Lab Reference Manual

ECEN 326 – Electronic Circuits

Texas A&M University
Department of Electrical and Computer Engineering

Contents

1. Circuit Analysis in PSpice

1.1 Transient and DC Analysis in PSpice

Start by placing the circuit components in the schematic screen. Bring up the part browser window as shown in figure 1 by going to "*Draw → Get New Part...*", by clicking on the button with the binoculars next to the parts list drop down menu, or by clicking *Ctrl+G*. See figure 1.1.
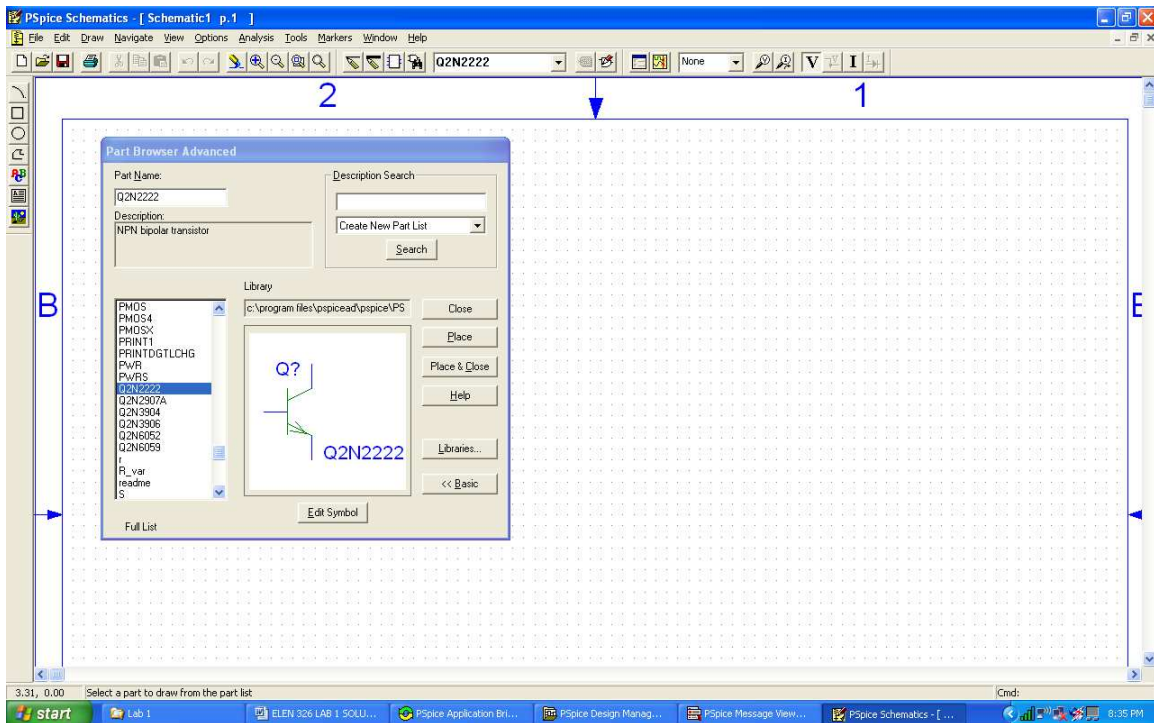

Figure 1.1: PSpice showing the Parts Browser window.

Select the appropriate part from the list on the left side of the window and click the "*Place*" button. Left click to place the part and right click to de-select. To change the values of a part, double click the part to bring up a dialog box. After editing a value, click the "*Save Attr*" button to save the entered value. (Be sure to place a ground node so that PSpice will have a reference voltage).

Use the shortcut key *Ctrl+R* to rotate a part counterclockwise and *Ctrl+F* to flip the part to its mirror image. To wire components together, click on the thin pencil button to draw. To name a node, double click the wire to bring up a dialog box. Name the signal input voltage "*vin*" and the output voltage "*vout*".

For input stimulus, use the component called "VSIN". Double click the component to bring up the attributes. Set DC to 0, VOFF to 0, and AC to 1 (this allows for frequency

3

domain analysis). VAMPL is the amplitude of the time domain input waveform, and FREQ is the operating frequency.

To set up the analysis, go to "*Analysis → Setup…*" or click on the blue and grey setup button to bring up the Analysis Setup window. Check the transient box to enable analysis in the time domain. Click on the "*Transient…*" button to edit the time domain and Fourier parameters. Set the "*Final Time*" field to show five complete cycles. So for an operating frequency of (1/T) Hz, the period is T seconds. Set the "*Final Time*" to 5T. Set the "*Print Step*" and "*Step Ceiling*" fields to a value between T/100 and T/1000 seconds. See figure 1.2.
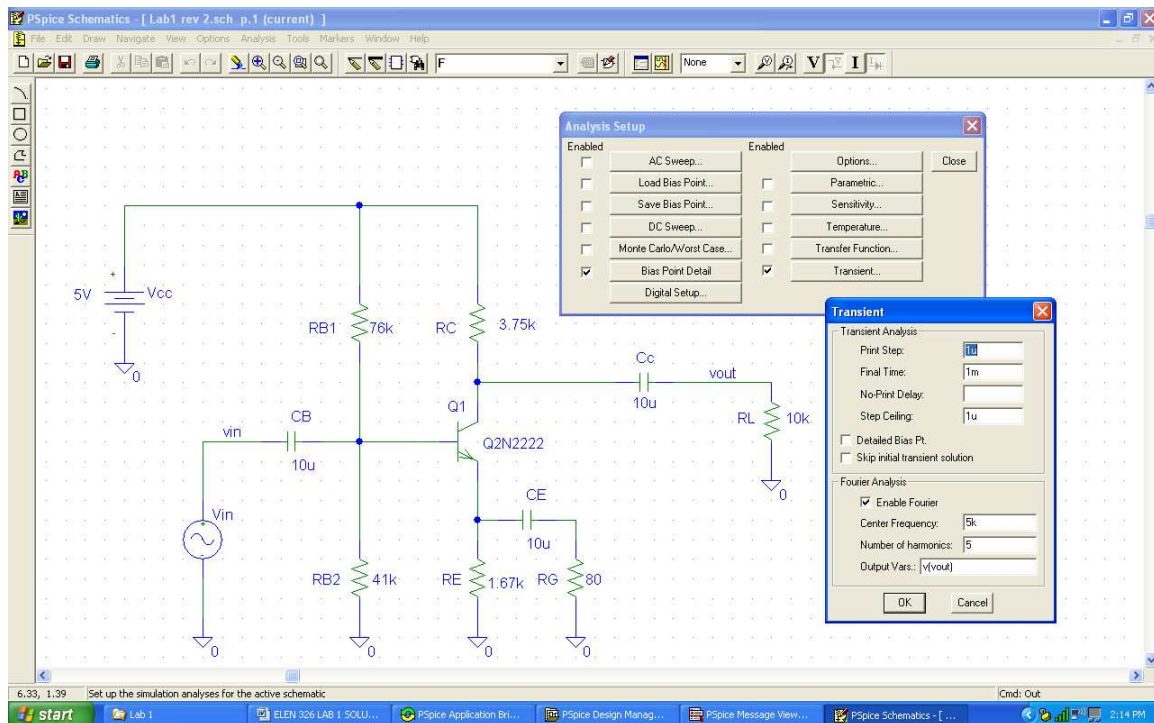

Figure 1.2: Transient Analysis setup.

To simulate the circuit, click the yellow simulate button or go to "*Analysis → Simulate*". Click the bold "V" and "I" buttons to display the DC bias values on the schematic. See figure 1.3.

Open the simulation results window (it should pop-up automatically when the simulation is run). Click "*Trace → Add Trace…*" or the "*Add Trace*" button to bring up a window to select which values are plotted. Click on "*V(vout)*" and "*V(vin)*" and press "*OK*". The transient waveforms are displayed in the results window. See figure 1.4.
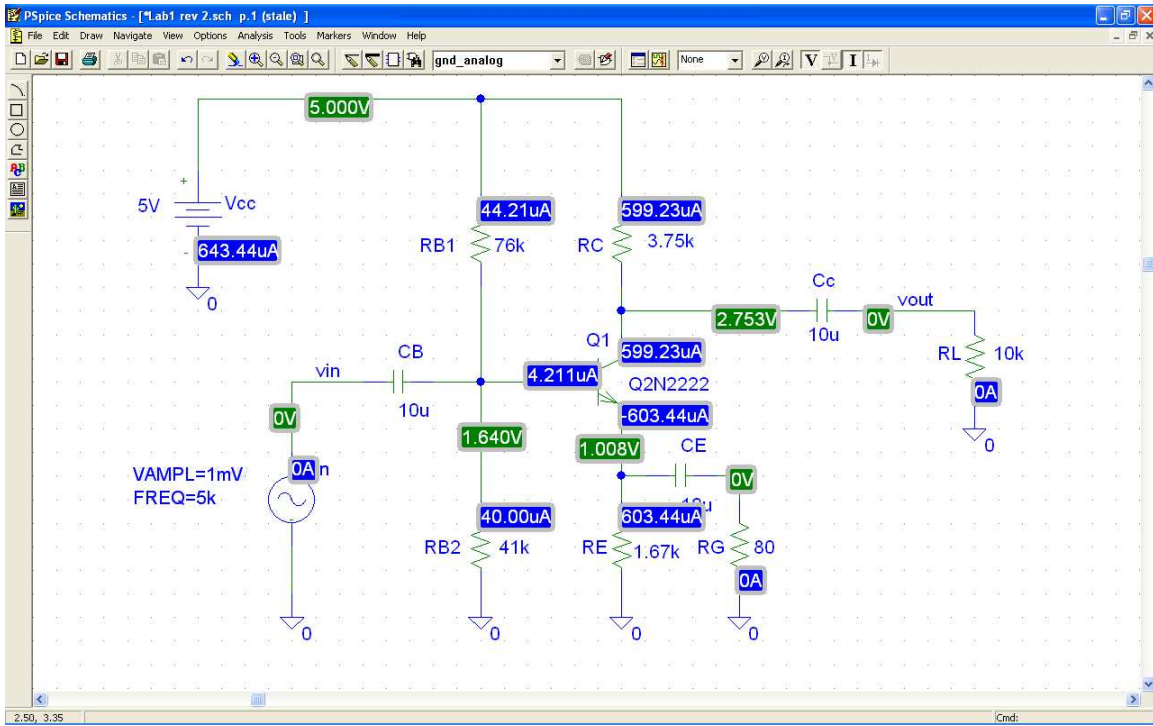
4

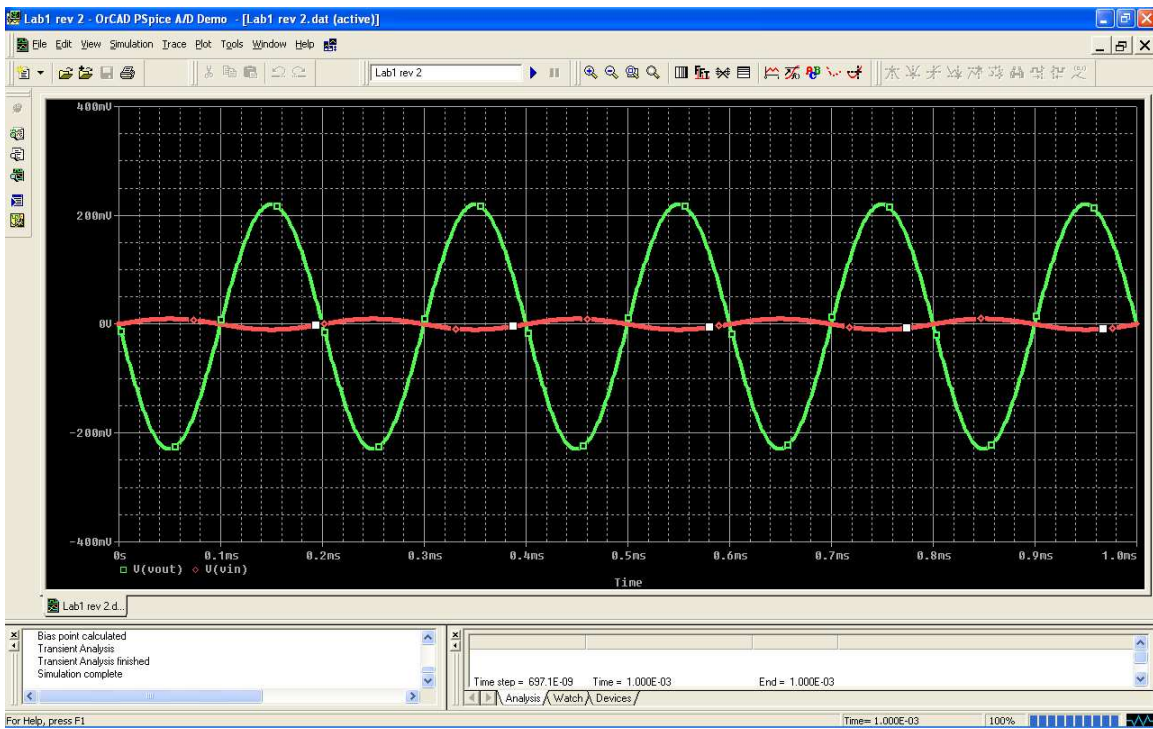Figure 1.3: Schematic with DC bias values shown.

Figure 1.4: Transient analysis showing input and output waveforms.

1.2   Measuring THD in PSpice

In the schematic window, go to "*Analysis → Setup...*" or click on the blue and grey setup button to bring up the Analysis Setup window. To determine Total Harmonic Distortion or THD, check the "*Enable Fourier*" box, set the "*Center Frequency*" to the operating frequency, set the "*Number of Harmonics*" to 5, and set the "*Output Vars*" field to "*v(vout)*" where "*vout*" is the name of the node at the circuit output. See figure 1.2 in "*Transient and DC Analysis in PSpice*".

Simulate the circuit, and open the simulation results window. To display the frequency components of the waveforms, click on the "*FFT*" button or go to "*Trace → Fourier*". Make the vertical axis log scaled by clicking on the "*Log Y Axis*" button or by going to "*Plot → Axis Settings...*" click on the Y-Axis tab and change the scale to "Log". See figure 1.5.
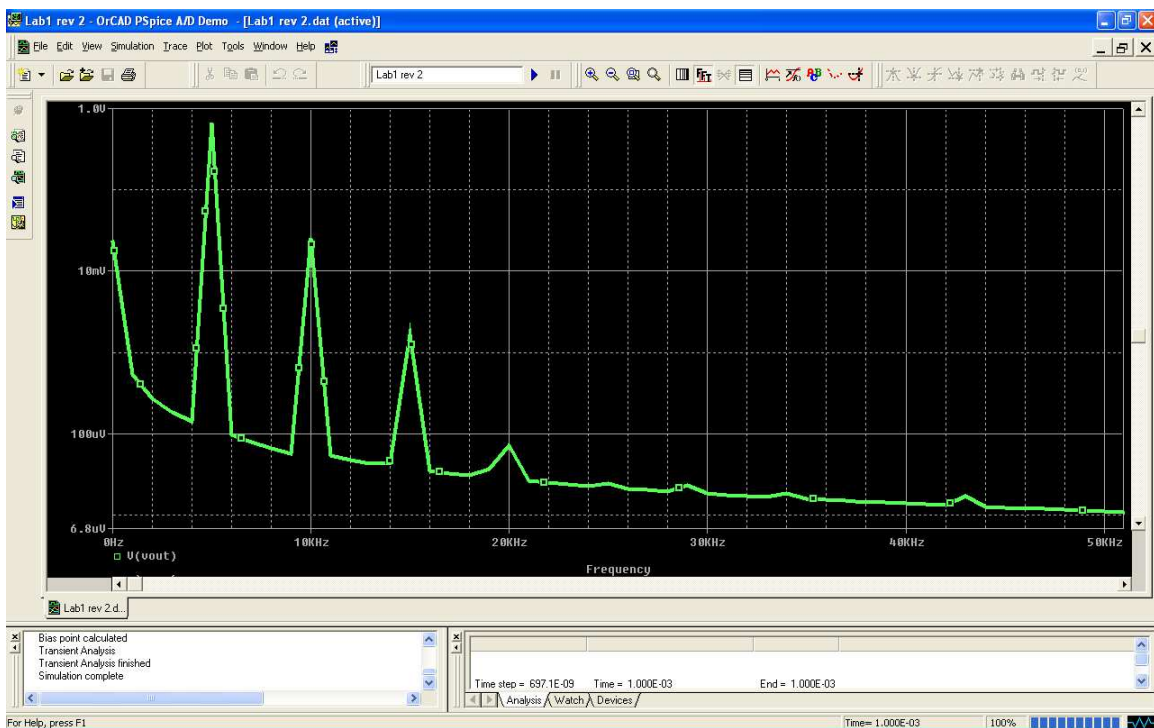

Figure 1.5: Harmonic Distortion

To view the numerical THD results, go to "*View → Output File*" and scroll all the way to the bottom. The Total Harmonic Distortion is given in percent. (Be careful to check the exponent because it is very easy to mistake having THD of 2.0% when it is actually 20%!) See figure 1.6.

To find the input voltage for 1% THD, go back to the schematic window and change the amplitude of the VSIN component, re-run the simulation, then go back and check the THD in the output file. Make any adjustments and re-simulate until the THD is 1%. Follow the same procedure for THD of 5%.
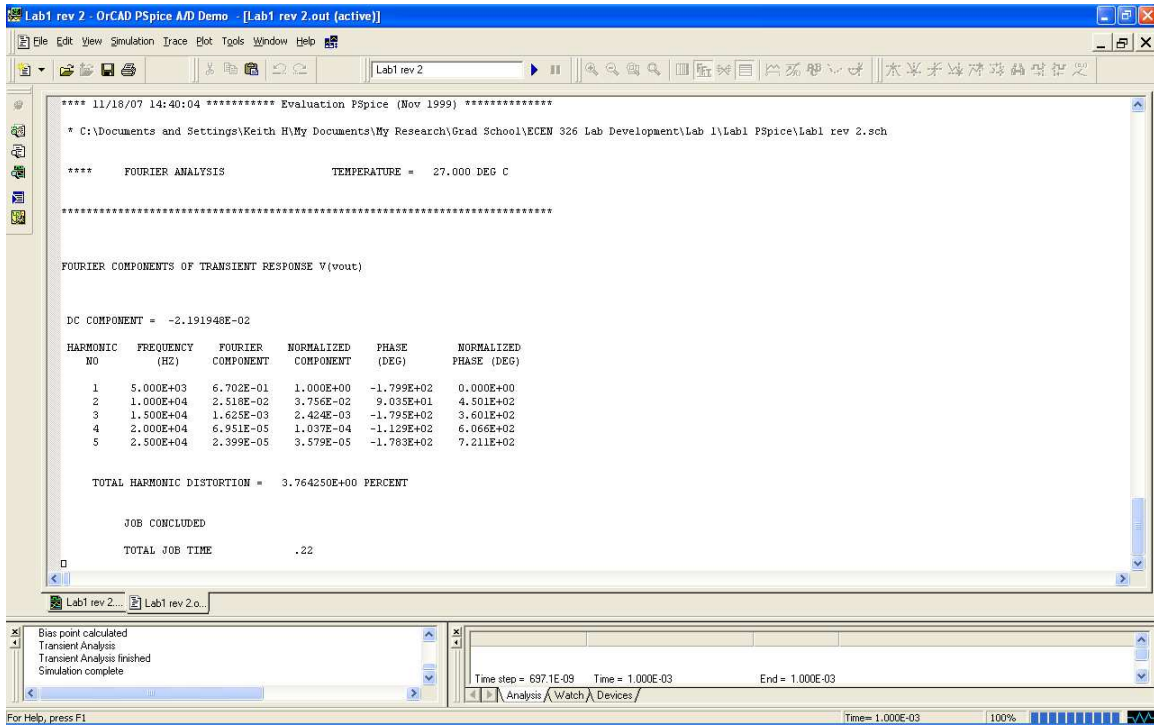
Figure 1.6: Output file showing THD and the distortion contribution of each harmonic

1.3 Voltage Swing Analysis in PSpice

To verify the output voltage swing, give the transient input voltage a relatively large value so that the output will be hard clipped. Run the simulation and examine the transient waveform to verify that there is hard clipping of the output signal. The waveform clipping defines the upper and lower values of the voltage swing range. See figure 1.7.
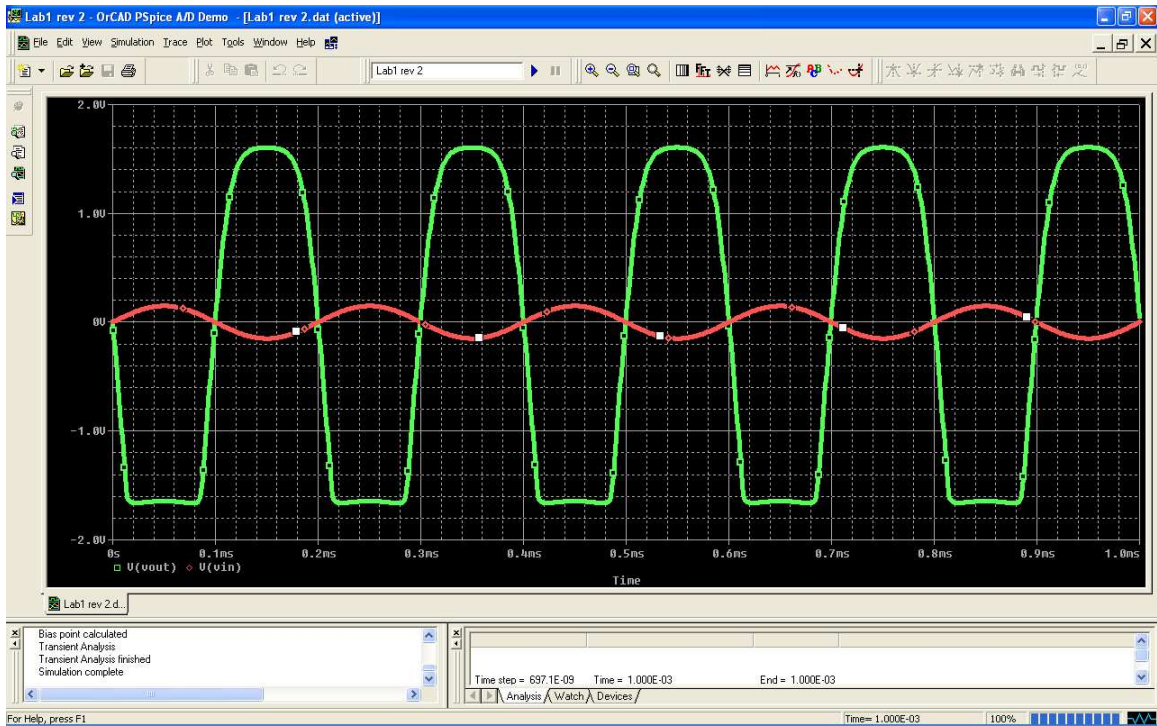

Figure 1.7: Analysis of output voltage swing. The swing of this waveform is 1.6V 0-to-peak.

1.4   Analysis of Input Resistance in PSpice

To verify the input resistance, open the schematic window and go to Analysis Setup. Check the box next to the "*AC Sweep…*" button, and then click on the button. In the window that opens, set a frequency range that includes the operating frequency.   See figure 1.7.
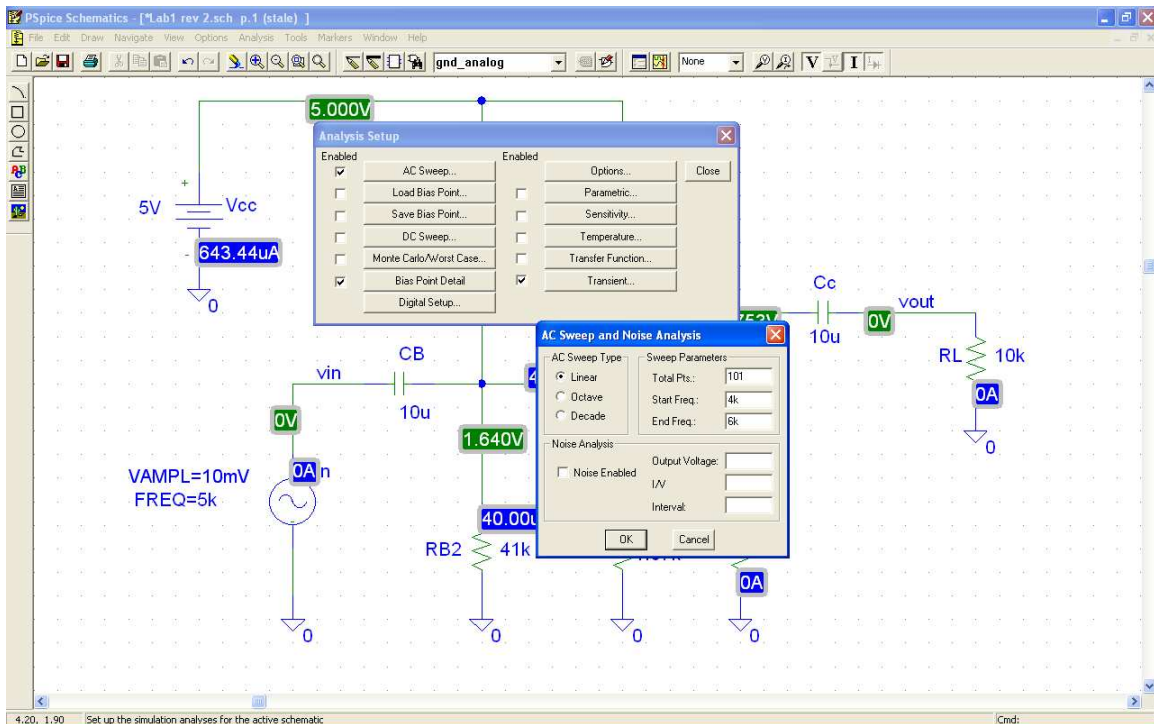

Figure 1.7: Setup for AC analysis.

Run the simulation, and then go to the results window. If you are also running a transient analysis as is being done in figure 1.7, you will see a window that allows you to choose AC or transient analysis. Choose AC, and then open the "*Add Trace*" window. To verify input resistance, select the voltage at the input node divided by the current generated by the input voltage source. Since input resistance is defined as V_{in} / I_{in}, the y-axis of the plot directly gives the value of the input resistance. See figure 1.8.
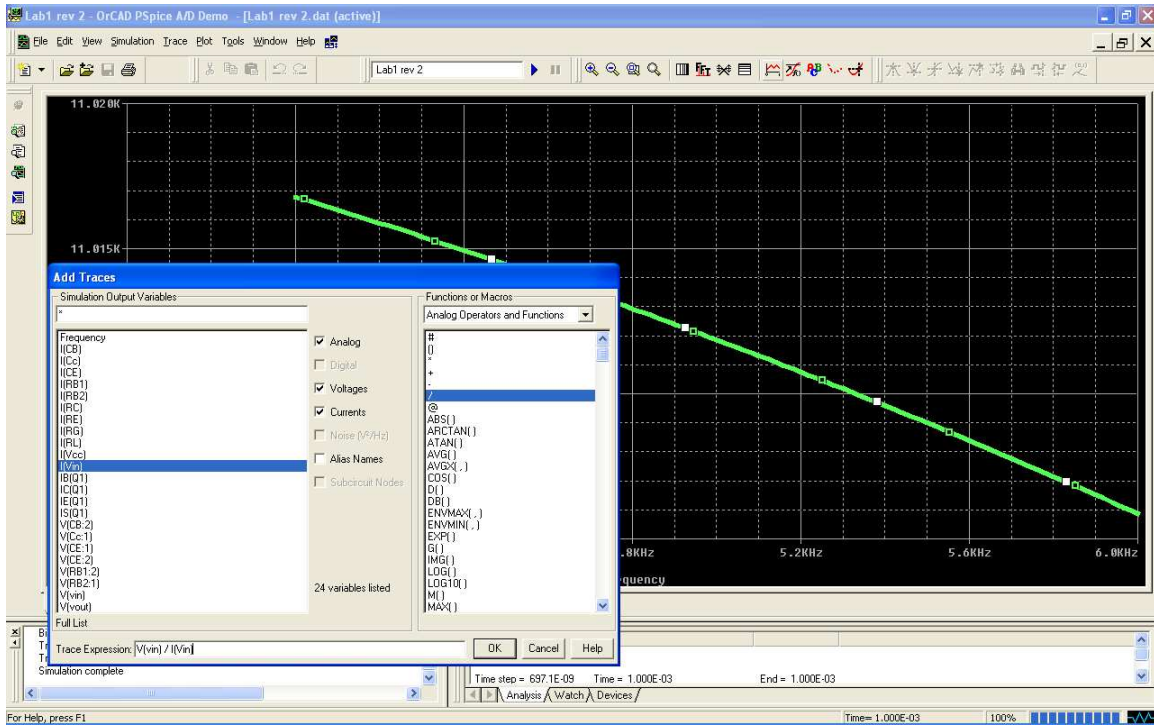
Figure 1.8: AC analysis to determine input resistance.

1.5   Analysis of Output Resistance in PSpice

Output resistance analysis is much the same as that of input resistance except that the AC input voltage must be set to 0 V, a voltage source must be connected to the output node, and the load impedance should be removed. Select AC = 1 for the output source. See figure 1.9.
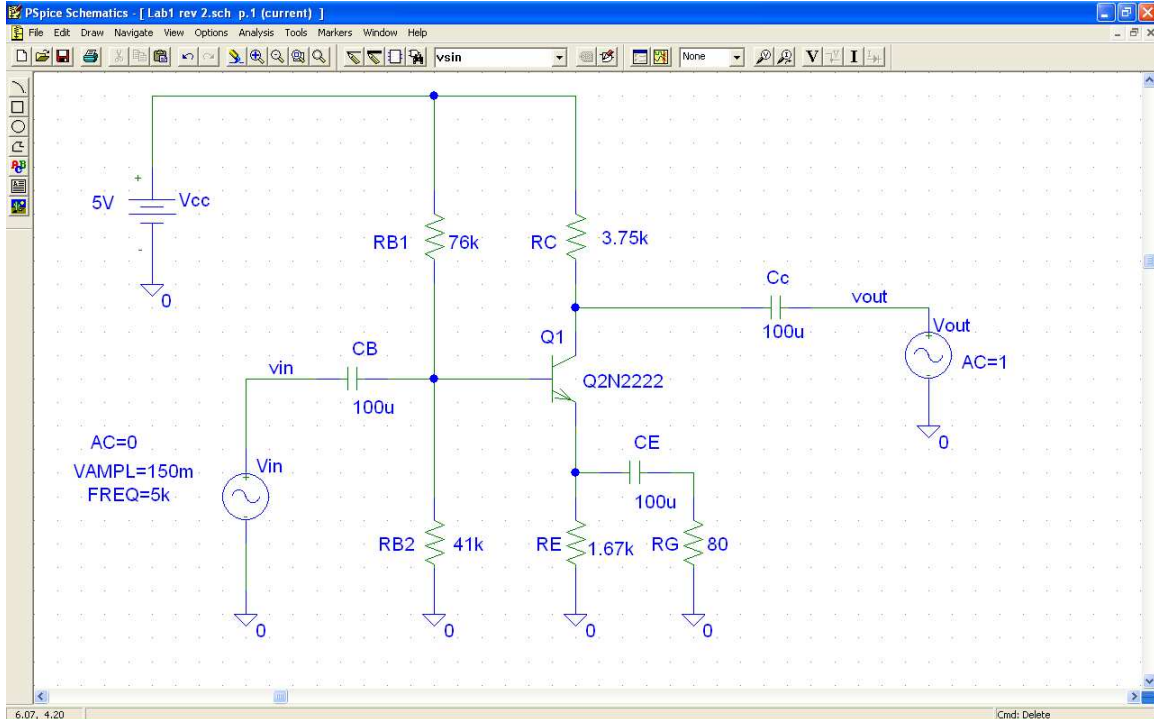

Figure 1.9: Circuit setup for determining output resistance.

Run the simulation, and then go to the results window. If you are also running a transient analysis, you will see a window that allows you to choose AC or transient analysis. Choose AC, and then open the "*Add Trace*" window. To verify output resistance, select the voltage of the output node divided by the current of the output AC voltage source. Since output resistance is defined as V_{out} / I_{out}, the y-axis of the plot directly gives the value of the output resistance. See figure 1.10.
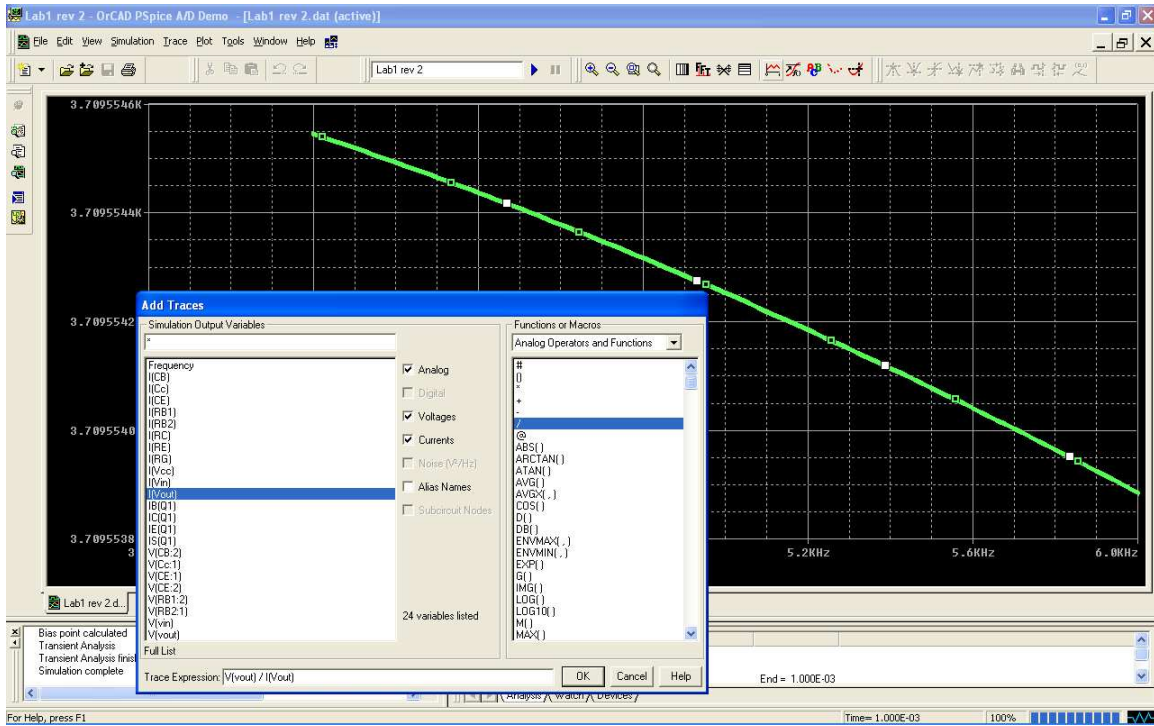
Figure 1.10: AC analysis to determine output resistance.

2. Lab Measurements

2.1 Circuit Construction and DC Analysis

First make sure the power supply to the NI ELVIS prototyping board is turned off (the switch on the front of the unit), while the system power is turned on (the switch on the back of the unit). Next, build the circuit that was designed in prelab onto the board. Note that each row of 5 holes is one electrical node (ie., 5 holes in a row are electrically connected). Also, the long vertical columns constitute a single node. These should be used as ground and power supply nodes. See figure 2.1.
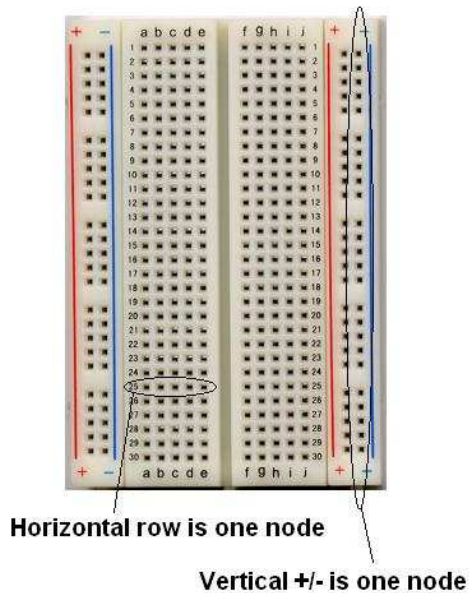

Figure 2.1: Nodes of a breadboard.

A 1-to-1 center-tapped isolation transformer should be placed at the circuit input. Connect the transformer across a ridge on the breadboard. The schematic for the transformer is shown in figure 2.2. Note the transformer only isolates the input signal, it should not be used for the power supply.
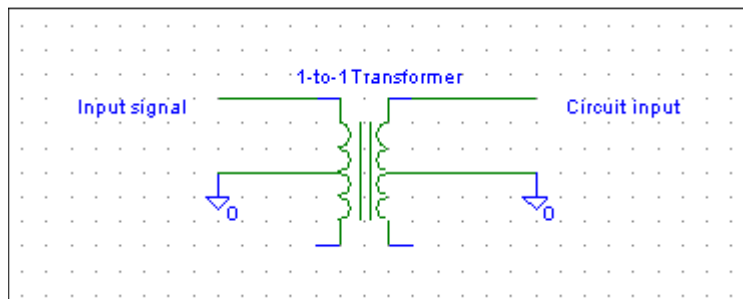

Figure 2.2: 1-to-1 transformer schematic

The coupling capacitors should be large enough that their impedance at the operating frequency is negligible. The equation for capacitive impedance is:

$$Z_C = \frac{1}{j\omega C}$$

If a coupling capacitor of 100 μF is chosen, then at an operating frequency of 5 kHz:

$$|Z_C| = \frac{1}{\omega C} = \frac{1}{\left(2\pi \cdot 5 \times 10^3\right)\left(100 \times 10^{-6}\right)} \cong 0.32[\Omega]$$

Large capacitors are usually electrolytic. Electrolytic capacitors have polarity that must be observed or the internal dielectric will break down and destroy the capacitor. Generally, the two capacitor leads are different in length, with the longer lead connecting to the more positive node. Also, there should be a light stripe with (-) signs painted on the side that connects to the more negative node.
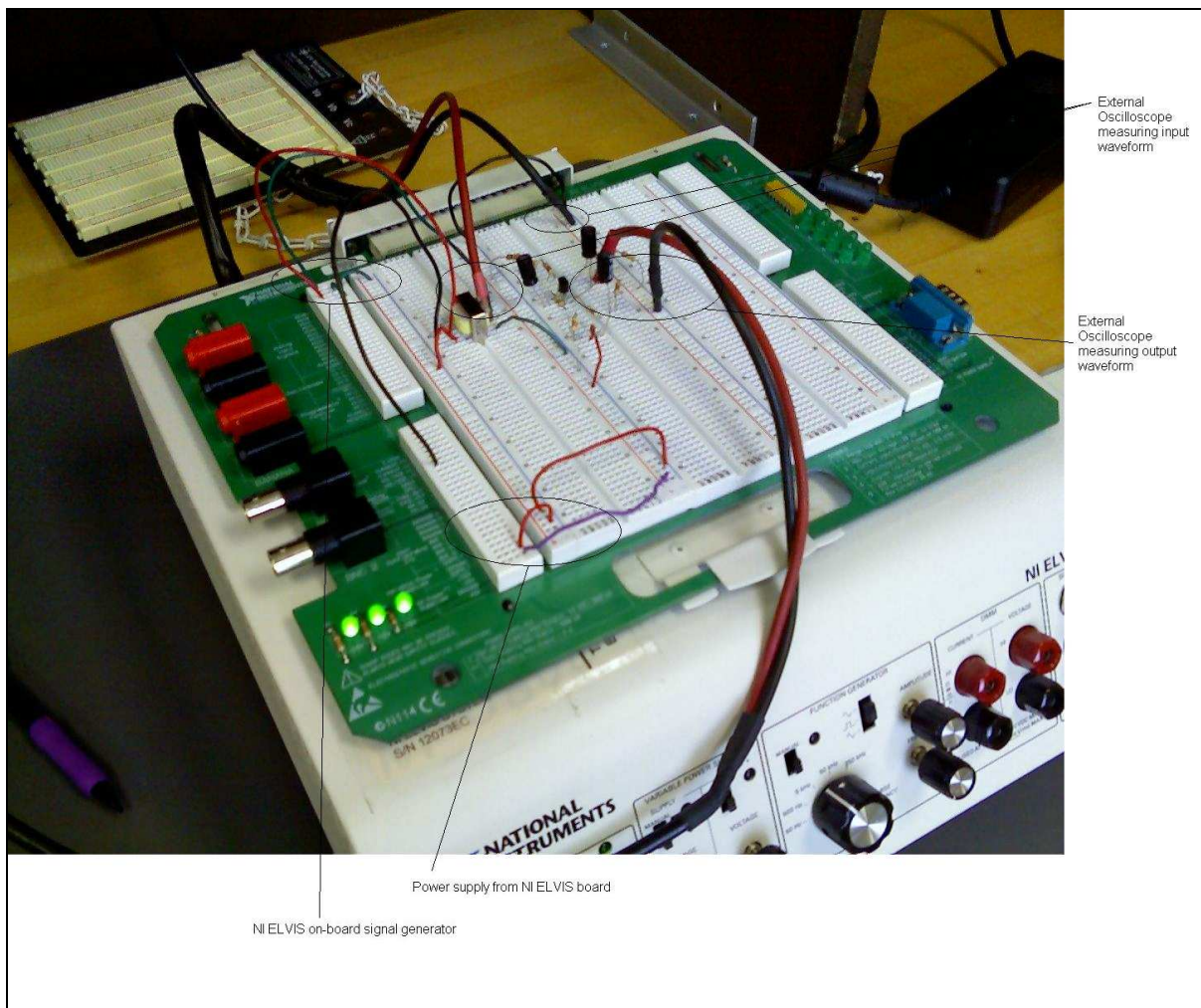

Figure 2.3: Circuit layout on NI ELVIS prototyping board with external oscilloscope measuring waveforms.

Connect a wire from one of the holes next to the +5 V supply located on the bottom left of the ELVIS board to a supply node on the circuit. Similarly, connect a wire from the ground supply (located right above the +5 V supply) to a ground node on the circuit. See the bottom of the breadboard in Figure 12. Now turn on the power supply switch located on the front of the ELVIS board. The three green LEDs marked +15 V, -15 V, and +5 V should now be lit.

Locate the digital multimeter on the ELVIS board (marked DMM[2]). See Figure 2.4. To measure current, first switch off the supply power switch on the front of the unit. Connect a wire to "CURRENT LO" and another to "CURRENT HI". Measuring current requires breaking the circuit at the node to be measured and then inserting the multimeter wires in the broken connection. Connect "CURRENT HI" to the side of the broken connection with the higher voltage and "CURRENT LO" to the lower voltage. If the wires are connected incorrectly, no damage will occur, but the multimeter will display a negative current.
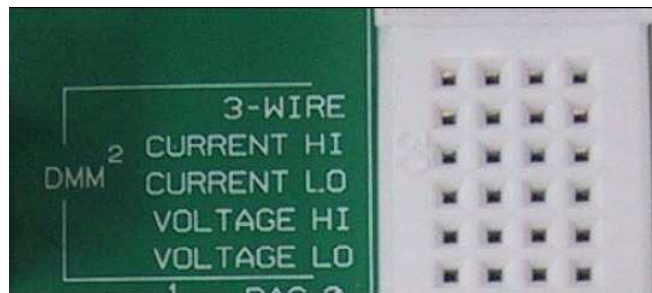

Figure 2.4: The ELVIS board multimeter connections

Go to the computer that is attached to the ELVIS board and launch the NI ELVIS software. When the main interface opens, click the Digital Multimeter, and then click the NULL option to calibrate the board. Click the DC Current button (the button marked "A" with the straight lines). Turn on the ELVIS board power supply. Record the measurement. See Figure 2.5.

After recording the current, turn off the ELVIS power supply. Remove the multimeter probes from the circuit and reconnect the circuit node the way it was before the measurement. Disconnect the next node to be measured and repeat the same process until all required currents are measured.

Figure 2.5: DC Current Measurement

To measure voltages, first switch off the supply power switch on the front of the unit. Connect a wire to "VOLTAGE LO" and another to "VOLTAGE HI". Unlike measuring current, voltage measurements require that the circuit not be broken. Instead, measurements are made in parallel to the existing circuit. Since we are referencing all voltage to ground, the "VOLTAGE LO" wire should be connected to a ground node of the circuit for all voltage measurements. Go to the computer and click on the digital multimeter window. Click the DC voltage button (the button marked "V" with the straight lines) and then click the NULL option to re-calibrate the meter. See figure 2.6.

Turn on the ELVIS board power supply. To measure the voltage at a node, simply connect the "VOLTAGE HI" wire to the node. To measure another node, just move the wire from the first node to the next node to be measured.
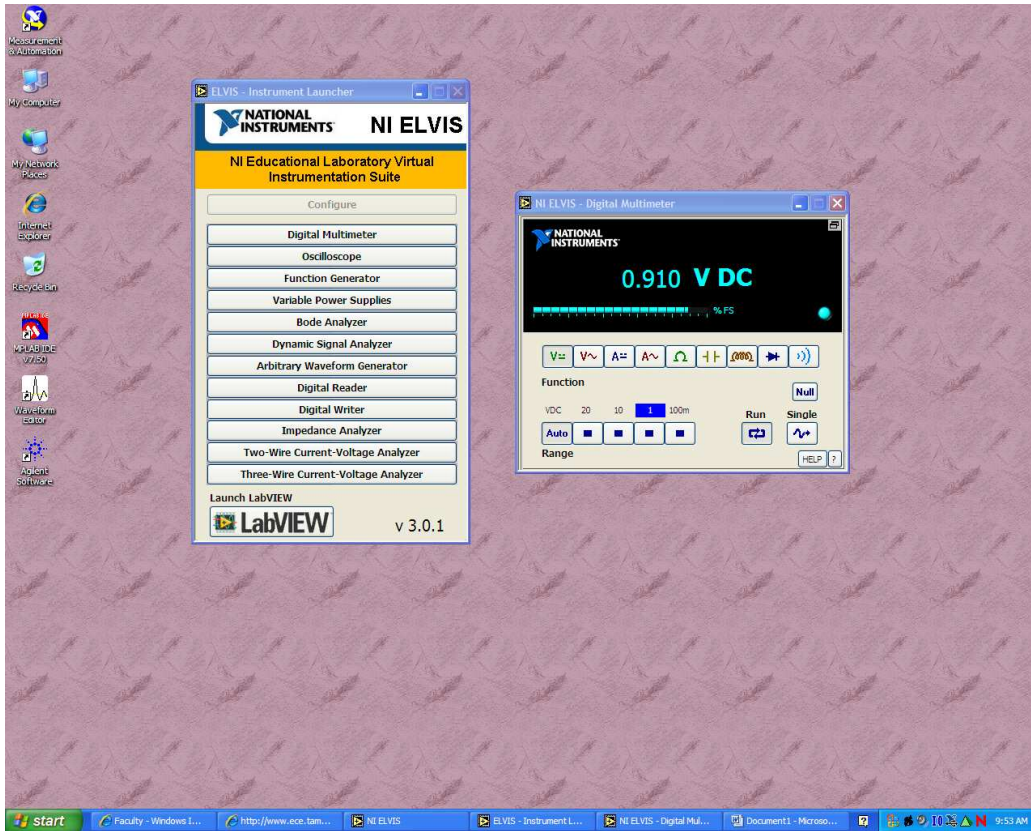
Figure 2.6: DC Voltage Measurement

2.2   THD Analysis with NI ELVIS

1. Construct the circuit on the NI ELVIS breadboard.

2. Turn on power to NI ELVIS (front and back).

3. Log in to your NOVELL account and start the NI ELVIS software. The ELVIS Instrument Launcher will pop up on the screen. See figure 2.7.
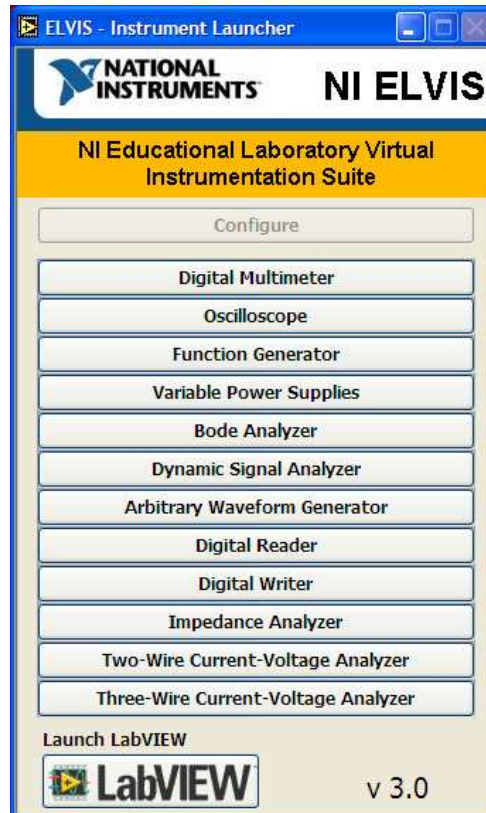

Figure 2.7: NI ELVIS Instrument Launcher window

4. On the ELVIS Instrument Launcher, click on the Function Generator button to bring up the Function Generator window (figure 2.8). Enter the desired operating frequency in the text box under the frequency fine tuning knob. Under "*Waveforms*", click on the sinusoid button. Make sure the DC offset is set to zero. Enter the desired input amplitude in the text box under the peak amplitude knob. The smallest input amplitude the NI ELVIS can generate is 10 mV. If a smaller amplitude is needed, simply make a 10:1 voltage divider at the input of the 1-to-1 transformer at the input of the circuit. See figure 2.9.

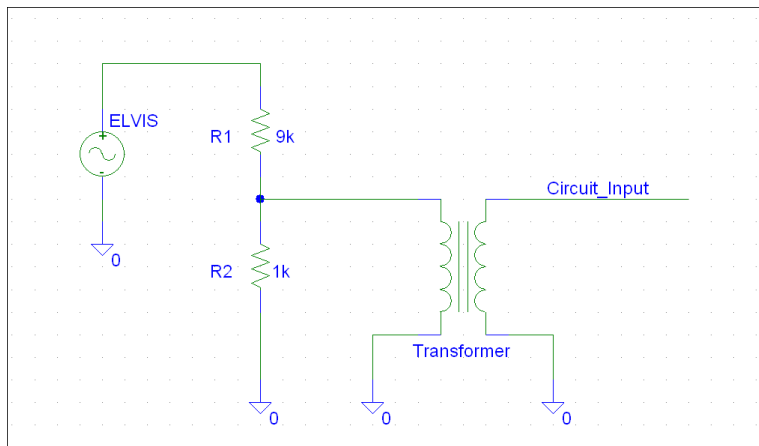Figure 2.8: NI ELVIS Function Generator window

Figure 2.9: 10:1 Voltage divider

5. Click on Dynamic Signal Analyzer on ELVIS Instrument Launcher. The window shown in figure 2.10 should pop up.

6. Set the Source Channel parameter to be "*ACH0*" and Trigger Source parameter to be "*FGEN SYNC OUT*".

7. Connect the output of the circuit to *ACH0+*, and connect *ACH0-* to ground.

19

8. Hit the "*Single*" button on the Dynamic Signal Analyzer to get a spectrum plot.

9. The window will display the THD in % under the spectrum plot. Record this value.
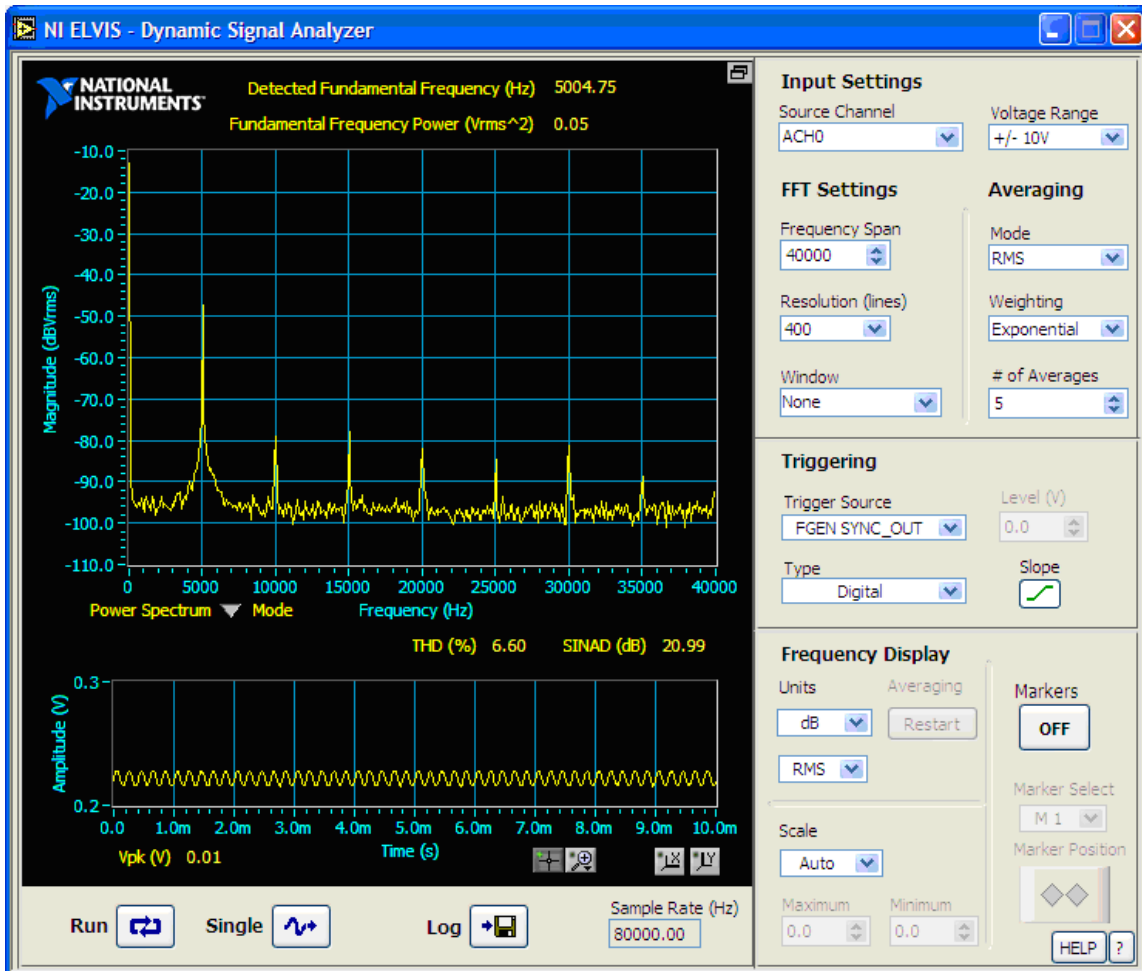

Figure 2.10: Distortion Analysis using NI ELVIS Dynamic Signal Analyzer

11. Save the Distortion Analysis window to include in the lab report. To copy a window, click on the window and then press "*Alt + Prnt Scrn*". This copies the window to the clipboard. Then paste the picture to the lab document.

Note: The ELVIS board tends to give larger variation of THD than the oscilloscope. For example, ELVIS can give a range of values from 4% to 8% THD for a single input amplitude. The oscilloscope has fewer variations than the ELVIS.

2.3 THD Analysis with HP Oscilloscope

1. Construct the circuit on the NI ELVIS breadboard.

2. Connect CH1 of the oscilloscope to the circuit input and CH2 to the output of the circuit.

3. Connect the function generator to the input of the circuit and set its frequency to the operating frequency (eg. 5 kHz) and the amplitude to the value calculated in the pre-lab for the given THD. Examine the transient waveform to verify the circuit is operating correctly. See figure 2.12 for a typical transient waveform.

4. Push the "±" button on the oscilloscope to pull up the function options. See figure 2.11 for a layout of the oscilloscope control panel.
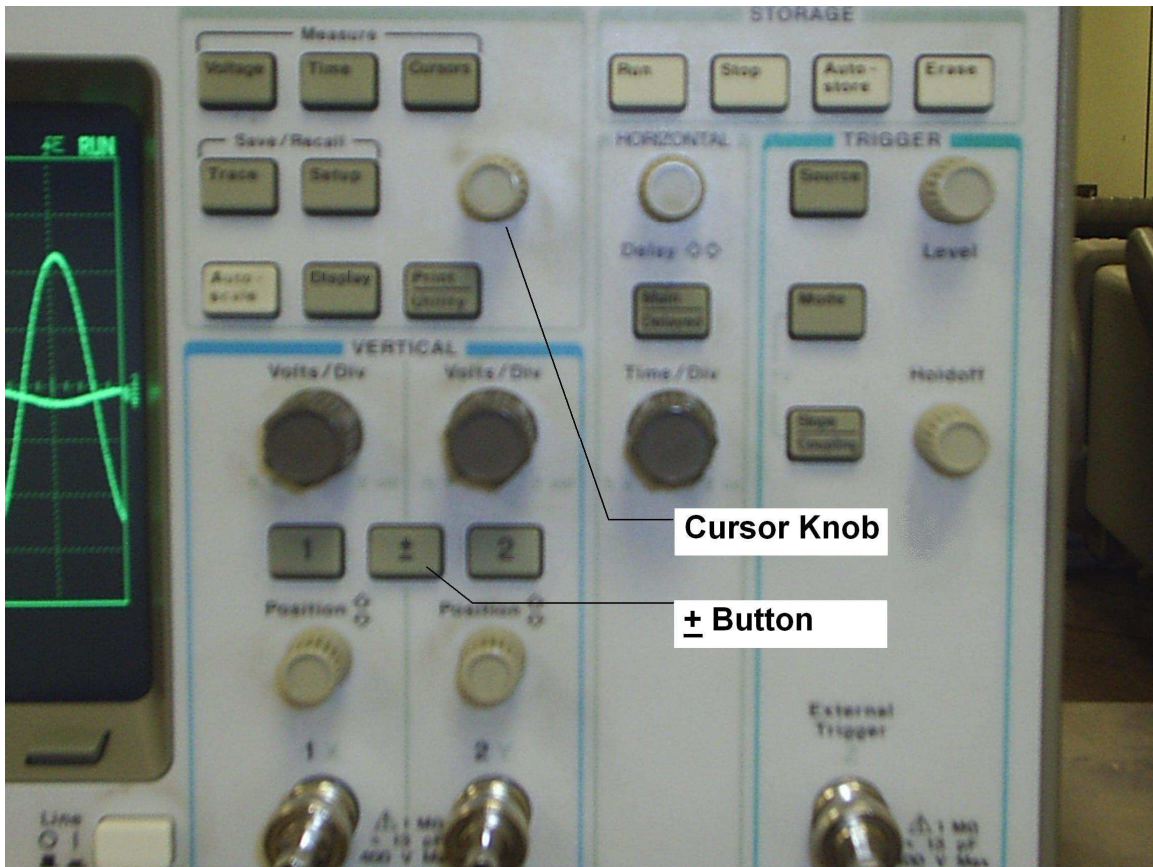

Figure 2.11: HP oscilloscope control panel

5. In the options on the bottom of the screen, toggle Function 2 to "On". Then under Function 2, select "Menu". See figure 2.12.
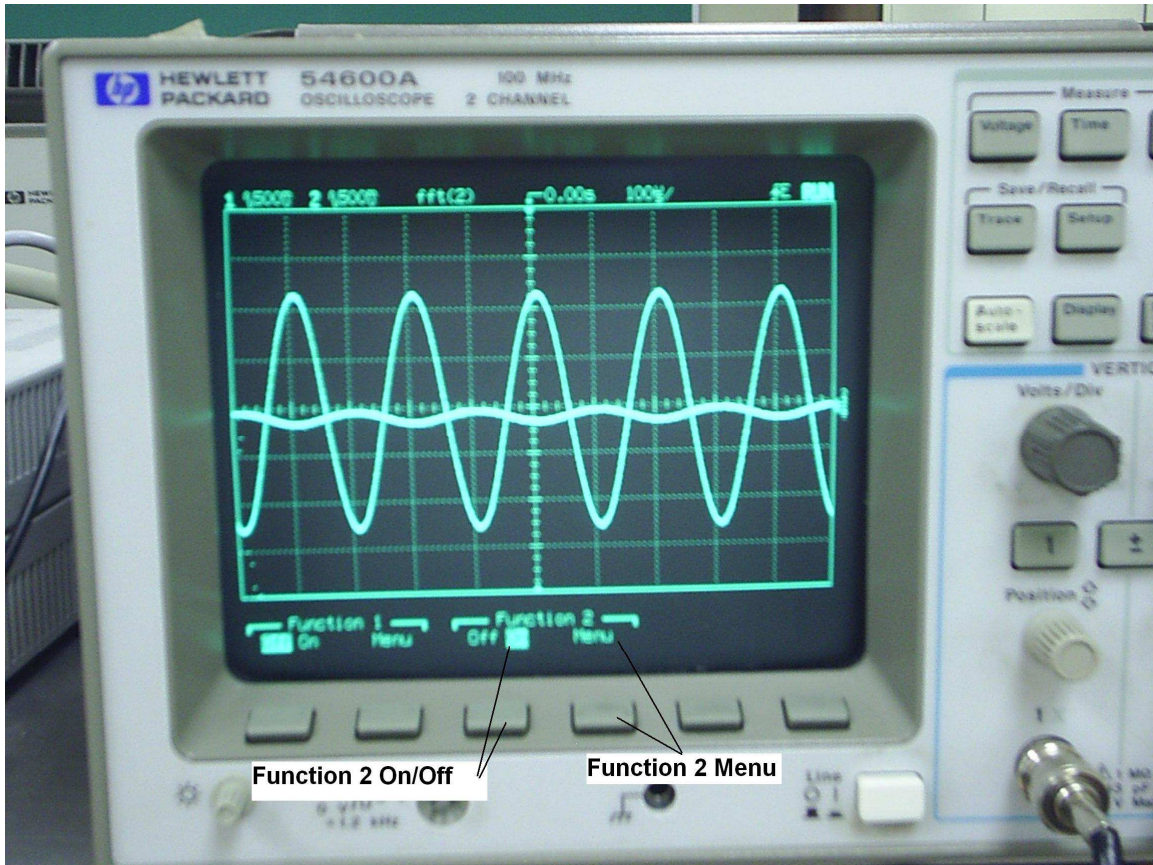

Figure 2.12: Setting the FFT function

6. Turn off the transient waveforms by pushing the "1" and "2" buttons on the control panel. This will take away the FFT menu. Push the "±" button again to bring the FFT menu back up.

7. In the FFT options at the bottom of the screen, under "*Operand*", toggle to "2". Under "*Operation*", toggle to "FFT". Then push the button under "*Units/div*" to set the vertical scale. When the value under "*Units/div*" is highlighted, turn the cursor knob on the control panel to change the vertical scale so that there is a measurable difference between peaks. A typical value for "*Units/Div*" is "*10.00 dB*". Then push the button under "*Ref Levl*" to set the absolute scale. When the value under "*Ref Levl*" is highlighted, turn the cursor knob on the control panel so that the peaks are all above the bottom of the screen. A typical value for "*Ref Levl*" is "*10.00 dBV*". See figure 2.13.

22

Figure 2.13: Setting the FFT options

8. Turn the "*Time/Div*" knob on the control panel to change the horizontal scale (frequency span) of the FFT waveform. Change the span so that 5 to 10 harmonics are visible on the screen. For example, for an operating frequency of 5 kHz, a total span of 30 kHz should suffice. See figure 2.14.
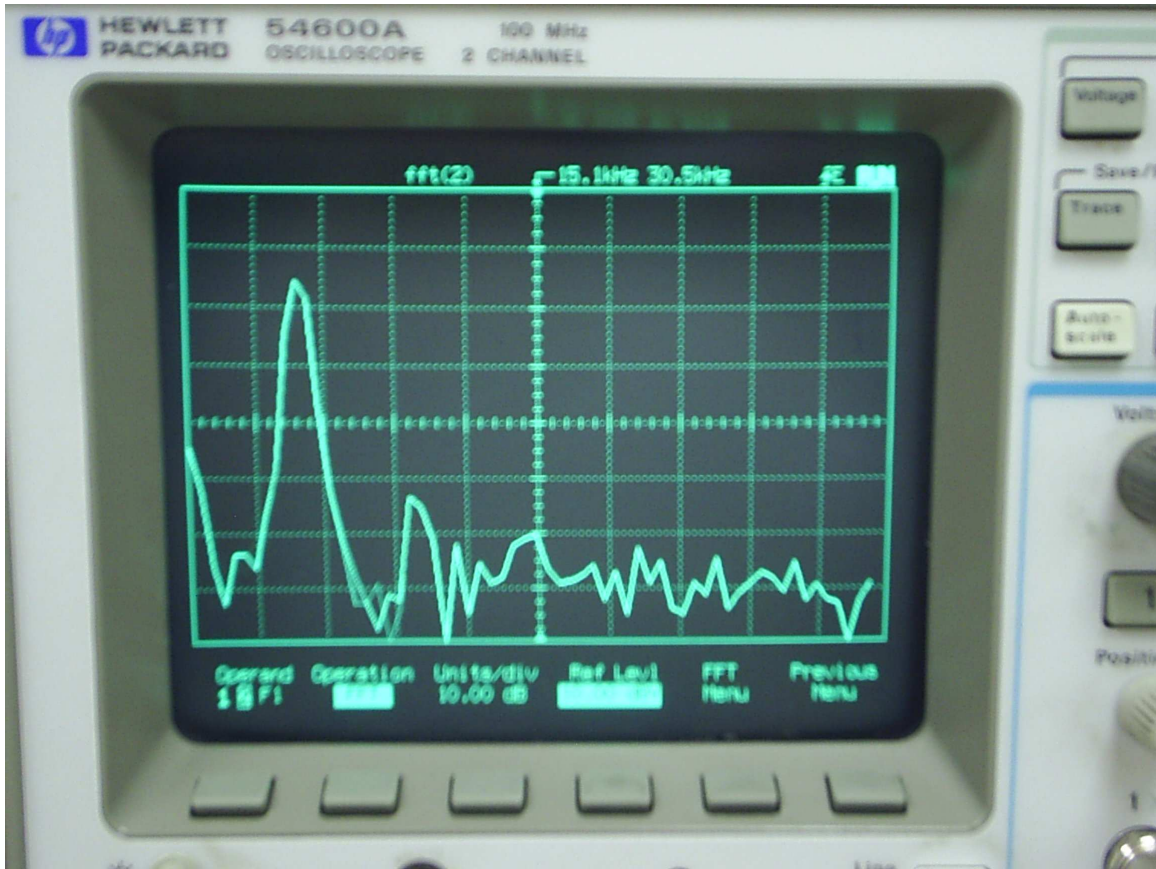
Figure 2.14: Setting the FFT options

9. Push the "*Cursors*" button on the control panel to bring up the cursor menu at the bottom of the screen. Under "*Active Cursor*" toggle until "*f1*" is highlighted. Turn the cursor knob until the vertical cursor on screen is at the operating frequency (this should be the largest peak). Then toggle the active cursor to "*V1*". Turn the cursor knob until the horizontal cursor is at the top of the operating frequency's peak. Toggle the active cursor to "*f2*" and move the cursor to the second harmonic. For an operating frequency of 5 kHz, the second harmonic is at 10 kHz. Toggle the active cursor to "*V2*" and move the cursor to the peak of the second harmonic. On the screen right below the waveform, the value absolute value of ΔV is the difference in dB between the fundamental tone and the measured harmonic. See figure 2.15. Record this value. Leave "*f1*" and "*V1*" at the operating frequency and move "*f2*" and "*V2*" to the next harmonic. Record its |ΔV| in dB. Repeat for a total of 5 harmonics.
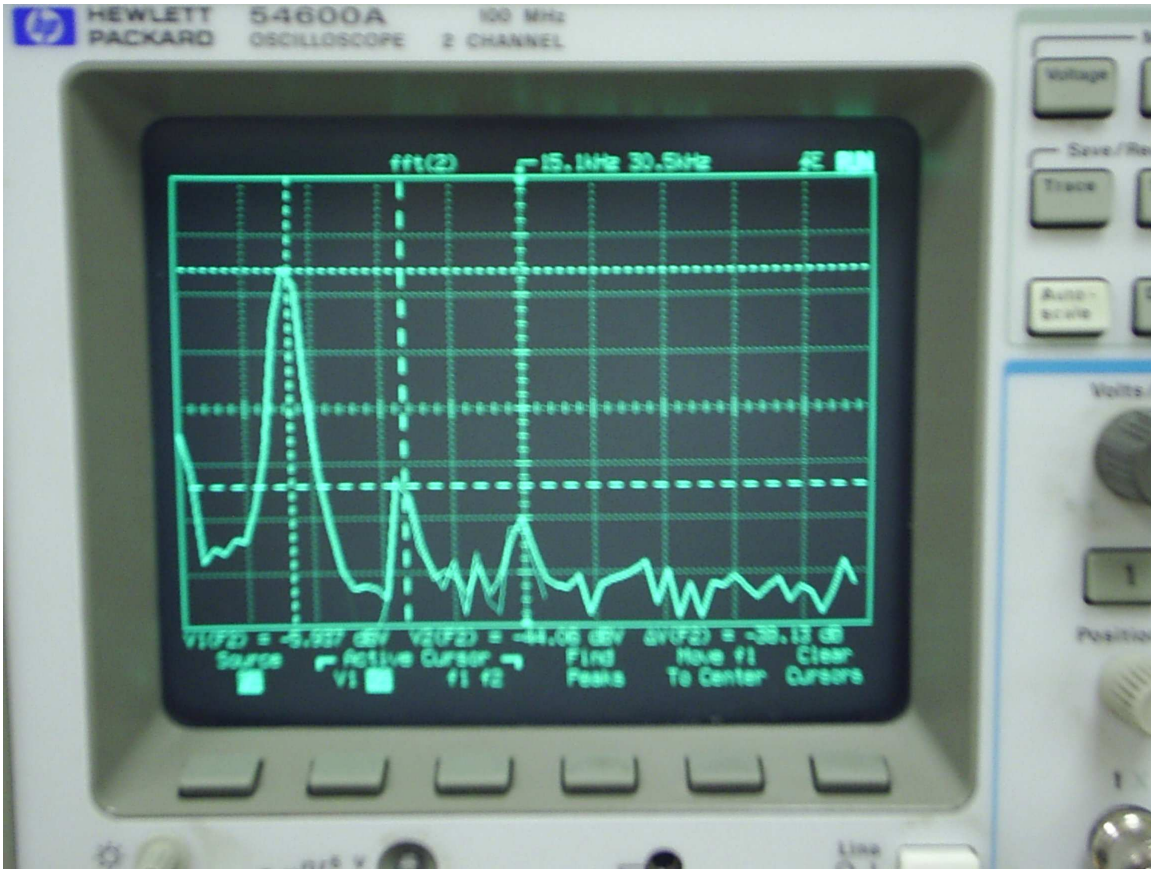
Figure 2.15: Using cursors on FFT window for THD measurements

10. Calculate the THD from the measured values of ΔV. The equation for THD is:

$$THD = \sqrt{HD2^2 + HD3^2 + HD4^2 + HD5^2 + ...}$$

$HDn = 10^{\frac{-\Delta V_n}{20}}$ where *n = 2,3,4,... and*

ΔV_n = *(Fundamental amplitude in dB) − (n_{th} harmonic amplitude in dB)*

11. To plot a screenshot of your waveform to the PC, open the program "*HP BenchLink Scope*". Click "*Image → New…*" then click "*Ok*" on the window that pops up. The program takes about 5 seconds to copy the oscilloscope window. See figure 2.16. To save the image, click "*File → Save As…*" and use the .TIF file extension.
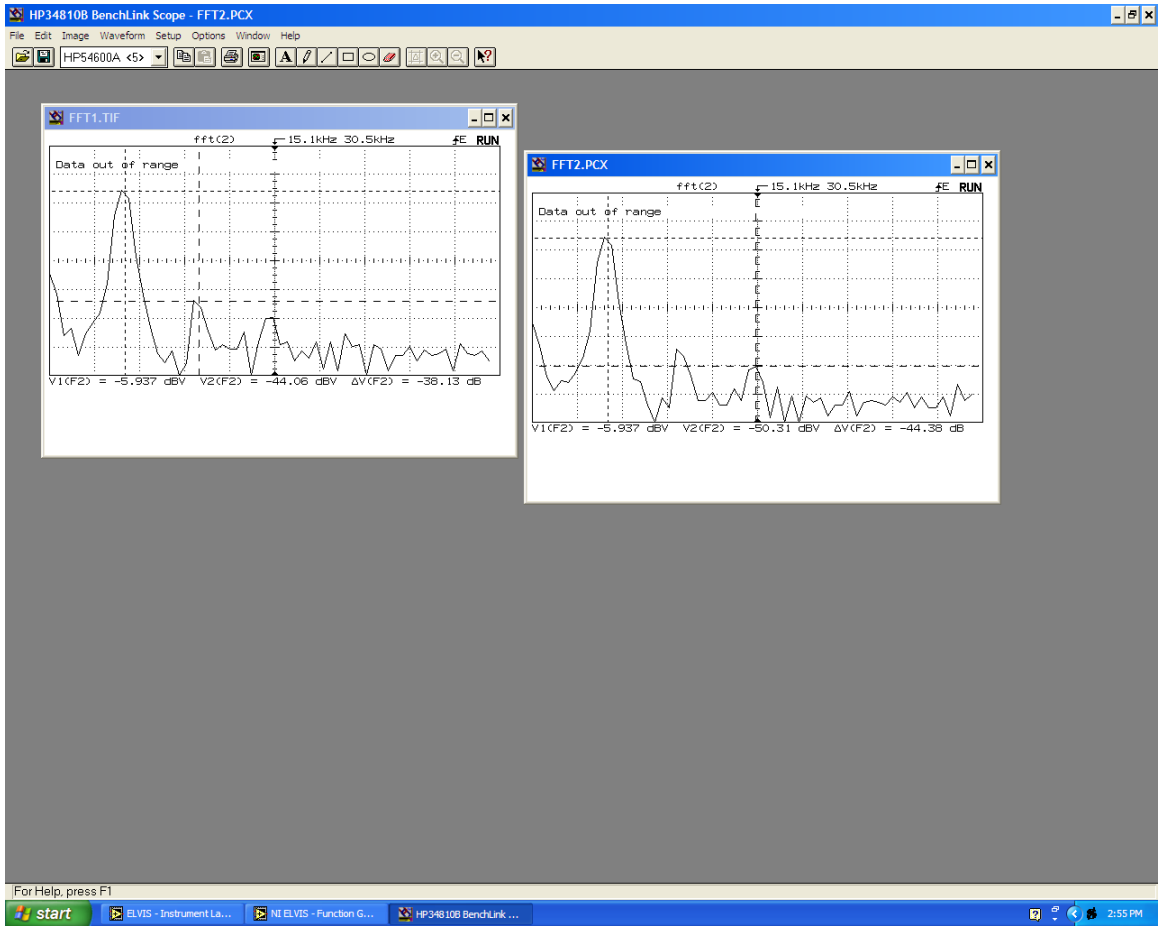
Figure 2.16: HP BenchLink Scope showing captured FFT plots

2.4 Measuring Maximum Voltage Swing

Apply a signal to the input of the circuit.

Turn on the oscilloscope by pushing the button at the front of the unit. Attach the channel 1 probe across the input and the channel 2 probe across the output of the circuit. Figure 2.3 shows the probes in place on the circuit. Push the "*CH 1 Menu*" and "*CH 2 Menu*" to turn on the channels for each probe. Use the "*Vertical Position*" knobs to align the waveforms on the screen. Use the "*Volts/Div*" knobs to scale the waveforms so they fit on the screen. Adjust the input until the signal on the oscilloscope is as large as possible without clipping. In figure 2.17, each grid square's vertical length represents 500 mV. So from figure 2.17, we can see the maximum unclipped output waveform is 1.5 V 0-to-peak.
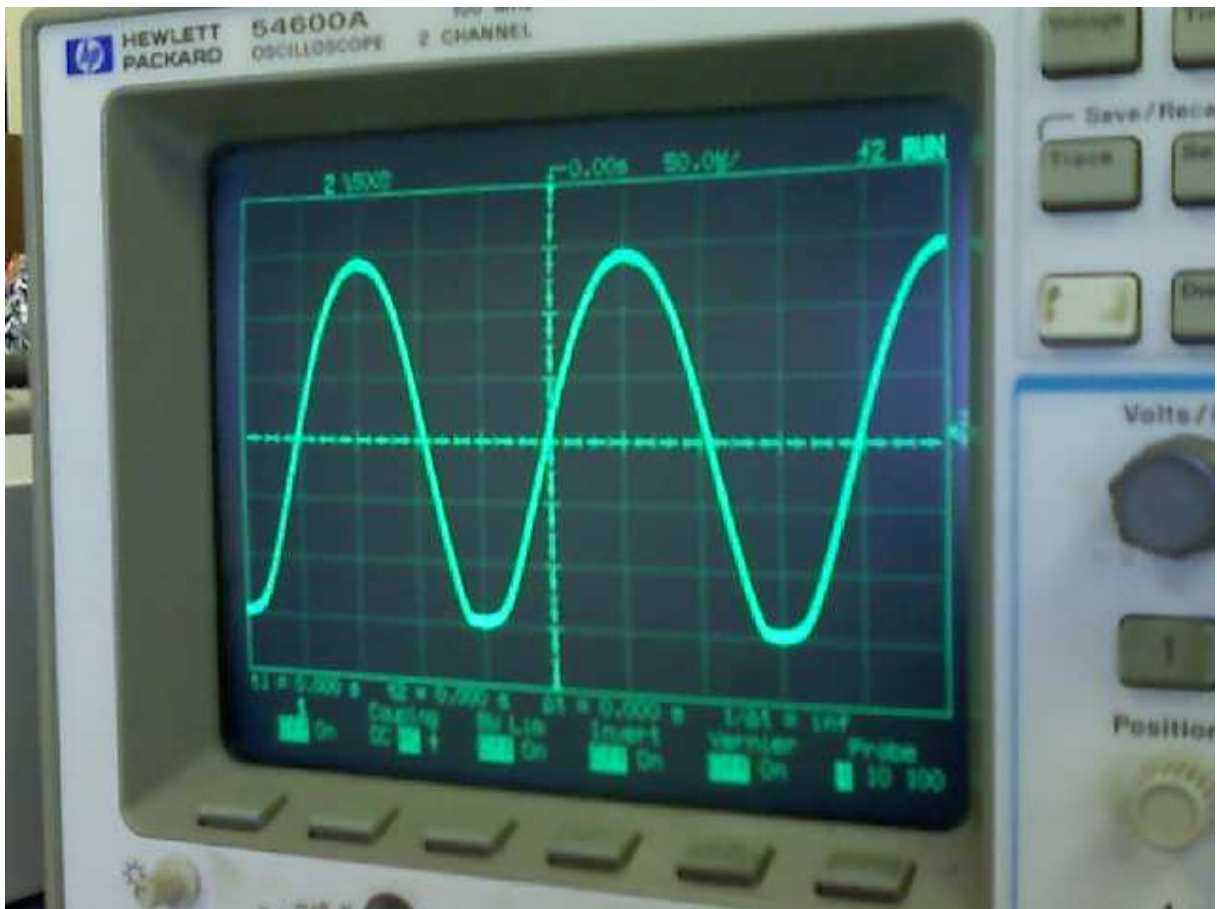

Figure 2.17: HP oscilloscope showing maximum unclipped output waveform

2.5 Measuring Input and Output Resistance

Apply a signal to the input using the ELVIS board by clicking on the "*Function Generator*" button. Make sure the manual mode button on the front of the unit is turned off so that all functions of the software function generator are available. Set the frequency of the function generator using the coarse control buttons and then the fine tuning knob. The operating frequency should be exactly 5 kHz. Select a sinusoidal wave type and set the amplitude low enough that there will not be any clipping at the output. See figure 2.18.

To measure the input resistance, turn off the power to the circuit, and then attach the multimeter probe to the input to measure the AC current. Switch on the power and record the result. The input current and the input voltage measured from the gain calculation combine to give the input resistance by the equation:

$$R_{in} = \frac{v_{in}}{i_{in}}$$

Measure output resistance by removing the load resistor, grounding the circuit input, and applying a signal to the output of the circuit. Measure the AC voltage and AC current at the output node to get the output resistance:
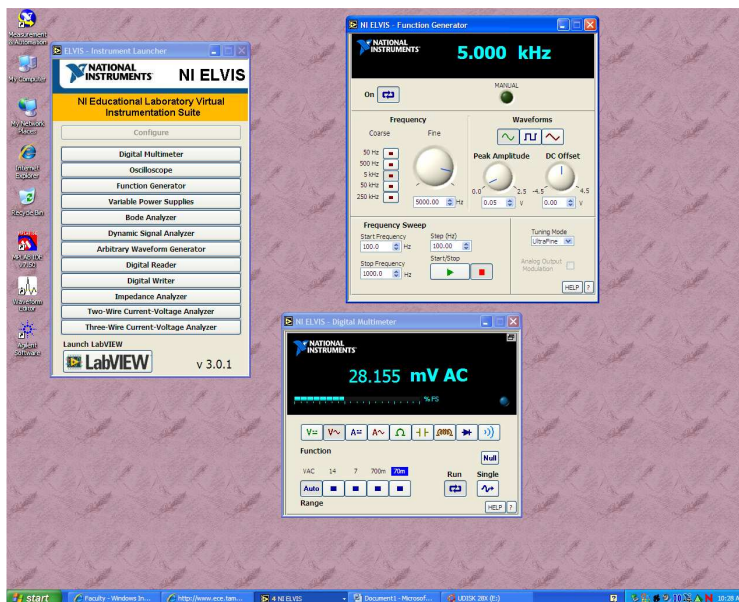
$$R_{out} = \frac{v_{out}}{i_{out}}$$

Figure 2.18: ELVIS function generator input signal