Errata

Title & Document Type: 35660A Dynamic Signal Analyzer Getting Started Guide

Manual Part Number: 35660-90005

Revision Date: July 1988

About this Manual

We've added this manual to the Agilent website in an effort to help you support your product. This manual provides the best information we could find. It may be incomplete or contain dated information, and the scan quality may not be ideal. If we find a better copy in the future, we will add it to the Agilent website.

HP References in this Manual

This manual may contain references to HP or Hewlett-Packard. Please note that Hewlett-Packard's former test and measurement, life sciences, and chemical analysis businesses are now part of Agilent Technologies. The HP XXXX referred to in this document is now the Agilent XXXX. For example, model number HP8648A is now model number Agilent 8648A. We have made no changes to this manual copy.

Support for Your Product

Agilent no longer sells or supports this product. You will find any other available product information on the Agilent Test & Measurement website:

www.agilent.com

Search for the model number of this product, and the resulting product page will guide you to any available information. Our service centers may be able to perform calibration if no repair parts are needed, but no other support from Agilent is available.

HEWLETT PACKARD

HP 35660A Dynamic Signal Analyzer
Getting Started Guide

Manual Part No. 35660-90005
Microfiche Part No. 35660-90205

Printed: July 1988

HEWLETT PACKARD

CERTIFICATION

Hewlett-Packard Company certifies that this product met its published specifications at the time of shipment from the factory. Hewlett-Packard further certifies that its calibration measurements are traceable to the United States National Bureau of Standards, to the extent allowed by the Bureau's calibration facility, and the calibration facilities of other International Standards Organization Members.

WARRANTY

This Hewlett-Packard product is warranted against defects in material and workmanship for a period of one year from date of shipment. During the warranty period, Hewlett-Packard Company will, at its option, either repair or replace products which prove to be defective.

For warranty service or repair, this product must be returned to a service facility designated by -hp-. Buyer shall prepay shipping charges to -hp- and -hp- shall pay shipping charges to return the product to Buyer. However, Buyer shall pay all shipping charges, duties, and taxes for products returned to -hp- from another country.

HP software and firmware products which are designated by HP for use with a hardware product, when properly installed on the hardware product, are warranted not to fail to execute their programing instructions due to defects in materials and workmanship. If HP receives notice of such defects during their warranty period, HP shall repair or replace software media and firmware which do not execute their programming instructions due to such defects. HP does not warrant that the operation of the software, firmware or hardware shall be uninterrupted or error free.

LIMITATION OF WARRANTY

The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by Buyer, Buyer-supplied software or interfacing, unauthorized modification or misuse, operation outside of the environmental specifications for the product, or improper site preparation or maintenance

NO OTHER WARRANTY IS EXPRESSED OR IMPLIED. HEWLETT-PACKARD SPECIFICALLY DISCLAIMS THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR PARTICULAR PURPOSE.

EXCLUSIVE REMEDIES

THE REMEDIES PROVIDED HEREIN ARE BUYER'S SOLE AND EXCLUSIVE REMEDIES. HEWLETT-PACKARD SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER BASED ON CONTRACT, TORT, OR ANY OTHER LEGAL THEORY.

ASSISTANCE

Product maintenance agreements and other customer assistance agreements are available for Hewlett-Packard products.

For any assistance, contact your nearest Hewlett-Packard Sales and Service Office. Addresses are provided at the back of this manual.

HEWLETT PACKARD

SAFETY SUMMARY

The following general safety precautions must be observed during all phases of operation, service, and repair of this instrument. Failure to comply with these precautions or with specific warnings elsewhere in this manual violates safety standards of design, manufacture, and intended use of the instrument. Hewlett-Packard Company assumes no liability for the customer's failure to comply with these requirements. This is a Safety Class 1 instrument.

GROUND THE INSTRUMENT

To minimize shock hazard, the instrument chassis and cabinet must be connected to an electrical ground. The instrument is equipped with a three-conductor ac power cable. The power cable must either be plugged into an approved three-contact electrical outlet or used with a three-contact to two-contact adapter with the grounding wire (green) firmly connected to an electrical ground (safety ground) at the power outlet. The power jack and mating plug of the power cable meet International Electrotechnical Commission (IEC) safety standards.

DO NOT OPERATE IN AN EXPLOSIVE ATMOSPHERE

Do not operate the instrument in the presence of flammable gases or fumes. Operation of any electrical instrument in such an environment constitutes a definite safety hazard.

KEEP AWAY FROM LIVE CIRCUITS

Operating personnel must not remove instrument covers. Component replacement and internal adjustments must be made by qualified maintenance personnel. Do not replace components with power cable connected. Under certain conditions, dangerous voltages may exist even with the power cable removed. To avoid injuries, always disconnect power and discharge circuits before touching them.

DO NOT SERVICE OR ADJUST ALONE

Do not attempt internal service or adjustment unless another person, capable of rendering first aid and resuscitation, is present.

DO NOT SUBSTITUTE PARTS OR MODIFY INSTRUMENT

Because of the danger of introducing additional hazards, do not install substitute parts or perform any unauthorized modification to the instrument. Return the instrument to a Hewlett-Packard Sales and Service Office for service and repair to ensure the safety features are maintained.

DANGEROUS PROCEDURE WARNINGS

Warnings, such as the example below, precede potentially dangerous procedures throughout this manual. Instructions contained in the warnings must be followed.

WARNING

Dangerous voltages, capable of causing death, are present in this instrument. Use extreme caution when handling, testing, and adjusting.

HEWLETT PACKARD

SAFETY SYMBOLS

General Definitions of Safety Symbols Used On Equipment or In Manuals.

⚠	Instruction manual symbol: the product will be marked with this symbol when it is necessary for the user to refer to the instruction manual in order to protect against damage to the instrument.
ϟ	Indicates dangerous voltage (terminals fed from the interior by voltage exceeding 1000 volts must be so marked.)
⏚ OR ⊜	Protective conductor terminal. For protection against electrical shock in case of a fault. Used with field wiring terminals to indicate the terminal which must be connected to ground before operating equipment.
⊜	Low-noise or noiseless, clean ground (earth) terminal. Used for a signal common, as well as providing protection against electrical shock in case of a fault. A terminal marked with this symbol must be connected to ground in the manner described in the installation (operating) manual, and before operating the equipment.
⫛ OR ⊥	Frame or chassis terminal. A connection to the frame (chassis) of the equipment which normally includes all exposed metal structures.
∼	Alternating current (power line).
⎓	Direct current (power line).
≂	Alternating or direct current (power line).

WARNING *The WARNING sign denotes a hazard. It calls attention to a procedure, practice, condition or the like, which if not correctly performed or adhered to, could result in injury or death to personnel.*

CAUTION *The CAUTION sign denotes a hazard. It calls attention to an operating procedure, practice, condition or the like, which, if not correctly performed or adhered to, could result in damage to or destruction of part or all of the product.*

NOTE The NOTE sign denotes important information. It calls attention to procedure, practice, condition or the like, which is essential to highlight.

Table of Contents

Chapter 1
Before You Begin

Please take a moment to read this introduction. Then go to Chapter 2, "Your First Measurement," to get comfortable with your new analyzer.

The Hewlett-Packard 35660A Dynamic Signal Analyzer helps you test, analyze, and design DC to 100 kHz electronic, electro-mechanical, and mechanical systems. If you've never used an FFT (Fast Fourier Transform) Dynamic Signal Analyzer before, you might think one difficult to use. Don't worry. Many measurements will be familiar to you — some are the same measurements made with swept-tuned analyzers.

How to Use this Book

Before you start using the analyzer, take a few minutes to read *Part I — The Basics*. This tells you what each measurement is all about — and a few hints that will help as you begin to use your analyzer. Even if you've used an FFT dynamic signal analyzer before, you might find a brief review of the basics useful.

If you already understand basic FFT measurements, you might want to proceed directly to the easy-to-follow measurement tasks. These are in two sections: *Part II — Making Spectrum Measurements* and *Part III — Making Network Measurements*. Each task steps you through a typical measurement procedure. To really learn how to use the analyzer, it's best if you gather the necessary equipment (outlined at the beginning of each task) and actually set up and make the measurements. The list of measurement tasks is by no means exhaustive — but after stepping through most of them, you'll have a good idea of what the analyzer can do for you.

After you've looked through the example measurements, spend some time with *Part IV — Beyond the Basics*. Here's where you'll learn about the analyzer's more sophisticated features.

And of course use, the *index* to quickly locate the information you need.

Where to find Additional Information

For quick information about specific hardkeys and softkeys, see the *HP 35660A Front-Panel Reference*. This book also contains softkey menu maps and a more detailed description of the analyzer's front panel.

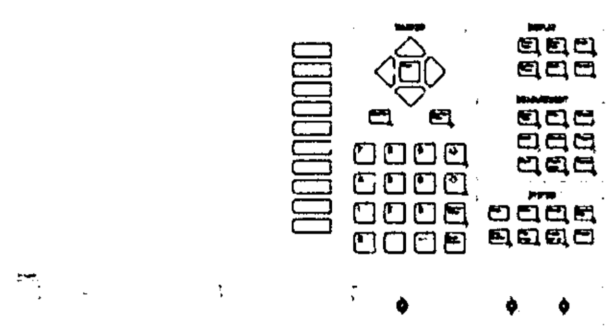
For specifications, installation instructions, and performance tests, see the *HP 35660A Installation Guide*.

To help you operate the analyzer remotely via HP-IB, see the *HP 35660A Programming Reference*.

Additionally, you will find applications information in numerous Hewlett-Packard Application Notes. These are available from your local HP Sales and Service Office. In particular, you might want to request a copy of the following application notes:

- AN 243 — The Fundamentals of Signal Analysis
- AN 243-1 -- Effective Machinery Maintenance Using Vibration Analysis
- AN 243-2 — Control System Development Using Dynamic Signal Analyzers
- AN 243-3 — The Fundamentals of Modal Testing

About the Analyzer



The HP 35660A Dynamic Signal Analyzer is really two instruments in one — a network analyzer and a spectrum analyzer. You can make network or two-channel spectrum measurements, from 244 μHz to 51.2 kHz; or single-channel spectrum measurements from 488 μHz to 102.4 kHz. There's also a built-in signal source with choice of random noise, periodic chirp (fast sine sweep), or fixed sine wave.

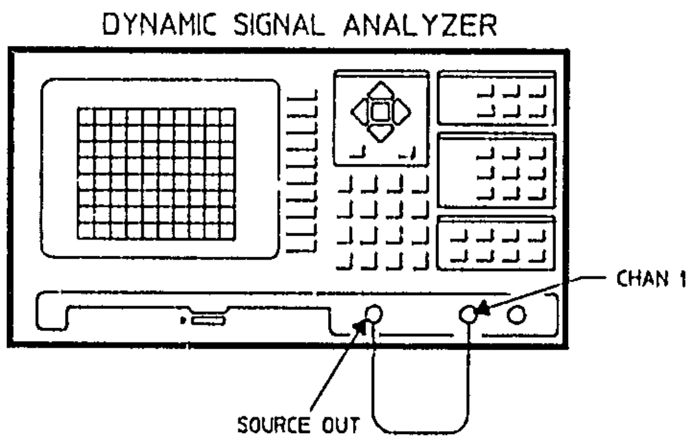
The analyzer has three connectors on the front panel. One connector is a signal source. The others two connectors are the channel 1 input and the channel 2 input. On the back panel, there's a connector for an external trigger and one for the HP-IB. To learn more about these connectors, see the *HP 35660A Front-Panel Reference*.

Chapter 2
Your First Measurement

If you haven't used the analyzer before, take a few minutes to make this first measurement. In this measurement, you will do the following:

- Look at the averaged power spectrum of a 1 kHz sine wave
- Check to see if the sine wave is really at 1 kHz
- Look for any harmonics of the fundamental frequency

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer — not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >

Pressing < Preset > returns most of the analyzer settings to the default positions.

2. Connect the analyzer's source to the Channel 1 input.

You do not need to terminate the analyzer's source, since the output impedance is less than 5Ω.

The analyzer's input channels have an impedance of 1 MΩ.

In this example you are using the analyzer's internal source as the test device. However, to test external signal sources designed to operate into a specific load (such as an oscillator with a 600Ω output) you must place an appropriate feedthrough terminator across the output of the test device.

3. Press < Source >

   [SOURCE ON/OFF]

   [FIXED SINE]

This turns on the analyzer's internal source and selects the fixed sine wave.

4. Press [SINE FREQ ENTRY]

   < 1 > [kHz]

This sets the sine frequency to 1 kHz.

5. Press [LEVEL]

   < 1 > [Vrms]

This sets the level of the sine wave to 1 Vrms.

6. Now look at the analyzer's screen. This is a display of the linear spectrum.

   This display appears in frequency domain.

Because this is a full span (0 to 102.4 kHz), the relatively low frequency of the source waveform (1 kHz) is at the extreme left of the display.

Because averaging is off, you will see the display change several times each second. Each display represents one FFT of a single time record.

7. Press < Freq >

   [SPAN]

   Now use the < ⇩ > hardkey to step through several spans.

   Stop when you reach 3.2 kHz — if you step down too far, simply use the < ⇧ > hardkey to go back up to 3.2 kHz.

The < ⇩ > and < ⇧ > hardkeys are located in the numeric keypad.

This changes the frequency span and lets us look at a smaller slice of the frequency spectrum. This gives a better view of the fundamental and the first two harmonics.

You can also use the numeric keypad to specify a span (the analyzer takes the nearest acceptable value).

7a. If you don't see any harmonics, press < Input > [CHANNEL 1 RANGE]. Then press < ⇩ > twice.

This intentionally overloads the analyzer's input to simulate a source with prominent harmonics.

8. Press < Average >

   [AVERAGE ON/OFF]

   The analyzer should start an averaged measurement right away.

Note how the word "ON" in this softkey label highlights when averaging is on.

The default averaging is rms averaging (with ten averages). For now, we'll stay with this type of average.

9. Press < Scale >

   [VERTICAL/DIV]

   < 1 > < 2 > [dB]

   This selects a vertical scale of 12 dB per division.

We've changed the scale to view both the top of the fundamental and the noise floor.

10. Press < Start >

Note how another averaged measurement begins.

11. Press < Marker >

    [MARKER TO PEAK]

This moves the marker to the largest frequency component on the display — in this case, the fundamental.

12. Note the frequency value indicated by the marker's x-axis position.

This value should be 1 kHz. This verifies that the sine source is indeed set at 1 kHz.

13. Note the amplitude value indicated by the marker's y-axis position.

This shows the absolute amplitude of the fundamental. In this case, the y-axis marker value indicates about 0 dBVrms — equivalent to about 1 Vrms.

14. Press the < ▲ > hardkey several times, until the marker moves to 3 kHz.

This moves the marker to the third harmonic.

The < ▲ > hardkey is the unlabeled key shaped like an up arrow in the MARKER group of front-panel keys.

Pressing < ▲ > jumps the marker to the next big peak to the right. Pressing < ▼ > works the same way, but jumps the marker to the left.

You can also use the < ◄ > and < ► > keys to move the marker.

15. Note the frequency value indicated by the marker's x-axis position.

This value should be about 3 kHz. Not surprising, since the third harmonic should be three times the fundamental's frequency.

16. Note the amplitude value indicated by the marker's y-axis position.

This value is about −70 dBVrms. Since the fundamental is about 0 dBVrms, the third harmonic is about 70 dB below the fundamental.

Chapter 3
Measurement Basics

Time Domain vs. Frequency Domain

If you haven't used a network analyzer or spectrum analyzer before, it's important to understand the difference between time-domain displays and frequency-domain displays.

Time-domain displays show a parameter (such as amplitude) versus *time*. This is the traditional way of looking at a signal. Oscilloscopes display signals in the time domain.

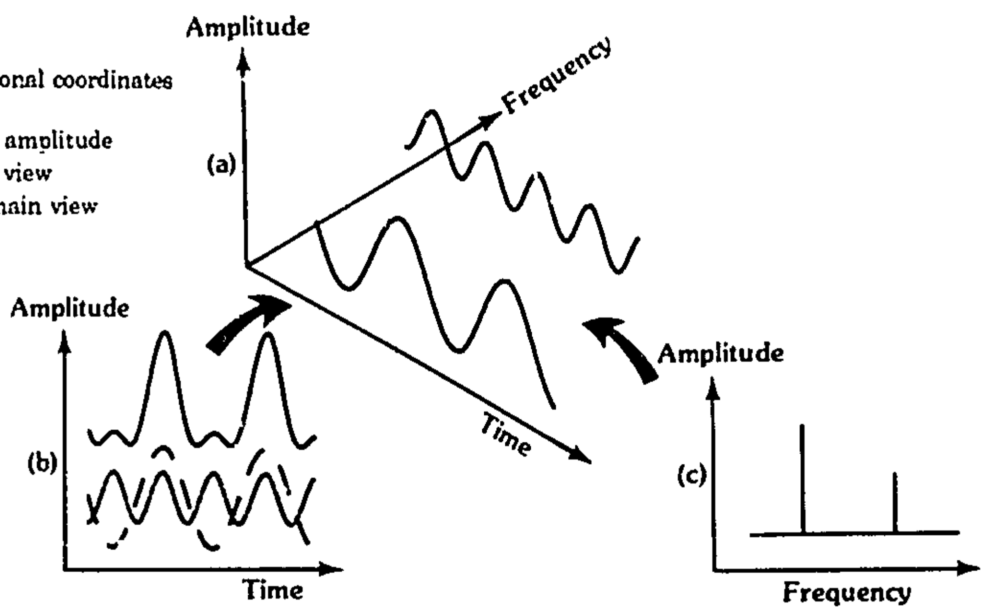
Frequency-domain displays show a parameter (again, such as amplitude) versus *frequency*. The analyzer uses an FFT (Fast Fourier Transform) algorithm to convert an analog input signal — a time-domain signal — to a signal displayed in the frequency domain. The tremendous advantage of frequency-domain displays is they can reveal very small signals not visible in time-domain displays — signals such as noise and distortion products.

All analyzer measurements appear in the frequency domain except time records.

One type of measurement that appears in the frequency domain is a spectrum measurement. Spectrum measurements show the energy of each frequency component at sampled points along the frequency spectrum. Now look at the figure and note the difference between the time-domain and frequency-domain displays of the same input signal.

The Relationship Between the Time and Frequency Domains.

(a) Three-dimensional coordinates showing time, frequency and amplitude
(b) Time domain view
(c) Frequency domain view



The Frequency Span

You can vary the size and the center frequency of the span to best suit your measurement needs. The HP 35660A Dynamic Signal Analyzer always presents data with a 401-point resolution – even when viewing very small spans. The tremendous advantage of FFT signal analyzers is that they have good frequency resolution for smaller frequency spans (they also make these measurements much more quickly than swept-tuned analyzers). And you can use much smaller spans with FFT analyzers as well.

Full-span measurements let you view the entire frequency spectrum on one display. For one-channel measurements, the spectrum will extend from dc to 102.4 kHz. For two-channel measurements, the spectrum will extend from dc to 51.2 kHz.

Alternatively, you may wish to view smaller slices of the frequency spectrum. You can select one of twenty different spans and position these spans where you want by specifying their start or center frequencies. This process of viewing smaller spans is sometimes called "band-selectable analysis." Measurements with spans that start at 0 Hz are called "baseband" measurements – those with spans that start at frequencies other than 0 Hz are called "zoomed" measurements.

There's more you should know about selecting an appropriate frequency span. We'll cover that later in this chapter.

First, the FFT

The Fast Fourier Transform (FFT) is a an implementation of the Discrete Fourier Transform, the math algorithm used for transforming data from the time domain to the frequency domain. Before the analyzer uses the FFT algorithm, it samples the input signal with an analog-to-digital converter (the Nyquist sampling theorem states that if samples are taken twice as fast as the highest frequency component in the signal, the signal can be reconstructed exactly). This transforms the continuous (analog) signal into a discrete (digital) signal.

Because the input signal is sampled, an *exact* representation of this signal is not available in either the time domain or the frequency domain. However, by spacing the samples closely, the analyzer provides an excellent approximation of the input signal.

The analyzer display appears to be one continuous trace. However, the display is really 401 discrete points connected together. Each point is called a frequency bin (or just bin for short).

Can the Analyzer Measure DC?

The analyzer is not designed to measure dc. However, it is designed to measure very low frequencies — as low as 244 μHz for two-channel measurements and 488 μHz for one-chennel measurements. The analyzer can, in fact, measure dc, but not without including a dc offset of its own that can contribute to (or obscure) a dc offset in the input signal. This internal offset is caused by residual dc that originates in the analyzer's input amplifiers. Thus, dc measurements are not guaranteed to be accurate.

As you use the analyzer, you will notice a dc offset when making baseband measurements (those with spans that start at 0 Hz). This offset is always present in the 0 Hz bin (sometimes called the dc bin). The feedthrough that cause the offset may also leak into the first several bins as well. If this is a problem, start the frequency span several bins above 0 Hz to avoid the feedthrough.

The Time Record

A time record is the amount of time-domain data the analyzer needs to perform one FFT operation. The time record and its FFT are the building blocks the analyzer needs for all subsequent measurements.

The analyzer takes 1024 samples of time data to produce 512 points of frequency domain data. The analyzer usually displays the first 401 points of this data and discards the rest (this accommodates the anti-aliasing filters, but that's beyond the scope of our current discussion).

The relationship between a time record and the frequency data is relatively straightforward. If a signal component completes one cycle within the time record, it will show up in the first frequency bin (the first point on the analyzer's display). If a component takes two cycles to complete, it will show up in the second bin. And so forth.

So if a time record is 1 second long (and you start at 0 Hz), then the period of the signal for the first bin is also 1 second. And its frequency (1/period) is 1 Hz. Since there are 401 bins displayed, the span will be 400 Hz. The effective sampling frequency is simply 1024 divided by the length of the time record.

Why a Time Record?

Essentially, the time record is a block of time-domain sample points. Now since the actual Fourier Transform does not have explicit time or frequency references (it simply operates on a sequential collection of points), FFT analyzers must assign arbitrary start and finish times for data to be transformed. These blocks of input data are called time records.

Measurement Speed vs. Time Record Length

Frequency Span (Hz)	Time Record Length (sec)	Resolution * (Hz)
102400	.00390625	256
51200	.0078125	128
25600	.015625	64
12800	.03125	32
6400	.0625	16
3200	.125	8
1600	.25	4
800	.5	2
400	1	1
200	2	.5
100	4	.25
50	8	.125
25	16	.0625
12.5	32	.03125
6.25	64	.015625
3.125	128	.0078125
1.5625	256	.00390625
.78125	512	.001953125
.390625	1024	.0009765625
.1953125	2048	.00048828125

* **Frequency Span/400**

The size of a time record is inversely proportional to the frequency span. So for smaller spans, the analyzer needs a longer time record and therefore takes longer to make a measurement. For larger spans, the analyzer needs a shorter time record and can therefore make a measurement much faster. These differences will become noticeable as you start making measurements. This characteristic is a natural part of the FFT process and is common to *all* FFT analyzers, not just the HP 35660A. By the way, swept-tuned analyzers have similar limitations (and are, in fact, much slower than FFT analyzers for comparable measurements).

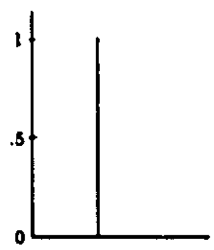
A dB Scale for the Y-Axis

Time-domain displays usually have a linear y-axis and a linear x-axis (think of an oscilloscope). However, frequency-domain displays must often use a *logarithmic* y-axis scale to show small signals with large signals.

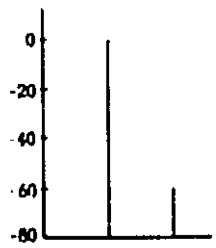
Let's look at the spectrum of a sine wave. Because the amplitude of any harmonic is small relative to the fundamental frequency, it's nearly impossible to view a harmonic on the same display as the fundamental unless the y-axis scale is logarithmic. So most magnitude measurements made with dynamic signal analyzers use a logarithmic y-axis scale with units based on decibels (dB). Because the dB scale is by definition logarithmic, there's no need to use logarithmically-spaced graticule lines.

The dB scale is convenient. It is also the scale you will probably use for most magnitude displays — particularly with spectrum or frequency response measurements.

Small Signals Can Be Measured with a Logarithmic Amplitude Scale.



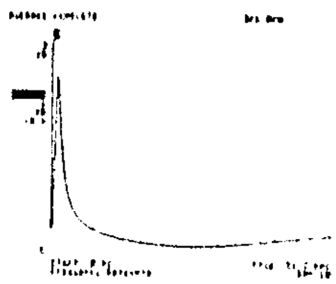
(a) Linear Amplitude Scale

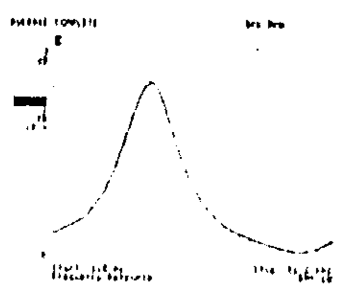


(b) Logarithmic Amplitude Scale

A Logarithmic Scale for the X-Axis

Sometimes it's convenient to use a logarithmic x-axis. Perhaps most familiar to you is the frequency response measurement. This is traditionally displayed with a log x-axis (frequency) versus a log y-axis (relative magnitude).





But most measurements do not require a logarithmic frequency scale. In fact, when making spectrum measurements it's easier to characterize harmonics with a linear x-axis scale since harmonics that are multiples of the same fundamental will appear at evenly-spaced intervals.

Here's what else you should know:

- The analyzer's frequency resolution is determined exclusively by the width of the span. So for the same span widths, frequency resolution for both linear and log scales is identical – both have a resolution of 401 points per display. The logarithmic scale simply displays these points on a logarithmic x-axis.
- For baseband measurements (spans that start at 0 Hz) the logarithmic scale shows the actual sta.t frequency (the first bin) of the current span – not the nominal value of 0 Hz. So if you're looking at a 51.2 kHz frequency span, the first frequency shown on the logarithmic scale will be labeled 128 Hz (the analyzer does not show a value at 0 Hz since the log of 0 is minus infinity). As you would for a linear scale, change to smaller span to view lower-frequency components.

Measurement Type

Changing the analyzer from a spectrum analyzer to a network analyzer (or two-channel spectrum analyzer) is easy. Simply press < Meas Type > and select either [1 CHANNEL 102.4 kHz] or [2 CHANNEL 51.2 kHz]. One-channel measurements are spectrum measurements. Two-channel measurements can be spectrum measurements with two channels *or* network measurements.

Use the following matrix to help you select an appropriate measurement. Notice how some measurements (such as frequency response) are available only when the analyzer is operating as a two-channel analyzer. Press < Meas Data > and you'll see a menu listing the available measurements.

Measurement Type

Measurement	One Channel	Two Channel
Spectrum CH1	Yes	Yes
Spectrum CH2	No	Yes
PSD CH1	Yes	Yes
PSD CH2	No	Yes
Time CH1	Yes	Yes
Time CH2	No	Yes
Frequency Resp.	No	Yes
Coherence	No	Yes (average on)
Cross Spectrum	No	Yes

The measurements are as follows:

- Spectrum
- Power Spectral Density (PSD)
- Time record
- Frequency Response
- Coherence
- Cross Spectrum

Trace Type

Once you've made a measurement there are a number of ways to display the measurement data. The measurement data is called a trace. Press < Trace Type > to see the available measurement traces. On some dynamic signal analyzers, these trace types are called "coordinate types."

It's important to understand that selecting a measurement and selecting a trace type are two different things. When you specify the measurement type, you are asking the analyzer to acquire input data and process it. When you select a trace type, you are specifying how you want the processed measurement data displayed. First select a measurement; then specify a trace type.

The logarithmic magnitude display is the most common way to view measurement data. However, the other trace types are also useful and can reveal information not visible from the traditional log magnitude display.

These are the trace types:

- Linear Magnitude
- Logarithmic Magnitude
- Phase
- Group Delay
- Real Part
- Imaginary Part

Spectrum Measurements

Softkeys: [SPECTRUM CHANNEL 1] and [SPECTRUM CHANNEL 2]

Linear Spectrum or Power Spectrum?

The term "spectrum measurement" is used to describe several different measurements. These include linear spectrum, averaged linear spectrum, and power spectrum.

A *linear spectrum* is the most basic spectrum measurement. This is the FFT of a single time record. The analyzer filters the input data (to the desired frequency span) and then performs an FFT on a single time record. The resulting display is in the frequency domain and shows the spectral content of the input signal.

An *averaged linear spectrum* is a vector-averaged linear spectrum. With vector averaging, the analyzer averages complex values, point-by-point, in the frequency domain. This lowers noise because the real and imaginary components of the random signals are not in phase and therefore cancel each other – increasingly so with each average. Frequency components that are periodic do not cancel and therefore do not diminish with successive averages. Vector averaging produces results similar to time averaging (a feature found on many FFT analyzers). You'll learn more about vector averaging in Chapter 4, "More Basics."

A *power spectrum* is an rms-averaged linear spectrum. The analyzer filters the input data (to the desired frequency span). It then performs an FFT on each time record, and multiplies each resulting linear spectrum by its complex conjugate. The final results are rms averaged to a single spectrum – the power spectrum.

Should I use Linear Spectrum or Power Spectrum?

Use linear spectrum measurements (both single and averaged) if you need single-channel phase information. Additionally, use averaged linear spectrum measurements when you want to reduce noise as much as possible. With enough averages, linear spectrum measurements have noise levels approaching zero, limited only by the dynamic range of the analyzer.

Use power spectrum measurements if you're interested in rms values of frequency components and noise. Since the power spectrum does not *reduce* noise but simply provides a good approximation of the *actual noise level*, it's an ideal measurement for characterizing the performance of many electronic devices – particularly for audio-frequency and communications applications.

Spectrum Measurements (continued)

Linear Spectrum for Single-channel Phase Information

The linear spectrum can reveal single-channel phase relationships. We'll see why this is useful in a moment. But first, let's discuss single-channel phase.

When we talk about phase, we usually associate it with two-channel measurements (such as frequency response). That's because in the traditional sense, phase is used to indicate time relationships between *two* signals – usually input and output – measured at the device under test. This produces a phase trace showing relative phase differences between the two signals at every point in the selected frequency span. And the HP 35660A can, of course, make this type of measurement. But there's also single-channel phase.

Instead of using phase to show time relationships between frequency components of two signals, we can use phase to show time relationships between individual frequency components of one signal and a *fixed time reference* (an external trigger signal). This is useful when you're trying to determine the relative phase of a particular component with respect to other frequency components – a situation common to vibration measurements.

However, linear spectrum measurements will not yield meaningful phase information unless you've met these conditions:

- *The input signal is periodic.* In other words, the input signal must repeat continuously. This is not a problem for rotating machinery measurements, since the measured signal (the machine's "frequency signature") repeats with each rotation.
- *The analyzer has a trigger signal with a fixed relation to the input signal.* For rotating machinery measurements, you'll need to provide an external trigger signal from a tachometer, proximity probe, or other device. A reliable trigger signal is important, since phase is measured relative to the trigger occurrence. The trigger signal is at the beginning of the time record (unless you've selected a pre- or post-trigger delay) but the actual timing is not critical as long as the trigger is consistent.

Linear Spectrum for Lower Noise Levels

Averaged linear spectrum measurements are sometimes used because they have better signal-to-noise ratios than power (rms) spectrum measurements. This is because linear spectrum measurements use vector averaging instead of rms averaging. With enough vector averages, the input noise level approaches zero – limited only by the analyzer's dynamic range.

But linear spectrum measurements are not just for mechanical measurements. You can also take advantage of their better signal-to-noise ratios for electrical measurements. If the input signal is periodic and there's a good trigger signal, you can use the linear spectrum. Though many linear spectrum measurements require an *external* trigger, you can also use input triggering – provided the input waveform will trigger the analyzer consistently.

Spectrum Measurements (continued)

Power Spectrum to Find RMS Average

Power spectrum measurements show rms values for frequency components (and noise) of the input signal. This is the most common measurement used for analyzing the spectrum of audio-frequency devices and communications equipment.

Power spectrum measurement do not contain any phase information. This information is lost during the transition from linear spectrum to power spectrum. But most measurements that call for rms power do not require phase information.

Let's look at an example. You can look at the power spectrum of an oscillator to determine rms values for the fundamental frequency and harmonics – measured in relative terms (such as dB) or absolute levels (such as dBm). You can then use the analyzer's marker functions to measure noise and harmonic distortion.

Remember, rms averaging does not eliminate noise. It simply produces a statistical average of the input signal including noise. Additional averages provide a better statistical average, but will not actually *reduce* noise.

With rms averaging, the magnitude of individual frequencies also includes noise. So for very small signals, noise can add significantly to that component's magnitude. If you want to reduce that noise, you'll have to use linear averaging instead of power averaging – but remember that magnitude values will no longer be rms values.

We'll learn more about averaging in the next chapter.

PSD (Power Spectral Density) Measurements

Softkeys: [PSD CHANNEL 1] and [PSD CHANNEL 2]

What is PSD?

PSD (Power Spectral Density) is similar to the power spectrum measurement, but the analyzer normalizes each component of the power spectrum to produce a display with values normalized to 1 Hz. Think of it this way – if you place the marker at a particular frequency, the marker value will show a value that *approximates the power within a 1 Hz band centered at the marker value*. This is true regardless of the frequency span you've selected.

PSD is sometimes called "noise density" or "spectral density," but the full name is "power spectral density."

Why Use It?

The magnitude of a measurement of white noise is proportional to the bandwidth the analyzer uses. So to make *comparative* noise measurements, the analyzer must use the same bandwidth to examine energy throughout the entire frequency span of interest. That's why PSD is so useful, since it uses a standard analysis bandwidth of 1 Hz.

Why Normalize to a 1 Hz Bandwidth?

Traditionally, swept-tuned analyzers used a tunable filter with a 1 Hz wide filter to produce a display with a resolution of 1 Hz. After a while, power spectrum measurements with a 1 Hz bandwidth became an industry standard.

However, FFT analyzers do not use tunable filters. In fact, the bandwidth at each frequency point varies with the frequency span and the window you've selected. So to simulate a 1 Hz bandwidth at each frequency point (401 points per display), the analyzer uses an algorithm that divides by the square root of the actual bandwidth. The algorithm also corrects for the type of window you're using.

Noise measurements made using PSD will approximate actual 1 Hz bandwidth measurements only if the noise is Gaussian (white noise).

PSD Measurements (continued)

What Do I Measure with PSD?

For electrical measurements, PSD can be used to make standardized noise measurements. For example, you can measure signal to the PSD of a typical point on the noise floor. This method yields a better apparent signal-to-noise ratio than referencing the test signal to wide-band noise, but may be preferable in cases where you want repeatable noise measurements referenced to a common standard.

PSD is also used to look at phase noise of high-frequency oscillators (such as microwave and radar) after these signals are mixed with a reference oscillator to bring the spectral components within the analyzer's range.

For mechanical measurements, PSD is routinely used to measure the energy of noise or other spectral components. The standard 1 Hz bandwidth allows meaningful comparison to measurements made with other analyzers (not necessarily FFT analyzers).

Time Record Measurements

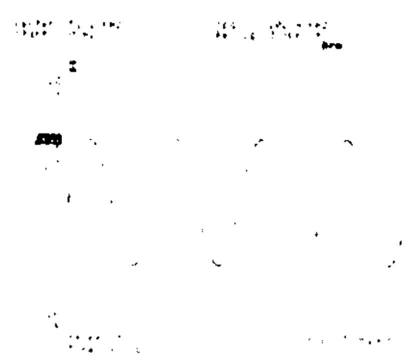
Softkeys: [TIME CHANNEL 1] and [TIME CHANNEL 2]

Time record displays appear in the time domain – that's why they look like oscilloscope traces. A time record is the amount of time-domain data the analyzer needs to perform one FFT operation. The time record and its FFT are the basic "building blocks" the analyzer needs for all subsequent measurements.

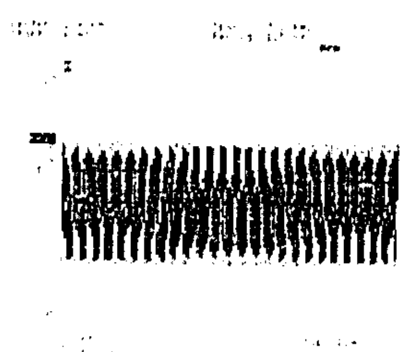
Time records are not calibrated, so they display only an approximate amplitude value. Still, time records are very useful. They show input data before the analyzer does any FFT processing.

Here's what you should know about input records:

- If you set the instrument to measure full span, the time record is called an "input time record." This is raw, unfiltered input data – the signal from which all subsequent measurements are based. Use the input time record to verify that there is indeed a signal. Additionally, you can use the time record when manually setting the input range.
- If you set the instrument to measure a specific bandwidth (something less than full span), the time record shows the raw input data after filtering. This lets you see if there's energy within the selected span.
- If the analyzer is making averaged measurements, the most recent time record added to the average is the one displayed. The analyzer does not show a time waveform that is a cumulative average, since all averaging is done after the time data has been transformed to the frequency domain.
- For zoomed time record displays (start frequency not equal to zero), the displayed amplitude is approximately one-half the actual amplitude.
- Although the time record is similar to an oscilloscope display, the analyzer is *not* a digital oscilloscope. The time record represents *samples* of a waveform. The samples have enough information to accurately reconstruct the input signal – but the human eye may not properly perform the reconstruction. In fact, for frequencies that are higher than about 10% of the frequency span, there will be noticeable *visible* distortion. However, in no way does this affect the accuracy of the measurement made from the time record.



Time Record of 1 kHz Sine Wave (102.4 kHz Span)



Time Record of 1 kHz Sine Wave (3.2 kHz Span)

Frequency Response Measurements

Softkey: [FREQUENCY RESPONSE]

Frequency response is one of the most useful measurements that two-channel analyzers perform. Frequency response shows how a system (a "network.") will respond to a particular input. The network might be electrical (a filter, for example) or mechanical (a model airplane in a wind tunnel).

Frequency response measurements show the ratio of the input stimulus to the measured output. A flat response means the network responds equally to all input frequencies (a truly linear device). You can also view the phase of a frequency response measurement – to look at phase shift or phase accuracy of the network. Naturally, frequency response measurements are displayed in the frequency domain.

Traditionally, most analyzers made frequency response measurements by calculating the ratio between the network's output linear spectrum to its input linear spectrum. However, analyzers such as the HP 35660A calculate frequency response differently – by measuring the ratio of the cross spectrum to the input (channel 1) power spectrum. This method is more accurate, but a bit harder to understand. It's a good idea to understand the cross spectrum measurement, as both frequency response and coherence measurements are derived from cross spectrum calculations.

Measuring frequency response by calculating the ratio of cross spectrum to the input power spectrum provides a better statistical estimate of true frequency response. Here's why:

- Anomalies in the input signal are minimized, because the analyzer measures an averaged input signal – the power spectrum (an rms averaged linear spectrum).
- Measuring the cross spectrum instead of the linear output spectrum minimizes non-coherent (spurious) information that may be present in the network under test. In fact, measuring frequency response this way produces a useful by-product – coherence. See "Coherence Measurements."
- No trigger is required for averaging frequency response measurements (unlike the traditional method).

Coherence Measurements

Softkey: [COHERENCE]

Coherence is derived from a series of averaged frequency response measurements – the more averages, the better the coherence measurement (you'll need to take at least five to ten averages to get good results).

The coherence display appears in the frequency domain. It shows the portion of the output power spectrum actually caused by an input signal – sort of an "integrity check."

Coherence values have no units, but are measured with simple linear scale (from 0.0 to 1.0). A coherence value of 1.0 (perfect coherence) means that all power at the output was caused by the input signal. A value of 0.0 (no coherence – an extreme case) means that none of the power at the output was caused by the input signal. Most coherence values are between these extremes.

Poor coherence can be caused by many things. Possible sources of poor coherence are leakage errors (see "windowing" in chapter 4), poor signal-to-noise ratios (perhaps caused by improper range settings), non-linearities in the device under test, and extraneous noise. Coherence is a complex measurement and should be used and interpreted with great care.

Coherence and frequency response measurements are often used together. After averaging several frequency response measurements, you can use the coherence display to find places along the output power spectrum where the measurement data may be questionable – in other words, places with poor coherence.

Ideally, of course, a network under test – electronic or mechanical – should exhibit perfect coherence. That is, the only stimulus to the network is what you apply and the only response is that caused by this controlled stimulus. But in many cases, it's just not possible to completely isolate the network from noise, interference, or other anomalies. Here are some examples:

- For electronic measurements, coherence is used to identify frequency components that cannot be removed from the device under test. For example, when measuring the frequency response of a switching power supply with a very large component at the switching frequency.
- For mechanical measurements, coherence is used to minimize measurement error from external sources. For example, when measuring the frequency response of a certain machine component, coherence (used carefully) can help isolate frequency components originating from nearby machinery – especially important when nearby machinery cannot be turned off.

If the spurious signal is common to both channels, the coherence measurement will not flag the problem. This can occur with 60 Hz power line components, for example.

Cross Spectrum Measurements

Softkey: [CROSS SPECTRUM]

Cross spectrum measurements are an intimate part of both frequency response and coherence measurements. In fact, the analyzer calculates cross spectrum (but doesn't display it) to derive both frequency response and coherence measurements. Like frequency response and coherence measurements, cross spectrum is a two-channel measurement.

Cross spectrum measurements are not used as often as other measurements. For most applications, cross spectrum (used without other measurements) is rarely used.

What Does It Show?

Cross spectrum (sometimes called "cross power spectrum") is a measure of the mutual power between two signals at each point in the current frequency span. Cross spectrum measurements reveal both phase and magnitude information.

The phase display of the cross spectrum measurement shows the *relative* phase – at each frequency – between two signals. Because the phase relationship is relative, you can make cross spectrum measurements without using a synchronized trigger.

The magnitude display of the cross spectrum measurement is the product of the magnitudes of the two signals. If both signals have a large magnitude, the cross product will be large – if both are small, the cross product will be small. This makes cross spectrum a sensitive tool for isolating major signals common to both signals.

Why Use It?

You can use cross spectrum to analyze phase relationships between signals. These might be caused by time delays in a system, propagation delays, or multiple signal paths between source and destination.

You can also use the cross spectrum measurement to calculate acoustic intensity. Acoustic intensity is a vector quantity that indicates the direction and magnitude of sound propagation at a point in space. It is proportional to the imaginary part of the cross spectrum measurement between two closely spaced microphones in the sound field.

Cross spectrum measurements do not necessarily reveal causal relationships. For example, if you're using the analyzer to measure a network, the cross spectrum may show signals at the output (channel 2) not caused by the input (channel 1). The coherence measurement is a much better indicator of causality.

Linear Magnitude Trace

Softkey: [LINEAR MAGNITUDE]

The linear magnitude trace type shows the magnitude of the measurement defined for the active trace on a linear y-axis scale.

Here are some characteristics of the linear magnitude trace:

- For frequency-domain measurements, frequency is the x-axis.
- For time -domain measurements (time records), time is the x-axis.
- For linear magnitude displays, all points on the y-axis have a linear relationship (regardless of the vertical unit you've selected). Usually, it's more convenient to assign a linear unit (such as V or Vrms) as the vertical unit on linear magnitude displays. But even if you select a logarithmic unit (such as dBVrms or dBm), the spacing between points on the y-axis still remains linear.

Logarithmic Magnitude Trace

Softkey: [LOG MAGNITUDE]

The logarithmic magnitude trace type shows the magnitude of the measurement defined for the active trace on a logarithmic y-axis scale.

Here are some characteristics of the logarithmic magnitude trace:

- For frequency-domain measurements, frequency is the x-axis.
- For time-domain measurements (time records), time is the x-axis.
- For logarithmic magnitude displays, all points on the y-axis have a logarithmic relationship (regardless of the vertical unit you've selected). Usually, it's more convenient to assign a logarithmic unit (such as dBVrms or dBm) as the vertical unit on logarithmic magnitude displays. But even if you select a linear unit (such as V or Vrms), the spacing between points on the y-axis still remains logarithmic.

Phase Trace

Softkey: [PHASE]

The phase trace type shows the phase of the measurement defined for the active trace.

Here are some characteristics of the phase trace:

- Frequency is the x-axis.
- Phase is the y-axis, displayed in degrees or radians. Unless you specify otherwise, the analyzer will scale the y-axis at ± 180 °.
- Unwrapping begins to occur for scaling greater than 45 ° per division. To change the vertical units per division, use the < Scale > hardkey and its associated softkeys.
- Phase accuracy is reduced for signal levels that are low relative to the full scale input range.

Group Delay Trace

Softkey: [GROUP DELAY]

The group delay trace type shows the group delay for the measurement defined for the active trace. Group delay is related to phase, but shows phase delays in time (seconds, milliseconds, or microseconds) rather than degrees of phase shift.

Group delay is actually the derivative of phase (the slope) with respect to frequency. The analyzer uses a smoothing aperture to define the resolution of the group delay display – you can change the resolution by selecting different apertures. Larger apertures have more of a smoothing effect than smaller ones. You can select the following smoothing apertures:

- 0.5% of span
- 1% of span
- 2% of span
- 4% of span
- 8% of span
- 16% of span

Here are some characteristics of the group delay trace:

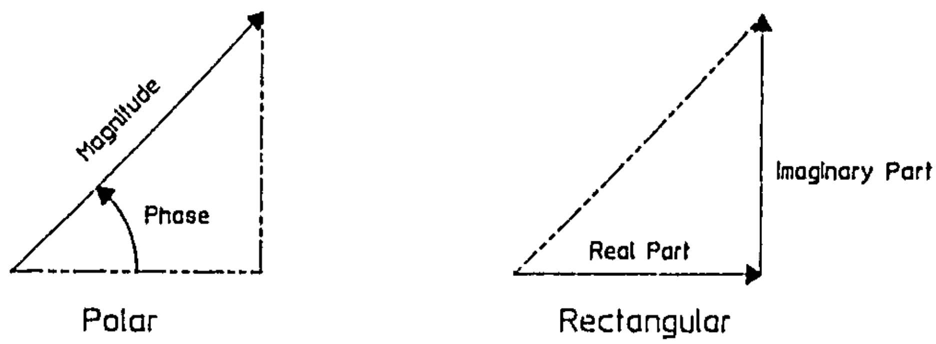
- Time is the y-axis.
- Group delay is plotted between the frequency points used to make the group delay measurement. For example, group delay for 100 Hz can be calculated by measuring the change in phase between 90 and 110 Hz. Therefore, no data is calculated for the end points of this segment (90 and 110 Hz). If you specified 90 as the start frequency, the first frequency point with any data will be 100 Hz – this means the trace will not extend to the left-hand side of the screen. This is more noticeable with larger group delay apertures.

Real and Imaginary Parts

Once an input signal is transformed from the time domain to the frequency domain, there are two ways to express values for the frequency components in each bin. One choice is to show the magnitude or phase of a component; the other is to show the real part or imaginary part of each component.

Polar Form (Magnitude and Phase). This is the most common way to characterize a frequency component. For example, when you select Log Mag and Phase as trace types, you are looking at magnitude and phase values for each frequency component. The magnitude represents the length of a vector and the phase is the angle of the vector.

Rectangular Form (Real and Imaginary parts). This is a less common way to characterize a frequency component. Still, it may be useful for some applications. For example, the imaginary part of the cross spectrum is used for acoustic intensity measurements.



The same frequency component can be expressed as a polar coordinate or a rectangular coordinate.

Real Part Trace

Softkey: [REAL PART]

The real part trace type shows the real part of the measurement defined for the active trace.

Here are some characteristics of the real part trace:

- Frequency (or time) is the x-axis.
- The real part of the active trace data is on the y-axis.
- For time record waveforms that are complex (zoomed measurements), the real part is scaled to be one-half the value of the waveform shown for real value (non-zoomed) time records.

Imaginary Part Trace

Softkey: [IMAGINARY PART]

The imaginary part trace type shows the imaginary part of the measurement defined for the active trace.

Here are some characteristics of the imaginary part trace:

- Frequency (or time) is the x-axis.
- The imaginary part of the active trace data is on the y-axis.
- If there's no imaginary data, the waveform will be a flat line, showing zero magnitude.
- For FFT data (all measurements excluding time records), the imaginary trace represents the imaginary part of the complex FFT data.
- For time waveforms, the imaginary trace represents the imaginary part of the Hilbert transform of the real part. For example, a 2 volt (peak) sine wave input in zoom mode will appear as a frequency-shifted 2 volt (peak) sine in the real part trace, and as a frequency- and phase-shifted 2 volt (peak) sine wave in the imaginary part trace.

Chapter 4
More Basics

Now that you've completed your first measurement and learned about basic measurements and trace types, it might be helpful to review some additional measurement basics. These include:

- Setting the input range
- Windowing
- Averaging
- Overlap processing
- Real-time bandwidth

Setting the Input Range

To make the best measurement possible, you should carefully consider the method you use to set the input range. You can set the input range automatically (using the autorange feature) or you can set the range manually. If you overload the current input range, an "Ovl1" or "Ovl2" message appears at the *top* of the analyzer's screen. If you exceed the analyzer's maximum range, an "OVLD" message also appears at the *bottom* of the screen.

Maximum Input Range:	27 dBVrms 30.01 dBV(peak) 22.39 Vrms 31.66 V(peak)
Minimum Input Range:	−51 dBVrms −47 dBV(peak) 2.818 mVrms 3.966 mV(peak)

The analyzer's input range extends from −51 dBVrms to +27 dBVrms.

Setting the Input Range with Autoranging

Autoranging for the HP 35660A is an "autorange up" feature. This means that when you start a measurement, the analyzer sets the input to the most sensitive range, and automatically steps through less-sensitive input ranges until the input channel is no longer overloaded.

If the input signal amplitude increases after the range is set (enough to overload the input), the analyzer will begin stepping through even less-sensitive ranges. Again, this stops when the input is no longer overloaded.

If the input signal amplitude decreases, the analyzer *will not* change to a different range. The input range will remain at the setting the analyzer found appropriate at the beginning of the measurement.

By the way, the analyzer does not autorange while averaging – so don't change the output of your test device during the averaging procedure. If an over-range condition occurs while averaging, an overload message appears but the analyzer does not abort the averaging procedure.

Setting the Range Manually

You can set the input range manually when you want to maintain a specific input range setting. Ideally, the signal peak should fall in the upper half of the currently-selected input range.

If you set the input range too low (more sensitive than necessary), the analyzer's input circuitry will introduce distortion into the measurement. But if you set the input range too high (less sensitive than necessary), the resulting loss of dynamic range will introduce additional noise – in some cases, the increase in the noise floor may even obscure low-level frequency components.

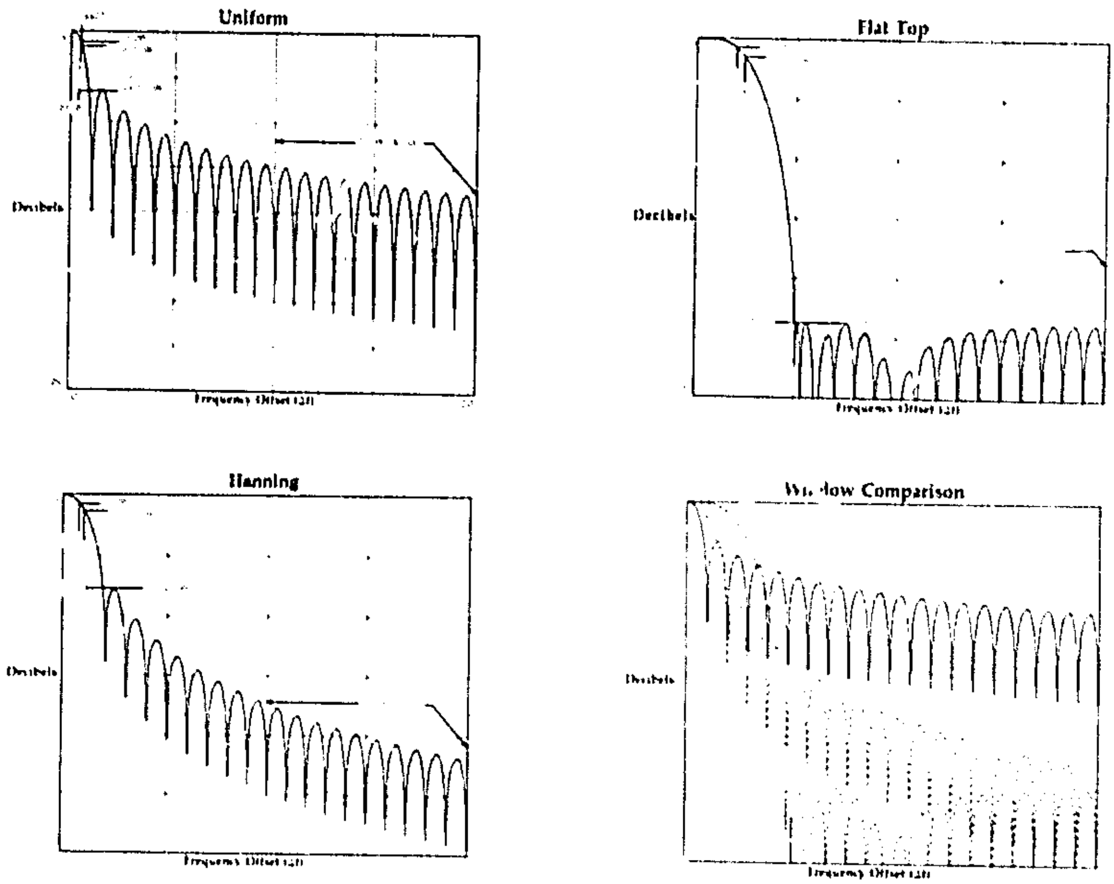
Windowing

A "window" is a time-domain weighting function applied to the input signal – essentially, a way to filter out signals that are not periodic (and therefore spurious) within the input time record. Depending on the window, the analyzer attenuates certain parts of the input time record, to prevent "leakage" – a smearing of energy across the frequency spectrum, caused by transforming signals that are not periodic within the time record.

To learn more about leakage and windowing, see *Hewlett-Packard Application Note 243* (available from your local HP Sales/Service Office).

Here are the windowing functions available with the HP 35660A:

- Hanning
- Flat Top
- Uniform
- Force
- Exponential



The HP 35660A functions as if the input signal were applied to a parallel bank of 401 narrow-band filters. The drawings here show the frequency-domain response of a single filter when using Uniform, Hanning, or Flat Top windows.

The left side of each drawing represents the center of each filter. Since the filters are symmetrical, only the right side is shown (the left side is a mirror image). The horizontal axis shows the frequency offset from the center of the filter, in units of Δf – or in other words, the number of frequency bins away from the one where the filter is centered.

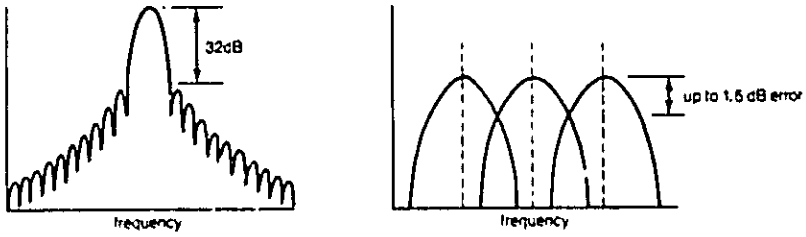
Think of each drawing as a template. If you position a sine frequency at the exact center of the filter, more of the sine wave's energy will show up at the center bin. Some of the energy will also show up in other bins. The amount of energy that spills into adjacent bins depends on the type of window you use. Notice how the Hanning window provides better frequency resolution than the Flat Top window – you can see how less energy spills into nearby bins with the Hanning window.

The Hanning Window

The Hanning window (sometimes called the Hann or Random window) attenuates the input signal at both ends of the time record. This forces the signal to appear periodic. The disadvantage of the Hanning window is some amplitude inaccuracy for sinusoidal signals (from 0 to minus 1.5 dB) compared to the Flat Top window. But its advantage is greater frequency resolution.

Here's what else you should know:

- The Hanning window is the most commonly-used window. It is particularly useful for random noise measurements.
- When you select the Hanning window, the function is applied to *both* input channels.



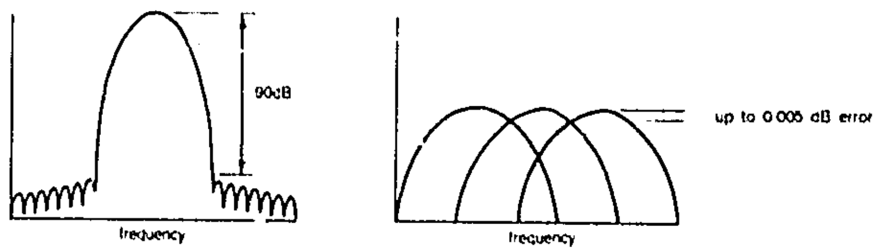
The Hanning Window

The Flat Top Window.

The Flat Top window (sometime called a sinusoidal window) compensates for the amplitude inaccuracy of the Hanning window. The flatter shape of the Flat Top window offers greater amplitude accuracy (plus or minus 0.005 dB). But the trade-off is lower frequency resolution.

Here's what else you should know:

- The Flat Top window is useful when you must measure the amplitude of a particular frequency component with great accuracy – for example, when using a fixed-sine source.
- When you select the Flat Top window, the function is applied to *both* input channels.



The Flat Top Window

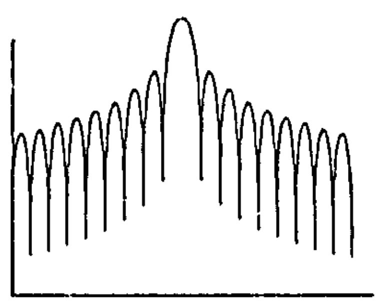
The Uniform Window

The Uniform window (sometimes called a transient window) has a rectangular shape that weights all parts of the time record equally. In other words, the Uniform window isn't really a window at all.

Because the Uniform window does not force the signal to appear periodic in the time record, it is normally used only with functions that are self-windowing, such a transients and bursts. The Uniform window has an amplitude accuracy uncertainty from 0 to minus 4.0 dB.

Here's what else you should know:

- For best results with the Uniform window, you should use signal sources that are periodic — for example, the analyzer's periodic chirp waveform.
- When you select the Uniform window, the function is applied to *both* input channels.



Uniform Window

Force Window

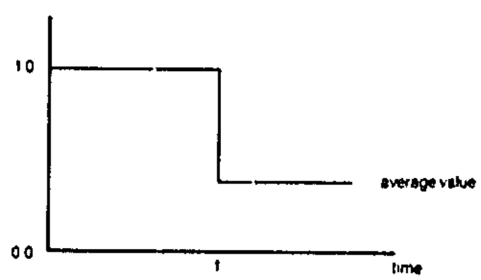
The Force window passes the first part of the time record and sets the last part to a fixed value. You can specify the width of the window, thus controlling where the fixed level begins. The width you specify determines how much of the signal is passed. Note that the width must be narrower than the time record for the force window to have any effect.

The analyzer calculates the average value of the time record's remaining data and sets the time record to this average level.

The force window is helpful in impact testing because is removes residual oscillations in lightly damped systems. It is often used with the Exponential window (see "Exponential Window").

Here's what else you should know:

- Unlike the other windows, you can apply the force or Exponential window to each channel individually. This allows you to mix the windows in measurements using both input channels, such as frequency response. This application is most commonly used when measuring propertie.- of mechanical structures during impact testing.
- If you apply the Force window to channel 1 and the Exponential window to channel 2, the data for channel 1 is multiplied by both the Force and the Exponential windows.
- If you are using trigger delay and you want to set the force width using the marker, remember that the time record starts in negative time for pre-triggering. You may have to adjust the window width to allow for this.



The Force Window

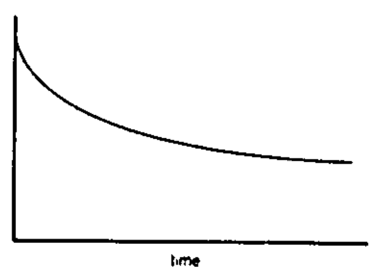
Exponential Window

The Exponential window attenuates the input signal at a decaying exponential rate determined by a specified time constant. You can enter a value between 0.1 μS and 9.99×10^6 Seconds.

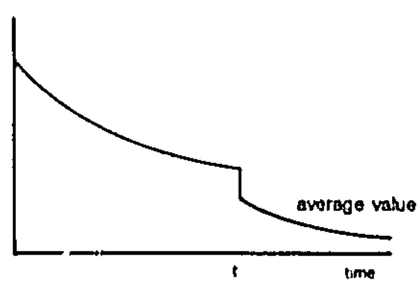
The Exponential window is often used in lightly damped systems with frequency responses that do not decay within one time record.

Here's what else you should know:

- Generally, the time constant should be set to one-fourth of the time record for the window to be effective.
- This window attenuates the input signal at a decaying exponential rate determined by the specified time constant.
- If you apply the Force window to channel 1 and the Exponential window to channel 2, the data for channel 1 is multiplied by both the Force and the Exponential windows.



The Exponential Window



The Combined Force and Exponential Windows

Type of Averaging

Although we introduced rms and vector averaging in the previous chapter, we will review these briefly and introduce some additional ways to make averaged measurements.

The HP 35660A has these types of averaging:

- Stable (normal) rms averaging
- Stable (normal) vector averaging
- Exponential averaging (either rms or vector)
- Peak hold averaging
- Fast averaging

RMS Averaging

To review, rms (power) averaging does not eliminate noise – it simply produces an approximation of the actual noise level. Increasing the number of rms averages provides a better statistical approximation of the noise, but will not actually reduce the noise.

Vector Averaging

With vector averaging, the analyzer averages complex values, point-by-point, in the frequency domain. This lowers noise because the real and imaginary components of the random signals are not in phase and therefore cancel each other – increasingly so with each average. Frequency components that are periodic do not cancel and therefore do not diminish with successive averages.

For mechanical applications, vector averaging is often used during vibration measurements to resolve low-level frequency components from background noise.

Vector averaging produces results similar to *time averaging*, a feature found on many FFT analyzers (time averaging means that the analyzer averages all time records first, then performs a single FFT on an averaged time record). Vector averaging accomplishes the same thing as time averaging, since the averaged linear spectrum derived from a series of vector-averaged linear spectra is equivalent to a single linear spectrum of time-averaged time records.

Although measurements made with vector averaging have better signal-to-noise ratios than rms averaging, there are some restrictions:

- The input signal must be periodic. In other words, the frequency components you want to measure must repeat with each time record. If these components are not periodic (not in phase with the start of each new time record), their real and imaginary values will cancel and the analyzer will not resolve these components.
- If you select vector averaging, you'll need to provide a trigger signal — from the analyzer's source or from an external signal. Of course, the analyzer will still make a measurement with continuous triggering (no trigger signal), but the amplitude of periodic signals will diminish with each successive average (since even periodic components have random phase with continuous triggering).

Exponential Averaging

You can select either rms exponential averaging or vector exponential averaging. Both work in a similar fashion. The only difference is that for vector exponential averaging, you'll need to provide a trigger signal.

Unlike stable (normal) averaging, exponential averaging weights new data more than old data. This is useful for tracking data that changes over time.

When using exponential averaging, the number of averages you select determines the weighting of old versus new data — not the total number of averages calculated. As the number of averages increases, new data is weighted less.

With exponential averaging, it's especially important to set the number of averages carefully — if there are too few averages in the measurement, the averaging will not smooth out variances. But if there are too many averages, the analyzer may not track subtle changes occurring within the data.

To calculate the exponential average, the analyzer uses this formula: $\{[(1/N)\times(\text{new})]+[((N-1)/N)\times(\text{old})]\}$, where N is a weighting factor (the number of averages you've specified).

When starting an exponential average, the analyzer sets N equal to 1 for the first analysis, then sets N equal to 2 for the second analysis, and so on — until N equals the number of averages you've specified.

Here's what else you should know:

- Once you start a measurement using exponential averaging, the measurement continues indefinitely. To stop the average, you must pause the measurement. This is different than stable averaging — stable averaging stops automatically after the specified number of averages are completed.
- For the first few averages, there's little difference between exponential averaging and stable averaging.

Peak-hold Averaging

When you request the peak-hold function, the analyzer will take data continuously, until you tell it to stop. The analyzer will compare each data point along the measured frequency span with the previous values. Only the largest values for each frequency bin will be saved.

Technically, peak-hold averaging is not really a type of averaging, since the results are not mathematically averaged. But it's still considered a type of averaging because it combines the results of several measurements into one final measurement result.

Here's what else you should know:

- With the peak-hold function, the analyzer mathematically compares each data point to its previous peak value. If the data point is larger than its last peak value, the new value is used. This is not the same thing as peak-holding the displayed trace.
- The peak-hold function works only with spectrum measurements, power spectral density (PSD) measurements, or time records.

Fast Averaging

Fast averaging is not a *type* of average. Rather, it's simply a way to have the analyzer make averaged measurements without having to update the screen after each average. You can use fast averaging with any type of averaging (rms, vector, rms and vector exponential, and peak-hold averaging).

You can specify the number of averages between screen updates for the fast average mode. If you enter an update rate of 5, for example, the analyzer will update the screen after every five averages. The update rate is important, since the number you select will affect the speed of the fast average (you can select values between 1 and 99,999).

By the way, fast averaging must be on to achieve maximum real-time bandwidths. See "Real-Time Bandwidth" later in this chapter.

Overlap Processing

As the span you select decreases, the corresponding time record length increases (see "Measurement Speed vs. Time Record Length" in Chapter 3, "Measurement Basics"). At some point, the time record length and the amount of time the analyzer needs to process each record are equal. If you continue to increase the record length, the FFT processor sits idle after processing the time record (while waiting for the next record to fill). But *overlap processing* allows you to overlap time records and compute the FFT from both previous *and* current time records.

Overlap processing offers several advantages. First of all, it lets you make a faster measurement (particularly with narrow spans). Overlap processing also reduces statistical variance caused by windowing. For a detailed discussion of overlap processing and real-time bandwidth, see *Hewlett-Packard Application Note 243* (available from your local HP Sales/Service Office).

Overlap processing is set in the < Average > menu. To specify the amount of overlap you want, use the [OVERLAP%] softkey. You can enter any value from 0 to 99%, in 1% increments.

Here's what else you should know:

- Overlap is not used if you're making triggered measurements. The analyzer must be in the continuous trigger mode.
- The amount of overlap possible varies with the frequency span. For wide spans (with short time records), little or no overlap is possible — the time record is small compared to the time it takes the analyzer to process the time record. For narrow spans (with long time records), considerable overlap is possible — the time record is long compared to the time it takes the analyzer to process the time record.
- The analyzer does not indicate the actual overlap percentage used. For example, if you specify an overlap of 90%, the analyzer will accept this value but may not actually use a 90% overlap if this is incompatible with the current frequency span.
- The analyzer will treat the overlap percentage as the maximum allowed. The actual overlap used depends on the current frequency span, the type of average selected, and how busy the analyzer is servicing the HP-IB and marker functions and key presses. The overlap percentage can change from time record to time record, but will always be less than or equal to the specified overlap percentage. If the analyzer indicates that the current measurement is in real time and the overlap percentage falls below 0%, the REAL TIME status message will be removed (if you're averaging and this occurs, no attempt will be made to re-enter real time until you start the average again — if averaging is off, real time processing will resume as soon as possible).

Real-Time Bandwidth

Overlap processing is easy to understand if you relate it to real-time bandwidth (RTBW). Real-time bandwidth is a specification used to characterize the performance of an FFT analyzer. The real-time bandwidth is the frequency span at which the FFT processing time equals the time record length – this means all input data is included in the average (in other words, there is no gap between the end of one time record and the beginning of the next). However, if you increase the span past the real-time bandwidth, the record length becomes shorter than the FFT processing time. Time records are no longer contiguous, and some input data is missed. Therefore, you can overlap records only when measuring below the real-time bandwidth, because the time record length must be longer than the FFT processing time to achieve any overlap.

The actual real-time bandwidth achieved varies with the amount of processing time the analyzer needs. As with overlap processing, this depends on the current frequency span, the type of average selected, and how busy the analyzer is servicing the HP-IB and marker functions and key presses. The following table shows typical real-time bandwidth for the HP 35660A:

	One-channel mode	Two-channel mode
Averaging Off	800 Hz span	400 Hz span
Fast Averaging	3.2 kHz span	1.6 kHz span

Typical Real-Time Bandwidths

Chapter 5
Spectral Purity of a Sine Wave

Task Overview — This chapter steps you through a series of measurements to characterize the spectral purity of a sine wave.

What you will need — gather the following items before starting this task:

- A connecting cable (BNC male to BNC male)

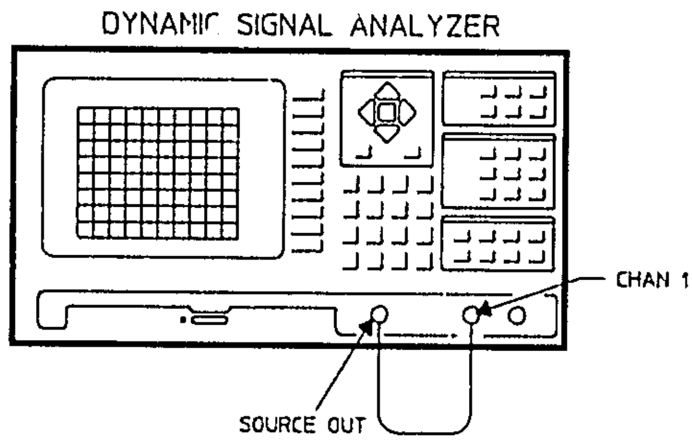
What you will measure — you will use the analyzer to do the following:

- Look for prominent harmonics of the fundamental frequency
- Measure Total Harmonic Distortion (THD)

What you will learn — In this chapter, you will be introduced to the following analyzer functions:

- Viewing a spectrum to reveal a fundamental frequency and its harmonics
- Setting the proper input range
- Selecting an appropriate scale
- Using the absolute marker, the offset marker, and the harmonic distortion marker

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer — not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >.

Pressing < Preset > returns most of the analyzer settings to the default positions.

2. Connect the analyzer's source to the Channel 1 input.

You do not need to terminate the analyzer's source, since the output impedance is less than 5Ω.

The analyzer's input channels have an impedance of 1 MΩ.

In the example here, you are using the analyzer's internal source as the test device. However, to test external signal sources designed to operate into a specific load (such as an oscillator with a 600Ω output), you must place an appropriate feedthrough terminator across the output of the test device.

3. Press < Source >

   [SOURCE ON/OFF]

   [FIXED SINE]

This turns on the analyzer's internal source and selects the fixed sine wave.

4 Press [SINE FREQ ENTRY]

   < 1 > [kHz]

This sets the sine frequency to 1 kHz.

5. Press [LEVEL]

   < 2 > [Vrms]

This sets the level of the sine wave to 2 Vrms.

6. Press < Input >

   [CHANNEL 1 AUTORANGE]

   This makes sure the input range is set correctly.

The default setting for the input is an "autorange up" feature. "Autorange up" means that when you start a measurement, the analyzer sets the input to the most sensitive range, and automatically steps through less-sensitive input ranges until the input channel is no longer overloaded. You'll see an "Auto-Ranging" message in the upper left corner of the screen.

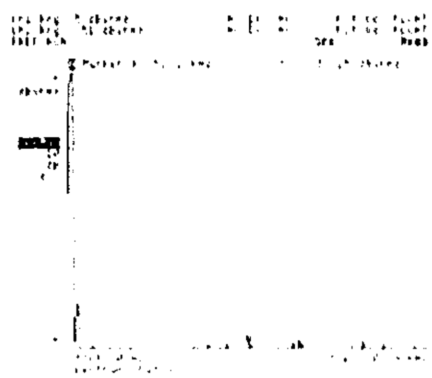
If the input signal amplitude increases after the range is set (enough to overload the input), the analyzer will begin stepping through even less-sensitive ranges. Again, this stops when the input is no longer overloaded.

If the input signal amplitude decreases, the analyzer *does not* change to a different range. The input range will remain at the setting the analyzer found appropriate at the beginning of the measurement.

If you *decrease* the output of your test signal during the measurement, you'll have to press < Input > and then press [CHANNEL 1 AUTORANGE].

7. Now look at the analyzer's screen. This is a display of the linear spectrum.

   This display appears in frequency domain.



Since this is a full span (0 to 102.4 kHz), the relatively low frequency of the signal (1 kHz) is at the extreme left of the display.

Because averaging is off, you will see the display change several times each second. Each display represents one FFT of a single time record.

8. Press < Meas Data >

   [TIME CHANNEL 1]

You are now viewing each time record as the analyzer acquires a new one.

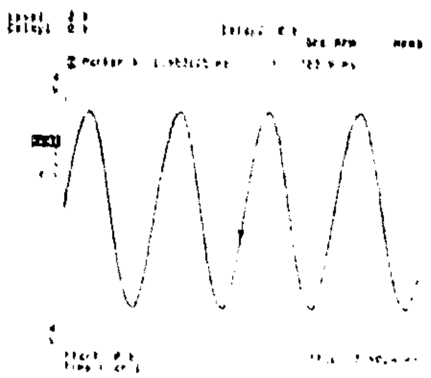
This is the time domain representation of the input data — that's why it looks like an oscilloscope trace.

Notice how the time record jitters. This is because the analyzer's trigger is set to continuous (the default condition). This means the analyzer will process time records as quickly as possible, without waiting for any kind of trigger signal.

Also note how some of the softkeys are de-emphasized. This means that these menu selections are not currently active. For example, [FREQUENCY RESPONSE] is a two-channel measurement and is not available right now because the analyzer is in the single-channel mode.

9. Press < Trigger >

   [CHANNEL 1 TRIGGER]



Notice how the time display becomes stable. With input triggering, the analyzer (when ready) does not begin a new measurement until the input signal reaches the predetermined trigger level.

The trigger level ranges from –100% to 100% of the input range (don't confuse this with the vertical units shown on the current display). The default value is 0% with a positive slope — this means the analyzer will trigger when the input signal crosses zero as the signal moves from negative to positive.

For now, it's not important to understand everything about triggering — but you should take comfort knowing that most of the analyzer's triggering features are similar to those found on standard oscilloscopes. To learn more about the analyzer's triggering capabilities, see the *HP 35660A Front-Panel Reference*.

10. Press < Meas Data >

    [SPECTRUM CHANNEL 1]

This returns you to the frequency domain.

11. Press < Freq >

    Press [SPAN]

    Now use the < ⇩ > hardkey to step through several spans.

The < ⇩ > and < ⇧ > hardkeys are located in the numeric keypad.

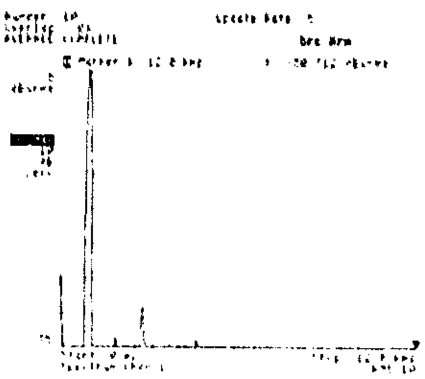
12. Stop when you reach 12.8 kHz — if you step down too far, simply use the < ⇧ > hardkey to go back up to a 12.8 kHz span.

This changes the frequency span and lets you look at a smaller slice of the frequency spectrum. This gives a better view of the fundamental and its harmonics.

You can also use the numeric keypad to specify a span (the analyzer takes the nearest acceptable value).

13. Press < Average >

[AVERAGE ON/OFF]



Note how the word "ON" in this softkey label highlights when averaging is on.

The default averaging is rms averaging (with ten averages). For now, this is the type of averaging you will use.

You are now looking at the averaged power spectrum. You can see the fundamental frequency and its harmonics. Because the span is large (full span), most of the harmonics are grouped at the left side of the display.

By the way, the analyzer does not autorange while averaging — so don't change the output of your test device (in this case, the analyzer's internal source) during the averaging procedure. If an over-range condition does occur while averaging, an overload message appears but the analyzer does not abort the averaging procedure.

13a. If you don't see any harmonics, press < Input >. [CHANNEL 1 RANGE] Then press < ⇩ > twice.

This intentionally overloads the analyzer's input to simulate a source with prominent harmonics.

14. Press < Start >

Note how another averaged measurement begins.

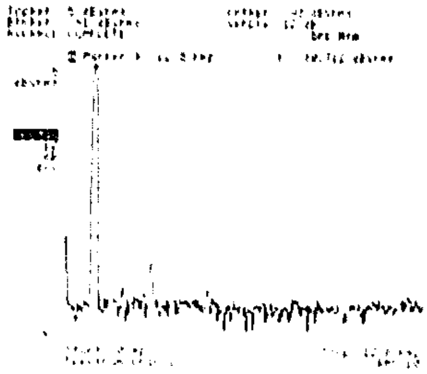
If you press < Pause/Cont > instead of < Start >, the analyzer makes another averaged measurement but adds the averages to the previous group of averages.

15. Press < Scale >

[VERTICAL/DIV]

< 1 > < 2 > [dB].

The display should now look like this. You changed the vertical scaling from 10 to 12 dB per division to see both the noise floor and the peak of the fundamental.



16. Press [AUTO SCALE]

Autoscaling means the analyzer automatically selects an appropriate scale for the input signal.

Alternatively, you could have pressed [VERTICAL/DIV] and entered the original scaling — in this case, 10 dB per division.

17. Press [VERTICAL/DIV]

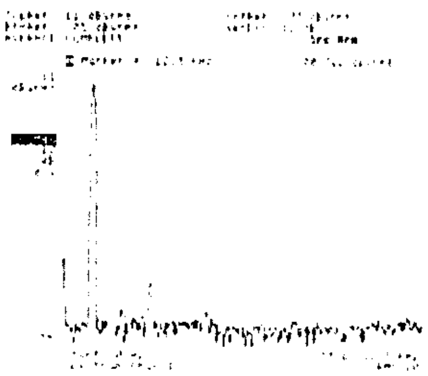
< 1 > < 2 > [dB].

It's convenient to see the noise floor, so you can leave the vertical scale at 12 dB per division.

18. Press [TOP REFERENCE]

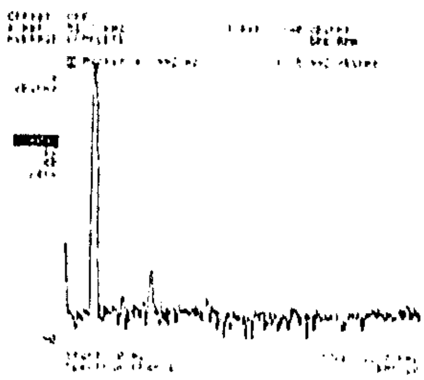
Now use the < ⇩ > and < ⇧ > hardkeys to shift the entire display up or down.

See how easy it is to adjust the display to your liking?



19. Press < Marker >

[MARKER TO PEAK]



This moves the marker to the largest frequency component on the display (in this case, the fundamental.)

The marker you are using is an absolute marker. That is, it indicates the absolute x-axis and y-axis coordinate. There's also an *offset* marker — but you'll learn about that in a few moments.

The analyzer says that the x-axis marker value is 992 Hz. Remember, the analyzer's frequency resolution changes with the frequency span you've selected. For the current span (12.8 kHz), the resolution is 32 Hz. Soon you will change to a smaller span to get better resolution.

20. Note the amplitude value indicated by the marker's y-axis position.

This shows the absolute amplitude of the fundamental. In this case, the y-axis marker value indicates about 6 dBVrms (2 Vrms).

While absolute amplitude values are useful, *relative* amplitude values are more important when characterizing the spectral purity of a signal source.

21. Make sure the marker is at the fundamental frequency. If it isn't, press [MARKER TO PEAK] again.

Press < Freq >

[CENTER]

< Marker Value >

Don't worry if nothing seems to be happening. The analyzer is simply waiting to start a new average.

Pressing < Marker Value > specifies the current marker value (in this case, the fundamental frequency) as the center frequency for the new span that you've about to enter.

The < Marker Value > hardkey is convenient, because it lets you enter numeric values quickly without using the numeric keypad.

Pressing < Marker Value > enters the x-axis, y-axis, or both values for the marker's current position.

If the analyzer needs an x-axis value, pressing < Marker Value > enters the x-axis value. If the analyzer needs a y-axis value, pressing < Marker Value > enters the y-axis value. If the analyzer needs both values, pressing < Marker Value > enters both.

22. Press [SPAN]

    Now use the numeric keypad or the < ⇩ > hardkey to specify a 100 Hz span.

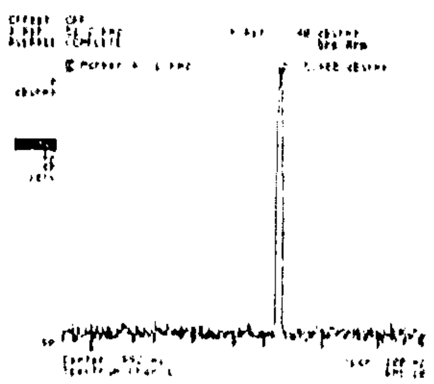
    < Start >

After pressing < Start >, you'll have to wait a for the time record to fill. But don't worry — the "Rec Lg" message at the top of the screen indicates how long it takes to fill the time record (in this case, 4 seconds).

Another message appears every few moments indicating time remaining to fill the time record.

23. Press < Marker >

    [MARKER TO PEAK]



After the first results are displayed, you can go ahead and check to see if the frequency resolution has improved. It's not necessary to wait for the analyzer to finish taking averages. You can use the marker during the averaging process.

The marker now indicates 1 kHz. This is a better approximation of the exact frequency.

24. Press < Freq >

    [SPAN]

    < 3 > [kHz]

    Press [ZERO START] to start the span at 0 Hz again.

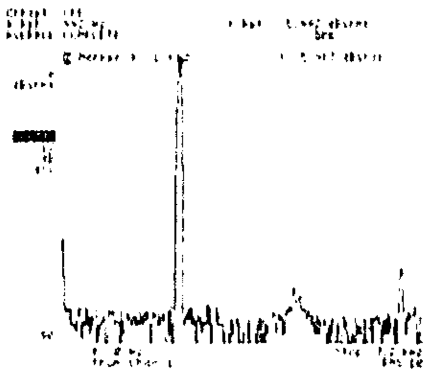
    Press < Start > to begin a new measurement.

This changes the span to more easily view the fundamental and the third harmonic.

When you specified the span, notice how the analyzer changed your numeric entry (3 kHz) to the nearest acceptable value (3.2 kHz).

25. Press < Marker >

[MARKER TO PEAK]

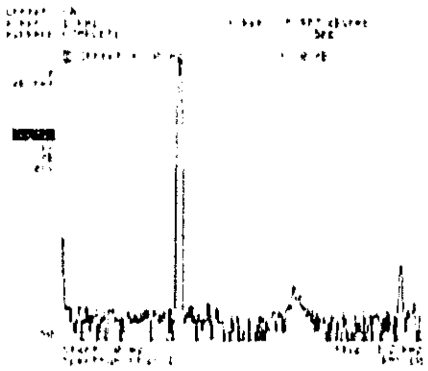


This ensures that the marker is at the fundamental frequency.

26. Press [OFFSET]

Press [OFFSET ON/OFF] to turn on the offset marker.

[OFFSET ZERO]



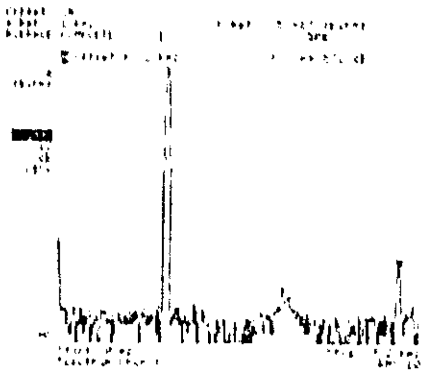
You've just turned on the offset marker, and zeroed it where the marker was. You will use the offset marker to find the amplitude and frequency of a harmonic relative to the fundamental.

On some analyzers, the offset marker is called a "relative marker."

Zeroing the offset marker at the fundamental frequency establishes the peak of the fundamental as a reference point for both x-axis and y-axis marker values. As long as the offset marker is on, both marker values will indicated the amount of offset from the zeroed point (in this case, the peak of the fundamental).

Until you reset the offset marker at another point, the zero position remains where you set it, even if you turn the offset marker off and then back on again. But the zero position will be lost if you press < Preset > or turn the analyzer off.

27. Press the < ▲ > hardkey several times, until the marker indicates an offset of 2 kHz.



This moves the marker to the third harmonic.

The < ▲ > hardkey is the unlabeled key shaped like an up arrow in the MARKER group of front-panel keys.

Pressing < ▲ > jumps the marker to the next big peak to the right. Pressing < ▼ > is similar, but jumps the marker to the left.

You can also use the < ◄ > and < ► > keys to move the marker. But that takes longer, since the marker steps through each "bin" at a time. Or you could use < ◄ > and < ► > with the < Fast > hardkey to move the marker faster.

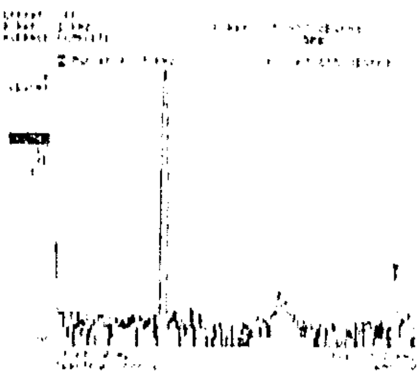
28. Note the frequency value indicated by the offset marker's x-axis position.

This value (actually the offset from the fundamental) is about 2 kHz. Not surprising, since the third harmonic should be offset from the fundamental by twice the fundamental's frequency.

29. Note the amplitude value indicated by the offset marker's y-axis position.

This value is about −70 dBVrms, referenced to the fundamental frequency's amplitude. In other words, the third harmonic is about 70 dB below the fundamental.

30. Press [OFFSET ON/OFF]



This turns off the offset marker. Notice how the marker values now reflect the absolute frequency of the third harmonic (3 kHz) and the absolute amplitude as well.

31. Press < Freq >

[SPAN]

< 1 > < 2 > [kHz]

Before measuring Total Harmonic Distortion (THD), let's change to a larger span. That way, you can see more harmonics.

32. Press < Average >

[NUMBER AVERAGES]

Now use the numeric keypad to specify 25 averages.

[ENTER]

Because the analyzer is set for rms averaging, taking more averages produces a better approximation of rms values. Using more averages also provides a cleaner noise floor.

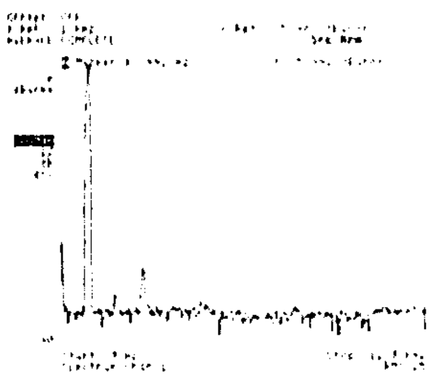
33. Press < Start >

< Marker >

[MARKER TO PEAK]

This begins another measurement and moves the marker to the fundamental frequency.

You can use either [MARKER TO PEAK] or the < ▼ > < ▲ > keys to move the marker.



34. Press < Marker Fctn >

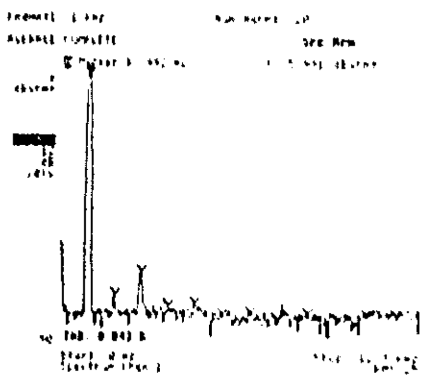
[HARMONIC]

[FNDMNTL FREQ]

< 1 > [kHz]

You have to specify the fundamental frequency for the analyzer to make the distortion calculation.

Once you enter the fundamental frequency, the analyzer starts the distortion calculation. The distortion results appear at the lower left corner of the screen. Our test signal shows about 0.04% THD.



Note how the harmonic markers appear over the harmonic frequencies. Also note the "Num Harms" message that appears at the top of the screen — this indicates the number of harmonics the analyzer used to calculate THD. Because you didn't specify a number, the analyzer used the default value of 20 harmonics (sometimes only five or ten harmonics are necessary).

The THD results reflect the harmonics found in the current frequency span. The number of harmonics you specify is the *maximum* number the analyzer will use in the THD calculation. For example, if you press [DEFINE NUM HARM] and enter 10 harmonics, the THD calculation will not include all 10 harmonics if some of these harmonics are out of the range of the current span.

The analyzer calculates THD by comparing the energy of the fundamental to the energy at the harmonics. Noise and other signals at other points along the frequency spectrum are not taken into account (unless they happen to occur at the fundamental frequency or at the harmonics). This is different than older distortion analyzers that simply rejected the fundamental frequency and measured any remaining energy as harmonic distortion (more accurately, harmonic distortion plus noise).

35. Press [DEFINE NUM HARM]

< 1 > [ENTER]

The THD measurement varies with the number of harmonics used for the distortion calculation.

Note how the THD reading is lower — about 0.01% — because the analyzer is only using the fundamental and the second harmonic to make the calculation.

Chapter 6
Amplifier Noise Level

Task Overview — This chapter steps you through a noise measurement for a typical audio-frequency amplifier.

What you will need — gather the following items before starting this task:

- Audio-frequency amplifier
- Feedthrough terminator to match output impedance of the amplifier
- Appropriate connecting cables

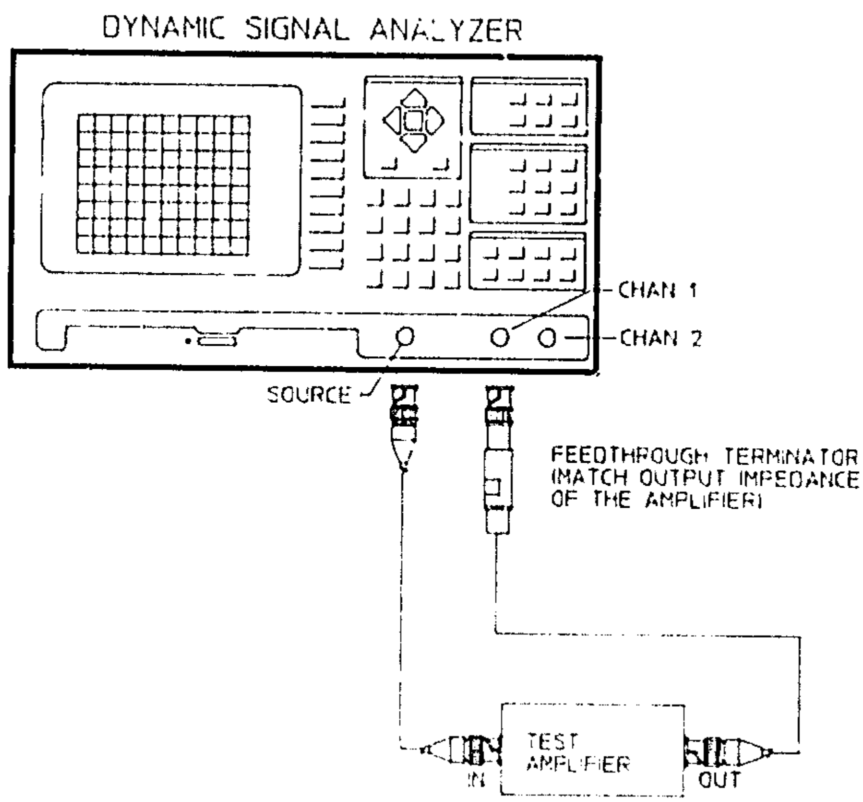
What you will measure — In this chapter, you will learn how to make these measurements:

- Wide-band noise
- Power Spectral Density (PSD) to obtain noise values normalized to a 1 Hz bandwidth
- Signal-to-noise ratios using both wide-band noise and PSD

What you will learn — In this chapter, you will be introduced to the following analyzer functions:

- Viewing a spectrum to reveal noise
- Using the band marker to calculate power within a specific frequency band
- Using the Power Spectral Density (PSD) measurement

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer – not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >.

   Pressing < Preset > returns most of the analyzer settings to the default positions.

2. Connect the analyzer's source to the input of your test amplifier.

   The test device for this example is a typical audio-frequency amplifier.

3. Connect the output of the amplifier to the analyzer's channel 1 input.

   To make a noise measurement valid for typical operating conditions, place an appropriate terminating resistor across the amplifier's output.

4. Press < Source >

   [SOURCE ON/OFF]

   [FIXED SINE]

   This turns on the analyzer's internal source and selects the fixed sine wave.

5. Press [SINE FREQ ENTRY]

   < 1 > [kHz]

   This sets the sine frequency to 1 kHz.

6. Press [LEVEL]

   < . > < 1 > [Vrms]

   This sets the output of the analyzer's source to 0.1 Vrms.

   When making signal-to-noise measurements, you should set the output of the analyzer's source to a level that simulates the typical operating conditions for the amplifier under test. In this example, we used 100 mVrms.

   Your test amplifier may require a different input level. If so, use the numeric keypad to specify a different level for the analyzer's source.

7. Press < Input >

   [CHANNEL 1 AUTORANGE]

This ensures that the input range is set correctly.

8. Press < Scale >

It's often necessary to press [AUTO SCALE] after each measurement. This provides the best display for each trace.

   [AUTO SCALE]

This autoscales the display.

9. Press [VERTICAL/DIV].

For signal-to-noise measurements, you should view both the noise floor and the peak of the test frequency.

   < 1 > < 2 > [dB]

In this example, we used a vertical scale of 12 dB per division. You can use a different scale if it's more convenient.

10. Press < Average >

This turns on averaging and selects 25 rms averages.

   [NUMBER AVERAGES]

The default setting is for 10 rms averages — but for noise measurements, using 25 (or even 50) averages provides a better approximation of the actual rms noise level.

   < 2 > < 5 >

   [ENTER]

   [AVERAGE ON/OFF]

11. Press < Freq >

This selects a span of 25.6 kHz.

   [SPAN]

   < 2 > < 5 > [kHz]

12. Press < Marker >

[MARKER TO PEAK]

Note the amplitude value indicated by the marker's y-axis position.

This moves the marker to the 1 kHz test frequency.

Write down the absolute amplitude of the test frequency – you will need it to make the first signal-to-noise measurement.

In this example, the amplitude of the test signal is about 4 dBVrms.

13. Press < Source >

[SOURCE ON/OFF]

This turns off the test signal.

14. Press < Start >

This starts another measurement – but this time, without the 1 kHz test signal.

15. Press < Marker Fctn >

    [BAND]

    [DEFINE LEFT FREQ]

    < 2 > < 0 > [Hz]

    [DEFINE RIGHT FREQ]

    < 2 > < 0 > [kHz]

    [RETURN]

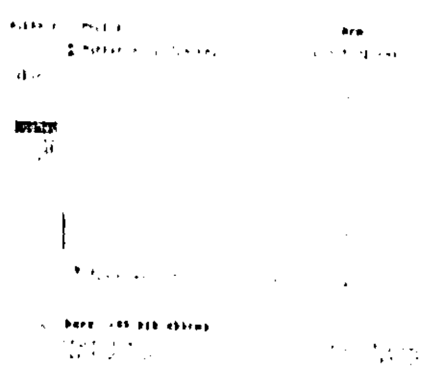
This turns on the band marker and defines the left edge of the band at 20 Hz and the right edge at 20 kHz. The band marker measures the total energy within the specified frequency band.

One way to measure signal-to-noise is to compare the level of the test signal to the amplifier's wide-band noise. Wide-band noise is usually defined as the total rms noise within the amplifier's 3 dB bandwidth — but for audio measurements, a standard bandwidth of 20 Hz to 20 kHz is often used. In this example, we used this standardized bandwidth because we didn't know the actual 3 dB bandwidth.

16. Note the band marker value in the lower left-hand corner of the screen.

This is the rms value of the wide-band noise.

In this example, the absolute value of the wide-band noise is −50 dBVrms.



17. Subtract the level of the test signal from the wide-band noise.

    Discard the minus sign from the final result.

This gives you the amplifier's signal-to-noise ratio.

In this example, we subtracted 4 dBVrms, or a signal-to-noise ratio of 54 dB.

To be more precise, the test amplifier has a signal-to-noise ratio of 54 dB, using a 100 mV, 1 kHz test signal referenced to wide-band noise.

18. Press [OFF]

This turns off the band marker.

19. Press < Meas Data >

[PSD CHANNEL 1]

This selects the PSD measurement (Power Spectral Density).

20. Press < Source >

[SOURCE ON/OFF]

This again turns on the analyzer's internal source.

Notice when you turned on the source, the analyzer retained the settings you used previously. You don't have to re-enter the sine frequency or the output level.

21. Press < Start >

This begins another measurement. You may have to readjust the vertical scale to see the noise floor. For example, 12 dB/division.

22. Press < Marker >

[MARKER TO PEAK]

This moves the marker to the 1 kHz test frequency.

23. Press [OFFSET]

[OFFSET ON/OFF]

[OFFSET ZERO]

[RETURN]

This turns on the offset marker and zeros it at the test frequency.

24. Press the < ▶ > until the marker moves to a typical point on the noise floor.

    Note the amplitude value indicated by the offset marker's y-axis position.

The offset marker shows the relative amplitude between the test signal and a particular point on the noise floor. Remember that with PSD measurements, amplitude values are normalized to a 1 Hz wide bandwidth. (See Chapter 3, Measurement Basics).

In this example, the signal-to-noise ratio is about 75 dB.

The test amplifier now seems to have less noise (the previous measurement indicated a signal-to-noise ratio of 54 dB). That's because you are now measuring the test signal to only one point on the noise floor — not the combined noise of a larger frequency band (as you did when measuring wide-band noise).

Measuring signal-to-noise with PSD does not take into account wide-band noise. But, since all analyzers that measure PSD are standardized to measure with a 1 Hz resolution, PSD may be preferable in cases where you want standardized, repeatable noise measurements.

Chapter 7
Characterizing Acoustic Noise

Task Overview – This chapter steps you through a typical acoustic noise measurement for rotating machinery. Although this particular measurement shows how to pinpoint the frequency of a rotating fan blade, the principles shown are common to other acoustic measurements.

What you will need – gather the following items before starting this task:

- Microphone
- Impedance-matching transformer (if using a low-impedance microphone)
- Appropriate connecting cables

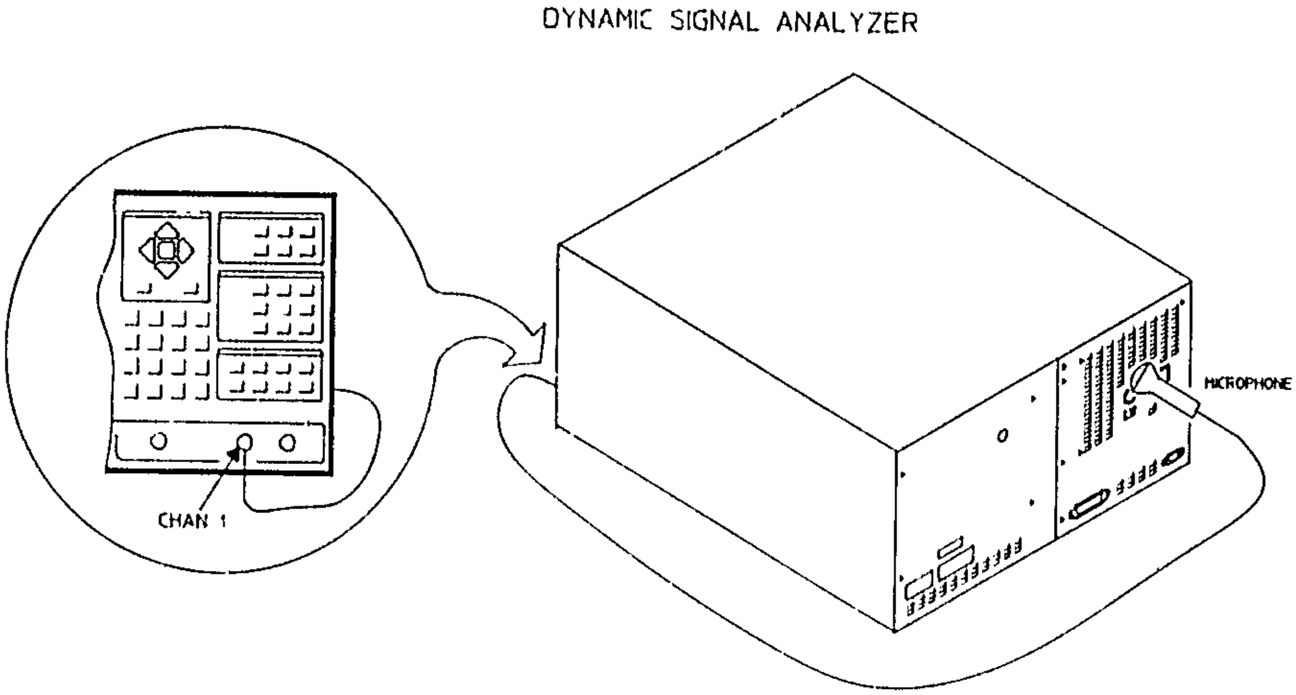
What you will measure – In this chapter, you will learn how to make these measurements:

- Power Spectrum to characterize acoustic noise (in this case, the noise from the analyzer's fan)

What you will learn – In this chapter, you will be introduced to the following analyzer functions:

- Creating a label for the displayed trace

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer — not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >.

   Pressing < Preset > returns most of the analyzer settings to the default positions.

2. Connect the microphone to the channel 1 input.

   The impedance match between the microphone and the analyzer is not critical for this particular measurement.

   If you're using a high-impedance microphone, you can connect it directly to the analyzer's input.

   But if you're using a low-impedance microphone, you may have to use a low-to-high impedance matching transformer.

3. Place the microphone near the analyzer's rear panel, at the fan exhaust.

   Make sure you find a place where the fan noise is loudest.

4. Press < Input >

   [CHANNEL 1 AUTORANGE]

   This makes sure the input range is set correctly.

   You may have to autorange several times while setting up this measurement task -- especially if you handle the microphone roughly or knock it against a hard surface.

5. Press < Freq >

   [SPAN]

   < 4 > < 0 > < 0 > Hz

You will need a relatively small span to resolve fan noise from the ambient noise.

6. Press < Average >

   [AVERAGE ON/OFF]

   [NUMBER AVERAGES]

   Now use the numeric keypad to specify 25 averages.

   [ENTER]

This turns on averaging and selects 25 rms averages.

The default setting is for 10 rms averages — probably not enough averages for the type of measurement. Increasing to 25 averages should be sufficient.

7. Press < Scale >

   [AUTO SCALE]

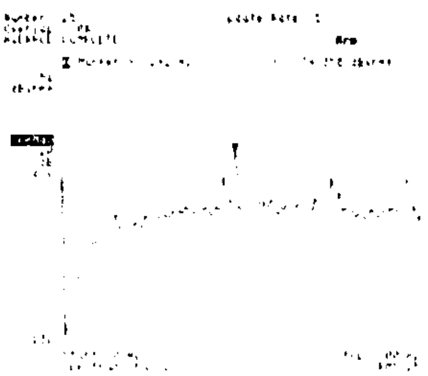
This autoscales the trace.

8. Press < Marker >

    [MARKER ON/OFF]

    [MARKER TO PEAK]

    Note the value indicated by the marker's x-axis position.



This turns on the marker and move the marker to the peak of the spectrum.

The peak should be around 200 Hz. There may be other noise components, but the largest one is the fundamental frequency of the rotating fan.

The magnitude of the peak is not important for this measurement.

9. Multiply the fundamental frequency by 60. Then divide this number by 5 (the number of fan blades).

This converts the fundamental frequency from Hz (cycles per second) to cycles per minute.

In our example, the fundamental frequency was 192 Hz, therefore :

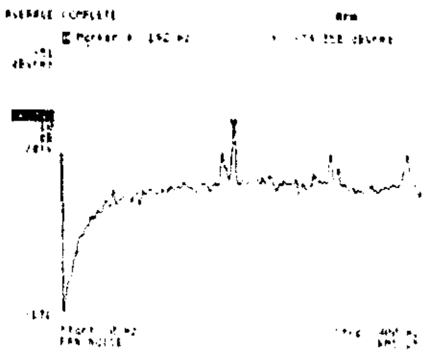
$$\frac{192 \times 60}{5} = 2304 \quad \text{(revolutions per minute)}$$

10. Press < Format >

[TRACE TITLE]

Then use both numeric and alpha-shifted keys to enter an appropriate trace title.

[ENTER]



This lets you label a display trace.

After pressing [TRACE TITLE], the analyzer automatically shifts certain hardkeys to alpha entry keys (note the alpha characters engraved on the front panel below these hardkeys).

If you make a mistake, you can use the appropriate edit softkeys to fix the title. You can also specify uppercase or lowercase letters.

After pressing [ENTER], the analyzer automatically returns the alpha-shifted hardkeys to their normal functions.

Chapter 8
Filter Characterization

Task Overview – This chapter steps you through a series of measurements to characterize the performance of a band-pass filter.

What you will need – gather the following items before starting this task:

- 1 kHz Band-pass filter (individual filter or graphic equalizer)
- Feedthrough terminator to match output impedance of filter (if necessary)
- Appropriate connecting cables and BNC "T" adapter

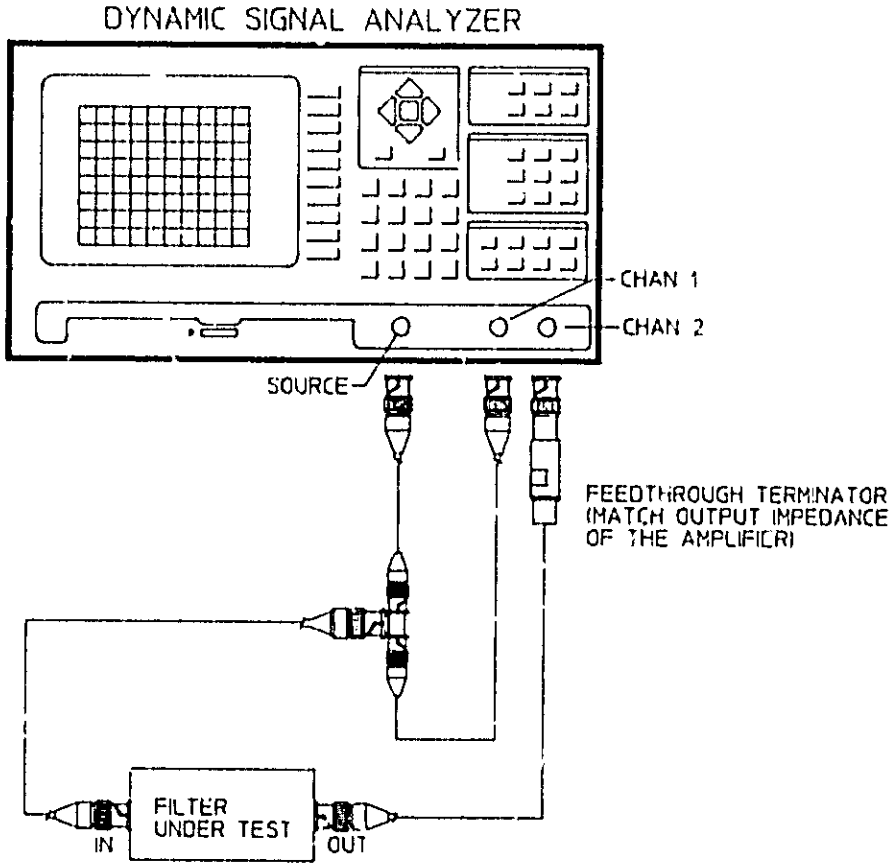
What you will measure – you will learn how to make these measurements:

- Frequency response (shape of filter)
- Resonant and passband frequencies
- Insertion loss
- Phase

What you will learn – In this chapter, you will be introduced to the following analyzer functions:

- Making frequency response measurements
- Viewing phase information
- Using different display formats
- Switching between the linear and logarithmic x-axis
- Using the marker search feature

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer — not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >.

Pressing < Preset > returns most of the analyzer settings to the default positions.

2. Place a BNC "T" adapter on the channel 1 input connector.

   Connect the analyzer's source to one side of the "T." Then connect the other side of the "T" to the input of the filter.

   Connect the output of the filter to the channel 2 input. If necessary, terminate the filter's output.

To make network measurements, the analyzer's source is routed to both the input of the device and to the analyzer's channel 1 input. The output of the test device is always connected to channel 2.

In the example here, the test device has an output impedance of 10 kΩ. So to best characterize the filter, you should place a 10 kΩ feedthrough terminator on its output.

3. Press < Meas Type >

   [2 CHANNEL 51.2 kHz]

This places the analyzer into the two-channel mode. Now you can make network measurements (or two-channel spectrum measurements) from DC to 51.2 kHz.

4. Press < Source >

   [SOURCE ON/OFF]

   [PERIODIC CHIRP]

   [LEVEL]

   < 1 > [Vrms]

This turns on the analyzer's internal source, selects the periodic chirp waveform, and sets the output to 1 Vrms.

The periodic chirp is a fast sine sweep over the current frequency span that repeats with the same period as the time record. Because it's periodic within the time record, no windowing is required.

The periodic chirp can characterize non-linearities because the device under test is excited in exactly the same manner every time record, and the nonlinear distortion averages to its mean value (it does not average to zero).

The periodic chirp is similar to the analyzer's random noise waveform, but the periodic chirp has a much higher rms-to-peak ratio.

5. Press < Input >

   [CHANNEL 1 AUTORANGE]

   [CHANNEL 2 AUTORANGE]

This makes sure the input ranges are set correctly.

6. Press < Trigger >

   [SOURCE TRIGGER]

The trigger for this measurement is from the internal source.

7. Press < Window >

   [UNIFORM]

Since you are using the periodic chirp as the excitation source, there's no need to use a window function — the periodic chirp is periodic within the time record.

The uniform window is really no window at all, since it does not attenuate any part of the time record.

8. Press < Meas Data >

   [FREQUENCY RESPONSE]

This displays the frequency response measurement.

9. Press < Trace Type >

   [LOG MAGNITUDE]

This makes sure the trace type is set to logarithmic magnitude. The analyzer should already be in this state, since this is a default setting.

It's important to understand that selecting a measurement and selecting a trace type are two different things.

Press < Meas Data > to select a measurement for display (such as spectrum or frequency response). Then press < Trace Type > to specify *how* you want the measurement data displayed (for example, the log magnitude or phase of the measurement data).

10. Press < Freq >

    [SPAN]

    < 3 > [kHz]

This changes the measurement frequency span to 3.2 kHz.

You could use another span, but the 3.2 kHz span provides good resolution for a 1 kHz band-pass filter.

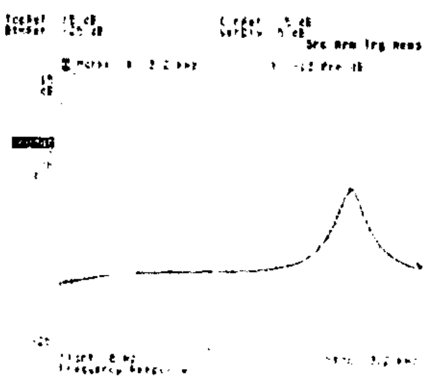
11. Press < Scale >

    [AUTO SCALE]

As you learned in Measurement Task 1, you could use [VERTICAL/DIV] to change the vertical scale. But for this example, the vertical scale selected by pressing [AUTO SCALE] is fine.

12. Press

[X-AXIS LIN/LOG]



This displays the x-axis on a logarithmic scale. The default scale is a linear x-axis.

The logarithmic x-axis is both a convenient and traditional way to display frequency response measurements.

13. Press < Average >

[AVERAGE ON/OFF]

This turns on averaging.

The default setting is for ten rms averages. This is adequate for the example here, so you don't have to select additional items from the averaging menu.

14. Press < Scale >

[AUTO SCALE]

This autoscales again.

15. Press < Marker >

    [MARKER ON/OFF]

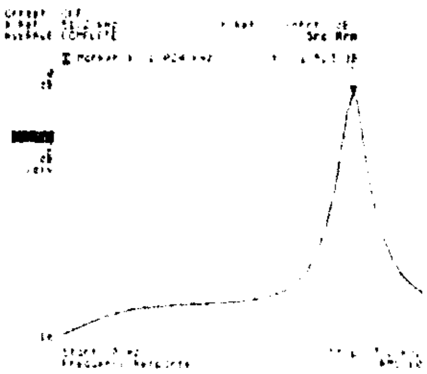
    [MARKER TO PEAK]

    Note the values indicated by the marker's x-axis and y-axis position.

This turns on the marker and moves it to the peak of the frequency response.

The marker's x-axis value is the resonant frequency of the band-pass filter. In this example, the resonant frequency is about 1 kHz.

The y-axis position is the insertion loss (or gain) of the filter. The y-axis value is about −1 dB, so the filter has an insertion loss of about 1 dB.



16. Press [OFFSET]

    [OFFSET ON/OFF]

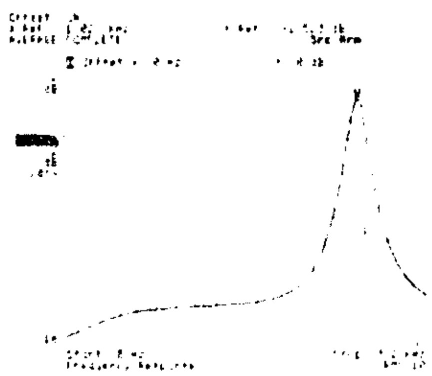
    [OFFSET ZERO]

    [RETURN]

This turns on the offset marker and zeros it at the resonant frequency.

Now you can make measurements relative to the resonant frequency.

Notice that pressing [RETURN] moves you back to the main menu.



17. Press [SEARCH]

[TARGET] < - > < 3 >

[ENTER]

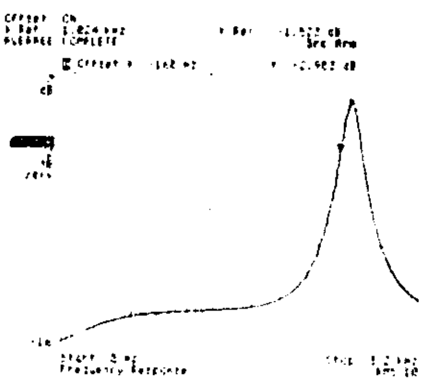
Pressing [Search] puts the analyzer into the search mode. This lets you move the marker quickly to a specific location.

The default setting for the search mode is [Target]. The target feature lets you search for a specific y-axis offset — in the example here, you will search for the 3 dB points on both sides of the resonant frequency.

You can use the [SEARCH] feature with both absolute and offset markers.

18. Press [LEFT]

Note the frequency value indicated by the offset marker's x-axis position.

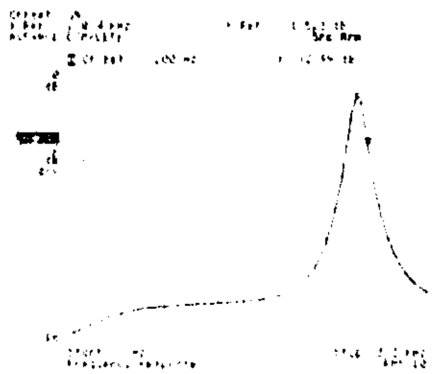


This moves the marker to the −3 dB point to the left of the resonant frequency.

In this example, the offset to the left of the resonant frequency is about 200 Hz.

19. Press [RIGHT]

Again, note the frequency value indicated by the offset marker's x-axis position.



This moves the marker to the −3 dB point to the right of the resonant frequency.

Again, this example shows an offset of 200 Hz to the right of the resonant frequency.

Adding the left offset frequency and the right offset frequency of the −3 dB points gives you the filter's 3 dB bandwidth.

In the example here, the 3 dB bandwidth is about 400 Hz.

20. Press [RETURN]

[OFFSET]

[OFFSET ON/OFF]

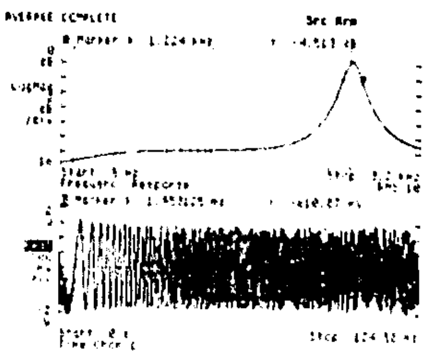
Pressing [RETURN] brings you back to the main marker menu. You can then turn off the offset marker.

You can press < Marker > and use the [SEARCH] and [TARGET] softkeys again, if you want to find the absolute frequency values of the −3 dB points.

21. Press < Format >

[UPPER/LOWER]

< Active Trace >



Pressing < Format > calls up a menu that lets you configure the analyzer's display.

Pressing [UPPER/LOWER] selects the upper/lower display format. Pressing < Active Trace > at this point selects Trace B (the lower one) as the active trace.

Trace A is the same log magnitude display of the frequency response measurement that you were looking at earlier.

Selecting the upper/lower format allows you to display two measurements (or different trace types) at once. In a few moments, you will put the log magnitude of the frequency response on the upper trace, and the phase of the frequency response on the lower trace.

Notice how the currently active trace has a highlighted box around the trace letter. Pressing < Active Trace > switches between Trace A and Trace B.

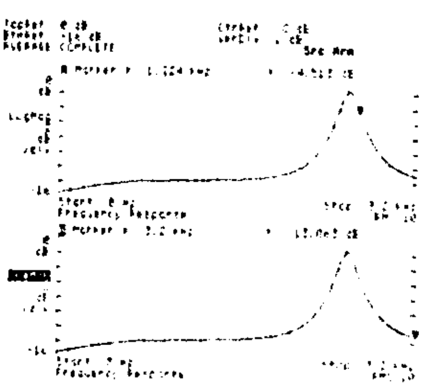
22. Press < Meas Data >

[FREQ RESPONSE]

< Scale >

[X-AXIS LIN/LOG]

[AUTO SCALE]



This puts the frequency response measurement on Trace B.

Because Trace A and Trace B are independent, you have to set up each trace individually.

23. Press < Trace Type >

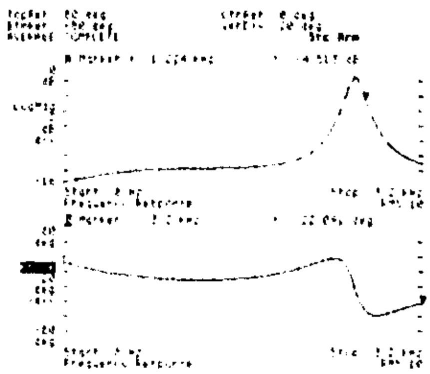
[PHASE]

This chenges the trace type on Trace B to show phase information. You are now looking at phase versus frequency for the test device.

24. Press < Scale >

[AUTO SCALE]

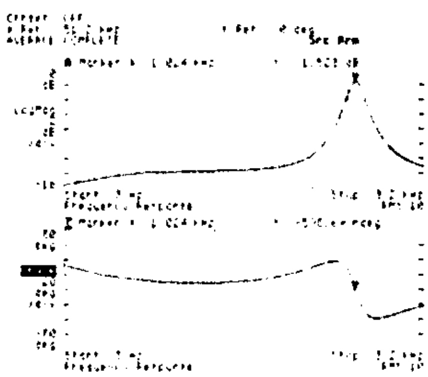
This autoscales the active trace.



25. Press < Marker >

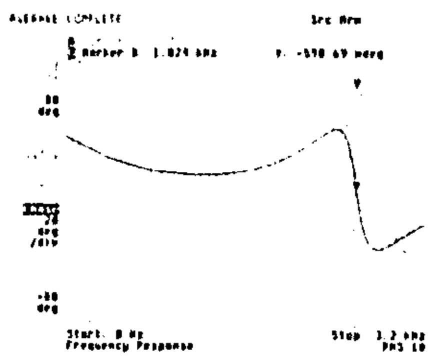
[COUPLED ON/OFF]

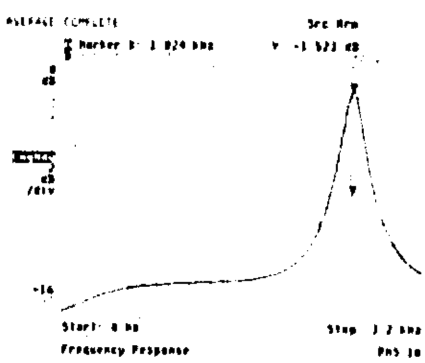
This turns on marker coupling. Marker coupling means that markers for both traces move together when you press the marker direction keys.



26. Press < Format >

[FRONT/BACK]





This selects the front/back display format. The two traces are now overlaid.

Notice how pressing < Active Trace > alternately brightens one trace and dims the other. The brighter trace is the currently active one.

Chapter 9
Impact Testing

Task Overview — This chapter steps you through a series of measurements to find the frequency response of a mechanical structure, using impact testing.

What you will need — gather the following items before starting this task:

- Test structure and support
- Impact hammer with built-in load cell
- Accelerometer with adhesive or threaded stud to mount the transducer
- Signal Conditioning (a current-source supply if you are using ICP devices, or a charge amplifier if you are using piezoelectric devices)
- Appropriate connecting cables

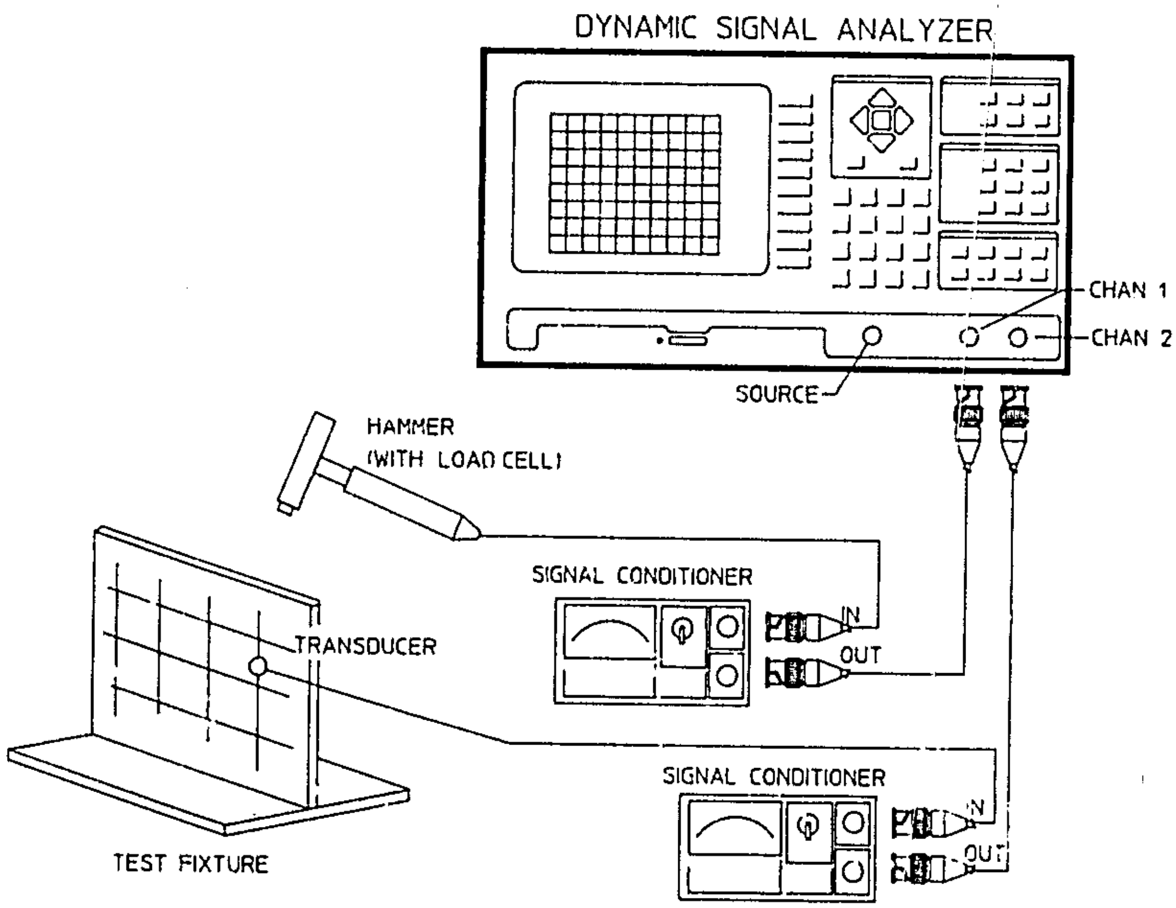
What you will measure — in this task, you will make these measurements:

- Frequency response function for a simple mechanical structure, using impact testing

What you will learn — In this chapter, you will be introduced to the following analyze functions:

- Setting input ranges manually
- Specifying a trigger delay
- Using the force window (and determining its duration)
- Using the exponential window (and determining its decay rate)

Measurement Setup



As you step through the following task, you may find that your measurement results differ slightly from those shown here. Keep in mind that the tasks are designed to help you learn about the analyzer – not to duplicate specific measurement results.

1. If you've already turned on the analyzer, press < Preset >.

   If the analyzer is off, turn it on and wait until it warms up and calibrates. Then press < Preset >.

   As always, you should preset the analyzer before beginning a new measurement task.

2. Connect the impact hammer to the analyzer's channel 1 input.

   Connect the transducer on the test structure to the analyzer's channel 2 input.

   As mentioned in the task overview, you will need to provide proper signal conditioning for the test equipment you are using.

   In our example we used a test kit comprising a hammer with load cell, transducer, and matching ICP current sources..

3. Press < Meas Type >

   [2 Channel 51.2 kHz]

   This places the analyzer in the two-channel mode.

4. Press < Freq >

   [SPAN]

   < 1 > < . > < 6 > [kHz]

   For this example, you will select a frequency span of approximately 2 kHz.

5. Press < Format >

   [UPPER/LOWER]

   To view the results of both the stimulus (the hammer taps) and the resultant response, you'll need to have two traces displayed.

   Trace A will show the hammer taps (in the time domain), while Trace B will show the response (in the frequency domain).

   It's important to monitor the hammer taps to ensure that you've made a clean hit. If the hammer bounces, (causing multiple hits within a single measurement), the response will be in error.

6. Press < Meas Data >

   [TIME CHANNEL 1]

   < Active Trace >

   [FREQUENCY RESPONSE]

This selects the channel 1 time record for display on Trace A.

Pressing < Active Trace > and [FREQUENCY RESPONSE] activates Trace B and then selects frequency response for display on Trace B.

You now have time domain data on one trace and frequency domain data on the other.

7. Press < Input >

   [CHANNEL 1 RANGE]

   Using the numeric keypad, enter a value appropriate for your measurement equipment.

   [CHANNEL 2 RANGE]

   Again, enter a value appropriate for your measurement equipment.

In this example we set the channel 1 range to 0.5V and the channel 2 range to 1V.

You'll have to set the input ranges to accommodate your particular test devices.

8. Press < Trigger >

   [TRIGGER SET UP]

   [LEVEL]

   < 1 > < 0 > [%]

The trigger level allows you to set the threshold level of the trigger to eliminate noise from your measurement.

Setting the trigger level to 10 percent of the input range is a good place to start. This should work well for many setups. However, if there's too much noise, you may need to use different trigger levels.

9. [CHANNEL 1 DELAY]

   < – > < 2 > [mSEC]

   [CHANNEL 2 DELAY]

   < – > < 2 > [mSEC]

   [RETURN]

   [CHANNEL 1 TRIGGER]

10. Press < Window >

   [FORCE EXPO]

   [FORCE CHANNEL1]

   < 1 > < 8 > [mSec]

   [EXPO CHN2]

   < 6 > < 0 > [mSec]

This places a minus 2 millisecond delay (2 millisecond pre-trigger) on channel 1. To be consistent, we've done the same with channel 2.

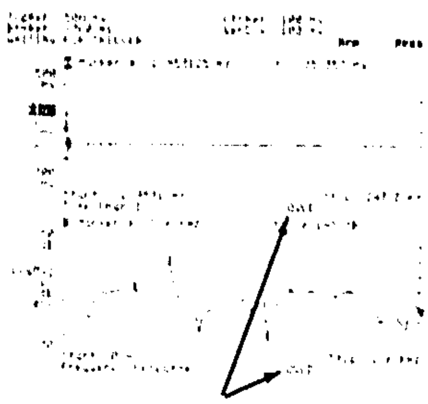
The delay allows you to see the leading edge of the hammer tap. This makes it easier to see if the hammer hit is good.

In this example [FORCE CHANNEL 1] is set to 18 mSec. This establishes a 20 millisecond duration for the force window (–2 mSec pre-trigger plus 18 mSec duration). You can vary the duration to best suit your needs.

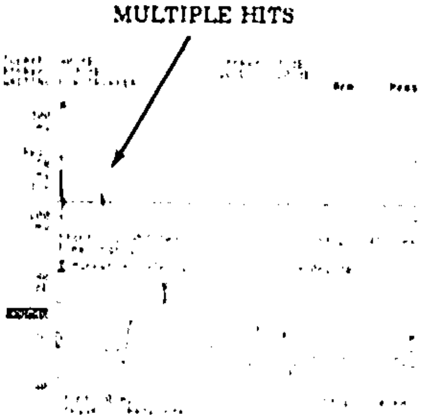
You must also specify an appropriate decay (the time constant) for the exponential window. As a rule of thumb, the period of the time record (displayed at the bottom of trace A) divided by 4 is a good place to start.

To learn more about force and exponential windows, see Hewlett-Packard Application Note 243-3 (*The Fundamentals of Modal Testing*).

11. The analyzer is now set up to receive data from your measurement.



Overloaded Inputs



Multiple Hammer Hits (Bounce)

When making your measurement, you must make sure that the input range for each channel is within the range settings of the analyzer.

Ideally, the dynamic range of the input should be just within the setting of the analyzer. If the stimulus is too high, you'll see an overload message (below each trace).

If the analyzer overloads repeatedly, reset the ranges of either or both channels. Each channel may require a different input range (as did our example here).

As we mentioned before, a good hammer tap should hit the test structure only once (there should be no bounce). After tapping your test object and autoscaling trace A, look to see if there is only one spike present. If there is more than one, it may be necessary to repeat the procedure.

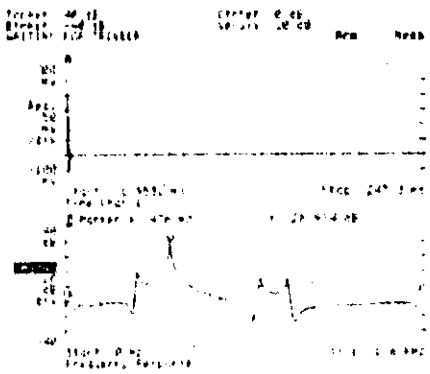
Examine the accompanying figures carefully; compare the test results on Trace B. Notice how overloading and multiple hammer taps have caused distortion in the measurement results.

12. Press < Scale >

[AUTO SCALE]

< Active Trace >

[AUTO SCALE]



Proper Measurement Technique

It's often necessary to press [AUTO SCALE] after each measurement. This provides the best display for each trace.

As in other measurements you have made, you may want to average your results. If you want to average, you will have to turn averaging on and, of course, press < Start > to begin each new series of measurements. To learn more about averaging modal tests, see Application Note 243-3 (*The Fundamentals of Modal Testing*).

Chapter 10

Plotting and Printing Measurement Results

Overview

You can use a variety of HP-IB plotters or printers to show measurement results. Contact your local Hewlett-Packard Sales/Service Office for a listing of currently-supported peripheral devices.

This chapter provides a brief overview of plotting and printing procedures. For more detailed information, see the *HP 35660A Front-Panel Reference.*

Preparing to Plot or Print

1. Connect the plotter/printer to the analyzer's HP-IB connector.

   The HP-IB connector is on the analyzer's rear panel.

2. Press < Local/HP-IB >

   [SYSTEM CONTROLLR]

   This sets the analyzer to be the system controller. The analyzer must have control of the HP-IB bus to plot or print anything.

   This procedure assumes that the analyzer is the only controller on the HP-IB. If you have more than one controller, see the *HP-IB Programming Reference*.

3. Determine the HP-IB address of the plotter/printer.

   You may need to refer to the operating guide for your particular plotter/printer.

   Make sure all external devices on the HP-IB have a unique address.

4. Press < Local/HP-IB >

   [PERIPHERL ADDRESS]

   This calls up a menu that lets you select the address of peripheral devices.

5. Press [PLOTTER ADDRESS]

   or [PRINTER ADDRESS]

   Now enter the appropriate address.

   The address you entered will be retained even if you turn off the analyzer's power.

Plotting or Printing

1. Check to see if the plotter/printer is ready.

   Make sure the plotter/printer is turned on, has paper, and is ON LINE.

2. Press < Plot/Print >

   Then press [PLOT SCREEN]

   or [PRINT SCREEN]

   Softkey labels do not appear on the plotted or printed results. Also, the analyzer does not send a form-feed command to the printer.

   If the "Plot/Print device not present" message appears, you need to:

   - Check the connection between the analyzer and the plotter/printer
   - Make sure that you have the correct address entered for the plotter/printer
   - Make sure there are no other controllers on the HP-IB

3. Press [ABORT PLOT]

   or [ABORT PRINT]

   This aborts the current plot or print in progress.

   When plotting or printing, the analyzer does not respond to any key presses, except [ABORT PLOT] or [ABORT PRINT].

Chapter 11
Save and Recall Operations

Overview

First, a brief overview. The analyzer lets you save (and later recall) the following:

- Traces
- Instrument setup states
- Math functions (and constants)
- Limit tables
- Data tables
- HP Instrument BASIC programs

There are three mass storage devices you can use:

- The analyzer's internal disc (this accepts standard 3.5" floppy discs)
- The analyzer's internal RAM disc (for fast, temporary storage)
- An external disc drive (must be HP-IB compatible)

Before doing any save or recall operations, make sure you've selected the correct mass storage device. Unless you've used a device specifier prefix (such as INT: for internal disc), the save/recall operation will use the currently-selected mass storage device.

Here are the device specifiers:

- INT – for internal disc
- EXT – for external disc
- RAM – for a RAM disc

The filename you use must have no more than ten characters. Also, all characters must be printable.

CAUTION *Files stored in internal RAM disc are temporary and will be lost when you turn off the analyzer. Use an external disc or the analyzer's internal disc drive for permanent storage.*

For detailed information about save and recall operations, see the *HP 35660A Front-Panel Reference.*

Saving and Recalling Traces

You can save (and recall) a trace to one of eight files in the current mass storage device – files 'TRACE1' through 'TRACE8'. You can also save the trace to a file with a name of your own choosing.

Saving and Recalling States

You can save (and recall) the current instrument state (or in other words, its configuration) to a file in the current mass storage device. Later, you can use this file to quickly set up the analyzer.

Here's what else you should know:

- When you save an instrument state, the analyzer remembers most settings, but does not remember some service tests and adjustments settings.
- You can save an instrument state to one of eight files in the current mass storage device – files 'STATE1' through 'STATE8'. You can also save the state to a file with a name of your own choosing.
- Because data tables and limit tables can use large amounts of memory (especially for larger, more complex tables), an "Insufficient disc space" message appears if the setup state is too large to save. If this happens, store each table to a separate file (for example, INT:LIMIT1), if you haven't done so already – do this by pressing < Save > and using the [SAVE DATA TBL] and [SAVE LIMIT] softkeys. Then clear all tables by pressing < Marker Fctn > and using the appropriate softkeys to delete all tables. This avoids duplicating the tables in the memory space allocated for saving setup states. An even better idea would be to store data tables and limit tables to another external mass storage device (to avoid running out of disc space).

Saving and Recalling Math Functions

You can save (and recall) a math function to one of eight files in the current mass storage device – files 'MATH1' through 'MATH8'. You can also save the trace to a file with a name of your own choosing. The five constants associated with each math function are also saved.

Saving and Recalling Limit Tables

You can save (and recall) a limit table to a file with a name of your own choosing.

Saving and Recalling Data Tables

You can save (and recall) a data table to a file with a name of your own choosing.

Typical Save and Recall Tasks

In a few moments you will step through typical save and recall operations, using the analyzer's internal disc drive. After completing these tasks, you can use similar procedures with the analyzer's internal RAM disc or an external disc.

Before starting the save and recall tasks, you must first designate the analyzer's internal disc as the current mass storage device. Afterwards, you will format a blank 3.5" floppy disc (if you haven't done so already).

Selecting the Current Mass Storage Device

1. Press < Recall >

   [STORAGE CONFIG]

   This calls up a menu that lets you designate the current mass storage device (sometimes called the "default" drive).

   The [STORAGE CONFIG] softkey is also available from the < Save > menu.

2. Press [INTERNAL DISC]

   This specifies the analyzer's internal disc drive as the current mass storage device.

3. If you have a disc already formatted, insert the disc in the analyzer's internal drive.

   Press [CATALOG ON/OFF]

   The analyzer will display the contents of the disc drive.

   To remove the catalog, press [CATALOG ON/OFF] once more.

Formatting a Blank Disc

1. Make sure the disc you're going to format is *not* write-protected.

   The write-protect tab should be covering the square hole at the lower-left hand corner of the floppy disc.

2. Insert the disc in the analyzer's internal disc drive.

   Note the eject button. To avoid damaging the disc, do not eject it when the "busy" light is on.

3. Press < Save >

   [DISC FUNCTIONS]

   Pressing < Recall > also displays the disc function menu.

4. Press [FORMAT OPTION]

   < 0 >

   This sets the format option to format ϕ. To learn more about format options, see the *HP 35660A Front-Panel Reference.*

5. Press [INTRLEAVE FACTOR]

   < 1 >

   The interleave factor is the spacing between sectors on a disc. Setting the interleave factor lets you maximize the efficiency of disc operations. Although setting the most efficient interleave factor is not critical for smaller files, it will save lots of time when reading or writing very large files.

   For the analyzer's internal disc drive, setting the interleave factor to 1 provides the most efficient disc operation.

6. Make sure you really want to format this disc.

   Formatting a disc destroys any information previously written to the disc. To abort the operation now, press [CANCEL/RETURN].

   Pressing [ABORT] after formatting has begun will *not* prevent loss of data.

7. Press [START FORMAT]

INT: should appear at the top of the screen (this indicates that the analyzer's internal disc is the default drive).

If RAM: or EXT: appears, press [CLEAR ENTRY] and enter INT: with the alpha-shifted hardkeys.

8. When INT: appears at the top of the screen, press [ENTER]

This starts the formatting operation.

If you'd like, you can use the alpha-shifted keys and enter a disc name after the INT: prefix.

Do not press [ENTER] until the INT: prefix appears.

Saving a Trace

1. Press < Save >

   [SAVE TRACE]

   Make sure the trace you want to save is the active trace. (The active trace is the one with the highlighted trace label.)

2. Press [INTO 'TRACE1']

   This saves the trace to a file called 'TRACE1'.

   You can save a trace to one of eight files in the current mass storage device — files 'TRACE1' through 'TRACE8.'

   You can also save the trace to a file with a name of your own choosing. For more information, see the *HP 35660A Front-Panel Reference*.

Recalling a Trace

1. Press < Recall >

   [RECALL TRACE]

   This calls up a menu that lets you recall a trace.

   You can also get this menu by pressing < Meas Data >, [MORE], and [RECALL TRACE].

2. Press [RCL FROM 'TRACE1']

   This recalls the trace from the file called TRACE1. The recalled trace now appears on the screen in place of the currently-active trace.

   Make sure the trace you're recalling is in the current mass storage device.

   To learn more about recalling traces, see the *HP 35660A Front-Panel Reference.*

Chapter 12
File Utilities, Application Utilities, and Special Functions

Overview

File Utilities

In addition to save and recall operations, the analyzer lets you do the following from a file utilities menu:

- Rename a file
- Delete a file
- Delete all files
- Pack files
- Rename catalog
- Format a disc
- Copy a disc
- Set storage configuration (select a mass-storage device)

At the end of this chapter, you will learn how to copy one disc to another, using the analyzer's internal disc drive. This is particularly useful for making backup copies.

To learn how to format a disc, see Chapter 11, "Save and Recall Operations." For detailed information about file utilities (and to learn about additional utilities), see the *HP 35660A Front-Panel Reference.*

Application Utilities

There's also an application utilities menu. From here, you can do the following:

- List all applications loaded in the analyzer
- Install individual applications
- Load all applications
- Turn on or off the autoload feature (if on, the analyzer loads all applications with the _LD suffix at power-up)

Don't confuse programs that run in HP Instrument BASIC with applications. Although HP Instrument BASIC is itself an application, the programs that run in it are not – rather, they are loaded and saved like data tables and limit tables. For more information, see the *HP Instrument BASIC Getting Started Guide.*

For detailed information about application utilities (and to learn about additional utilities), see the *HP 35660A Front-Panel Reference.*

Special Functions

A special function menu lets you select:

- Calibration options
- Memory allocation
- Turning on the beeper
- Setting the clock and calendar
- Self-test functions

For detailed information about special functions, see the *HP 35660A Front-Panel Reference.*

Disc to Disc Copying

This procedure shows how to copy the contents of one disc to another, using the analyzer's internal disc drive. After completing this task, you can use a similar procedure to do related operations (for example, copying the analyzer's internal RAM disc to a disc in the internal disc drive).

Copying a Disc

1. Press < Save >

   [FILE UTILITIES]

   Pressing < Recall > also displays the file utilities menu.

2. Press [COPY DISC]

   The copy disc routine will erase all files on the destination disc (the disc you're going to copy to files *to*) before performing the copy operation.

   To add files to a disc, copy the files one at a time.

3. Press [SOURCE DISC]

   INT: should appear at the top of the screen (this indicates that the analyzer's internal disc is the default drive).

   If RAM: or EXT: appears, press [CLEAR ENTRY] and enter INT: with the alpha-shifted hardkeys.

4. When INT: appears at the top of the screen, press [ENTER]

   This specifies the source disc as the analyzer's internal disc drive.

   The source disc is the disc you're going to copy *from*.

5. Press [DESTN DISC]

   INT: should appear at the top of the screen (this indicates that the analyzer's internal disc is the default drive).

   If RAM: or EXT: appears, press [CLEAR ENTRY] and enter INT: with the alpha-shifted hardkeys.

6. When INT: appears at the top of the screen, press [ENTER]

   This specifies the destination disc as the analyzer's internal disc drive.

   For this particular operation (copying one internal disc to another), the internal disc is both the source and destination disc.

7. Insert the source disc into the analyzer's internal disc drive.

Make sure the source disc is write-protected.

This disc is write-protected if you can see through the square hole at the lower-left hand corner of the floppy disc.

8. Press [START COPY]

As the operation progresses, the analyzer will ask you to remove the source disc and insert the destination disc (possibly several times).

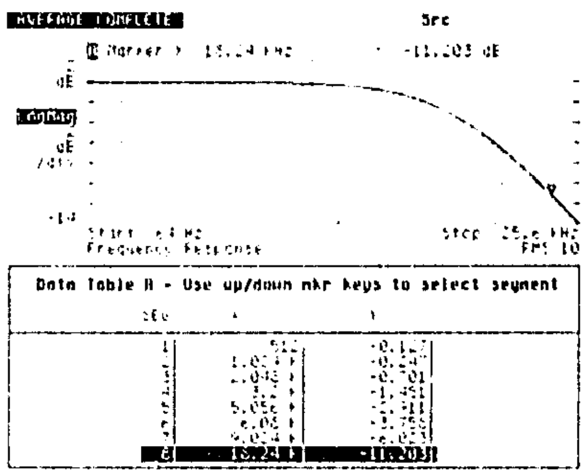
The copied disc is an image copy, so it will be an exact duplicate of the original disc.

Chapter 13
Data Tables

Overview

A data table is a list of x-axis values. For each x-axis value that you enter into a data table, the analyzer calculates the corresponding y-axis value.

There are two data tables; one for Trace A and one for Trace B. When you call up a data table calculation, the analyzer will select the appropriate data table for the trace that's active. And when you turn on data table calculation, the analyzer will automatically calculate the y-axis values.



Typical Data Table

Data tables are useful for quickly characterizing a measurement result by taking a "snapshot" of key points along the x-axis. Data tables save time because they let you easily record measurement results (in numeric form) without having to move the marker around and manually record marker values at various points. For more information about data tables, see the *HP 35660A Front-Panel Reference.*

You can also store a data table and apply it to subsequent measurement traces. This is particularly useful when using the analyzer remotely. For more information, see the *HP 35660A HP-IB Programming Reference.*

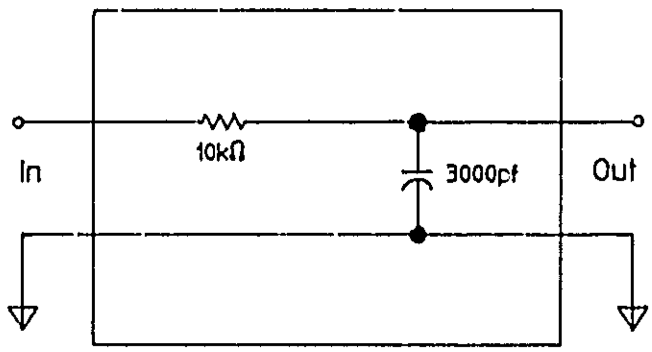
Here's what else you should know about the data table:

- You can specify 401 x-axis values for the data table. Of course, the more values you specify, the longer it takes for the analyzer to fill in the y-axis values.
- If data table calculations are on, the analyzer will update the data table after each measurement. Also, the data table does not have to be displayed for the calculation to occur.
- If data table calculations are off, the analyzer will not update the data table's y-axis values. However, the x-axis entries will remain unchanged.
- The analyzer does not store unit labels in the data table. For example, an x-value of 1.2 kHz is stored simply as "1.2K" and a y-value of −35 dBVrms as "−35." Before recalling a data table for use again, make sure the analyzer is set to use the same vertical units that you used when building the table initially. Otherwise, the calculated data table values will be in different units than the original data table. It's also a good idea to use the same frequency span.
- If you've turned on the offset marker, the analyzer will calculate y-axis values with respect to the current offset zero point (the little square). Otherwise, the analyzer shows absolute y-axis values.
- When editing the data table, use the < ▲ > and < ▼ > hardkeys to move to a particular x-axis value (if there's more than one page, pressing < ▲ > < Fast > moves to the previous page and < ▼ > < Fast > moves to the next page). If you're at the last entry, press < ▼ > to add a new one.
- The analyzer does not show markers for any of the data table points.
- You can use the data table for both frequency domain and time domain measurements.

A Simple Data Table

With this task, you will learn how to create a data table. Although this example uses a 5 kHz low-pass filter, you can use another type of filter if it's more convenient (the goal of this task is to learn how to build a data table – the choice of test device is not important).

The example low-pass filter has just two components and is very easy to build. The same device is also used in the first waveform math task (Chapter 15, "Waveform Math").



Low-pass Filter

A Simple Data Table (continued)

1. Connect the 5 kHz low-pass filter to the analyzer.

If you use another type of filter, you can follow along with this task, but you may need to use a different frequency span.

2. Make a frequency response measurement.

If you need review, use the procedures outlined in chapter 9.

3. Press < Freq >

   [ZERO START]

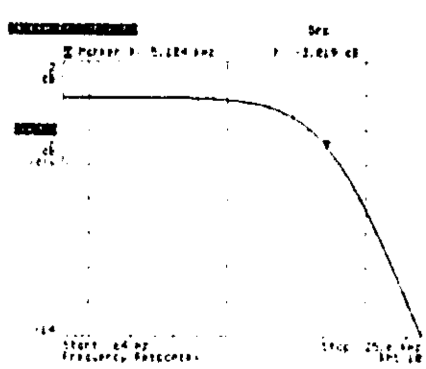
   [SPAN]

   < 2 > < 5 > [kHz]

This sets the span to 25.6 kHz.

4. Press < Scale >

   [X-AXIS LIN/LOG]



This selects the logarithmic x-axis.

You can use the linear x-axis instead, if it's more convenient.

5. Press < Marker Fctn >

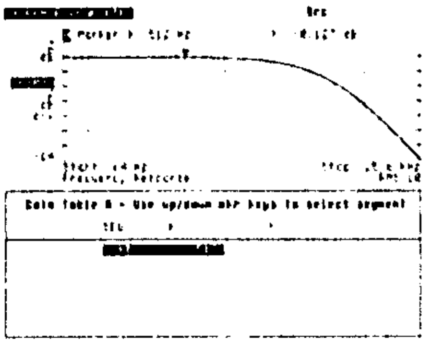
   [DATA TABLE]

To delete an existing data table, press [DELETE ALL] and then [DO DELETE].

6. Move the marker to a point at the left side of the trace.

In the example here, we moved the marker to 512 Hz.

7. Press < Marker Value >



This enters 512 Hz as the first x-axis value in the data table.

You can also enter an x-axis value with the numeric keypad. However, if the x-axis value you specify is not a frequency bin for the current span, the analyzer will use the nearest available bin to calculate the y-axis value.

In the current span, for example, there isn't a bin at 500 Hz (but there are bins at 448 Hz and 512 Hz). So if you enter 500 Hz, the analyzer will put 500 Hz into the data table but the y-axis value it supplies will be the value at 512 Hz, since that's the nearest available frequency bin.

Of course, if you change to a narrower frequency span, the bins will be spaced more closely and the calculated y-axis values in the data table may be somewhat different. So if you're using a data table to characterize similar devices (for example, several filters of the same design), it's best to use a consistent frequency span.

8. Press < ► >

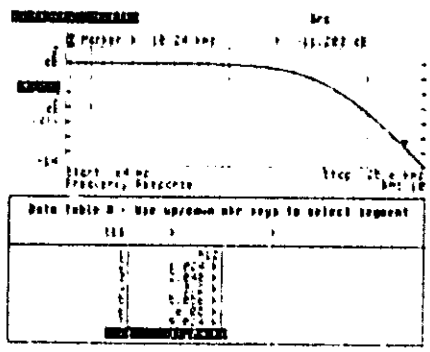
This moves the marker a little to the right.

Press < ▼ >

Pressing < ▼ > adds another x-axis value.

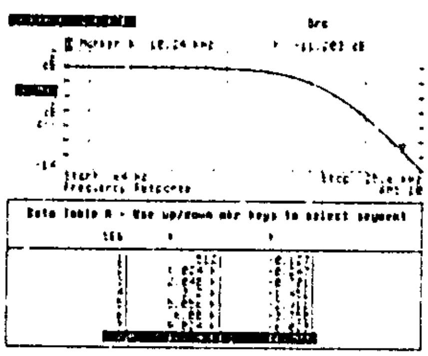
< Marker Value >

9. Repeat the previous step several more times.



Stop after you've entered a few more x-axis values.

10. Press [CALC ON/OFF]



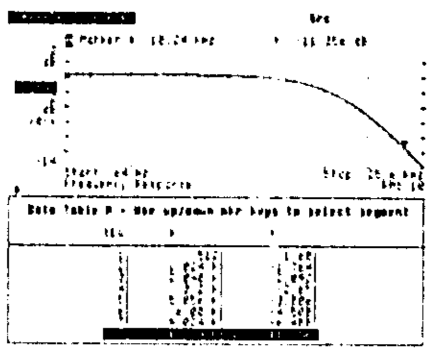
This turns on the calculation and displays the calculated y-axis values. These values will change as new measurement data appears on the active trace.

If data table calculations are on, the analyzer will update the data table after each measurement. Also, the data table does not have to be displayed for the calculation to occur.

If data table calculations are off, the analyzer will not update the data table. The entries in the data table will remain unchanged.

11. Modify the output of the test device

Press < Start >



In the example here, we put a 50 kΩ resistor across the output of the filter.

As the new measurement completes, note the corresponding difference in calculated y-axis values.

12. Press [RETURN TBL DOWN]

This exits the data table display.

To save the data table, press < Save >, [SAVE MORE]. Then press [SAVE DATA TABLE]. The analyzer will assign a filename for you. Then press [ENTER].

To use a different filename, enter a new name with the alphanumeric keypad. Then press [ENTER]. Be sure to use the proper prefix if you want the limit to go to a different (non-current) mass storage device.

13. Press < Format >

[SINGLE]

This returns to the single-trace display format.

Chapter 14
Limit Tables

Overview

A limit table is a list of values (referenced to their respective x and y coordinates) that the analyzer uses to compare with a current measurement or a stored trace. A limit appears as a line (or lines) defined by a series of line segments. These line segments are defined by points that you specify for each limit table.

Limit tables are useful for go/no go checking — they quickly tell you if a particular measurement result passes or fails the limits outlined with a particular limit table.

You can create both upper and lower limit lines for each limit table. When you turn on the limit testing feature, the analyzer indicates a "fail" condition if the trace you're testing exceeds an upper limit (or goes below a lower limit). If the trace is within the limit lines, the test passes. By the way, you don't have to use both upper and lower limit lines — for some types of testing, it may be more convenient to use only upper (or lower) limits.

Here's what else you should know:

- The analyzer does not store unit labels in the limit table. For example, an x-value of 1.2 kHz is stored simply as "1.2K" and a y-value of −35 dBVrms as "−35." Before using a limit table again, make sure the analyzer is set to use the same vertical units that you used when building the table initially. Otherwise, the limit testing will not work properly. It's also a good idea to use the same frequency span.
- When editing the limit table, use the < ▲ > and < ▼ > hardkeys to move to a particular segment (if there's more than one page of segments, pressing < ▲ > < Fast > moves to the previous page and < ▼ > < Fast > moves to the next page). If you're at the last segment, press < ▼ > to add a new segment.
- When adding a new segment, x-start and ystart values are copied from the x-stop and y-stop values of the previous segment. This lets you conveniently add a connecting segment to the previous one. There's no need to re-enter the x-start and y-start values — simply move the marker (with the < ► > hardkey) to the desired end point for the new segment. Then press [X-STOP] < Marker Value > and [Y-STOP] < Marker Value > .
- To copy a limit from Trace A to Trace B, use < Save > and related softkeys to save the limit for Trace A to a file (such as 'LIMIT1'). Then make Trace B active. Now, using < Recall >, recall 'LIMIT1' (you may have to backspace and use the numeric keypad to specify 'TRACE1'). You now have identical limits in both tables. To save the newly-created Trace B limit, again use < Save > and related softkeys; this time, save the limit to 'LIMIT2.'

For more information about limit tables, see the *HP 35660A Front-Panel Reference.*

To learn about operating the analyzer remotely, see the *HP 35660A HP-IB Programming Reference.*

A Simple Limit Table (upper limit only)

With this task, you will learn how to create a simple limit table (using just one limit line) to test the spectral purity of a sine source. Specifically, you're going to check the absolute amplitude of the second and third harmonics.

A Simple Limit Table (continued)

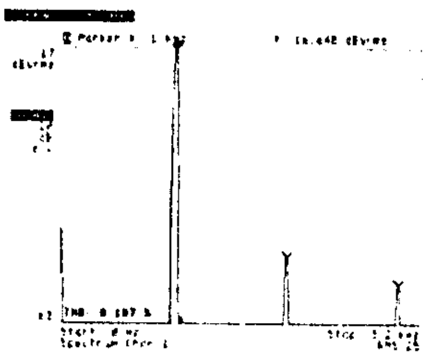
1. Connect a sine source to the analyzer's channel 1 input.

   Set the output to 1 kHz.

In the example here, we've used an external oscillator.

If you use the analyzer's internal source, you can overload the analyzer to simulate a sine source with prominent harmonics. See Chapter 4, "Spectral Purity of a Sine Wave."

2. Make a spectrum measurement of the 1 kHz test signal.

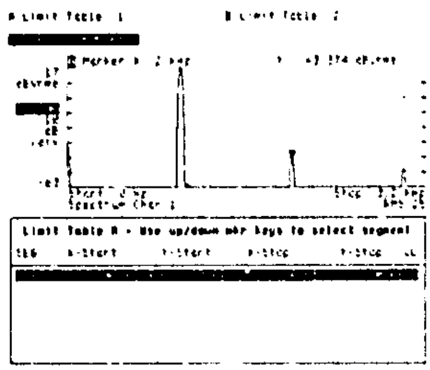


Use a span of 3.2 kHz.

If you need review, use the procedures outlined in chapter 4.

3. Press < Marker Fctn >

   [LIMIT]



Note how a limit table appears at the bottom of the screen. The analyzer automatically shifts to the upper/lower display format.

4. Press [LIMIT CONFIG]

This defines the x-axis starting point of the upper limit segment.

   [X-START]

   < 1 > < . > < 8 > [kHz]

1.8 kHz is a convenient starting point because it's a little before the second harmonic.

5. Press [Y-START]

This defines the y-axis starting point of the upper limit segment.

   < – > < 4 > < 0 >

   [ENTER]

Entering –40 (dBV) specifies a point just above the current second harmonic.

Since the limit table stores only absolute values, the entry is recorded as –40, not –40 dBV.

6. Press [X-STOP]

This defines the x-axis end point of the upper limit segment.

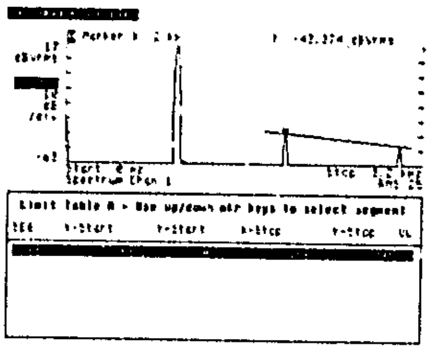
   < 3 > < . > < 1 > [kHz]

3.1 kHz is a convenient stopping point because it's a little after the third harmonic.

7. Press [Y-STOP]

   <−> <5> <0>

   [ENTER]



This defines the y-axis starting point of the upper limit segment.

Entering −50 (dBV) specifies a point just above the current third harmonic.

8. Press [OFFSET]

   [Y ADJUST ALL SEGS]

   <1> <0>

   [ENTER]

The offset feature lets you move an entire limit segment up or down.

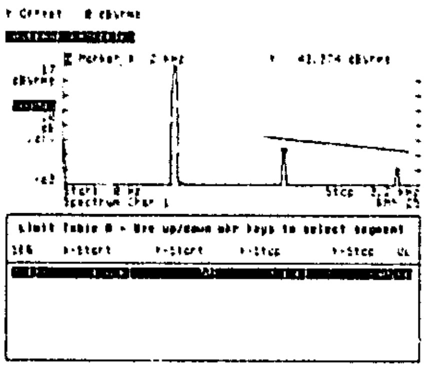
See how easy it was to move the segment up 10 dBV? Also, notice how the analyzer automatically adjusted the values in the limit table to reflect this offset.

In similar fashion, you can use [X ADJUST ALL SEGS] to move a segment left or right.

9. Press < – > < 1 > < 0 >

[ENTER]

[RETURN]



This removes the 10 dBV offset and returns you to the original limit setting.

10. Press [LIMIT CONFIG]

[TEST EVAL ON/OFF]

This turns on (or off) the limit testing.

When testing is on, a PASS or FAIL label appears on the screen.

11. Press [RETURN]

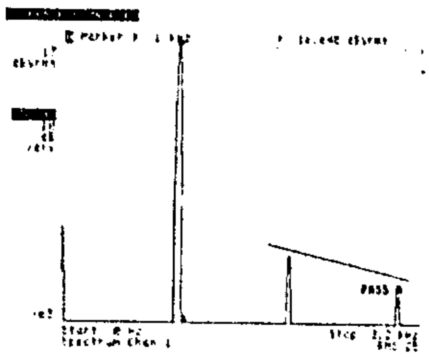
[RETURN TBL DOWN]

This exits the limit table display.

Unless you turn off limit testing (using the [TEST EVAL ON/OFF] softkey), limit testing remains in effect. Also, limit lines remain on the screen unless you turn them off as well.

12. Press < Format >

[SINGLE]



This returns to the single-trace display format.

The test signal should pass the limit test. If it doesn't, go back and readjust the limit segment accordingly.

13. Press < Input >

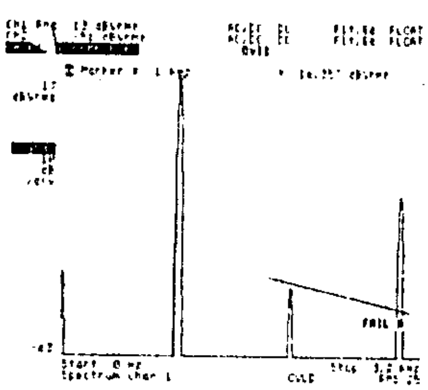
[CHANNEL 1 RANGE]

Then press < v > twice.

This intentionally overloads the analyzer's input to produce a distorted signal.

If you're using the analyzer's internal source (and you can't increase the harmonics any more), return to the limit menu. Then use [OFFSET] and related softkeys to lower the limit segment until the test fails.

14. Press < Start >



As the analyzer completes the measurement, note how the test signal now fails the limit test.

If you're not making an averaged measurement, you do not have to press < Start >.

15. This completes the limit test.

If you're finished with limit testing, go back to the limit menu. Then use [TEST EVAL ON/OFF] and [LINES ON/OFF] to turn off limit testing and the limit lines.

Another Limit Table (upper and lower limits)

With this task, you will learn how to create a simple limit table to check the shape of a band-pass filter. Although this example uses a 1 kHz band-pass filter, you can use another type of filter if it's more convenient (the goal of this task is to learn how to build a limit table – the choice of test device is not important). After a bit of practice, you should be able to build a limit table very easily.

Make sure you've read and understood the previous task ("A simple limit table") before starting this task.

Another Limit Table (continued)

1. Connect a 1 kHz band-pass filter to the analyzer.

   This is the same test device used in Chapter 9, "Filter Characterization."

   If you use another type of filter, you can follow along with this task, but you may need to use a different frequency span.

2. Make a frequency response measurement.

   If you need review, use the procedures outlined in chapter 9.

3. Press < Freq >

   [ZERO START]

   [SPAN]

   < 2 > < 5 > [kHz]

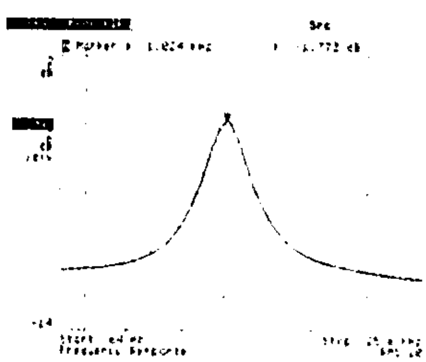
   This sets the span to 25.6 kHz.

4. Press < Scale >

   [X-AXIS LIN/LOG]

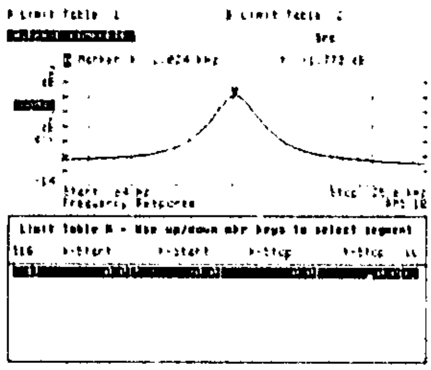
   This selects the logarithmic x-axis.

   You can use the linear x-axis instead, if it's more convenient.



5. Press < Marker Fctn >

   [LIMIT]



To delete an existing limit, press [DELETE] and then [DELETE ALL].

6. Move the marker to the left of the filter peak.

   Press [X-START]

   < Marker Value >

   [Y-START]

   < Marker Value >

This defines the start of the first upper limit segment.

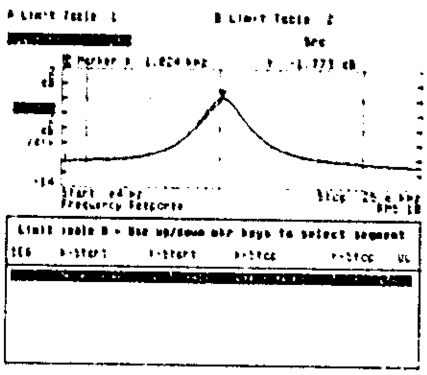
7. Move the marker to the filter peak.

   Press [X-STOP]

   < Marker Value >

   [Y-STOP]

   < Marker Value >



This defines the end of the first upper limit segment.

8. Press < ▼ >

Note how the x-stop and y-stop values are copied from the last segment.

When adding a new segment, the x-start and the y-start values are copied from the preceding segment. This lets you conveniently add a connecting segment to the previous one.

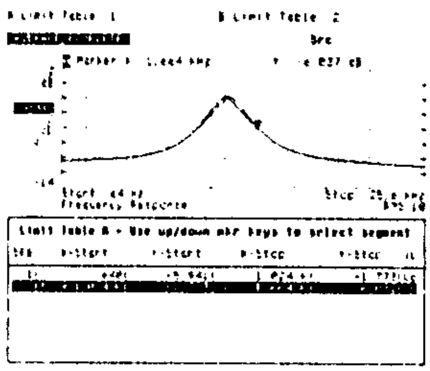
9. Move the marker to the right of the filter peak.

Press [X-STOP]

< Marker Value >

[Y-STOP]

< Marker Value >



The defines the end of the second upper limit segment.

There's no need to re-enter the x-start and y-start values, since the analyzer has already entered them.

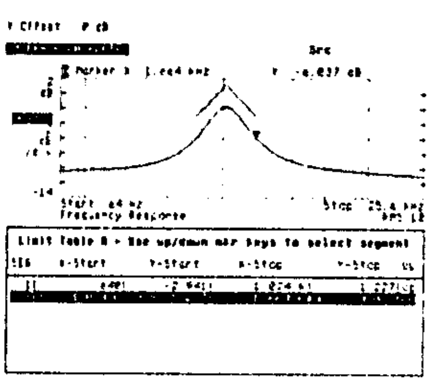
10. Press [OFFSET]

[Y ADJUST ALL SEGS]

< 3 >

[ENTER]

[RETURN]



This establishes an offset of 3 dB.

Later, you will use another offset to move both upper and lower limits at the same time.

11. Again, move the marker to the left of the filter peak.

    Press < ▼ >

    You are about to create the lower limit line.

12. Press [LIMIT UPPER/LOW]

    This designates the following segments as lower limits.

13. Press [X-START]

    < Marker Value >

    [Y-START]

    < Marker Value >

    This defines the start of the first lower limit segment.

14. Move the marker to the filter peak.

    Press [X-STOP]

    < Marker Value >

    [Y-STOP]

    < Marker Value >

    This defines the end of the first lower limit segment.

15. Press < ▼ >

    This adds yet another segment.

16. Move the marker to the right of the filter peak.

    Press [X-STOP]

    < Marker Value >

    [Y-STOP]

    < Marker Value >

    The defines the end of the second lower limit segment.

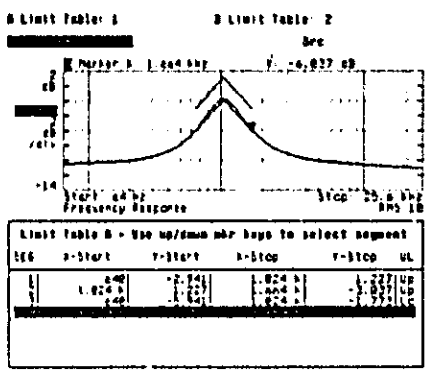
17. Press [OFFSET]

[Y ADJUST ALL SEGS]

< – > < 1 > < . > < 5 >

[ENTER]

[RETURN]

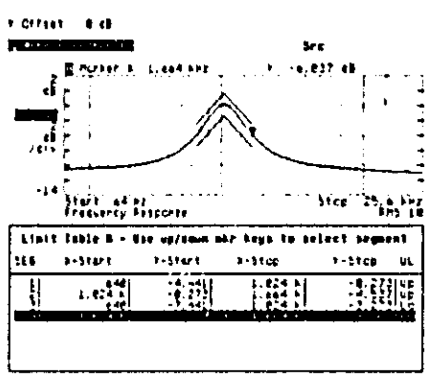


This establishes an offset of –1.5 dB.

The limits are now centered around the frequency response curve. If another filter has a response that differs by more than plus or minus 1.5 dB, the limit test will fail.

18. Press [LIMIT CONFIG]

[TEST EVAL ON/OFF]



This turns on the limit testing.

19. Press [RETURN]

[RETURN TBL DOWN]

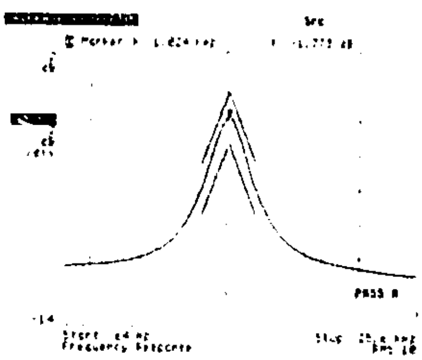
This exits the limit table display.

To save the limit table, press < Save >, [SAVE MORE]. Then press [SAVE LIMIT]. The analyzer will assign a filename for you. Then press [ENTER].

To use a different filename, enter a new name with the alphanumeric keypad. Then press [ENTER]. Be sure to use the proper prefix if you want the limit to go to a different (non-current) mass storage device.

20. Press < Format >

[SINGLE]



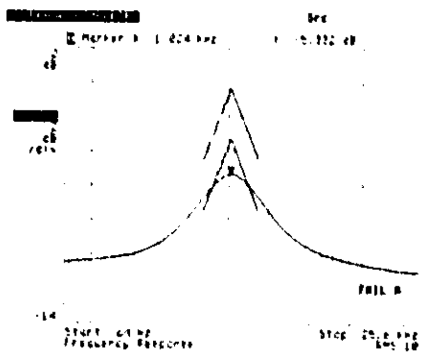
This returns to the single-trace display format.

The test signal should pass the limit test. If it doesn't, go back and readjust the limit segments accordingly.

By the way, the limit lines in this example are not very smooth — that's because each line is defined by only two segments. To get smoother-looking limit lines, you would have to enter more segments to define each limit.

21. If possible, adjust the filter to change its frequency response.

Then press < Start >



As the analyzer completes the measurement, note how the test signal now fails the limit test.

If your filter does not have an adjustment, you can simply attenuate the filter's output to make the response curve fall outside the limit lines. Alternatively, you can go back and change the limit lines (using the offset softkeys).

22. This completes the limit test.

If you're finished with limit testing, go back to the limit menu. Then use [TEST EVAL ON/OFF] and [LINES ON/OFF] to turn off limit testing and the limit lines.

Chapter 15
Waveform Math Operations

Overview

Waveform math lets you perform a variety of operations on current (or stored) traces. Here's what you should know:

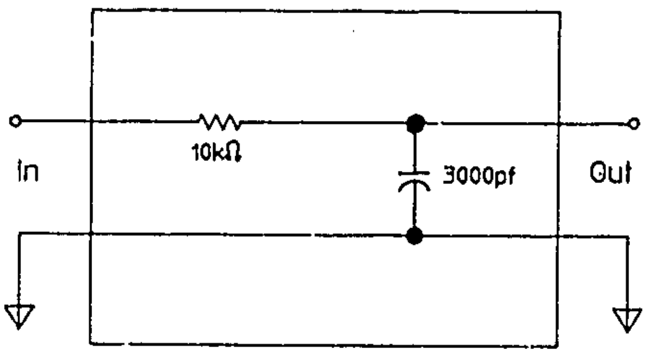
- Functions are specified by entering the definition with operands and operators in infix (standard algebraic) notation. After you enter a function, you can use it to perform waveform (trace) math operations with a combination of measurement data, stored trace data, and constants for display in the currently-active trace.
- Constants can be defined as real or complex quantities. To accomplish this, you are given the choice of defining the real part, imaginary part, magnitude, and phase of the constant — all independently.
- To view the results of a math operation, press < Meas Data > and use the appropriate softkeys to call up the math results.
- To exit any math menu without affecting any function of constant definitions, simply press any hardkey.
- If any data resulting from a math operation overflows the analyzer's floating point limits, an OVFLW message appears. For example, this occurs when a math function involves a divide-by-zero operation.
- When performing a math operation with stored traces, unexpected results may occur if the stored traces are not in the same domain (either the frequency domain or the time domain) or if the stored traces have different frequency spans (frequency-domain traces) or time record lengths (time-domain traces). For example, when multiplying two stored spectrum traces, make sure both traces have the same frequency span and the same start frequency.

To learn more about math operations, see the *HP 35660A Front-Panel Reference.*

A Simple Math Operation

Using waveform math, you will take a frequency response trace and invert it. Then you will multiply the frequency response by the inverted curve to produce a flat frequency response trace. Although this example uses a 5 kHz low-pass filter, you can use another type of filter if it's more convenient (the goal of this task is to learn how to perform a simple waveform math operation – the choice of test device is not important).

The example low-pass filter has just two components and is very easy to build. The same device is also used in Chapter 13, "Data Tables."



Low-pass Filter

A Simple Math Operation (continued)

1. Connect the 5 kHz low-pass filter to the analyzer.

If you use another type of filter, you can follow along with this task, but you may need to use a different frequency span.

2. Make a frequency response measurement.

If you need review, use the procedures outlined in chapter 9.

3. Press < Freq >

   [ZERO START]

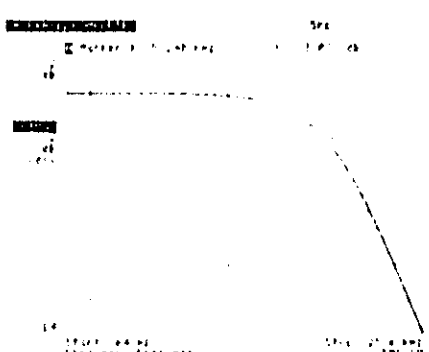
   [SPAN]

   < 2 > < 5 > [kHz]

This sets the span to 25.6 kHz.

4. Press < Scale >

   [X-AXIS LIN/LOG]



This selects the logarithmic x-axis.

You can use the linear x-axis instead, if it's more convenient.

5. Press < Save >

   [SAVE TRACE]

   [INTO FILE 'TRACE1']

   This saves the frequency response trace to the 'TRACE1' file.

6. Press < Math >

   [DEFINE K1]

   Make sure the real part of constant K1 is set to 1.

   If it isn't, use the numeric keypad to enter 1, then press [ENTER].

7. Press [RETURN]

   < Math >

   [DEFINE F1]

   You are about to define math function F1:

8. Press [CONSTANT (K1-K5)]

   [CONSTANT K1]

   [/]

   [STORED DATA]

   [FILE 'TRACE1']

   [ENTER]

   This defines math function F1 as:

   K1/(stored frequency response)

   Since K1 (real part) is 1, the math expression is really:

   1/(stored frequency response).

9. Press < Meas Data >

   [MORE]

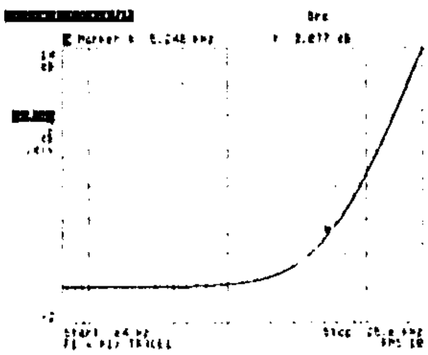
   [FUNCTION (F1-F5)]

   [FUNCTION F1]

   This performs the math function and displays the resulting trace.

10. Press < Scale >

[AUTO SCALE]



You may need to readjust the scale to display the entire trace.

In the example here, we used <⇧> and <⇩> to make additional adjustments to the trace.

Note how the trace is an exact copy of the original frequency response, but inverted. The label underneath the trace shows the math function used to produce this trace.

11. Press < Math >

[DEFINE F2]

[MEAS DATA]

[FREQUENCY RESPONSE]

[*]

[FUNCTION (F1-F5)]

[FUNCTION F1]

[ENTER]

This defines math function F2 as:

(current freq. response)*(F1)

In other words, when you display math function F2, the resulting trace will be the current frequency response data multiplied by function F2 (the inverted frequency response trace).

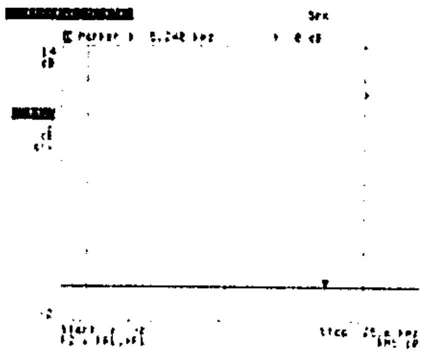
In this example, the purpose of math function F2 is to use the inverted trace to cancel the effects of the low-pass filter (the resulting trace should show a flat response).

12. Press < Meas Data >

[MORE]

[FUNCTION (F1-F5)]

[FUNCTION F2]



This performs the math function and displays the resulting trace.

If you did everything correctly, the resulting trace should show a flat response.

If you press < Start > again, math function F2 will be applied to the next frequency response trace (if you have time, modify the test device and take another frequency response measurement — the resulting trace from function F2 will not be flat anymore).

Another Math Operation

In this task, you will take a spectrum measurement and divide it by $j\omega$ (where $\omega = 2\pi f$). This operation is useful for mechanical measurements because it converts signals proportional to acceleration to a signal proportional to velocity.

Although this math operation is performed entirely in the frequency domain, the effect is the same as integrating a time-domain signal. Conversely, multiplying by $j\omega$ has the effect of differentiating a time-domain signal.

Another Math Operation (continued)

1. Connect a BNC cable from the analyzer's source to the channel 1 input.

   Turn on the periodic chirp and set it at 1 Vrms.

   By now, you should already know how to do this. If you need review, go back and complete the tasks in chapter 2, chapter 4, and chapter 9.

2. Select the Uniform window; then turn on rms averaging.

   Ten rms averages is sufficient to produce a smooth trace.

3. Press < Freq >

   [SPAN]

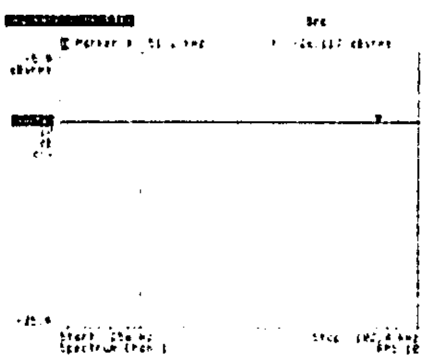
   [FULL SPAN]

   The span should be set to 102.4 kHz.

4. Press < Scale >

   [X-AXIS LIN/LOG]

   This selects the logarithmic x-axis.



5. Press < Math >

   [DEFINE F1]

You are about to define math function F1.

6. Press [MEAS DATA]

   [SPECTRUM CHANNEL 1]

   [/]

   [JOMEGA]

   [ENTER]

This divides the current channel 1 spectrum by $j\omega$.

7. Press < Meas Data >

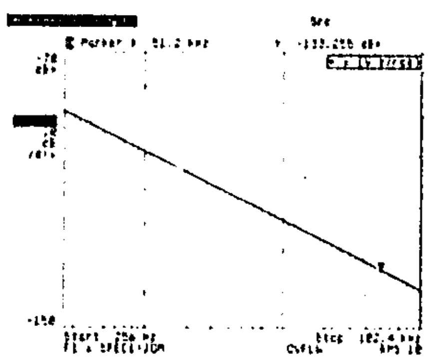
   [MORE]

   [FUNCTION (F1-F5)]

   [FUNCTION F1]

This shows the results of the math function.

The "OVFLW" message underneath the trace indicates an overflow during a math operation. This is normal for this operation, since $j\omega$ is 0 at 0 Hz, and thus generates a divide-by-zero error.



8. Press < Scale >

   [AUTO SCALE]

You may need to readjust the scale to display the entire trace.

Index

N

O

P

R

S

T

U

W

Z

Hewlett-Packard Sales and Service Offices

To obtain Servicing information or to order replacement parts, contact the nearest Hewlett-Packard Sales and Service Office listed in HP Catalog, or contact the nearest regional office listed below:

In the United States

California
P.O Box 4230
1421 South Manhattan Avenue
Fullerton 92631

Georgia
P.O. Box 105005
2000 South Park Place
Atlanta 30339

Illinois
5201 Tollview Drive
Rolling Meadows

New Jersey
W. 120 Century Road
Paramus 07652

In Canada
Hewlett-Packard (Canada) Ltd.
17500 South Service Road
Trans-Canada Highway
Kirkland, Quebec H9J 2M5

In France
Hewlett-Packard France
F-91947 Les Ulis Cedex
Orsay

In German Federal Republic
Hewlett-Packard GmbH
Vertriebszentrale Frankfurt
Berner Strasse 117
Postfach 560 140
D-6000 Frankfurt 56

In Great Britain
Hewlett-Packard Ltd.
King Street Lane
Winnersh, Wokingham
Berkshire RG11 5AR

In Other European Countries
Switzerland
Hewlett-Packard (Schweiz) AG
7, rue du Bois-du-Lan
Case Postale 365
CH-1217 Meyrin

In All Other Locations
Hewlett-Packard Inter-Americas
3155 Porter Drive
Palo Alto, California 94304