



TINA-TI™ 7.0 – Op Amps

**Presented by Keith Nicholas
Analog Field Applications**



Agenda

- Introduction / Overview
- Let's Design a Circuit
- Other Cool Analysis Features
- Reference Material



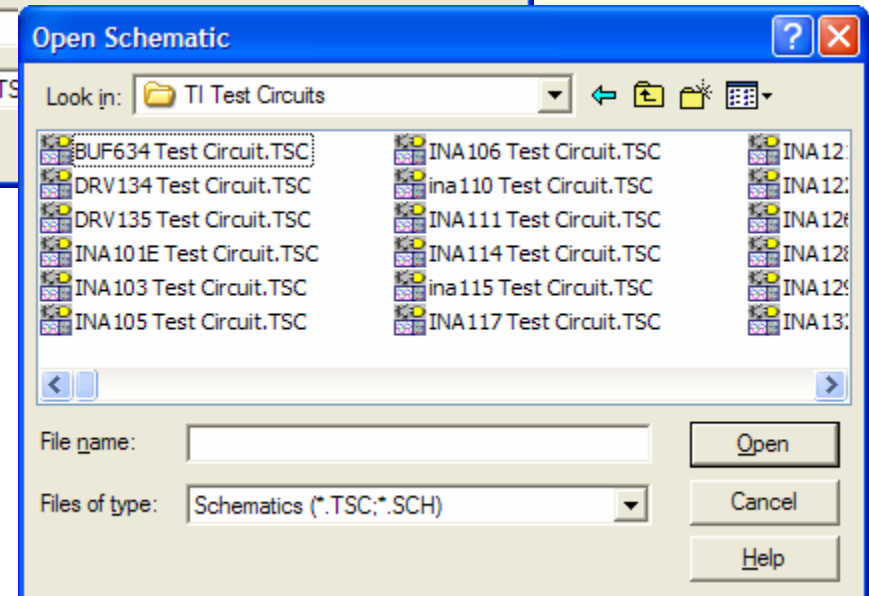
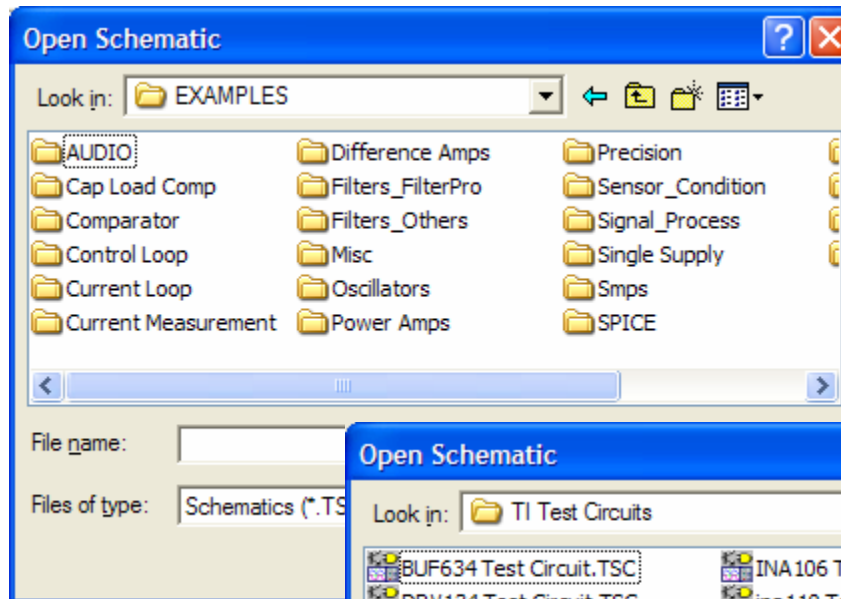
Now Available: TINA-TI™ Version 7.0!

- TINA is an easy-to-use, but powerful, circuit simulation program based on the SPICE engine.
- TINA provides all the conventional DC, transient and frequency domain analysis of SPICE.
- TINA-TI is a fully functional version of TINA, loaded with a library of TI macro-models plus passive and active models.
- Free distribution by TI.
- TINA-TI 7.0 provides support for switching power supply devices.
- TINA-TI 7.0 has **NO LIMIT** to circuit size.
- Circuits developed in TINA-TI 7.0 will work with TINA Industrial version 7.
- TINA-TI Version 6.0 is forward compatible with Version 7.0, but not visa versa.



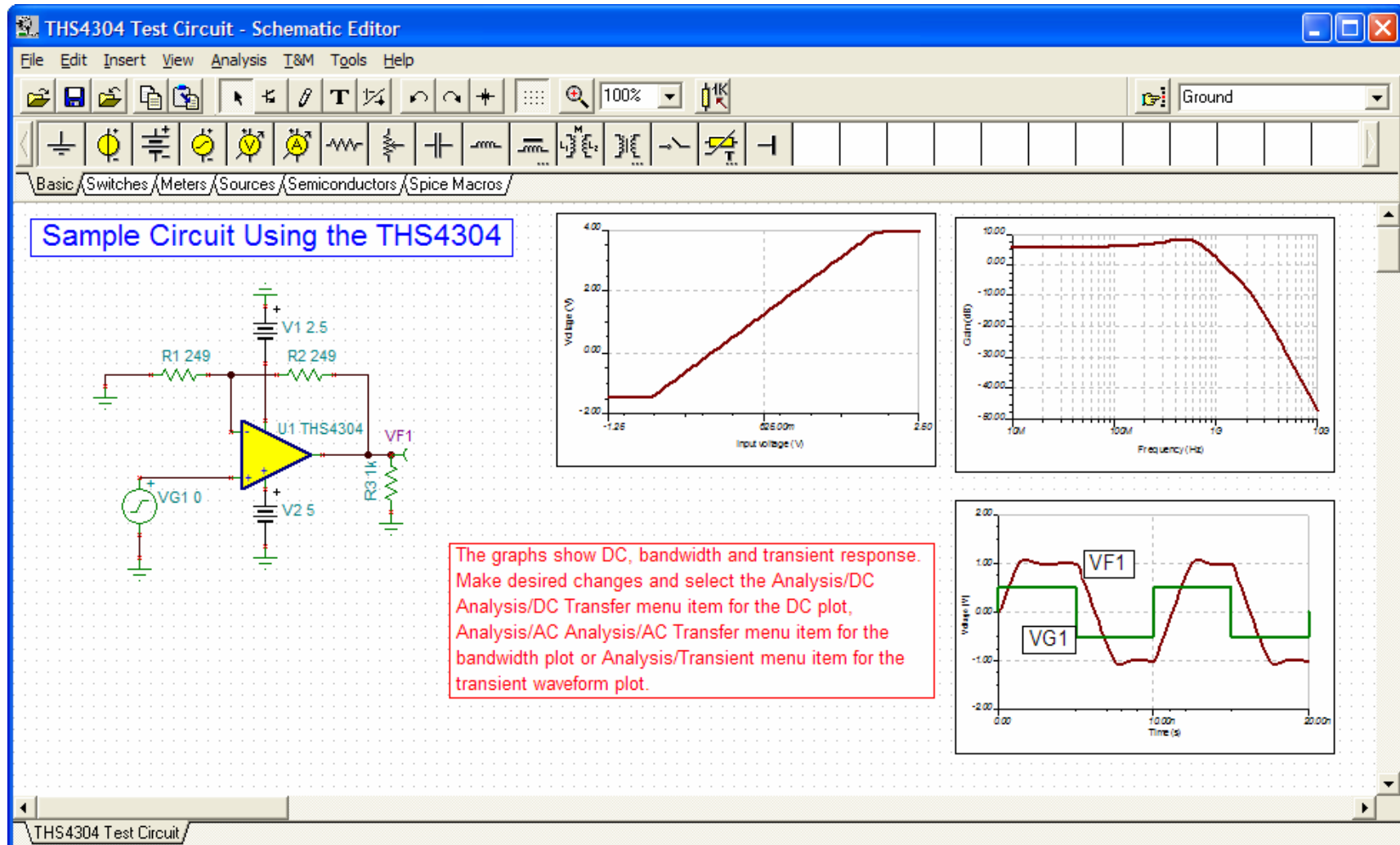
C:\Program Files\DesignSoft\Tina 7 - TI\EXAMPLES

- TINA-TI includes a directory with several example circuits.
- TINA-TI also has test circuits for most TI devices; this is an excellent starting point for a simulation.





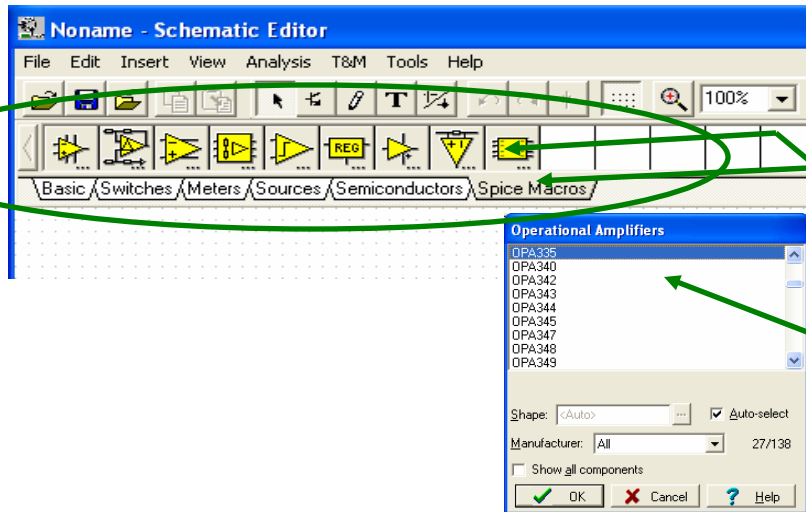
Tina Circuit Analysis/Simulation



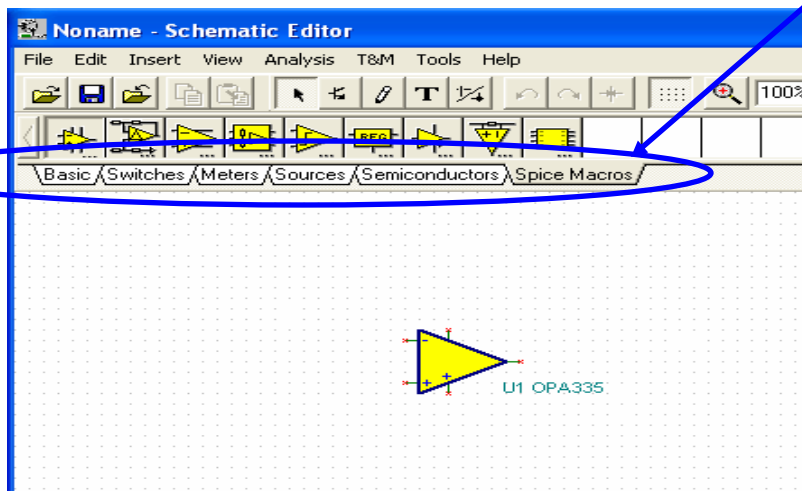


TINA Circuit Simulator

Getting Started



- Click to bring up list of part types
- Click a part type to bring up list of devices of a type
- Place the device on the layout sheet

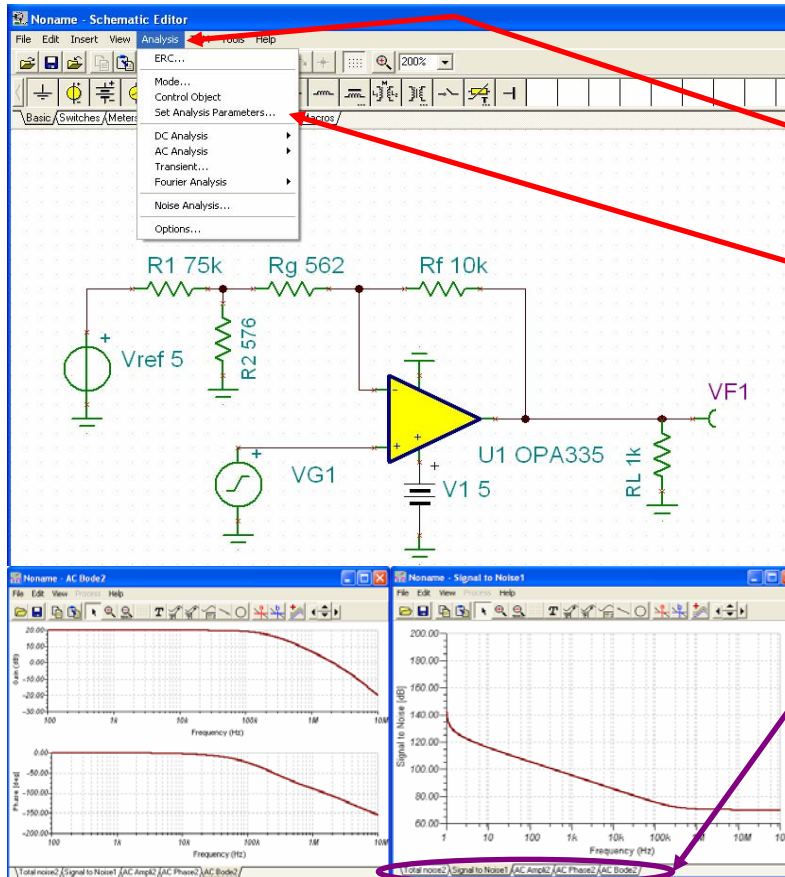


- Use other tabs and selections to find other circuit elements
- Build a working circuit
- Working test circuits for many parts can be opened by going to file/open.
- Other example circuits can be downloaded from the TINA webpage



TINA Circuit Simulator

Initial Design



- Click 'Analysis' to pull up list of simulations
- 'Set Analysis Parameters' to change global analysis settings
- AC, DC, Fourier, and Noise analysis are available
- Graphical results will pop-up in a new window
- All graphs are available by tabbed browsing
- Other tools (oscilloscope, function generator) are available under tools

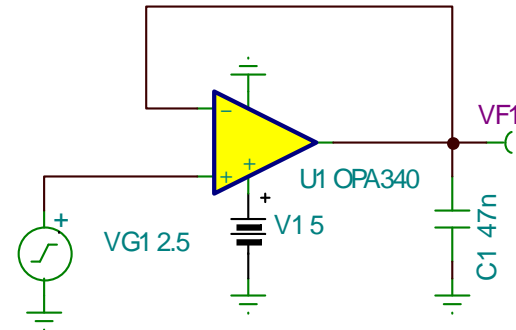


Let's Design a Circuit

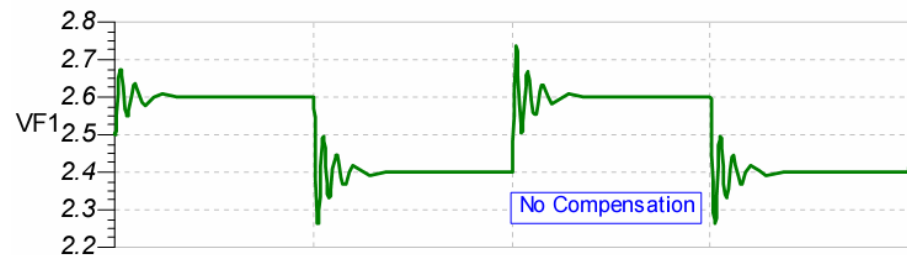


Driving Capacitive Loads

- Ugh!
- Amplifier does not like 47nF load.
- Oscillators won't but evidently amplifiers will!



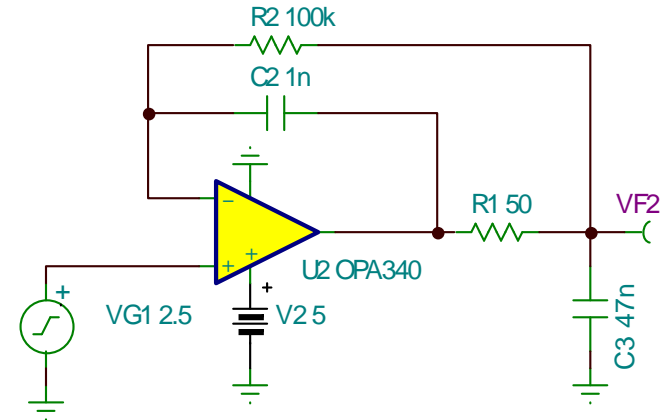
Op Amp Without Cloud Compensation



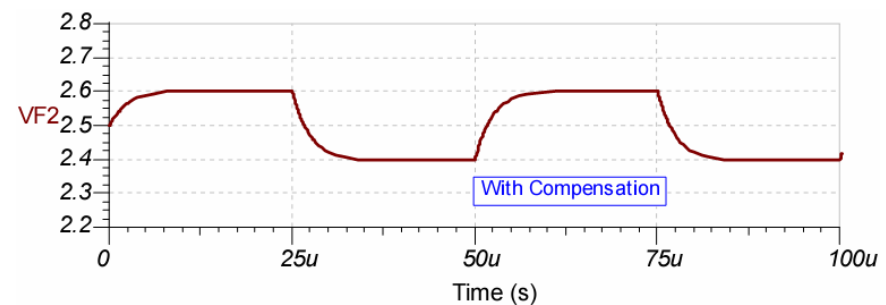


Driving Capacitive Loads

- Much Better
- Transient Analysis looks good.
- Now let's enter the circuit and try some other options.

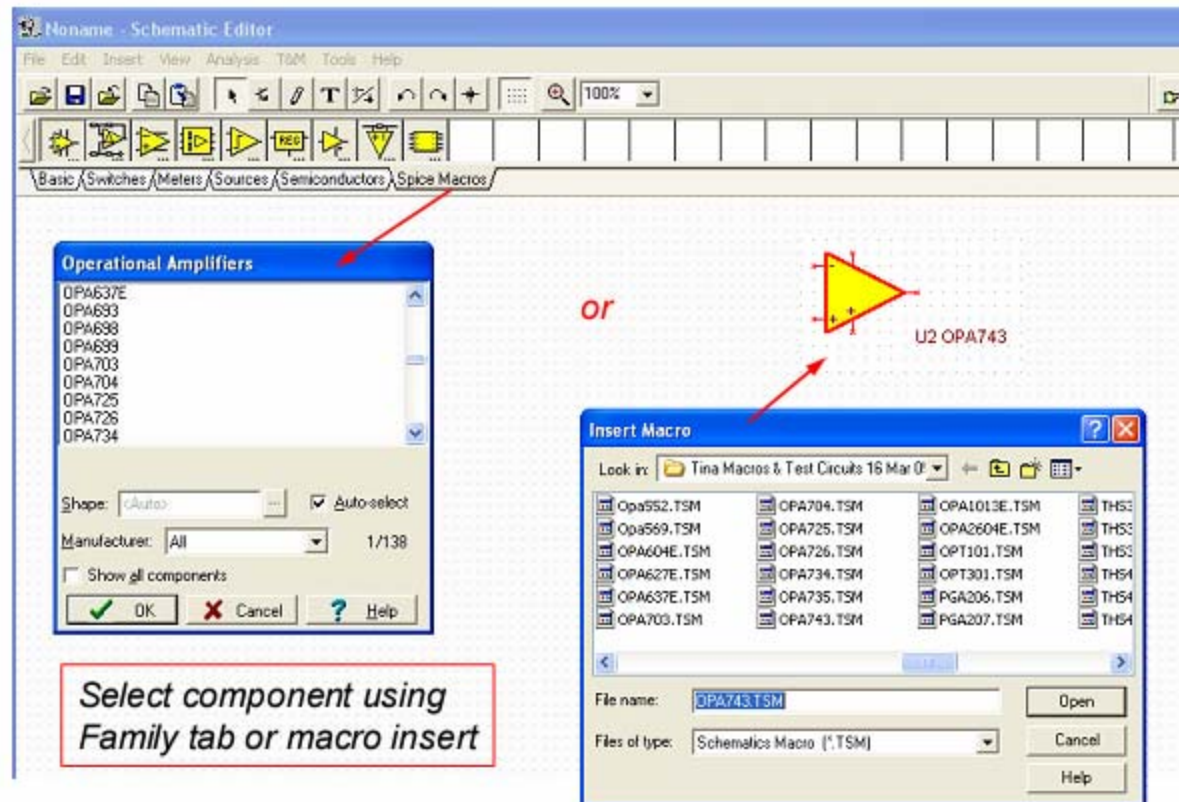


Op Amp With Cload Compensation



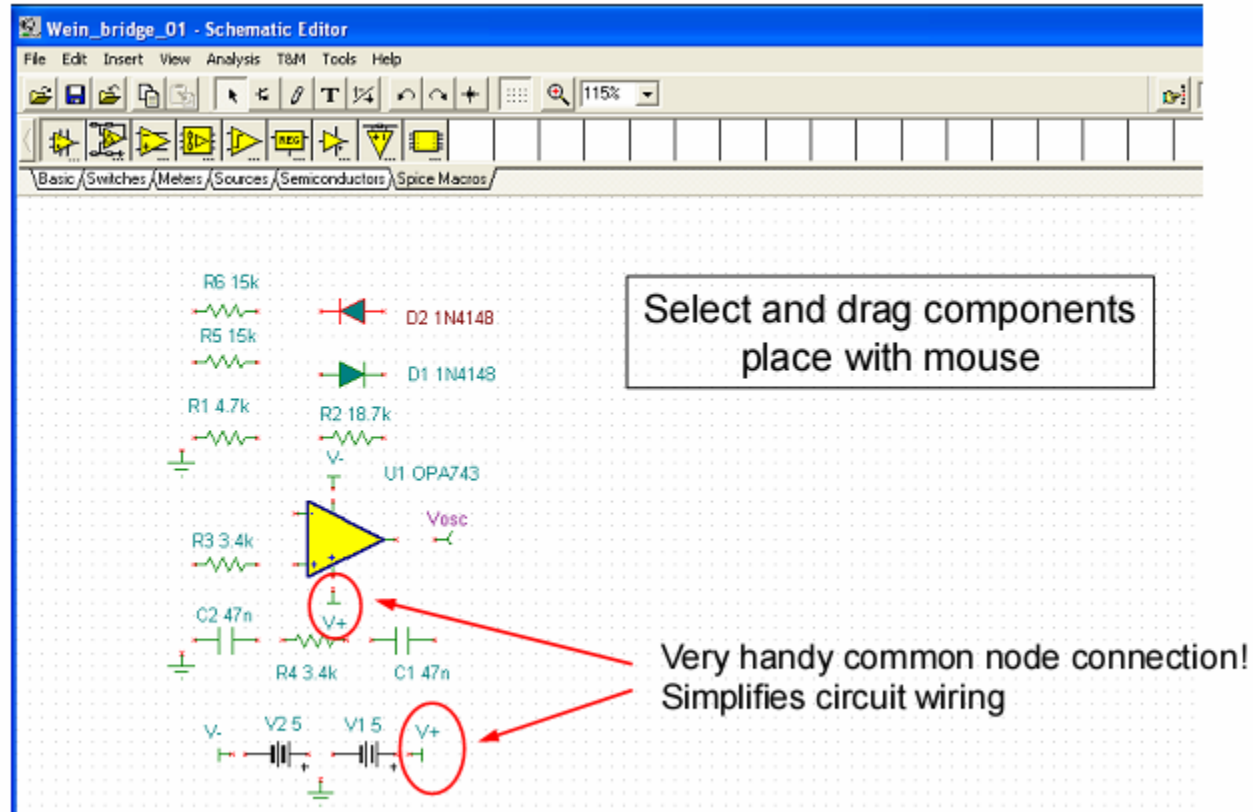


Place Amplifier Macro



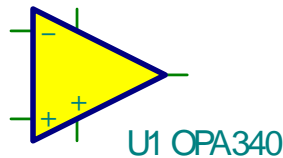


Enter Passives/etc.





Enter Passives/etc.



J1
—|

J1 is a connection jumper.

Use for Power Supplies, Reference, IO, etc.

This 'node' does not show up in analysis results.

VF1
—(

VF1 is a 'Voltage Meter'.

Use this to view analysis results.



DC Analysis

Cloud_Example_TechDay2007 - Schematic Editor

File Edit Insert View Analysis TSM Tools Help

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

Calculate model voltages
Table of DC results
DC Transfer Characteristic...
Temperature Analysis...

STARTING FOR A CAPACITIVE LOAD

Op Amp With Cloud Compensation

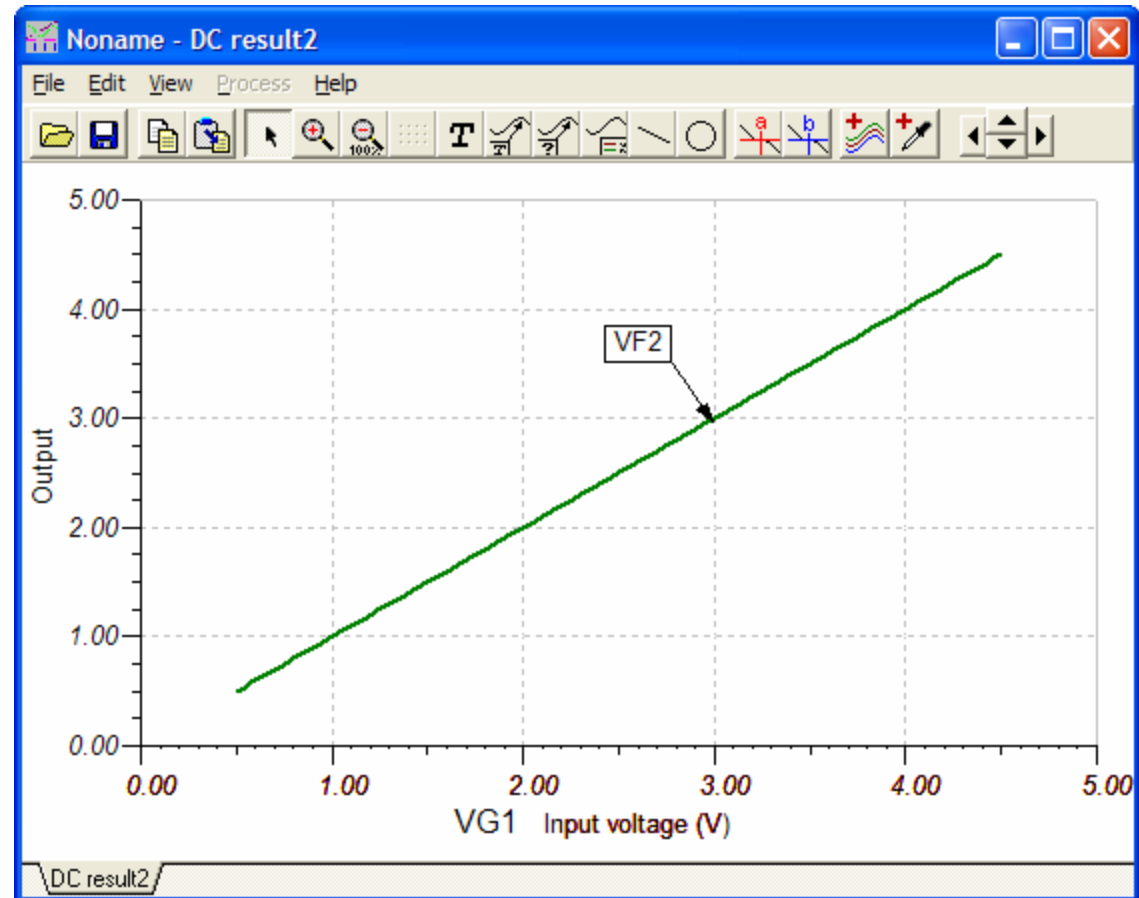
DC Transfer Characteristic

Start value 0.5 [V]
End value 4.5 [V]
Number of points 100
Input VG1
☐ Enable hysteresis run

OK
Cancel
Help



DC Analysis





AC Analysis

Cloud_Example_TechDay2007 - 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...

Calculate nodal voltages
Table of AC results
AC Transfer Characteristic...

COM

MACROS

Op Amp With Cload Compensation

AC Transfer Characteristic

Start frequency 10 [Hz]
End frequency 10M [Hz]
Number of points 100

Sweep type
☐ Linear ☒ Logarithmic

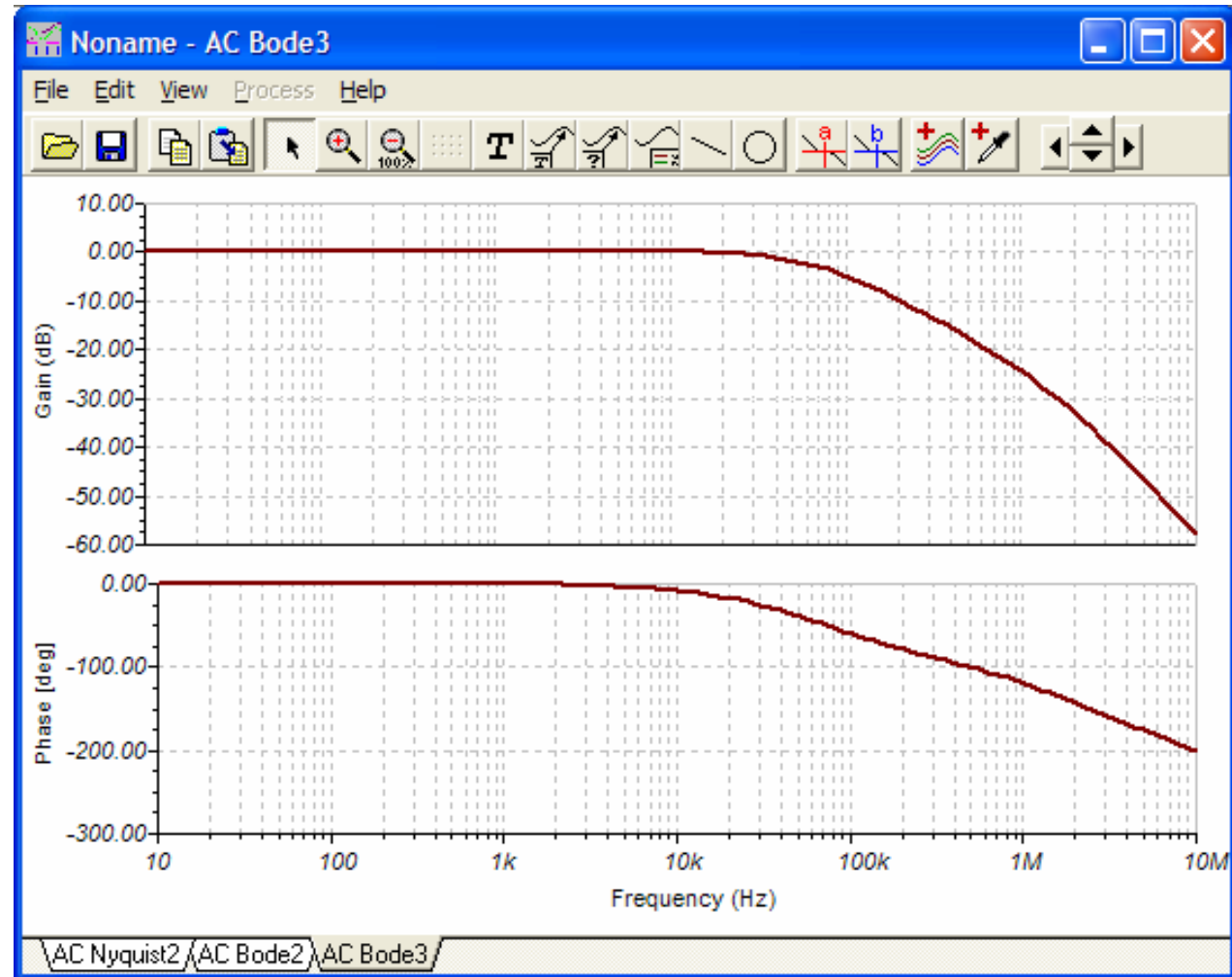
Diagram
☐ Amplitude ☒ Nyquist
☐ Phase ☐ Group Delay
☒ Amplitude & Phase

OK Cancel Help

Schematic components: VG1 2.5, U2 OPA340, V2 5, C2 1n, R1 50, VF2, C3 47n.



AC Analysis





Fourier Analysis

Fourier Series

Sampling start time: 0

Base frequency: 20k

Number of samples: 4096

Number of harmonics: 16

Format: RMS, Phase

Output: VF2

Transient initial condition:

- ☒ Calculate operating point
- ☐ Use initial conditions
- ☐ Zero initial values

Buttons: Calculate, Cancel, Help, Draw

Fourier Series

Sampling start time: 0

Base frequency: 20k

Number of samples: 4096

Number of harmonics: 16

Format: RMS, Phase

Output: VF2

Transient initial condition:

- ☒ Calculate operating point
- ☐ Use initial conditions
- ☐ Zero initial values

Fourier coefficients

k	Amplitude (C)	Phase (φ)
0.	2.5	0
1.	88.64m	-102.63
2.	6.11m	-30.63
3.	23.38m	-120.43
4.	4.57m	-49.98

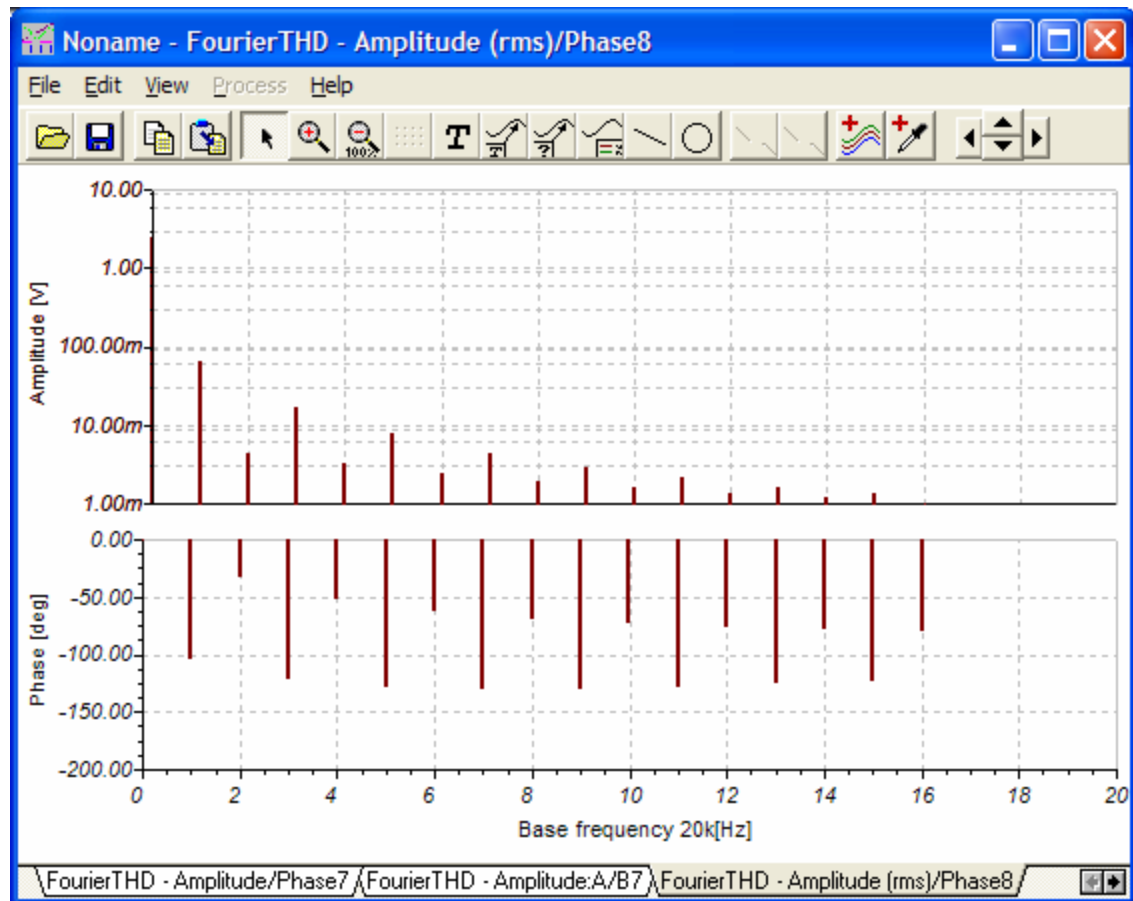
Harmonic distortion : 32.482%

Buttons: Calculate, Cancel, Help, Draw



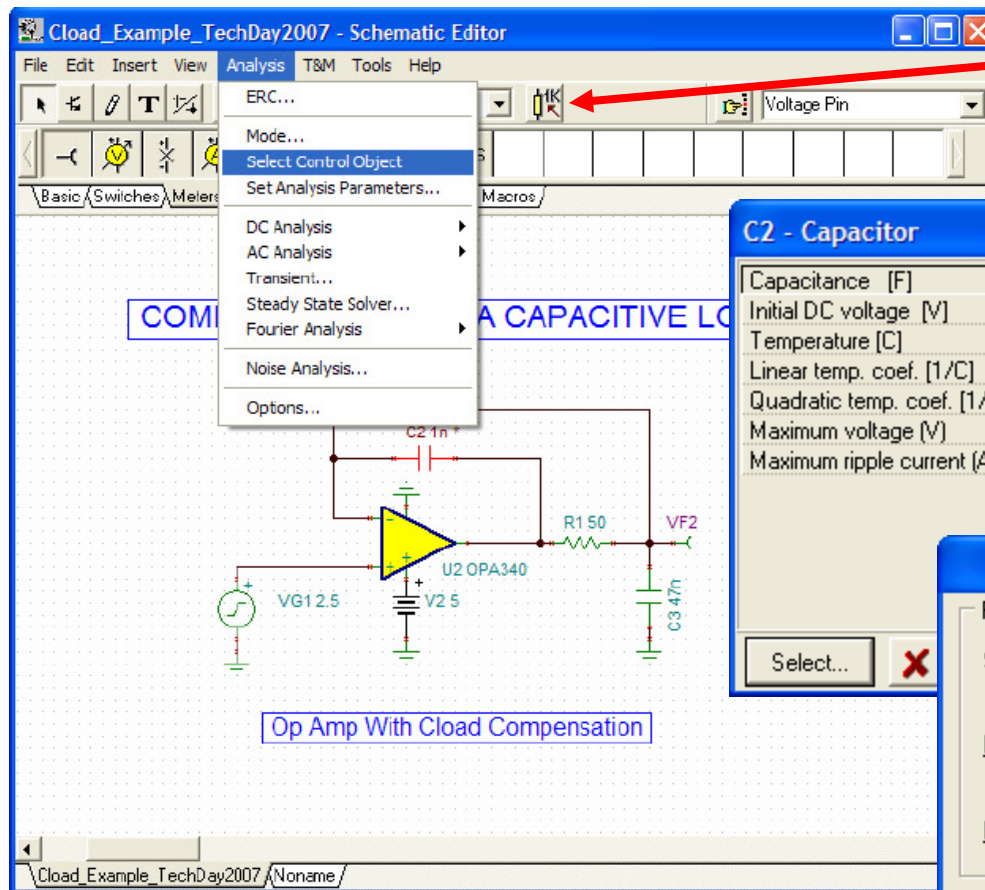
Fourier Analysis

- Based on 20KHz, Square Wave.





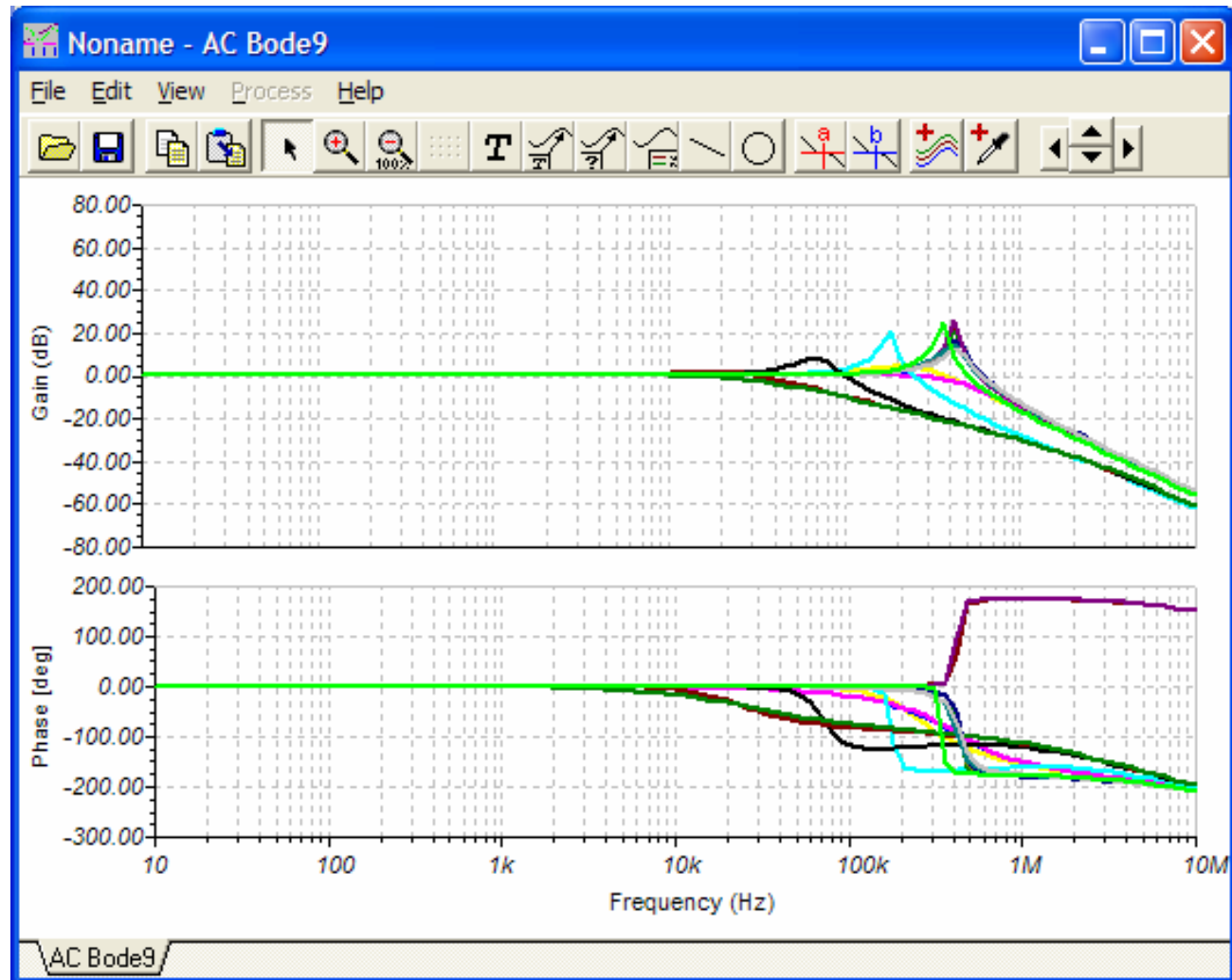
AC Analysis with Parameter Step



Can use button instead of Menu.



Parameter Stepping





Transient Analysis

Cloud_Example_TechDay2007 - 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...

COM A CAPACITIVE LOAD

Op Amp With Cloud Compensation

Transient Analysis

Start display 0 [s]
End display 100u [s]

☒ Calculate operating point
☐ Use initial conditions
☐ Zero initial values

☐ Draw excitation

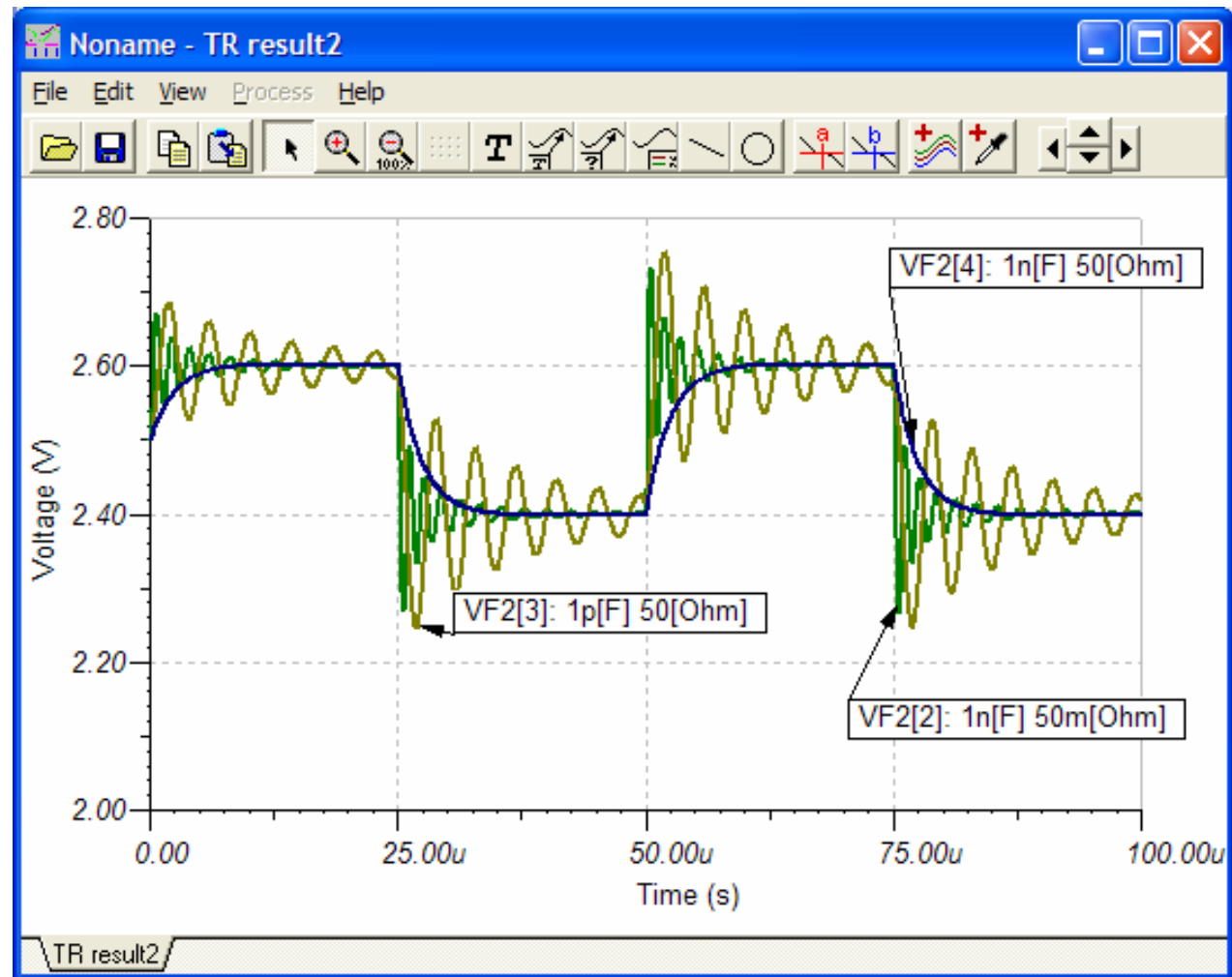
Integration method
☒ Trapezoidal ☐ Gear

Integration order 2

OK Cancel Help



Transient Analysis





Noise Analysis

Cloud_Example_TechDay2007 - 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...

COMPEN APACITIVE LOAD

Op Amp With Cloud Compensation

Noise Analysis

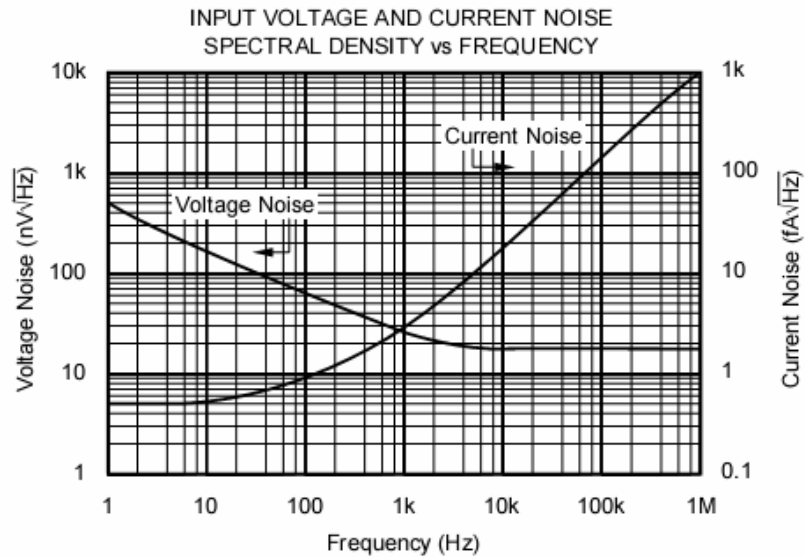
Start frequency 1 [Hz]
End frequency 1M [Hz]
Number of points 100
S/N Signal Amplitude 1

Diagrams
☒ Output Noise ☐ Total Noise
☐ Input Noise ☐ Signal to Noise

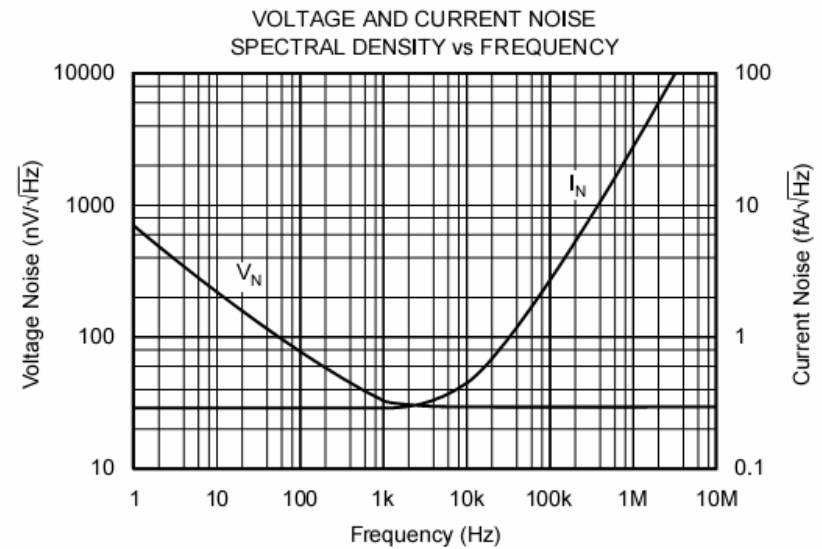
OK Cancel Help



Noise Analysis



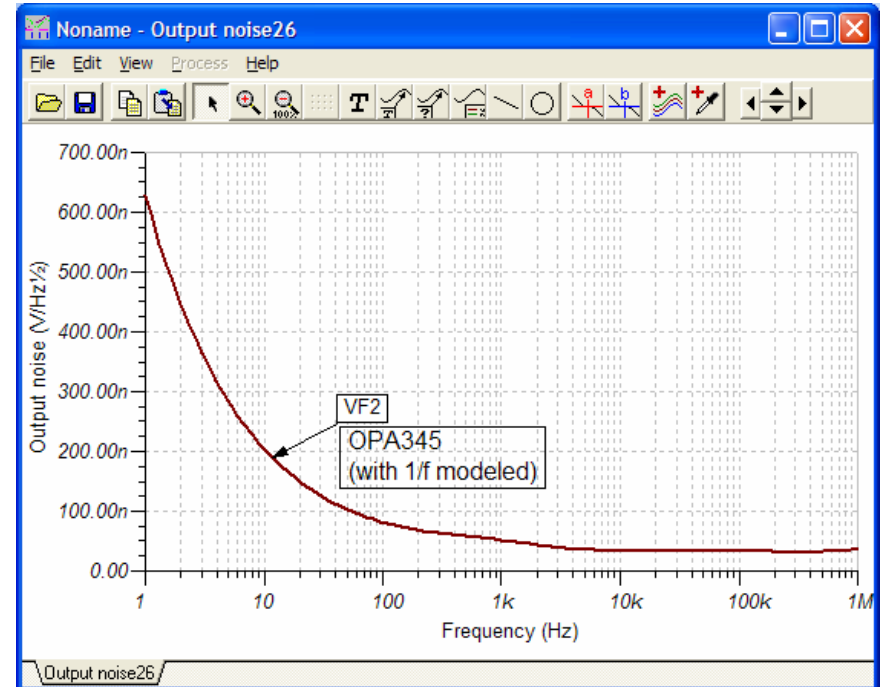
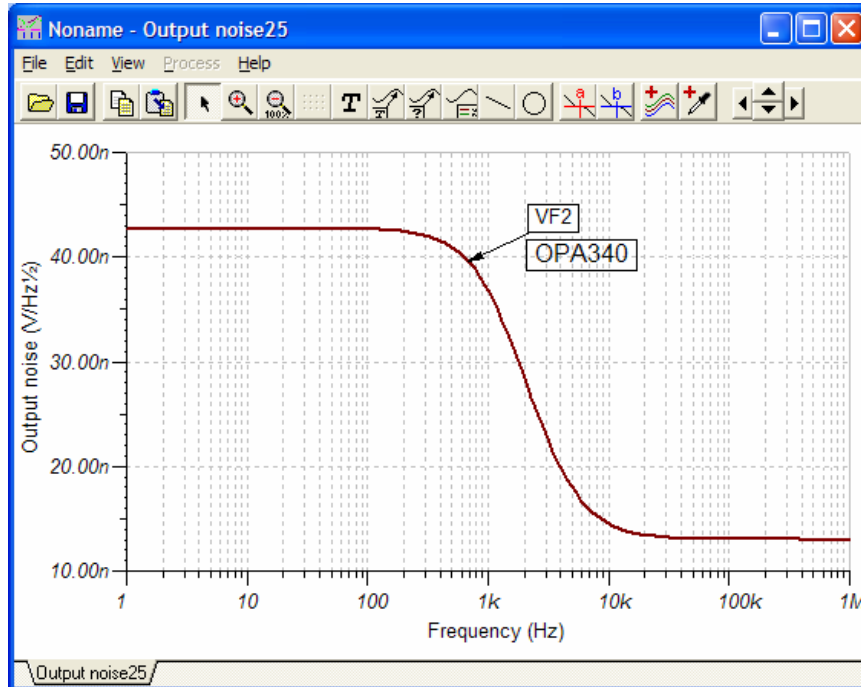
OPA340



OPA345



Noise Analysis



Many OPAMP macro's do not model noise, some only model wideband 'white' noise but not 1/f. Check Models!

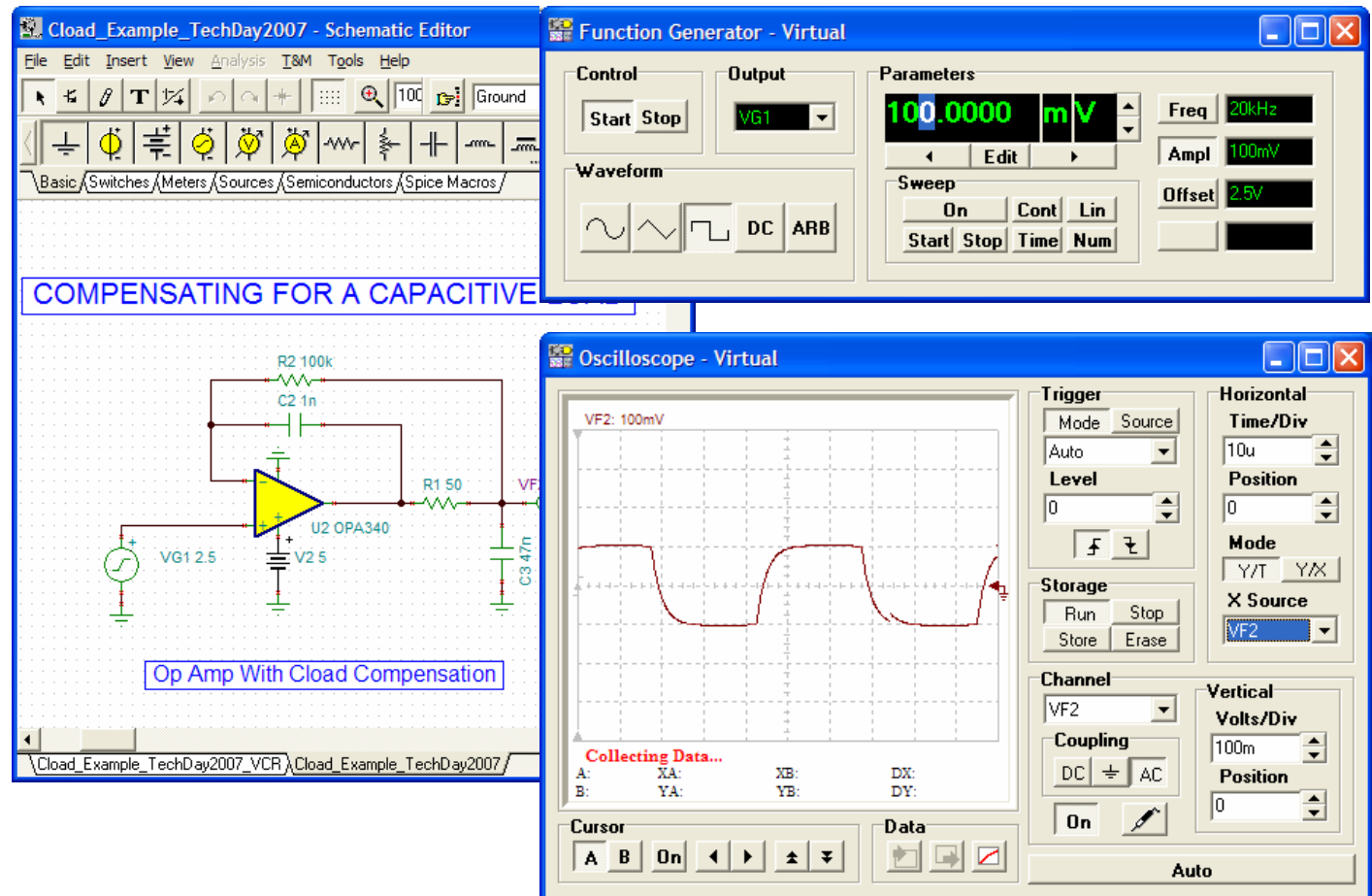


Other Cool Features



Test & Measurement Tools

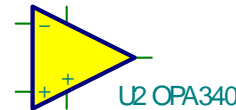
- TINA has a set of interactive T&M tools.
- These work much the same way as lab equipment.
- Function Generator, Oscilloscope, Multimeter, etc.
- Allows interactive changes to stimulus and view results 'on-the-fly'.





View Macro's

- Double Click on Schematic Symbol.
- Click 'Enter Macro'.
- View Macro Header. For TI macro's, parameters modeled are listed.



U2 - OPA340

Label	U2
Parameters	[Parameters]
SubCkt-(Shape)	Amp5-T
SubCkt-(Content)	
SubCkt-Parameters	<input type="checkbox"/>
SubCkt-Optional nodes	

OK Cancel ? Help Enter Macro

<Cloud_Example_TechDay2007:U2 [MACRO]> - Netlist Viewer

File Edit Analysis Help

```

*
*               non-inverting input
*               |   inverting input
*               |   |   positive supply
*               |   |   |   negative supply
*               |   |   |   output
*               |   |   |   |
*               |   |   |   |
* .SUBCKT OPA340  +   -   V+   V-   OUT
* INPUT STAGE
*
i1 V+ 5 150u
m7 550 vswitch 5 5 pix l=6u w=25u m=4
m8 550 550 V- V- nix l=6u w=25u m=4

```

Line: 1 Col: 1

Cloud_Example_TechDay2007\U2\



Netlist Editor

- Netlist Editor basically replaces the schematic entry tool.
- Handy to cut/paste circuits from different websites; will work with most simulators.
- With TINA-TI, this is the only way to generate your own MACRO's. (You need to upgrade to the full TINA-7.0 to use the schematic editor for MACRO's.)

```
<cloud_example_techday2007.cir> - Netlist Editor
File Edit Analysis Help
[Icons] Compile
[Icons] Set Parameters...
CLOAD
*****
** Th
**
*****
.LIB "
.LIB "
.TEMP
.AC DEC 16 10 10MEG
.TRAN 200N 100U
.DC LIN VG1 0 1 10M

.OPTIONS ABSTOL=1U ITL1=100
.PROBE V(5)

V2          4 0 5
VG1         1 6 DC 0 AC 1 0
+ SIN( 0 1 20K 0 0 0 )
VG1_DC      6 0 2.5
XU2         1 2 4 0 3 OPA340_0
C2          2 3 1N IC=0

2007 (TINA Netlist Editor format)
*****
Created by TINA
DesignSoft, Inc.
*****
...LES\SPICE\TSPICE.LIB"
...LIB\Operational Amplifiers.LIB"

Line: 1 Col: 1
```



Tricks for custom Macro/devices

- TINA-TI does not allow creation of Macro's or creation of libraries, must upgrade to TINA-7.0 Classic Edition.
- However, you can create custom Macro's by viewing and editing the contents of a Macro.
- Each Macro placed into the schematic has a unique local copy that can be edited with any valid SPICE statements.
- With this capability, you can 'load' different device models or create a unique model.



-
- Noname - Schematic Editor
- File Edit Insert View Analysis T&M Tools Help
- TLV411XA
- Basic Switches Meters Sources Semic
- U1 TLV411XA
- Open this macro and modify the internal netlist.
- <Noname:U1 [MACRO]> - Netlist Viewer
- File Edit Analysis Help
- ```

* Replace 5-pin OPAMP macro with this text, TINA keeps local copy.
*
* Voltage Controlled Resistor/Inductor
* connection: negative control input (OPAMP -input)
* | positive control input (OPAMP +input)
* | | positive resistor (OPAMP positive supply)
* | | | negative resistor (OPAMP negative supply)
* | | | | reference resistor (OPAMP output)
* | | | |
.subckt VCres 1 2 3 4 5
E1 3 6 POLY(2) (1,2) (5,0) 0 0 0 0 1
V1 6 4 0
F1 0 5 V1 1
R1 1 2 10G
.ends

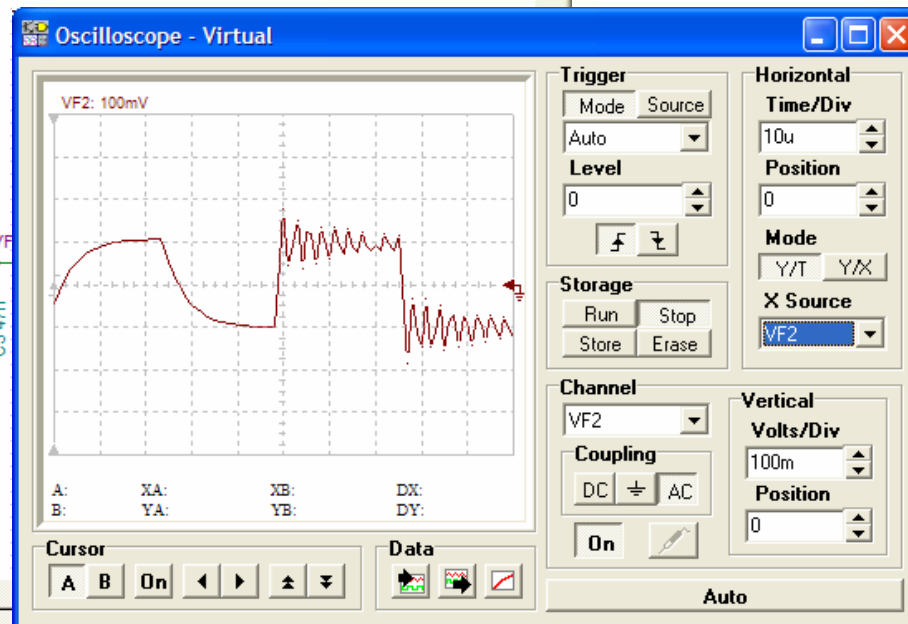
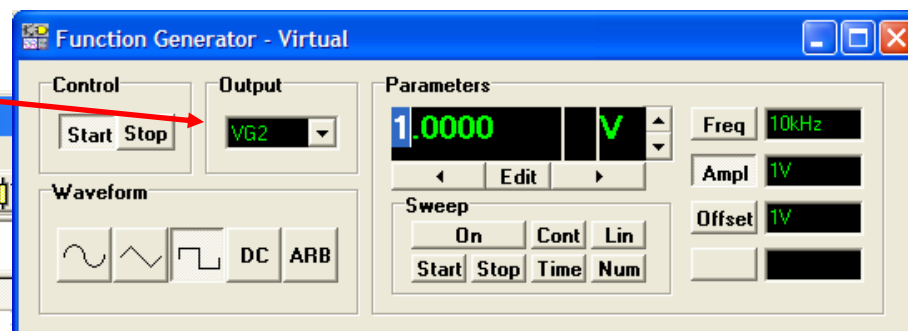
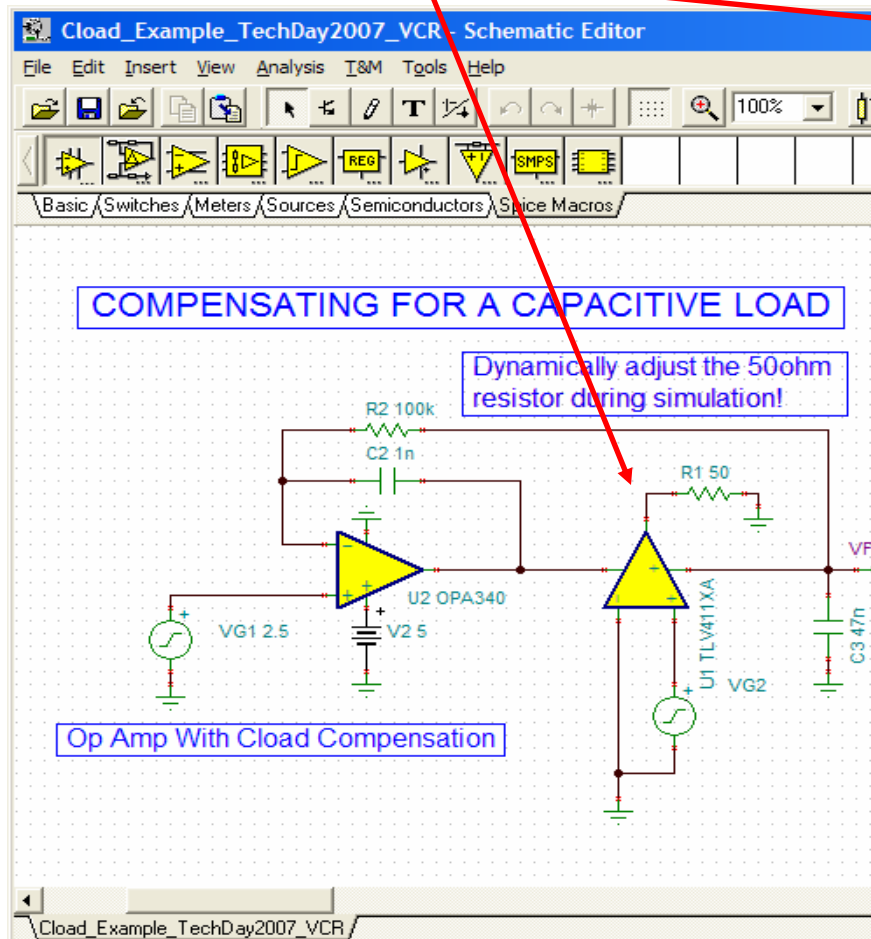
```
- Line: 1 Col: 1
- C:\Cloud\_Example\_TechDay2007\_VCR\C\Cloud\_Example\_TechDay2007\Noname\U1





# Dynamic Adjustment during Simulation

- VG2 controls 50ohm resistor,  $R_{out} = R1 \cdot VG2$





# References/Links

- <http://www.ecircuitcenter.com/index.htm>
- <http://www.tina.com/>
- **SPICE: A Guide to Circuit Simulation and Analysis Using PSpice**  
by [Paul W. Tuinenga](#)



<http://focus.ti.com/docs/toolsw/folders/print/tina-ti.html>

SPICE-Based Analog Simulation Program - TINA-TI - TI Tool Folder - Microsoft Internet Explorer

File Edit View Favorites Tools Help

Address <http://focus.ti.com/docs/toolsw/folders/print/tina-ti.html> Go Links

TEXAS INSTRUMENTS Technology for Innovators™

products applications design support buy

search TI.com Enter Keyword Go Enter Part Number Go

Contact Us | TI Worldwide: United States | my.TI Login

TI Home > Semiconductors > Amplifiers and Linear >

### SPICE-Based Analog Simulation Program

TINA-TI, Status: ACTIVE

Texas Instruments

Alert me to changes to this product

|                 |                      |                     |
|-----------------|----------------------|---------------------|
| Description     | Support Software     | Technical Documents |
| Features        | Available Updates    | Order Options       |
| What's Included | Compatibility Issues | Related Products    |

| TINA-TI      |                                       |
|--------------|---------------------------------------|
| Name         | SPICE-Based Analog Simulation Program |
| Status       | ACTIVE                                |
| Price (US\$) | Free                                  |
|              | Order Options                         |
|              | <a href="#">Download</a>              |

**Product Information**

Description

[View TINA-TI Screen](#)

[http://www.ti.com/hdr\\_jbm](http://www.ti.com/hdr_jbm)

Local intranet

**Analog eLab Design Center**

**Support**

- KnowledgeBase
- Contact Technical Support
- Training

**Customer Tags - What's This?**

Most Popular Tags for TINA-TI:

- [tina-ti](#) (6)
- [behavioral simulation](#) (1)
- [nilsson](#) (1)
- [View All Tags](#)

[Create a Tag](#)