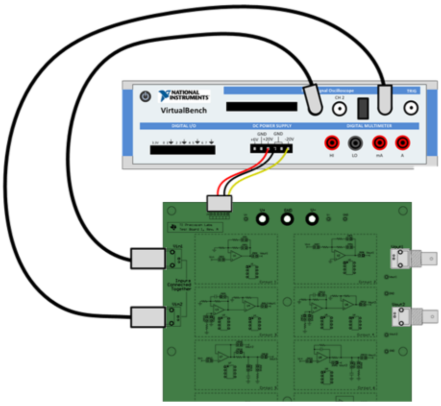
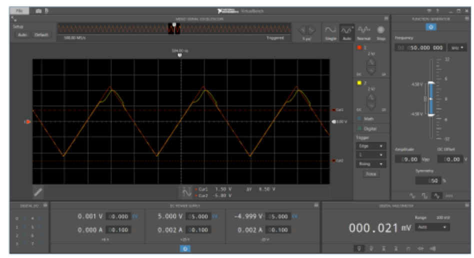
Lab Manual

TI Precision Labs – Op Amps





TI Designs
PRECISION

Texas Instruments

Hello, and welcome to the TI Precision Labs presentation which gives an overview of the National Instruments VirtualBench. The VirtualBench is a very powerful piece of hardware which combines multiple pieces of traditional lab equipment into one compact and easy-to-use device. This presentation will describe the features of the VirtualBench, as well as give a tutorial on how to configure the VirtualBench hardware and software.

VirtualBench Specifications

Mixed-Signal Oscilloscope	
Bandwidth	100 MHz
Channels	2 analog, 34 digital
Ranges	10mV/div, 100mV/div,….10V/div
Sampling Rate	1 GS/s (single channel), 500 MS/s/ch (dual channel)
Waveform Measurements	cursors, 22 automatic measurements
Waveform Math	add, subtract, multiply, FFT
Record Length	1 million samples

Function Generator	
Max Frequency	20 MHz (sine), 5 MHz (square)
Channels	1
Waveform Types	sine, square, ramp, triangle, DC

I won't go into detail on all of the specifications, but they are copied here. Some key specs to note are the 100 MHz oscilloscope bandwidth, up to 1 gigasample per second sampling rate, and up to 20MHz sinusoidal function generator.

VirtualBench Specifications

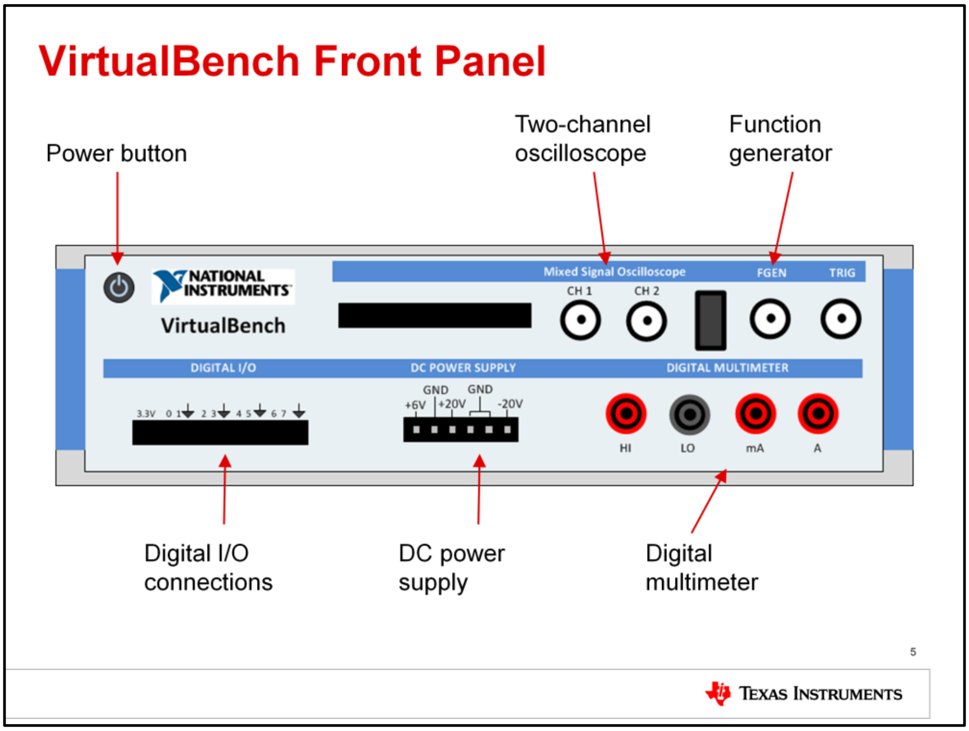
Digital Mulitimeter	
Resolution	5 ½ digits
Measurement Functions	VDC, VAC, IDC, IAC, continuity, resistance, diode
Max Voltage	300 V max input voltage
Max Current	10 A max input current
Basic Accuracy	up to 0.015% VDC

Programmable DC Power Supply	
Channels	3
Voltage/Current (Ch1)	0 to +6V / 0 to 1A
Voltage/Current (Ch2)	0 to +25V / 0 to 0.5A
Voltage/Current (Ch3)	0 to -25V / 0 to 0.5A

4

TEXAS INSTRUMENTS

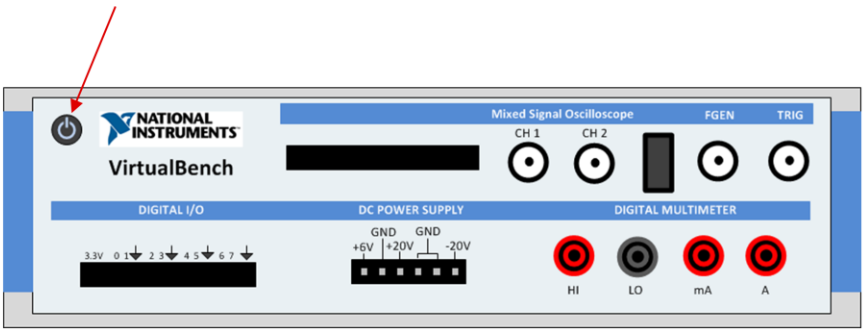
The specs are continued here for the multimeter and power supply. The multimeter has 5 and a half digits of resolution, and the three-channel power supply can support both single supply and split supply devices.



The front panel of the virtual bench is used to make connections to your system under test. You must turn on the VirtualBench by pressing the power button on the top left before using the device. Then, you can connect to the oscilloscope, function generator, digital I/O, DC power supply, and digital multimeter as required in your experiments.

VirtualBench Front Panel

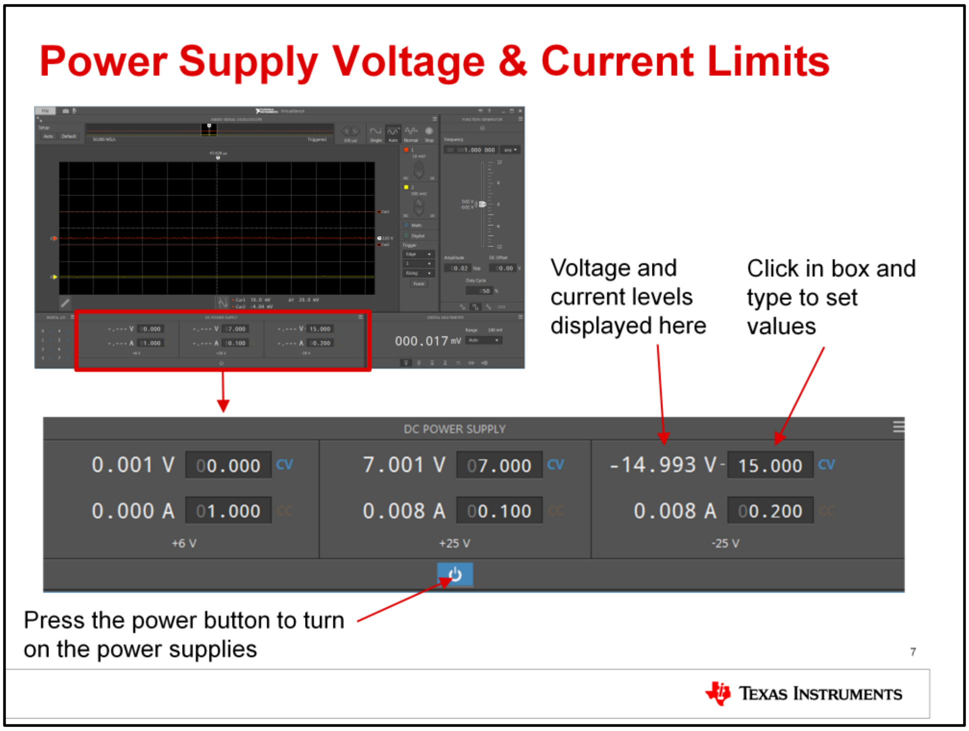
Only press the On/Off Power button when instructed. This button will cause the software to auto boot on the laptop. Software booting is slow, please be patient.



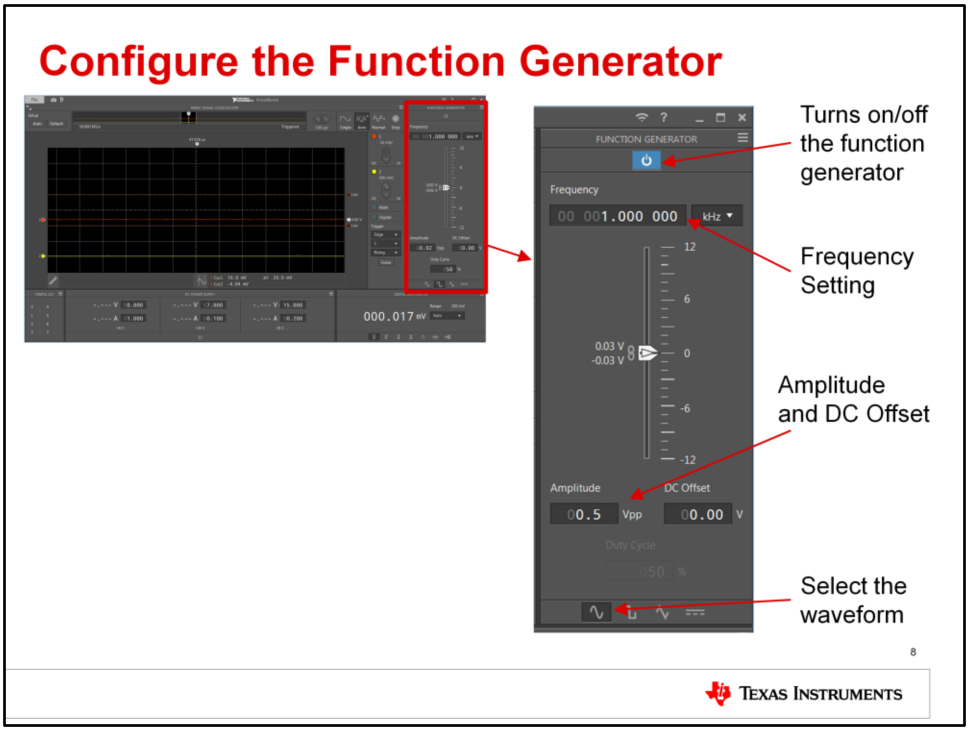
6

TEXAS INSTRUMENTS

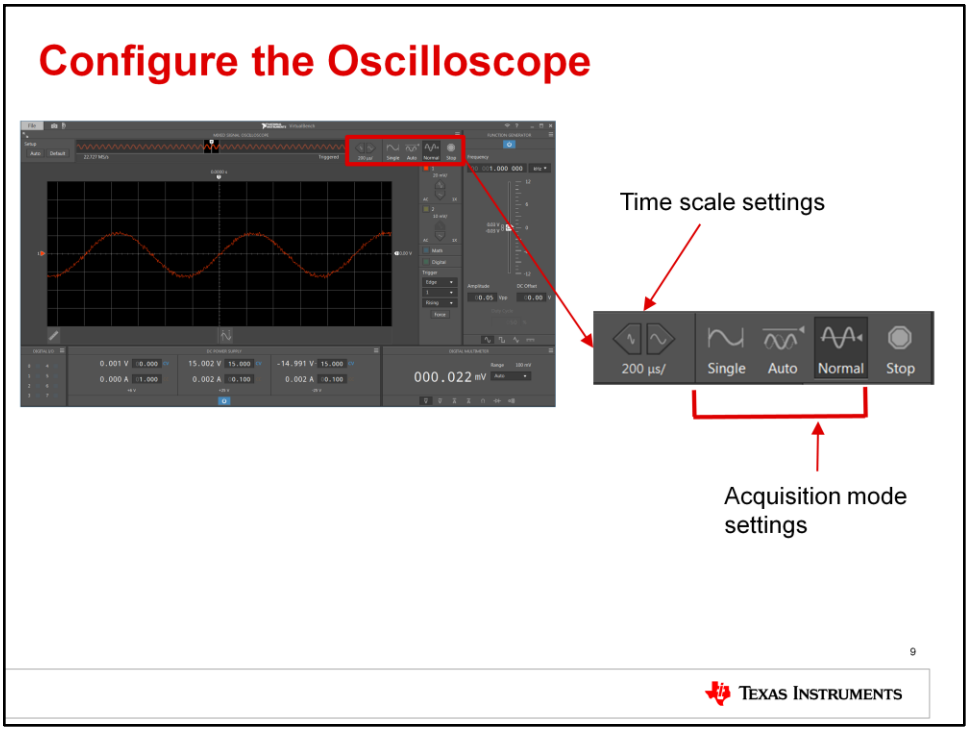
Make sure that you only press the power when instructed. Power to the circuit is controlled using the software interface on the laptop. The front pannel button is only used when you want to boot the software from initial start up.



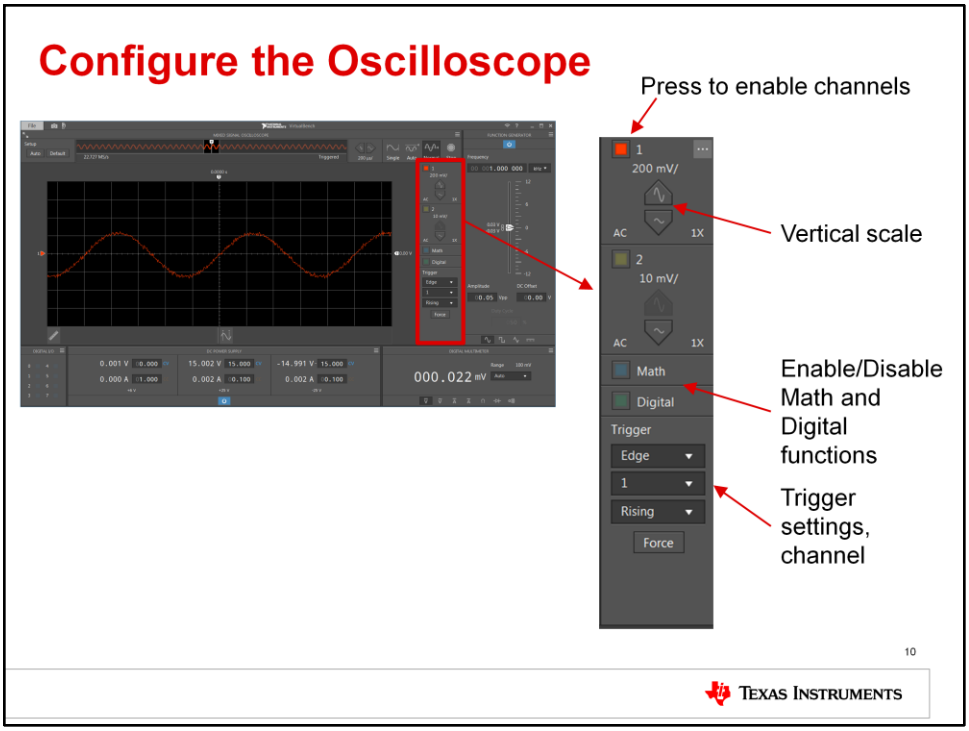
This slide shows the location of power supply voltage and current settings. Also, the button in the center turns on and off the supplies.



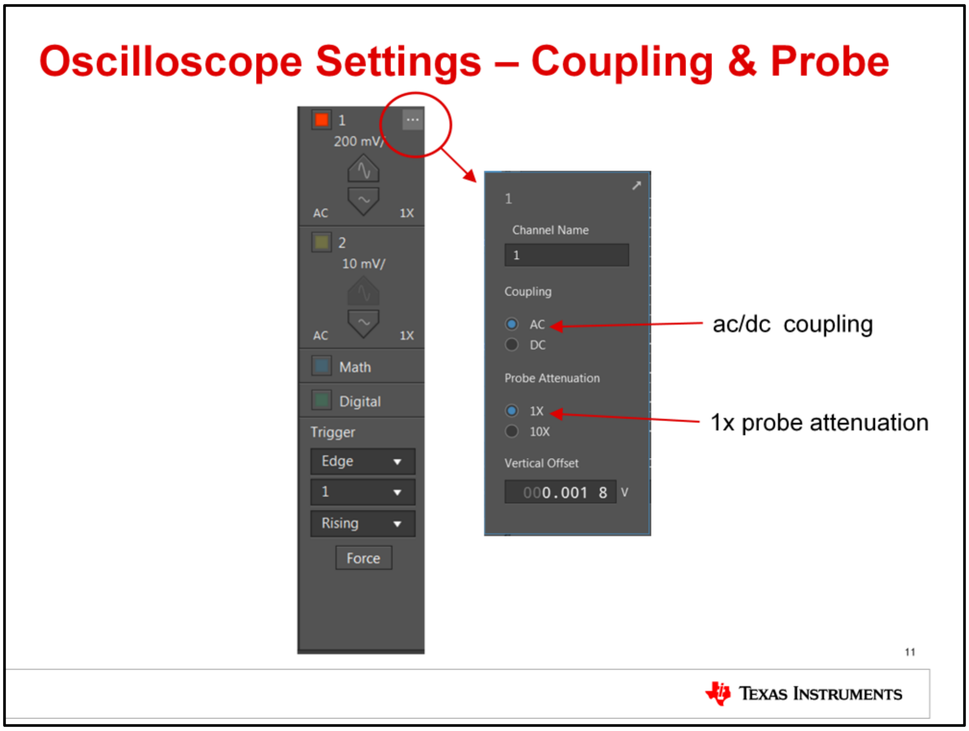
This slide shows how to set the function generator.



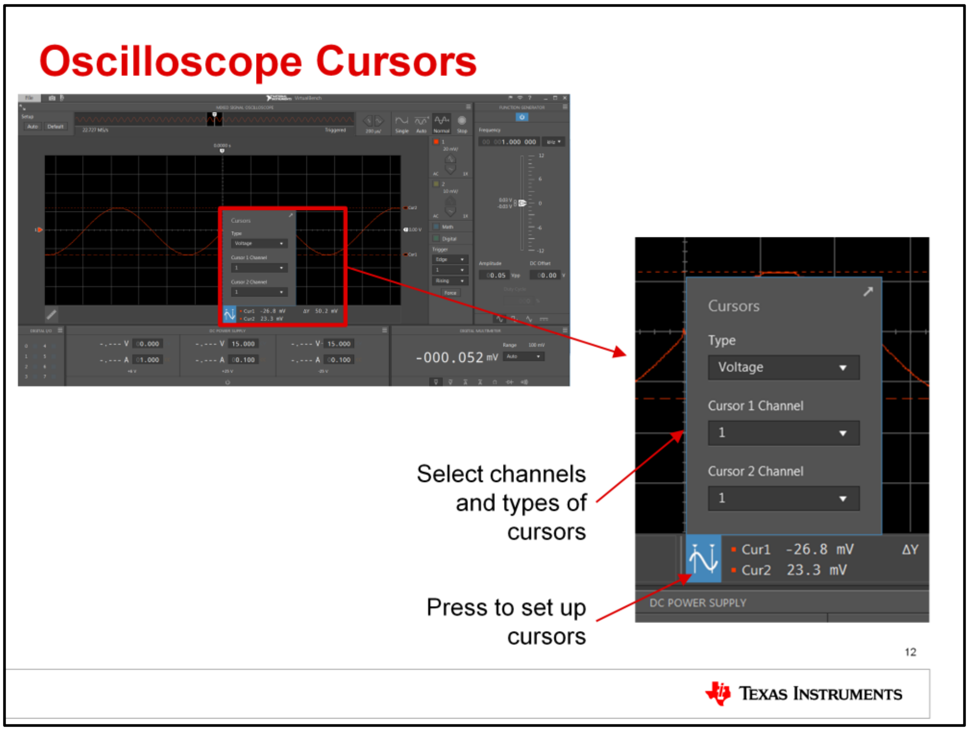
Here we set the horizontal time base on the scope as well as the triggering mode. “Auto” is untriggered roll or automatic triggering. Normal is triggered. For periodic signals it is best to use normal mode.



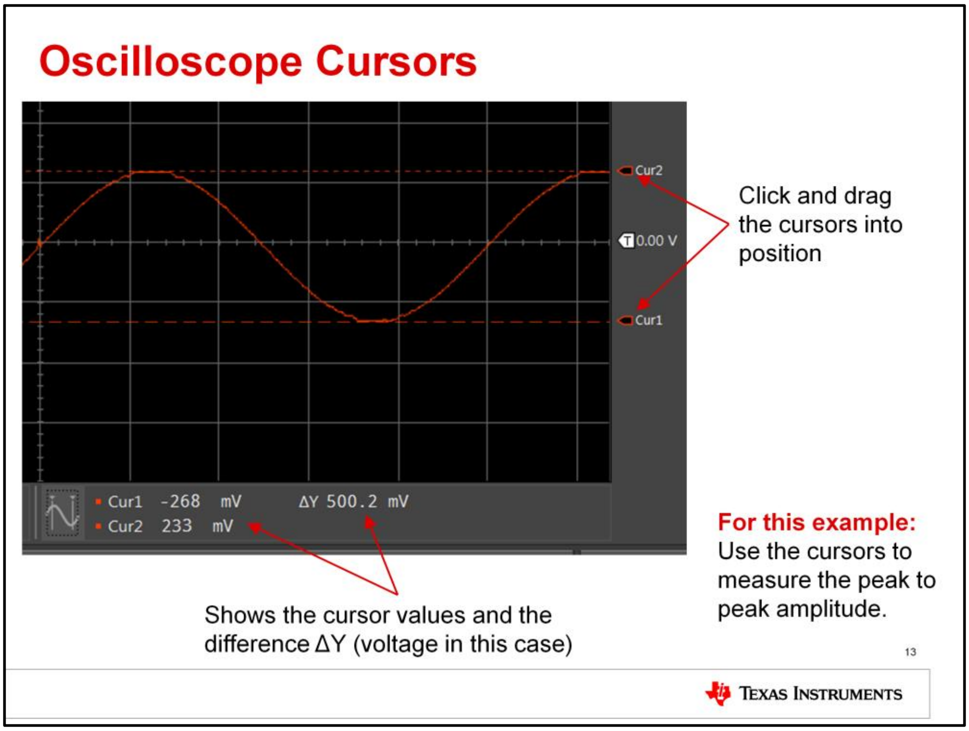
Here we can set the vertical scale on the scope



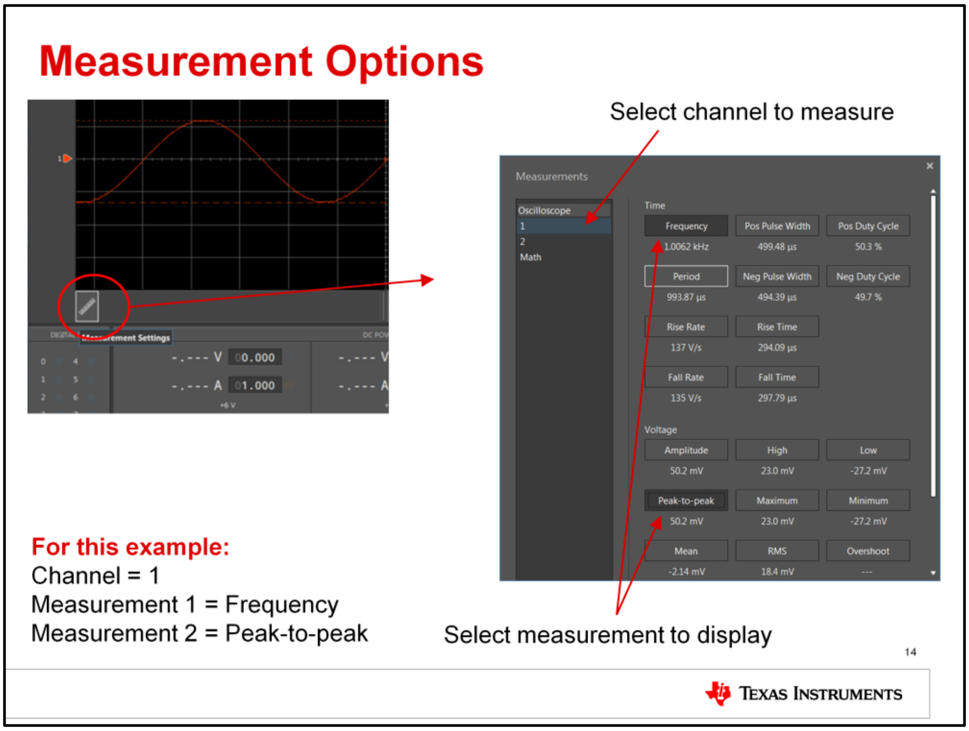
The ac/dc coupling is in a hidden button. Hover your mouse above the corner and click for these menus



Cursors may be used to measured oscilloscope signals. Click the cursor icon, located above the power supply settings, to open the cursor menu. Select voltage type, then select the channel for the cursors.

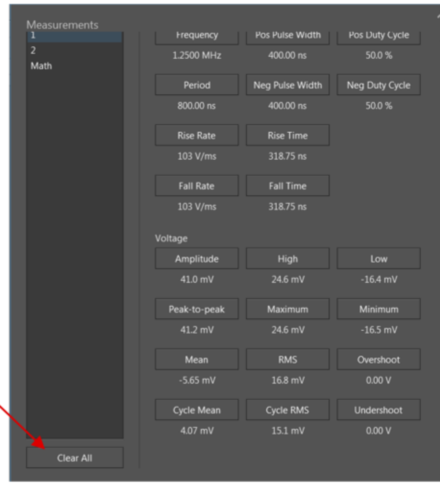


The cursors location may be adjusted to make a measurement. Click and drag the cursor indicators into position at the top and bottom of the sinusoidal waveform. The voltage level at each cursor, as well as the delta Y, in this case voltage difference, will be displayed next to the cursor icon. Use the cursors to measure the signal's peak-to-peak amplitude.



The VirtualBench can also make measurements automatically. Click the ruler icon, above the digital I/O settings, to open the measurement options window. Here you can select what channel to measure, and which measurement you want to display.

More on the Measurement Options

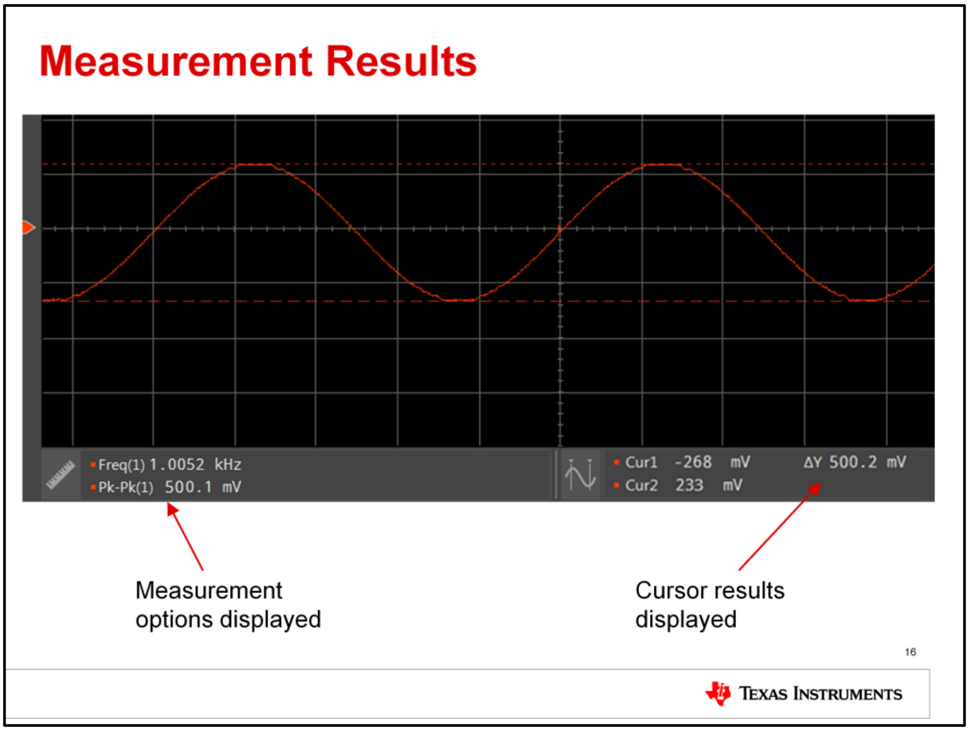


Use "Clear All" to remove old measurements

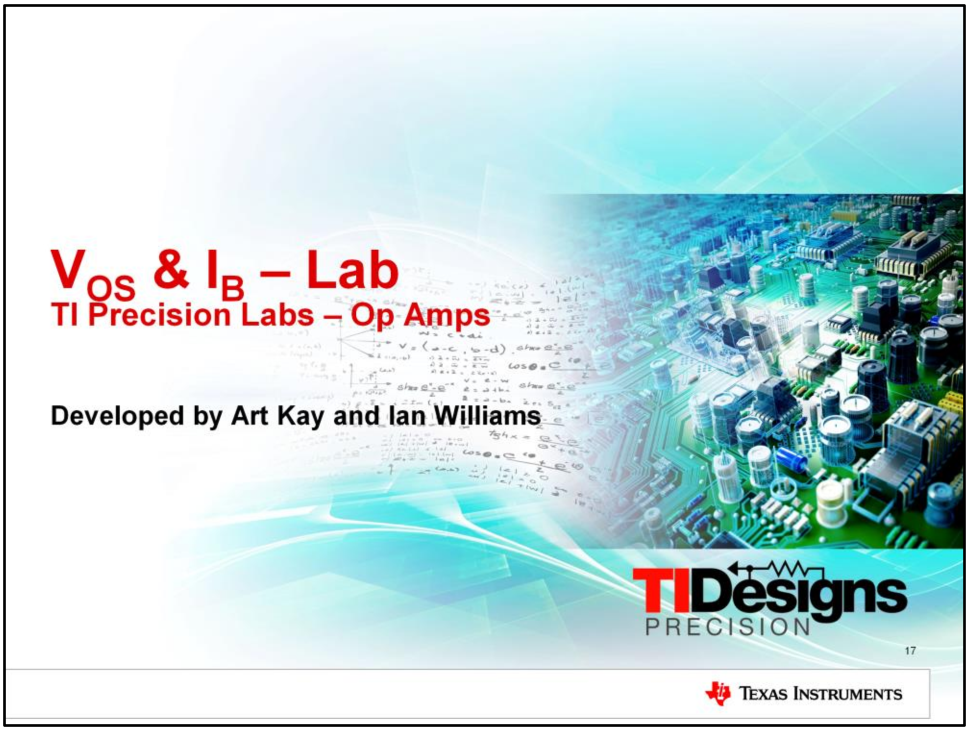
Use slider to see all options

Texas Instruments

"Clear All" and the slider feature can help to use this effectively.



The measurement results are displayed next to the ruler icon. You should read a result of 1kHz, 500mVpp – the same as the function generator. The amplitude should be the same as the delta Y measurement from the cursor.



Hello, and welcome to the presentation for the TI Precision Lab supplement for op amp input offset voltage (VOS) and input bias current (IB). This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the op amp VOS and IB lecture.

Required/Recommended Equipment

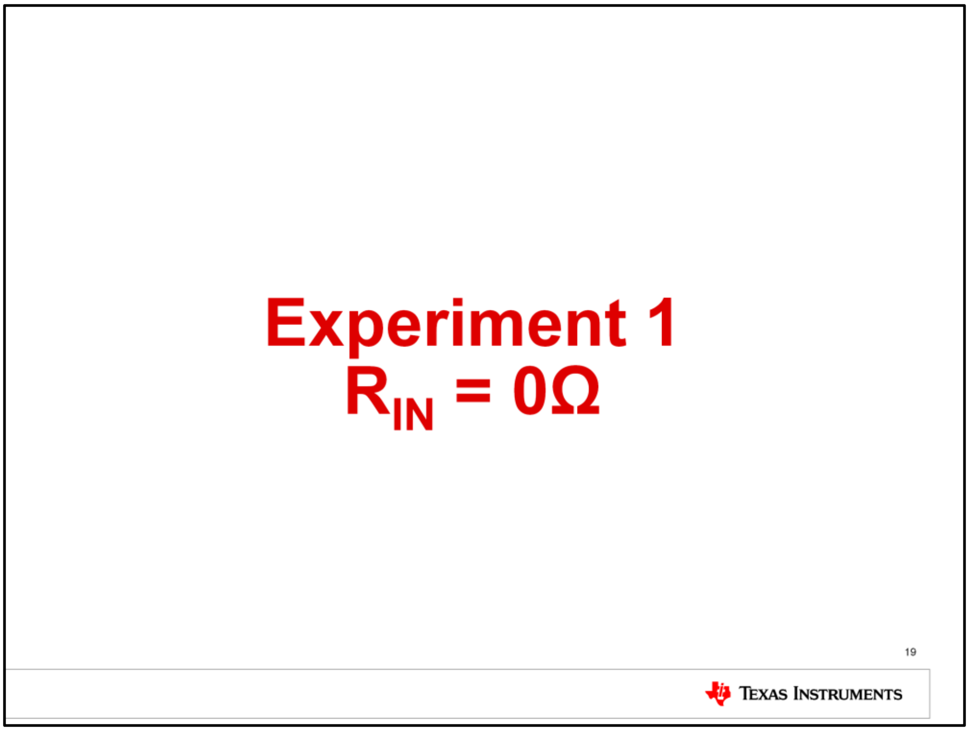
- Calculation
  - Pencil and paper
  - **Recommended:** MathCAD, Excel, or similar
- Simulation
  - SPICE simulation software
  - **Recommended:** TINA-TI™
- Measurement
  - TI Precision Labs PCB from Texas Instruments
  - Oscilloscope
  - ±12V power supply
  - **Recommended:** National Instruments VirtualBench™

Texas Instruments 18

The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly.

The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

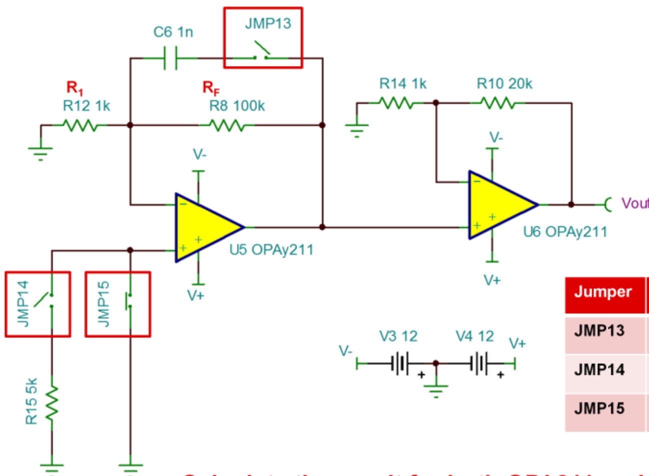
Finally, the real-world measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any oscilloscope and ±12V power supply. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.



In experiment 1, we'll determine the effects of VOS and IB in a circuit where the input resistance, RIN, is equal to 0 ohms.

Calculation – $R_{IN} = 0\Omega$

Calculate the total output voltage due to V_{OS} and I_B for the circuit shown below. Use the typical and maximum values for V_{OS} and I_B given on the next slide.



Jumper	Position	Description
JMP13	OPEN	C5 disconnected
JMP14	OPEN	R15 disconnected
JMP15	CLOSED	VIN+ connected to GND

Calculate the result for both OPA211 and OPA188!

Texas Instruments 20

First, calculate the expected total output voltage due to VOS and IB for the circuit shown here, using the techniques and equations given in the VOS and IB lecture. Take note of the jumper positions in the table. JMP13 and JMP14 are open, and JMP15 is closed. JMP15 shorts the non-inverting input of U5 to ground, causing RIN to be 0 ohms.

Calculate the output voltage twice – first with the OPA211 selected for U5 and U6, then with the OPA188. The different parameters of these op amps will give you different results.

Calculation – R_{IN} = 0Ω

PARAMETER		OPA211			
		MIN	TYP	MAX	UNIT
Input Offset Voltage	V_{OS}		±30	±125	µV
Input Bias Current	I_B		±60	±175	nA

PARAMETER		OPA188			
		MIN	TYP	MAX	UNIT
Input Offset Voltage	V_{OS}		±6	±25	µV
Input Bias Current	I_B		±160	±1400	pA

Device	Typical Output	Maximum Output
OPA188	±13mV	±56mV
OPA211	±191mV	±635mV

21

Texas Instruments

In order to perform the calculations, you need to know the typical and maximum values of VOS and IB for each op amp. Those values are given here.

Enter your answers in the table at the bottom of the slide. The solutions are already provided to allow you to check your work.

Calculation – $R_{IN} = 0\Omega$

OPA211

Calculate typical output error from Vos & Ib

$i_b = \pm 60\text{nA}$

$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1\text{k}\Omega$

$V_{ib} = i_b R_{eq} = \pm 60\mu\text{V}$

$V_{os} = \pm 30\mu\text{V}$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 30\mu\text{V} + \pm 60\mu\text{V})$

$V_{out\_error} = \pm 191\text{mV or } \pm 64\text{mV}$

$V_{out\_error} = \pm 191\text{mV}$ Choose the largest

Calculate maximum output error from Vos & Ib

$i_b = \pm 175\text{nA}$

$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1\text{k}\Omega$

$V_{ib} = i_b R_{eq} = \pm 175\mu\text{V}$

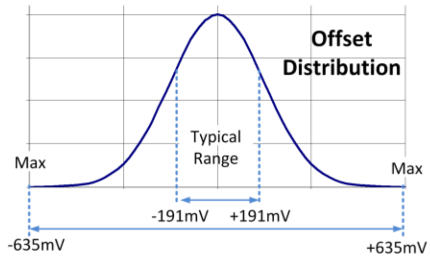
$V_{os} = \pm 125\mu\text{V}$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 125\mu\text{V} + \pm 175\mu\text{V})$

$V_{out\_error} = \pm 106\text{mV or } \pm 636\text{mV}$

$V_{out\_error} = \pm 635\text{mV}$ Choose the largest



22 TEXAS INSTRUMENTS

One important point to note is that IB and VOS can be positive or negative. This means that there are different possibilities for the output voltage due to VOS and IB which must all be considered.

First, calculate R_EQ, the equivalent input resistance, then multiply R_EQ by IB to determine the input voltage due to IB. Next, use the equation Vout = G1, gain of the first stage, times G2, gain of the second stage, times the sum of VOS and VIB to calculate the total output. Again, there are four possibilities. Pick the largest value.

Repeat the same steps, using the maximum values instead of the typical values.

Calculation – $R_{IN} = 0\Omega$

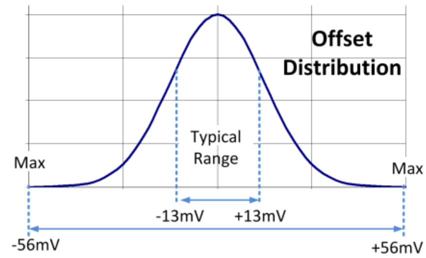
OPA188

Calculate typical output error from Vos & Ib

$$i_b = \pm 160pA$$
$$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1k\Omega$$
$$V_{ib} = i_b R_{eq} = \pm 0.16\mu V$$
$$V_{os} = \pm 6\mu V$$
$$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$$
$$V_{out\_error} = 101 * 21(\pm 6\mu V + \pm 0.16\mu V)$$
$$V_{out\_error} = \pm 13mV \text{ or } \pm 12mV$$
$V_{out\_error} = \pm 13mV$ Choose the largest

Calculate maximum output error from Vos & Ib

$$i_b = \pm 1400pA$$
$$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1k\Omega$$
$$V_{ib} = i_b R_{eq} = \pm 1.4\mu V$$
$$V_{os} = \pm 25\mu V$$
$$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$$
$$V_{out\_error} = 101 * 21(\pm 25\mu V + \pm 1.4\mu V)$$
$$V_{out\_error} = \pm 56mV \text{ or } \pm 50mV$$
$V_{out\_error} = \pm 56mV$ Choose the largest



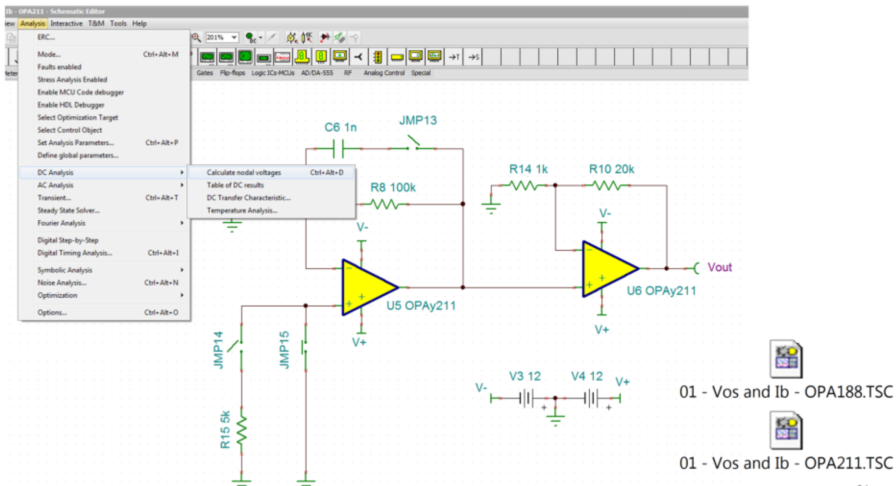
23

Texas Instruments

Repeat the same procedure for the OPA188. The different specifications of the OPA188 will give a different output voltage result in both the typical and maximum case.

Simulation Setup – R_{IN} = 0Ω

Click **Analysis** → **DC Analysis** → **Calculate nodal voltages** to simulate the total output voltage from V_{OS} and I_B.

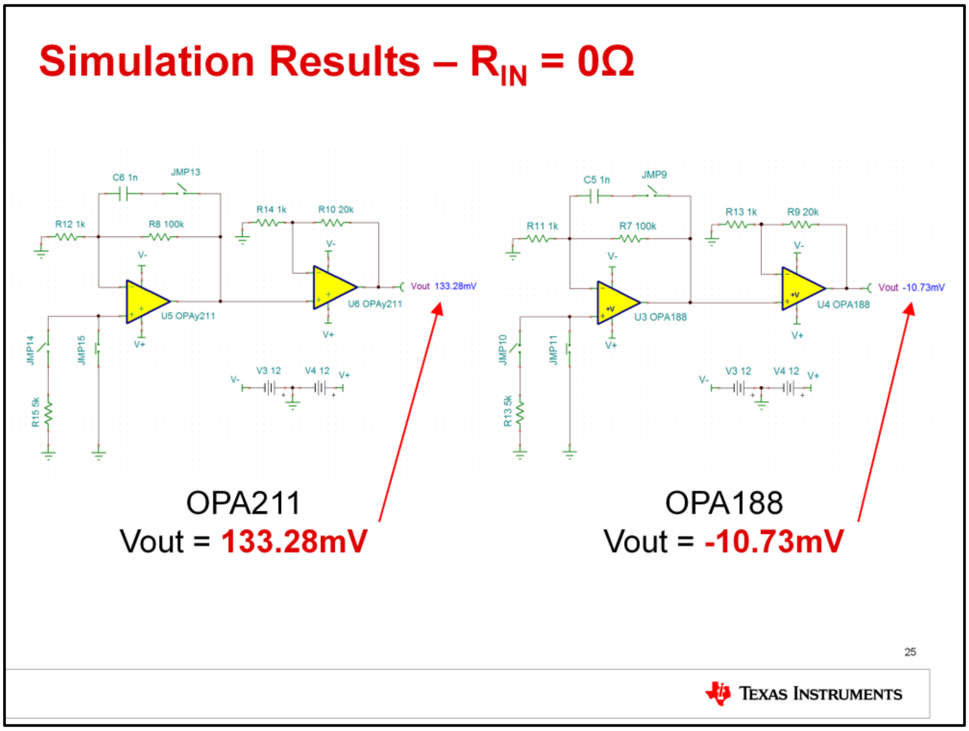


24

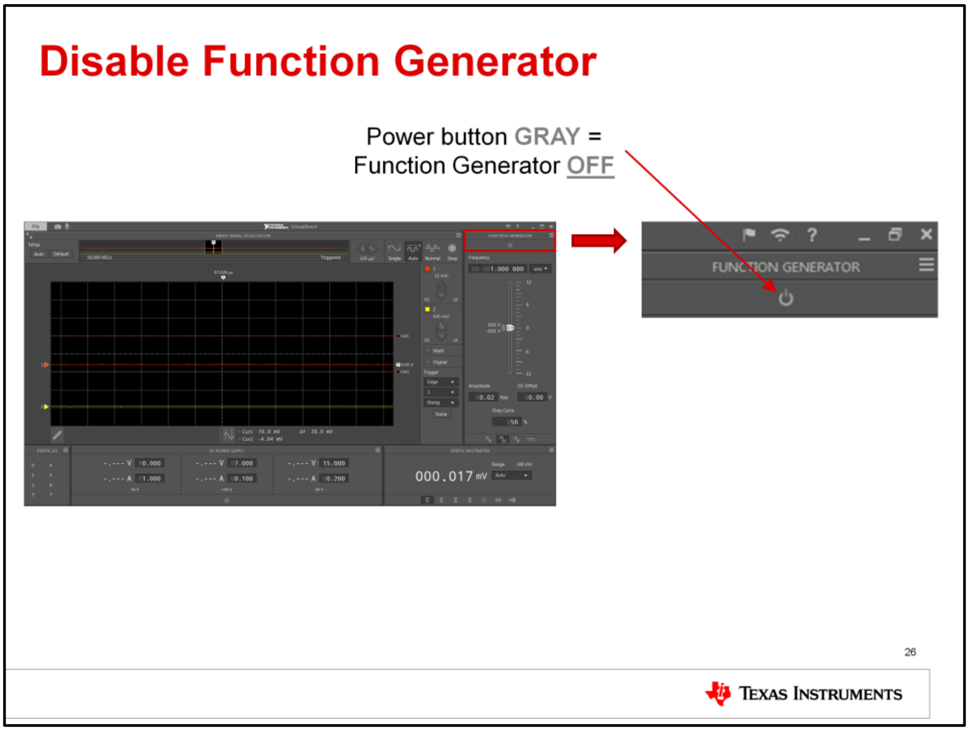
TEXAS INSTRUMENTS

The next step is to run a SPICE simulation analysis for the total DC output voltage. The necessary TINA-TI simulation schematics are embedded in this slide set – simply double-click the icons to open them. Ensure that the jumpers are set correctly. In the OPA211 circuit, JMP13 and JMP14 are open, and JMP15 is closed. In the OPA188 circuit, JMP9 and JMP10 are open, and JMP11 is closed.

To simulate the output voltage, click Analysis → DC Analysis → Calculate nodal voltages.

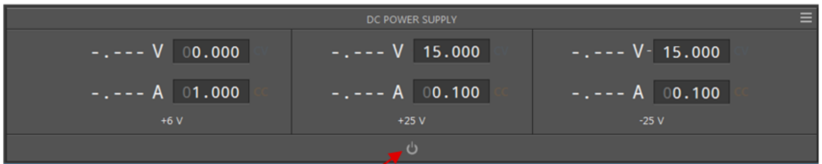


For the OPA211, you should get a result of around 133.28mV. For the OPA188, you should get a result of about -10.73mV.

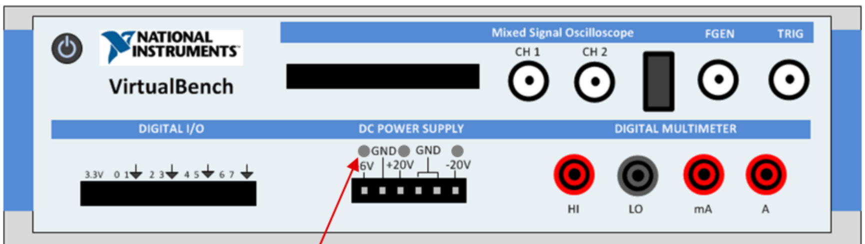


Also make sure the Function Generator is OFF.

Disable DC Power Supply



Power button GRAY = DC power supply OFF

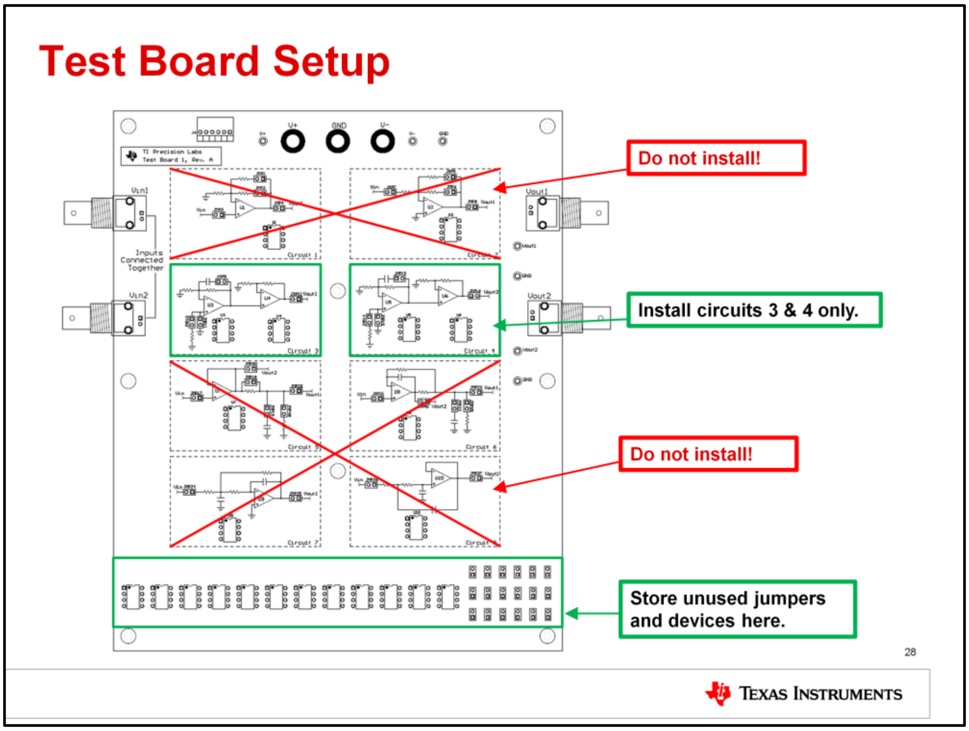


LEDs OFF = DC power supply OFF

27

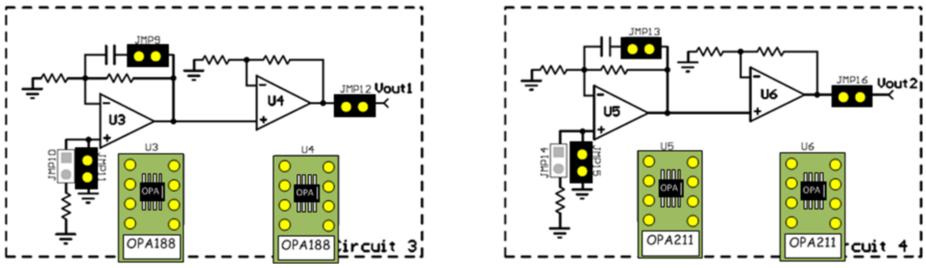
TEXAS INSTRUMENTS

Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuits 3 and 4. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

Test Board Setup – Jumpers



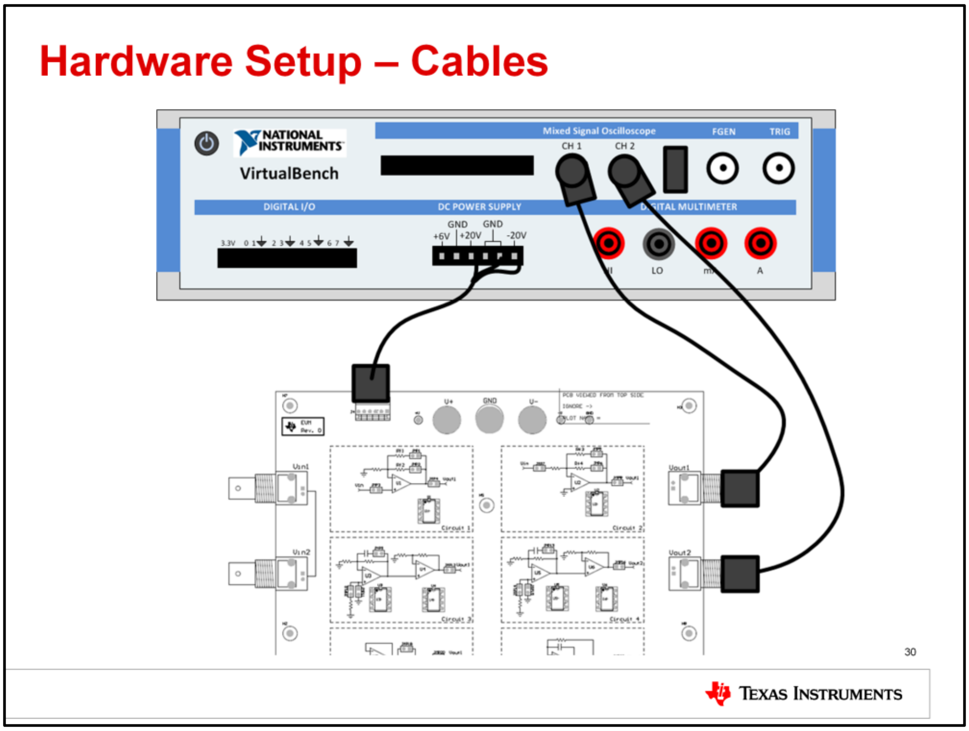
Jumper, Device	Description
JMP9, JMP13	Filter noise with feedback capacitor
JMP11, JMP15	Connect VIN+ to GND
JMP12, JMP16	Connect Circuit 3 to Vout1, connect Circuit 4 to Vout2
U3, U4	Install OPA188
U5, U6	Install OPA211

29

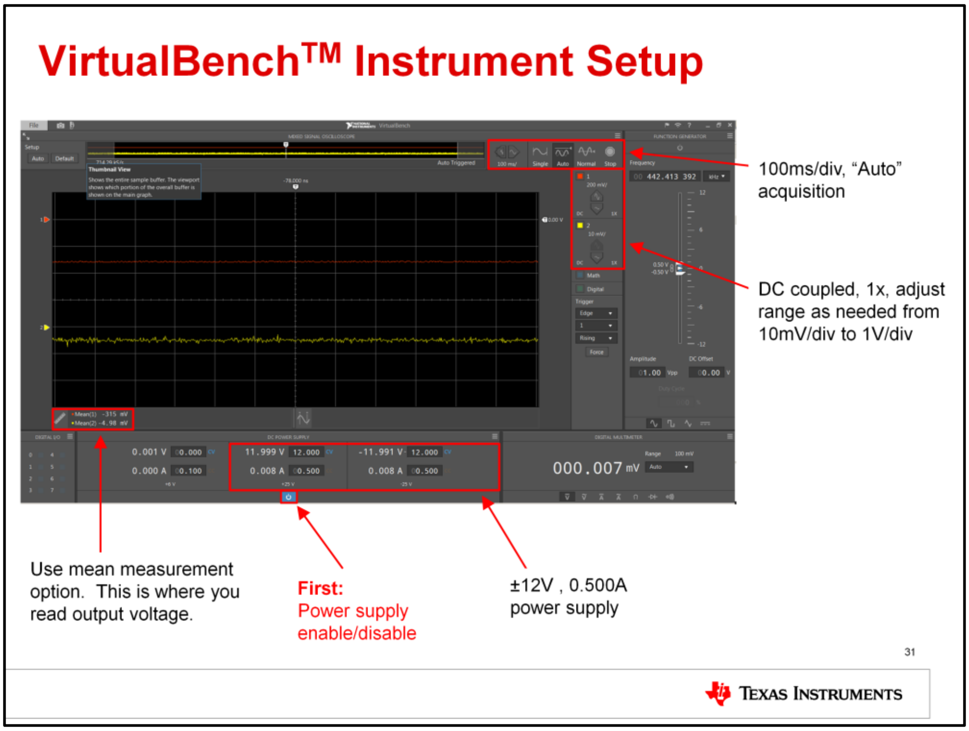
TEXAS INSTRUMENTS

To prepare the test board for the measurement, install the jumpers and devices on circuit 3 and circuit 4 as shown here.

On circuit 3, install JMP9, JMP11, and JMP12, as well as the OPA188 in sockets U3 and U4. On circuit 4, install JMP13, JMP15, and JMP16, as well as the OPA211 in sockets U5 and U6.



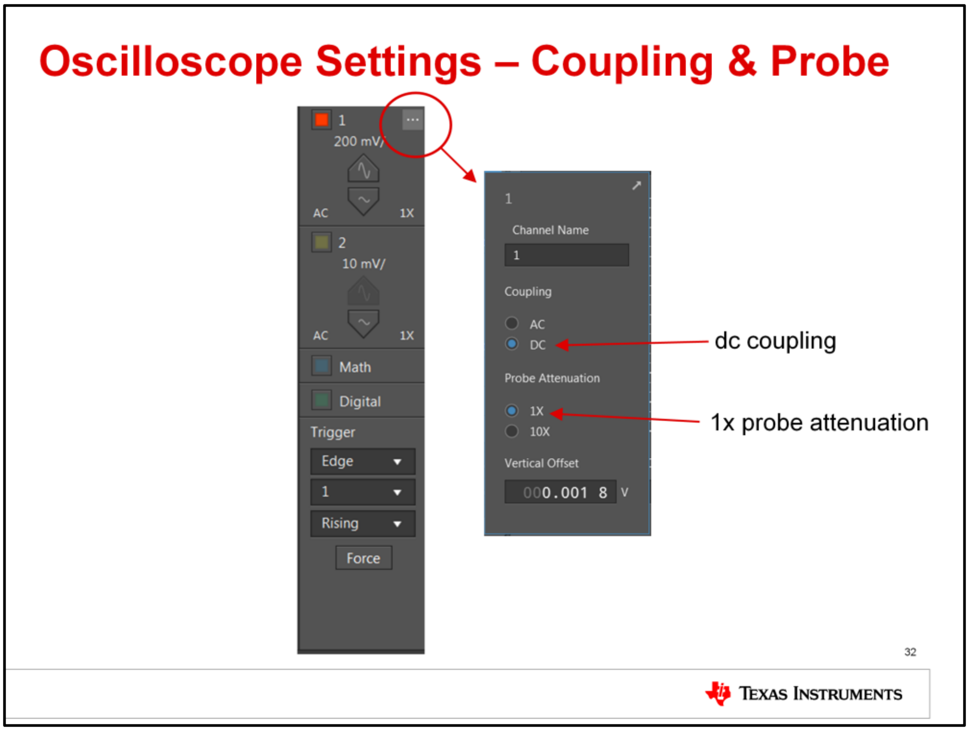
This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board. Connect Vout1 on the test board to VirtualBench oscilloscope channel 1, and Vout2 on the test board to Virtual Bench oscilloscope channel 2, using BNC cables.



Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

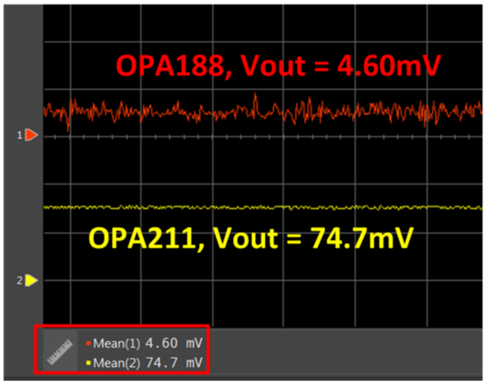
Set the time scale to 100ms per division, with the acquisition mode set to "Auto." Enable channels 1 and 2 on the oscilloscope, and set them to 1x, DC-coupled mode. Adjust the vertical scale as needed from 10mV/div to 1V/div. Set the +25V power supply to +12V, 0.500A. Set the -25V power supply to -12V, 0.500A. Press the power button to turn on the power supply rails.

Enable "mean" measurements on both channels in order to read the output voltage of each circuit.



The icon at the top right of the oscilloscope opens a new window with more options for each channel. Each channel can be given a custom name. Set the coupling mode to dc. Set probe attenuation to 1x.

Measurement Results – R_{IN} = 0Ω



Measured result should be less than calculated maximum

Device	Calculated Maximum	Calculated Typical	Simulated Typical	Measured
OPA188	±56mV	±13mV	-10.73mV	4.60mV
OPA211	±635mV	±191mV	+133.28mV	74.7mV

33

Texas Instruments

The expected output voltage results from the measurement are shown here. The OPA211 has a measured output voltage of 74.7mV, and the OPA188 has a measured output voltage of 4.60mV. You may have different results in your experiment.

How did the measured and simulated results compare to the typical hand calculated results? Take a moment to look over the previous results and draw your own conclusions.

Experiment 2
R_{IN} = 5kΩ

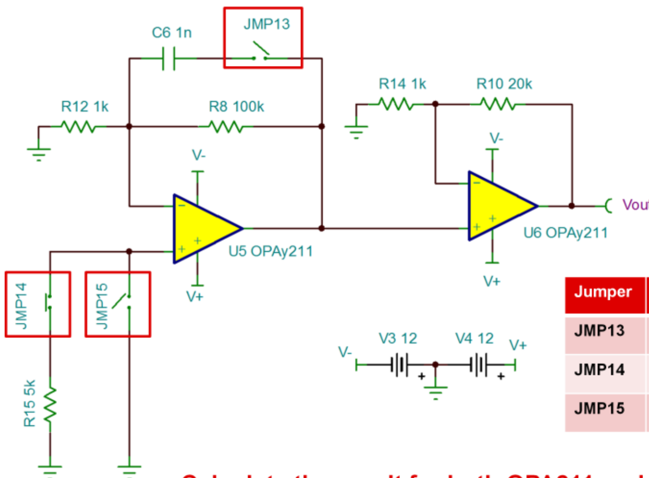
34

TEXAS INSTRUMENTS

For the next part of the lab, we'll repeat the same procedure as experiment 1, but this time with 5kΩ of input resistance. This will emphasize the effects of input bias current, IB.

Calculation – R_{IN} = 5kΩ

Calculate the total output voltage due to V_{OS} and I_B for the circuit shown below. Use the typical and maximum values for V_{OS} and I_B given on the next slide.



Jumper	Position	Description
JMP13	OPEN	C5 disconnected
JMP14	CLOSED	R15 connected
JMP15	OPEN	VIN+ connected to R15

Calculate the result for both OPA211 and OPA188!

35

Texas Instruments

As shown in the schematic, jumper JMP15 shorting the positive input of U5 to ground is now removed. Jumper JMP14 is now installed in order to connect the positive input of U5 to a 5kΩ resistor. The IB of U5 will now flow through this resistor, developing a DC voltage due to Ohm's law and increasing the amount of offset voltage.

As before, calculate the total output voltage due to VOS and IB for this circuit, using both the OPA211 and OPA188.

Calculation – R_{IN} = 5kΩ

PARAMETER		OPA211			
		MIN	TYP	MAX	UNIT
Input Offset Voltage	V_{OS}		±30	±125	μV
Input Bias Current	I_B		±60	±175	nA

PARAMETER		OPA188			
		MIN	TYP	MAX	UNIT
Input Offset Voltage	V_{OS}		±6	±25	μV
Input Bias Current	I_B		±160	±1400	pA

Device	Typical Output	Maximum Output
OPA188	±15mV	±71mV
OPA211	±191mV	±2492mV

36

Texas Instruments

The data sheet parameters for both devices are provided again for reference. Enter your calculated results in the lower table. The answers have been provided so that you can check your work.

Calculation – R_{IN} = 5kΩ

OPA211

Calculate typical output error from Vos & Ib

$i_b = \pm 60\text{nA}$

$R_{eq} = \dfrac{R_f R_1}{R_f + R_1} \approx 1\text{k}\Omega$

$R_{in} = 5\text{k}\Omega$

$V_{ib} = i_b\,R_{eq} + i_b\,R_{in} = \pm 360\mu\text{V}$

$V_{os} = \pm 30\mu\text{V}$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 30\mu\text{V} + \pm 360\mu\text{V})$

$V_{out\_error} = \pm 827\text{mV or } \pm 700\text{mV}$

$V_{out\_error} = \pm 827\text{mV}$ Choose the largest

Calculate maximum output error from Vos & Ib

$i_b = \pm 175\text{nA}$

$R_{eq} = \dfrac{R_f R_1}{R_f + R_1} \approx 1\text{k}\Omega$

$R_{in} = 5\text{k}\Omega$

$V_{ib} = i_b\,R_{eq} + i_b\,R_{in} = \pm 1050\mu\text{V}$

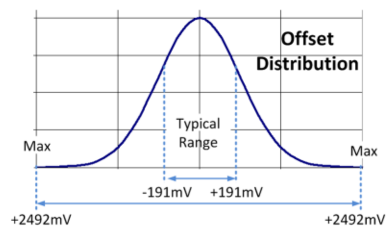
$V_{os} = \pm 125\mu\text{V}$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 125\mu\text{V} + \pm 1050\mu\text{V})$

$V_{out\_error} = \pm 2492\text{mV or } \pm 1962\text{mV}$

$V_{out\_error} = \pm 2492\text{mV}$ Choose the largest



37

TEXAS INSTRUMENTS

With RIN = 5k, the calculations change slightly since the voltage caused by IB is now affected by R_IN. Use the new equation VIB = IB*R_EQ + IB*R_IN. Otherwise, the steps are the same as in experiment 1. Repeat the calculations for the maximum values.

Calculation – R_{IN} = 5kΩ

OPA188

Calculate typical output error from Vos & Ib

$i_b = \pm 160pA$

$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1k\Omega$

$R_{in} = 5k\Omega$

$V_{ib} = i_b R_{eq} + i_b R_{in} = \pm 0.96\mu V$

$V_{os} = \pm 6\mu V$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 6\mu V + \pm 0.96\mu V)$

$V_{out\_error} = \pm 15mV \text{ or } \pm 11mV$

$V_{out\_error} = \pm 15mV$ Choose the largest

Calculate maximum output error from Vos & Ib

$i_b = \pm 1400pA$

$R_{eq} = \frac{R_f R_1}{R_f + R_1} \approx 1k\Omega$

$R_{in} = 5k\Omega$

$V_{ib} = i_b R_{eq} + i_b R_{in} = \pm 8.4\mu V$

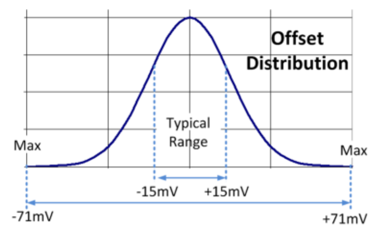
$V_{os} = \pm 25\mu V$

$V_{out\_error} = G_1 G_2 (V_{os} + V_{ib})$

$V_{out\_error} = 101 * 21(\pm 25\mu V + \pm 8.4\mu V)$

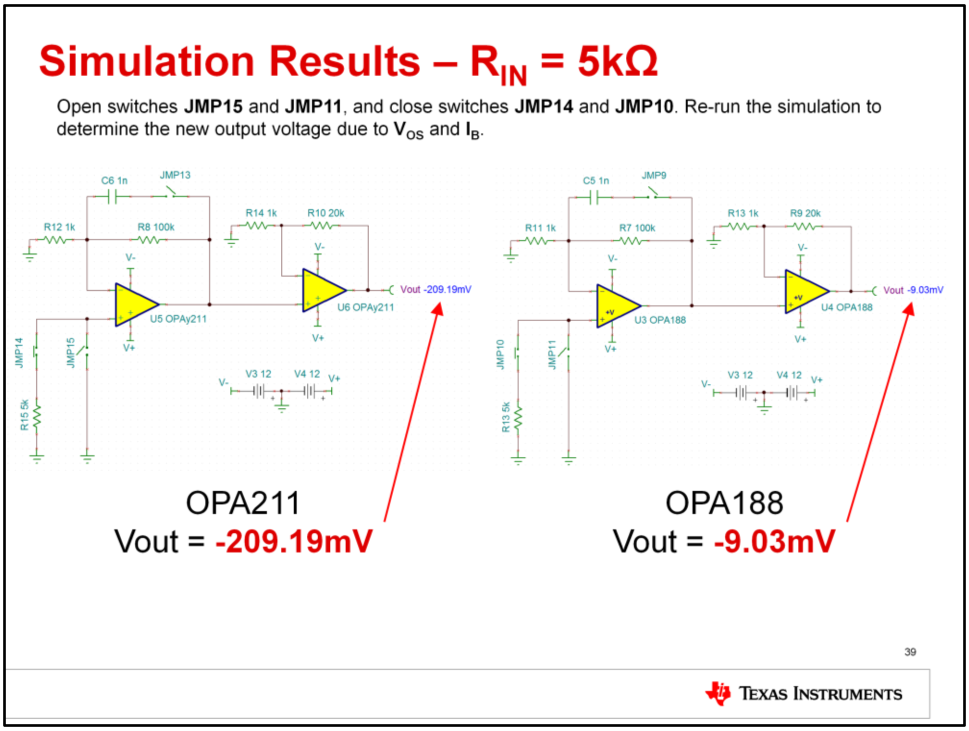
$V_{out\_error} = \pm 71mV \text{ or } \pm 35mV$

$V_{out\_error} = \pm 71mV$ Choose the largest



Texas Instruments

Repeat both sets of calculations for the OPA188, again using typical and maximum values. As before, the different electrical characteristics of the OPA188 will result in different output voltage calculations.

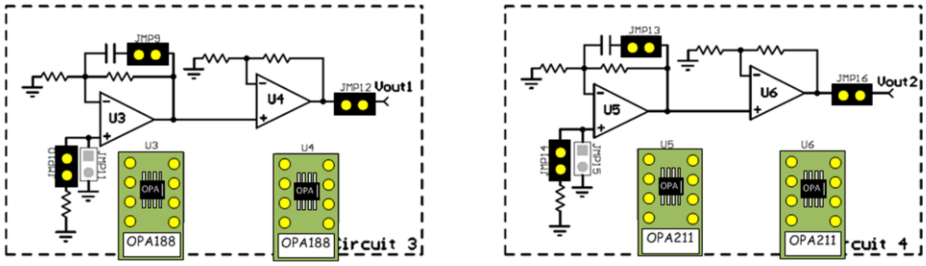


Re-run the simulated DC nodal voltage analysis, making sure to use the proper jumper settings.

In the OPA211 circuit, JMP13 and JMP15 are open, and JMP14 is closed.
In the OPA188 circuit, JMP9 and JMP11 are open, and JMP10 is closed.

Test Board Setup – Jumpers

Remove jumpers **JMP11** and **JMP15**, and install jumpers **JMP10** and **JMP14**.



The only change →

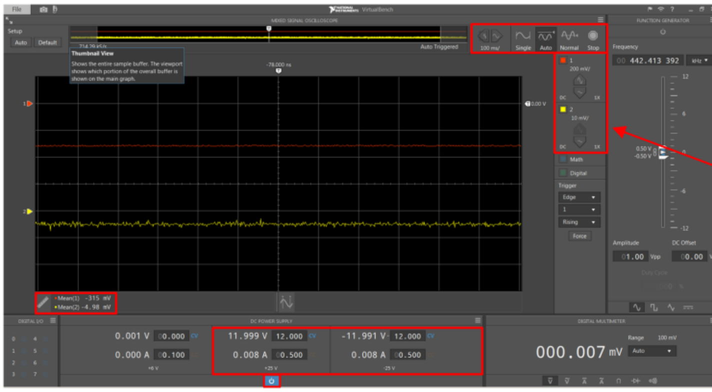
Jumper, Device	Description
JMP9, JMP13	Filter noise with feedback capacitor
JMP10, JMP14	**Connect VIN+ to 5kΩ**
JMP12, JMP13	Connect Circuit 3 to Vout1, connect Circuit 4 to Vout2
U3, U4	Install OPA188
U5, U6	Install OPA211

40

TEXAS INSTRUMENTS

The jumper settings on the test board must be modified before re-running the bench measurement. Remove jumpers JMP11 and JMP15, and install jumpers JMP10 and JMP14. All other jumpers and devices remain the same from the previous experiment.

VirtualBench™ Instrument Setup



May need to adjust range
10mV/div to 1V/div

41

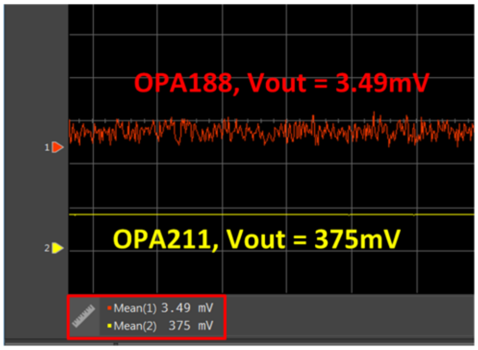
Texas Instruments

Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

Set the time scale to 100ms per division, with the acquisition mode set to “Auto.” Enable channels 1 and 2 on the oscilloscope, and set them to 1x, DC-coupled mode. Adjust the vertical scale as needed from 10mV/div to 1V/div. Set the +25V power supply to +12V, 0.500A. Set the -25V power supply to -12V, 0.500A. Press the power button to turn on the power supply rails.

Enable “mean” measurements on both channels in order to read the output voltage of each circuit.

Measurement Results – R_{IN} = 5kΩ



Measured result should be less than calculated maximum

Device	Calculated Maximum	Calculated Typical	Simulated Typical	Measured
OPA188	±71mV	±15mV	-9.03mV	3.49mV
OPA211	±2492mV	±827mV	-209.19mV	375mV

In experiment 2, the OPA188 has a measured output voltage of 3.49mV, and the OPA211 has a measured output voltage of 375mV. You may have different results in your experiment.

How did the measured and simulated results compare to the hand calculated results? In this example, the OPA211 output was greater than the calculated and simulated typical values, but less than the calculated maximum value. The OPA188 output was less than the calculated and simulated typical values.

Final Results

How did the change in input resistance affect the output voltage measurement?

Device Rin = 0Ω	Calculated Maximum	Calculated Typical	Simulated Typical	Measured
OPA188	±56mV	±13mV	-10.73mV	4.60mV
OPA211	±635mV	±191mV	133.28mV	74.7mV

Device Rin = 5kΩ	Calculated Maximum	Calculated Typical	Simulated Typical	Measured With Rin
OPA188	±71mV	±15mV	-9.03mV	3.94mV
OPA211	±2492mV	±827mV	-209.19mV	375mV

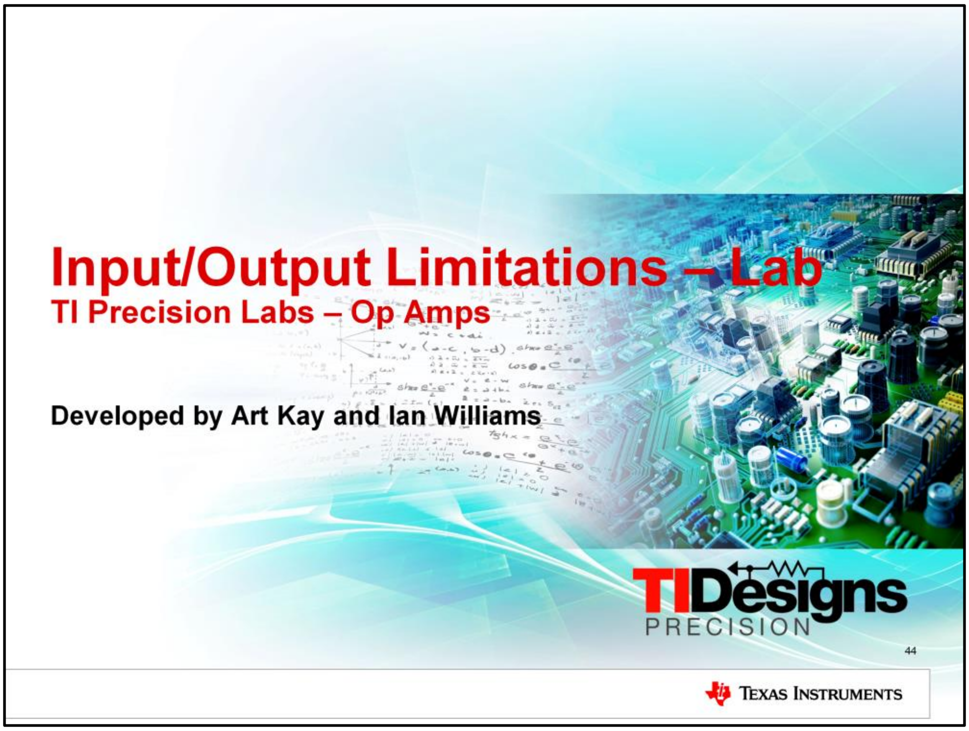
Answer:	Dramatic increase for the OPA211, small change for OPA188. This is due to the OPA211 having much larger I_B than the OPA188.

43

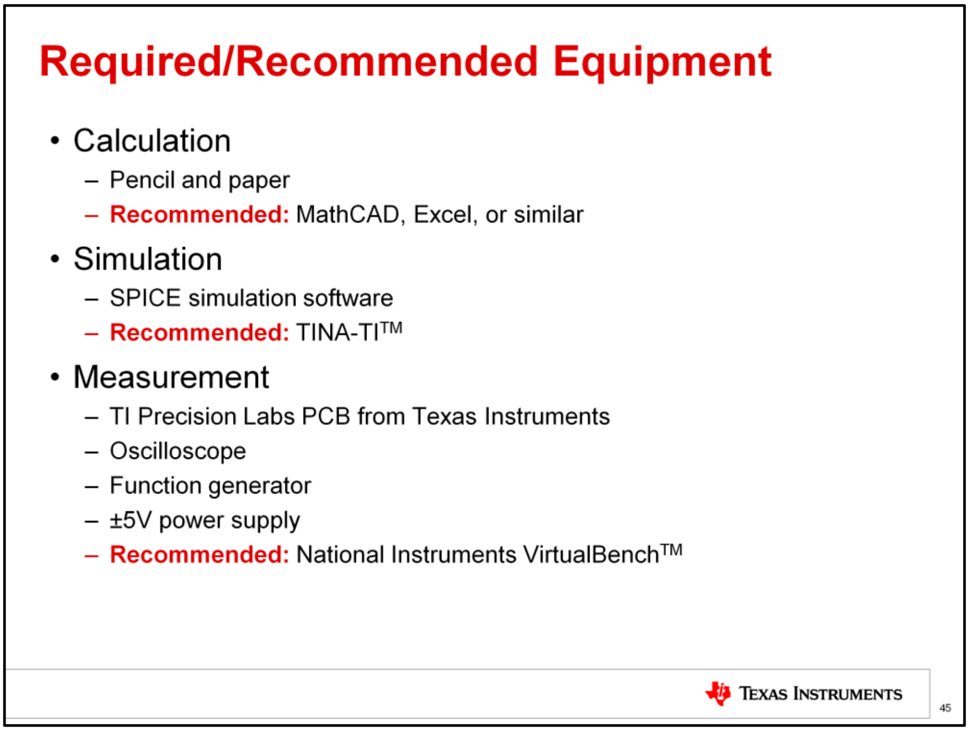
TEXAS INSTRUMENTS

Let’s now compare the results of both experiments. How did the change in input resistance affect the output voltage measurement?

In the OPA211, increasing the input resistance caused a dramatic increase in output voltage. However, the OPA188 did not see such a large increase. This is because the OPA211 has a much larger input bias current (IB), than the OPA188.



Hello, and welcome to the presentation for the TI Precision Lab supplement for op amp input and output limitations. This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the op amp input and output limitations lecture.



The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly.

The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

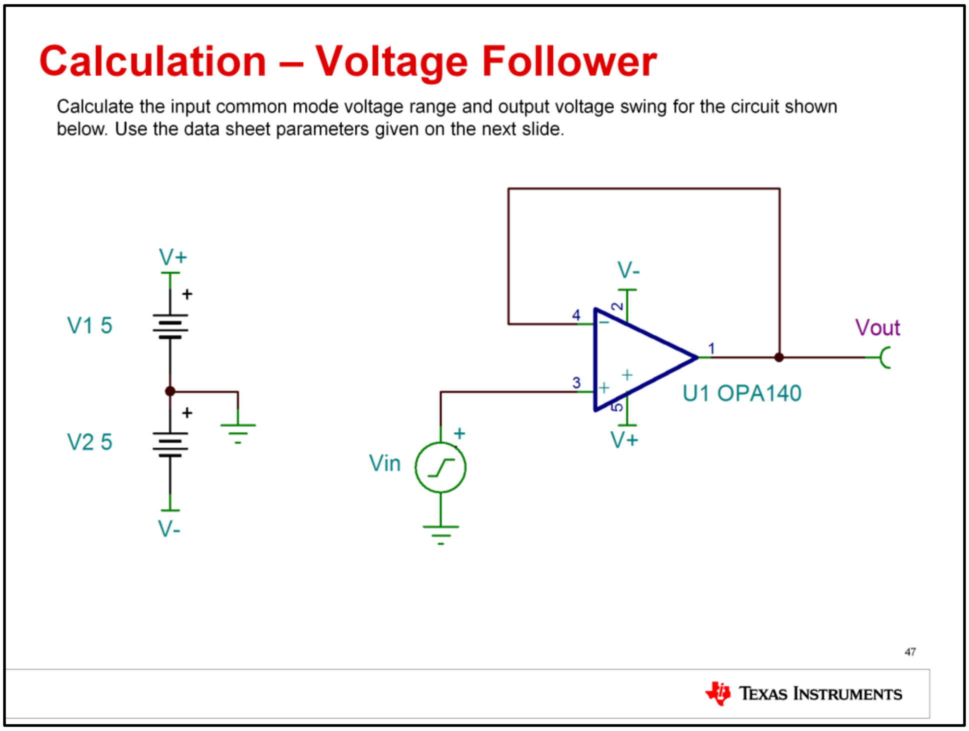
Finally, the real-world measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any oscilloscope, function generator, and ±5V power supply. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.

Experiment 1
Voltage Follower

46

TEXAS INSTRUMENTS

In experiment 1, we'll determine the effects of input and output limitations in a basic voltage follower, or unity-gain buffer, circuit.

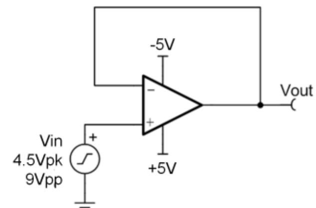


First, calculate the input common mode voltage range and output voltage swing for the circuit shown here, using the techniques and equations given in the input/output limitations lecture. Use the data sheet parameters given on the next slide.

Calculation – Voltage Follower

PARAMETER		OPA140			
		MIN	TYP	MAX	UNIT
Input Voltage Range	V_{CM}	(V-) – 0.1		(V+) – 3.5	V
Voltage Output Swing from Rail	V_O	(V-) + 0.2		(V+) – 0.2	V

Calculated Answers	Min	Max
Common Mode Input Range	-5.1	+1.5V
Output Swing Range	-4.8	4.8V



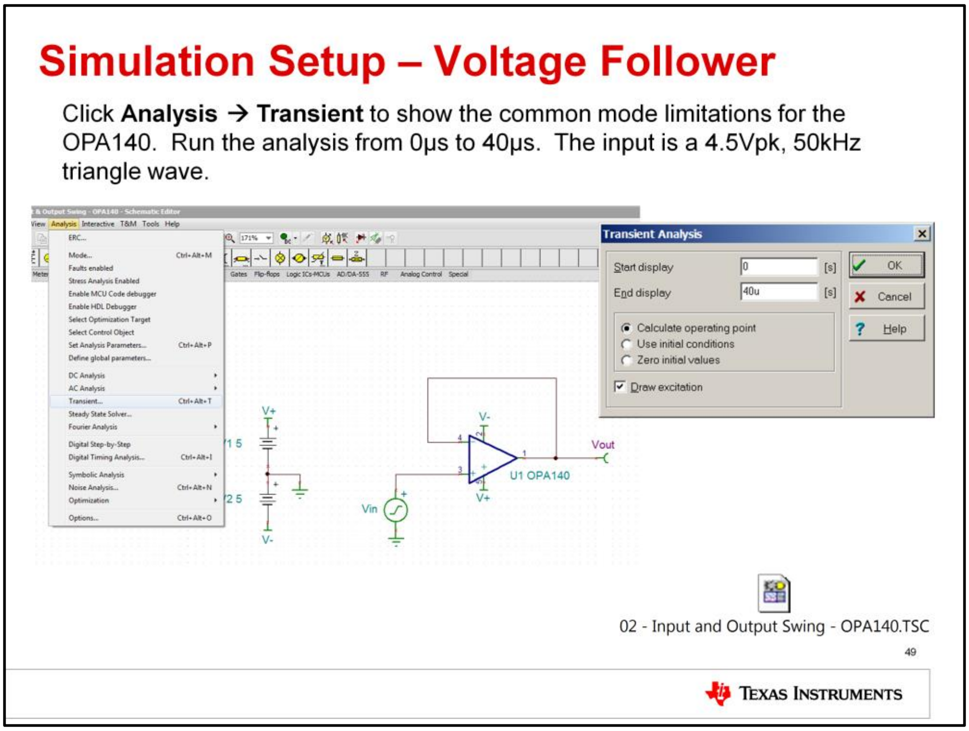
With a 9Vpp (4.5Vpk) input signal, is the circuit limitation caused by input common mode range or output voltage swing?

Answer:	The maximum input common mode voltage of +1.5V is the limit, since the peak input signal is +4.5V.

This circuit uses the OPA140. In order to perform the calculations, you need to know the input voltage range and voltage output swing values for that device. Those values are given here.

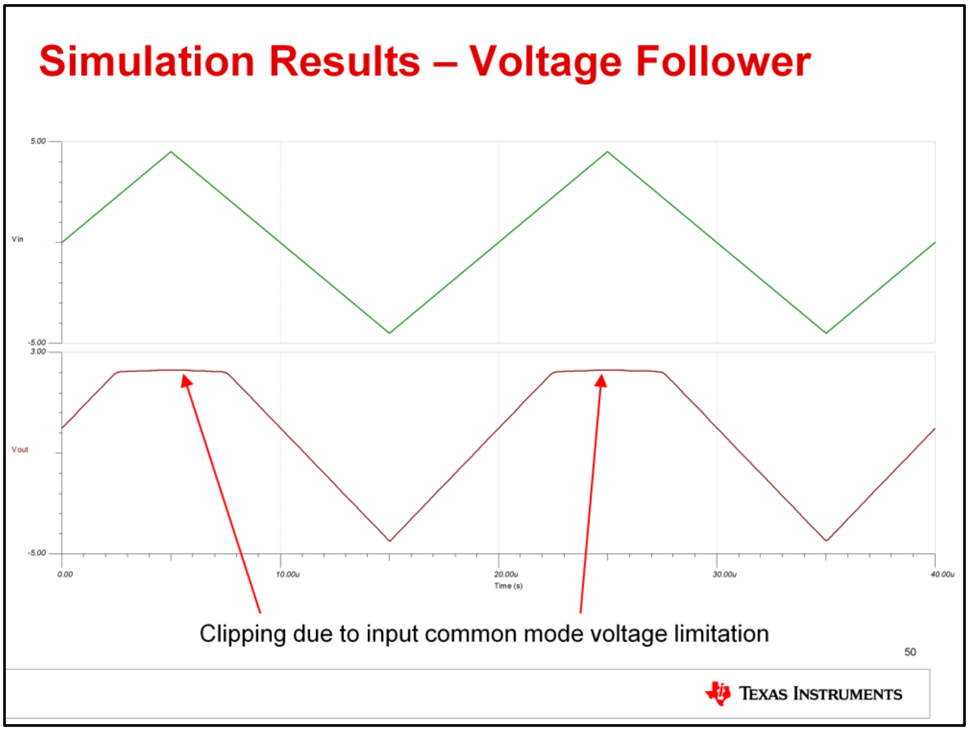
Enter your answers in the table in the middle of the slide. The solutions are already provided to allow you to check your work.

Also answer the question at the bottom of the slide. Again, the solution is already provided.

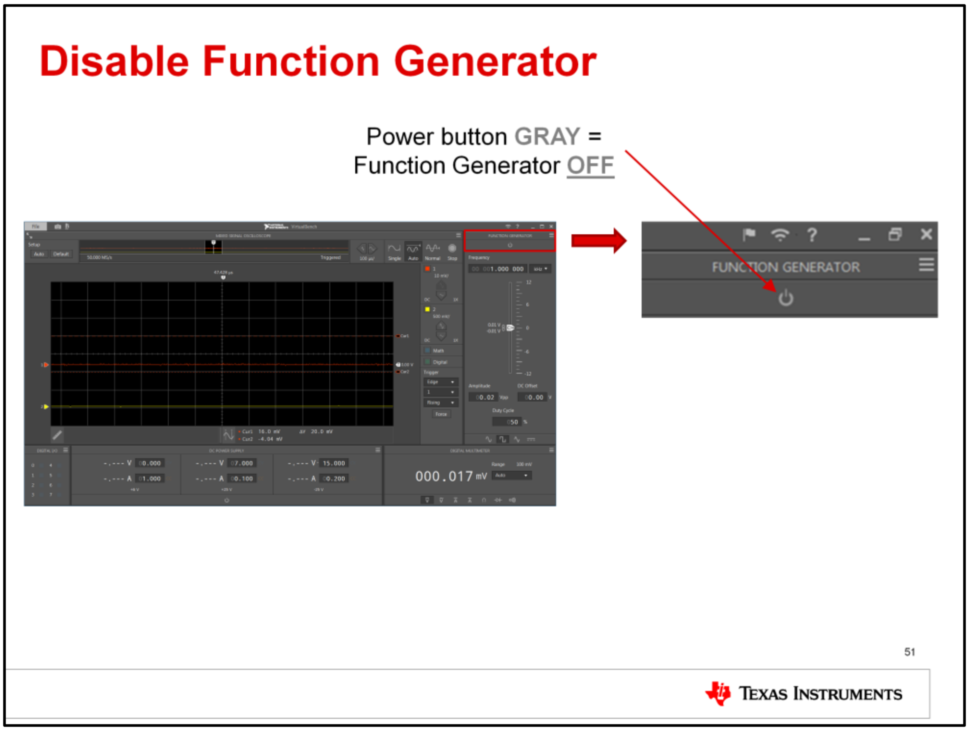


The next step is to run a SPICE simulation analysis for the transient output voltage behavior. This will allow us to see the op amp's output voltage response for a specified input signal, which in this case is a 2Vp, 1kHz triangle wave.

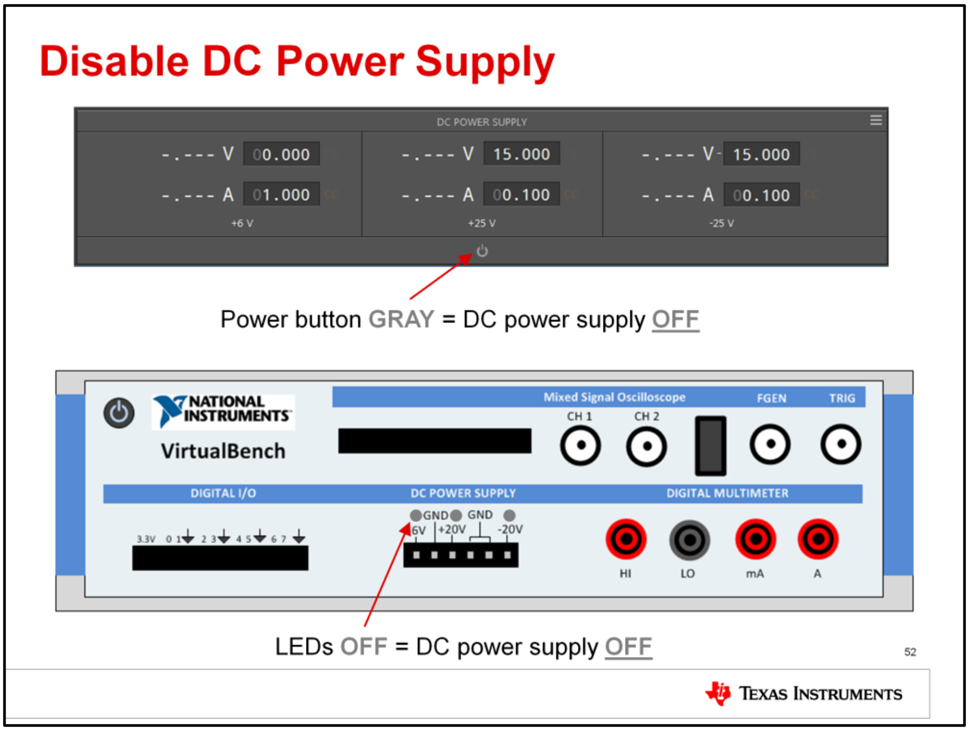
The necessary TINA-TI simulation schematic is embedded in this slide set – simply double-click the icon to open it. To simulate the transient output behavior, click Analysis → Transient. Run the analysis from 0ms to 2ms.



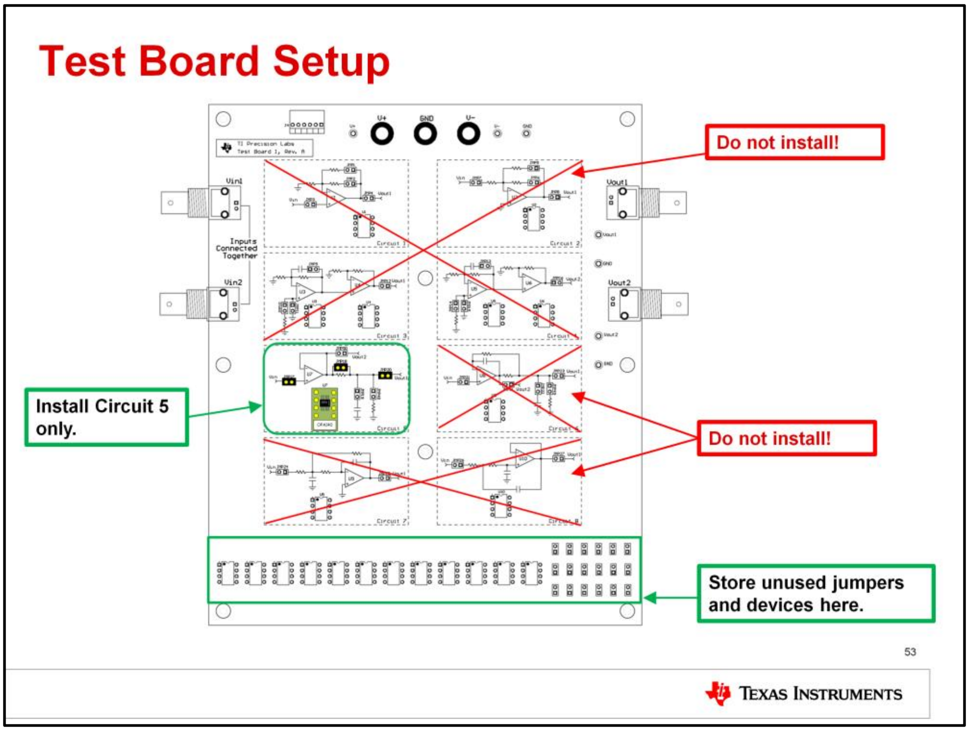
You should see a result similar to this. Vin is a triangle wave, as expected, but Vout cannot exceed +1V. This is due to the input common mode voltage limitation of the OPA735.



Also make sure the Function Generator is OFF.

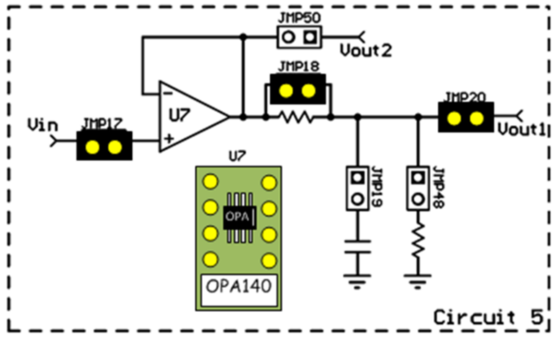


Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuit 5. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

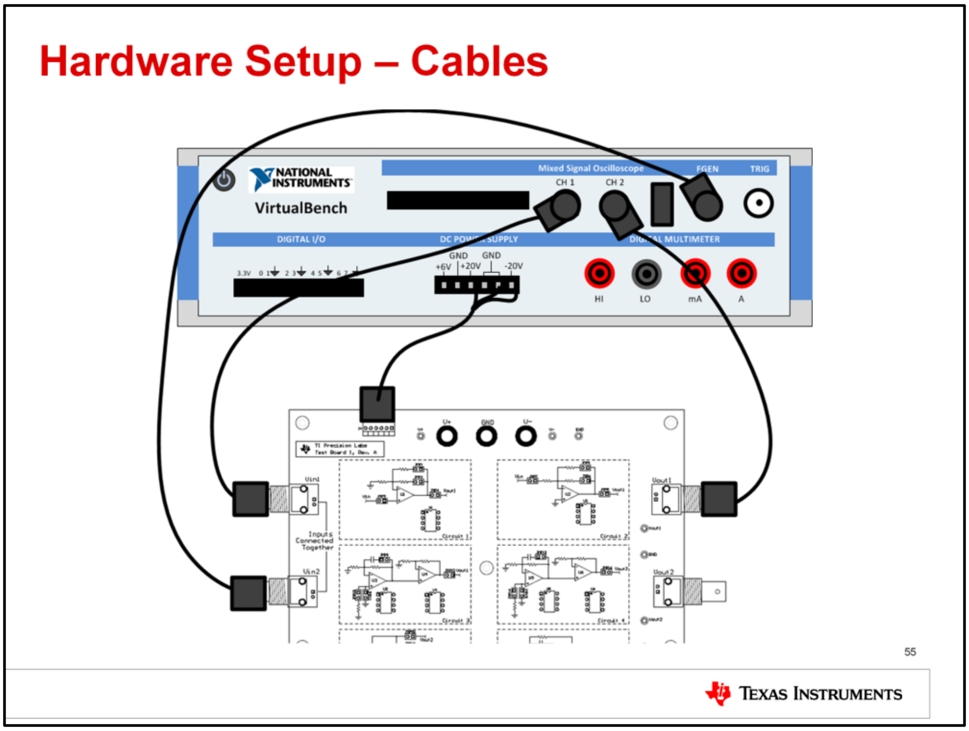
Test Board Setup – Jumpers



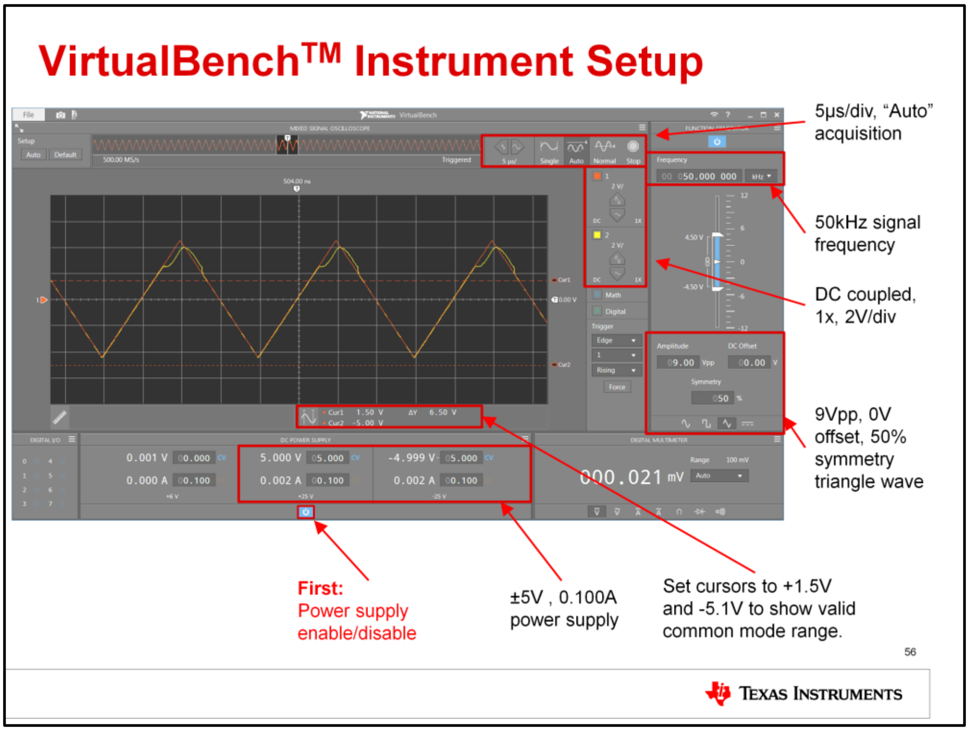
Jumper, Device	Description
JMP17	Connect input to signal source.
JMP18	Short output resistance
JMP20	Connect Circuit 5 output to Vout1
U7	Install OPA140

To prepare the test board for the measurement, install the jumpers and devices on circuit 5 as shown here.

Install JMP17, JMP18, and JMP20, as well as the OPA735 in socket U7.



This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board. Connect Vin2 on the test board to VirtualBench channel FGEN, or function generator. Then connect Vin1 on the test board to VirtualBench oscilloscope channel 1, and Vout1 on the test board to VirtualBench oscilloscope channel 2.



Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

Set the time scale to 5us per division, with the acquisition mode set to "Auto." Enable channels 1 and 2 on the oscilloscope, and set them to 1x, DC-coupled mode, 2V/div. Enable the function generator and setup the signal as follows:

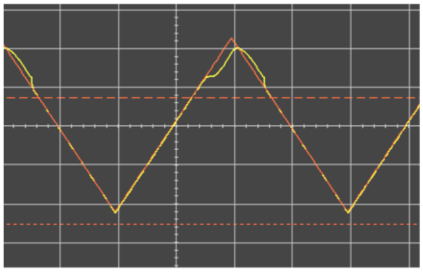
50kHz frequency, 9Vpp, 0V offset, 50% symmetry triangle wave.

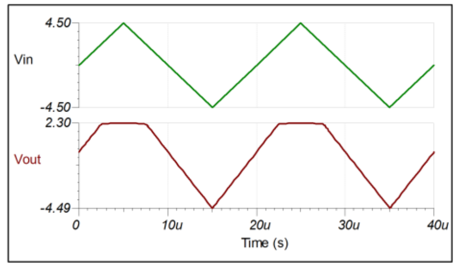
Enable the cursors and set them to +1.5V and -5V to show the valid input common mode range.

Set the +25V power supply to +5V, 0.500A. Set the -25V power supply to -5V, 0.500A. Press the power button to turn on the power supply rails.

Measurement Results – Voltage Follower

1. Compare TINA-TI™ simulation results to measured results.





2. Use the cursors on the VirtualBench and TINA-TI™ tool to measure the voltage where Vout becomes limited (clipped). Compare this to your calculation.

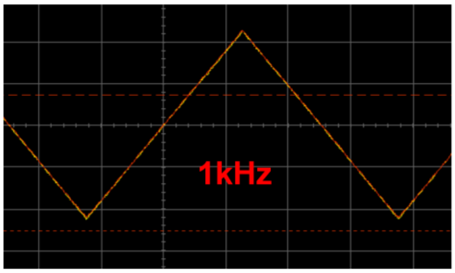
Answer (calculation):	clipped at +1.5V
Answer (simulation):	clipped at +2.3V
Answer (measurement):	visible distortion above +2.5V

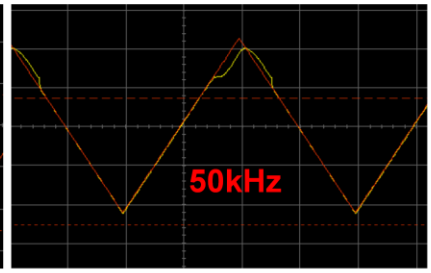
The expected measurement results are shown here. Compare the oscilloscope display of the VirtualBench to the simulation results from TINA-TI. Also, use the cursors on the VirtualBench and TINA-TI tool to measure the voltage where Vout becomes limited, or clipped. Compare this to your calculation.

The results have already been entered into the table to allow you to check your results. You may have different results in your experiment.

Extra Experiment: 1kHz vs. 50kHz Input

3. Change the triangle waveform frequency to 1kHz and compare the common mode range to the 50kHz wave. Change the time scale to 200µs/div.

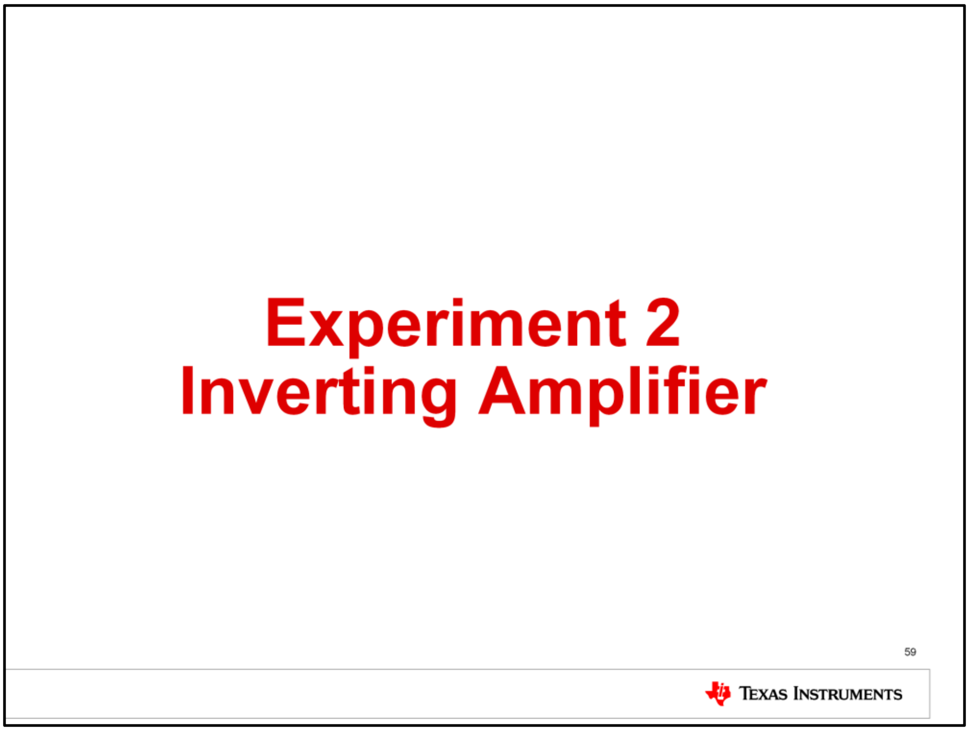




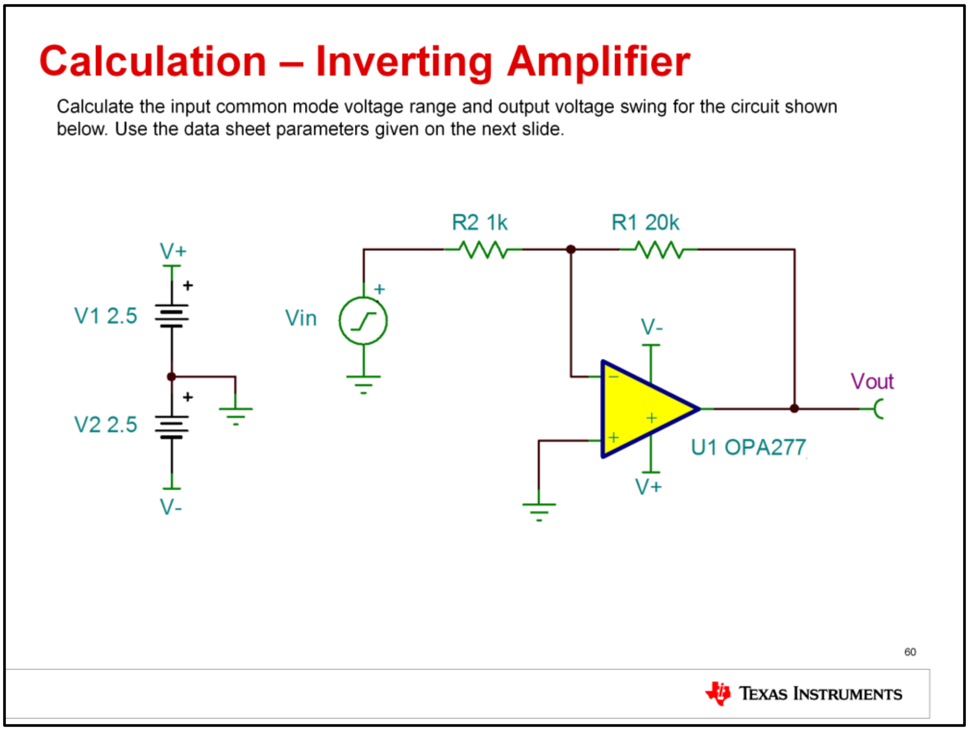
4. What conclusion do you draw from the measurement?

The 1kHz waveform does not show the common mode limitation, but the 50kHz waveform does. This device is actually “rail-to-rail” for low frequencies, but common mode performance ~2V from the positive rail degrades at higher frequencies.

Texas Instruments



In experiment 2, we'll determine the effects of input and output limitations in an inverting amplifier circuit with gain.

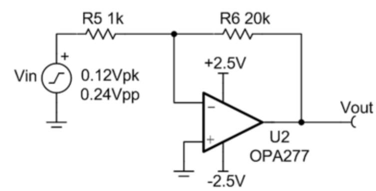


First, calculate the input common mode voltage range and output voltage swing for this inverting amplifier circuit, using the techniques and equations given in the input/output limitations lecture. Use the data sheet parameters given on the next slide.

Calculation – Inverting Amplifier

PARAMETER (data sheet)		CONDITION	OPA277			
			MIN	TYP	MAX	UNIT
Input Voltage Range	V_{CM}		(V-) +2		(V+) –2	V
Voltage Output	Vo	RL = 10kΩ	(V-) +0.5		(V-) –1.2	V
Voltage Output		RL = 2kΩ	(V-) +1.5		(V-) –1.5	

Answers	Min	Max
Common Mode Input Range	-0.5V	0.5V
Output Swing Range	-2.0V	1.3V



With a 0.24Vpp (0.12Vpk) input signal, is the circuit limitation caused by input common mode range or output voltage swing?

Answer:	The output wants to be ± 2.4Vpk. This violates both the negative and positive output swing limit. There's no issue with the input common mode range.

61

Texas Instruments

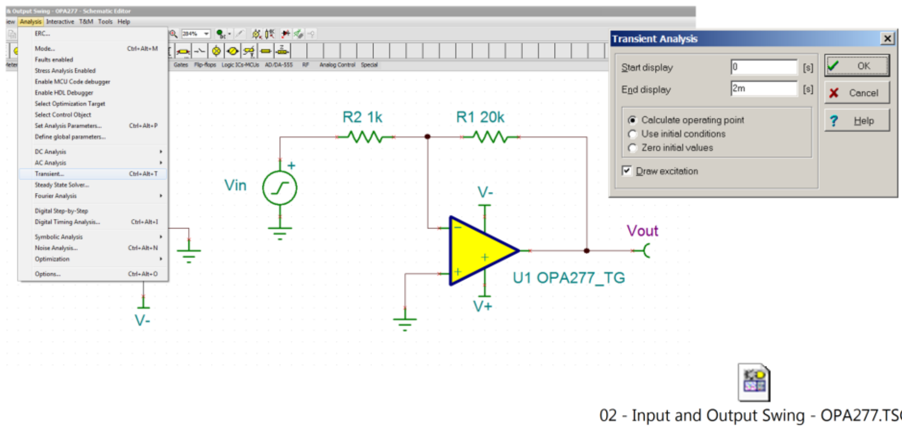
This circuit uses the OPA277. In order to perform the calculations, you need to know the input voltage range and voltage output swing values for that device. Those values are given here.

Enter your answers in the table in the middle of the slide. The solutions are already provided to allow you to check your work.

Also answer the question at the bottom of the slide. Again, the solution is already provided.

Simulation Setup – Inverting Amplifier

Click **Analysis → Transient** to show the common mode limitations for the OPA277. Run the analysis from 0ms to 2ms. The input is a 120mVpk, 1kHz triangle wave.

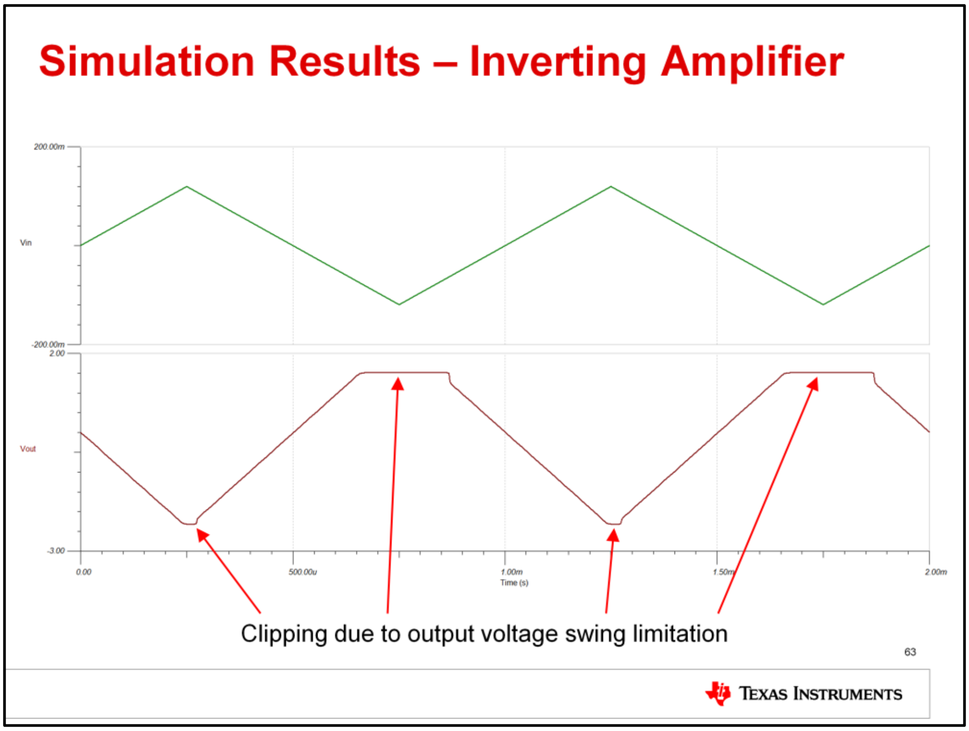


62

TEXAS INSTRUMENTS

Run a SPICE simulation as before, but now using this inverting amplifier circuit with the OPA277.

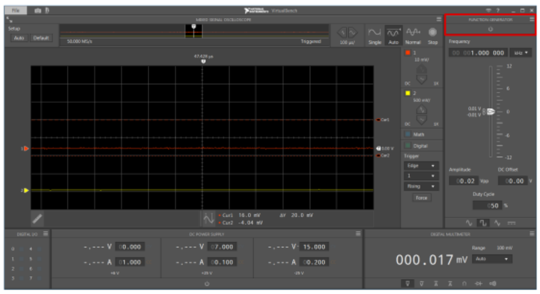
The necessary TINA-TI simulation schematics are embedded in this slide set – simply double-click the icon to open them. Click Analysis → Transient and run the transient from 0ms to 2ms. The input signal to this circuit is a 120mVp, 1kHz triangle wave.

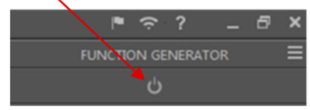


You should see a result similar to this. Vin is a triangle wave, as expected, but Vout clips at both the positive and negative ends of the triangle wave due to output voltage swing limitations.

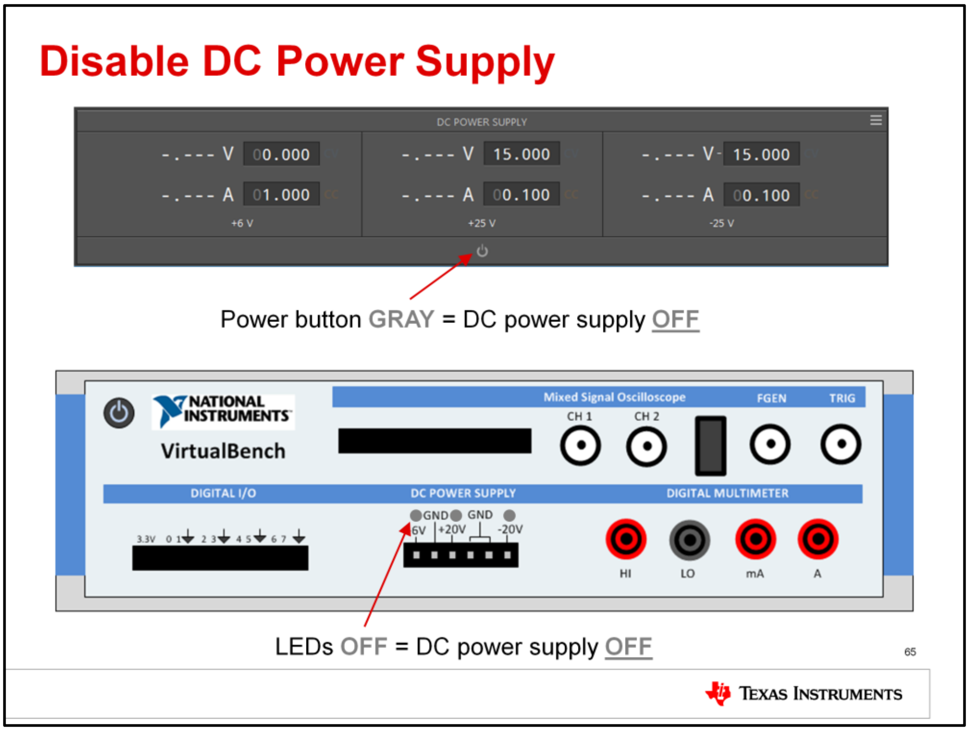
Disable Function Generator

Power button GRAY =
Function Generator OFF

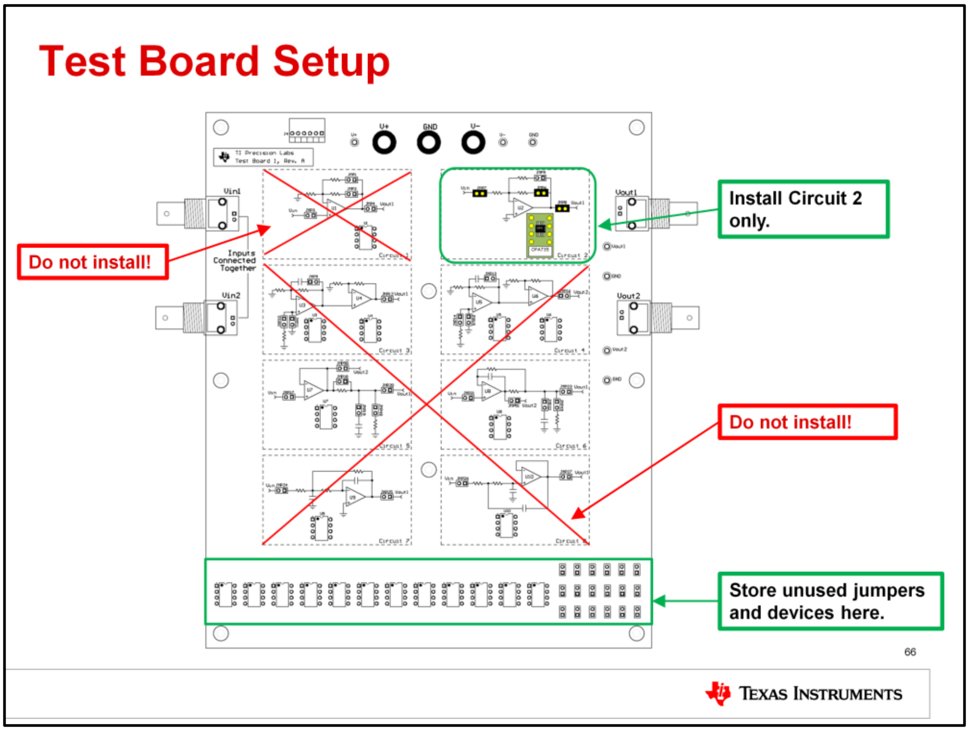




Also make sure the Function Generator is OFF.

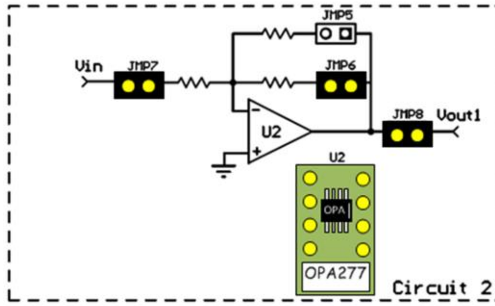


Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For this measurement, only circuit 2 is used. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

Test Board Setup – Jumpers



Jumper, Device	Description
JMP6	Connect Rf = 20kΩ for a gain of 20V/V
JMP7	Connect input to signal source
JMP8	Connect Circuit 2 output to Vout1
U2	Install OPA277

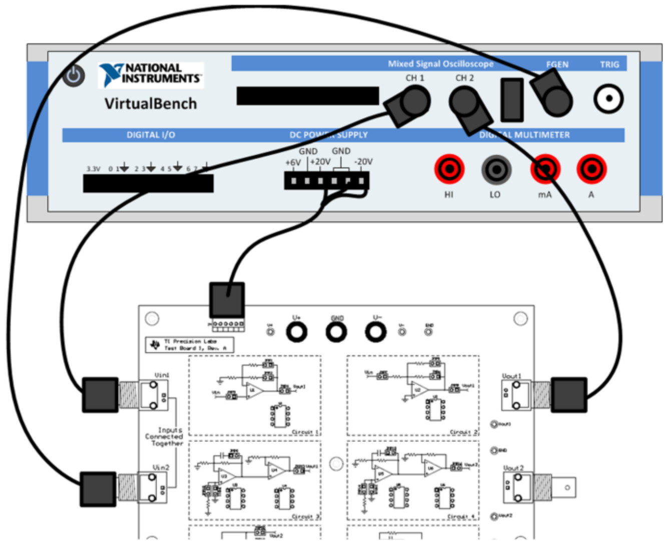
67

TEXAS INSTRUMENTS

To prepare the test board for the measurement, install the jumpers and devices on circuit 2 as shown here.

Install JMP6, JMP7, and JMP8, as well as the OPA277 in socket U2.

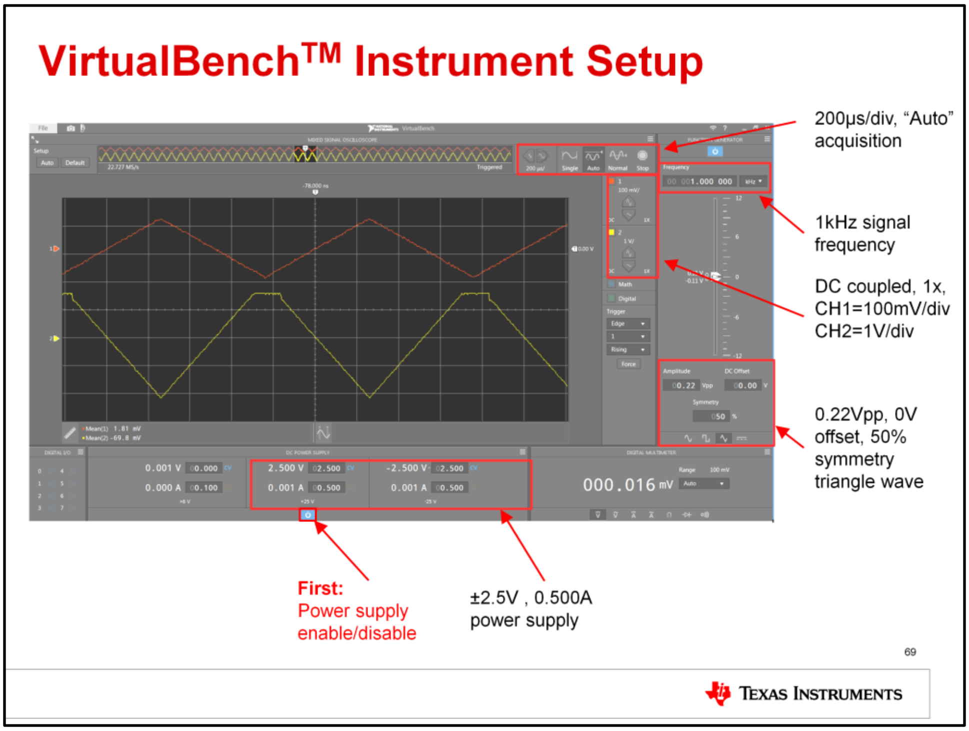
Hardware Setup – Cables



68

TEXAS INSTRUMENTS

The cable connections to the VirtualBench are exactly the same as in experiment 1. No changes are necessary.



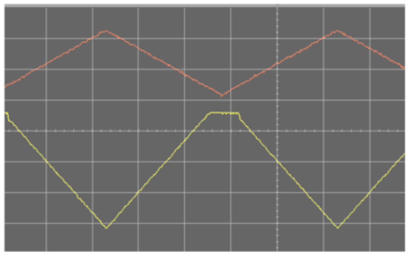
The VirtualBench instrument setup is very similar to experiment 1. Make the following changes:

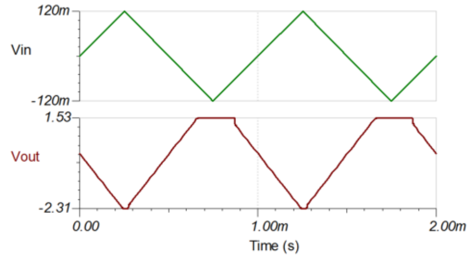
Set the vertical scale of CH1 to 100mV/div. Keep the vertical scale of CH2 at 1V/div.

Set the function generator to a 0.22Vpp, 0V offset, 50% symmetry triangle wave at 1kHz.

Measurement Results – Inverting Amplifier

1. Compare TINA-TI™ simulation results to measured results.

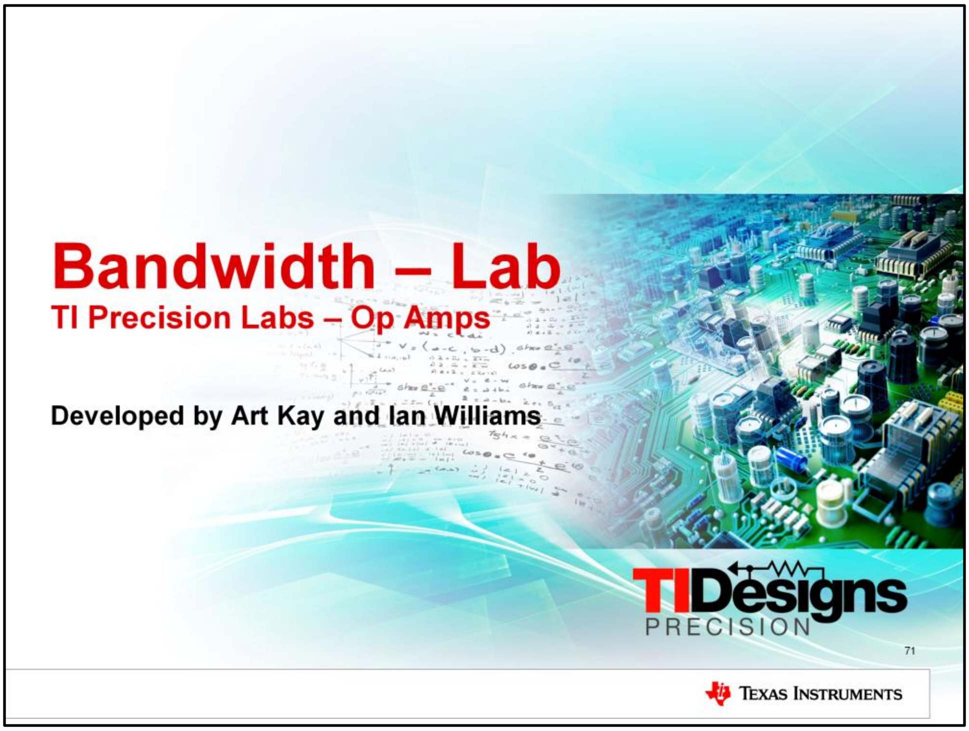




2. Use the cursors on the VirtualBench and TINA-TI™ tool to measure the voltage where Vout becomes limited (clipped). Compare this to your calculation.

Answer (calculation):	clipped at -2.0V and +1.3V
Answer (simulation):	clipped at -2.31V and +1.53V (it may distort before the clip)
Answer (measurement):	clipped at -2.40V and +1.59V (it may distort before the clip)

Compare the TINA simulation results to your measured results. The shape of the output waveform should look very similar, with hard clipping at the top of the waveform. Your device may or may not clip at the bottom of the waveform.



Hello, and welcome to the presentation for the TI Precision Lab supplement for op amp bandwidth. This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the op amp input and output limitations presentation.

Required/Recommended Equipment

- Calculation
  - Pencil and paper
  - **Recommended:** MathCAD, Excel, or similar
- Simulation
  - SPICE simulation software
  - **Recommended:** TINA-TI™
- Measurement
  - TI Precision Labs PCB from Texas Instruments
  - Bode plotter
  - ±15V power supply
  - **Recommended:** National Instruments VirtualBench™

Texas Instruments 72

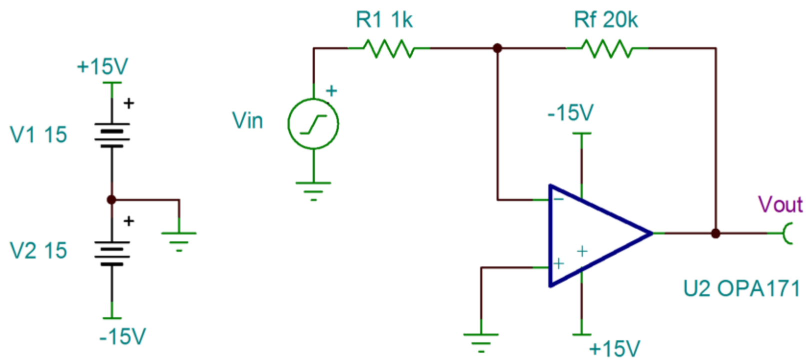
The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly.

The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

Finally, the real-world measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any Bode plotter and ±15V power supply. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.

Calculation – Bandwidth

Calculate the closed loop bandwidth and DC signal gain for the circuit shown below. Use the data sheet parameters given on the next slide.



73

TEXAS INSTRUMENTS

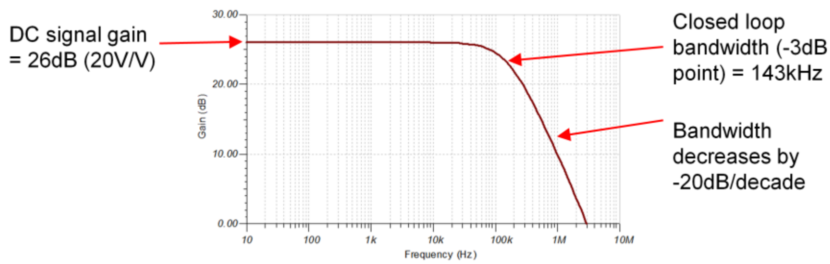
First, calculate the closed loop bandwidth and DC signal gain for the circuit shown here, using the techniques and equations given in the bandwidth lecture. Use the data sheet parameters given on the next slide.

Calculation – Bandwidth

1. Use excerpt from data sheet below to fill in the table and AC transfer function:

PARAMETER		OPA171			
		MIN	TYP	MAX	UNIT
Gain bandwidth product	GBP		3.0		MHz
Slew rate	SR		1.5		V/μs

Closed Loop Bandwidth:	GBP/G_n = 3MHz/21= 143kHz.
Signal Gain at DC	G_{CL} = -(Rf/R1) = -(20k/1k) = -20V/V



74

TEXAS INSTRUMENTS

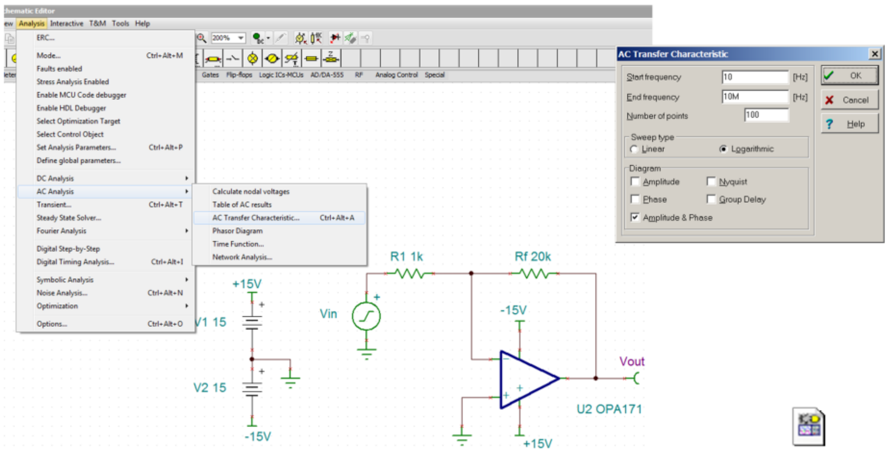
This circuit uses the OPA171. In order to perform the calculations, you need to know the gain bandwidth product and for that device. That value is given here.

Enter your answers in the table in the middle of the slide. The solutions are already provided to allow you to check your work.

Also complete the AC transfer function for this circuit. Again, the solution is already provided.

Simulation Setup – Bandwidth

Click **Analysis → AC Analysis →AC Transfer Characteristic** to show the AC response for the OPA171 circuit. Run the analysis from 10Hz to 10MHz.



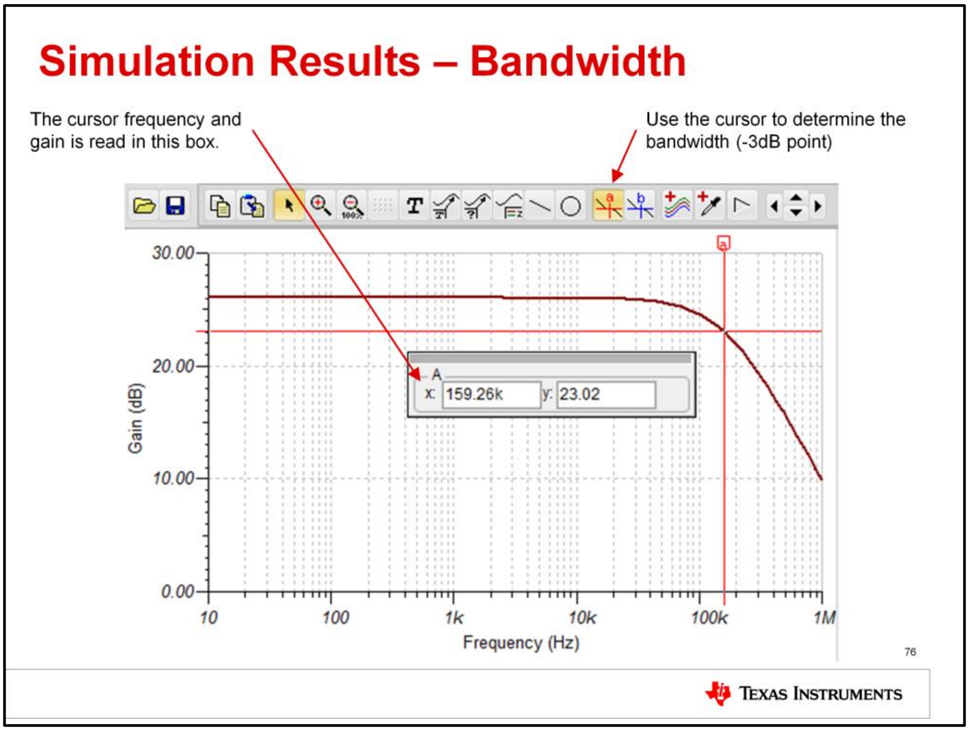
03 - Bandwidth - OPA171.TSC

75

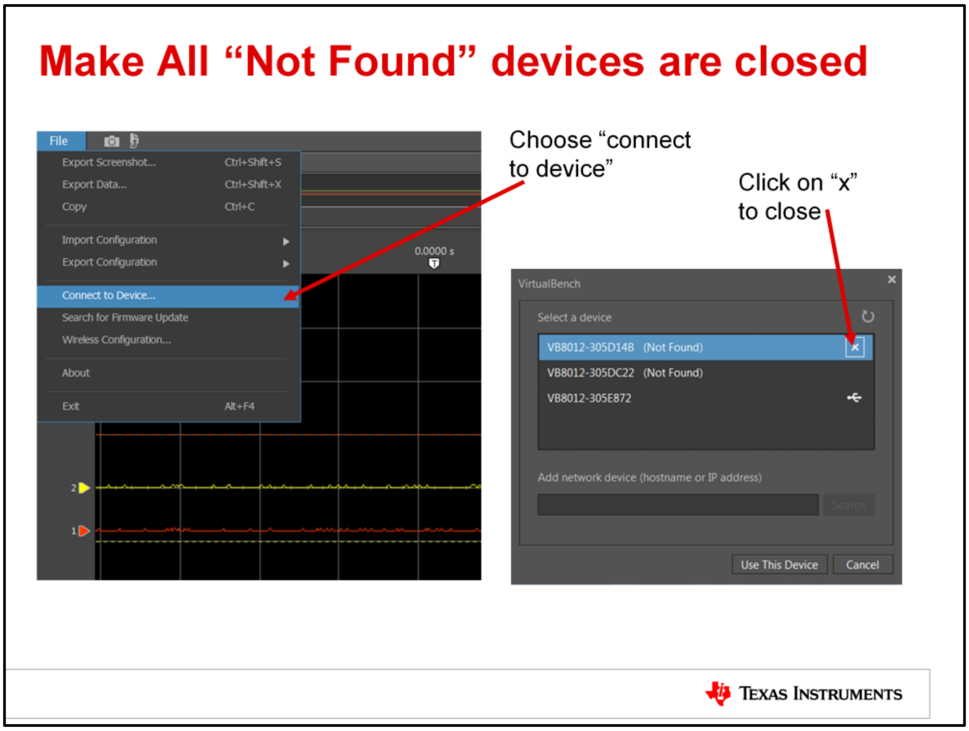
Texas Instruments

The next step is to run a SPICE simulation analysis for the AC transfer characteristic. This will allow us to see the op amp's closed loop bandwidth and DC gain in this configuration.

The necessary TINA-TI simulation schematic is embedded in this slide set – simply double-click the icon to open it. To run the analysis, click Analysis → AC Analysis → AC Transfer Characteristic. Run the analysis from 10Hz to 10MHz.



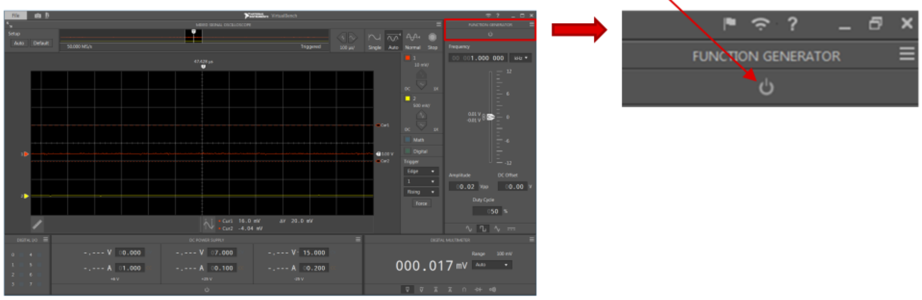
You should see a result similar to this. Enable the cursor, then check the DC gain, or gain at minimum frequency. The result will be 26dB, or 20V/V. Next, find the -3dB point, or the frequency where gain drops to 23dB. This occurs at 159kHz.



We will be using a new software package. To prevent problems, make sure that all “not found” devices are closed.

Disable Function Generator

Power button GRAY =
Function Generator OFF

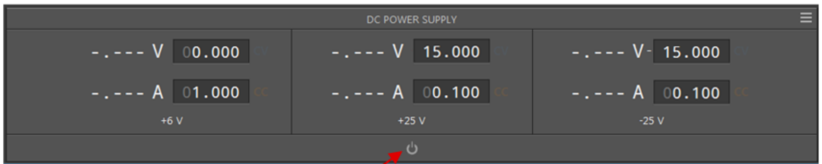


78

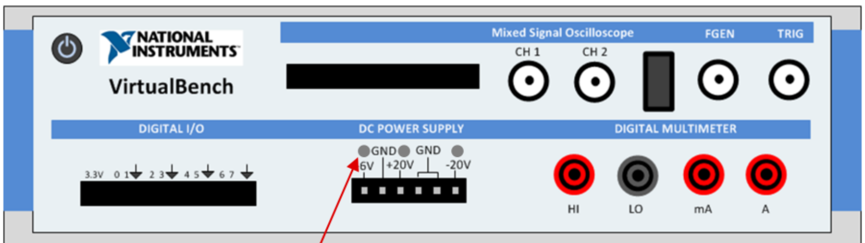
Texas Instruments

Also make sure the Function Generator is OFF.

Disable DC Power Supply



Power button GRAY = DC power supply OFF

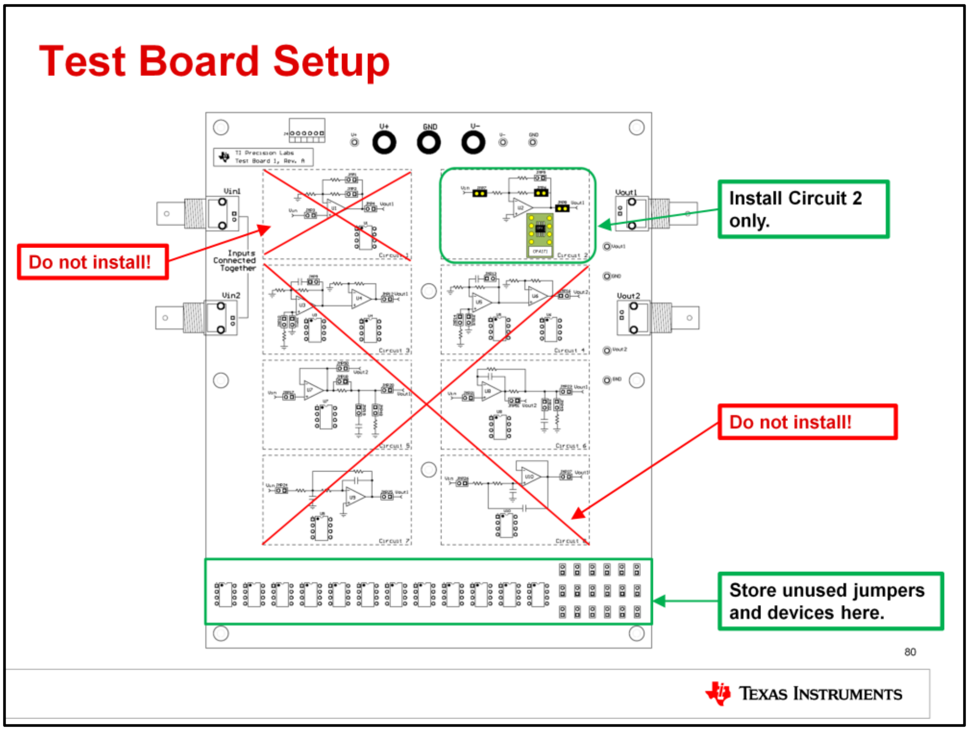


LEDs OFF = DC power supply OFF

79

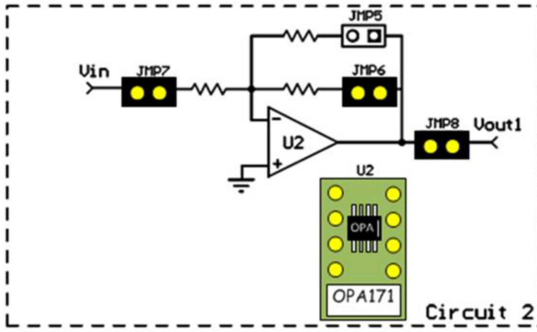
Texas Instruments

Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For this measurement, only circuit 2 is used. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

Test Board Setup - Jumpers



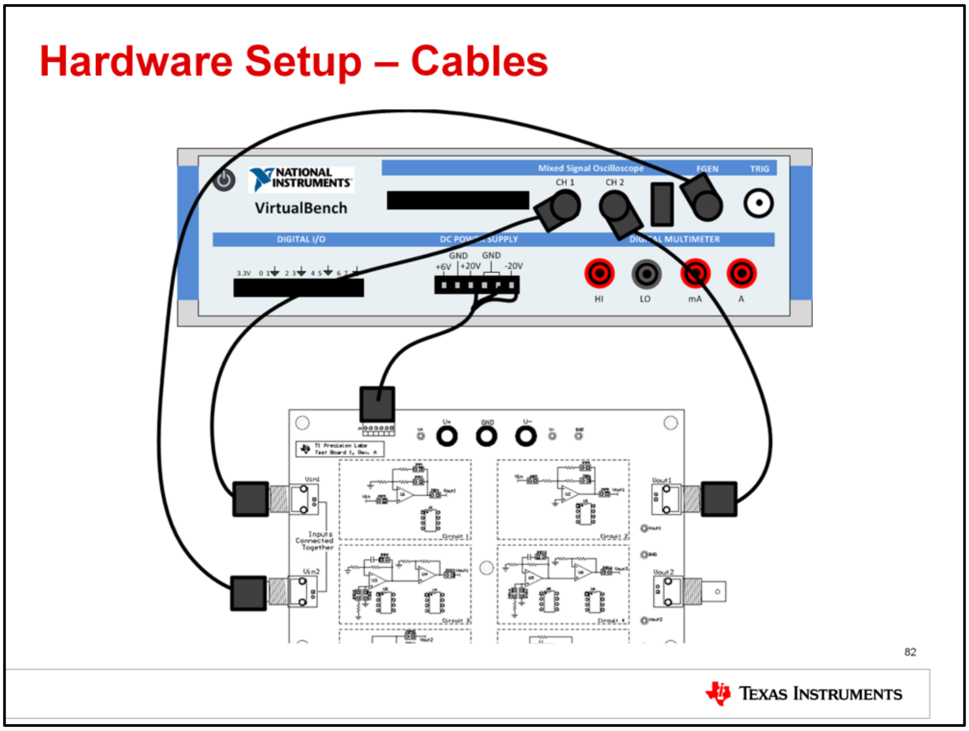
Jumper, Device	Description
JMP6	Select Rf = 20kΩ for gain = -20V/V
JMP7	Connect input to signal source
JMP8	Connect Circuit 2 output to Vout1
U2	Install OPA171

81

TEXAS INSTRUMENTS

To prepare the test board for the measurement, install the jumpers and devices on circuit 2 as shown here.

Install JMP6, JMP7, and JMP8, as well as the OPA171 in socket U2.



This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board. Connect Vin2 on the test board to VirtualBench channel FGEN, or function generator. Then connect Vin1 on the test board to VirtualBench oscilloscope channel 1, and Vout1 on the test board to VirtualBench oscilloscope channel 2.

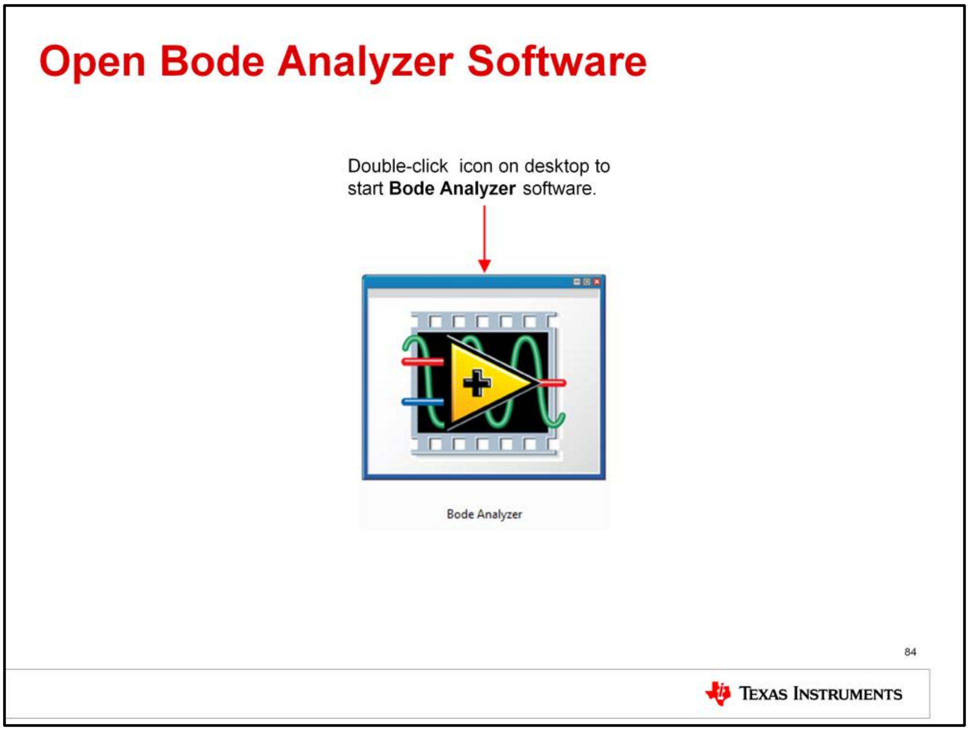
Close VirtualBench Software

Click the '**X**' to close

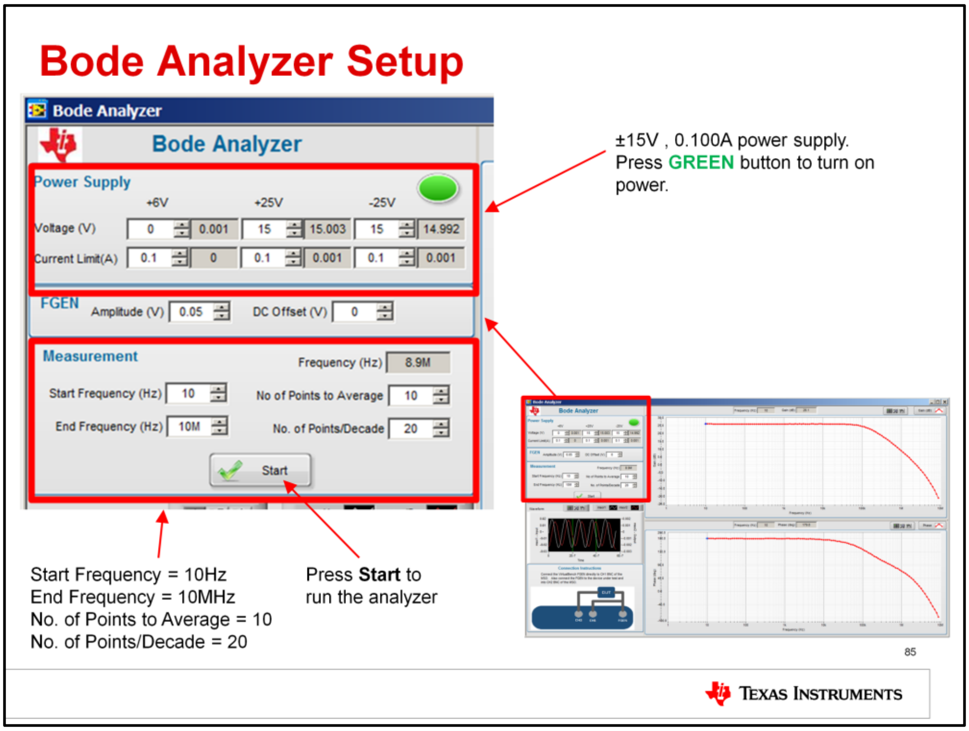
83

Texas Instruments

The VirtualBench software must be closed before continuing with the lab. Click the 'X' in the top-right corner of the software to close.

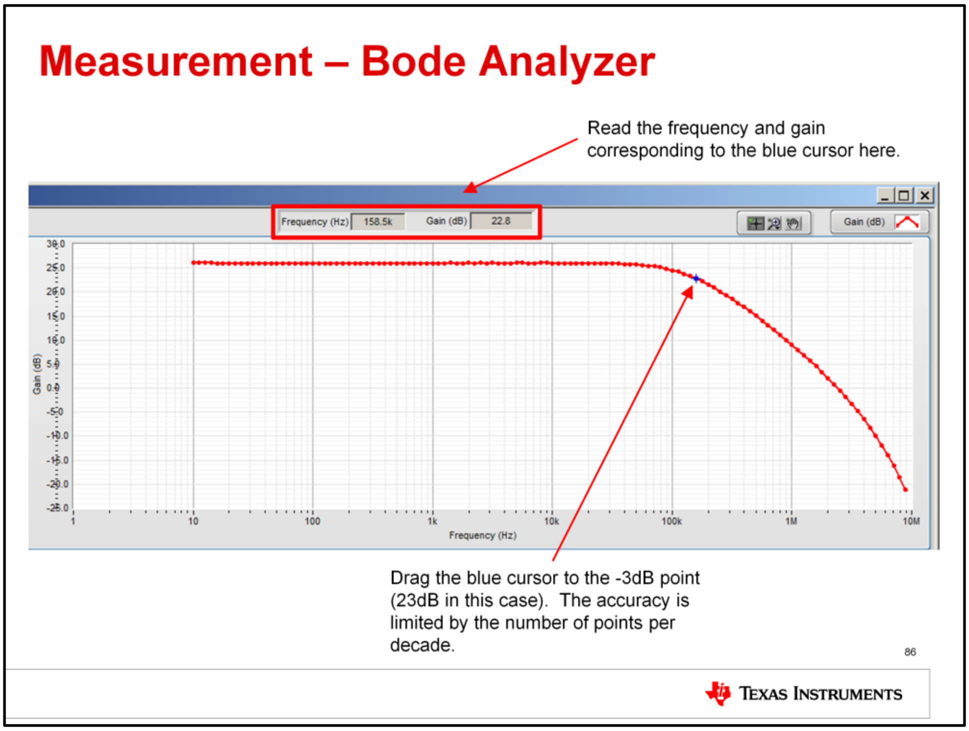


This lab requires additional Bode analyzer software. Install the software, then run it by double-clicking the Bode Analyzer icon on the desktop. You may also run the software by clicking Start → All Programs → Bode Analyzer → Bode Analyzer.



In the configuration panel, set the power supply to ±15V, 0.1A. Press the green button to turn on the power.

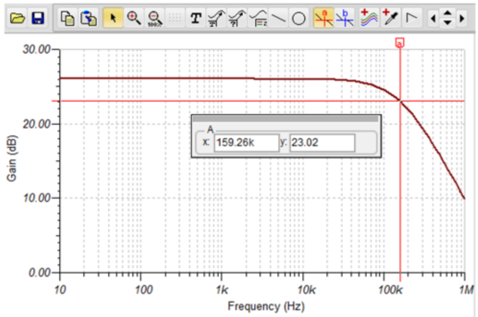
Set the start frequency to 10Hz and the end frequency to 10MHz. Set the number of points to average to 10, and the number of points per decade to 20. Press “Start” to run the Bode analyzer.

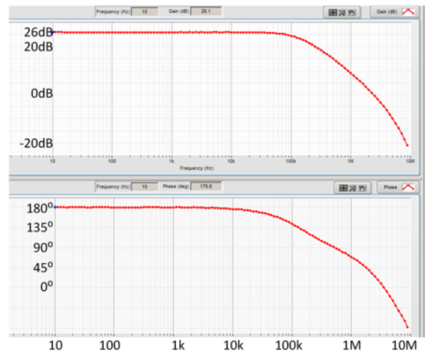


You should see a result similar to this. Enable the cursor, then drag the cursor to the -3dB point, or 23dB in this case. Take note of the frequency – in this case the result is 159kHz, although your results may vary slightly.

Measurement Results – Bandwidth

1. Compare TINA-TI™ simulation results to measured results.





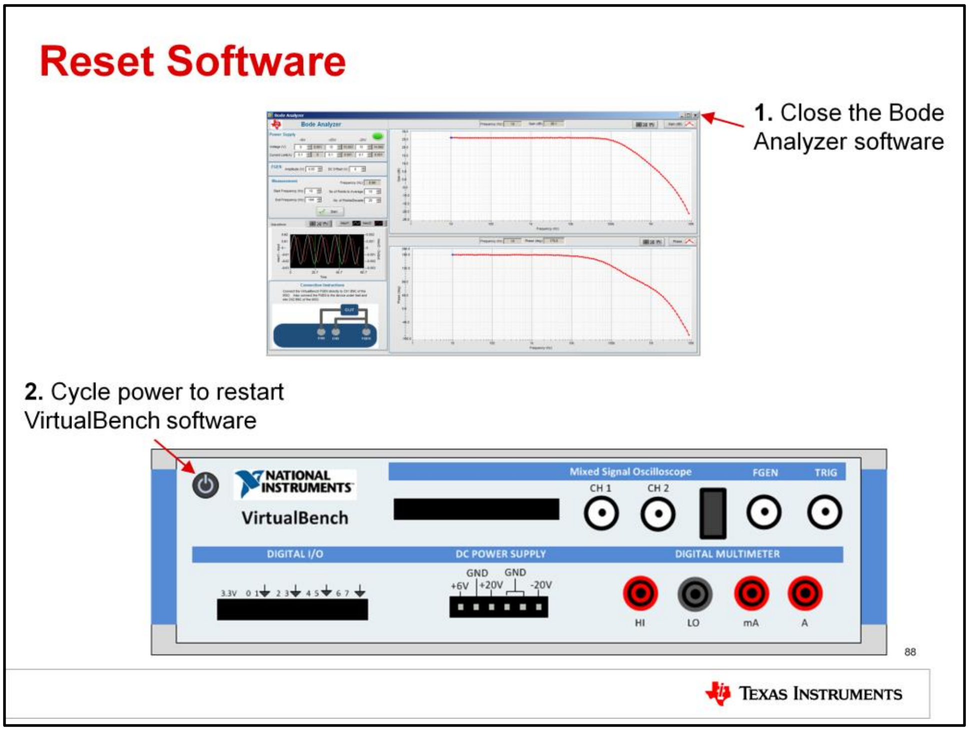
2. Compare the calculated, simulated, and measured bandwidth for the OPA171.

Device	Calculated	Simulated	Measured
OPA171	143kHz	159kHz	159kHz

87

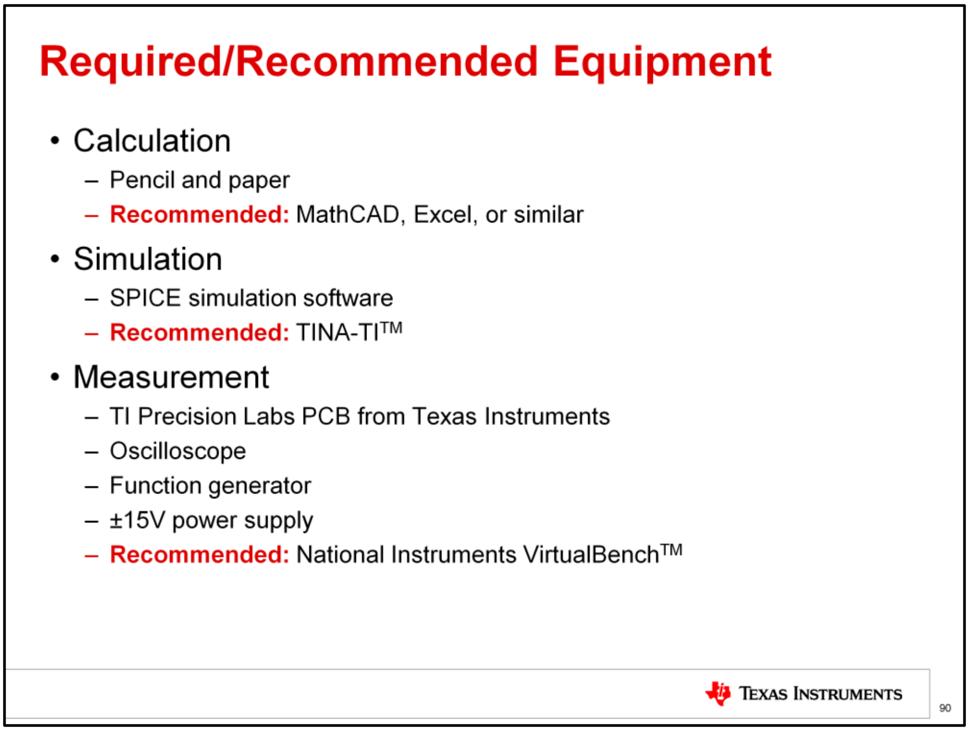
Texas Instruments

Compare the bandwidth measurement of the VirtualBench to the simulated results from TINA-TI. Compare this to your calculated results. You should see very good correlation between all three values, although your numbers may vary slightly.



The bandwidth lab is now complete. Before continuing to the next lab, you must reset the software. First, close the Bode Analyzer software by clicking the 'X' in the top-right corner. Next, cycle power on the VirtualBench by turning power off and on. This will restart the VirtualBench software.

Hello, and welcome to the presentation for the TI Precision Lab supplement for op amp slew rate. This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the slew rate lecture.



The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly.

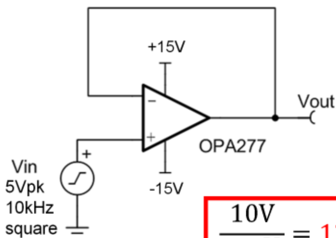
The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

Finally, the real-world measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any oscilloscope, function generator, and ±15V power supply. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.

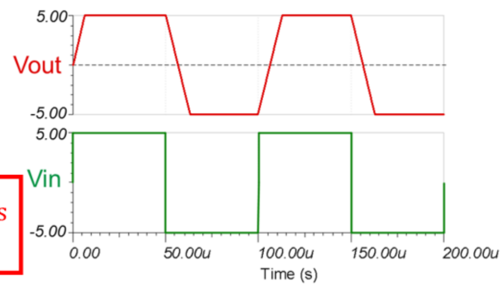
Calculation – Slew Rate

Draw the output voltage waveform of the circuit shown below, based on the data sheet slew rate parameters given in the table.

PARAMETER		OPA277			
		MIN	TYP	MAX	UNIT
Gain-Bandwidth Product	GBW		1.0		MHz
Slew Rate	SR		0.8		V/µs



$$\frac{10\text{V}}{0.8\,\frac{\text{V}}{\mu\text{s}}} = 12.5\mu\text{s}$$



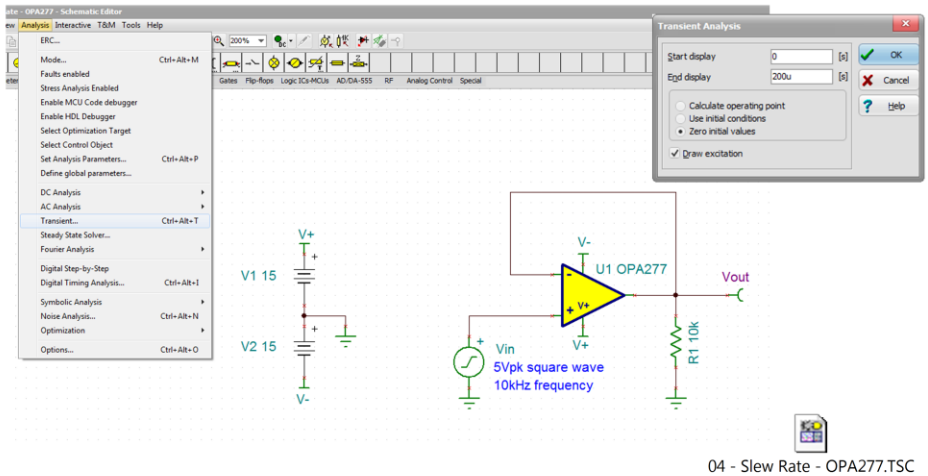
91

TEXAS INSTRUMENTS

First, draw the output voltage waveform of the circuit shown below. Use the slew rate specifications given in the table and the techniques from the slew rate lecture.

Simulation Setup – Slew Rate

Click **Analysis → Transient** to show the slew rate limitations for the OPA277.
Run the analysis from 0µs to 200µs. The input is a 5Vpk, 10kHz square wave.

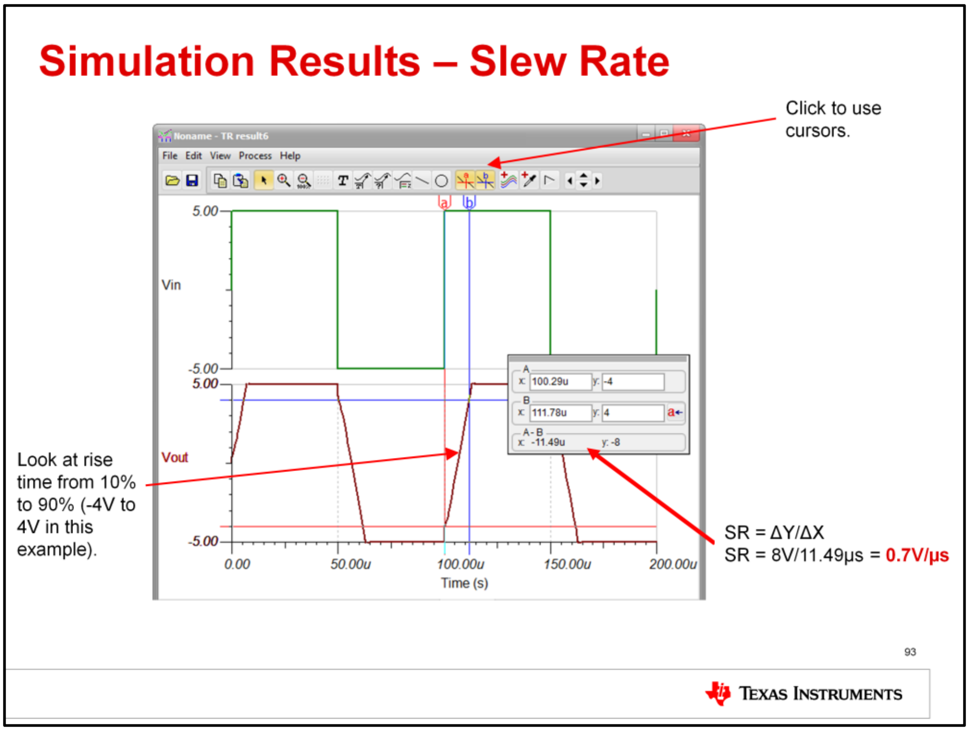


92

TEXAS INSTRUMENTS

The next step is to run a SPICE simulation analysis for the transient output voltage behavior. This will allow us to see the op amp's output voltage response for a specified input signal, which in this case is a 5Vp, 10kHz square wave.

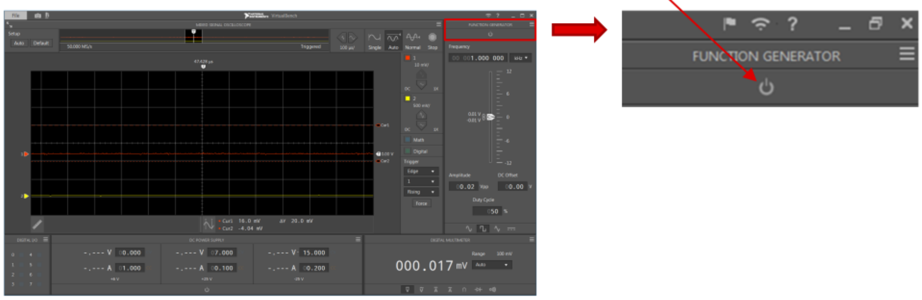
The necessary TINA-TI simulation schematic is embedded in this slide set – simply double-click the icon to open it. To simulate the transient output behavior, click Analysis → Transient. Run the analysis from 0µs to 200µs.



You should see a result similar to this. Vin is a square wave, as expected, but Vout must obey the slew rate limitations of the OPA277. You can see this effect in the rise time from 10% to 90% of the output, or -4V to 4V. Use the cursors to measure the time required to slew from -4V to 4V, and use the equation that slew rate = delta y over delta x to calculate the simulated slew rate of 0.7V/µs.

Disable Function Generator

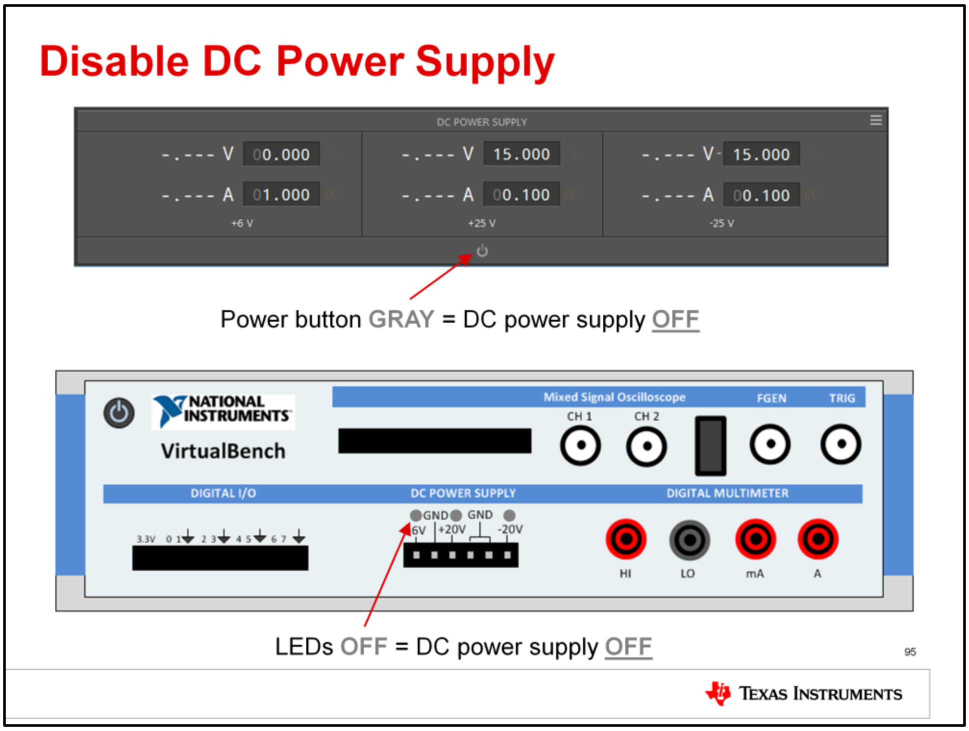
Power button GRAY =
Function Generator OFF



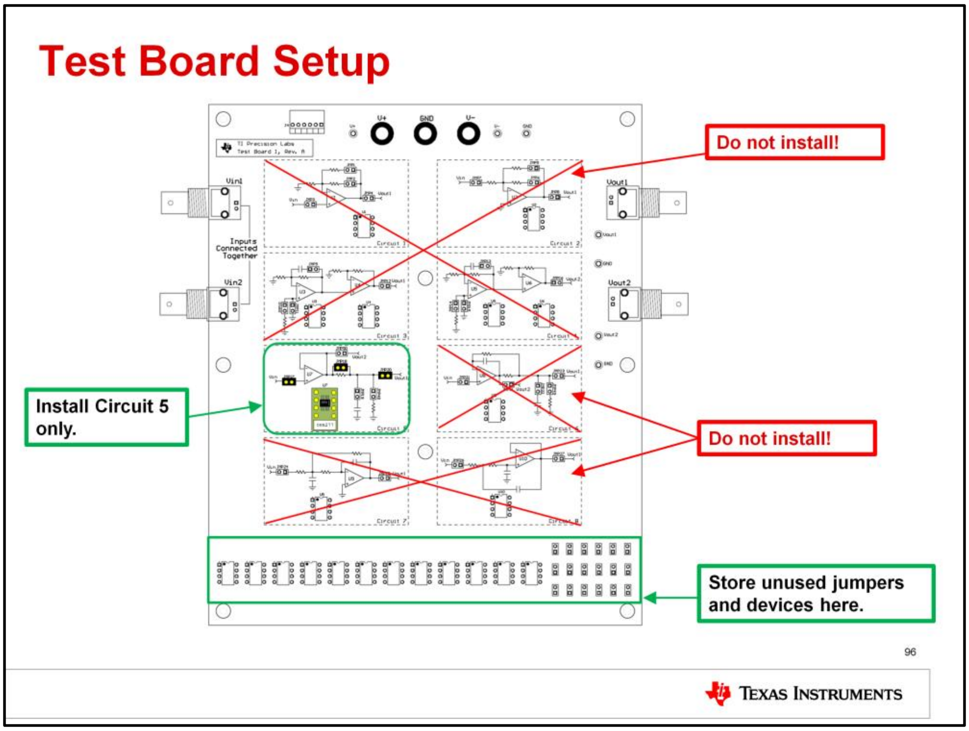
94

TEXAS INSTRUMENTS

Also make sure the Function Generator is OFF.

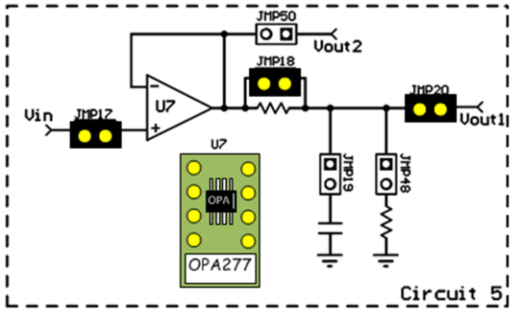


Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuit 5. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

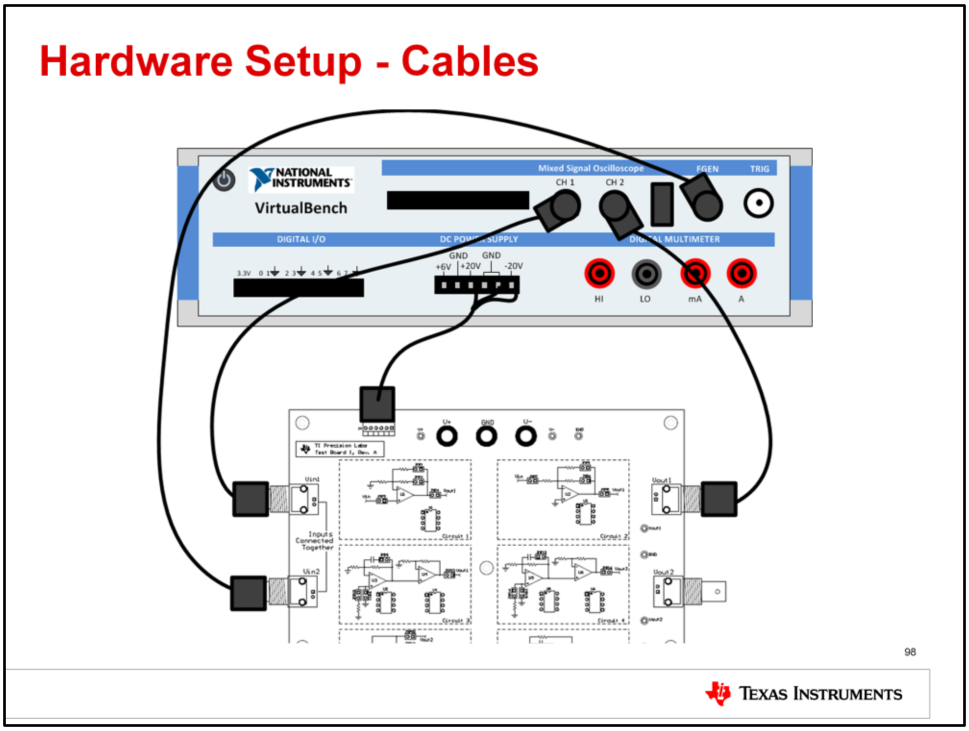
Test Board Setup – Jumpers



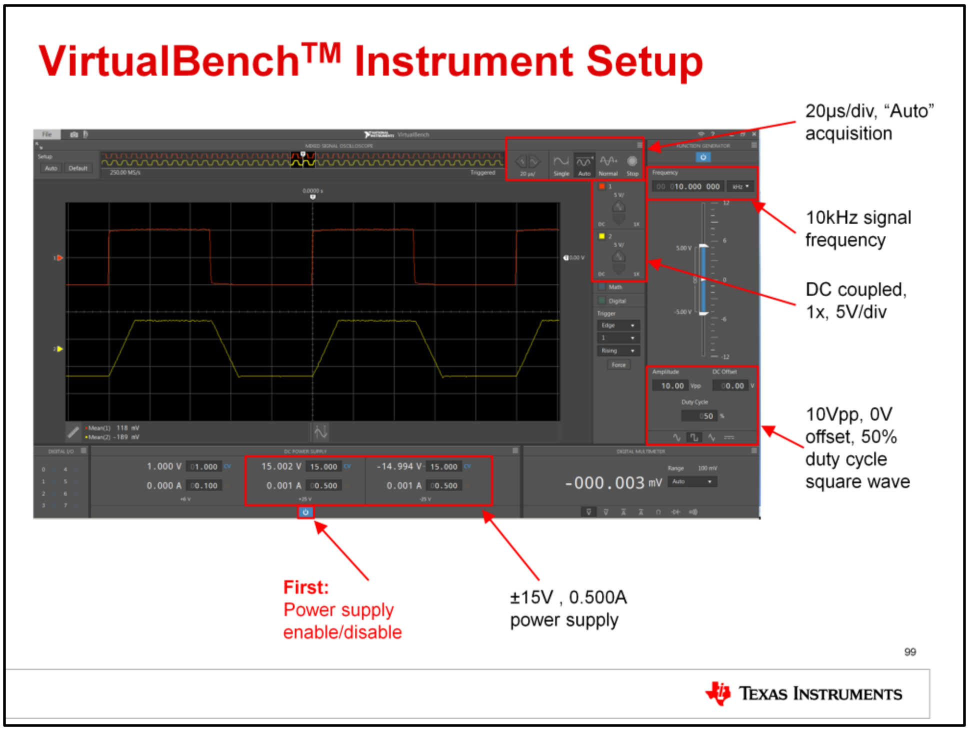
Jumper, Device	Description
JMP17	Connect input to signal source.
JMP18	Short output resistance
JMP20	Connect Circuit 5 output to Vout1
U7	Install OPA277

To prepare the test board for the measurement, install the jumpers and devices on circuit 5 as shown here.

Install JMP17, JMP18, and JMP20, as well as the OPA277 in socket U7.



This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board. Connect Vin2 on the test board to VirtualBench channel FGEN, or function generator. Then connect Vin1 on the test board to VirtualBench oscilloscope channel 1, and Vout1 on the test board to VirtualBench oscilloscope channel 2.

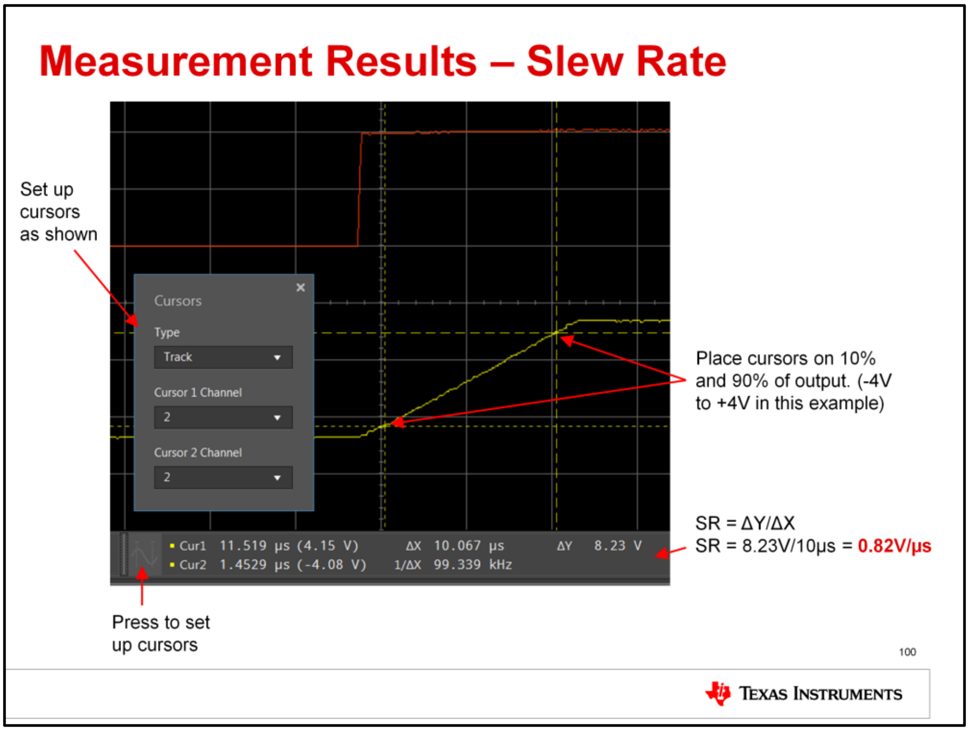


Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

Set the time scale to 20us per division, with the acquisition mode set to "Auto." Enable channels 1 and 2 on the oscilloscope, and set them to 1x, DC-coupled mode, 5V/div. Enable the function generator and setup the signal as follows:

10kHz frequency, 10Vpp, 0V offset, 50% duty cycle square wave.

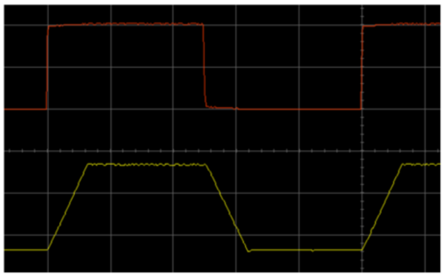
Set the +25V power supply to +15V, 0.500A. Set the -25V power supply to -15V, 0.500A. Press the power button to turn on the power supply rails.

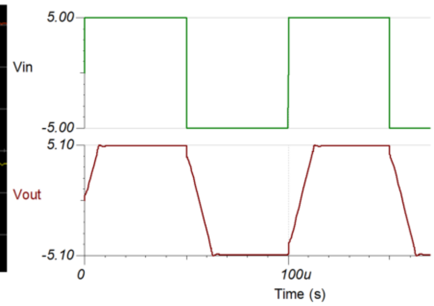


Let's now determine the measured slew rate. Enable cursors, set them up as shown, then place the cursors on 10% and 90% of the output signal. Take note of the time and voltage difference, and then calculate the slew rate using the equation given. In this example, a slew rate of 0.82V/us was measured. You may have different results in your experiment.

Measurement Results – Slew Rate

1. Compare TINA-TI™ simulation results to measured results.



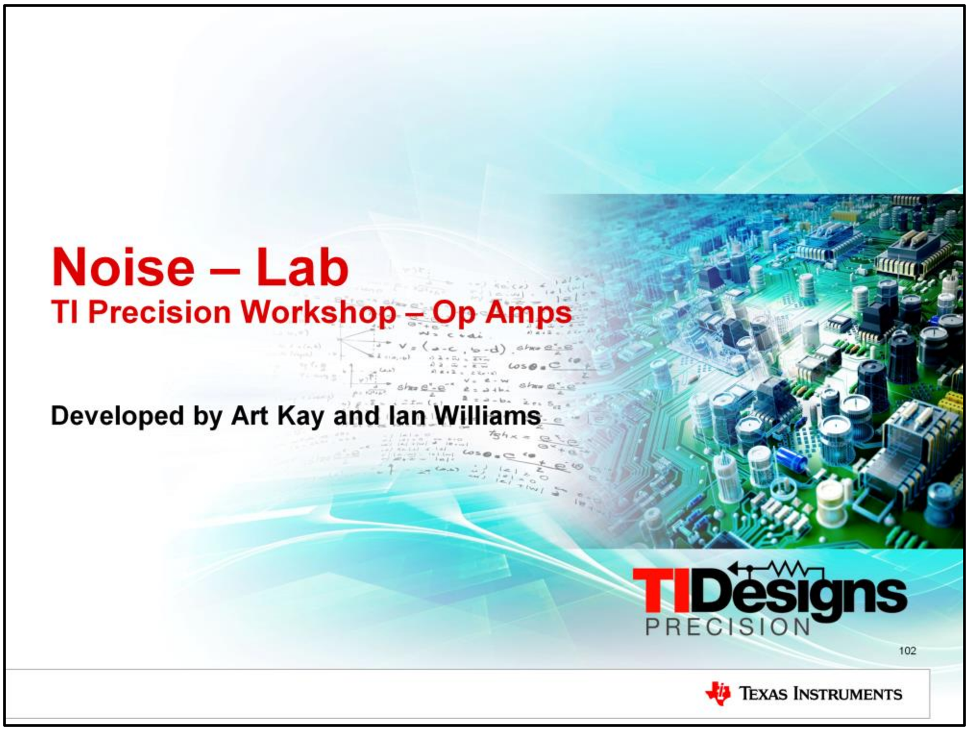


Slew Rate (data sheet):	0.8V/µs
Slew Rate (simulation):	8V / 11.49µs= 0.7V/µs
Slew Rate (measurement):	8.23V / 10µs = 0.82V/µs

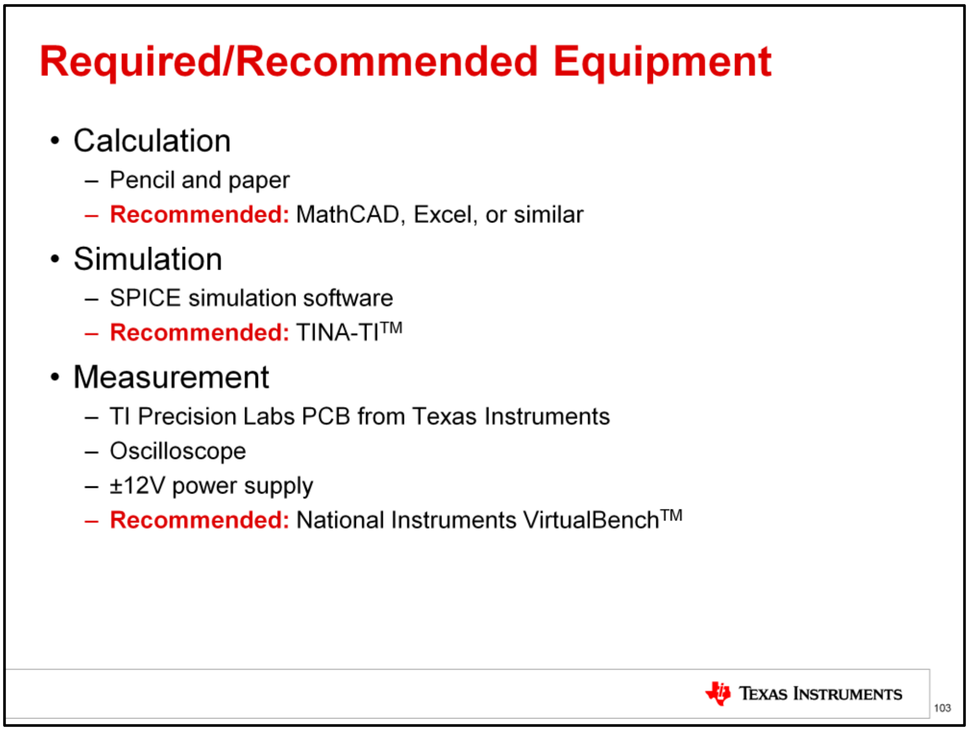
101

TEXAS INSTRUMENTS

Compare the oscilloscope display of the VirtualBench to the simulation results from TINA-TI. Also compare the slew rate values from the data sheet, simulation, and measurement. They should all be similar, although you may get slightly different results.



Hello, and welcome to the presentation for the TI Precision Lab supplement for intrinsic op amp noise. This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the noise lecture.



The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly since noise calculations can involve many steps.

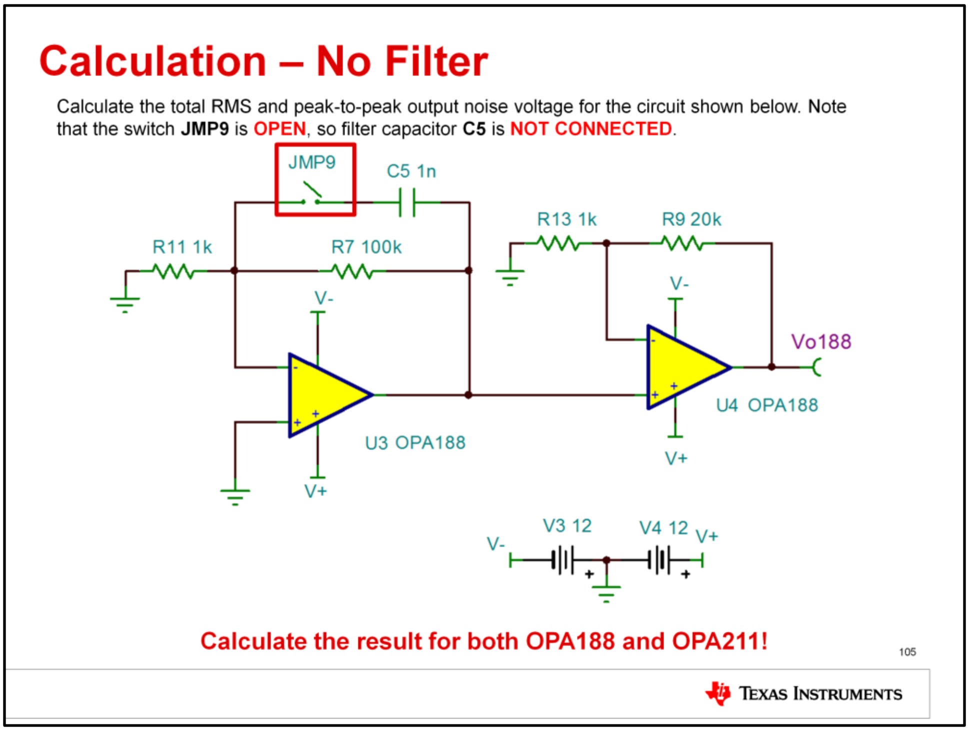
The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

Finally, the real-world noise measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any ±12V power supply and oscilloscope. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.

Experiment 1
Output Noise – No Filter

Texas Instruments

In experiment 1, we'll determine the total output voltage noise in a circuit with no filtering.



First, calculate the expected total RMS and peak-to-peak output noise voltage for the circuit shown here, using the techniques and equations given in the noise lecture. Note that the switch **JMP9** is open, so filter capacitor C5 is not connected to the circuit. Make this calculation twice – first with the OPA188 selected for U1 and U2, then with the OPA211. The different parameters of these op amps will give you different noise results.

Calculation – No Filter

1. Use the data sheet parameters given below to calculate output noise.

PARAMETER		CONDITIONS	OPA188			
			MIN	TYP	MAX	UNIT
Input Voltage Noise Density	e_n	f = 1kHz		8.8		nV / √Hz
Input Current Noise Density	i_n	f = 1kHz		7		fA / √Hz
Gain Bandwidth	GBW			2		MHz

PARAMETER		CONDITIONS	OPA211			
			MIN	TYP	MAX	UNIT
Input Voltage Noise Density	e_n	f = 1kHz		1.1		nV / √Hz
Input Current Noise Density	i_n	f = 1kHz		3.2		pA / √Hz
Gain Bandwidth	GBW			80		MHz

Answers	OPA188	OPA211
Total RMS output noise	$3.6mV_{RMS}$	$10.7mV_{RMS}$
Total peak-to-peak noise	$22mV_{PP}$	$64mV_{PP}$

106

Texas Instruments

In order to perform the noise calculations, you need to know certain parameters of each op amp. The key parameters are the op amp gain bandwidth product, or GBW, input voltage noise density, or e_n, and input current noise density, or i_n. This slide gives a table of these key specs for both the OPA188 and OPA211.

Enter your answers in the table at the bottom of the slide. The solutions are already provided to allow you to check your work.

Calculation – No Filter, OPA188

OPA188 noise, no filter

Thermal noise calcualtion

$k_n := 1.38 \cdot 10^{-23}$ $T_n := 273 + 25$ $R_n := 1 \cdot 10^3$

$e_{n\_r} := \sqrt{4k_n \cdot T_n \cdot R_n} = 4.056 \times 10^{-9} \frac{V}{\sqrt{Hz}}$

op-amp current and voltage noise

$e_{n\_opa} := 8.8 \times 10^{-9} \frac{V}{\sqrt{Hz}}$ From OPA188 Data sheet

$i_n := 7 \times 10^{-15}$ From OPA188 Data sheet

$e_{n\_i} := R_n \cdot i_n = 7 \times 10^{-12}$

Add thermal noise and voltage noise calcualtion

$e_{n\_total} := \sqrt{e_{n\_r}^2 + e_{n\_opa}^2 + e_{n\_i}^2} = 9.69 \times 10^{-9}$

Gain from two stage amp

$G_{n1} := 101$ $G_{n2} := 21$

$GBW := 2 \cdot 10^6$ From OPA188 Data sheet

Closed loop bandwidth (-3dB)

$f_c := \frac{GBW}{G_{n1}} = 19.802 \times 10^3$ Hz

Noise bandwidth (-3dB)

$BW_n := 1.57 \cdot f_c = 31.089 \times 10^3$ Hz

$E_n := e_{n\_total} \cdot \sqrt{BW_n} = 1.708 \times 10^{-6}$ Vrms

Total rms noise

$E_{n\_out} := G_{n1} \cdot G_{n2} \cdot E_n = 3.624 \times 10^{-3}$ Vrms

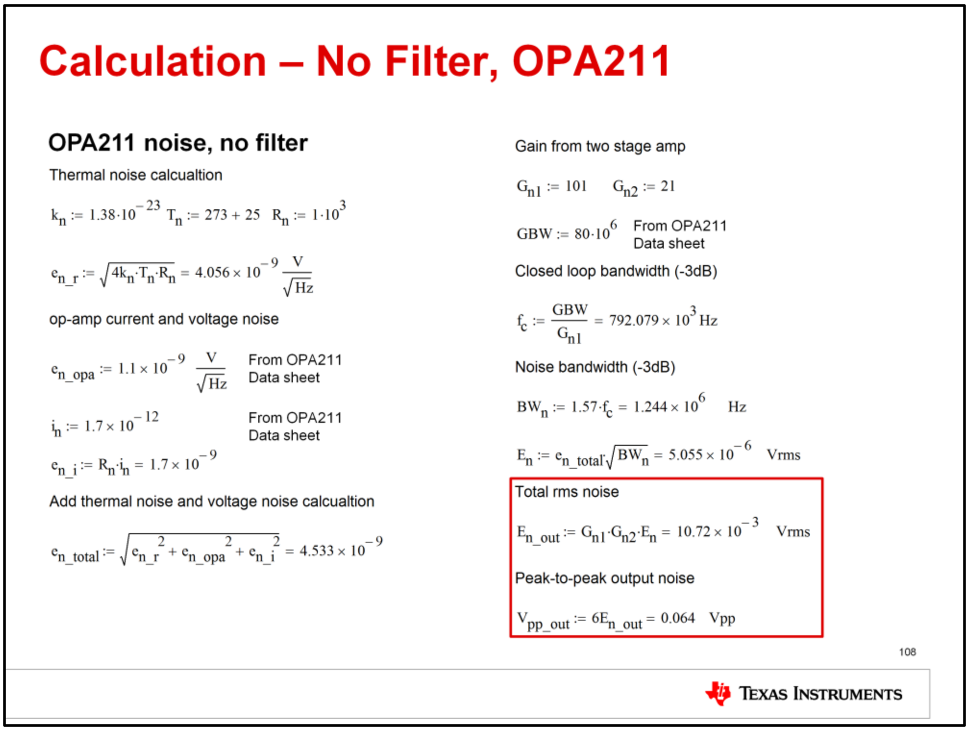
Peak-to-peak output noise

$V_{pp\_out} := 6E_{n\_out} = 0.022$ Vpp

107

TEXAS INSTRUMENTS

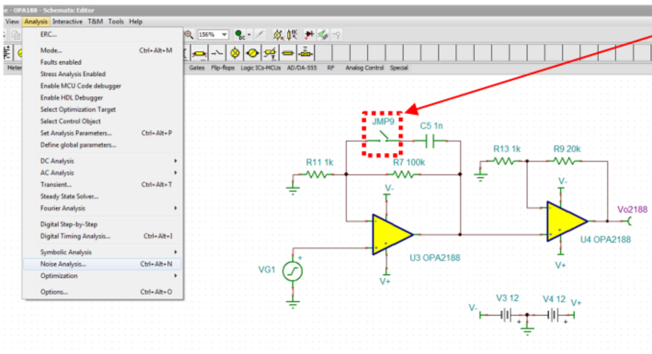
I won't go through the entire calculation, but will give an overview of the key steps. First, the resistor thermal noise spectral density is calculated using the circuit resistor values. Next, the total input noise spectral density is computed based on the resistor thermal noise, op amp input voltage noise, and op amp input current noise once it's been converted to voltage. The op amp's GBW and the circuit's closed-loop bandwidth are used to calculated the noise bandwidth, which enables you to determine the total input-referred RMS noise. Finally, the input-referred noise is multiplied by the closed-loop gain in order to compute the total RMS and peak-to-peak output noise. This slide shows the complete calculation for the OPA188.



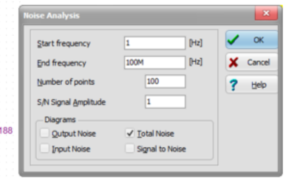
This slide shows the calculation for the OPA211. The op amp's input voltage and current spectral density, as well as the op amp gain bandwidth product, are different, but the steps of the calculation are exactly the same. Simply substitute in the new values and take note of the result.

Simulation Setup – No Filter

Click **Analysis → Noise Analysis** to compute the total output noise. Run the noise analysis from 1Hz to 100MHz and select the **Total Noise** diagram. The filter capacitor is disconnected.



Note that the capacitor is disconnected.



05 - Noise - OPA188.TSC

05 - Noise - OPA211.TSC

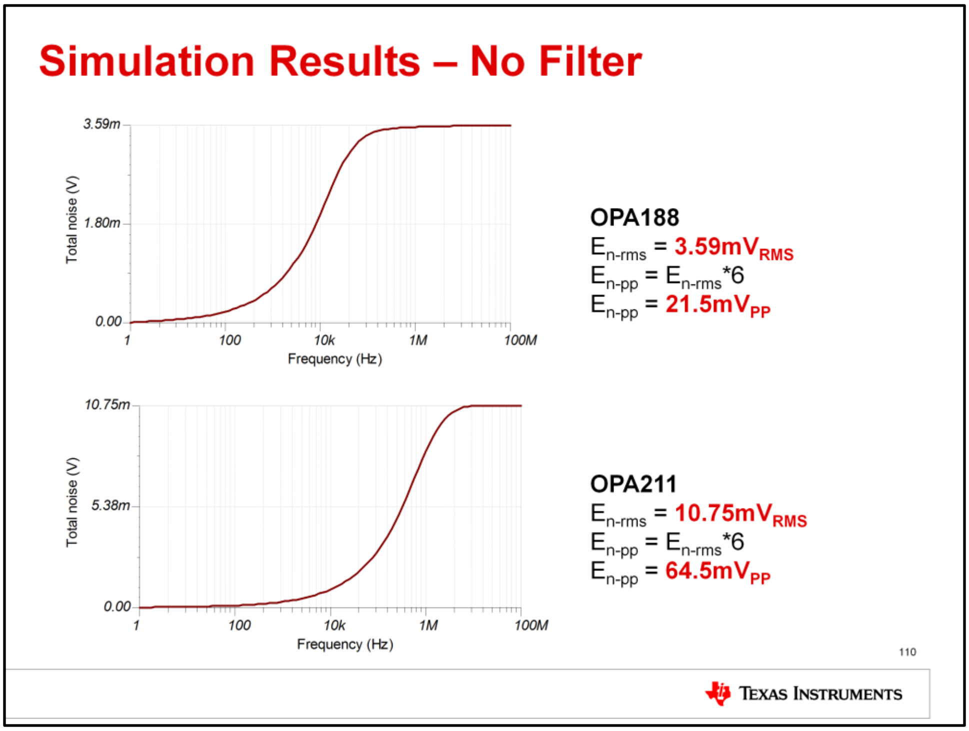
Run the simulation for both the OPA188 and OPA211!

109

Texas Instruments

The next step is to run a SPICE simulation analysis for the total output noise. Simply open the TINA-TI simulation schematics embedded in the presentation, ensure that filter capacitor jumper is open, then select Analysis, followed by Noise Analysis. Make sure that “total noise” is selected, then run the analysis from 1Hz to 100MHz.

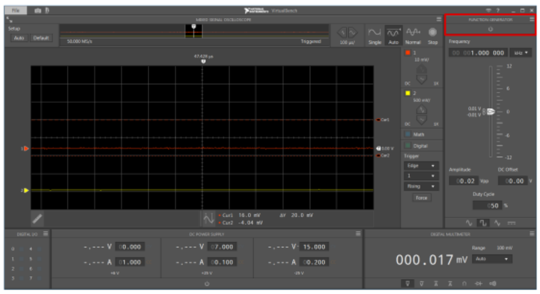
Run the simulation for both the OPA188 and OPA211!

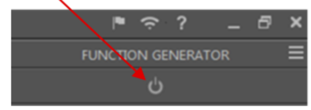


Over the 100MHz frequency range, the OPA188 circuit has a total noise of 3.59mVrms, or 21.5mVpp. The OPA211 circuit has a total noise of 10.75mVrms, or 64.5mVpp.

Disable Function Generator

Power button GRAY =
Function Generator OFF

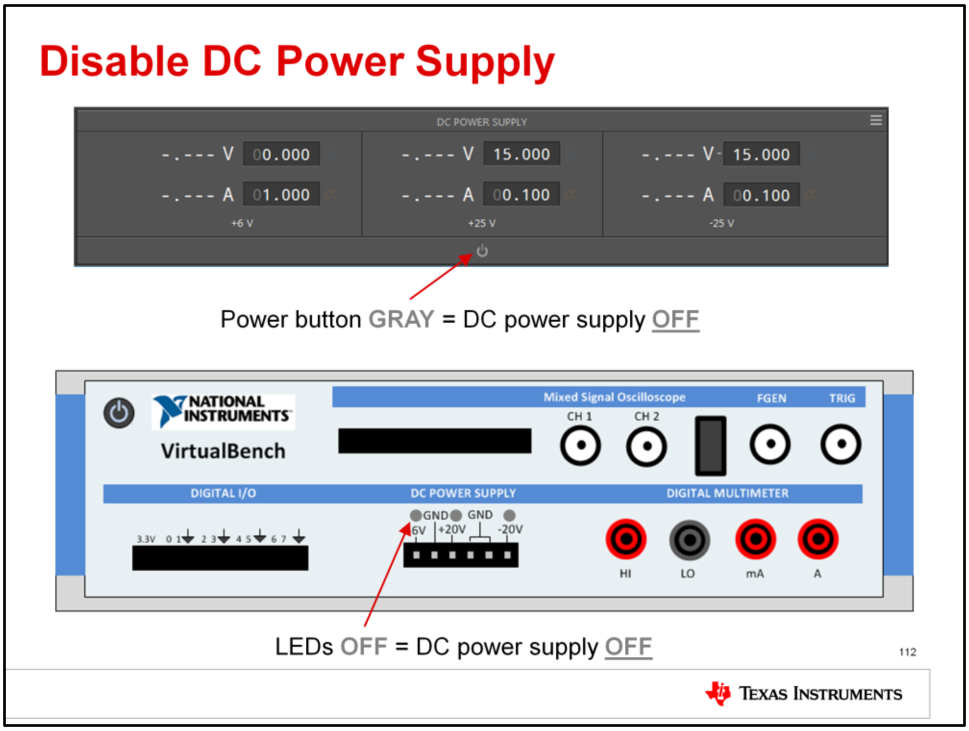




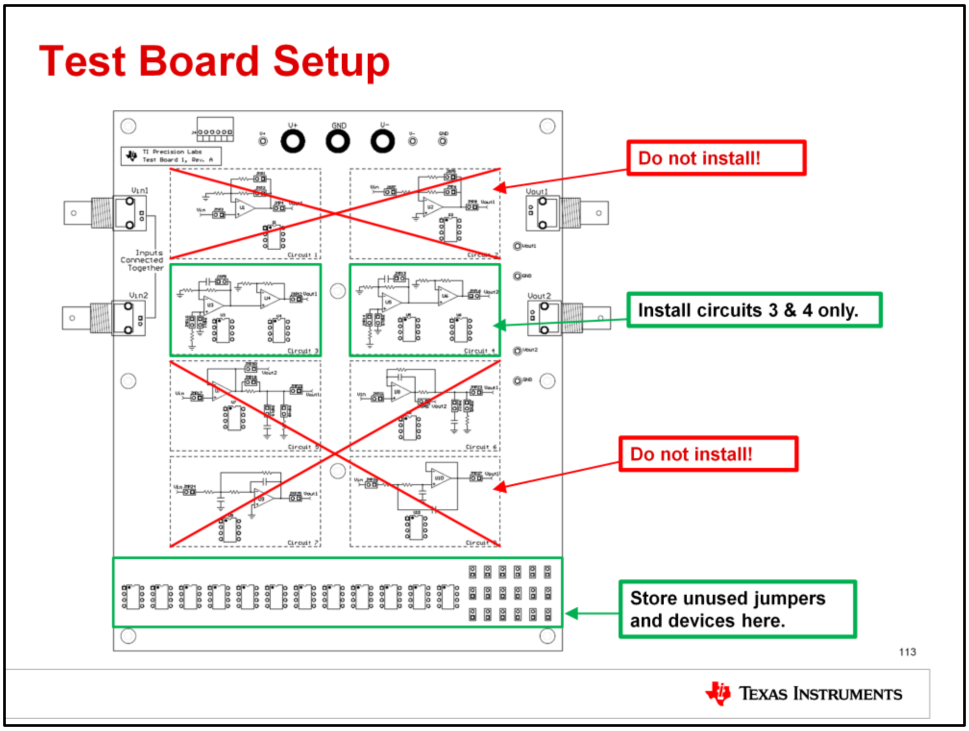
111

TEXAS INSTRUMENTS

Also make sure the Function Generator is OFF.

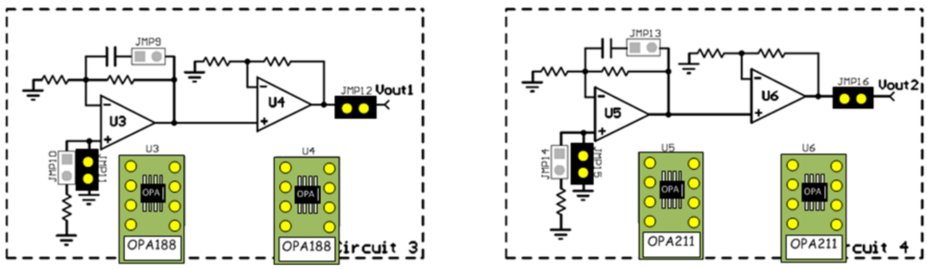


Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuits 3 and 4. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

Test Board Setup - Jumpers



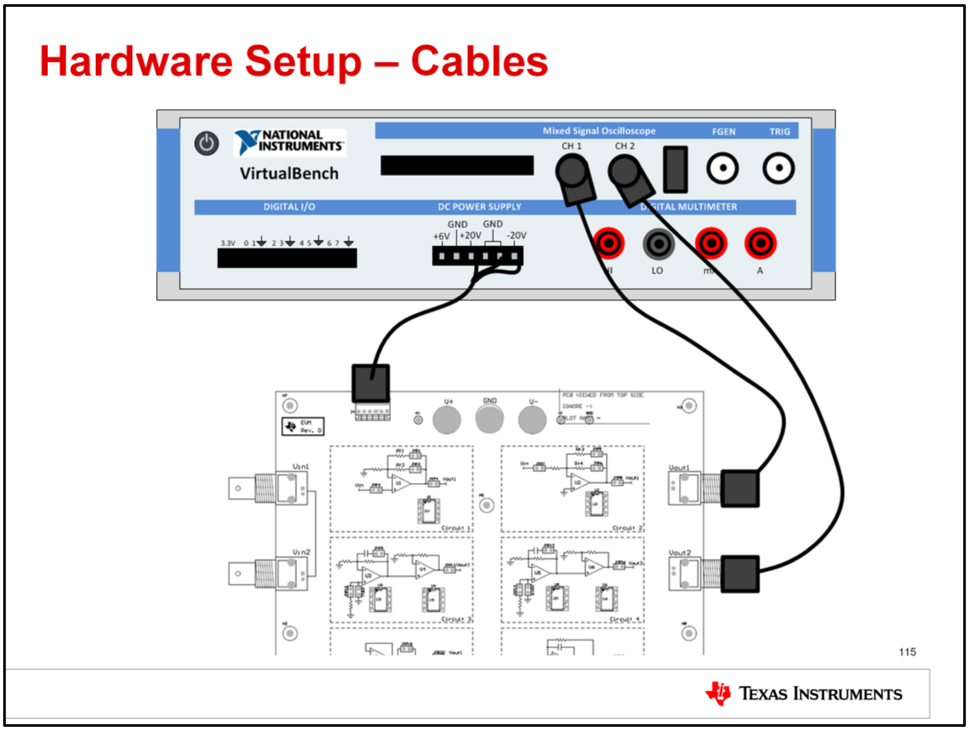
Jumper, Device	Description
JMP11, JMP15	Connect VIN+ to GND
JMP12, JMP16	Connect Circuit 3 to Vout1, connect Circuit 4 to Vout2
U3, U4	Install OPA188
U5, U6	Install OPA211

114

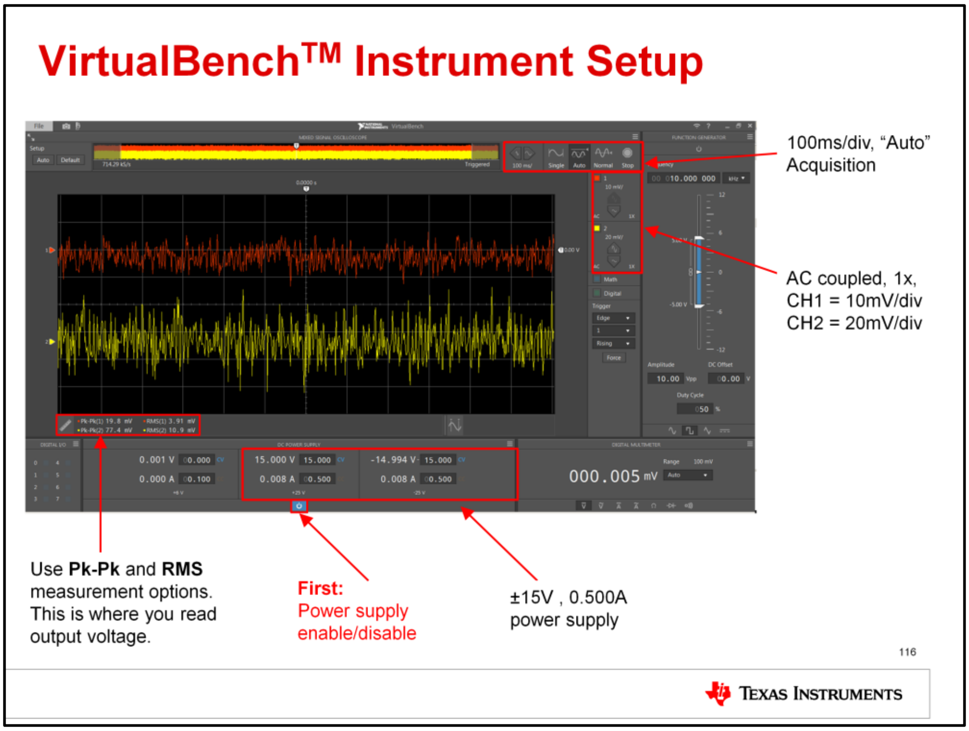
TEXAS INSTRUMENTS

To prepare the test board for the measurement, install the jumpers and devices on circuit 3 and circuit 4 as shown here.

On circuit 3, install JMP11 and JMP12, as well as the OPA188 in sockets U3 and U4. On circuit 4, install JMP15 and JMP16, as well as the OPA211 in sockets U5 and U6.



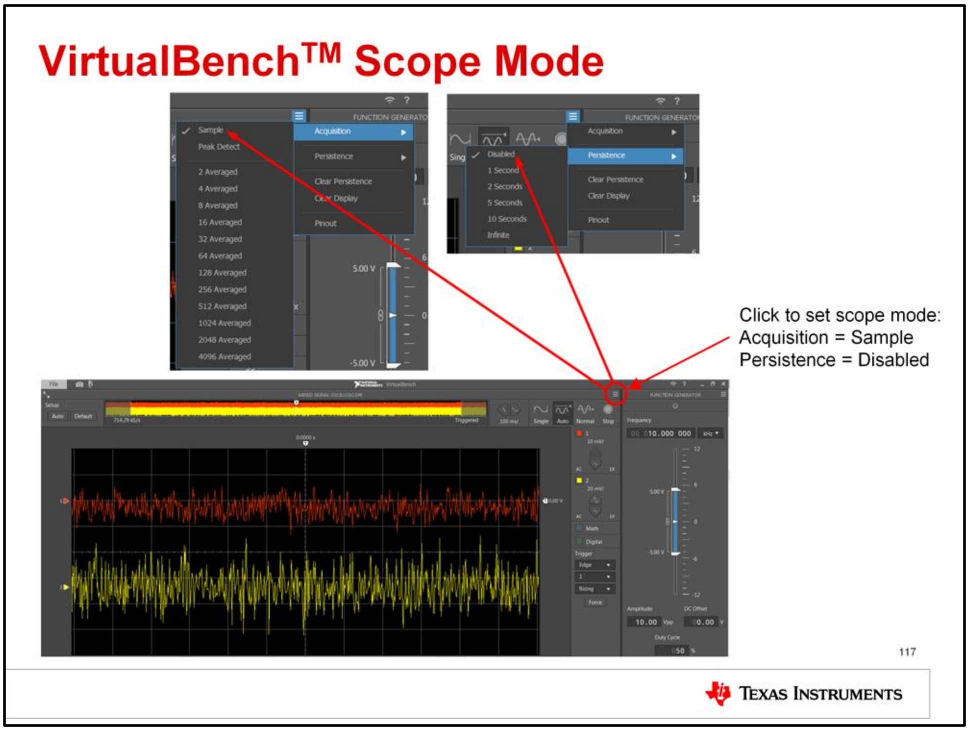
This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board. Connect Vout1 on the test board to VirtualBench oscilloscope channel 1, and Vout2 on the test board to Virtual Bench oscilloscope channel 2, using BNC cables.



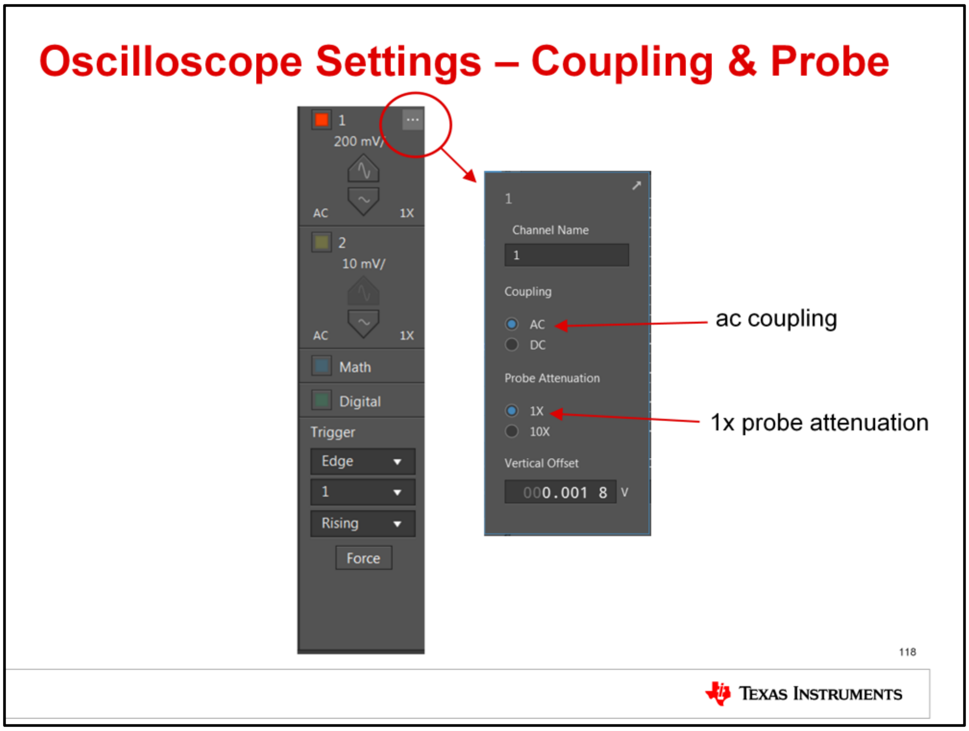
Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

Set the time scale to 100ms per division, with the acquisition mode set to "Auto." Enable channels 1 and 2 on the oscilloscope, and set them to 1x, AC-coupled mode. Set the vertical scale on channel 1 to 10mV/div, and on channel 2 to 20mV/div. Set the +25V power supply to +15V, 0.500A. Set the -25V power supply to -15V, 0.500A. Press the power button to turn on the power supply rails.

Enable Peak-to-peak and RMS measurements on both channels in order to read the output voltage of each circuit.



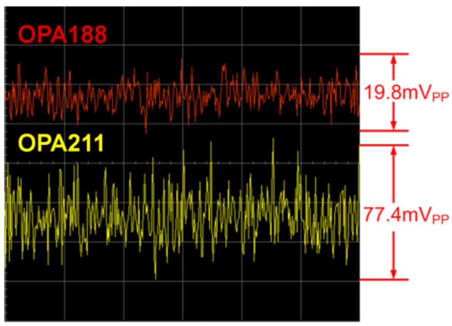
You must also set the mode of the Virtual Bench oscilloscope. Click the button shown on the front panel, then set Acquisition to “Sample” and Persistence to “Disabled.”

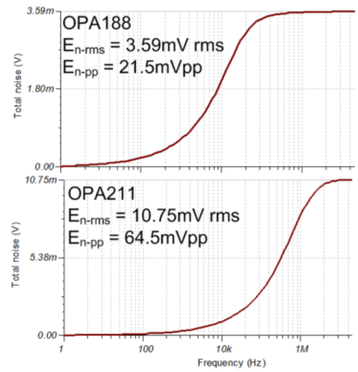


The icon at the top right of the oscilloscope opens a new window with more options for each channel. Each channel can be given a custom name. Set the coupling mode to AC. Set probe attenuation to 1x.

Measurement Results – No Filter

1. Compare TINA-TI™ simulation results to measured and calculated results.





Device	Calculated Noise (RMS)	Simulated Noise (RMS)	Measured Noise (RMS)
OPA188	$3.62mV_{RMS}$	$3.59mV_{RMS}$	$3.3mV_{RMS}$
OPA211	$10.72mV_{RMS}$	$10.75mV_{RMS}$	$12.9mV_{RMS}$

Note: Measured RMS noise is determined using measured peak-to-peak noise divided by 6.

119

Texas Instruments

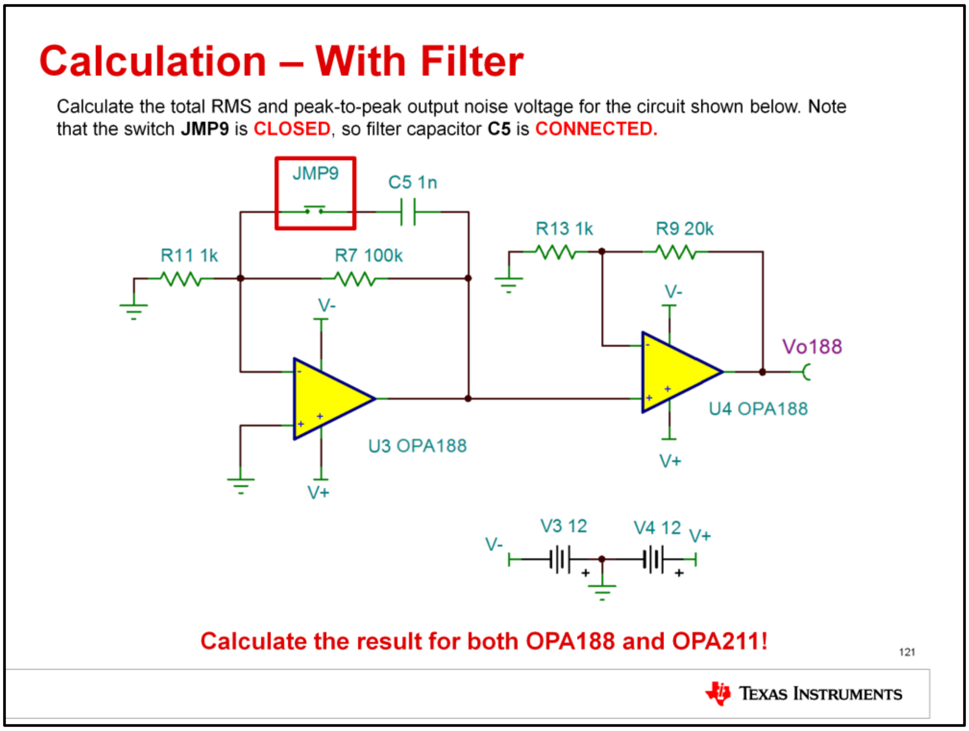
The expected output voltage noise results from the measurement are shown in the screenshot at the top left. The OPA188 has a measured noise of 19.8mVpp, or 3.3mVrms, and the OPA211 has a measured noise of 77.4mVpp, or 12.9mVrms. As you can see from the table, this agrees extremely well with the results from calculation and simulation!

Experiment 2
Output Noise with Filter

120

TEXAS INSTRUMENTS

In experiment 2, we'll determine the total output voltage noise in a circuit with a filter capacitor in the feedback network.



Perform another total noise calculation on the circuit shown. However, this time a filter capacitor is added to the feedback network of the first amplifier stage. This capacitor will reduce the circuit's noise bandwidth, so the overall noise performance will be significantly improved. As before, do the calculations with both the OPA188 and OPA211.

Calculation – With Filter

1. Use the data sheet parameters given below to calculate output noise.

PARAMETER		CONDITIONS	OPA188			
			MIN	TYP	MAX	UNIT
Input Voltage Noise Density	e_n	f = 1kHz		8.8		nV / √Hz
Input Current Noise Density	i_n	f = 1kHz		7		fA / √Hz
Gain Bandwidth	GBW			2		MHz

PARAMETER		CONDITIONS	OPA211			
			MIN	TYP	MAX	UNIT
Input Voltage Noise Density	e_n	f = 1kHz		1.1		nV / √Hz
Input Current Noise Density	i_n	f = 1kHz		3.2		pA / √Hz
Gain Bandwidth	GBW			80		MHz

Answers	OPA188	OPA211
Total RMS output noise	$1mV_{RMS}$	$0.48mV_{RMS}$
Total peak-to-peak noise	$6mV_{PP}$	$2.88mV_{PP}$

122

TEXAS INSTRUMENTS

The key parameters of op amp gain bandwidth product, or GBW, input voltage noise density, or e_n, and input current noise density, or i_n are given again for your reference.

Enter your answers in the table at the bottom of the slide. The solutions are already provided to allow you to check your work.

Calculation – With Filter, OPA188

OPA188 noise, with filter

Thermal noise calcualtion

$k_n := 1.38 \cdot 10^{-23}$ $T_n := 273 + 25$ $R_n := 1 \cdot 10^3$

$e_{n\_r} := \sqrt{4k_n \cdot T_n \cdot R_n} = 4.056 \times 10^{-9} \frac{V}{\sqrt{Hz}}$

op-amp current and voltage noise

$e_{n\_opa} := 8.8 \times 10^{-9} \frac{V}{\sqrt{Hz}}$ From OPA188 Data sheet

$i_n := 7 \times 10^{-15}$ From OPA188 Data sheet

$e_{n\_i} := R_n \cdot i_n = 7 \times 10^{-12}$

Add thermal noise and voltage noise calcualtion

$e_{n\_total} := \sqrt{e_{n\_r}^2 + e_{n\_opa}^2 + e_{n\_i}^2} = 9.69 \times 10^{-9}$

Gain from two stage amp

$G_{n1} := 101$ $G_{n2} := 21$

$GBW := 2 \cdot 10^6$ From OPA188 Data sheet

Closed loop bandwidth (-3dB)

$f_c := \frac{1}{2 \cdot \pi \, 100 \cdot 10^3 \cdot 1 \cdot 10^{-9}} = 1.592 \times 10^3$ Hz

Noise bandwidth (-3dB)

$BW_n := 1.57 \cdot f_c = 2.499 \times 10^3$ Hz

$E_n := e_{n\_total} \sqrt{BW_n} = 4.844 \times 10^{-7}$ Vrms

Total rms noise

$E_{n\_out} := G_{n1} \cdot G_{n2} \cdot E_n = 1.027 \times 10^{-3}$ Vrms

Peak-to-peak output noise

$V_{pp\_out} := 6E_{n\_out} = 6.164 \times 10^{-3}$ Vpp

123

Texas Instruments

The steps in the calculation are nearly the same as before. However, this time the filter capacitor must be taken into account when calculating the circuit's closed-loop bandwidth. With the filter capacitor connected, the closed-loop bandwidth is computed to be 1.59kHz. Without the capacitor, the bandwidth was 19.8kHz! As expected, this helps to reduce the total noise.

Calculation – With Filter, OPA211

OPA211 noise, with filter

Thermal noise calcualtion

$k_n := 1.38 \cdot 10^{-23}$ $T_n := 273 + 25$ $R_n := 1 \cdot 10^3$

$e_{n\_r} := \sqrt{4k_n \cdot T_n \cdot R_n} = 4.056 \times 10^{-9} \frac{V}{\sqrt{Hz}}$

op-amp current and voltage noise

$e_{n\_opa} := 1.1 \times 10^{-9} \frac{V}{\sqrt{Hz}}$ From OPA211 Data sheet

$i_n := 1.7 \times 10^{-12}$ From OPA211 Data sheet

$e_{n\_i} := R_n \cdot i_n = 1.7 \times 10^{-9}$

Add thermal noise and voltage noise calcualtion

$e_{n\_total} := \sqrt{e_{n\_r}{}^2 + e_{n\_opa}{}^2 + e_{n\_i}{}^2} = 4.533 \times 10^{-9}$

Gain from two stage amp

Gain from two stage amp

$G_{n1} := 101$ $G_{n2} := 21$

$GBW := 80 \cdot 10^6$ From OPA211 Data sheet

Closed loop bandwidth (-3dB)

$f_c := \frac{1}{2 \cdot \pi \cdot 100 \cdot 10^3 \cdot 1 \cdot 10^{-9}} = 1.592 \times 10^3$ Hz

Noise bandwidth (-3dB)

$BW_n := 1.57 \cdot f_c = 2.499 \times 10^3$ Hz

$E_n := e_{n\_total} \sqrt{BW_n} = 2.266 \times 10^{-7}$ Vrms

Total rms noise

$E_{n\_out} := G_{n1} \cdot G_{n2} \cdot E_n = 480.62 \times 10^{-6}$ Vrms

Peak-to-peak output noise

$V_{pp\_out} := 6E_{n\_out} = 2.884 \times 10^{-3}$ Vpp

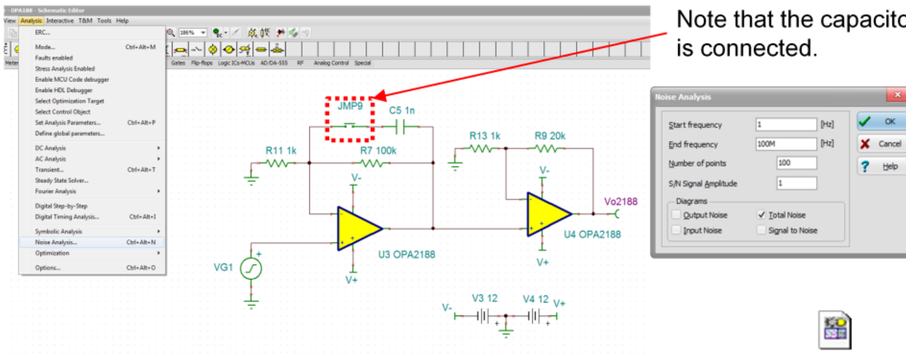
124

Texas Instruments

Step through the calculation again for the OPA211, substituting in for the noise and bandwidth parameters of that device.

Simulation Setup – With Filter

Click **Analysis → Noise Analysis** to compute the total output noise. Run the noise analysis from 1Hz to 100MHz and select the **Total Noise** diagram. Make sure JMP9 is closed so the filter capacitor C5 is connected!



Note that the capacitor is connected.

05 - Noise - OPA188.TSC

05 - Noise - OPA211.TSC

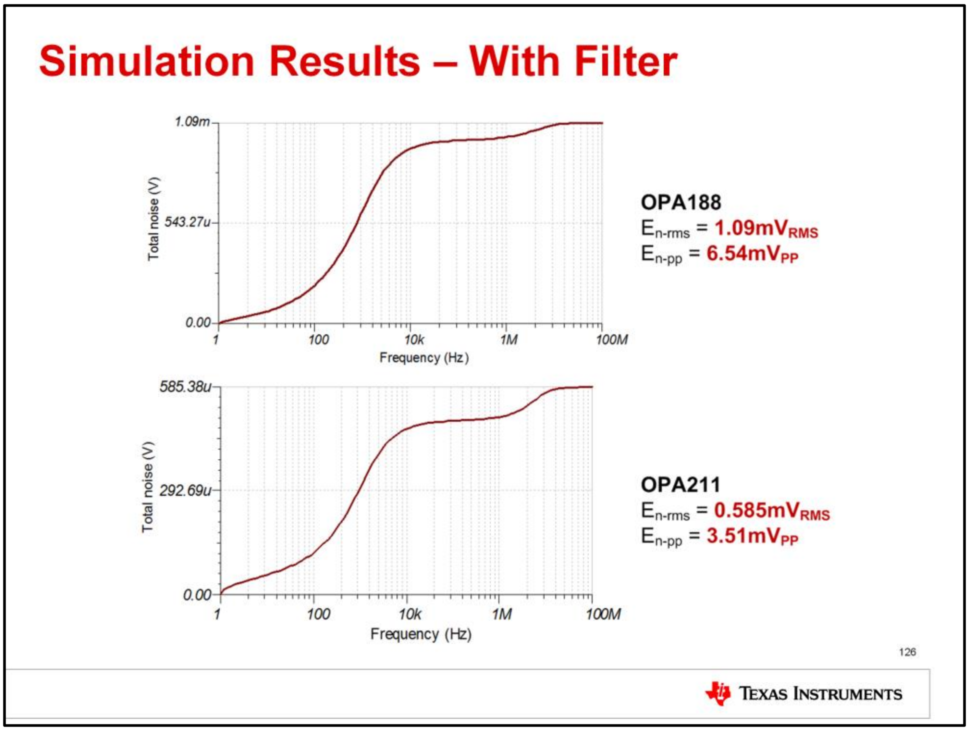
Run the simulation for both the OPA188 and OPA211!

125

Texas Instruments

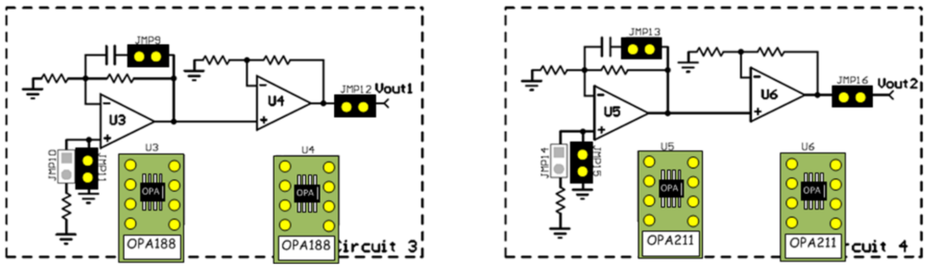
Run another SPICE simulation analysis for the total output noise. This time, ensure that the filter capacitor jumper is closed so that the capacitor is connected! As before, select Analysis, followed by Noise Analysis. Make sure that “total noise” is selected, then run the analysis from 1Hz to 100MHz.

Remember, run the simulation for both the OPA188 and OPA211!



With the filter capacitor connected, the OPA188 circuit has a total noise of 1.09mVrms, or 6.54mVpp. The OPA211 circuit has a total noise of 0.585mVrms, or 3.51mVpp.

Test Board Setup – Jumpers



The only change. Can be "hot plugged". →

Jumper, Device	Description
JMP9, JMP13	Connect filter capacitor in Circuit 3 and Circuit 4
JMP11, JMP15	Connect VIN+ to GND
JMP12, JMP16	Connect Circuit 3 to Vout1, connect Circuit 4 to Vout2
U3, U4	Install OPA188
U5, U6	Install OPA211

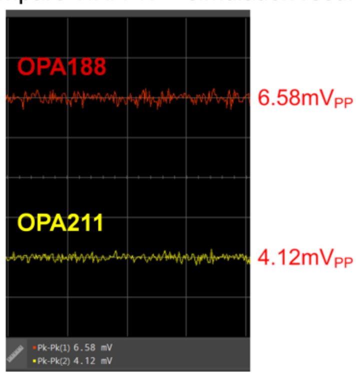
127

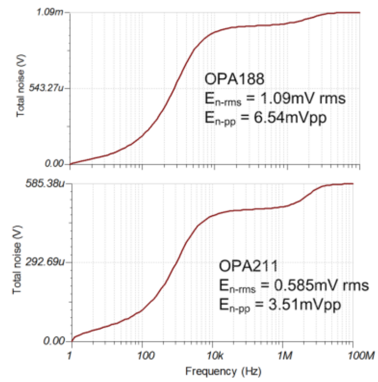
Texas Instruments

The jumper settings on the test board must be modified before re-running the bench measurement. Simply install jumpers JMP9 and JMP13. All other jumpers and devices remain the same from the previous experiment.

Measurement Results – With Filter

1. Compare TINA-TI™ simulation results to measured and calculated results.





Device	Calculated Noise (RMS)	Simulated Noise (RMS)	Measured Noise (RMS)
OPA188	$1mV_{RMS}$	$1.09mV_{RMS}$	$1.09mV_{RMS}$
OPA211	$0.48mV_{RMS}$	$0.585mV_{RMS}$	$0.69mV_{RMS}$

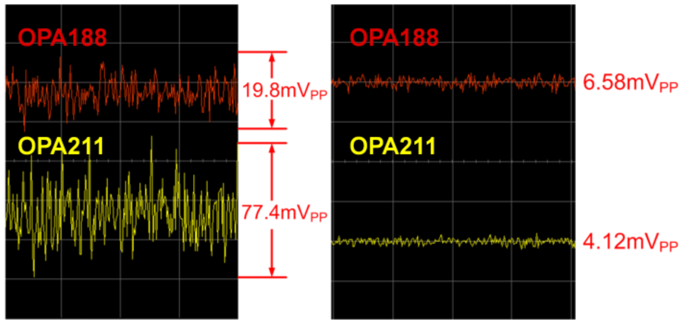
Note: Measured RMS noise is determined using measured peak-to-peak noise divided by 6.

128

Texas Instruments

The expected output voltage noise results from the measurement with the filter are shown here. The OPA188 has a measured noise of 6.58mVpp, or 1.09mVrms, and the OPA211 has a measured noise of 4.12mVpp, or 0.69mVrms. Like before, the calculated, simulated, and measured results all match quite closely. The real question is: how much benefit did we get by adding the filter capacitor?

Noise Comparison – Filter vs. No Filter



Device	Calculated (no filter)	Calculated (with filter)	Simulated (no filter)	Simulated (with filter)	Measured (no filter)	Measured (with filter)
OPA188	$3.62mV_{RMS}$	$1mV_{RMS}$	$3.59mV_{RMS}$	$1.09mV_{RMS}$	$3.3mV_{RMS}$	$1.09mV_{RMS}$
OPA211	$10.72mV_{RMS}$	$0.48mV_{RMS}$	$10.75mV_{RMS}$	$0.59mV_{RMS}$	$12.9mV_{RMS}$	$0.69mV_{RMS}$

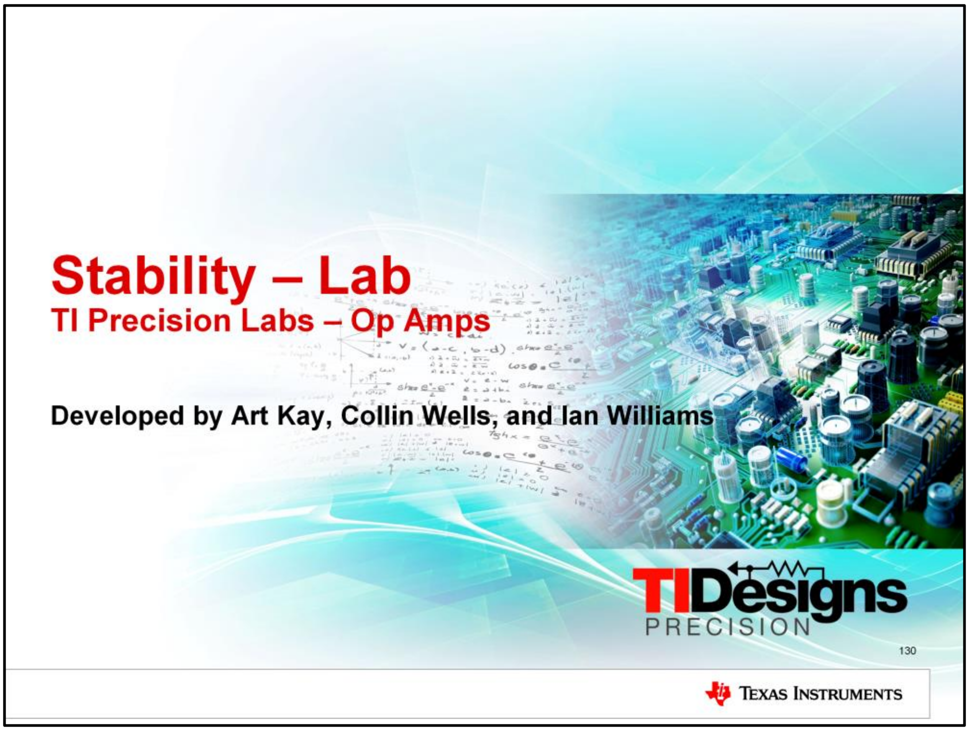
OPA188 → 3x noise improvement!
OPA211 → 19x noise improvement!

129

Texas Instruments

This slide compares the noise performance of both the OPA188 and OPA211, with and without the filter. It should be clear just from the oscilloscope capture that the filter gave us a massive improvement in noise. If you compare the peak-to-peak noise values in the table, you can see that with the filter, we achieved a 3x improvement with the OPA188 and a 19x improvement with the OPA211!

You may wonder why we didn't see more of a reduction in noise with the OPA188. One key reason is that the gain-bandwidth product of the OPA188 at 2MHz is much lower than that of the OPA211 at 80MHz, so the closed-loop bandwidth of the OPA188 circuit is lower as well. When we added the filter and reduced the closed-loop bandwidth of both circuits to 1.6kHz, this was much more of a percentage reduction for the OPA211 than the OPA188. Therefore, the filter had a bigger effect for the OPA211 circuit. The OPA188 also has higher input voltage noise spectral density than the OPA211.



Hello, and welcome to the presentation for the TI Precision Lab supplement for op amp stability. This lab will walk through detailed calculations, SPICE simulations, and real-world measurements that greatly help to reinforce the concepts established in the stability lecture.

Required/Recommended Equipment

- Calculation
  - Pencil and paper
  - **Recommended:** MathCAD, Excel, or similar
- Simulation
  - SPICE simulation software
  - **Recommended:** TINA-TI™
- Measurement
  - TI Precision Labs PCB from Texas Instruments
  - Oscilloscope
  - Function generator
  - Bode plotter
  - ±15V power supply
  - **Recommended:** National Instruments VirtualBench™

Texas Instruments 131

The detailed calculation portion of this lab can be done by hand, but calculation tools such as MathCAD or Excel can help greatly.

The simulation exercises can be performed in any SPICE simulator, since Texas Instruments provides generic SPICE models of the op amps used in this lab. However, the simulations are most conveniently done in TINA-TI, which is a free SPICE simulator available from the Texas Instruments website. TINA simulation schematics are embedded in the presentation.

Finally, the real-world measurements are made using a printed circuit board, or PCB, provided by Texas Instruments. If you have access to standard lab equipment, you can make the necessary measurements with any oscilloscope, function generator, Bode plotter, and ±15V power supply. However, we highly recommend the VirtualBench from National Instruments. The VirtualBench is an all-in-one test equipment solution which connects to a computer over USB or Wi-Fi and provides power supply rails, analog signal generator and oscilloscope channels, and a 5 ½ digit multimeter for convenient and accurate measurements. This lab is optimized for use with the VirtualBench.

Experiment 1
Buffer with Cap Load

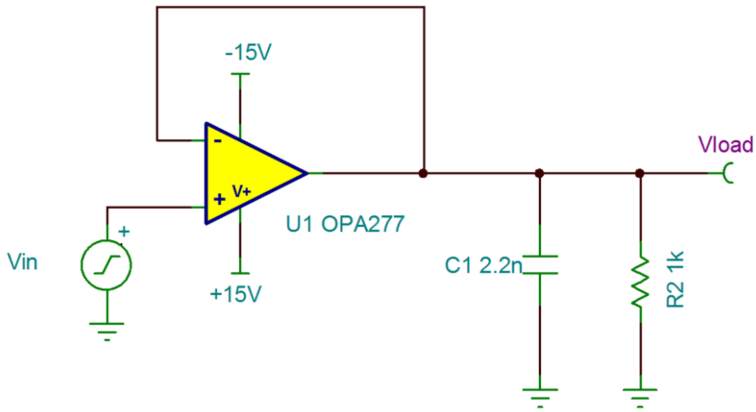
132

Texas Instruments

In experiment 1, we'll determine the phase margin, and therefore the stability, of a buffer circuit which is being used to drive a large capacitive load. We'll determine the phase margin by observing the transient overshoot as well as the AC transfer characteristic.

Calculation – Capacitive Load

Calculate the phase margin and percentage overshoot for the circuit shown below.

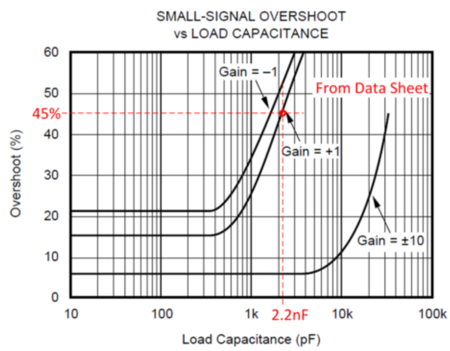


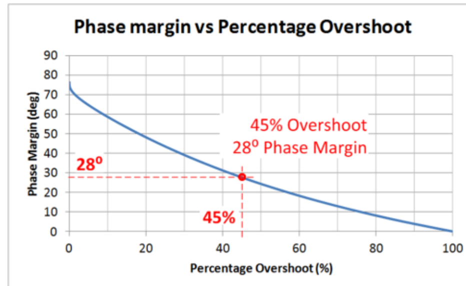
133

TEXAS INSTRUMENTS

First, calculate the phase margin and percentage overshoot for the circuit shown here, using the techniques and equations given in the stability lecture. Use the plots given on the next slide.

Calculation – Capacitive Load





Answers	OPA277
Percent overshoot	45%
Phase margin	28°

134

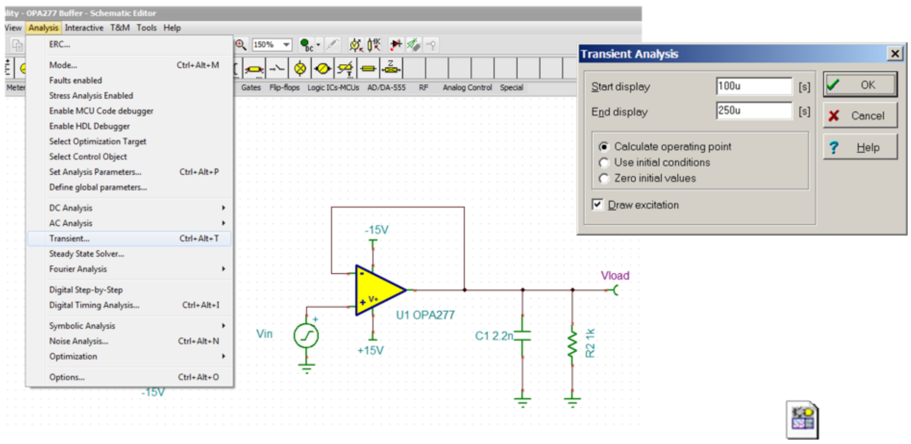
Texas Instruments

This circuit uses the OPA277. In order to perform the calculations, you need to know the percent overshoot versus load capacitance for that device, shown on the top left. Then, use the plot on the right to determine the phase margin from that percentage overshoot.

Enter your answers in the table at the bottom of the slide. The solutions are already provided to allow you to check your work.

Simulation Setup – Overshoot

Click **Analysis → Transient** to run a transient simulation showing overshoot.
Run the analysis from 100µs to 250µs. The input is a 20mVpk, 10kHz square wave.



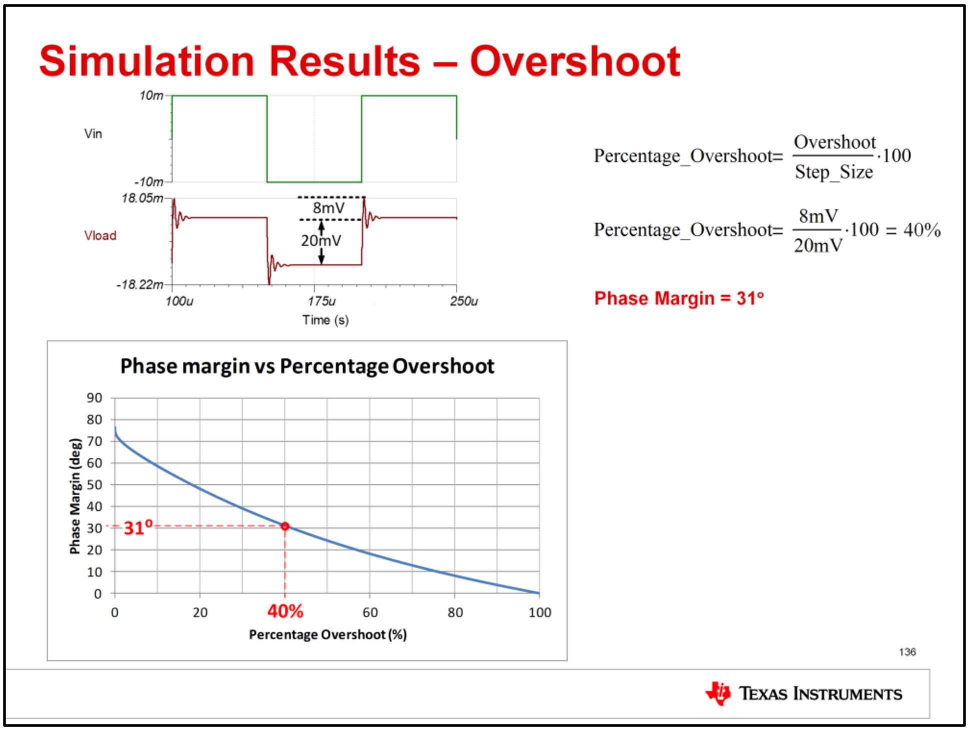
06 - Stability - OPA277 Buffer.TSC

135

Texas Instruments

The next step is to run a SPICE simulation analysis for the transient overshoot.

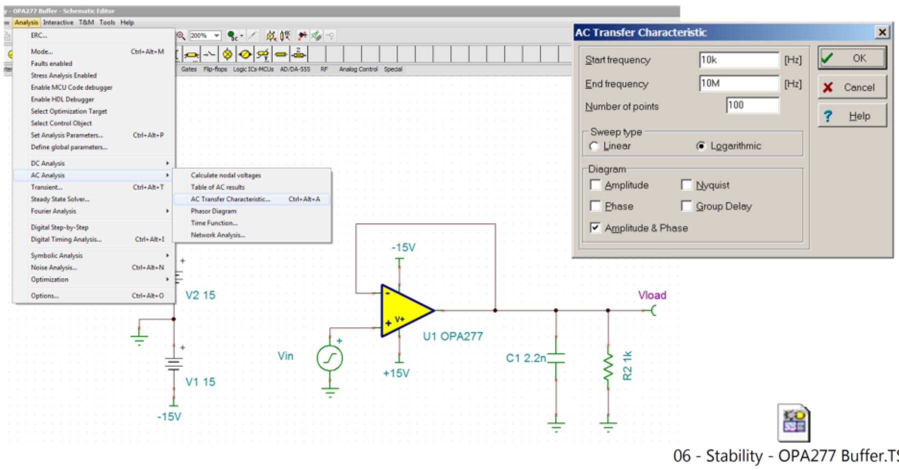
The necessary TINA-TI simulation schematic is embedded in this slide set – simply double-click the icon to open it. To run the analysis, click Analysis → Transient, and run the analysis from 0ms to 150us. The input is a 20mVpk, 10kHz square wave.



You should see a result similar to this. Use the simulated percentage overshoot of 55% to calculate the phase margin, which comes out to 21 degrees.

Simulation Setup – Bode Plot

Click **Analysis → AC Analysis → AC Transfer Characteristic** to generate the bode plot. Run the AC analysis from 10kHz to 10MHz.



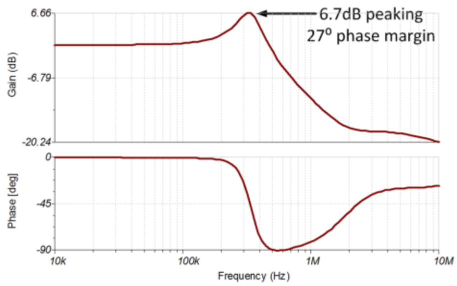
137

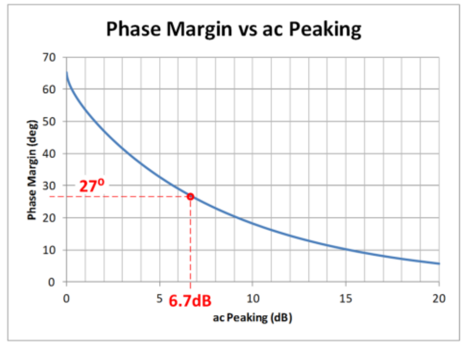
TEXAS INSTRUMENTS

Next, run a SPICE simulation analysis for the AC transfer characteristic. This will allow us to see the op amp's AC peaking, which is another indicator of phase margin.

Use the same TINA-TI simulation schematic as before. To run the analysis, click Analysis → AC Analysis → AC Transfer Characteristic. Run the analysis from 10kHz to 10MHz.

Simulation Results – Bode Plot





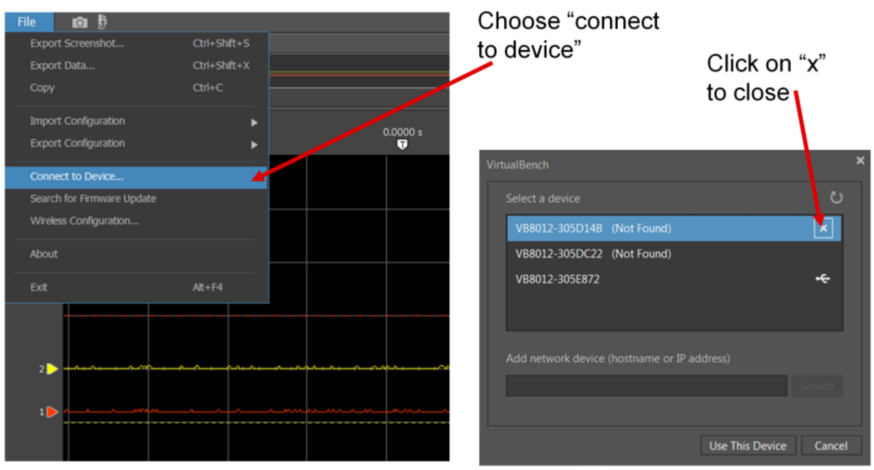
Phase Margin = 27°

138

TEXAS INSTRUMENTS

You should see a result similar to this. The 9dB of simulated AC peaking results in a phase margin of approximately 20 degrees.

Make All "Not Found" devices are closed

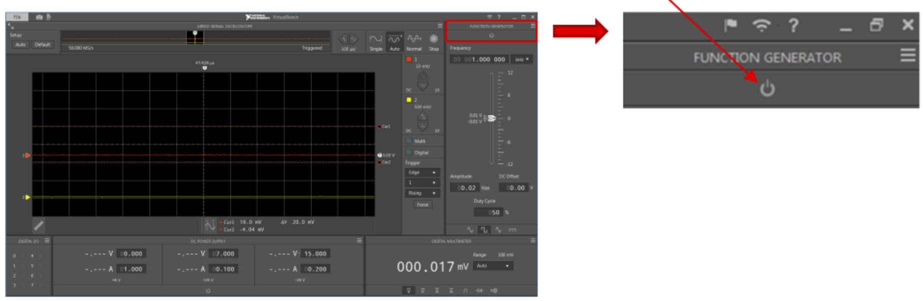


Texas Instruments

We will be using a new software package. To prevent problems, make sure that all "not found" devices are closed.

Disable Function Generator

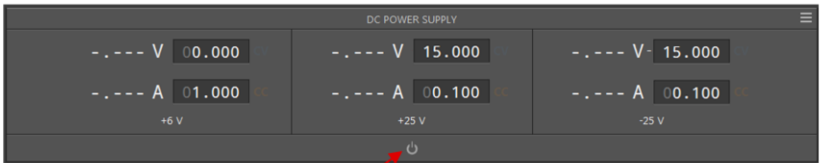
Power button GRAY =
Function Generator OFF



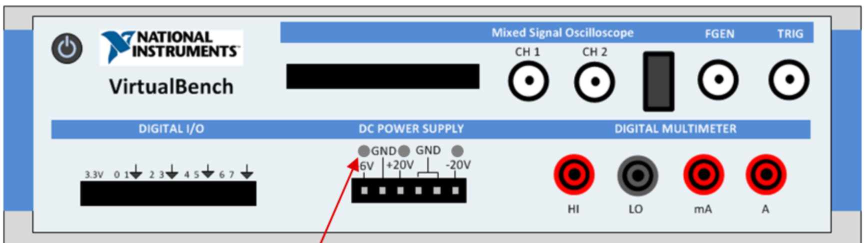
TEXAS INSTRUMENTS

Also make sure the Function Generator is OFF.

Disable DC Power Supply

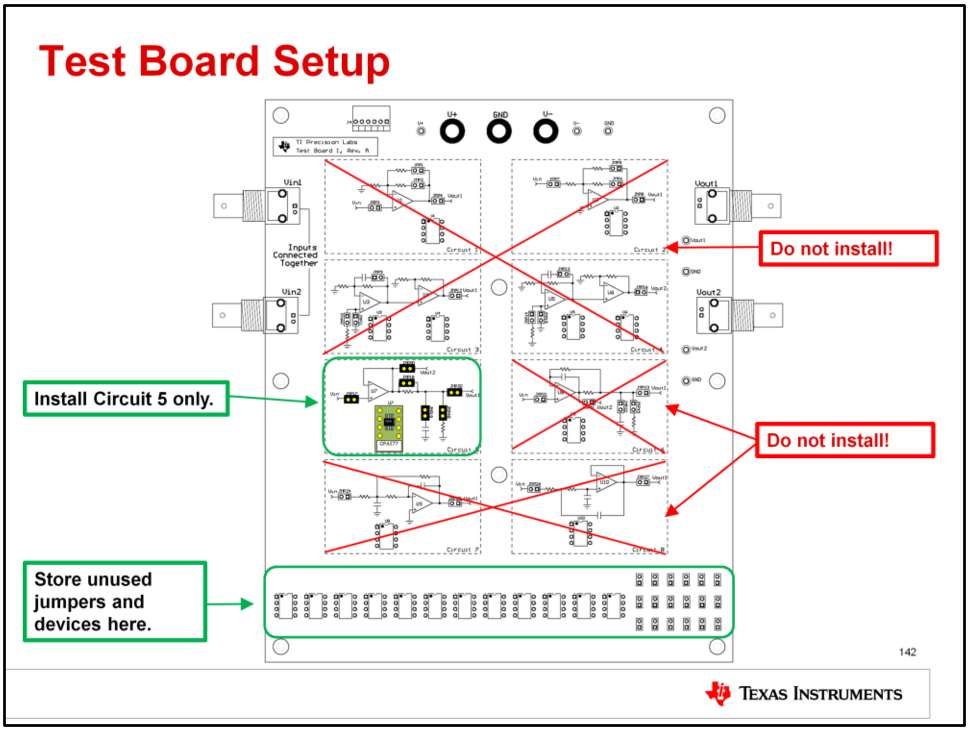


Power button GRAY = DC power supply OFF



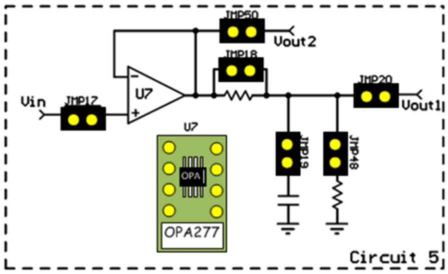
LEDs OFF = DC power supply OFF

Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuit 5. Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

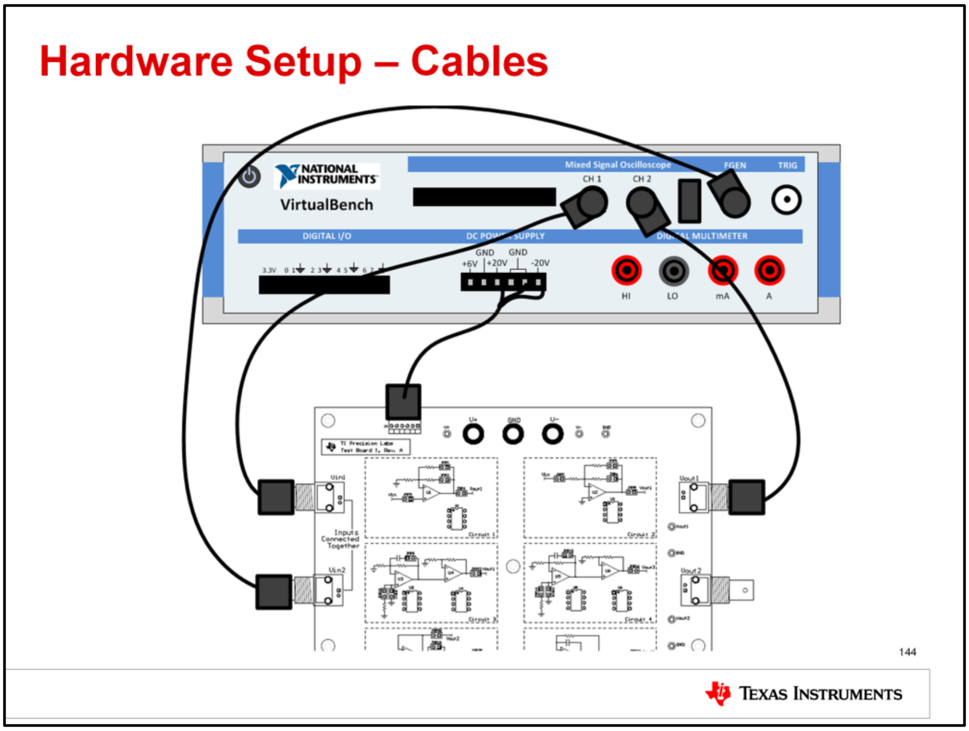
Test Board Setup – Jumpers



Jumper, Device	Description
JMP17	Connects input to U7
JMP18	Shorts isolation resistor. Riso = 0Ω
JMP19	Connects 2.2nF load capacitance
JMP20	Monitor output across load
JMP48	Connects 1kΩ load resistance
JMP50	Monitor directly at U7 output
U7	Install OPA277

To prepare the test board for the measurement, install the jumpers and devices on circuit 5 as shown here.

Install JMP17, JMP18, JMP19, JMP20, JMP48, and JMP50, as well as the OPA277 in socket U7.

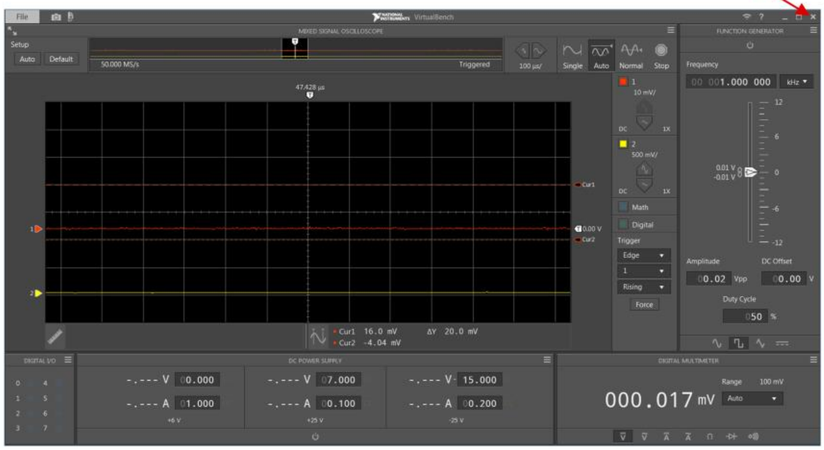


This slide gives the connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board.

Connect Vin1 on the test board to VirtualBench oscilloscope channel 1, and Vin2 on the test board to the VirtualBench FGEN. Connect Vout1 on the test board to VirtualBench oscilloscope channel 2.

Close VirtualBench Software

Click the '**X**' to close



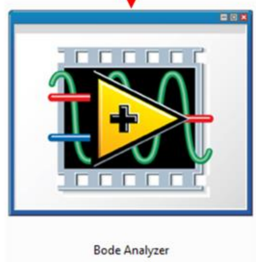
145

TEXAS INSTRUMENTS

The VirtualBench software must be closed before continuing with the lab. Click the 'X' in the top-right corner of the software to close.

Open Bode Analyzer Software

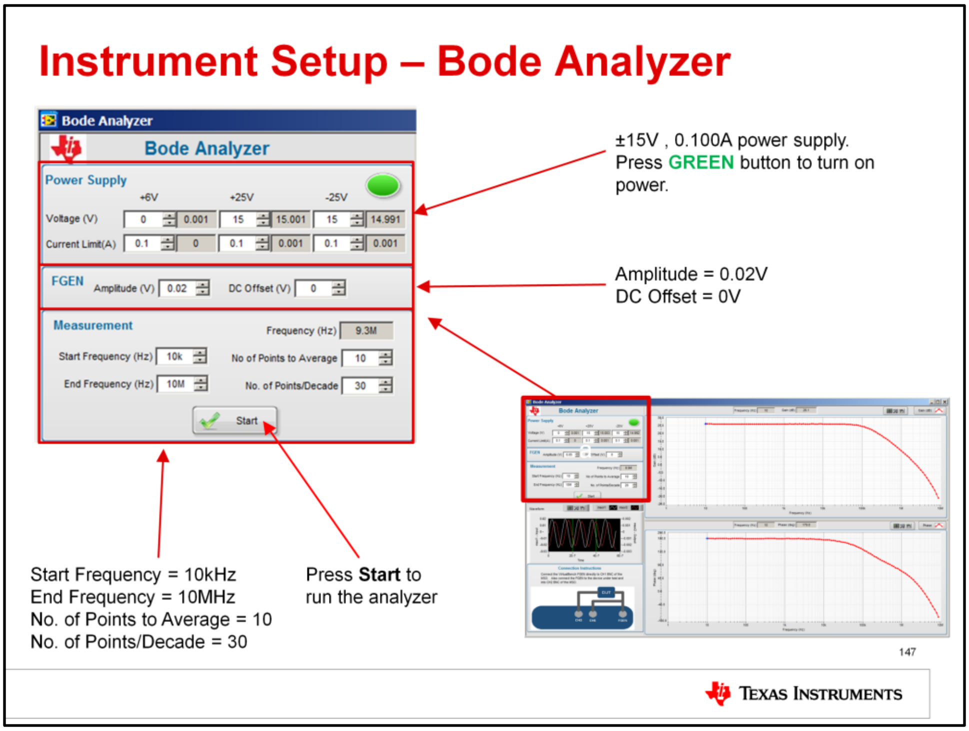
Double-click icon on desktop to start **Bode Analyzer** software.



146

Texas Instruments

This lab requires additional Bode analyzer software. Install the software, then run it by double-clicking the Bode Analyzer icon on the desktop. You may also run the software by clicking Start → All Programs → Bode Analyzer → Bode Analyzer.

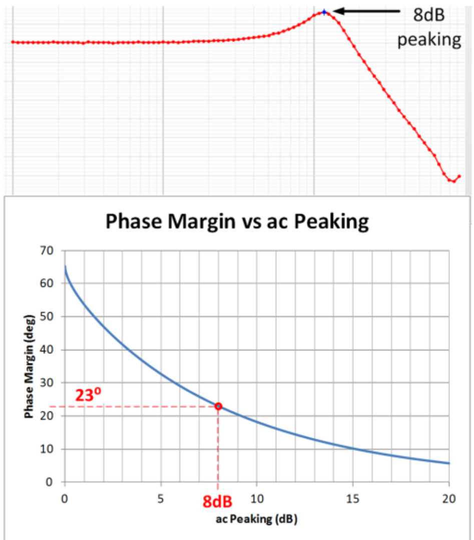


In the configuration panel, set the power supply to ±15V, 0.1A. Press the green button to turn on the power.

Set the FGEN amplitude to 0.02V, and DC offset to 0V.

Set the start frequency to 10kHz and the end frequency to 10MHz. Set the number of points to average to 10, and the number of points per decade to 30. Press “Start” to run the Bode analyer.

Measured AC Peaking – Bode Analyzer

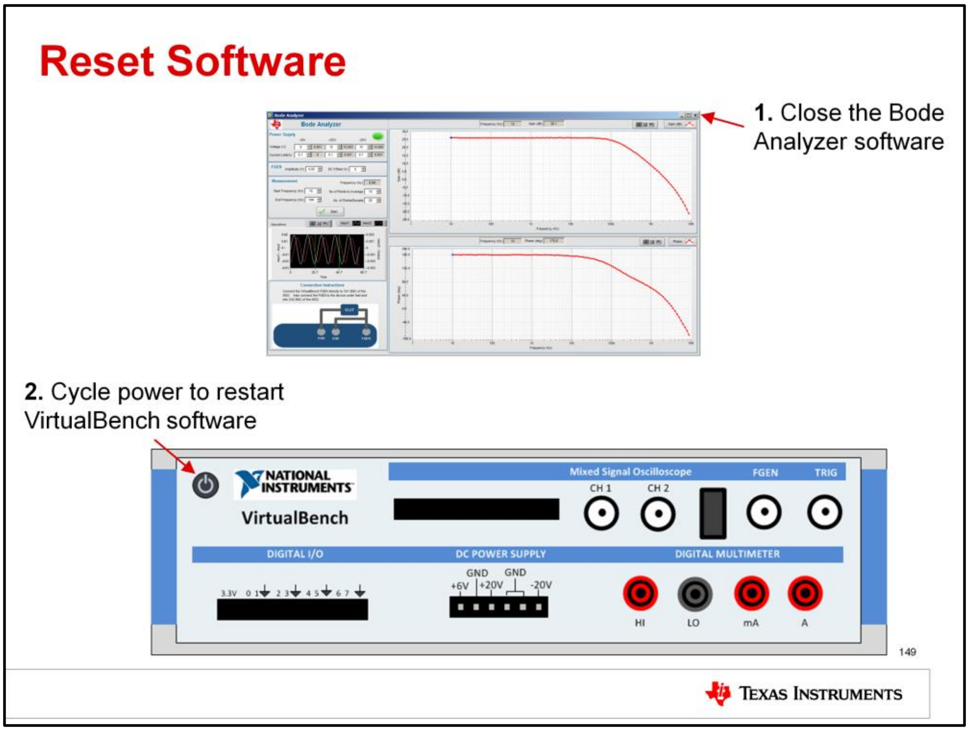


Answers	OPA277
AC Peaking	8dB
Phase margin	23°

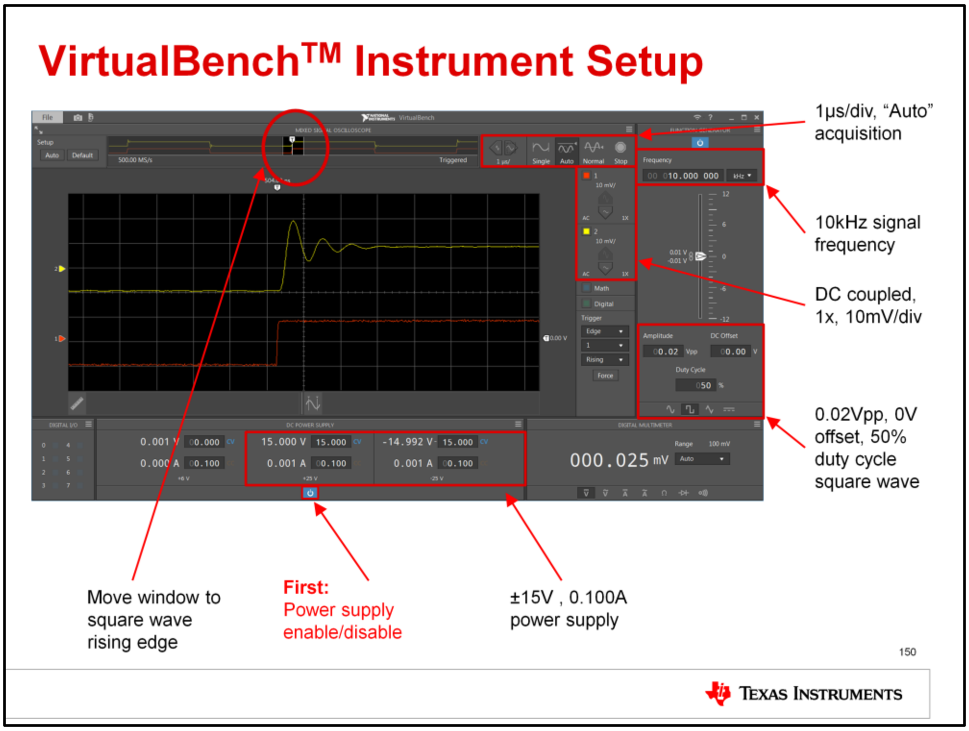
148

Texas Instruments

You should see a result similar to this. Enable the cursor, then drag the cursor to the maximum value to measure the AC peaking. In this measurement, AC peaking of 8dB resulted in a phase margin of 23 degrees. Your results may vary slightly.



The bandwidth lab is now complete. Before continuing to the next lab, you must reset the software. First, close the Bode Analyzer software by clicking the 'X' in the top-right corner. Next, cycle power on the VirtualBench by turning power off and on. This will restart the VirtualBench software.



Next apply power to the National Instruments VirtualBench and connect it to your computer with a USB cable. The hardware should be detected as a virtual CD drive, and you can run the VirtualBench software directly from the drive. Once the software opens, configure the software as follows:

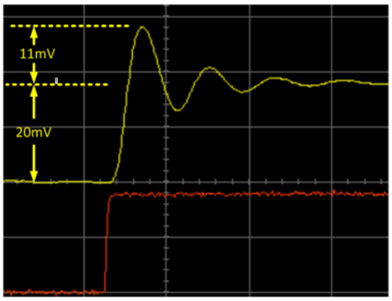
Set the time scale to 1us per division, with the acquisition mode set to “Auto.” Enable channels 1 and 2 on the oscilloscope, and set them to 1x, DC-coupled mode, 10mV/div. Enable the function generator and setup the signal as follows:

10kHz frequency, 20mVpp, 0V offset, 50% duty cycle square wave.

Set the +25V power supply to +15V, 0.100A. Set the -25V power supply to -15V, 0.100A. Press the power button to turn on the power supply rails.

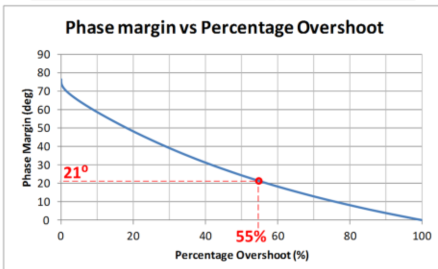
Move the display window to the rising edge of the square wave. This allows you to observe the overshoot and ringing of the op amp output.

Measurement Results – Overshoot



$$\text{Percentage\_Overshoot} = \frac{\text{Overshoot}}{\text{Step\_Size}} \cdot 100$$

$$\text{Percentage\_Overshoot} = \frac{11\text{mV}}{20\text{mV}} \cdot 100 = 55\%$$



Answers	OPA277
Percent overshoot	55%
Phase margin	21°

151

Use cursors to measure the amount of overshoot. The expected measurement results are shown here. The measured overshoot of 55% results in a phase margin of 21 degrees. Your results may vary slightly.

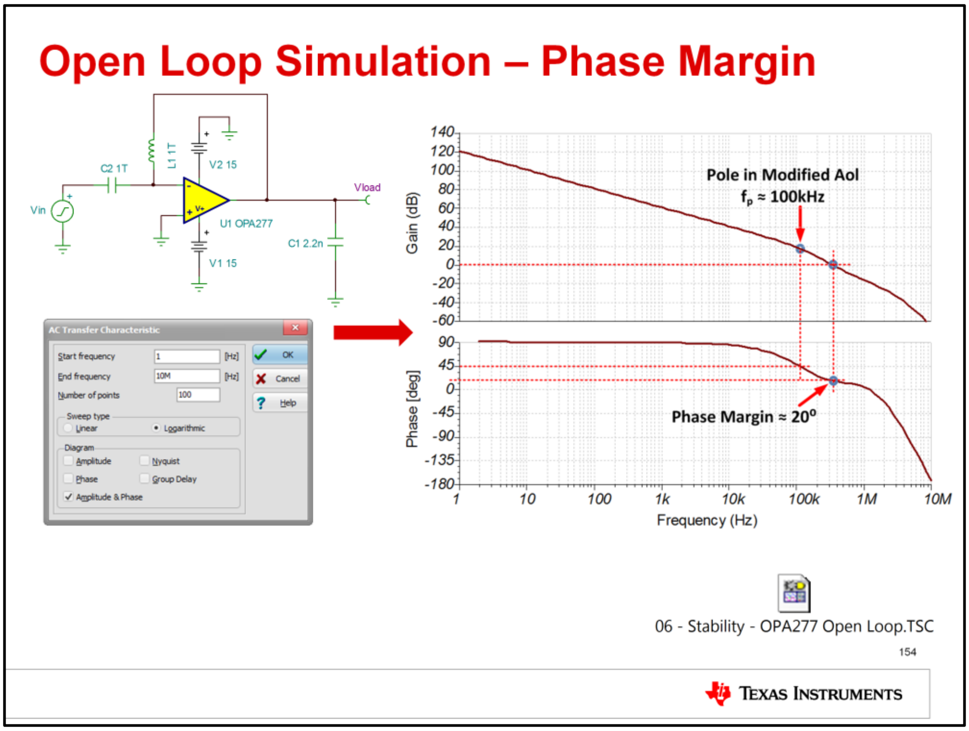
Results Comparison

1. Compare the phase margin results from calculations, transient and AC response simulations, and transient and AC response measurements.

Test	Phase Margin
Calculated from data sheet	28°
Simulated from transient overshoot	31°
Simulated from AC peaking	27°
Measured from transient overshoot	21°
Measured from AC peaking	23°

Now, compare the phase margin results from your hand calculations, transient and AC response simulations, and transient and AC response measurements. While there is some slight variation to the results, the phase margin values compare very well at approximately 23 degrees.

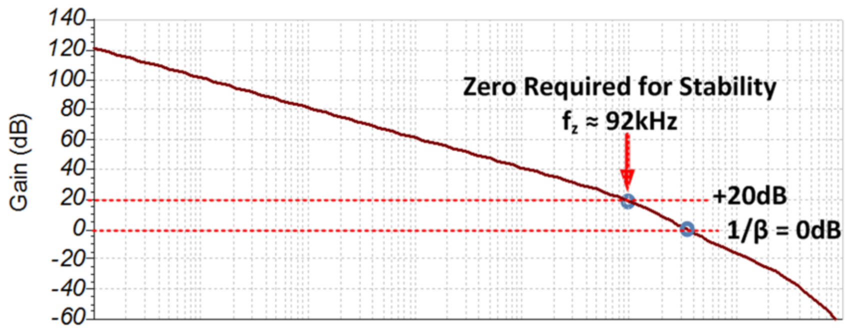
In experiment 2, we'll use an isolation resistor to increase the phase margin of the circuit from Experiment 1 and therefore improve its stability.



To determine the value of isolation resistor Riso, we must first know the open-loop AC response of the circuit. Here we show the TINA-TI simulation schematic and AC response results, which you can verify using the embedded file.

To measure the phase margin, find the frequency where the gain measures 0dB. Then, measure the phase at that same frequency, which in this example is 20 degrees.

Calculation – R_{ISO}



$$R_{iso} = \frac{1}{2\cdot\pi\cdot f_z\cdot C_L} = \frac{1}{2\cdot\pi\cdot 92\text{kHz}\cdot 2.2\text{nF}} = 786\ \Omega$$

R_{ISO} = 787 Ω (nearest standard value)

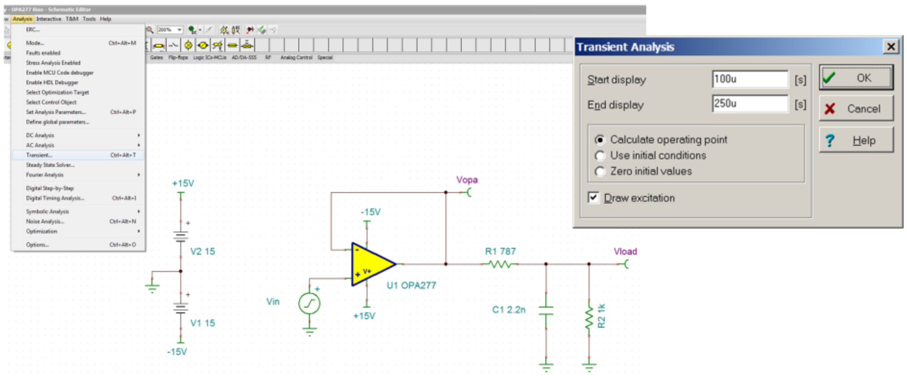
155

TEXAS INSTRUMENTS

For best results, Riso should create a zero in the loop response 20dB greater than the frequency where the open-loop gain intersects with the closed-loop gain. This circuit is a buffer with a closed-loop gain of 0dB, so the zero should occur at 20dB. The open-loop gain is equal to 20dB at 92kHz. Use this value and the load capacitance to calculate Riso, which is 786 ohms in this example. 787 ohms is the nearest standard value resistance.

Simulation Setup – R_{ISO} Overshoot

Click **Analysis → Transient** to run a transient simulation, showing overshoot.
Run the analysis from 100µs to 250µs. The input is a 20mVpk, 10kHz square wave.



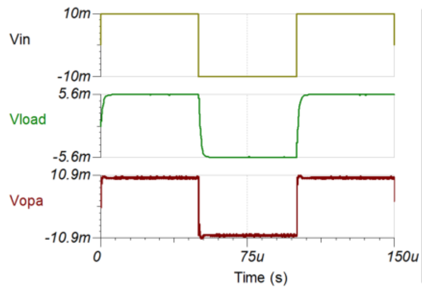
06 - Stability - OPA277 Riso.TSC

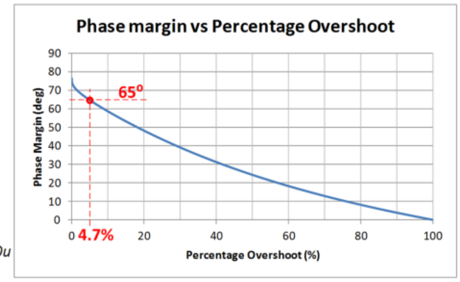
156

Texas Instruments

Let’s now simulate the transient overshoot with Riso included in the circuit. As before, click Analysis → Transient to run the simulation. Run the analysis from 0ms to 150us. The input is a 20mVpk, 10kHz square wave.

Simulation Results – R_{ISO} Overshoot





$$\text{Percentage\_Overshoot} = \frac{\text{Overshoot}}{\text{Step\_Size}} \cdot 100$$

$$\text{Percentage\_Overshoot} = \frac{0.93\text{mV}}{20\text{mV}} \cdot 100 = 4.7\%$$

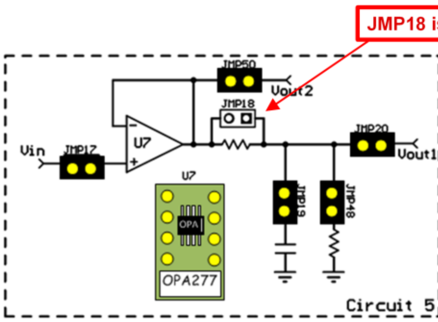
Phase Margin = 65°

157

TEXAS INSTRUMENTS

You should see results similar to this. With Riso added to the circuit, the overshoot was reduced to only 4.7%. This results in a phase margin of 65 degrees, indicating a stable circuit.

Test Board Setup – Jumpers

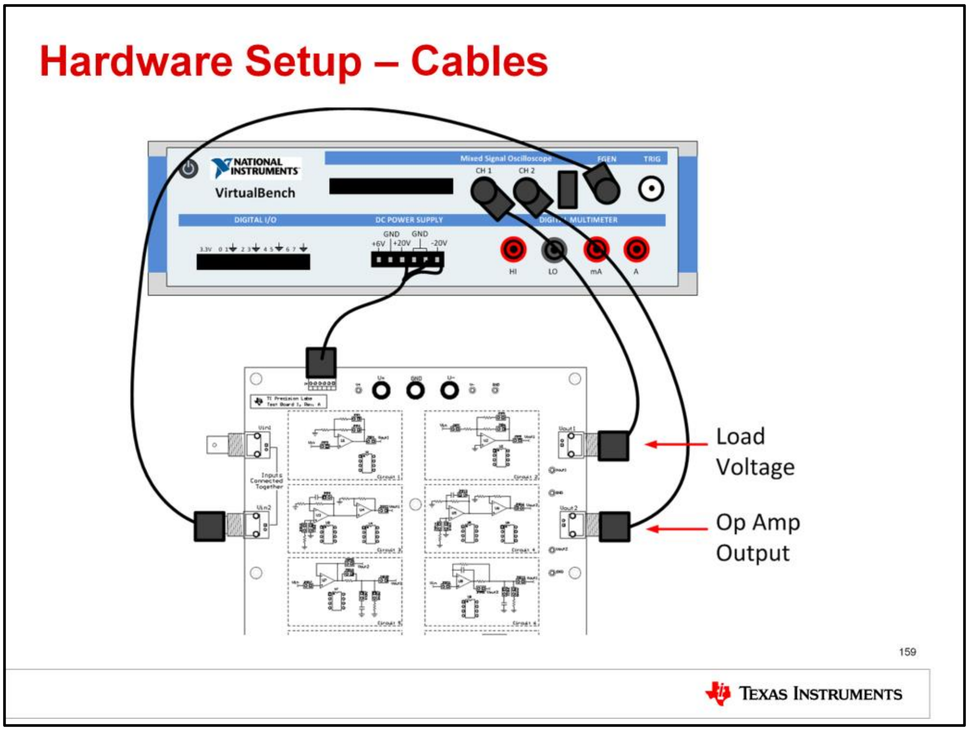


Jumper, Device	Description
JMP17	Connects input to U7
JMP18	**Removed to connect isolation resistor Riso = 787Ω**
JMP19	Connects 2.2nF load capacitance
JMP20	Monitor output across load
JMP48	Connects 1kΩ load capacitance
JMP50	Monitor directly at U7 output
U7	Install OPA277

158

TEXAS INSTRUMENTS

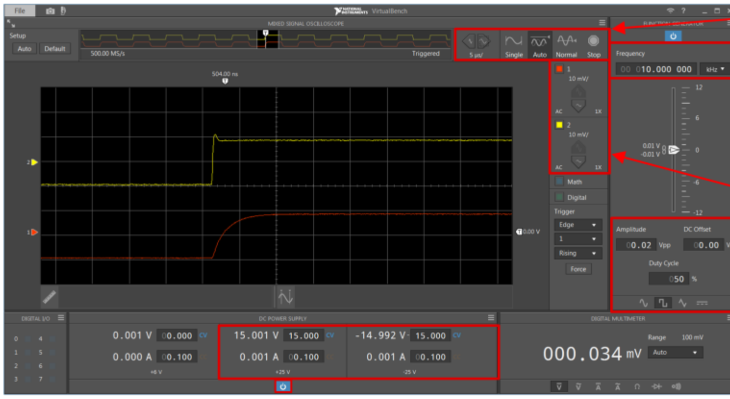
Let's now re-run the overshoot measurement, this time with Riso. The jumper setup is almost the same. The only change is to remove JMP18, which connects the isolation resistor of 787 ohms.



This slide gives the new connection diagram between the TI Precision Labs test board and the National Instruments VirtualBench. Connect the provided power cable to the DC power supply of the Virtual Bench and power connector J4 on the test board.

Connect Vin2 on the test board to the VirtualBench FGEN. Connect Vout1 on the test board to VirtualBench oscilloscope channel 1 to measure the load voltage, and connect Vout2 on the test board to VirtualBench oscilloscope channel 2 to measure the unloaded op amp output voltage.

VirtualBench^{TM} Instrument Setup



5µs/div, "Auto" acquisition

10kHz signal frequency

DC coupled, 1x, 10mV/div

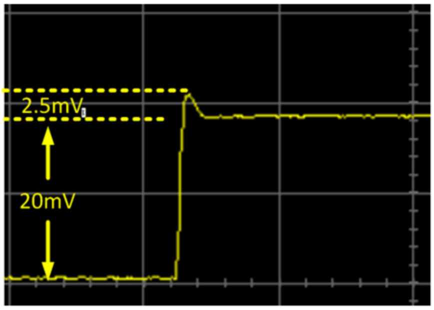
0.02Vpp, 0V offset, 50% duty cycle square wave

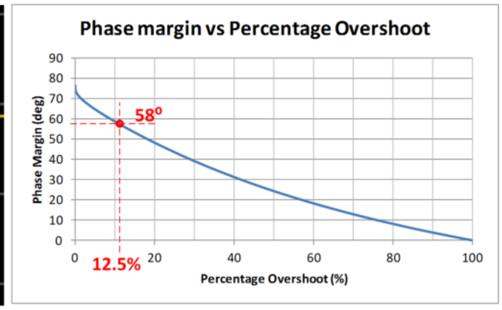
160

Texas Instruments

The VirtualBench setup is almost the same as before. Only change the time scale to 5us/div. All other settings must remain the same.

Measurement Results – R_{ISO} Overshoot





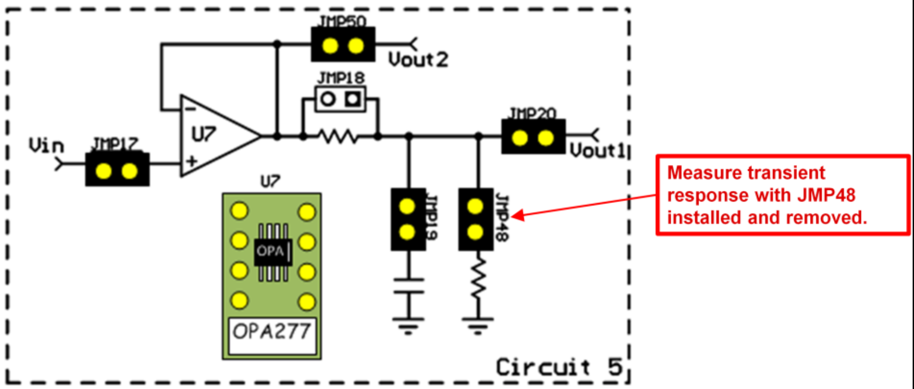
$$\text{Percentage\_Overshoot} = \frac{\text{Overshoot}}{\text{Step\_Size}} \cdot 100$$

$$\text{Percentage\_Overshoot} = \frac{2.5\text{mV}}{20\text{mV}} \cdot 100 = 12.5\%$$

Phase Margin = 58°

Use cursors to measure the amount of overshoot. You should see results similar to this. With Riso, the measured unloaded output overshoot was reduced to 12.5%, resulting in a phase margin of approximately 58 degrees which indicates a stable circuit.

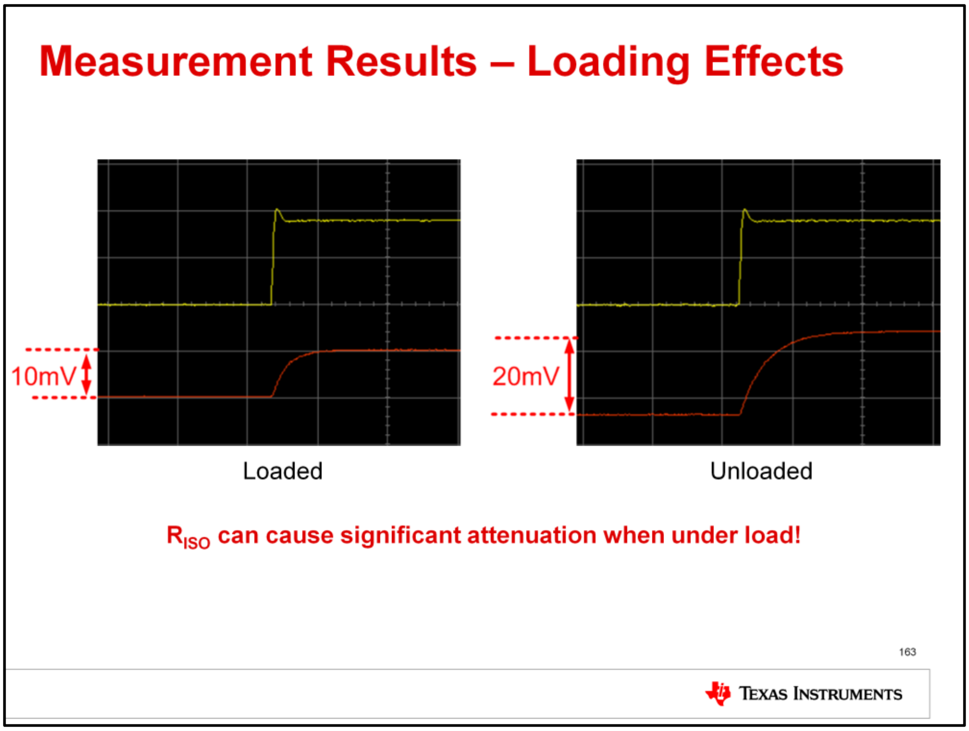
Measurement Setup – Loading Effects



162

TEXAS INSTRUMENTS

As an additional experiment, measure the transient response again with JMP48 installed and removed. This will connect and disconnect the 1k load resistor from the circuit.



As you can see, the load voltage changes dramatically if the circuit is loaded or unloaded! In fact, when loaded, the circuit with Riso shows attenuation of approximately 50%! This is simply due to the voltage divider effect of Riso and the load resistance.

Experiment 3
Dual Feedback

164

Texas Instruments

For the final experiment, we'll analyze a circuit which uses Riso as well as a dual feedback network to achieve stability as well as output voltage accuracy.

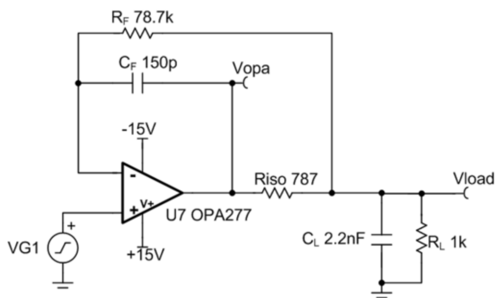
Calculation – Dual Feedback

$R_F \geq 100 \times R_{iso} = 100 \times 787\Omega = 78.7k\Omega$ **Choose R_F**

$$\frac{5 \times R_{iso} \times C_L}{R_F} \leq C_F \leq \frac{10 \times R_{iso} \times C_L}{R_F}$$ **Find the range of C_F**

$110pF \leq C_F \leq 220pF$

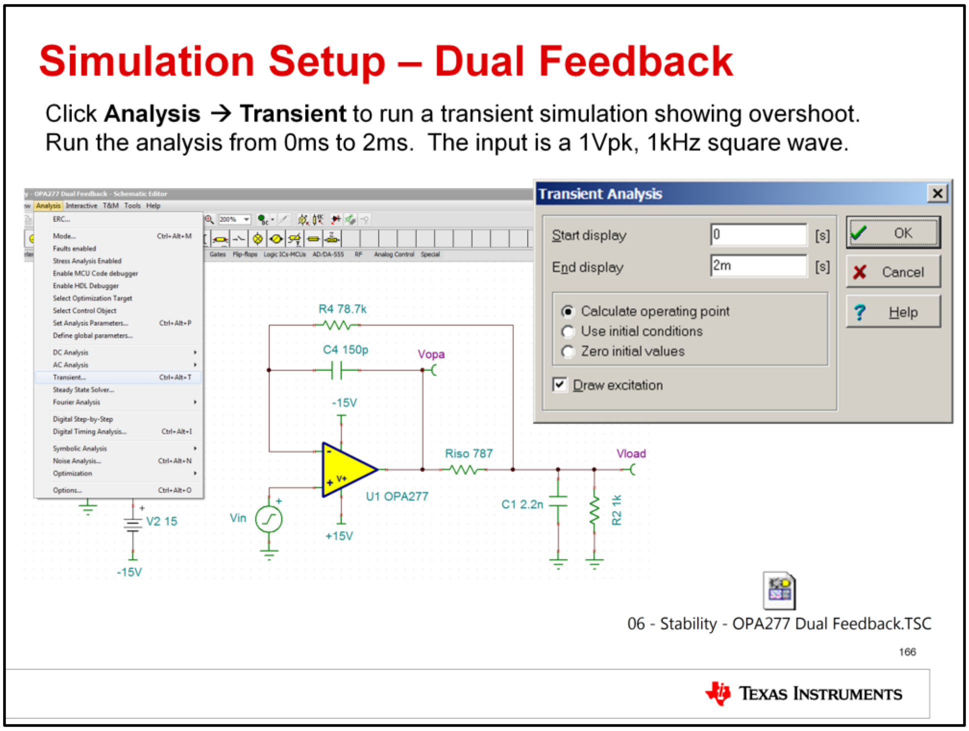
$C_F = 150pF$ **Select standard value of C_F**



165

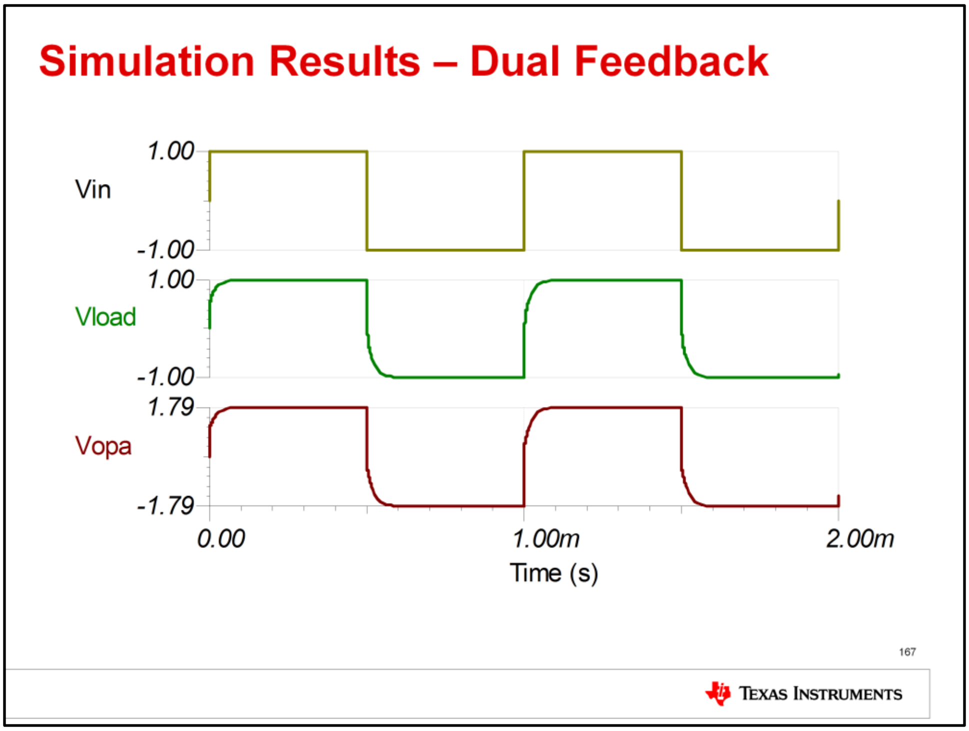
TEXAS INSTRUMENTS

To select the components of the dual feedback network, use the equations given on this slide. For this example, Rf was chosen to be 78.7k, and Cf was chosen to be 150pF.



Next, run a SPICE simulation analysis for the transient overshoot with dual feedback.

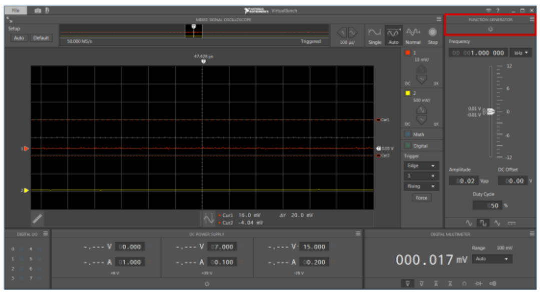
The necessary TINA-TI simulation schematic is embedded in this slide set – simply double-click the icon to open it. To run the analysis, click Analysis → Transient, and run the analysis from 0ms to 2ms. The input is a 1Vpk, 1kHz square wave.

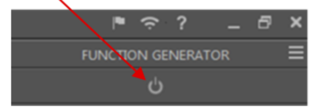


You should see results similar to this. As you can observe, in this configuration the loaded output voltage matches very well with the input. To achieve this, the unloaded output voltage Vopa must increase to compensate for the attenuation caused by Riso and the load resistance.

Disable Function Generator

Power button GRAY =
Function Generator OFF

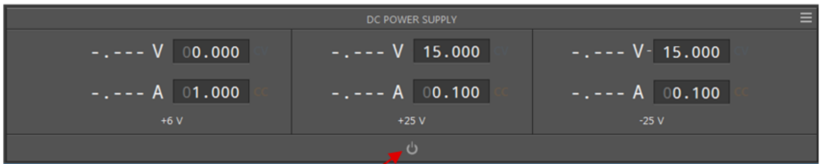




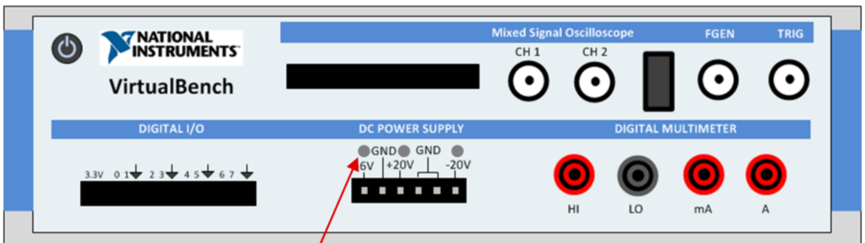
Texas Instruments

Also make sure the Function Generator is OFF.

Disable DC Power Supply



Power button GRAY = DC power supply OFF

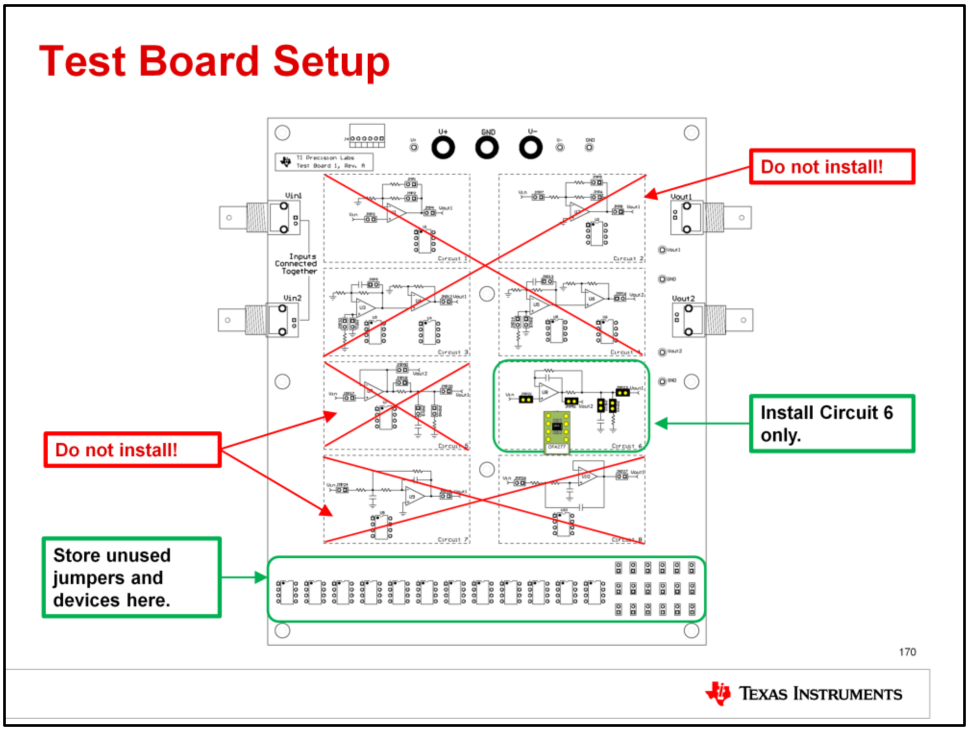


LEDs OFF = DC power supply OFF

169

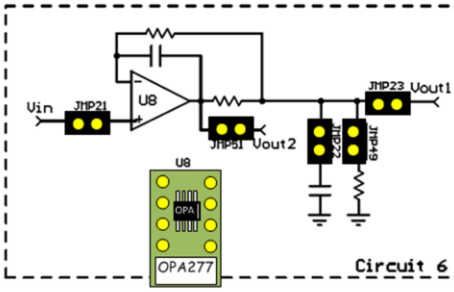
Texas Instruments

Make sure to disable the DC power supply before setting up the test PCB! In the VirtualBench software, click the power button in the DC Power Supply area to turn off the power. Check the front panel of the VirtualBench unit to make sure the LEDs are OFF!



For the test board to function properly, it is important that you only install jumpers and devices in circuit 6! Do not install any jumpers or devices in any other circuits on the PCB! Remove any jumpers or devices from the unused circuits and store them in the storage area at the bottom of the test board.

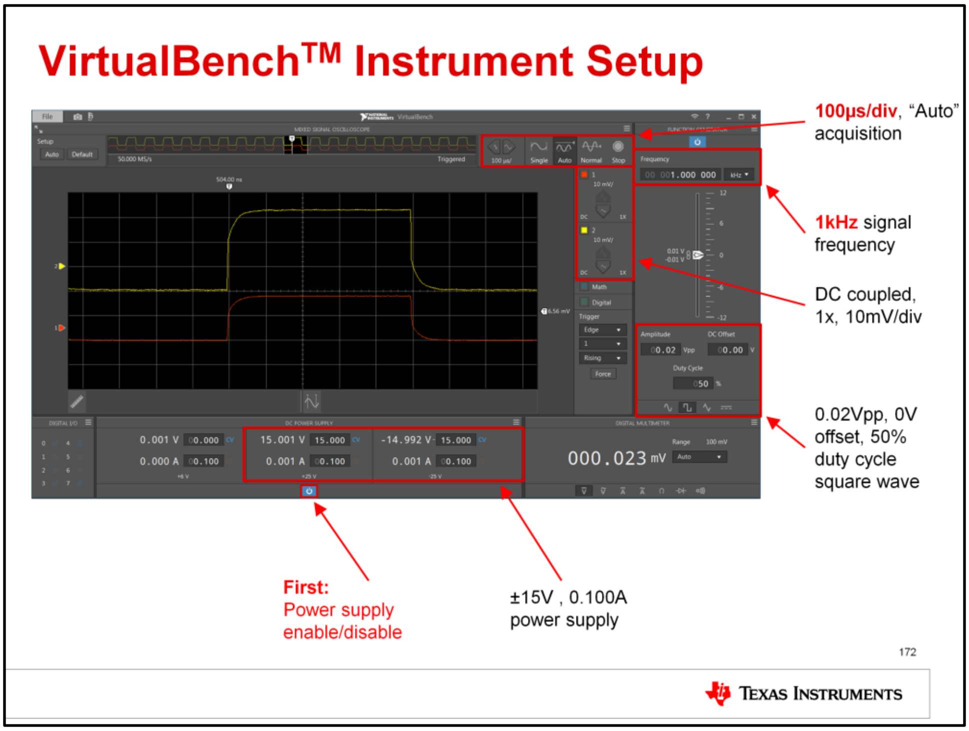
Test Board Setup – Jumpers



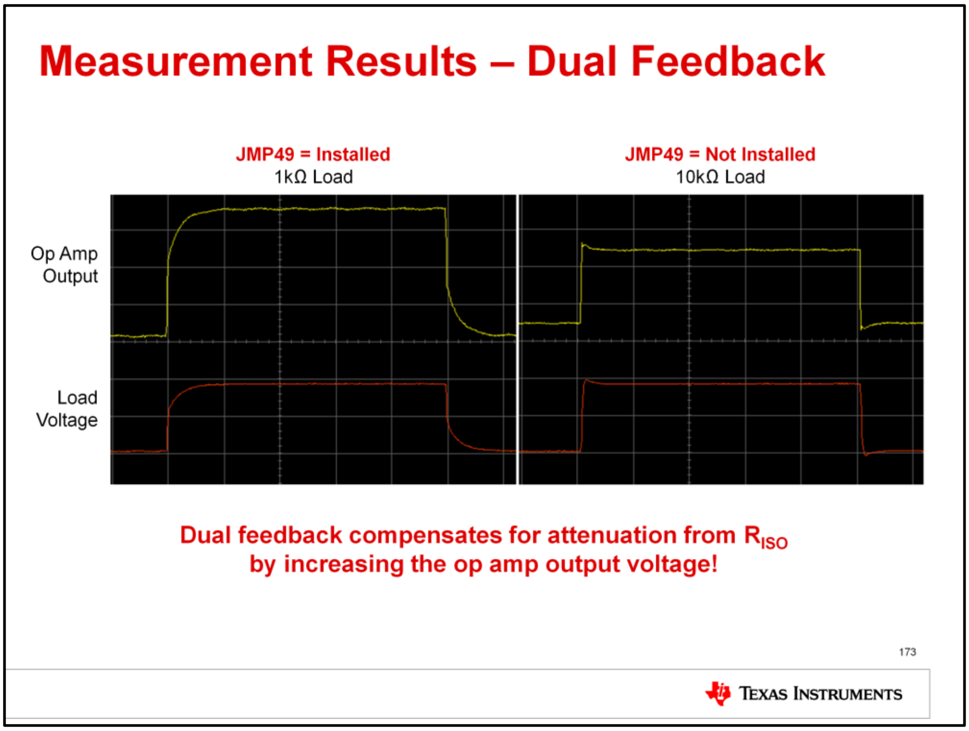
Jumper, Device	Description
JMP21	Connects input to U8
JMP22	Connects 2.2nF load capacitance
JMP23	Monitor across the load RC.
JMP49	Connects 1kΩ load resistance
JMP51	Monitor directly at U8 output.
U8	Install OPA277

To prepare the test board for the measurement, install the jumpers and devices on circuit 6 as shown here.

Install JMP21, JMP22, JMP23, JMP49, and JMP51. Install the OPA277 in socket U8.



The VirtualBench setup is almost the same as before. Only change the time scale to 100us/div. All other settings must remain the same.



You should see results similar to this. As you can see, the load voltage remains accurate, even with changing load, due to the compensation provided by the dual-feedback network. The unloaded op-amp output voltage must increase as the load resistance decreases to minimize the output voltage divider effect.

That concludes the lab – thank you for your time!

Updates

- Changed order of stability lab. Do body plot first to minimize resetting of equipment
- Eliminated redundant hardware setup in noise part 2
- Added animation to IO limit, BW, and Slew Rate hand calculation area
- Corrected slew rate calculation to be 0.8V/us
- Corrected amplifier name in slew rate calculation

Texas Instruments

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as "components") are sold subject to TI's terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI's terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers' products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers' products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI's goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or "enhanced plastic" are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have ***not*** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265