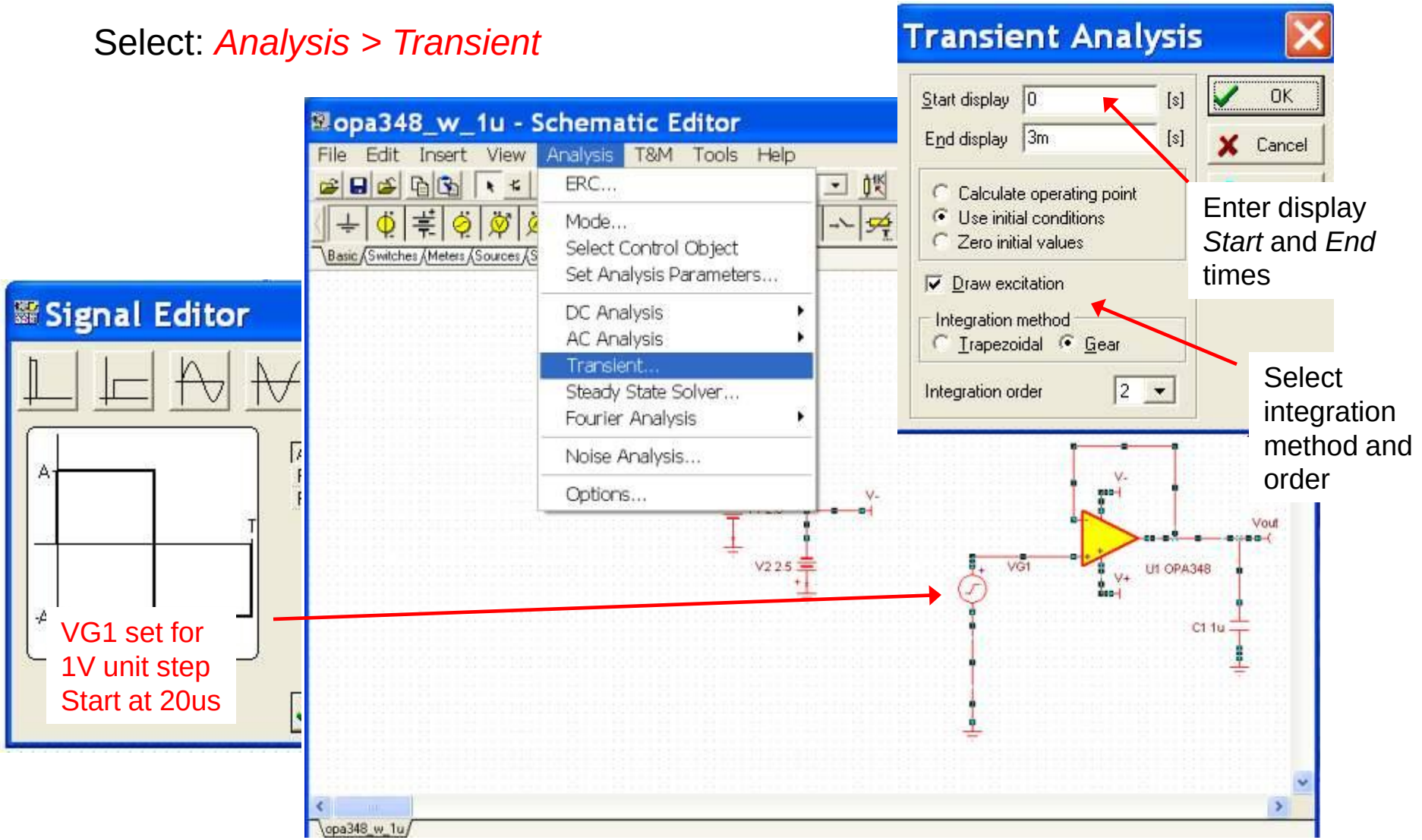


Enter axis limits and other settings



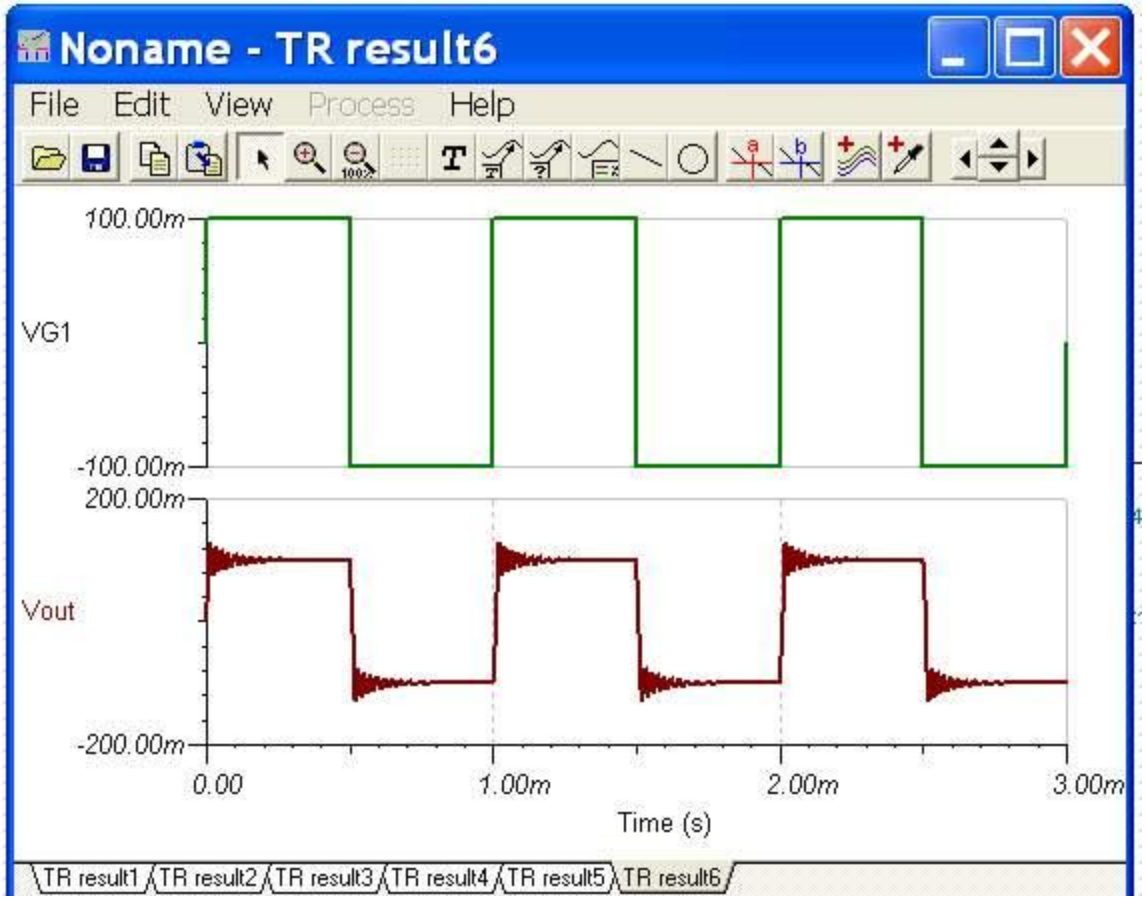
Transient Analysis

Select: *Analysis > Transient*

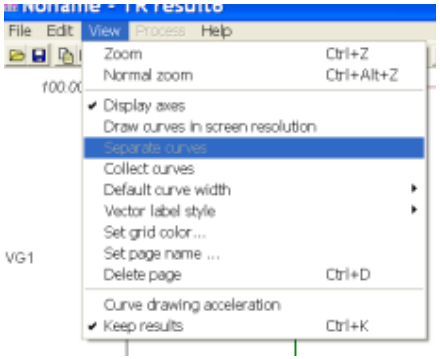


Time domain Graphs

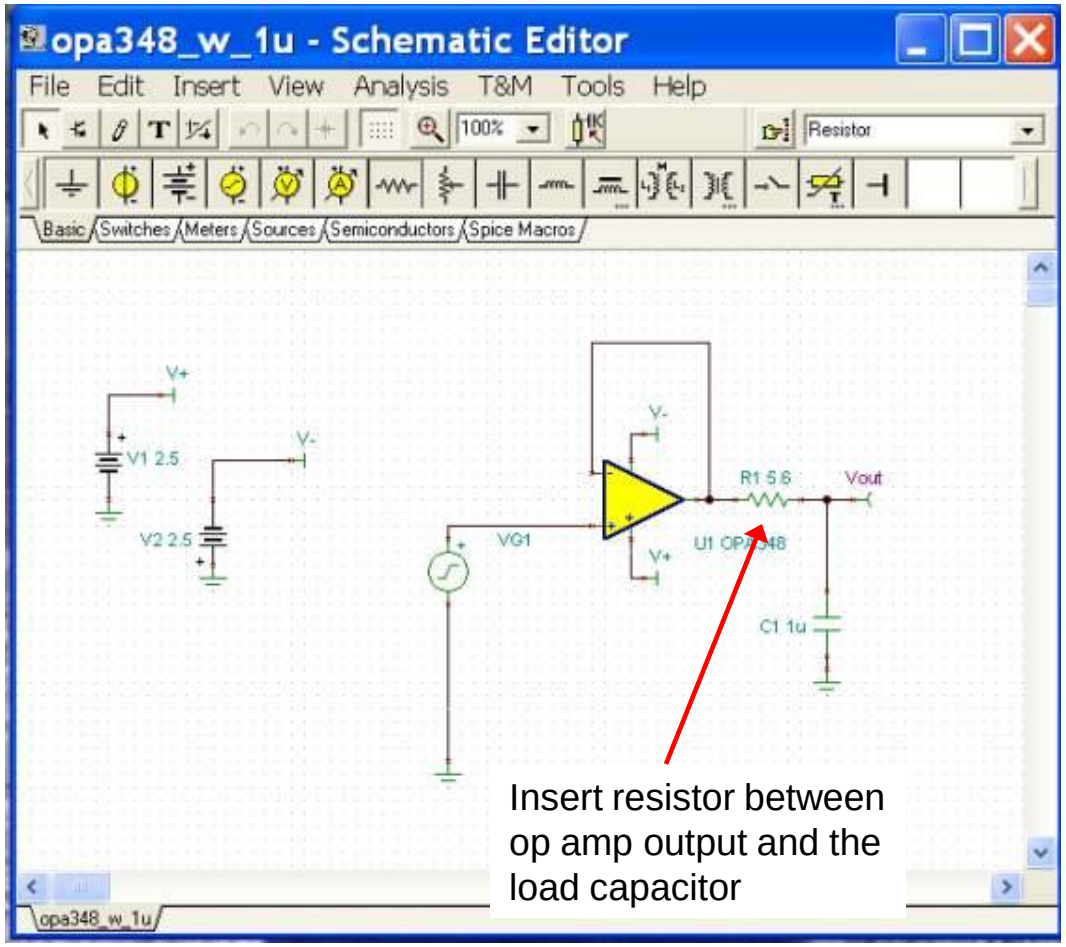
Square Wave Response – OPA348 Buffer with 1 uF load



Separate Curves



Design a Stable OPA348 Buffer



$GBWP_{OPA348}$ = 1 MHz

C_L = 1 uF

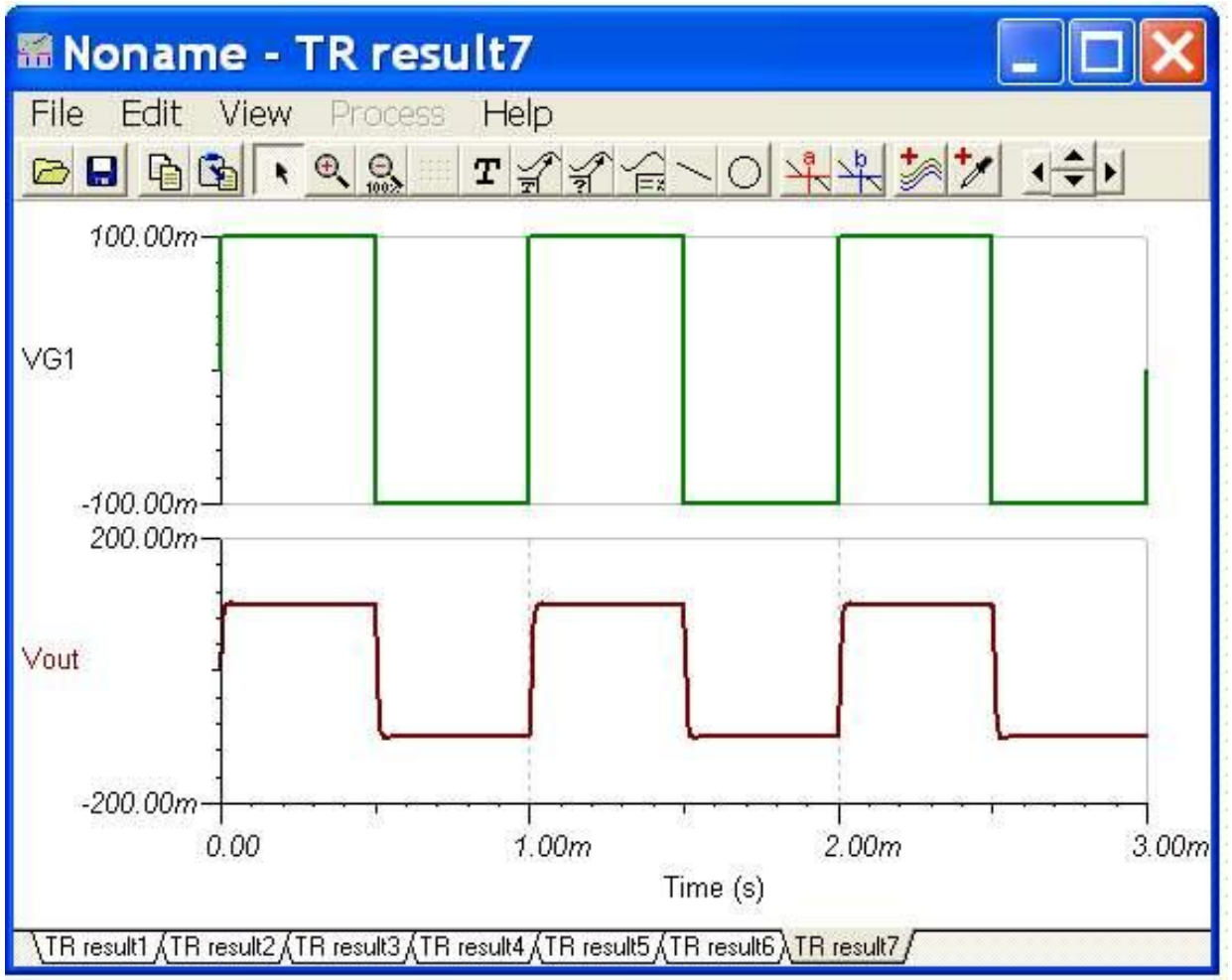
$R_L \geq$ 2 ohms

also

R_O of OPA348 ~ 50 ohms

$R_L \geq$ 5.6 ohms

Stable System - Transient Signal Results

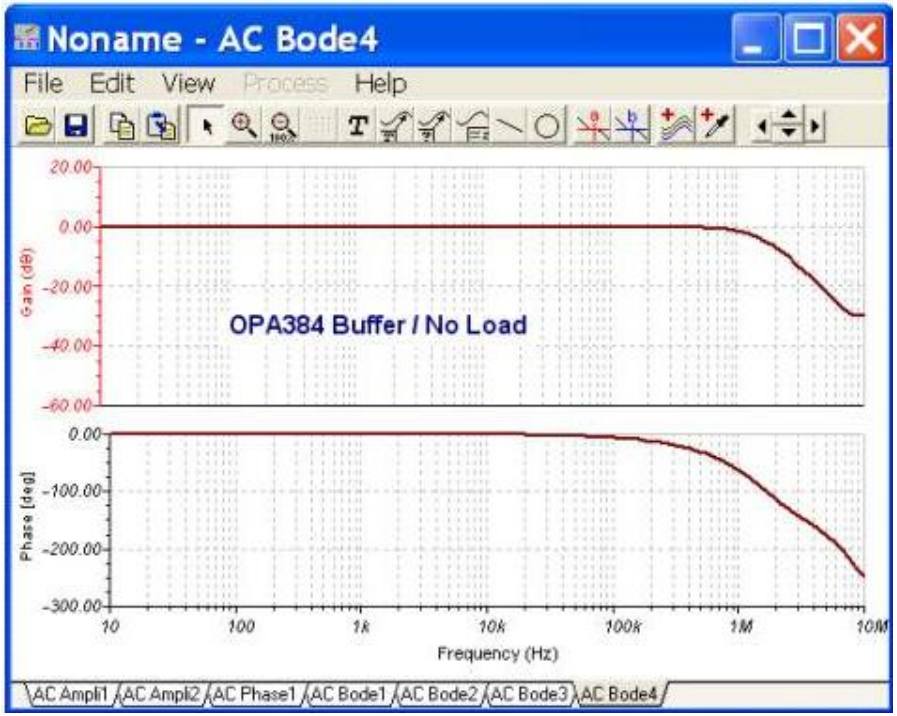


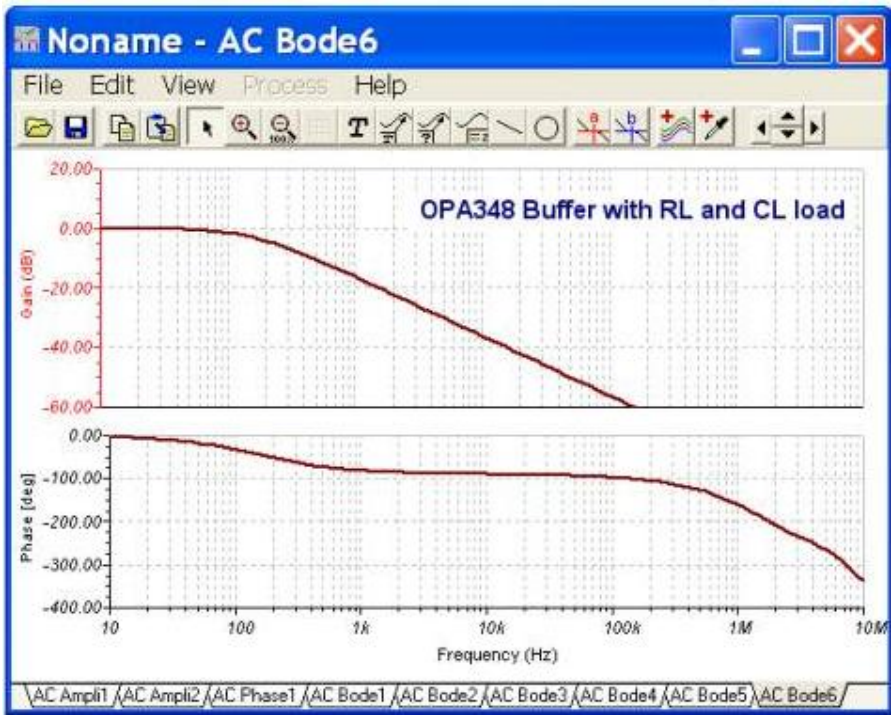
C_L = 1 uF

R_L = 5.6 ohms

Go Back to the AC Analysis

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





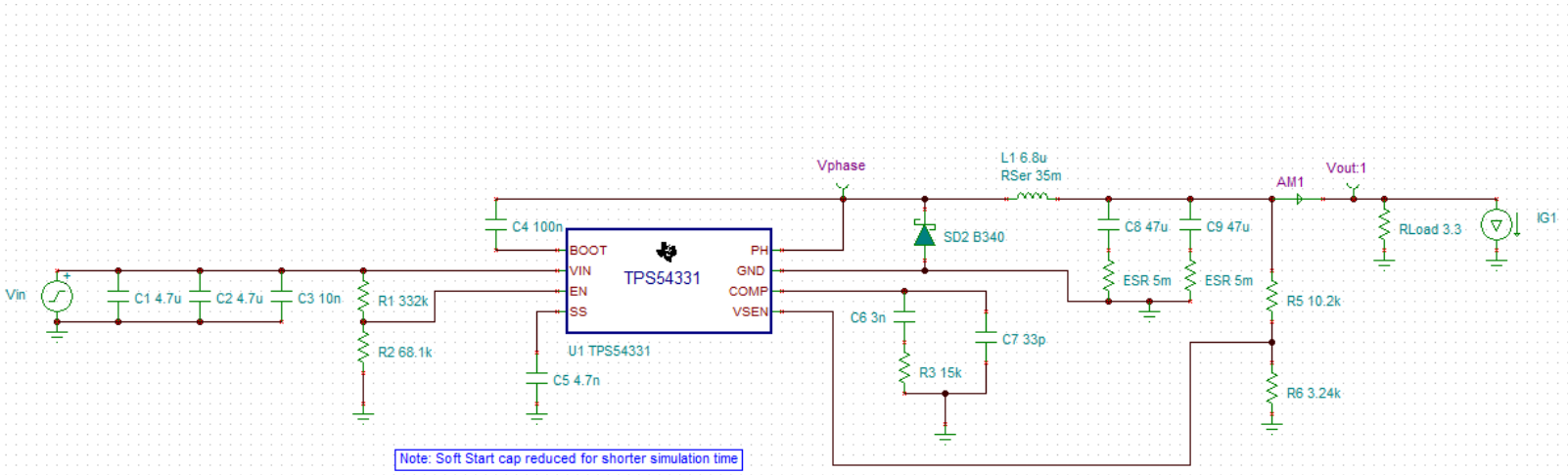
Power Modeling in TINA-TI v9

- 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

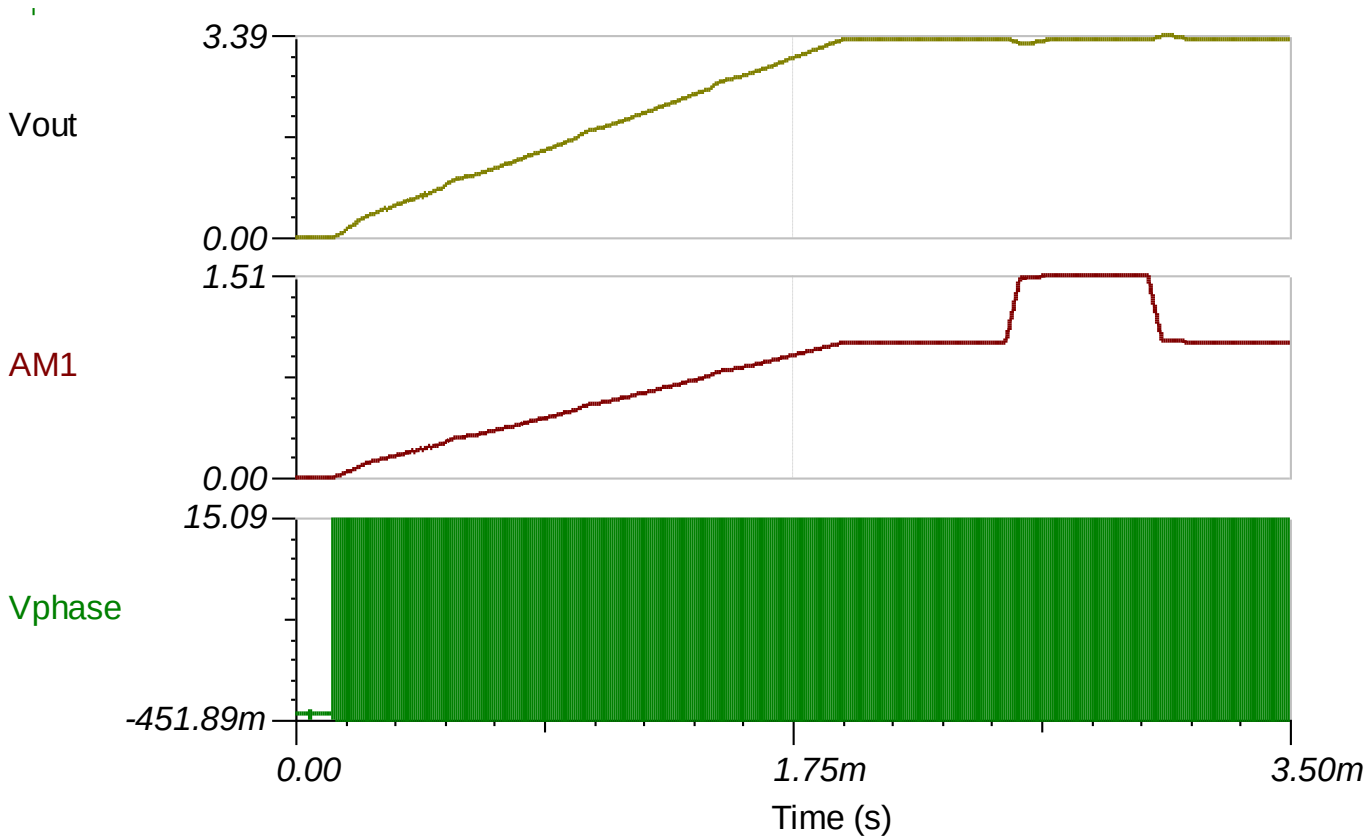
Power Example

A Typical Load transient simulation for the TPS54331



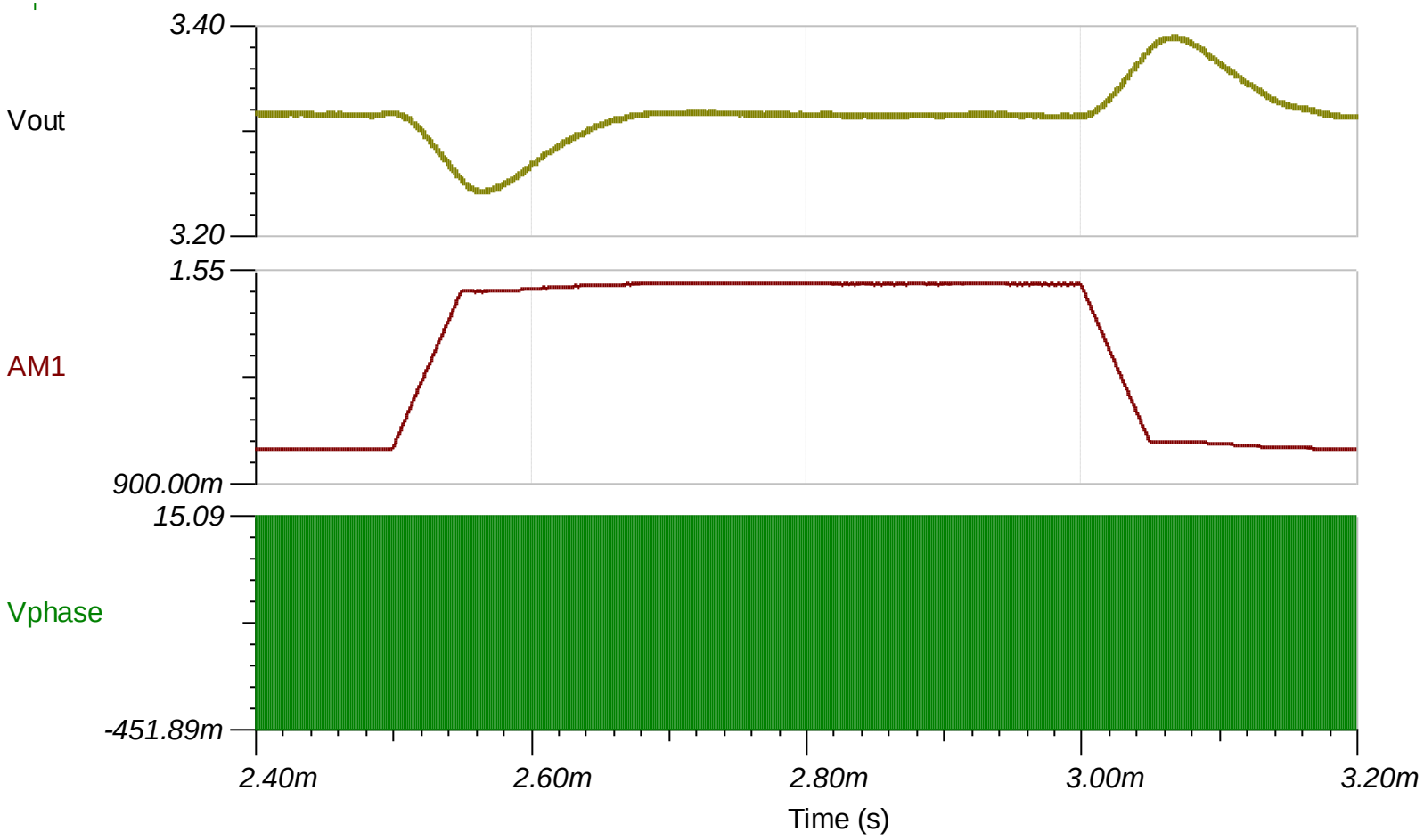
Power Example

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

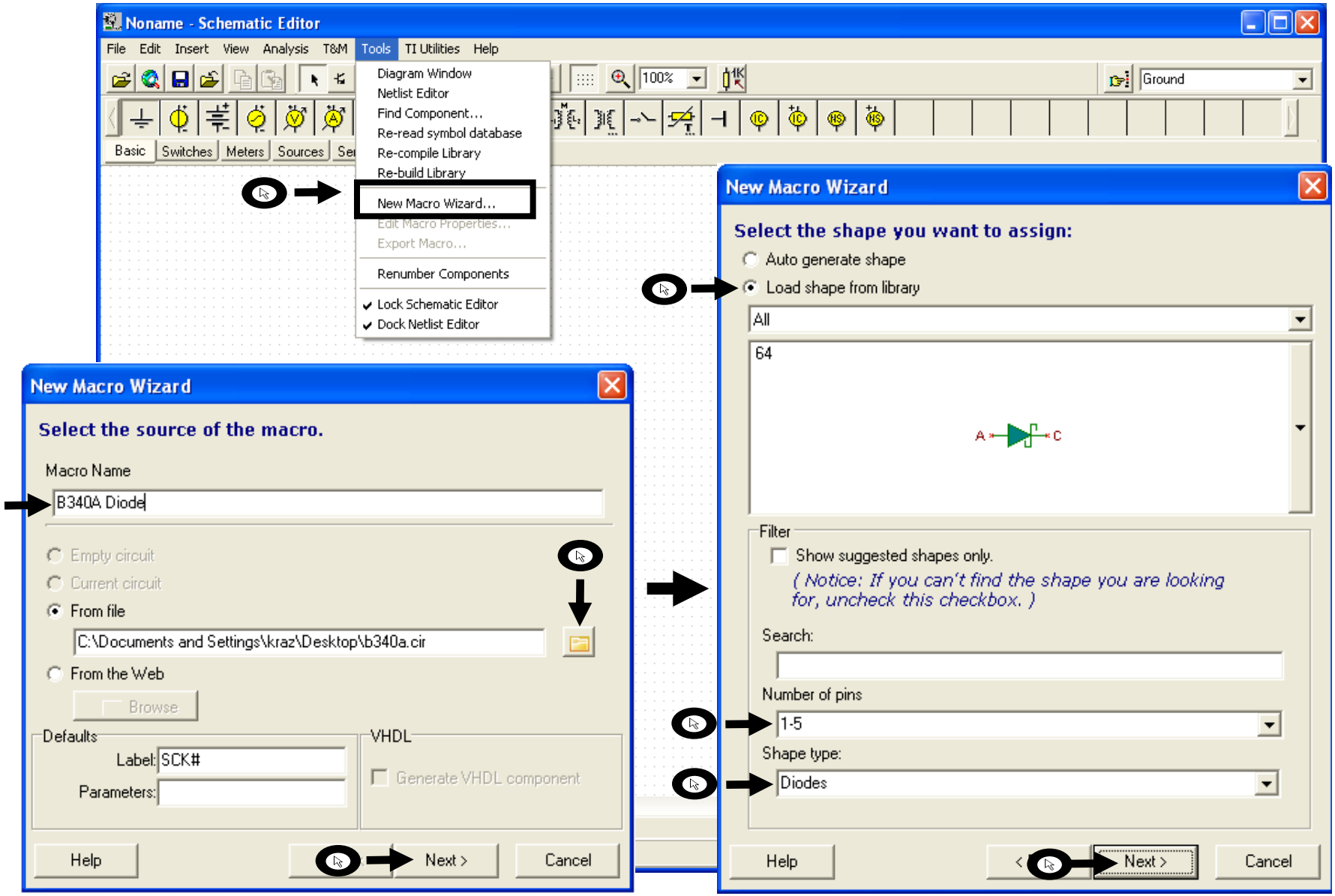
Importing a 3rd Party Model into TINA-TI

b340a.cir - Notepad

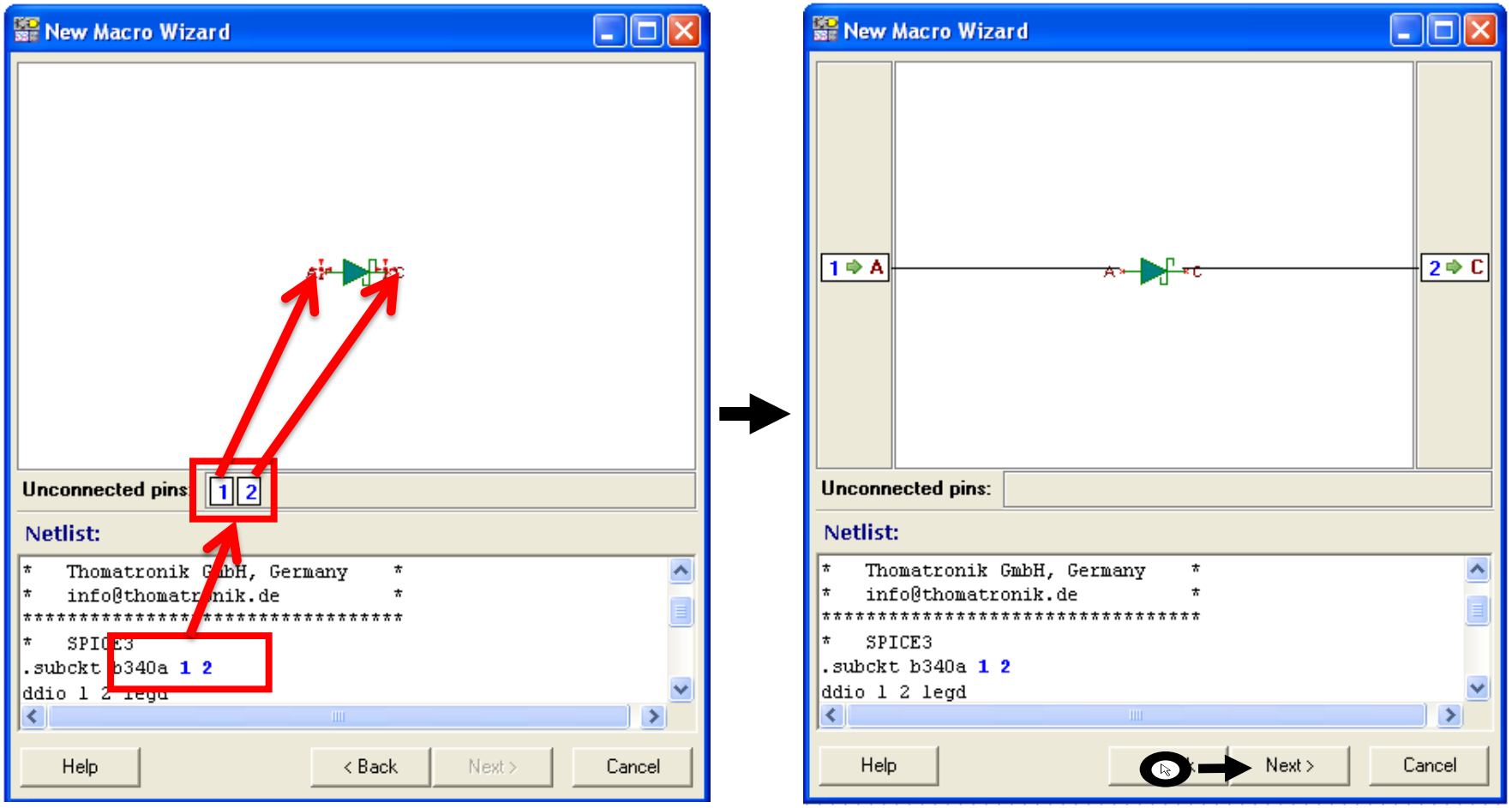
File Edit Format View Help

```
**************************************
* 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
```

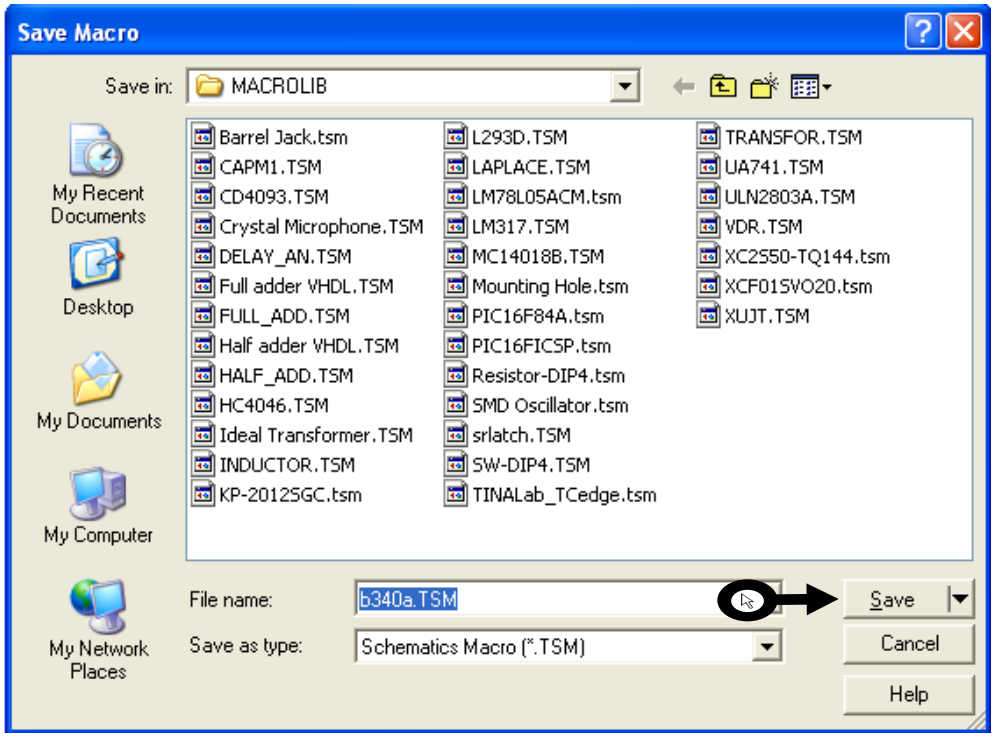
Importing a 3rd Party Model into TINA-TI

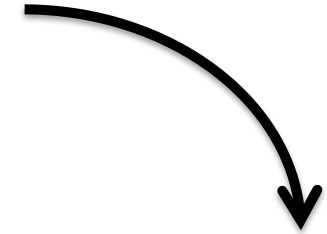


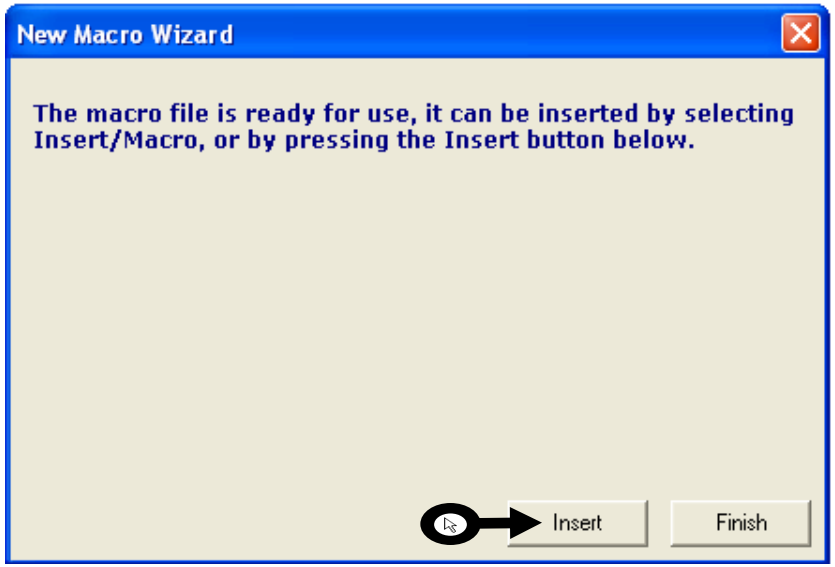
Importing a 3rd Party Model: Getting the Pinout Correct

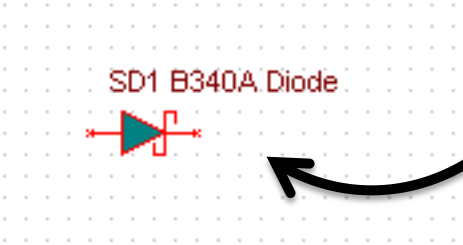


Importing a 3rd Party Model

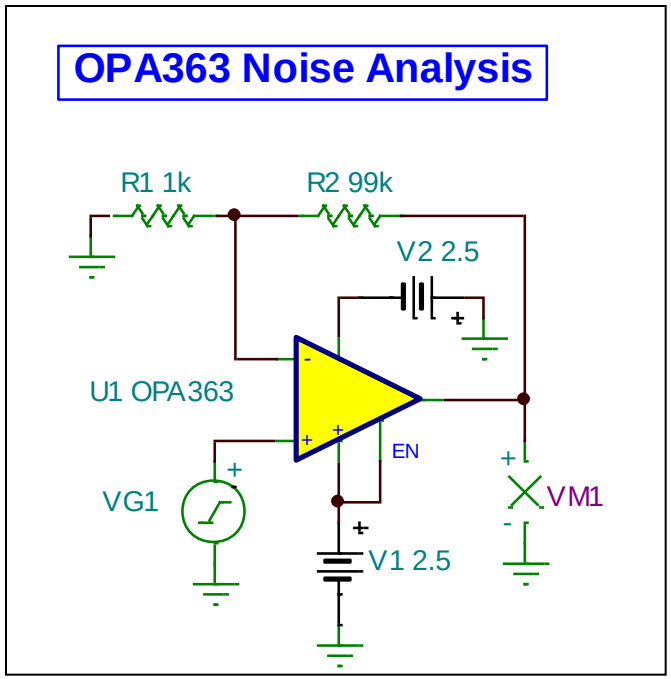




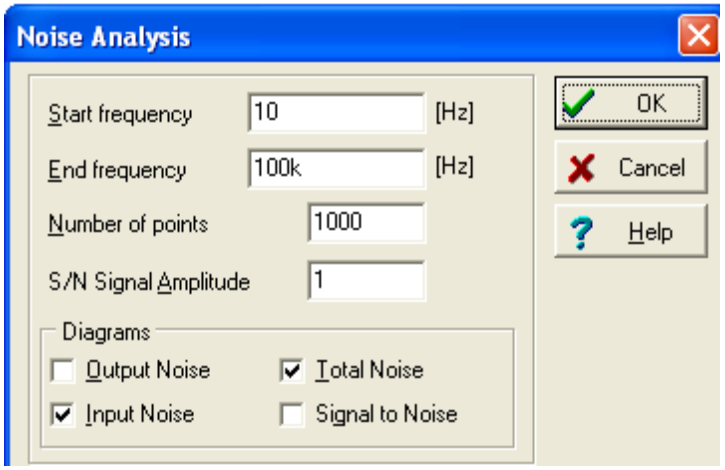




Noise Analysis
OPA363 + resistor noise



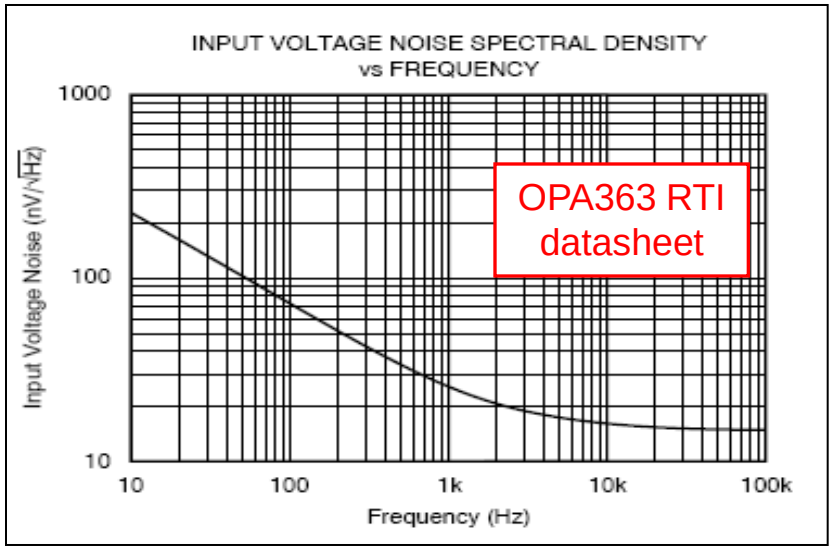
Select: *Analysis > Noise Analysis*

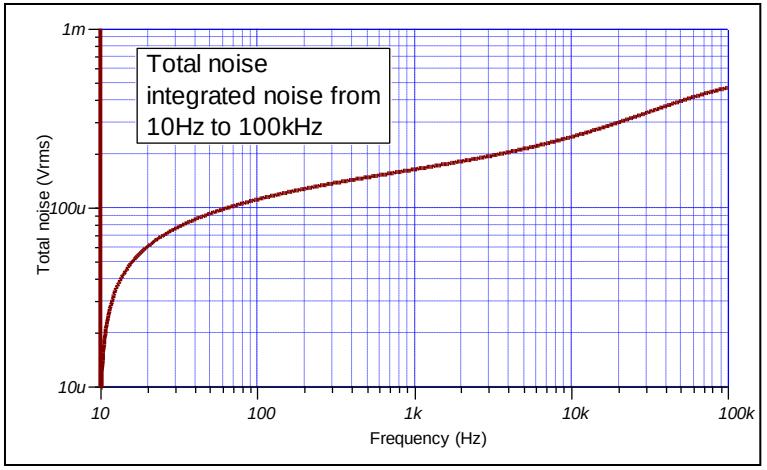


Input Noise and Total Noise responses are selected for simulation

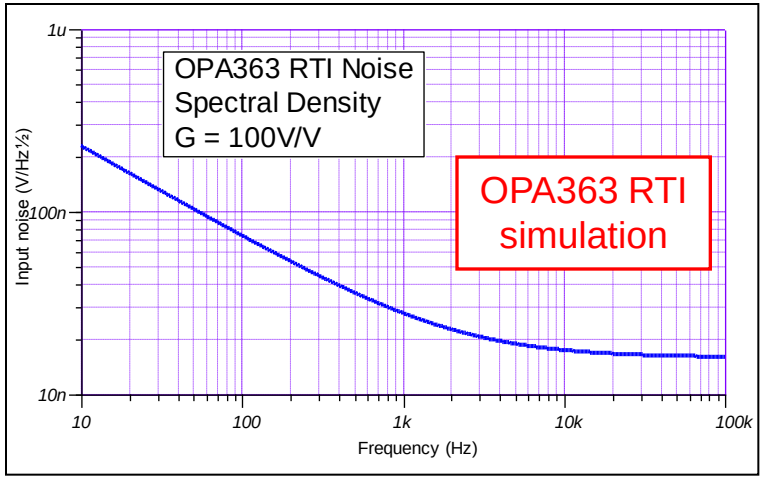
Datasheet vs. Simulation

Noise Analysis

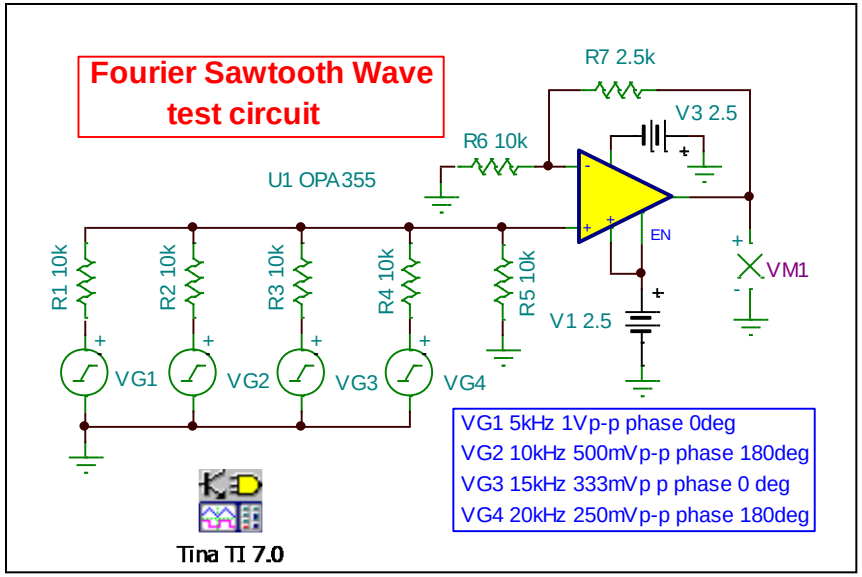




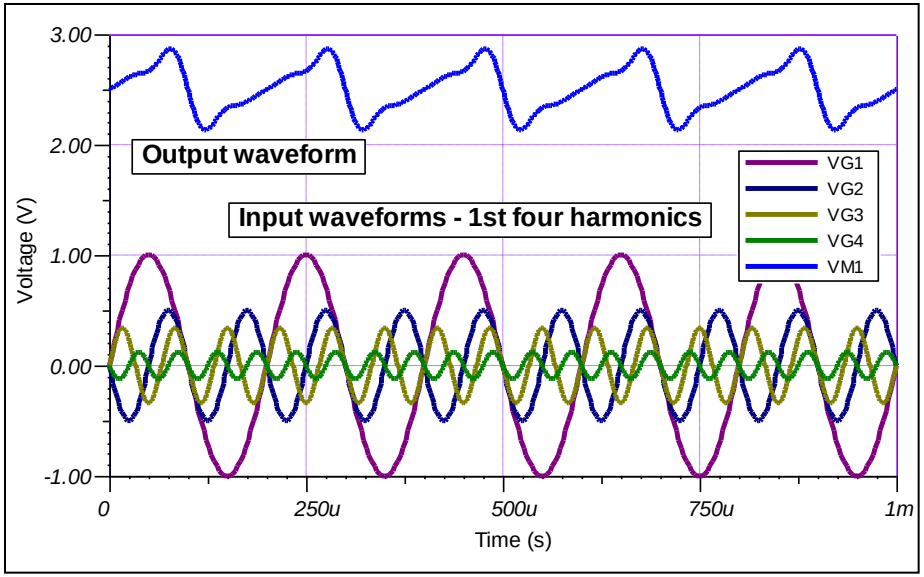
OPA363 Total noise simulation



Fourier Analysis



First Step: perform a Transient Analysis



Fourier Series for sawtooth wave

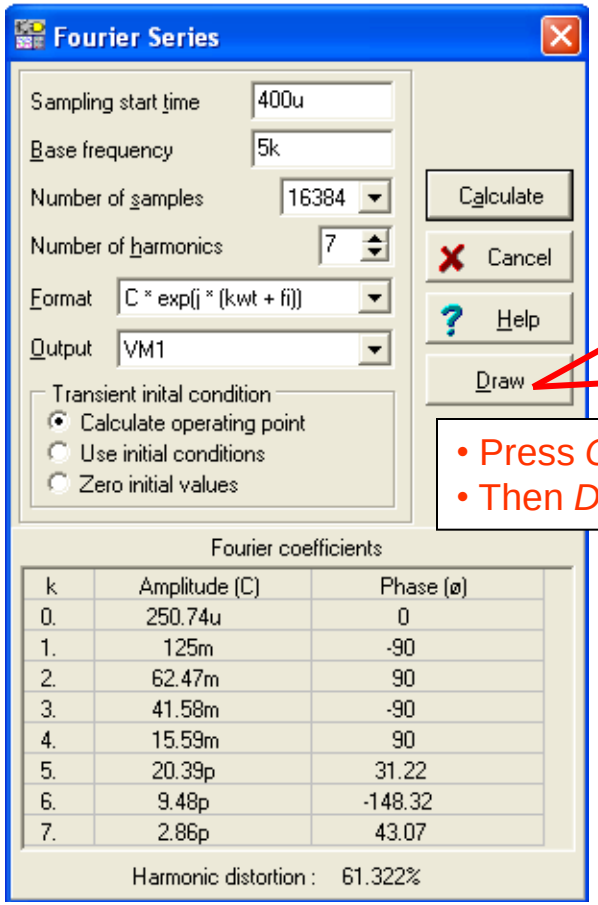
$$x(t) = \frac{2v}{\pi}\left(\sin\omega_o t - \frac{1}{2}\sin 2\omega_o t + \frac{1}{3}\sin 3\omega_o t - \frac{1}{4}\sin 4\omega_o t + \ldots\right)$$

$$where: \omega_o = \frac{2\pi}{T} \quad T = period \quad v = V_{peak}$$

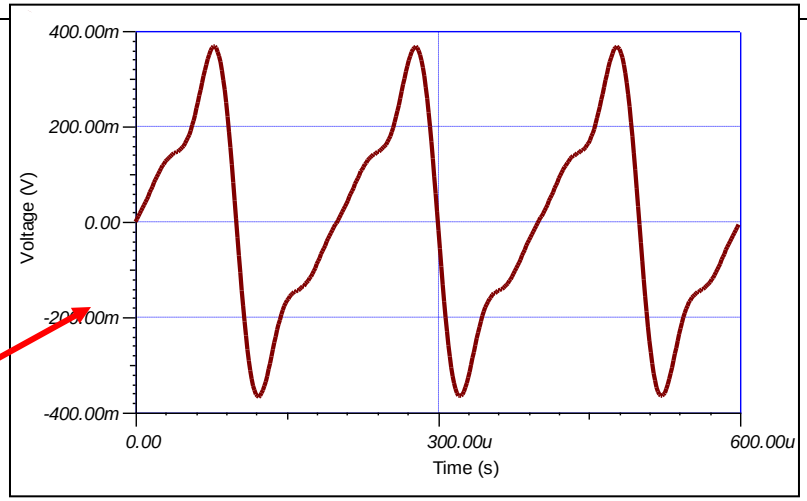
Result from TINA-TI Transient Analysis

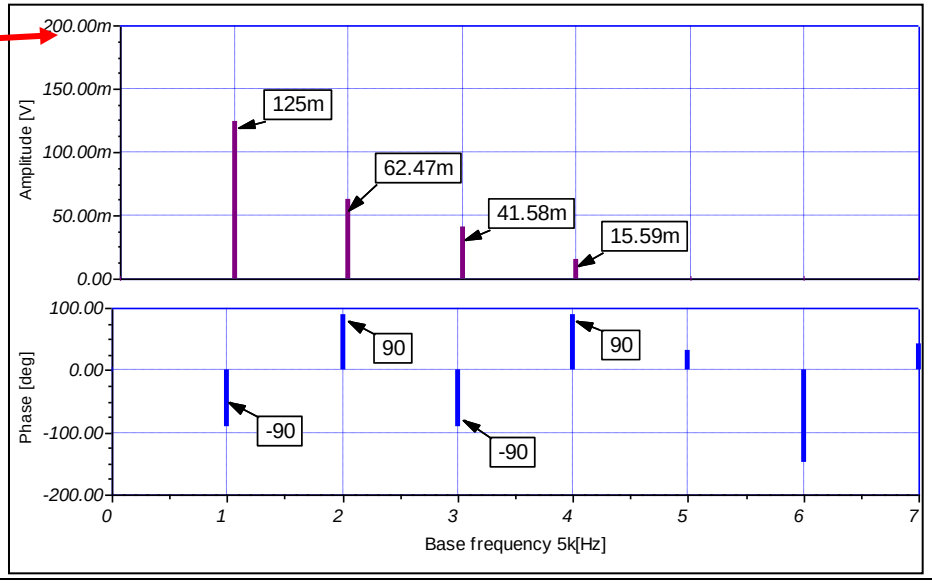
Frequency, Amplitude and Phase Fourier Analysis

Fourier Analysis > Fourier Series

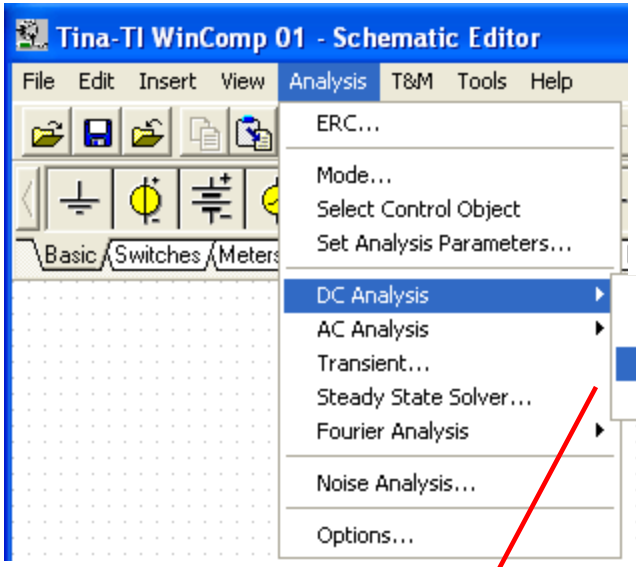


- Press *Calculate*
- Then *Draw*



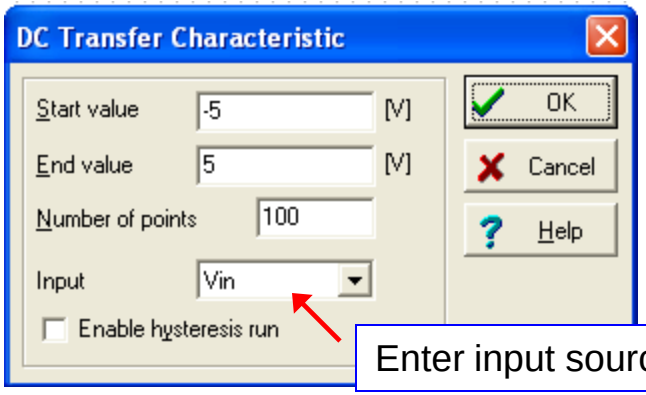


DC Analysis –
DC Sweep With a Window Comparator

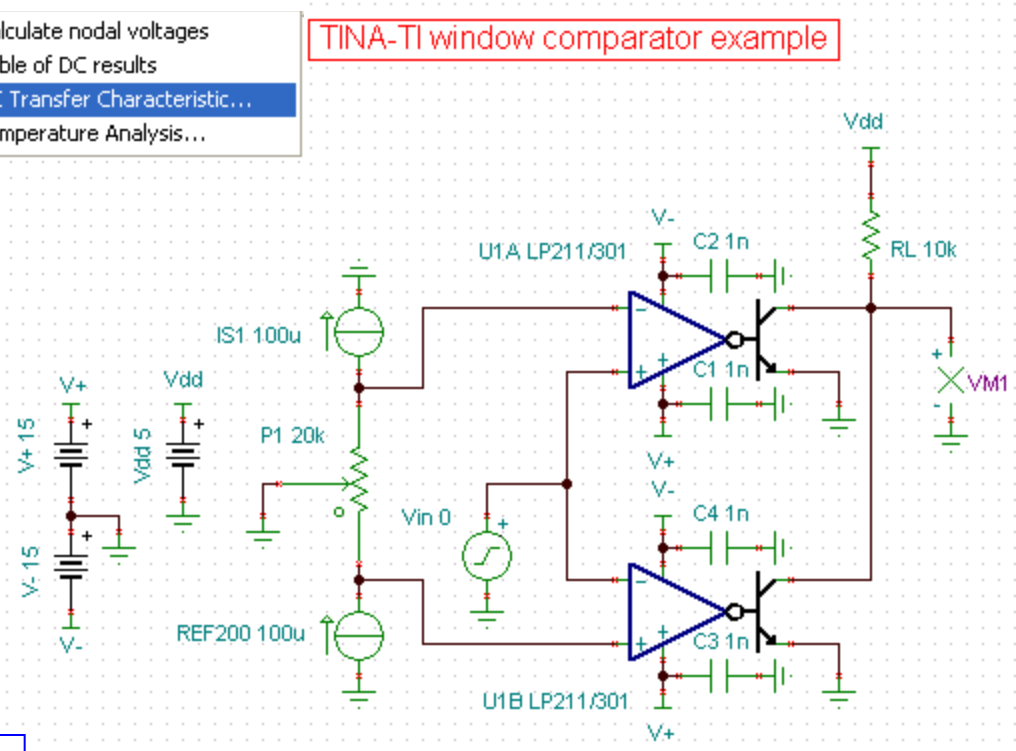


Select: *Analysis > DC Analysis > DC Transfer…*

Enter start and end sweep voltages

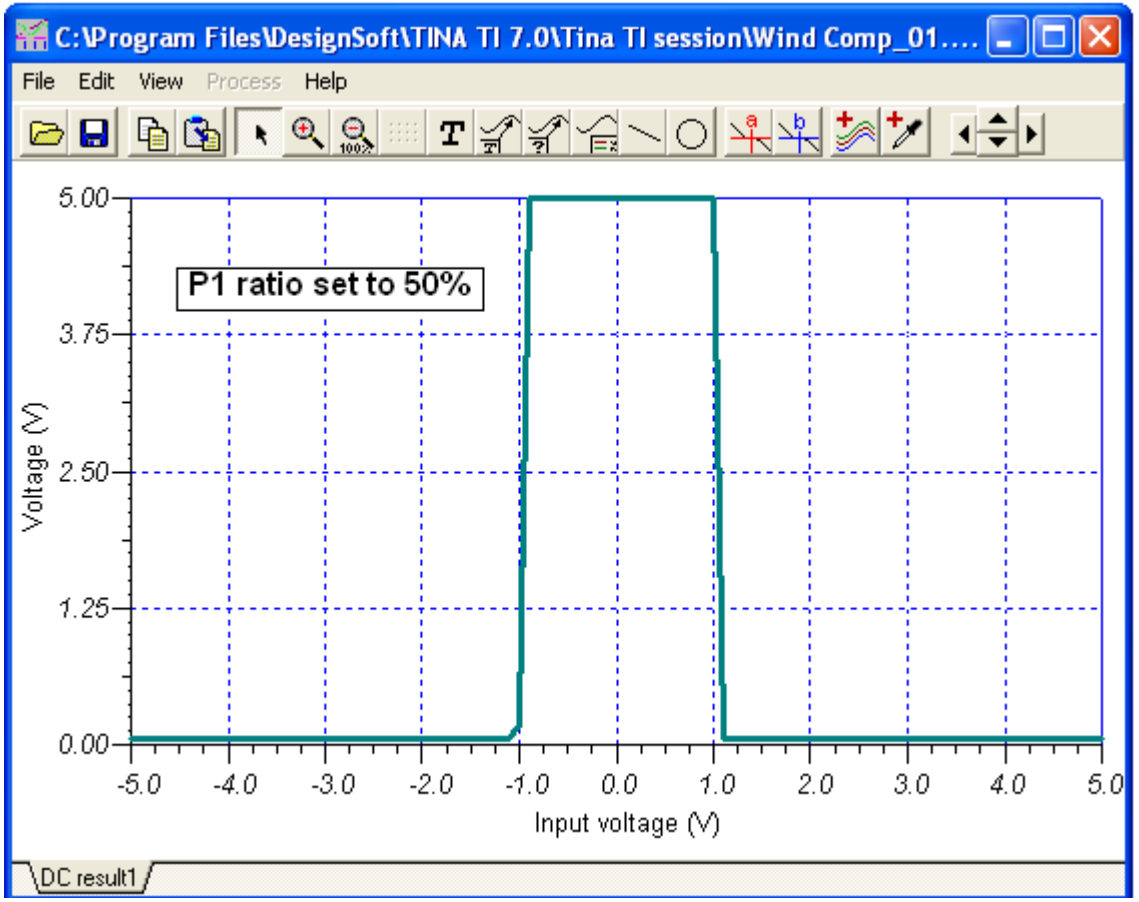


Enter input source



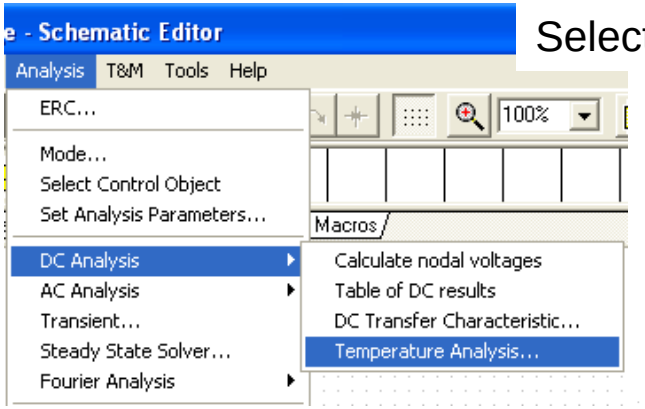
Comparator Output Response vs. Input Source

Vertical scale: The output response



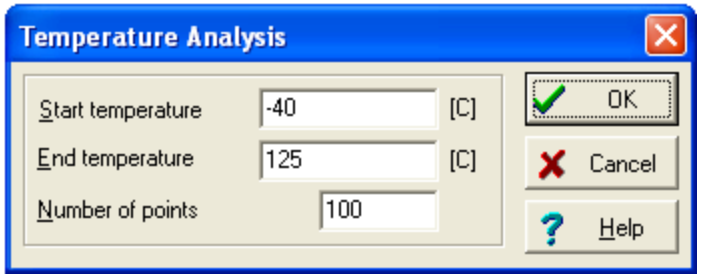
Horizontal scale: The input source, V_{IN}, is swept from -5V to +5V

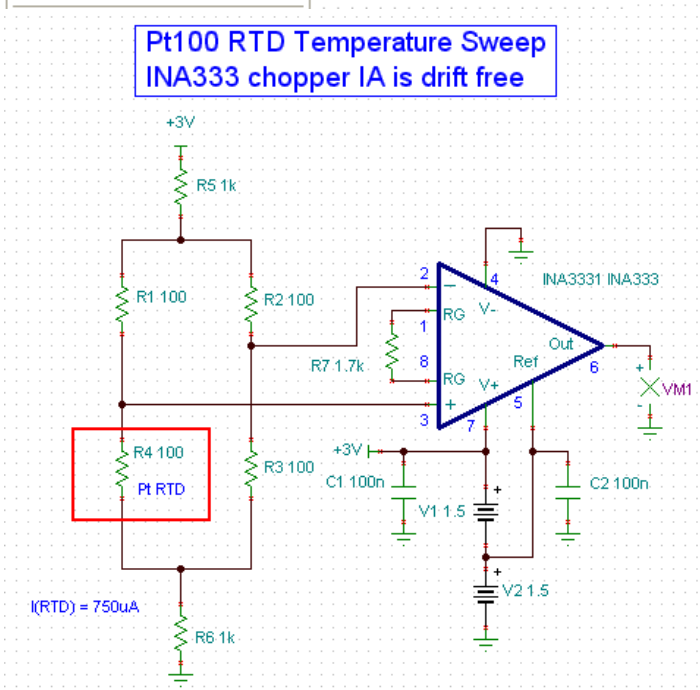
DC Analysis: Temperature Analysis

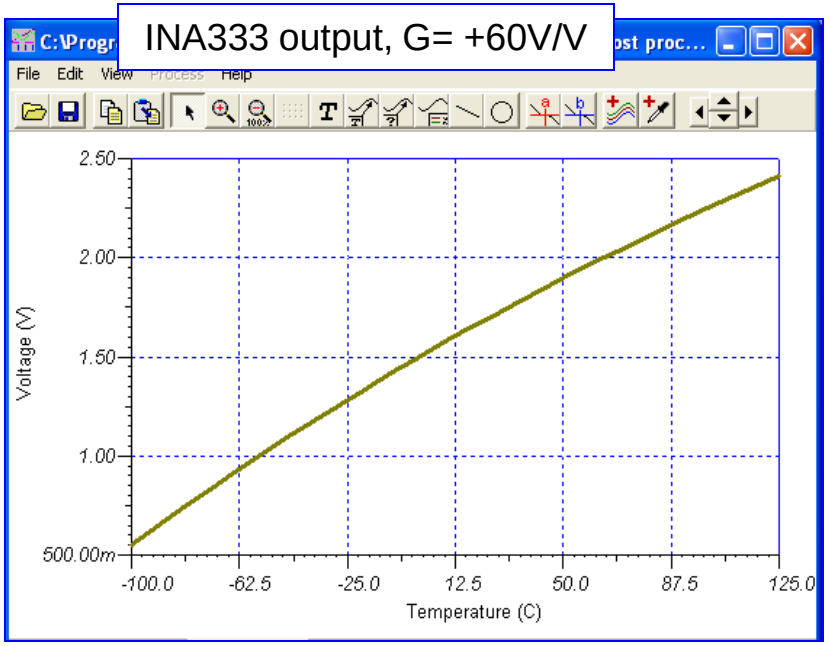


Select: *Analysis > DC Analysis >Temperature Analysis*

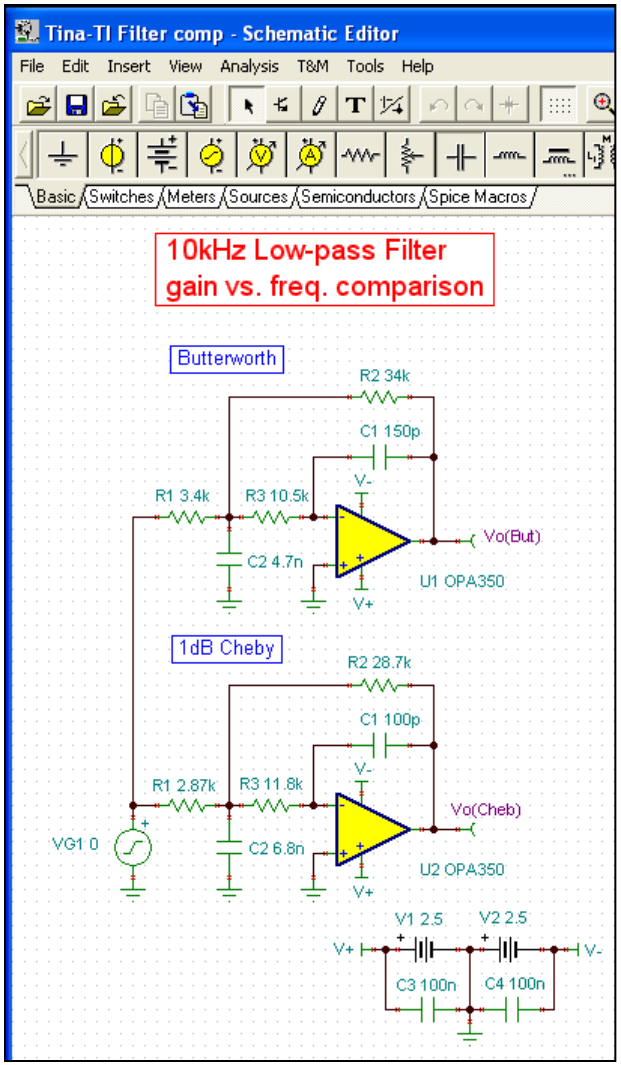
Enter *Start* and *End* temperatures





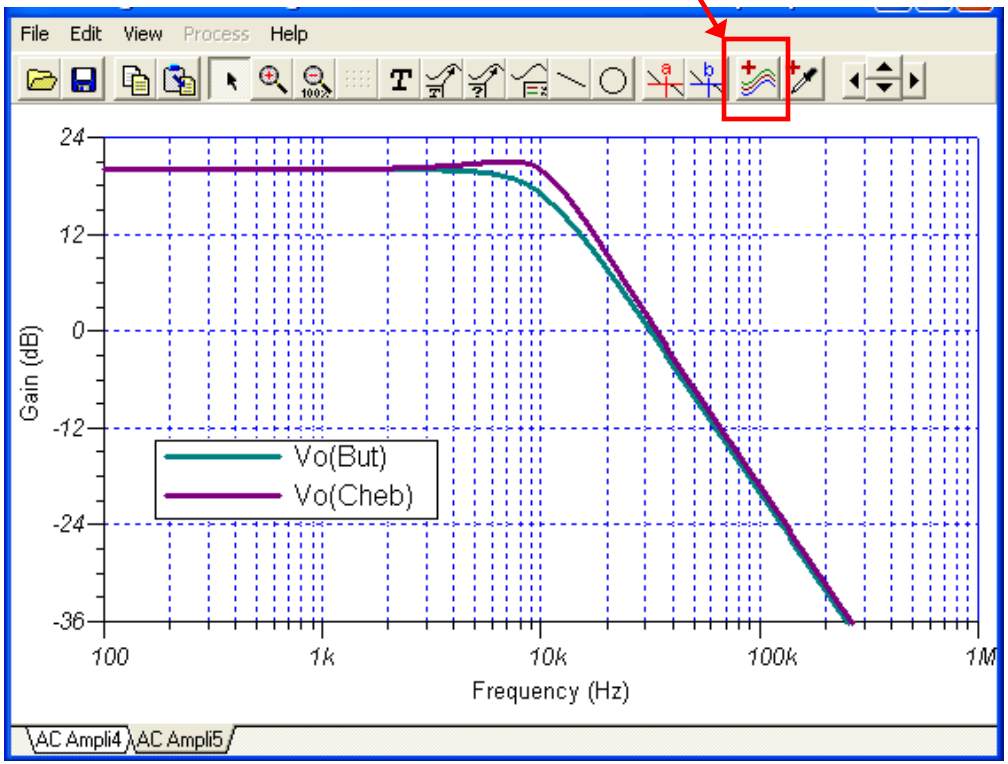


Post Processing Analysis

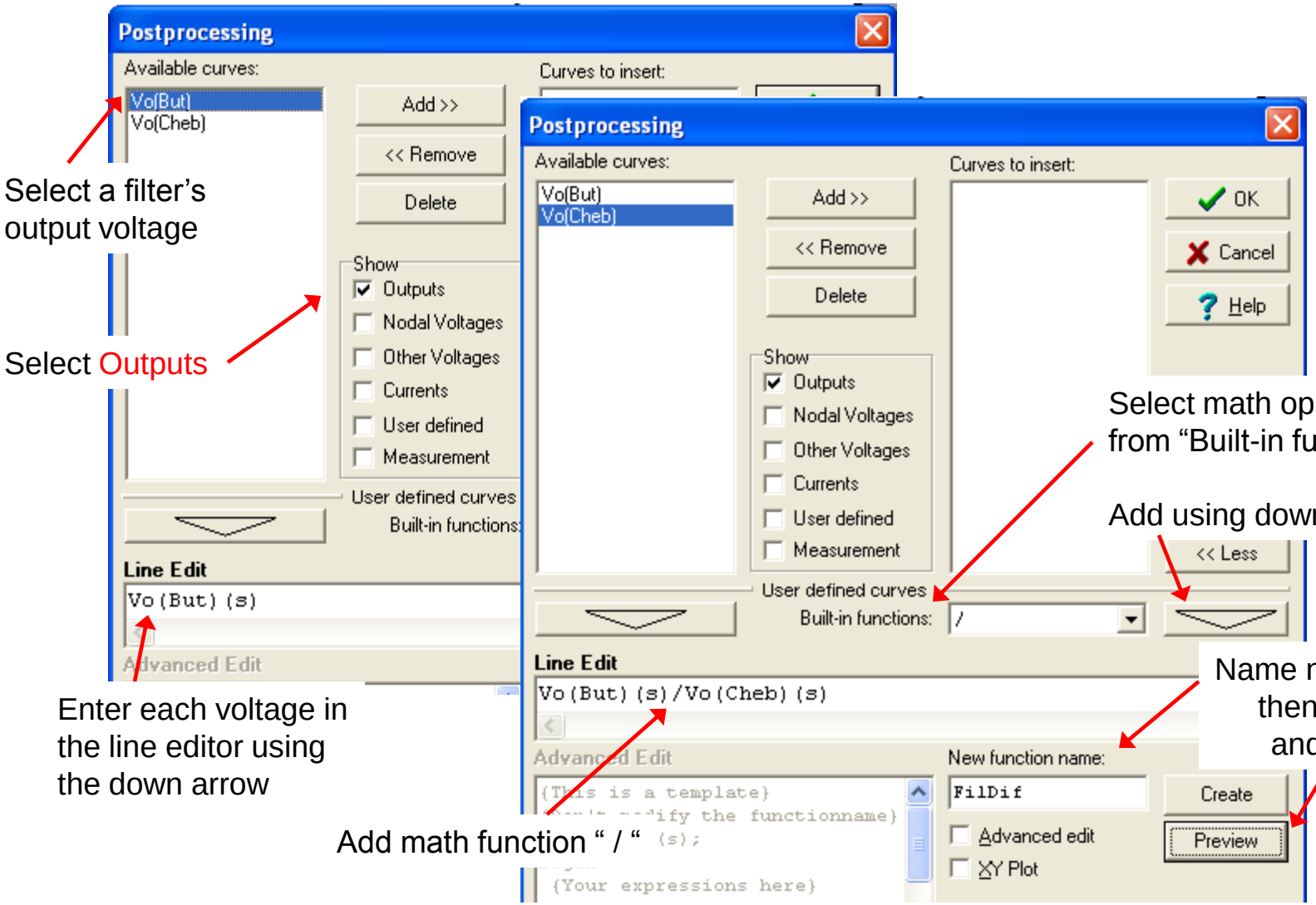


Goal: Find the difference in the two filter gains across frequency

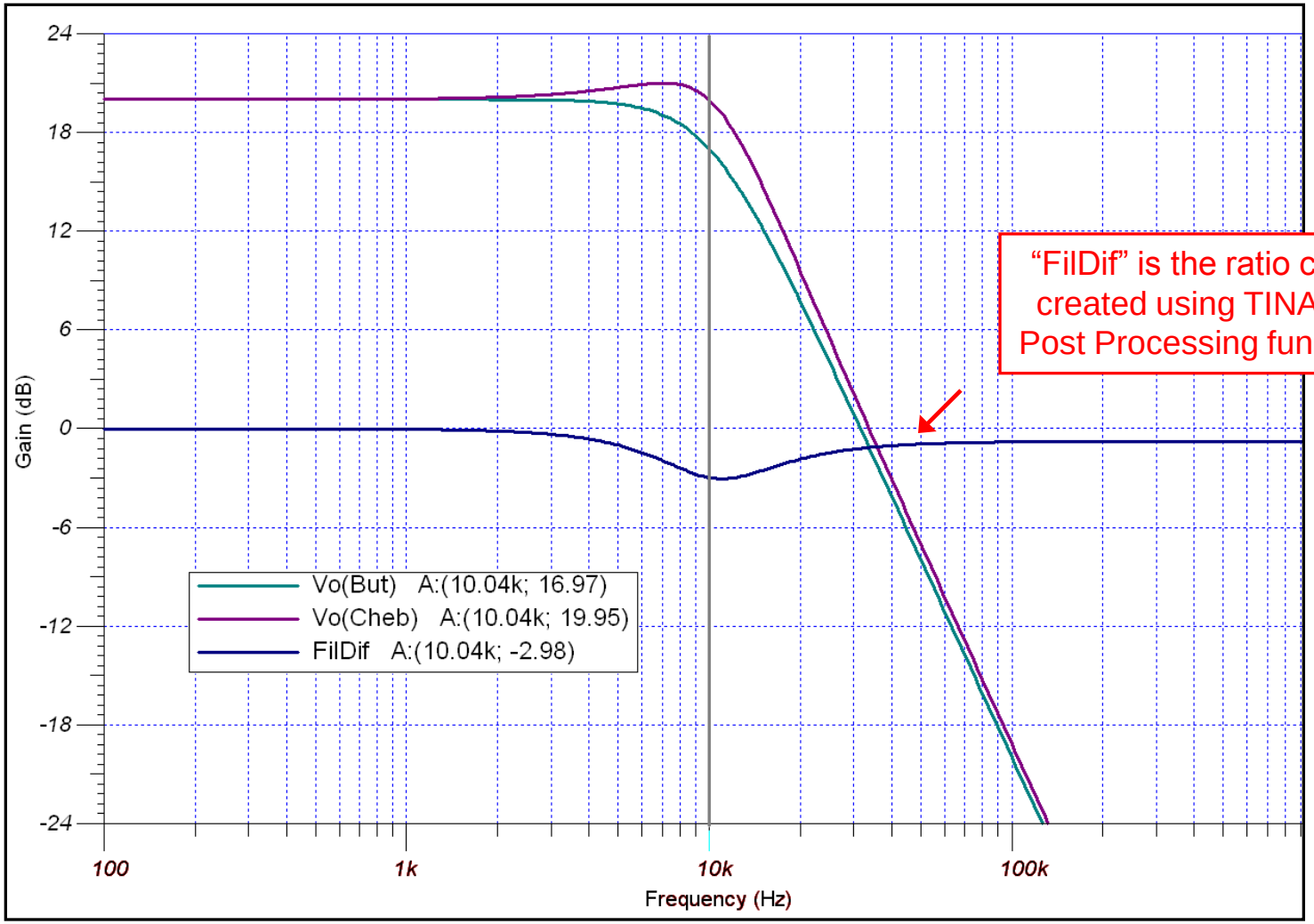
Select the *Post Processing* function icon



Math Tool Applied after an Analysis



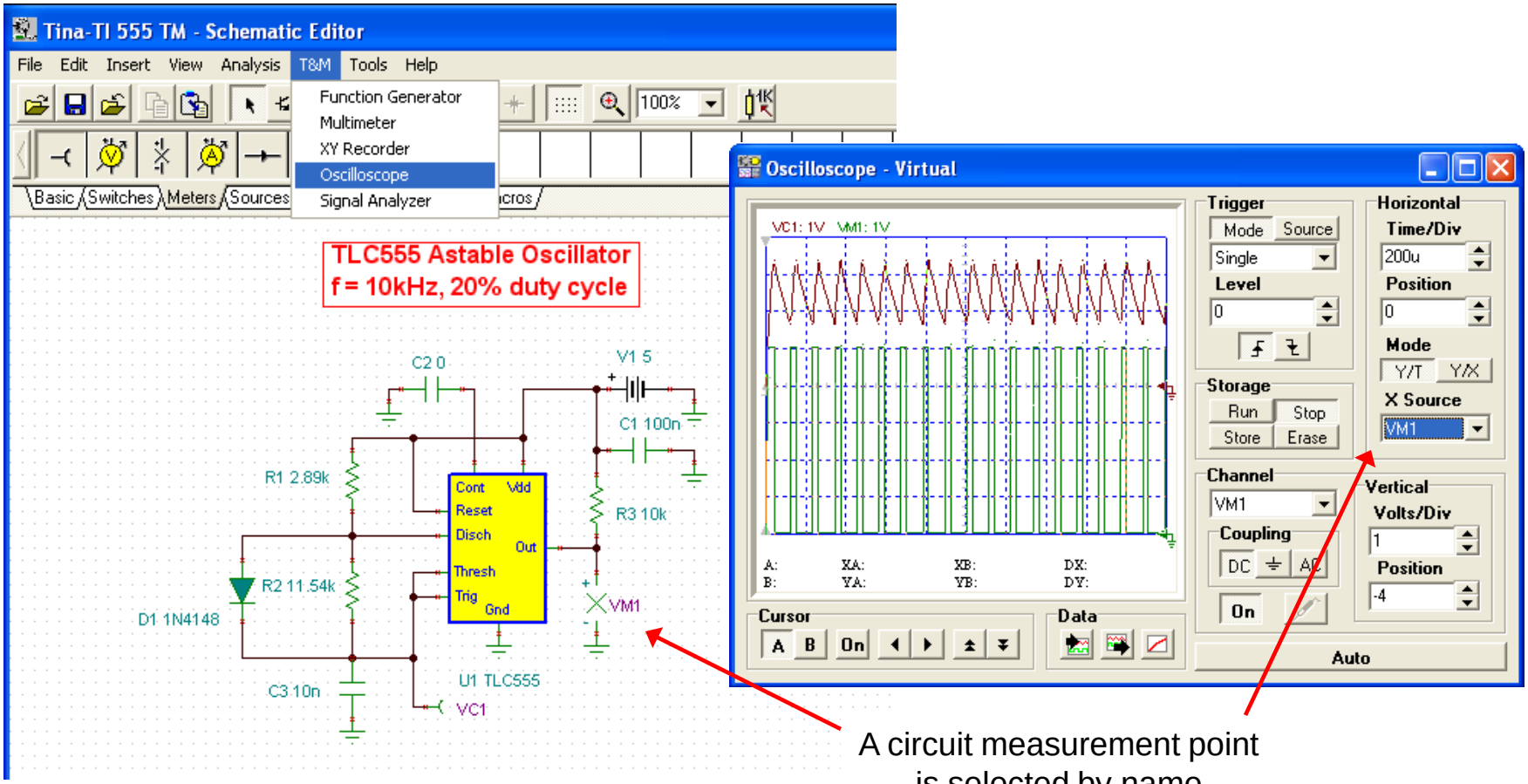
Ratio of Two Curves



"FilDif" is the ratio curve created using TINA-TI's Post Processing function

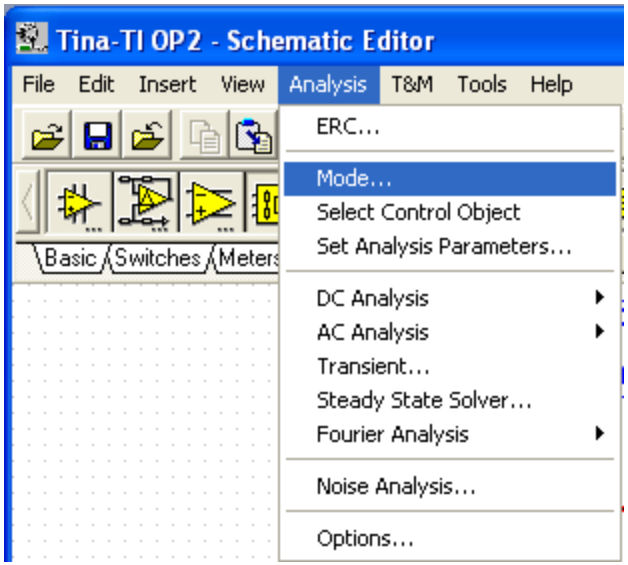
Test & Measurement: Using Virtual Instruments

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

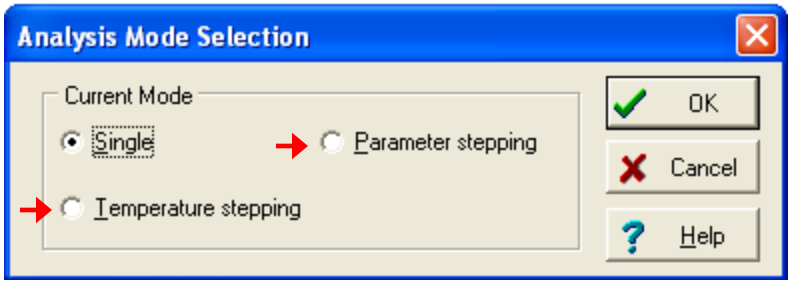


A circuit measurement point is selected by name

Temperature and Parameter Stepping



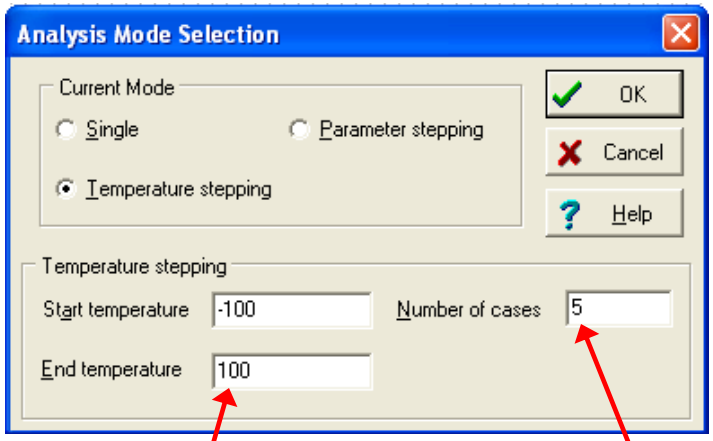
Select: *Analysis > Mode*



Select: *Temperature stepping or Parameter stepping*

Temperature Stepping Applied to a DC Sweep

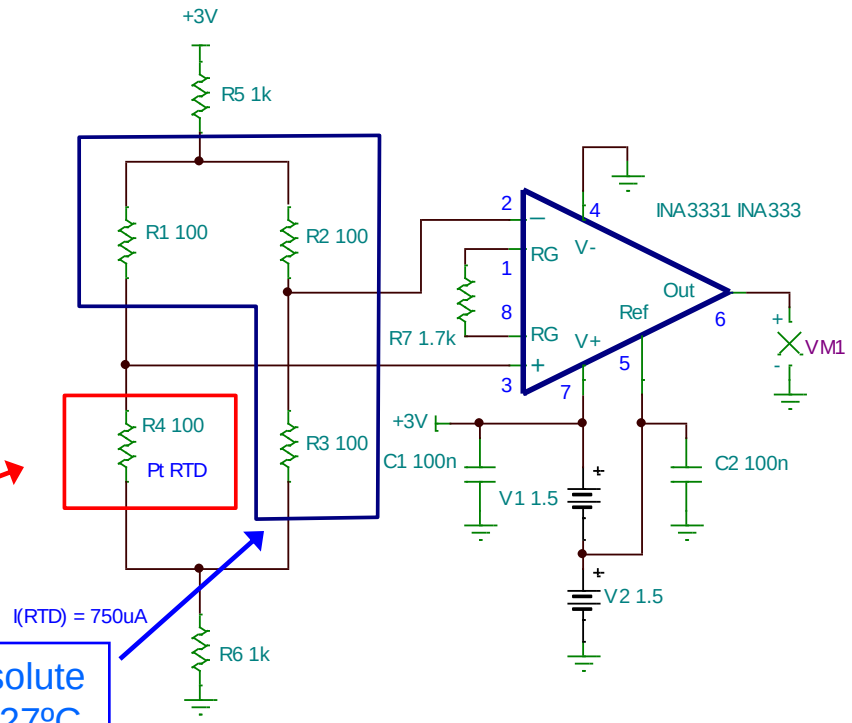
Temperature Stepping is selected



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

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

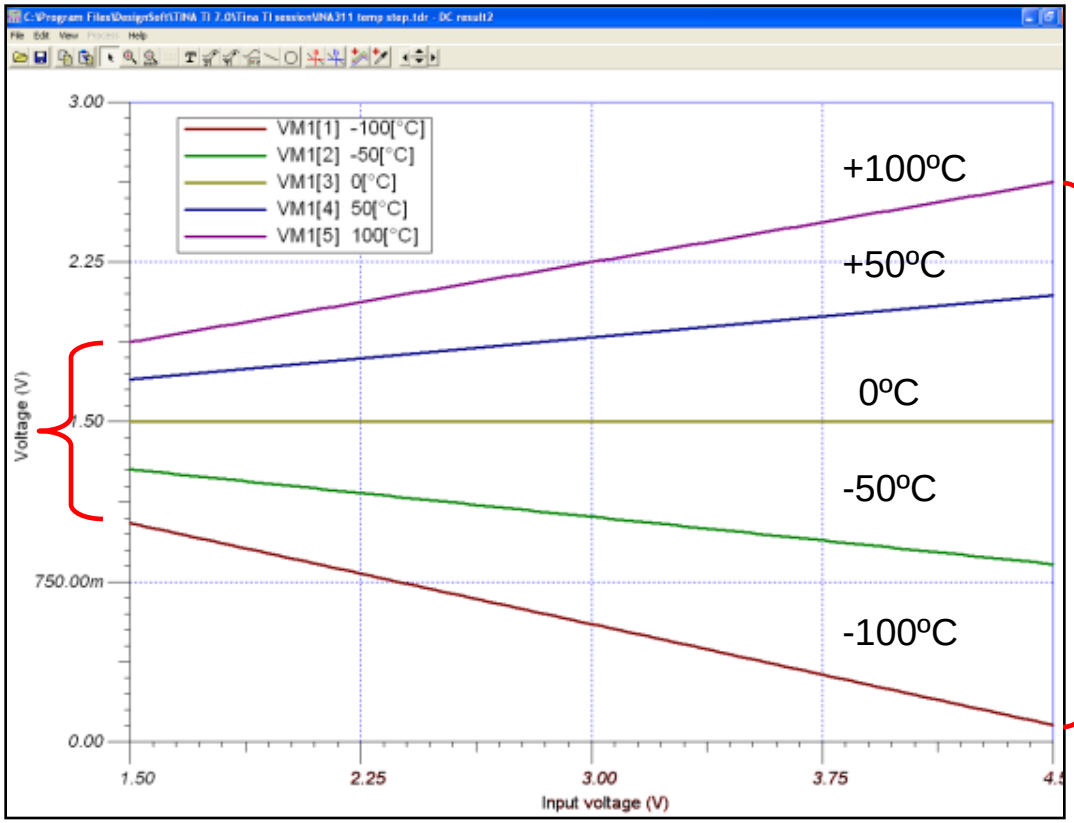


Set RTD Relative Temperature to 0ºC

Set R1– R3 Absolute Temperature to 27ºC

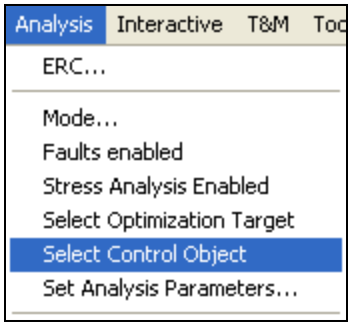
Voltage Out vs. Temperature vs. Input Voltage

Vm1 output span with V_{S1} at 1.5V



Vm1 output span with V_{S1} at 4.5V

Parameter Stepping: DC and AC Example

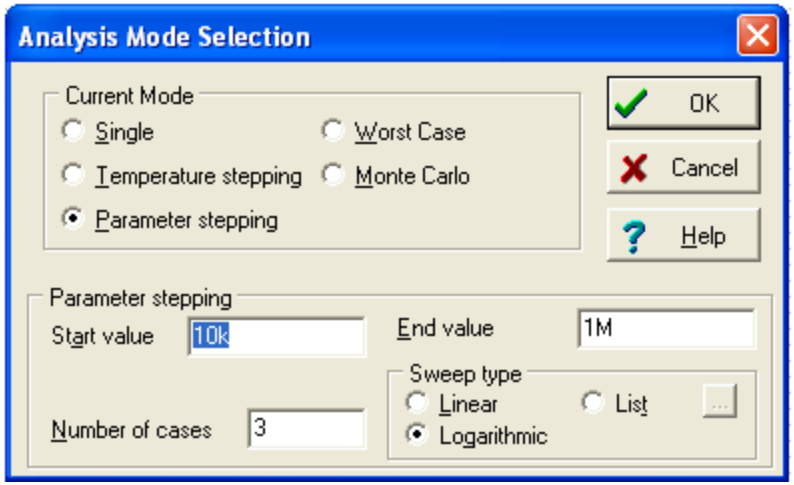


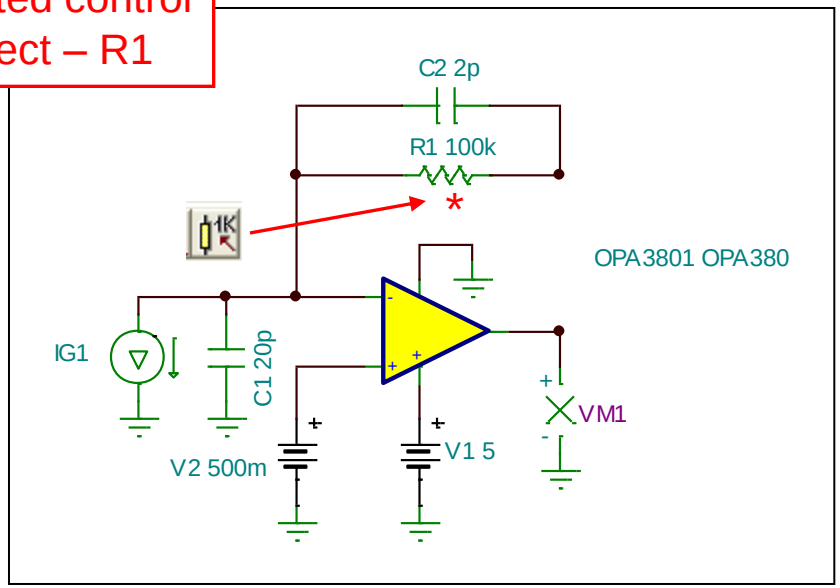
OPA380 Transimpedance Amplifier

R1 is Stepped to Affecting Gain and AC Response

Selected control object – R1

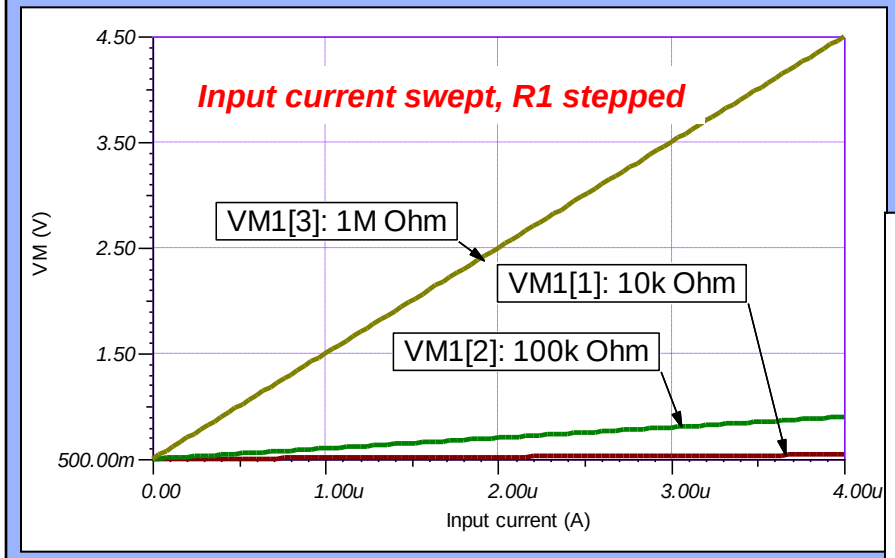
or



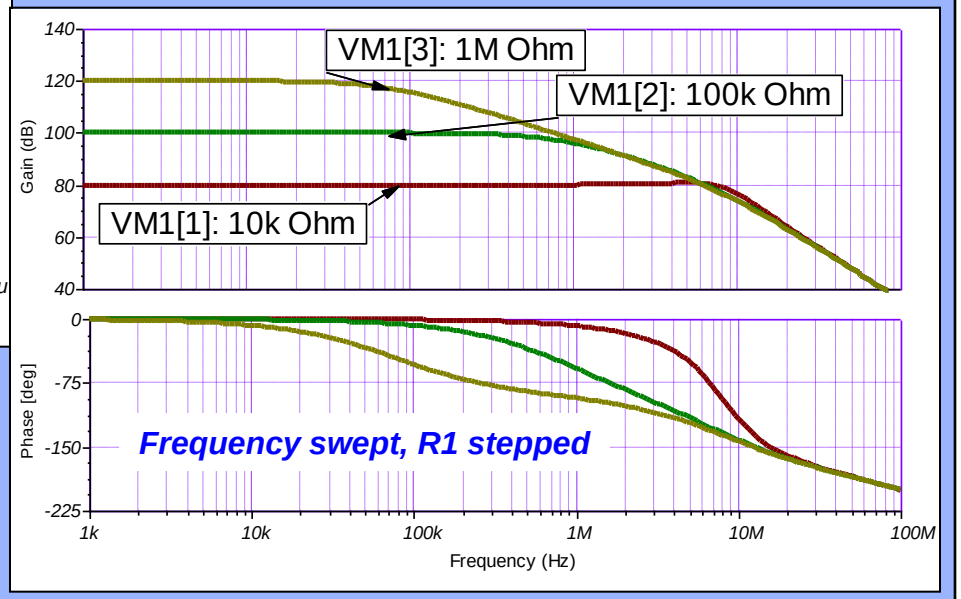


Parameter Stepping: DC and AC Example

OPA380 transimpedance amplifier example
R1 Log stepped 10kΩ, 100kΩ, 1MΩ

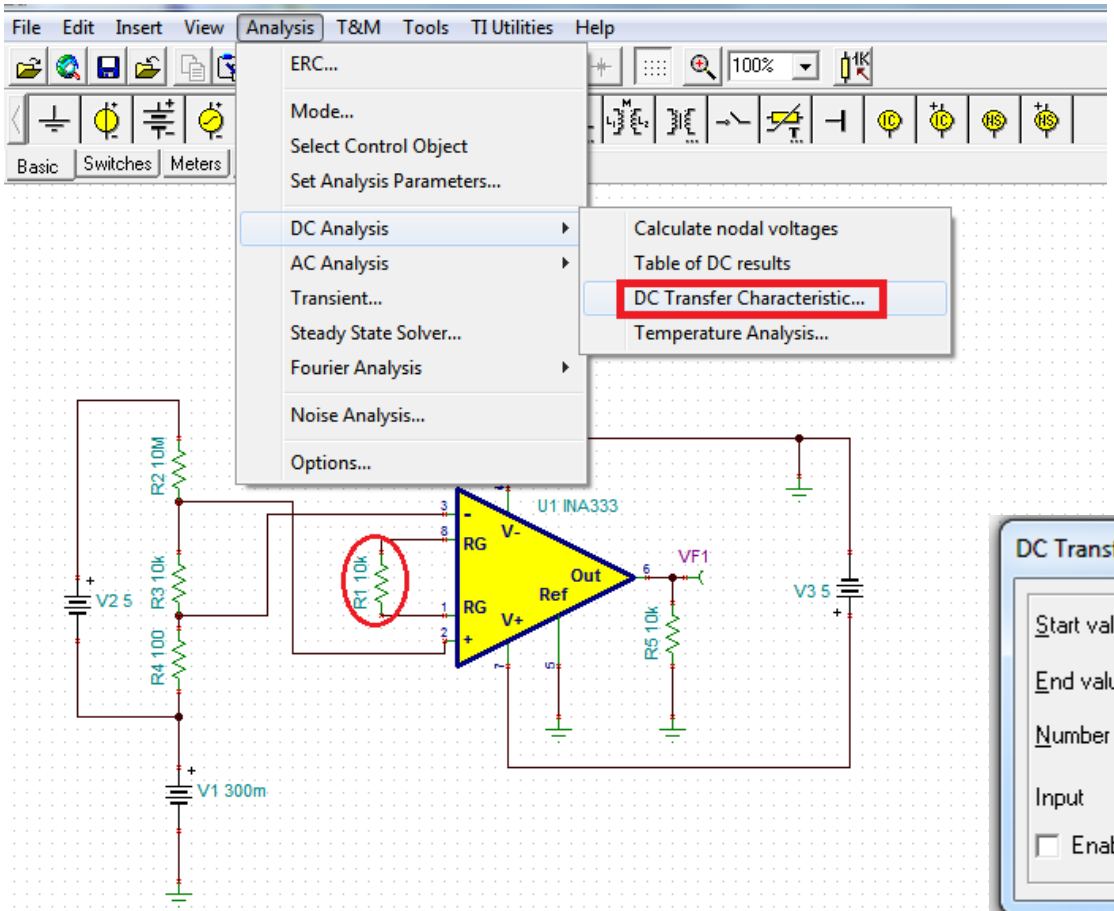


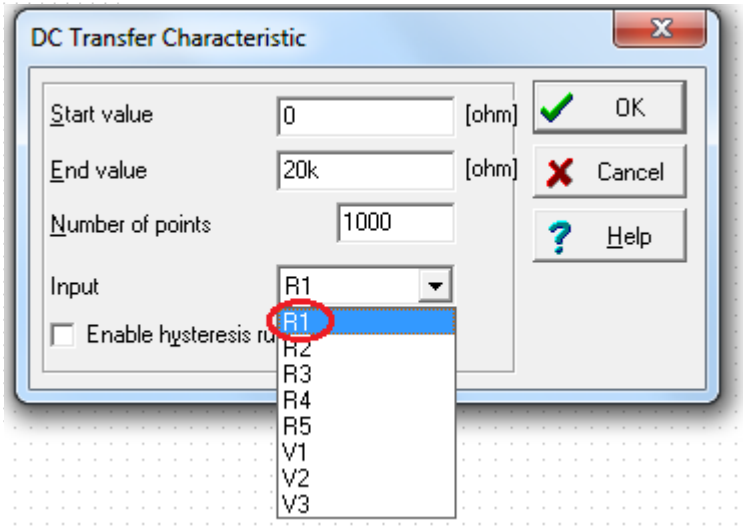
Select: *AC Analysis > AC transfer characteristic*



Select: *DC Analysis > DC transfer characteristic*

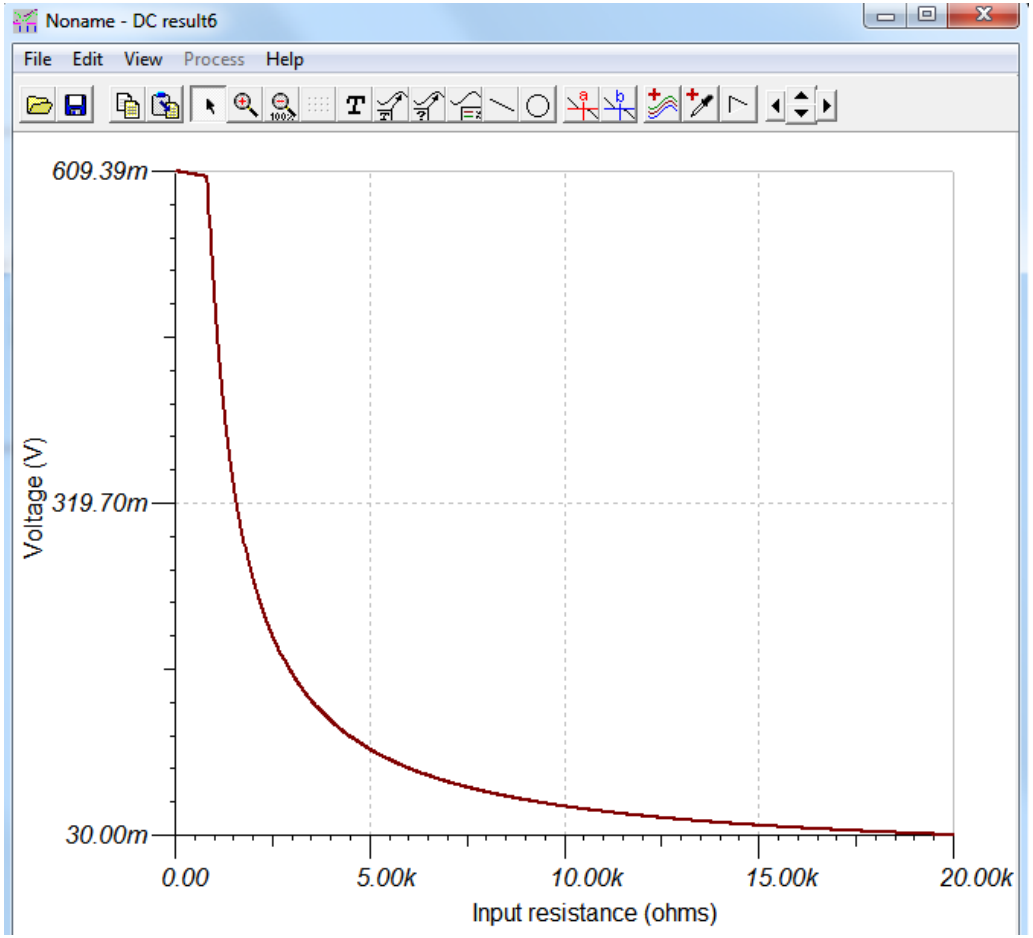
DC transfer for parameter stepping



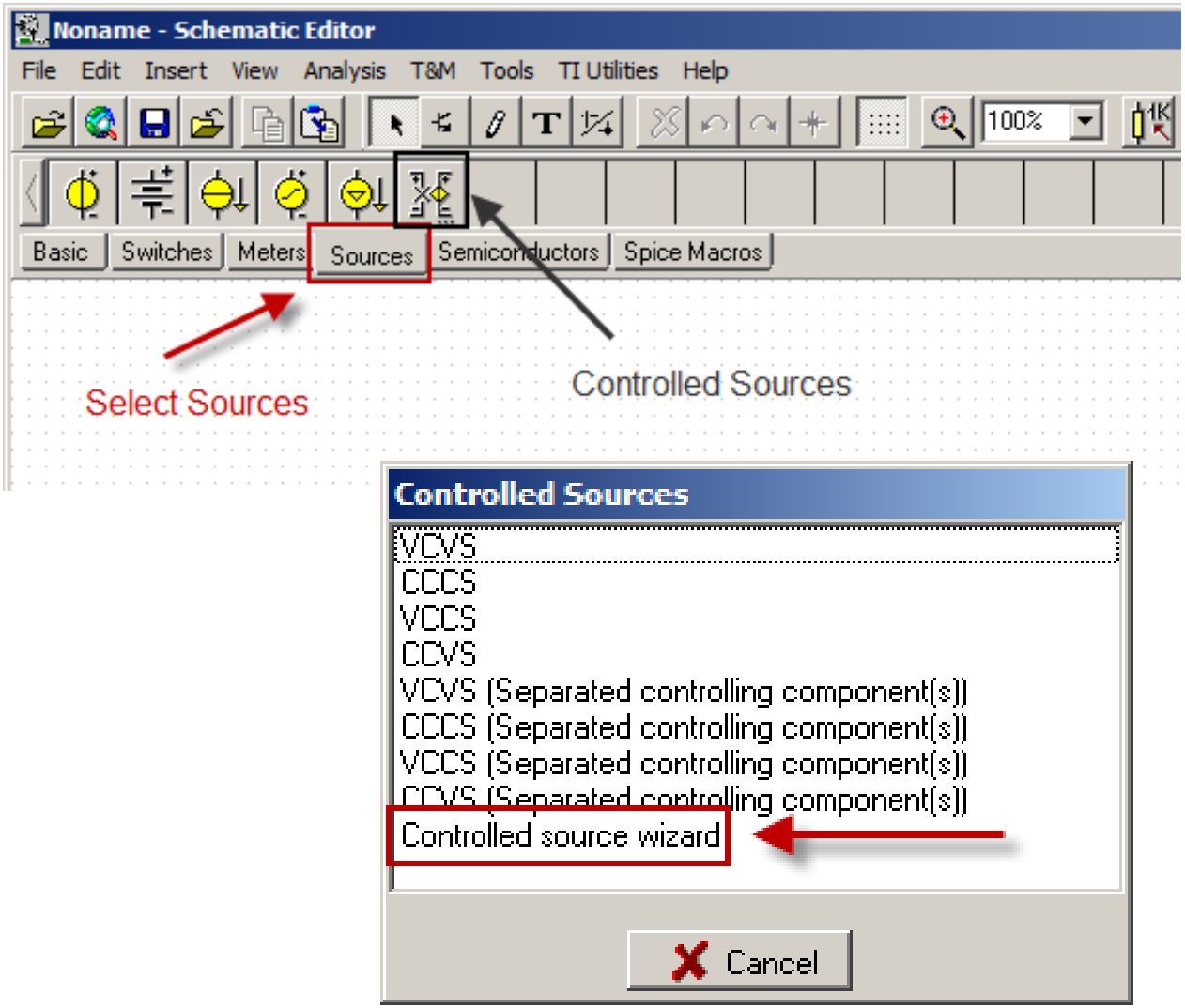


DC transfer for parameter stepping

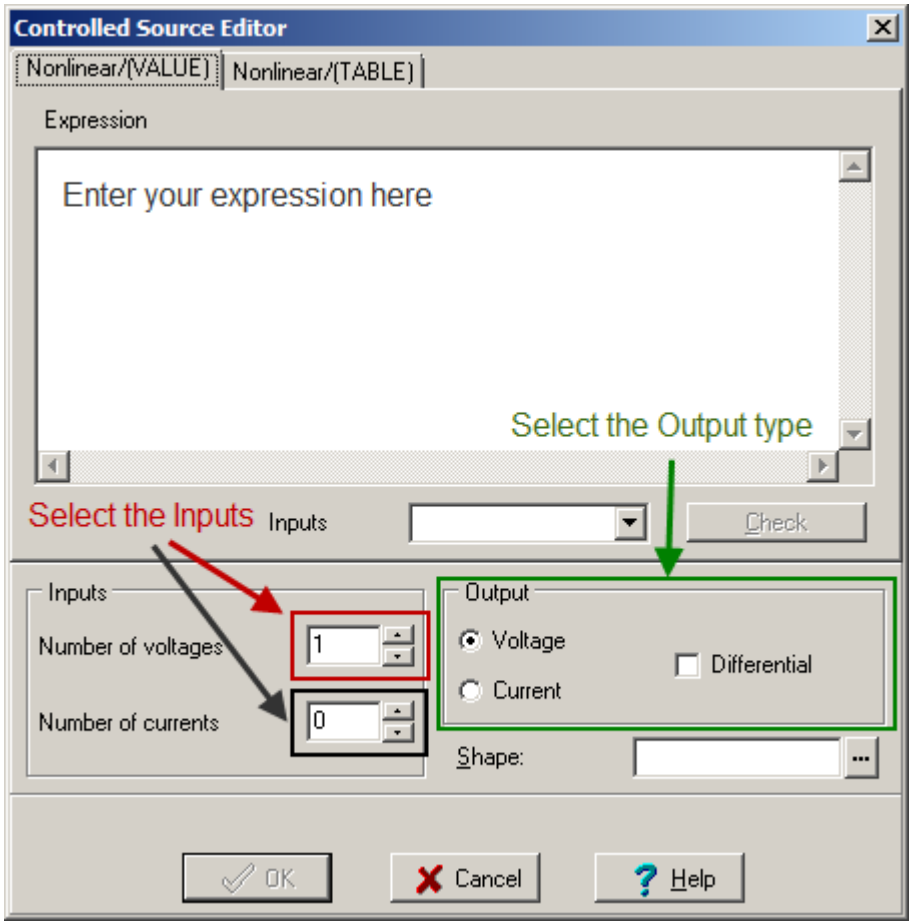
The result of R1 stepping



Controlled Source Wizard



Controlled Source Wizard

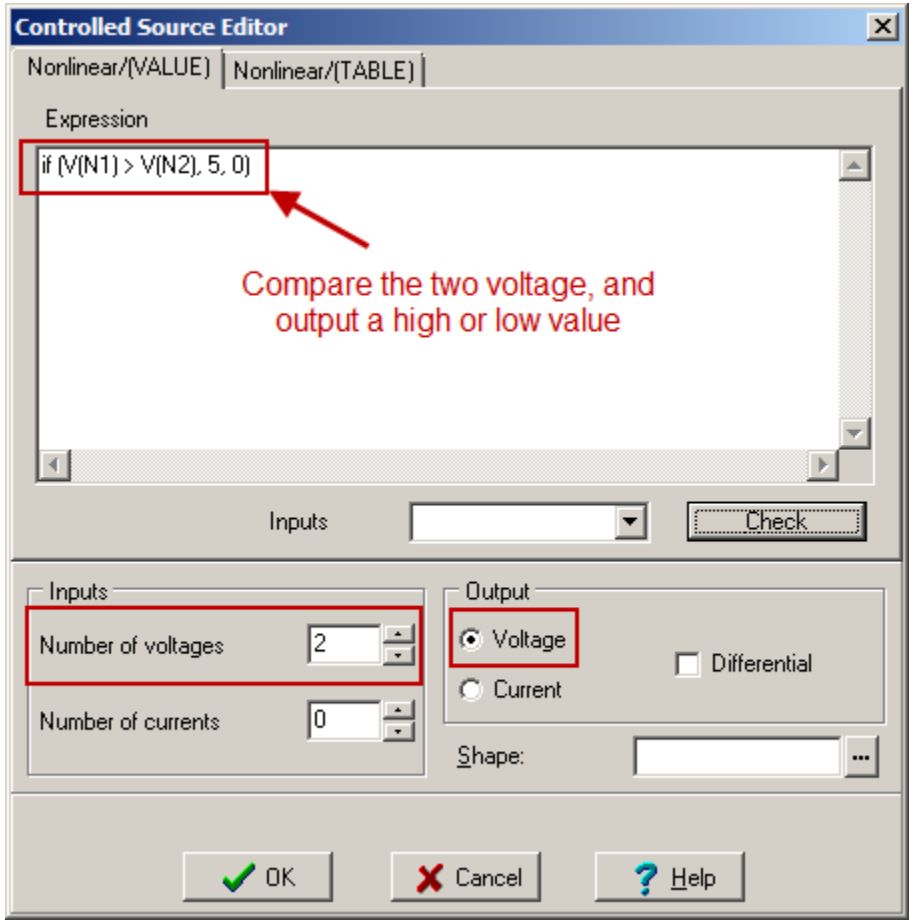


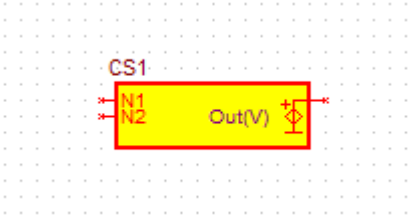
Expressions are entered in the dialog area. The expression will relate the inputs to the outputs.

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

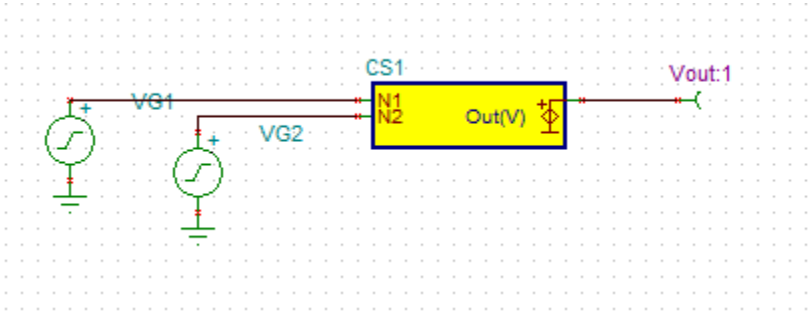
In the Table mode, the block uses input and output data pairs.

Controlled Source Wizard (Comparator)

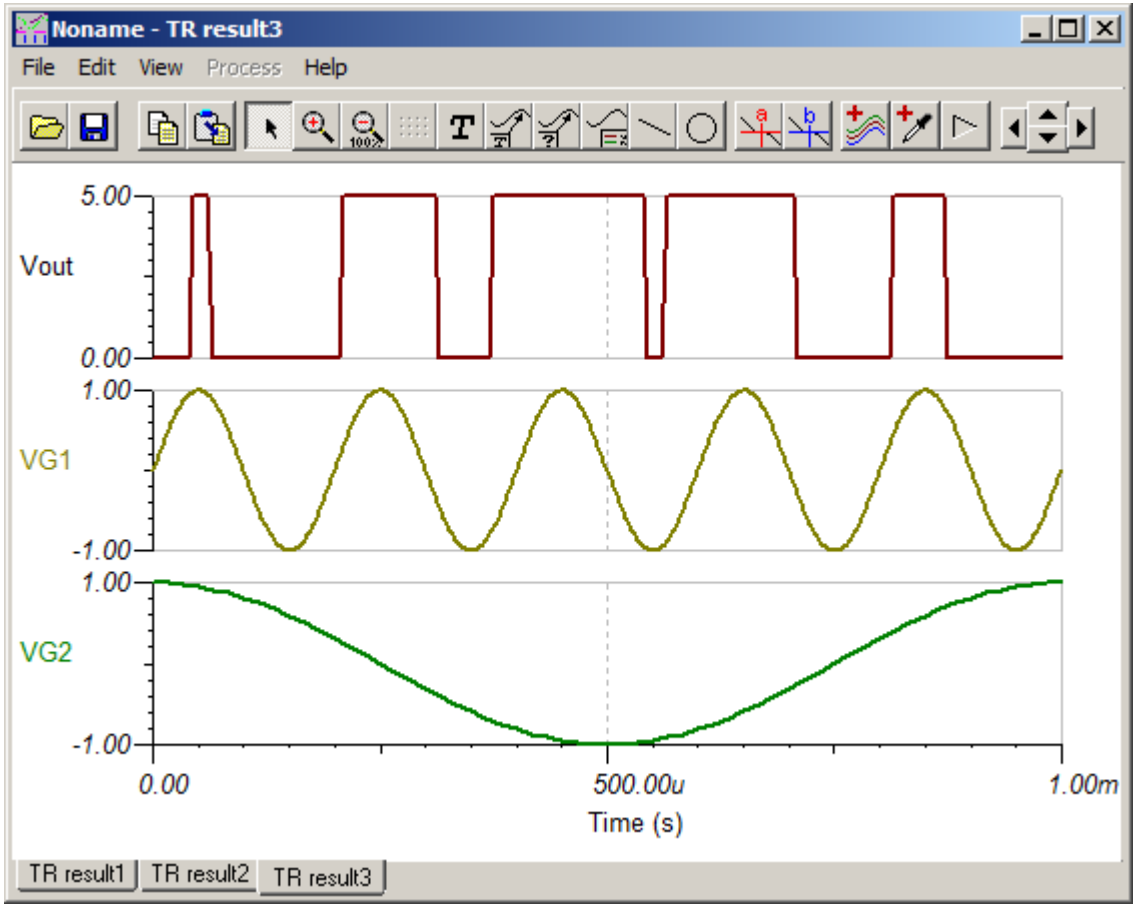




Place the controlled source and connect the two voltages and monitor the output.



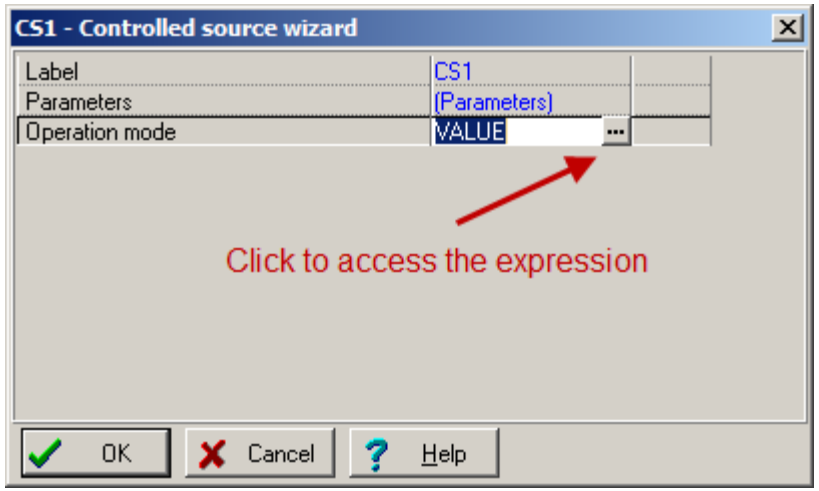
Controlled Source Wizard (Comparator)

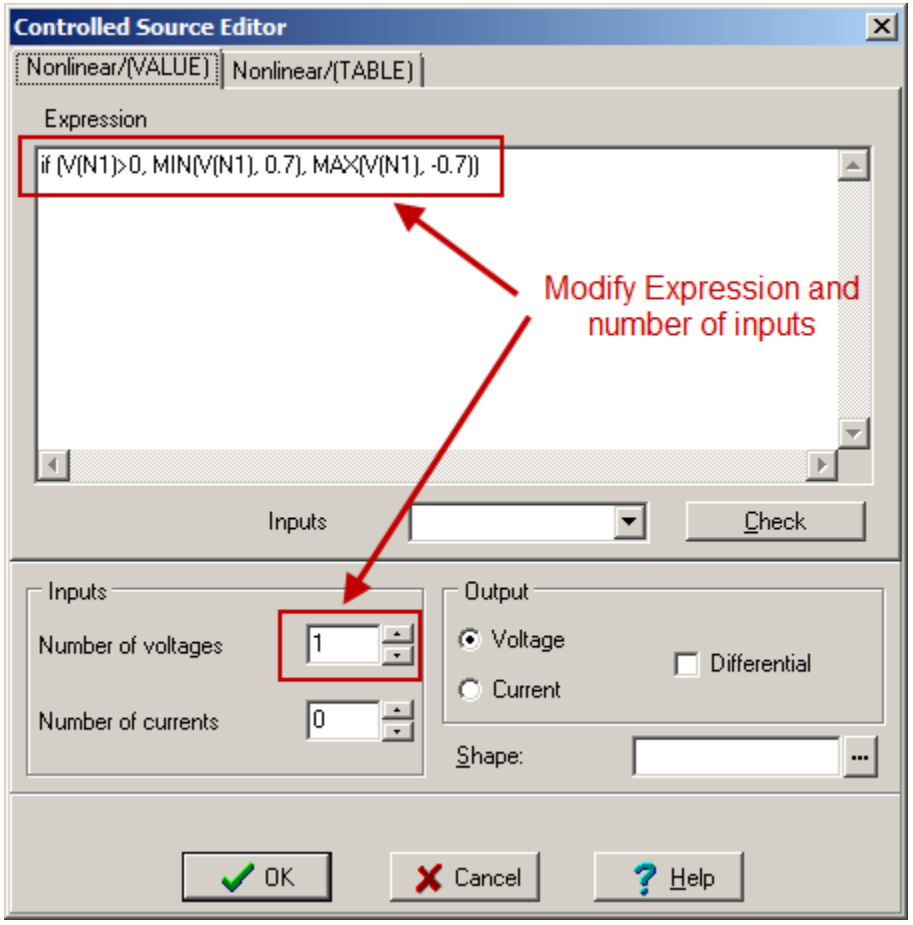


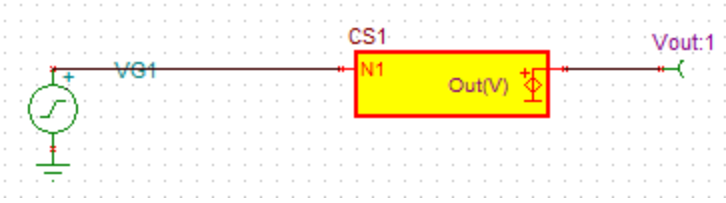
The output is the comparison of the two different frequency sine and cosine waves.

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

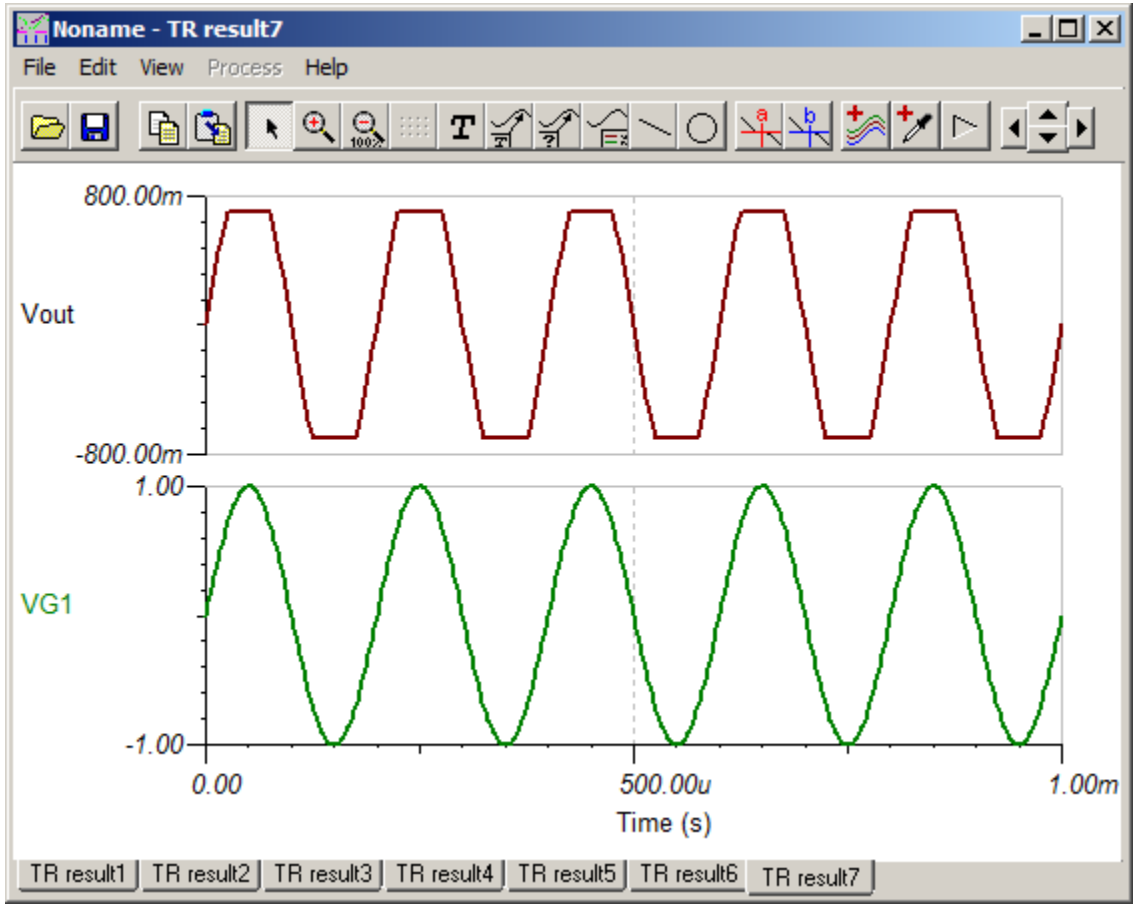
Controlled Source Wizard (Limiter)







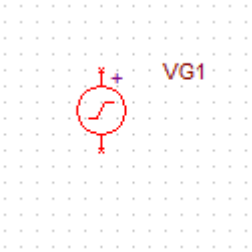
Controlled Source Wizard (Limiter)

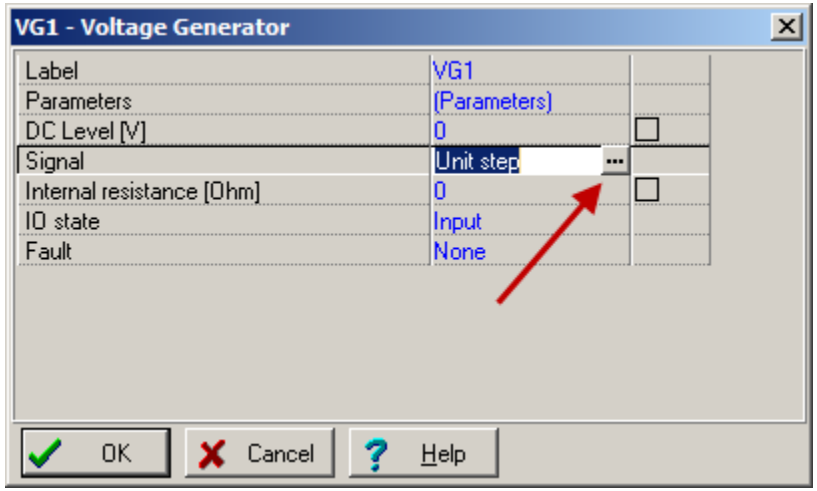


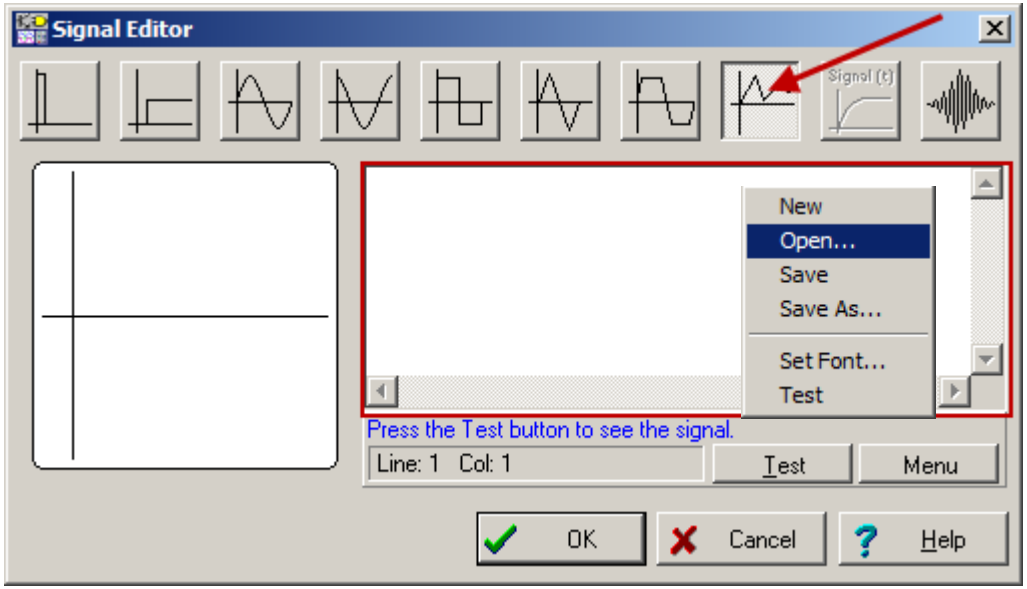
The voltage is now limited between 0.7 and -0.7.

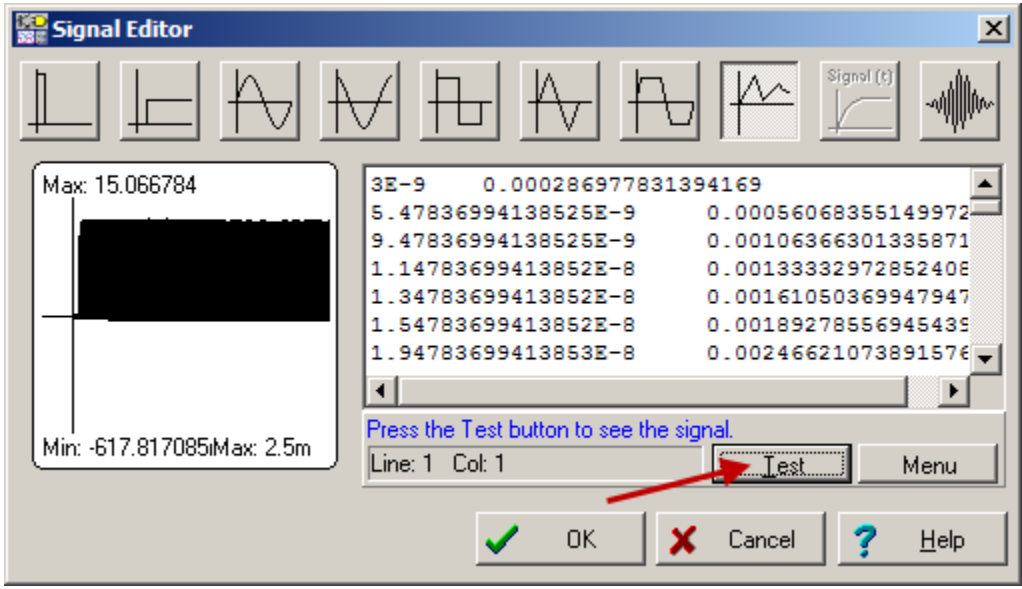
Note that the voltage between the limits is not effected. It could be modified again by adding another controlling block or by use of a more complicated expression.

Piecewise Linear Source from file



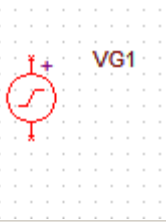




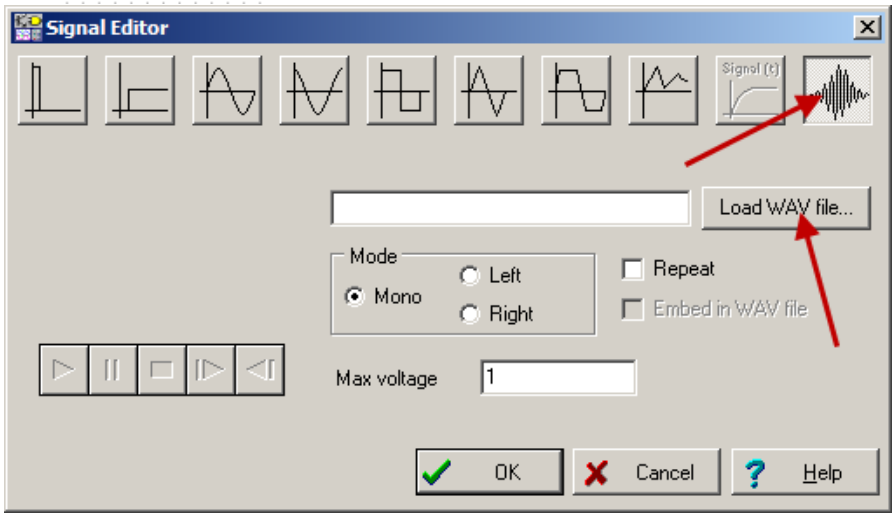


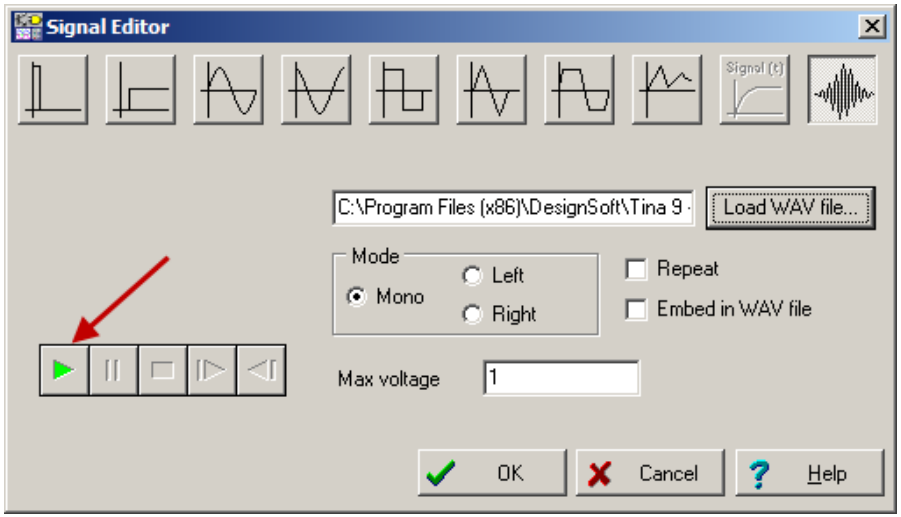
Navigate to the file location and load it into the Signal Editor. This may take a few seconds for a large file. Click Test to see the waveform.

WAV file used as an input

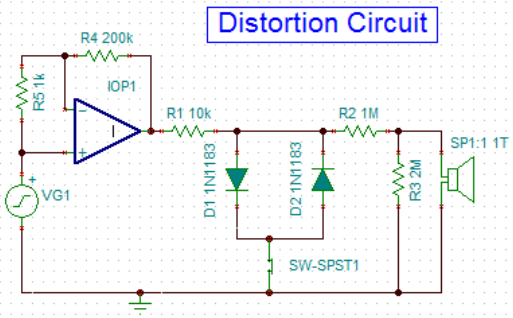


Once the WAV file is selected, navigate to the location for the .wav file to be used. Once loaded, the play arrow will become green and you can test it if you wish.





WAV file used as an input - Example

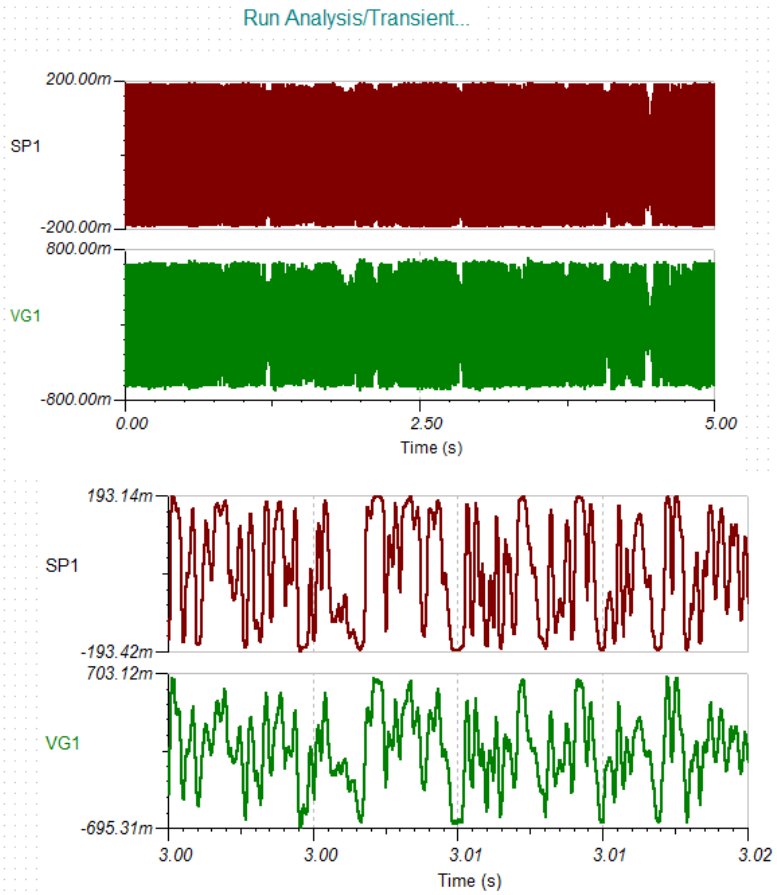


With the settings of this distortion circuit you can not only see the waveforms but also listen to the effect of distortion.

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

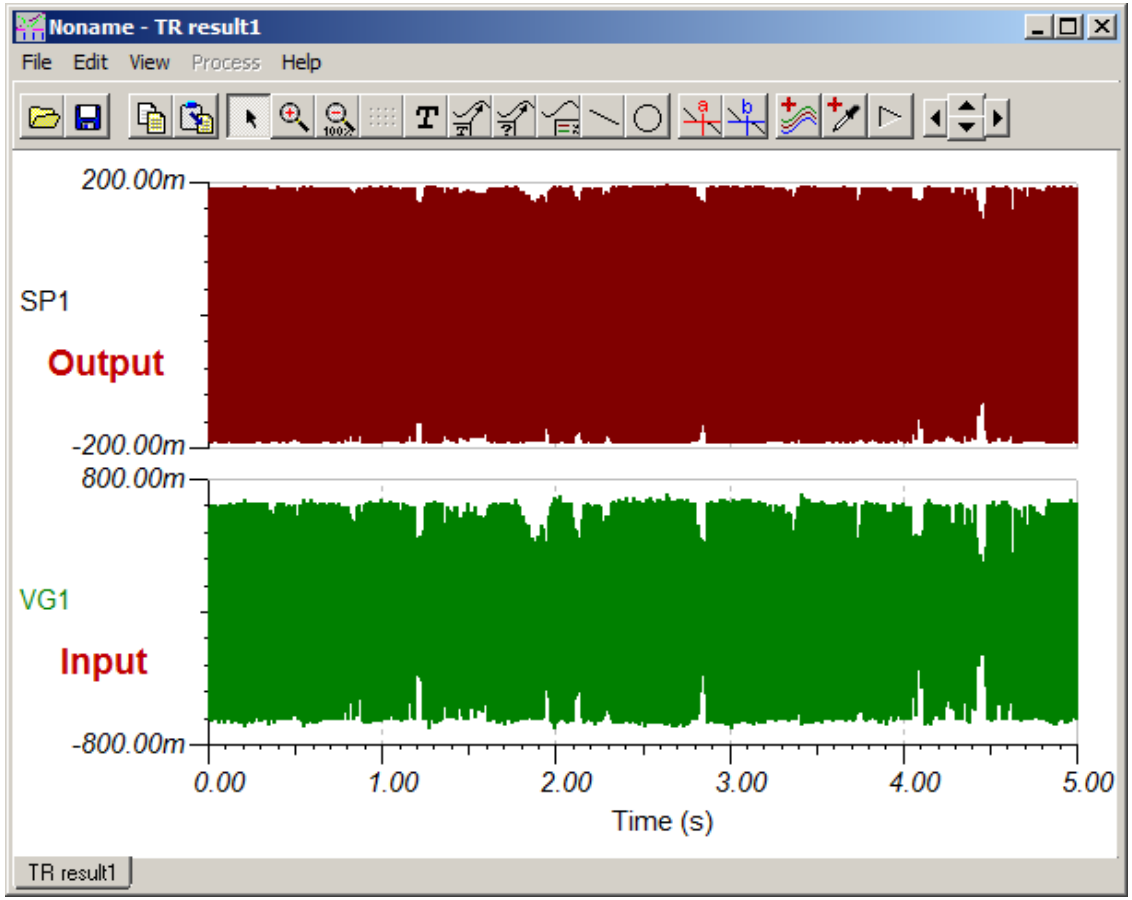
Press the A key on the keyboard or change the switch to disable or enable distortion.

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.



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.

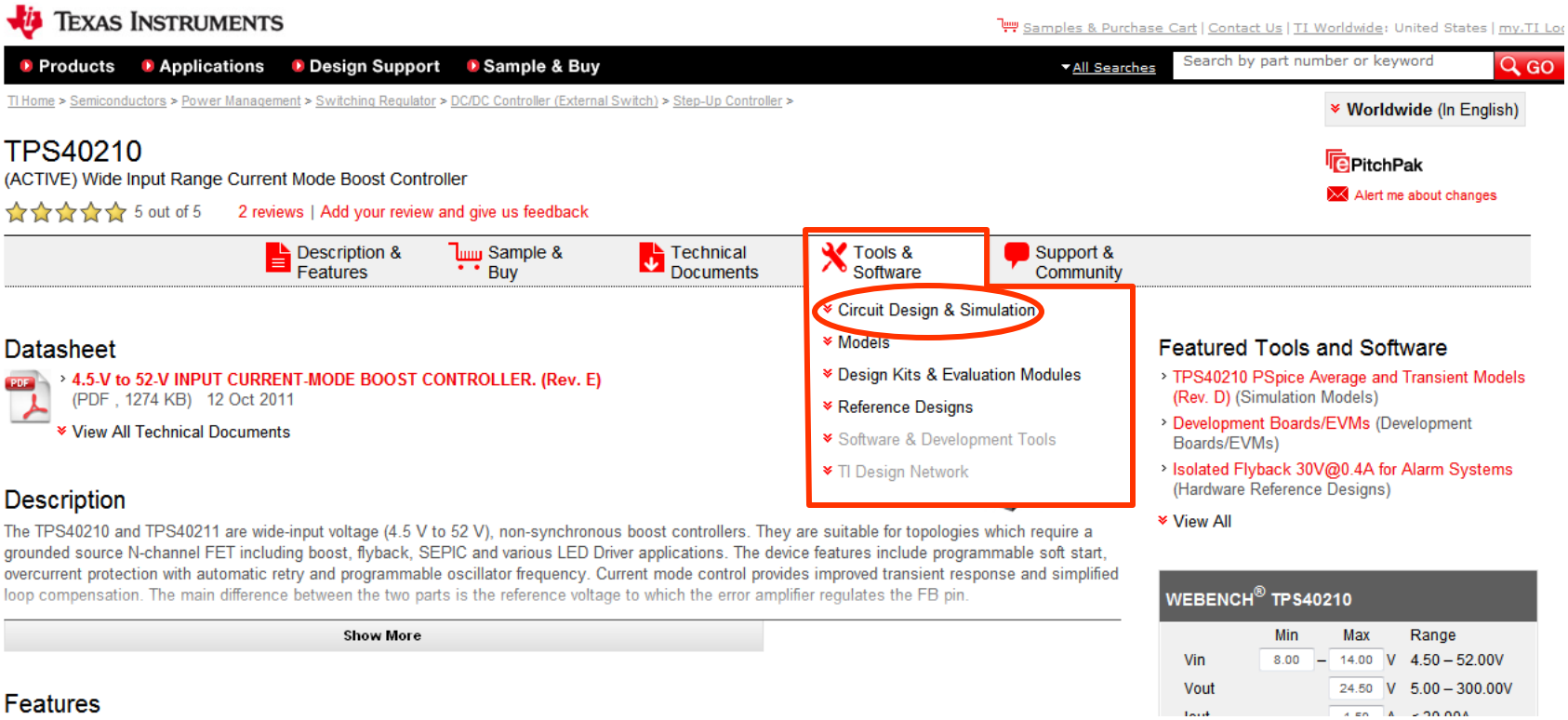
Where to find Models for TI Products

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



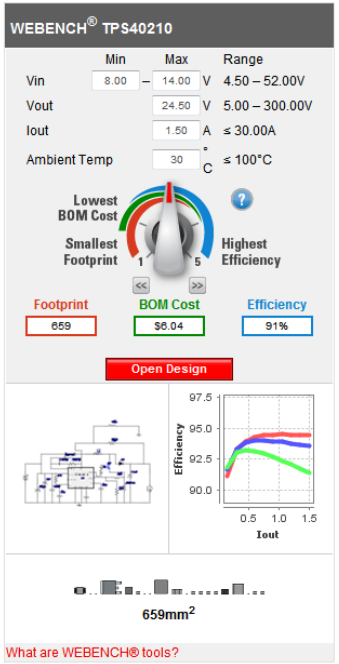
Example product folder for the TPS40210. Clicking on “Circuit Design & Simulation” under “Tools & Software” takes you to the section in the product folder where the models are located.

Different model types and Reference Designs may be available.

Models in Product Folder

Tools & Software

Circuit Design & Simulation



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

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

Show More

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

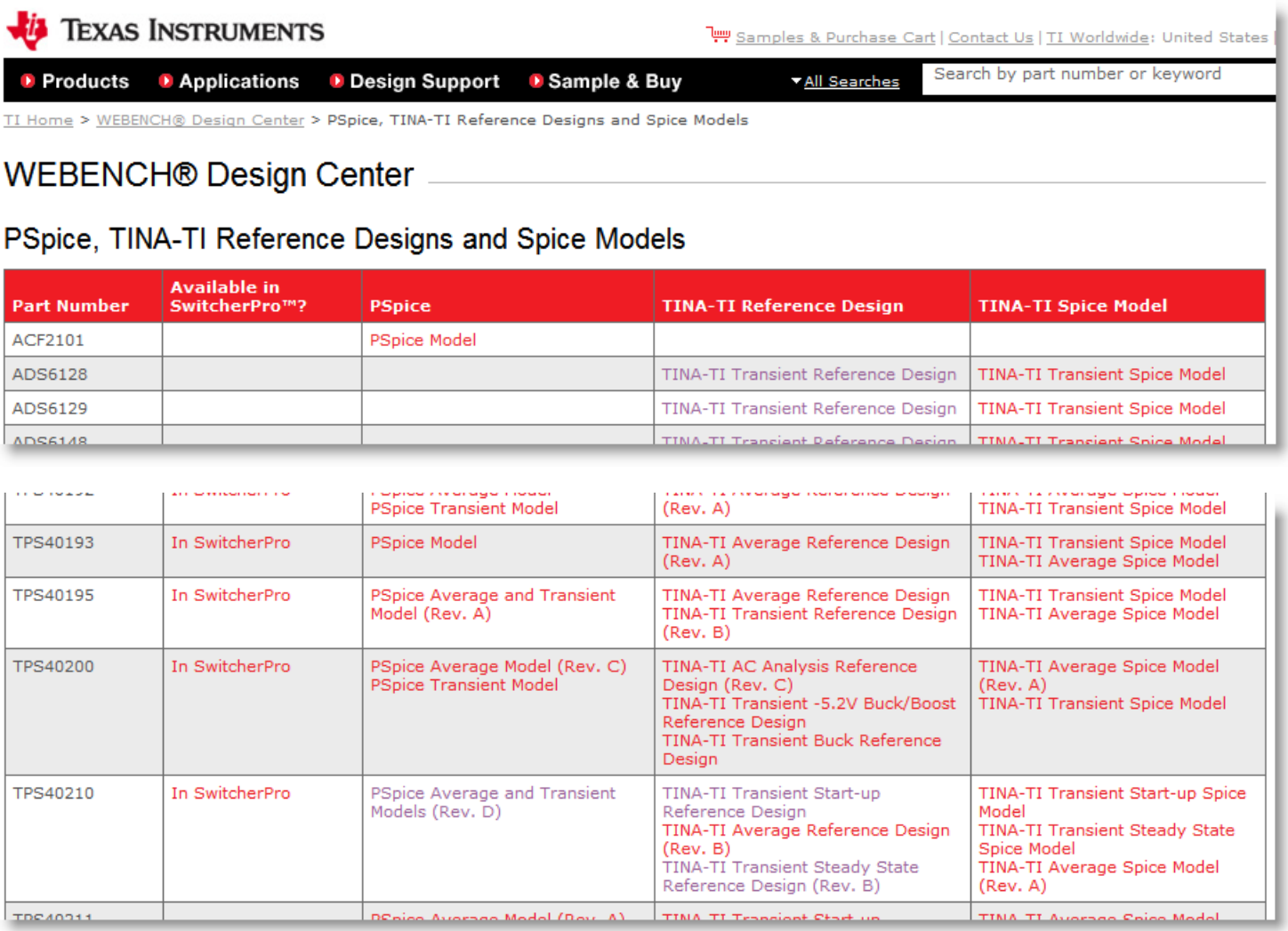
Show More

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



TEXAS INSTRUMENTS

Samples & Purchase Cart | Contact Us | TI Worldwide: United States

Products | Applications | Design Support | Sample & Buy | All Searches | Search by part number or keyword

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
		PSpice Transient Model	(Rev. A)	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.

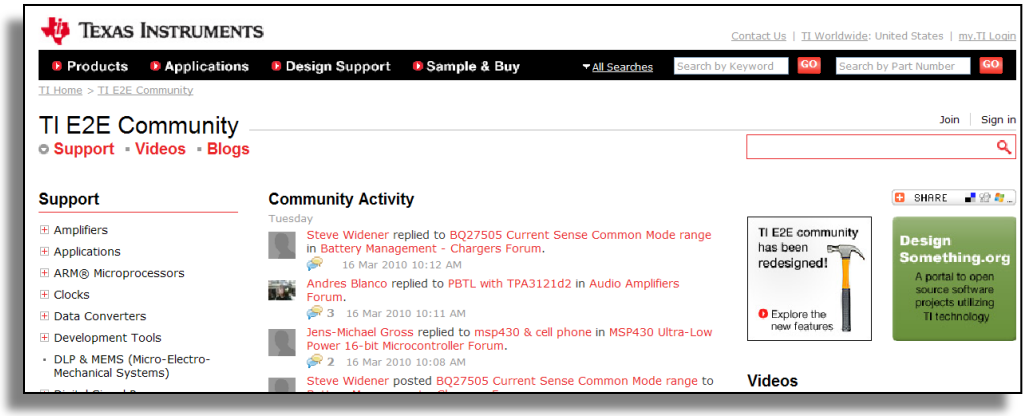
Additional Resources

TI E2E™ Community

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

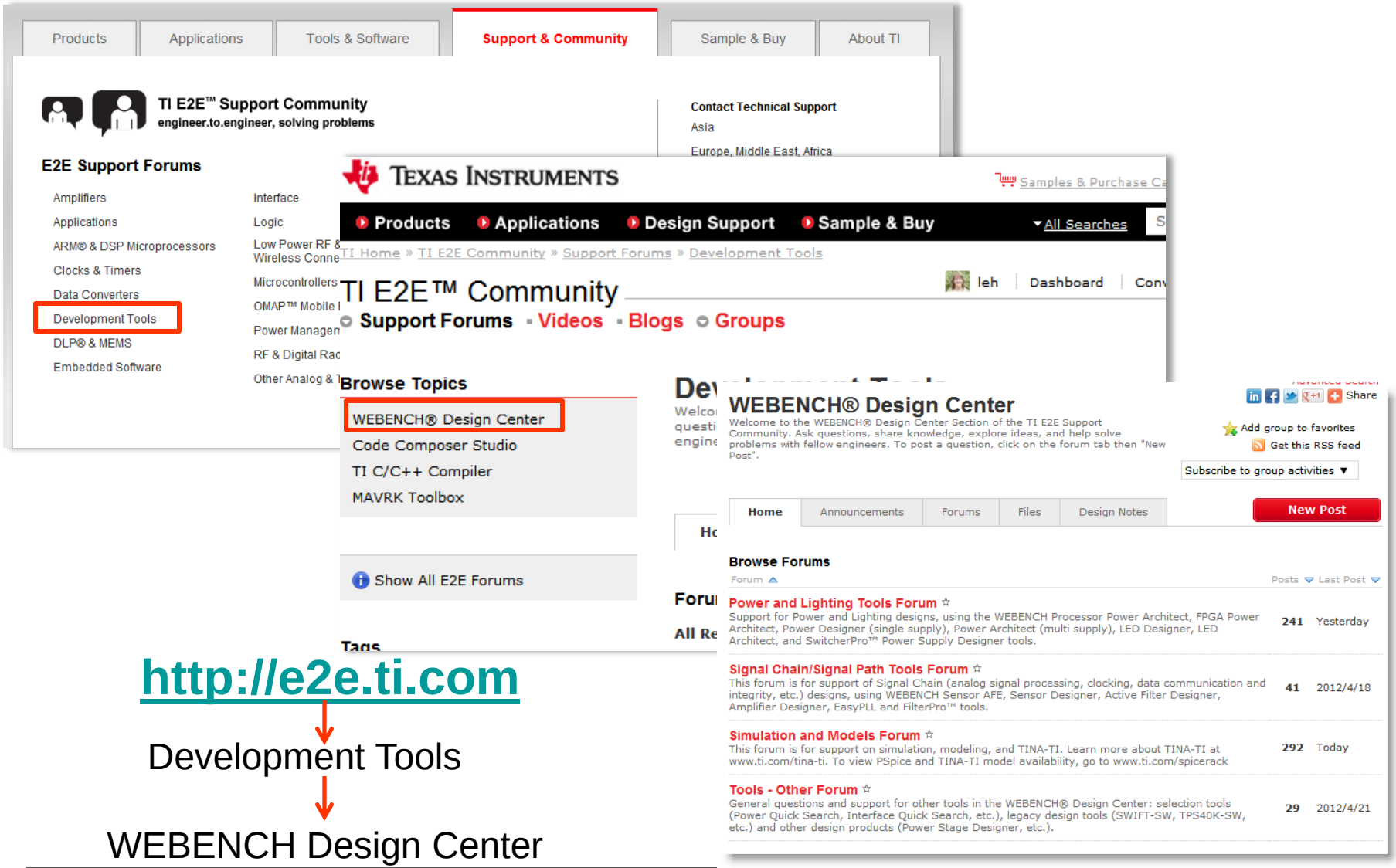
- **Access to a rich library of technical content**
- **Facilitates connections with like-minded engineers**
- **Takes personalization to a new level**
- **Advanced search functionality**
- **Recognizes the involvement of active members**

TI E2E™ Community



http://e2e.ti.com

TI E2E™ Community – Questions on Tina-TI



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?