

Development of Analog Circuit Labs using TDS7000 Series Digital Phosphor Oscilloscopes

In this lab paper, we describe the use of TDS7000 Series digital phosphor oscilloscopes (DPOs) in the development of analog circuit laboratory experiments for senior classes. These experiments are designed to offer:

- Practical insight into the topics covered in the class
- Solid understanding of issues involved in building the actual circuits designed in the class, like proper biasing and debugging
- Experience with measurement equipment similar to the equipment used in the industry

The TDS7000 Series' simple user interface and built-in, automated waveform measurements allow the students to take measurements easily and accurately. These measurement features, coupled with the oscilloscope's calculator-type math features, can be used to evaluate expressions involving the measurements taken. The TDS7000 Series' spectrum-analyzer-style interface allows students to perform frequency domain analysis.

Acquisition

Before taking any measurements, you must first acquire a signal or waveform. The basic acquisition mode of a digital oscilloscope is the sample mode, where the waveform is sampled in time and the amplitude of each sample is digitized and displayed. With the use of interpolation, these samples can be connected to create a continuous waveform display.

A digital storage oscilloscope (DSO) or digital phosphor oscilloscope (DPO) can also digitally process the signal before it is displayed, making complex measurements much easier. For example, you can use the oscilloscope's average mode to remove the effects of random noise to make precise amplitude measurements. The averaging function smoothes out the waveform by averaging multiple waveforms, while the high-resolution mode filters the samples taken during an acquisition to create a higher-resolution, lower-bandwidth signal. The average and high-resolution acquisition modes are useful when measuring signals of small amplitudes. Other types of acquisition modes include peak detect and envelope. Refer to the Oscilloscopes User Manual for more information.

In this paper, we describe the setup to perform two of the most common analog circuit laboratory experiments:

- Lab I: Measuring the Gain of a Common Emitter Amplifier
- Lab II: Frequency Response of a Common Emitter Amplifier

Lab I: Measuring the Gain of a Common Emitter Amplifier

One of the important measurements of an amplifier is its gain. The following is the setup to measure the gain of a common emitter amplifier, in addition to the description of the automated measurement using LabVIEW.

Setup

1. Set up the circuit as shown in the Figure 1. The resistors R1, R2 are just to attenuate the signal so that a small input signal (few mVs) is applied to the amplifier.

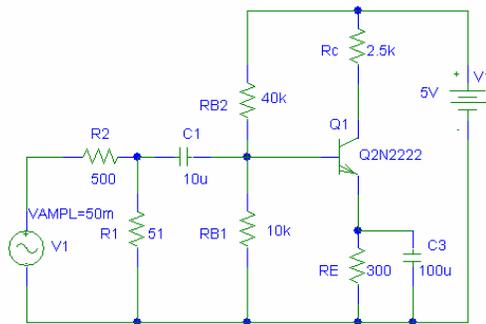


Figure 1: Schematic circuit for Common Emitter Amplifier

2. Measure the DC bias points using Digital Multimeter. Make sure the amplifier is biased properly and operating in the forward active region.
3. Connect the input signal of the amplifier (at the base of the transistor) to one the channel on the oscilloscope (say Channel 1). Connect the output signal of the amplifier (at the collector of the transistor) to other channel on the oscilloscope (say Channel 2). These signals are AC signals superposed on the DC bias signal. The DC signal will normally be large compared to the AC signals, and one may not able to see the AC signals clearly. The AC signal without the DC component can be displayed by setting the coupling to AC. To do this, select Vertical → Vertical Setup. A graphical palette appears similar to the Figure 2. Select coupling as AC for both the channels.
4. The input waveforms can be noisy because of its small amplitude of the signals. To average out the noise select Horiz/Acq → Acquisition → Average/HiRes.

Figure 3 shows the acquired input, output waveforms. Channel 1 (yellow trace) displays the input waveform and channel 2 (blue trace) displays the output waveform.

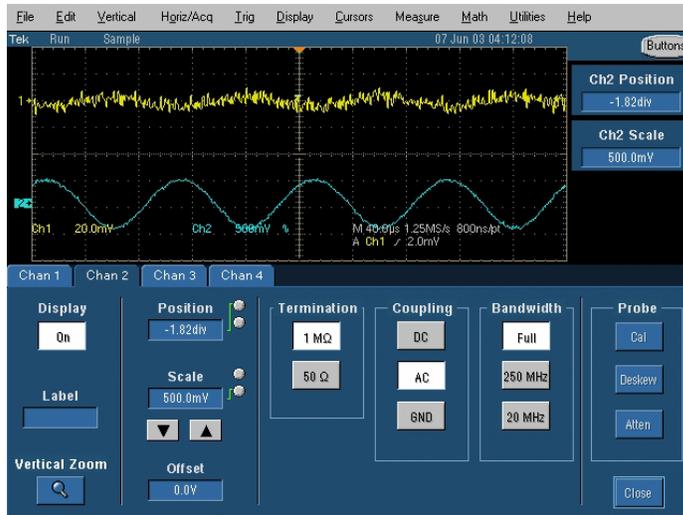
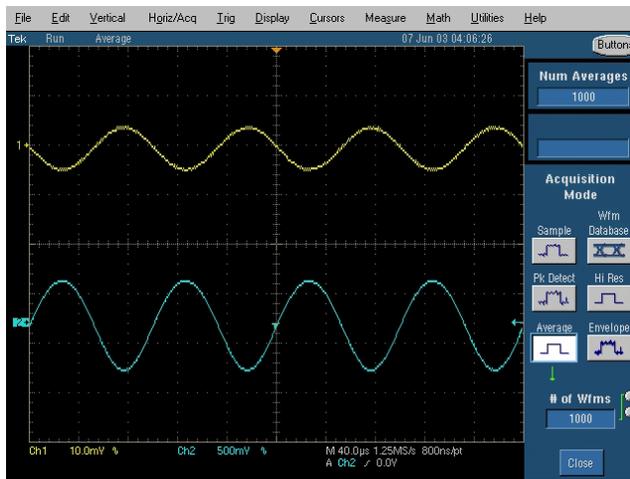
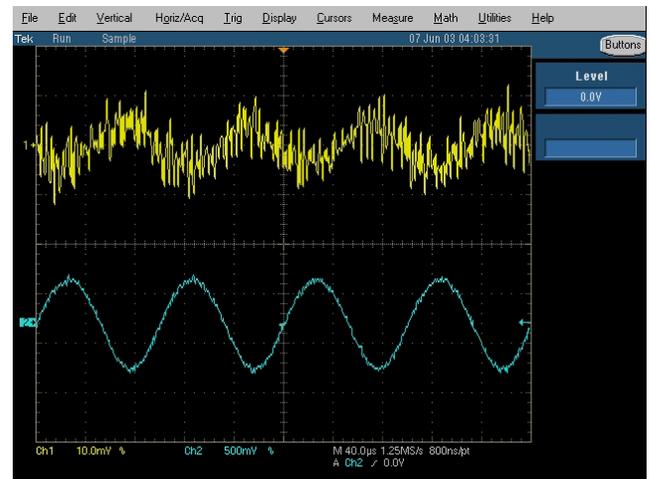


Figure 2: AC Coupling Setup



(a) Sample Acquisition Mode



(b) Average Acquisition

Figure 3: Input and Output waveforms

To measure the gain of the amplifier, we need the amplitude of the input and output signals. One can measure the input and output signal amplitudes manually using the cursors. Using the TDS7000 Series DPOs, these measurements can be obtained automatically.

To do this, Select Measure → Measurement Setup. Select amplitude and the source as shown in the Figure 4.

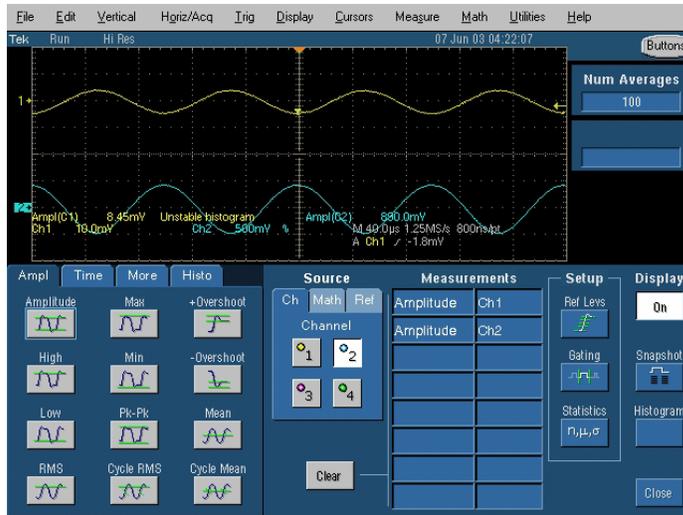


Figure 4: Measuring the amplitude of the signals

Gain is calculated as the ratio of the output to the input amplitudes. The Math feature allows you to calculate the gain from the measurements taken. Select Math → Math setup and enter the expression for the gain as shown in the Figure 5. The expression is displayed as the waveform. The gain can be measured as the mean of the Math waveform. To calculate the mean, setup the measurement as described earlier, but select Math as the source instead of channel.

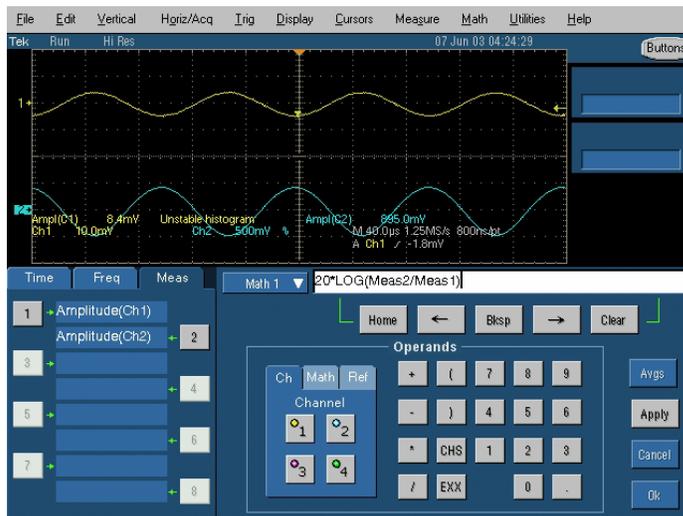


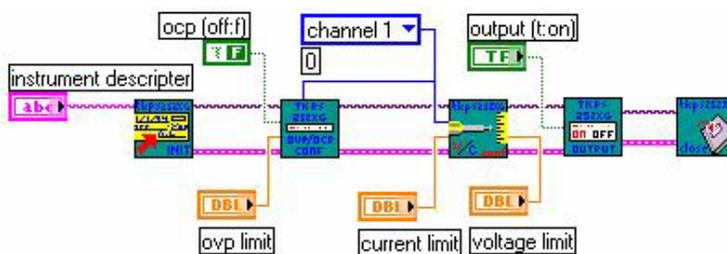
Figure 5: Equation Editor

Automating Measurements Using LabVIEW Software

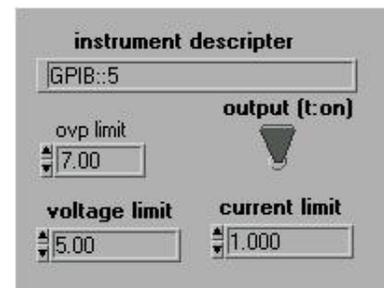
You can use LabVIEW software on the TDS7000 Series' Windows-based oscilloscopes to automate the equipment(s) connected to the oscilloscope to perform a sequence of tasks, and analyze and process the measurements taken. The automation greatly alleviates several common laboratory problems, such as insufficient laboratory time, bad laboratory data, and calculations. The automation increases student productivity by accelerating routine data-taking. This section assumes that the reader has some familiarity with the LabVIEW environment and has worked with instrument drivers before. In this section, the automation for the gain measurement is described using the TDS7104 digital phosphor oscilloscope, coupled with LabVIEW software, connected via GPIB to an AFG310 arbitrary function generator and PS2521G DC power supply. The programming should be similar for other instruments. Download the required drivers for these instruments and make sure you are able to see these devices in the Functions Palette → Instrument I/O → Instrument Drivers. To determine whether the drivers are installed and working properly, open Getting Started.vi of the instrument and run it.

Setting up the PS2521G DC Power Supply

1. From the Functions Palette, open Instrument I/O → Instrument Drivers → tkps252x VIs and find Initialize.vi. Configure OVP-OCP.vi, Configure Current-Voltage.vi, Output on-off.vi, Close.vi and place them on the block diagram.
2. From the Tools Palette, select Connect Wires, right click on the resource name of Initialize.vi and create a control (by selecting Create → Control). Similarly, create a control for ovp limit, constants for ocp and channel of Configure OVP-OCP.vi. Connect the rest as shown in the Figure 6. Controls are inputs to the vi and indicators are the outputs of the vi. The controls can be used instead of constants and corresponding values must be set via the front panel.
3. Run the vi to see if the Power Supply is controlled properly and to check for any errors up to this point.



(a) Block diagram
Front panel

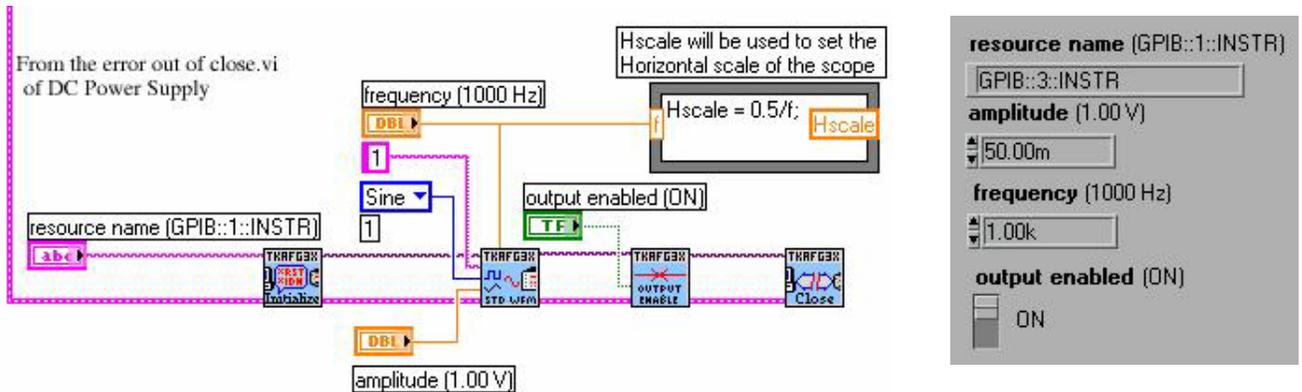


(b)

Figure 6: VI part for DC power supply

Setting up the AFG310/AFG320 Arbitrary Function Generator

1. From the Functions Palette, open Instrument I/O → Instrument Drivers → kafg3x0 and find Initialize.vi, Configure Standard Waveform.vi, Configure Output Enabled.vi, Close.vi and place them on the block diagram.
2. Create controls, constants and connect the vis placed as shown in the Figure 7.
3. Create a formula node and enter the expression to get the Horizontal Scale required for the oscilloscope to capture about five cycles.
4. Run the vi to see if the Function Generator is also controlled properly and to check for any errors up to this point.



(a) Block diagram
(b) Front panel

Figure 7: VI part for Arbitrary Function Generator

Setting up the Oscilloscope

1. From the Functions Palette, open Instrument I/O → Instrument Drivers → tkpds7k and find Initialize.vi. Set Vertical Parameters.vi, Set Wfm Display.vi. You will need two each for the Set Vertical Parameters.vi and Set Wfm Display.vi, one for the channel connected to the input signal of the amplifier and the other for the output signal.
2. Place Measurement source.vi, Measurement Type.vi, Measurement State.vi. Again, you will need two each of these vis, one for measuring input amplitude and the other for output amplitude.
3. Place Autoset.vi, Set Acquisition Mode.vi, Set Horizontal Scale.vi, Set Trigger A Edge Parameters.vi, Set Horizontal RecLength.vi.
4. Place two of Get Measurement Value.vi, Close.vi
5. Create a formula node and enter the expression for gain.
6. Create constants, controls, and indicators and connect as shown in the Figure 8.
7. The front panel can be arranged as shown in the Figure 9.

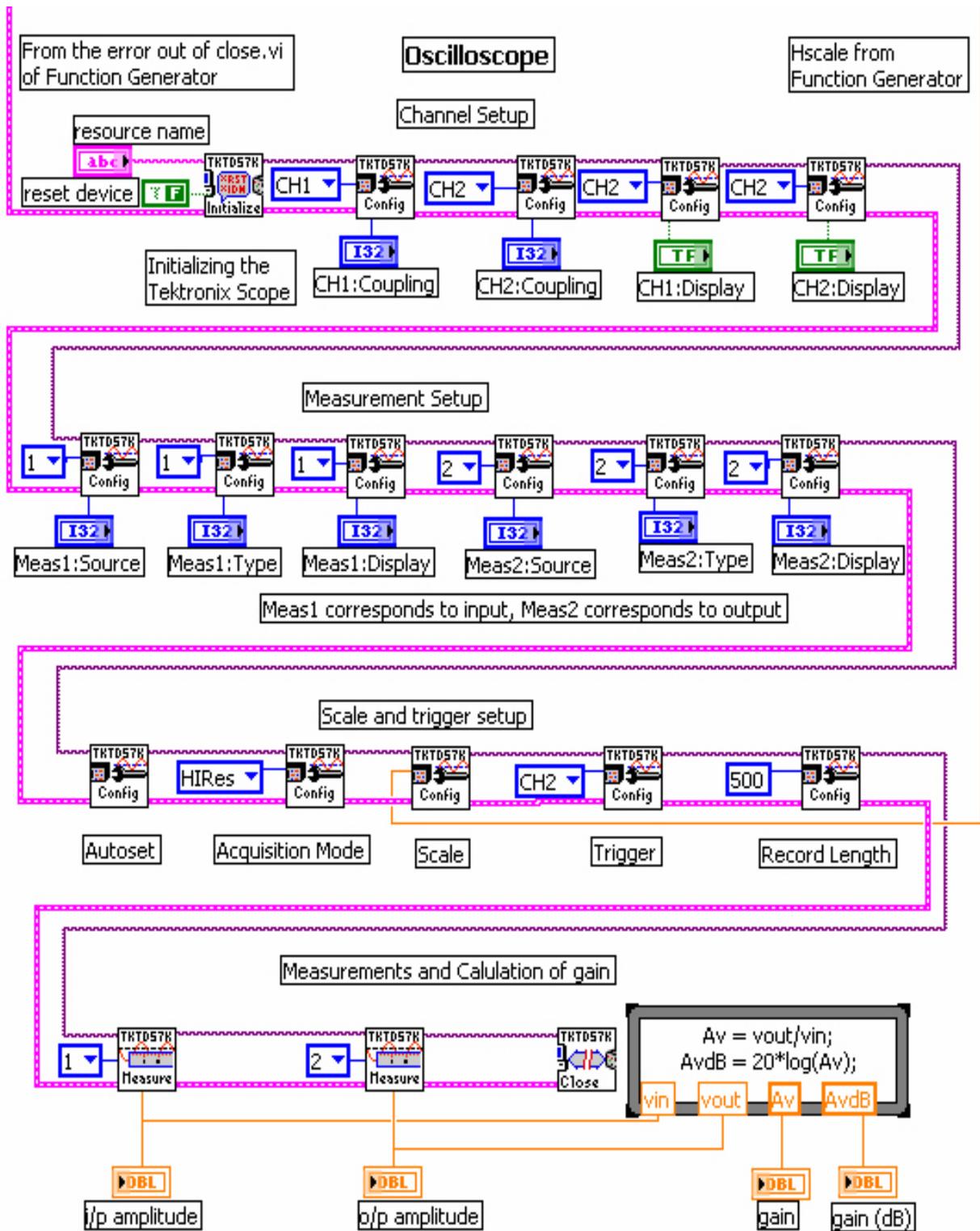


Figure 8: VI part for the oscilloscope

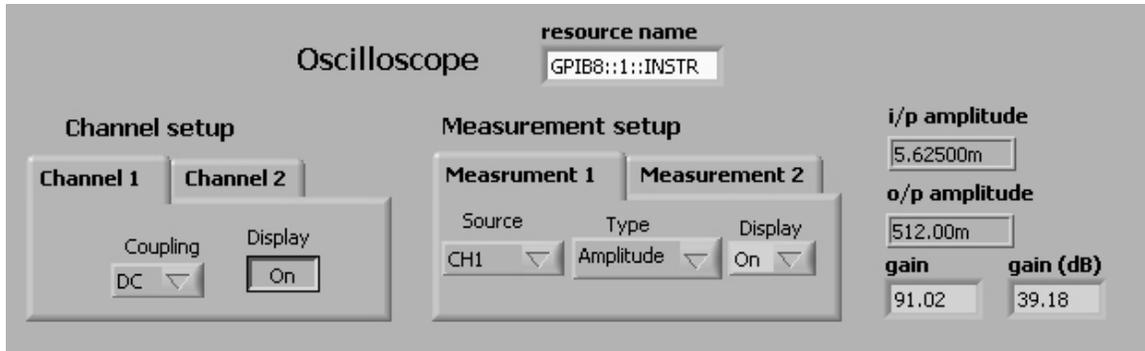


Figure 9: Oscilloscope front panel

Other Features of the Oscilloscope

The built-in, Windows-based platform of the TDS7000 Series oscilloscopes simplifies the transfer of data from the oscilloscope to a personal computer (PC). Waveforms can be exported as bitmap or jpeg files and used directly in reports or presentations using the export command under the File menu. The Select for Export command allows you to specify whether to export numerical waveform and measurement data, or bitmap images of the graticule or entire screen area. Numerical data can also be exported for further analysis. The Export Setup command allows you to choose the file format; bitmap (.bmp) and jpeg (.jpg) formats are available for the waveforms, while .txt, .csv, and .dat formats are available to export numerical data to applications including word document, spreadsheets (Excel, Lotus 1-2-3, Quattro Pro), and mathematical applications (MATLAB, Mathcad). Finally, the Export command allows you to save the file onto the system or disk. These features allow you to document waveforms and test results easily. These Windows-based oscilloscopes integrate the power and flexibility of application software like LabVIEW, LabWindow/CVI, Matlab or high-level application development languages like C++ and Visual Basic, eliminating the need for separate computers.

Lab II: Frequency Response of Common Emitter Amplifier

The traditional way to observe the frequency response of a common emitter amplifier is to apply the input at different frequencies, take measurements and plot the response. The TDS7000 Series' Fast Fourier Transform (FFT) feature and intensity-graded display, coupled with the AFG310/320 arbitrary function generator's sweep mode provide an easy way to observe the frequency response. (Note: This is possible only when the harmonics at a frequency in the range of interest is less than the fundamental at that frequency for all frequencies of interest). The function generator's sweep mode generates signals of different frequencies within a time duration called sweep time. When the function generator output is connected to the setup as shown in the Figure 1, the response of the amplifier to various frequencies can be observed on the oscilloscope using the FFT feature one at a time. The oscilloscope's intensity-graded display can be used to more easily interpret the response of all the frequencies.

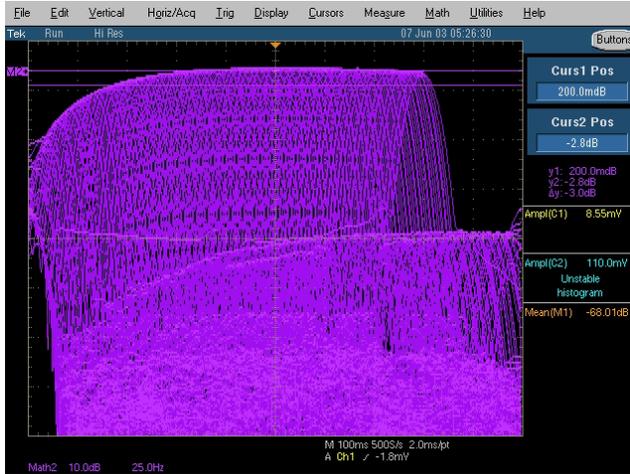
Setting up the Function Generator in Sweep Mode

1. Press MODUL button on the front panel. Use \vee and \wedge buttons to scroll through until SWP (sweep) mode is displayed and press enter.
2. In the sweep mode, enter three parameters: SWP START (start frequency), SWP STOP (stop frequency) and SWP TIME (sweep time). Use \ll and \gg to scroll through until the desired parameters are displayed, enter the parameter and press enter. Enter SWP START as 1Hz, SWP STOP as 200Hz and SWP TIME as 50s.

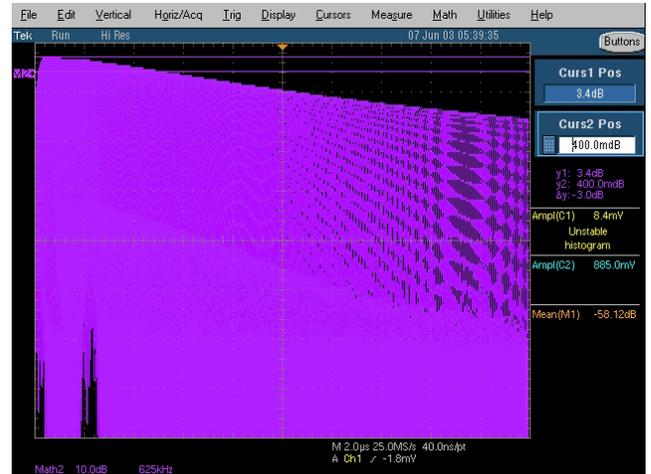
Setting up the Oscilloscope

1. Select Math \rightarrow Spectral Setup.
2. Select Magnitude and the source as channel 2.
3. Enter the sampling rate more than the Nyquist Rate (say 1ks), span frequency as 250Hz and center frequency as 125Hz.
4. Select Display \rightarrow Display Format \rightarrow Infinite Persistence.

The above setup is for the low frequency response. The high frequency response can be obtained similarly by the changing the frequencies appropriately. Figure 10 shows the frequency response of the amplifier.



(a) Low frequency response



(b) High frequency

Figure 10: Frequency response of the amplifier

The automation of this experiment can be done similar to the one shown in the previous experiment by placing the vis responsible for generating input frequency and the measurements in a loop.

Conclusion

This paper demonstrates the use of the TDS7000 Series digital phosphor oscilloscopes to develop analog circuit laboratory experiments. The oscilloscope's measurement and math features allow you to easily measure the gain of the amplifiers. The intensity-graded display, coupled with the sweep mode of the AFG3x0 function generators provides a way to observe the frequency response in a single step. The waveforms, numerical data, and measurements can be moved to the Windows-based platform within the oscilloscope for documentation or further analysis. The Windows-based oscilloscope integrates the applications software into the instruments, which can be used to automate the equipment(s) to perform a sequence of tasks, and analyze and process acquired signals.