HP 35665A Dynamic Signal Analyzer Demo

prepared by Nate Matter
Stanford University
Oct. 15, 2000

1 What is the HP35665A?

Dynamic Signal Analyzers are instruments which process an input signal to give you various information about the signal (as opposed to an oscilloscope, which simply displays the input signal.) All Signal analyzers have an **FFT** mode, where they calculate and display the FFT (Fast Fourier Transform, which is an efficient algorithm for finding the frequency spectrum of a time domain signal.) This is what the HP3561A (on every lab bench) does.

The HP35665A has an additional mode of operation called **Swept Sine** mode. In swept sine mode, the analyzer outputs a pure sinusoid which it sweeps in frequency. The analyzer then measures the amplitudes of the two input signals, divides them, and plots the result. If the source (the swept sine output) is connected to input #1 (in addition to the input of the circuit under test) and the output of the circuit under test is connected to input #2, then the analyzer effectively displays the frequency response of the circuit. The HP35664A is a vector signal analyzer, which means it measures both magnitude and phase, as opposed to a scaler signal analyzer which measures only magnitude.

2 HP35665A Demo

The following instructions demonstrate how to plot the frequency response of a circuit using the HP35665A:

Syntax Key: Words in **Boldface** represent "hard" keys, or buttons on the front panel of the analyzer. Words in *Italics* represent "soft" keys, or the commands represented by F1-F10 as displayed along the right edge of the screen.

1. Turn on the HP35665A. If the analyzer is already on and you would like to start from the default preset mode, you can select **Preset**
2. Connect the Source of the analyzer to both the input of your test circuit and Channel 1.
3. Connect the output of your test circuit to Channel 2.
4. Select **Inst Mode** → *Swept Sine*.
5. Define the source amplitude:
   - Select **Source** → Level.
   - Enter a value for the source amplitude and select the appropriate units using the softkeys. Consider the gain of your circuit and keep the magnitude of Channel 2 less than 1V.
6. Set the frequency range:

- Select **Freq** → *Sweep Log*
- Select **Freq** → *Start*
- Enter the starting frequency for the sweep and select units using the softkeys. (As you use frequencies less than 10 Hz, the measurement will start to take much longer to complete. (Why?)
- Select **Freq** → *Stop*
- Enter the end frequency.

7. Set the input ranges (the input range sets the sensitivity of the inputs–a small input needs to be amplified more than a large input in order to get an accurate A/D measurement):
   - **Input** → *Ch1 Auto Range*.
   - **Input** → *Ch2 Auto Range*.
8. Set the display to show two plots (so you can view both magnitude and phase.) Select **Disp Format** → *Upper/Lower*.
9. Select the top trace using **Active Trace**.
10. Set the top trace to display magnitude:
    - **Meas Data** → *Frequency Response*
    - **Trace Coord** → *dB Magnitude*
    - **Scale** → *Autoscale ON*
11. Set the bottom trace to display the phase by repeating the above step, but choose *Unwrapped Phase* instead of *dB Magnitude*
12. Run the measurement by hitting **Start**
13. Use the dial to move the cursor and take measurements.

3 Plotting and Printing Results in MATLAB

Unfortunately, our current setup leaves no simple way to print your results. If you are willing to take an extra 15 minutes, you can generate very nice plots by using MATLAB:

1. Save your results to a floppy disk:
   - Insert DOS format floppy.
   - **Save/Recall** → *Save Data* → *Save Trace* → *Into File*
   - Enter Filename (use *.dat extension, although this is arbitrary.)
   - *Enter*
2. Convert the data to MATLAB file format:
   - On a computer in the lab, open a DOS prompt and go to `D:\program files\sdftoml\`

- Execute the program which performs the data conversion by typing
  `sdftoml <filein.dat> <fileout.mat> /X`
  The /X option instructs the program to include x-axis data in the matlab file (i.e. include the frequency values associated with each output data point.)

3. FTP your MATLAB data file, `<fileout.mat>` to your leland account.

4. Open a telnet session to your leland account, go to the directory where you saved the data file, open an x-windows session, and execute MATLAB. (If you need help with this step, please ask someone!)

5. View and plot the data in MATLAB: (for information on any MATLAB command, type `help <command name>`)

   - To load the data, type `load <filename.mat>`
   - View the variables contained in MATLAB memory by typing `who`. You should see the names of the two vectors loaded from the data file. One vector has the frequency data (probably called o2i1x) and the other vector has the real/imaginary pair data points which represent the gain data (probably called o2i1).
   - You can view the data by just typing the vector name.
   - Plot the magnitude and phase on the same page by typing the following commands in order:
     - `figure(1)`
     - `subplot(2,1,1)`
     - `semilogx(o2i1x, 20*log10(abs(o2i1)))`
     - `ylabel('Magnitude')`
     - `subplot(2,1,2)`
     - `semilogx(o2i1x, unwrap(angle(o2i1)))`
     - `ylabel('Phase')`
     - `xlabel('Frequency')`
     - `subplot(2,1,1)`
     - `title('Circuit Gain')`
   - You can also execute the above commands by writing them in a text file titled with a .m extension and then type the name of the file in MATLAB. This is useful if you want to generate multiple plots.

6. Print your plot by either going to Sweet Hall or following this procedure in the lab:

   - In the MATLAB plot window, select File → Page Setup and adjust the settings as you please.
   - Select File → Print
   - In the print window, check the box to print to a file and execute. MATLAB will save the plot to a PostScript file.
   - FTP the printed file to a lab computer.
   - View and print the file using Ghostview.