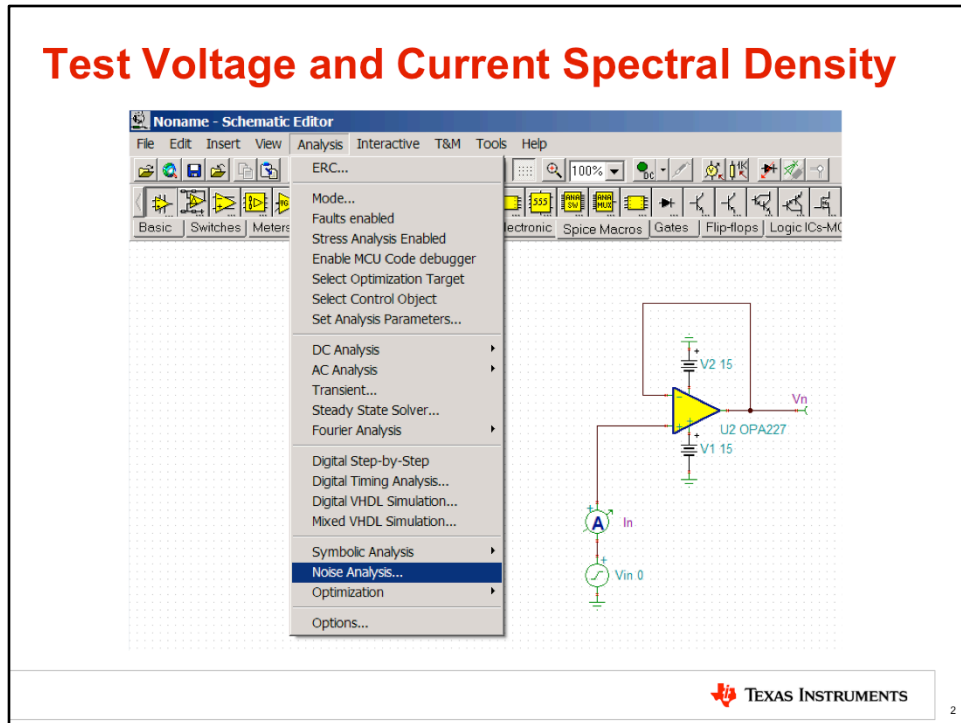




Hello, and welcome to the TI Precision Lab discussing intrinsic op amp noise, part 6.

In the last noise video we covered the basics of noise analysis using SPICE. During that video we mentioned that it is important to make sure that the noise model is accurate, and also suggested looking at the net list to check if noise is included in the model. In this video, we show a more comprehensive method for verifying that the noise model is accurate. Furthermore, we show a method for creating our own model if an accurate model does not already exist.

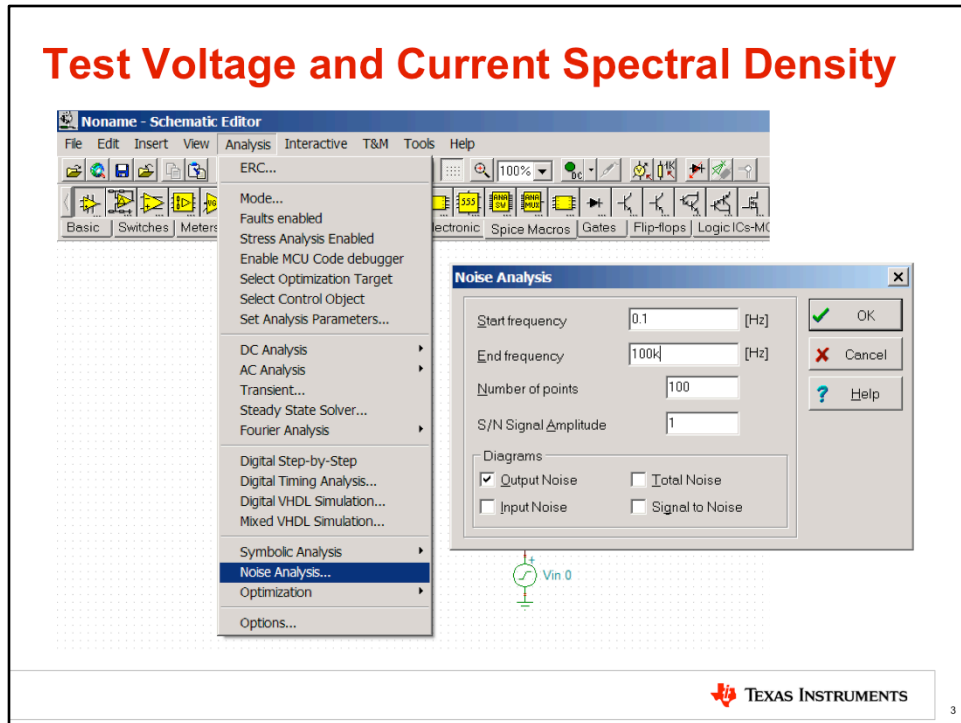
## Test Voltage and Current Spectral Density



This is the standard test circuit for confirming the noise model of an amplifier. The amplifier is placed in a voltage follower configuration, which allows the input voltage noise to be buffered so that it can be measured at the output probe  $V_N$ . A current meter  $I_N$  is placed in series with the non-inverting input to measure the current noise of the amplifier.

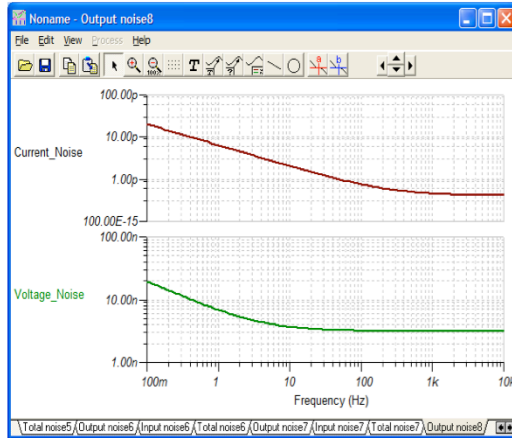
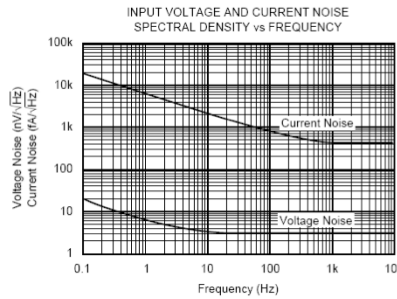
To begin the noise analysis, click "Analysis" in the menu bar, then click "Noise Analysis."

## Test Voltage and Current Spectral Density



After selecting “Noise Analysis,” you will need to enter the start and end frequency to match the plot in the amplifier’s data sheet. You should also select “Output Noise” under the diagrams section to generate the noise spectral density curve.

## OPA227 Matches the Data Sheet

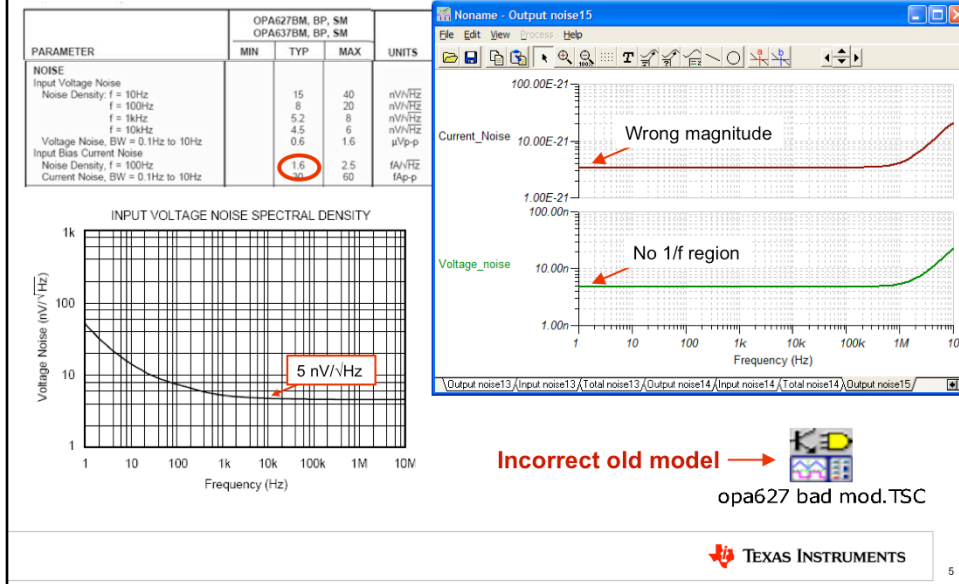


TEXAS INSTRUMENTS

4

This slide shows the result of the noise simulation. Make sure that you compare several points on the voltage and current noise curves from the data sheet to the simulated results. In general, modern Texas Instruments amplifier models accurately model noise, so the simulated results should closely match the data sheet curves. However, some old models or obscure devices may not properly model noise. What do we do if there is an issue with a noise model or if the model doesn't exist? The answer is that we can make our own model. The remainder of this video covers the step-by-step procedure for creating a noise model.

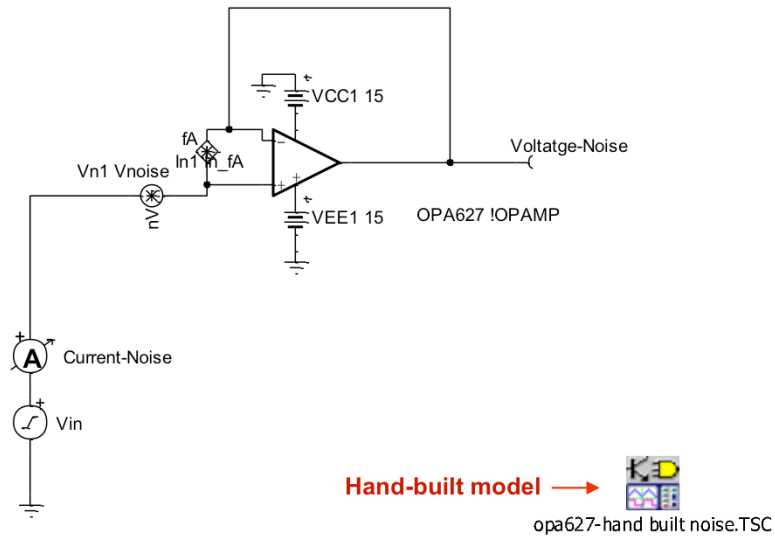
## Old Version of OPA627 Does Not Match!



This slide shows an example of an older model for the OPA627 that did not accurately cover noise. Thankfully, the model has been updated so that the current version available on the TI website shows accurate performance. Nevertheless, the old version is embedded in this presentation as an example of a model that does not properly include noise.

In this example, if you compare the simulated voltage noise spectral density to the data sheet graph you can see that 1/f noise is not modeled. Furthermore, the simulated current noise should be 1.6fA/√Hz but is the wrong magnitude. What do we do in the case that we do not have a good noise model?

## Build Your Own Noise Model

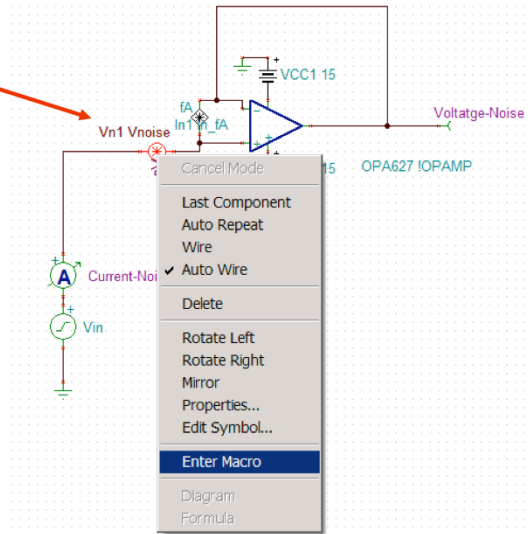


TEXAS INSTRUMENTS

Texas Instruments provides a Tina SPICE model that you can customize in order to achieve accurate noise performance. The model includes a noise voltage source, a noise current source, and a generic op amp. The  $1/f$  and broadband noise in the models can be adjusted according to the data sheet specifications for the op amp you want to model. Also, the op amp's open loop gain and bandwidth can be adjusted so that the AC response is correct. Included in this presentation is an embedded link to the Tina SPICE circuit with the noise sources and generic op amp. Now, let's see how to edit the sources and op amp to match the data sheet.

## Edit Voltage Noise Source

Enter magnitude of  $1/f$  and broadband noise into the macro.



First, let's edit the voltage noise source. Right click on it, then select "Enter Macro." This will open a net list viewer.

## 1/f Region

Look for a point in the 1/f region. Enter the frequency and magnitude at this point.

```

opa627 noise source mod:U1 [MACRO] - Schematic Editor
File Edit Insert View Analysis T&M Tools Help
Basic/Switches/Meters/Sources/Semiconductors/Optoelectronics/Spice Macros/Gates/
* BEGIN PROG NSE NANOVOLT/RT-HZ
.SUBCKT VNSE 30 40
* BEGIN SETUP OF NOISE GEN - NANOVOLT/RT-HZ
* INPUT THREE VARIABLES
* SET UP VNSE 1/F
* NV/RHZ AT 1/F FREQ
.PARAM NLF=50
* FREQ FOR 1/F VAL
.PARAM FLW=1
* SET UP VNSE FB
* NV/RHZ FLATBAND
.PARAM NVR=5
* END USER INPUT
* START CALC VALS
.PARAM GLF={FLW^0.25*NLF/1164}
.PARAM RNV={1.184*NVR^2}
.MODEL DVN D KF={FLW^0.5/1E11} IS=1.0E-16
* END CALC VALS
I1 0 7 10E-3
I2 0 8 10E-3
D1 7 0 DVN
D2 8 0 DVN
E1 3 6 7 8 {GLF}
E3 30 40 3 4 1
R1 3 0 1E9
R2 3 0 1E9
  
```

TEXAS INSTRUMENTS

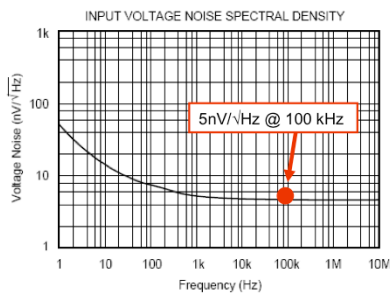
The net list for the noise voltage source should look like the screenshot on the right.

First let's enter the 1/f noise. 1/f noise is controlled by two values: the noise spectral density in nV/√Hz and the frequency at which it's measured. In this case, the noise is 50nV/√Hz, so 50 is entered for the parameter NLF. This noise value is measured at 1Hz, so the frequency FLW is set to 1. When selecting the 1/f noise voltage, make sure that this point is selected at the lowest frequency on the curve. This will insure that the 1/f component in the noise model is dominant and the broadband noise isn't affecting the value much. This will give the most accurate result for your simulation.



## Broadband Region

Look for a point in the broadband region.  
Enter the magnitude at this point.

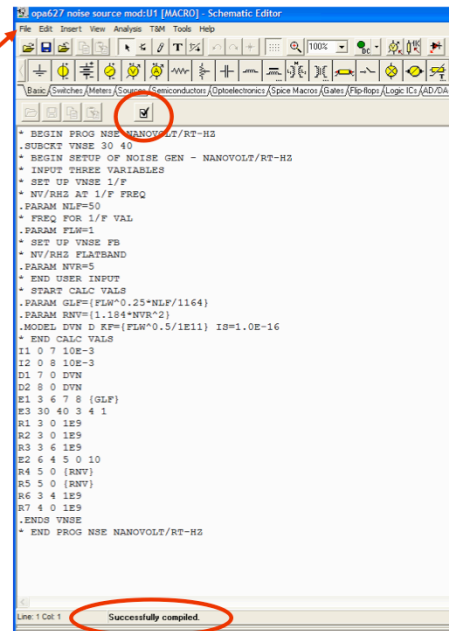


```
opa627 noise source mod:U1 [MACRO] - Schematic Editor
File Edit Insert View Analysis T&M Tools Help
Basic/Switches/Meters/Sources/Semiconductors/Optoelectronics/Spice Macros/Gates/
* BEGIN PROG NSE NANOVOLT/RT-HZ
.SUBCKT VNSE 30 40
* BEGIN SETUP OF NOISE GEN - NANOVOLT/RT-HZ
* INPUT THREE VARIABLES
* SET UP VNSE 1/F
* NV/RHZ AT 1/F FREQ
.PARAM NLF=50
* FREQ FOR 1/F VAL
.PARAM FLW=1
* SET UP VNSE FB
* NV/RHZ FLATBAND
.PARAM NVR=5
* END USER INPUT
* START CALC VALS
.PARAM GLF={FLW^0.25*NLF/1164}
.PARAM RNV={1.184*NVR^2}
.MODEL DVN D KF={FLW^0.5/1E11} IS=1.0E-16
* END CALC VALS
I1 0 7 10E-3
I2 0 8 10E-3
D1 7 0 DVN
D2 8 0 DVN
E1 3 6 7 8 {GLF}
E3 30 40 3 4 1
R1 3 0 1E9
R2 3 0 1E9
```

The broadband region only requires one entry in the model. The voltage noise spectral density in nV/√Hz in the broadband region is entered under the parameter NVR. In this example the broadband noise is 5nV/√Hz, so NVR is set to 5. Note that the frequency does not need to be entered because broadband noise is constant over frequency.

## Compile the Macro

After macro is compiled press "File → Close Netlist Viewer" to return to schematic editor.



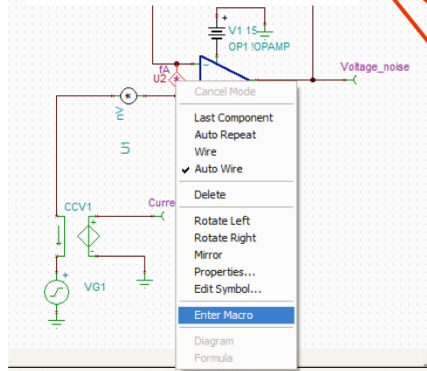
```
* BEGIN PROG NSE NANOVOLT/RT-H2
.SUBCKT VNSE 30 40
* BEGIN SETUP OF NOISE GEN - NANOVOLT/RT-H2
* INPUT THREE VARIABLES
* SET UP VNSE 1/F
* NV/RHZ AT 1/F FREQ
.PARAM NLF=50
* FREQ FOR 1/F VAL
.PARAM FLN=1
* SET UP VNSE FB
* NV/RHZ FLATBAND
.PARAM NVR=5
* END USER INPUT
* START CALC VALS
.PARAM GLF=(FLN*0.25*NLF/1164)
.PARAM RNV=(1.184*NVR*2)
.MODEL DYN D FF=(FLN*0.5/1E11) IS=1.0E-16
* END CALC VALS
I1 0 7 10E-3
I2 0 8 10E-3
D1 7 0 DYN
D2 8 0 DYN
E1 3 6 7 8 {GLF}
E3 30 40 3 4 1
R1 3 0 1E9
R2 3 0 1E9
R3 3 6 1E9
E2 6 4 5 0 10
R4 5 0 {RNV}
R5 5 0 {RNV}
R6 3 4 1E9
R7 4 0 1E9
.ENDS VNSE
* END PROG NSE NANOVOLT/RT-H2
```

Line: 1 Col: 1 **Successfully compiled.**

The final step for the voltage noise source is to click the compile check box. If the updated net list has no errors, you will see the message "Successfully compiled" at the bottom of the window. Once the macro is compiled, click "File" then "Close Netlist Viewer" to return to the schematic editor.

## Current Noise Source

Follow the same procedure for current noise. This example has no 1/f component, so set FLWF = 0.001.



```
* BEGIN PROG NSE FEMTO AMP/RT-HZ
* SUBCKT FEMTO 1 2
* BEGIN SETUP OF NOISE GEN - FEMPTOAMPS/RT-HZ
* INPUT THREE VARIABLES
* SET UP INSE 1/F
* FA/RHZ AT 1/F FREQ
PARAM NLFF=2.5
FREQ FOR 1/F VAL
PARAM FLWF=0.001
* SET UP INSE FB
* FA/RHZ FLATBAND
PARAM NVRF=2.5
* END USER INPUT
* START CALC VALS
```

PARAMETER	OPA627BM, BP, SM OPA637BM, BP, SM			UNITS
	MIN	TYP	MAX	
<b>NOISE</b>				
Input Voltage Noise				
Noise Density, f = 10Hz		15	40	nV/√Hz
f = 100Hz		8	20	nV/√Hz
f = 1kHz		5.2	8	nV/√Hz
f = 10kHz		4.5	6	nV/√Hz
Voltage Noise, BW = 0.1Hz to 10Hz		0.6	1.6	μVp-p
Input Bias Current Noise				
Noise Density, f = 100Hz	1.6	2.5	4	fA/√Hz
Current Noise, BW = 0.1Hz to 10Hz	30	60	100	fAp-p

TEXAS INSTRUMENTS

11

The same procedure is done for the current noise source. In this example the current noise source does not have a 1/f region. So, set the 1/f noise the same as the broadband noise and enter the 1/f frequency at a very low frequency value, such as 0.001.

## Important Op Amp Characteristics

<b>OPEN-LOOP GAIN</b>					
Open-Loop Voltage Gain	$V_O = \pm 10V, R_L = 1k\Omega$	112	120		dB
Over Specified Temperature	$V_O = \pm 10V, R_L = 1k\Omega$	106	117		dB
SM Grade	$V_O = \pm 10V, R_L = 1k\Omega$	100	114		dB
<b>FREQUENCY RESPONSE</b>					
Slew Rate: OPA627	$G = -1, 10V \text{ Step}$	40	55		V/ $\mu$ s
OPA637	$G = -4, 10V \text{ Step}$	100	135		V/ $\mu$ s
Settling Time: OPA627 0.01%	$G = -1, 10V \text{ Step}$		550		ns
0.1%	$G = -1, 10V \text{ Step}$		450		ns
OPA637 0.01%	$G = -4, 10V \text{ Step}$		450		ns
0.1%	$G = -4, 10V \text{ Step}$		300		ns
Gain-Bandwidth Product: OPA627	$G = 1$		16		MHz
OPA637	$G = 10$		80		MHz
Total Harmonic Distortion + Noise	$G = +1, f = 1kHz$		0.00003		%

$$A_{OL} = 10^{(120\text{dB}/20)} = 1E6 \text{ (from data sheet)}$$

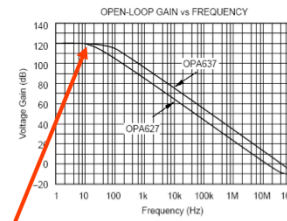
$$GBW = 16\text{MHz} \text{ (from data sheet)}$$

$$\text{Dominant Pole} = GBW / OLG = 16\text{MHz}/1E6 = 16\text{Hz}$$

where:

**GBW** – Unity Gain-Bandwidth Product

**$A_{OL}$**  – Open Loop Gain



**Dominant Pole**

TEXAS INSTRUMENTS

12

The final requirement is to make some modifications to the generic op-amp model. First, find the linear equivalent of the decibel open loop gain, or  $A_{OL}$ , given in the data sheet. In this example 120dB translates to  $1 \times 10^6$ . Next, find the op amp's gain bandwidth product spec, or GBW. Divide the GBW by the linear  $A_{OL}$  to calculate the dominant pole. In this example, the GBW of 16 MHz divided by the linear  $A_{OL}$  of  $10^6$  gives a dominant pole of 16 Hz. This information will be directly entered into the model.

## Edit Generic Op Amp Macromodel

The image shows a circuit schematic with an operational amplifier (OP1) and various components (resistors R1, R2, capacitors V1, V2, voltage source VG1, and nodes V627). Two dialog boxes are overlaid on the schematic:

- OP1 - Operational amplifier**: A properties window for the OP1 component. It shows fields for Label Name (OP1), Parameters (Parameters), Type (IOPAMP), Compensation node1, Compensation node2, Temperature (Relative), Temperature [°C] (0), and Fault (None).
- Catalog Editor**: A window for editing the model parameters. It shows a list of models (Standard, IOPAMP, LM318, LM709, LM318, LM709, TL061C, TL071C, TL081C, uA741, uA741A, uA741C) and a table of parameters for the selected model (IOPAMP):

Parameter	Value
Usage: General	
Open loop gain	1M
Input resistance [Ohm]	2M
Output resistance [Ohm]	55
Maximum slew rate [V/s]	5M
Dominant pole [Hz]	1E
Second pole [Hz]	100G
Input offset voltage [V]	0
Input bias current [A]	0
Input offset current [A]	0
Offset voltage tco [V/C]	0
Current doubling int. [C]	10
Outp. offs. lim. [Vcc+][V]	2

1. Double Click on Op amp
2. Press "Type" Button
3. Edit "Open loop gain" and "Dominant Pole" values

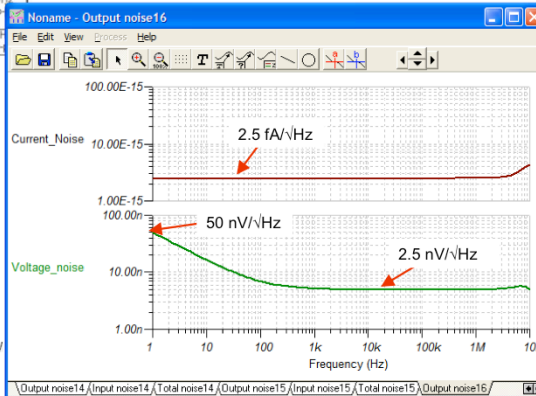
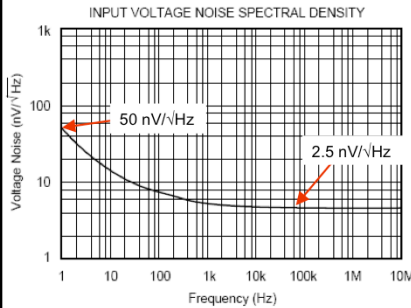
1/13

TEXAS INSTRUMENTS

Double-click on the op amp schematic symbol to open the properties window. Click the button in the "Type" field to access the op amp specifications. Here, enter the open loop gain and dominant pole into the generic op-amp model. At this point, the op amp and noise sources contain all the information needed for noise simulations. Let's do a final simulation to confirm that we didn't make any mistakes.

## Verify the Noise Model

PARAMETER	OPA627BM, BP, SM OPA637BM, BP, SM			UNITS
	MIN	TYP	MAX	
<b>NOISE</b>				
Input Voltage Noise				
Noise Density, $f = 10\text{Hz}$		15	40	$\text{nV}/\sqrt{\text{Hz}}$
$f = 100\text{Hz}$		8	20	$\text{nV}/\sqrt{\text{Hz}}$
$f = 1\text{kHz}$		5.2	8	$\text{nV}/\sqrt{\text{Hz}}$
$f = 10\text{kHz}$		4.5	6	$\text{nV}/\sqrt{\text{Hz}}$
Voltage Noise, BW = 0.1Hz to 10Hz		0.6	1.6	$\mu\text{Vp}$
Input Bias Current Noise		1.6	2.5	$\text{fA}/\sqrt{\text{Hz}}$
Noise Density, $f = 100\text{Hz}$		30		$\text{fA}/\sqrt{\text{Hz}}$
Current Noise, BW = 0.1Hz to 10Hz				



TEXAS INSTRUMENTS

14

Using the noise test circuit introduced at the beginning of this video, re-run the noise simulation. After our adjustments, the model now matches the data sheet! This op amp model can now be used in any application circuit to measure noise. After you build a few of these noise models, you will find that it is a relatively simple procedure. In some cases, you may even choose to develop your own noise model even if an accurate one exists. The reason is that a customized noise model provides the flexibility to adjust or eliminate noise sources. This gives you the ability to experiment and determine the dominant source of noise.

**Thanks for your time!  
Please try the quiz.**

15



That concludes this video – thank you for watching! Please try the quiz to check your understanding of this video’s content.