ELECTRICAL ENGINEERING DEPARTMENT

LABORATORY MANUAL

University of Wisconsin – Platteville

Eleventh Edition

2008

TABLE OF CONTENTS

First Edition released September 1989.

Second Edition, released January 1991, added material to sections on network analyzer and measurement of s-parameters.

Third Edition, released January 1992, corrected errors in section on use of oscilloscope.

Fourth Edition, released August 1993, added section on laboratory safety.

Fifth Edition, released January 1994, deleted all references to specific Electrical Engineering courses.

Sixth Edition, released August 2001, deleted sections on Tektronix Model 2213 Oscilloscope and Gain-Phase Meter, added sections on the Tektronix Model TDS 210/220 oscilloscopes and Fluke/Philips Model PM 5193 Function Generator, enhanced the sections on measurement of s-parameters, added a section on the measurement of input impedances with a network analyzer, and updated the section on PC Board Fabrication.

Seventh Edition, released September 2004, deleted references to David Wood and Software Engineering.

Eighth Edition, released February 2005, added instructions for the Agilent 4395A and Hewlett-Packard 4195A network analyzers.

Ninth Edition, released July 2006, added instructions for careful use of Agilent 41800A active probe, and added sections on the Agilent E3630A Triple-Output DC Power Supply and Tektronix CFG253 3-MHz Function Generator.

Tenth Edition, released July 2008, refined the instructions for the setup and calibration of the Agilent 4395A Network/Spectrum/Impedance Analyzer.

Eleventh Edition, released August 2008, added a section on the Agilent E4411B 9-kHz to 1.5-GHz Spectrum Analyzer and Agilent E4403B 9-kHz to 3-GHz Spectrum Analyzer, and the section on the measurement of s-parameters was updated for the Agilent 4395A network analyzer.

THE DIGITAL MULTIMETER (DMM) AND POWER SUPPLY

I. Purpose

In this chapter you will learn to use your digital multimeter (DMM), the HP 6236B triple-output power supply, and the Agilent E3630A triple-output power supply in the laboratory.

II. Use of the HP 6236B power supply

A. Description

The power supply has three output jacks (+6 V, +20 V, and –20 V) and two analog meters on its front panel. The meter on the left measures the output voltage and the meter on the right measures the output current.

B. Operation

1. Flip on the "Line" switch. The lamp next to the switch should glow. If it does not, check the power cord and the laboratory bench power outlet switches.

2. Turn the +6-V Voltage and ±20-V Voltage knobs fully counterclockwise (CCW).

3. The +6-V output:

   a. Select +6 V with the meter switch in the upper left corner of the front panel. (The meters on the front panel will now show the voltage and the current at the +6-V output jack.)

   b. Set the output voltage to ≈ 5 V with the + 6-V Voltage knob on the left side of the panel and the "Volts" meter (use the bottom scale on the meter).

   c. Measure the voltage between the +6-V output jack and the chassis ground jack (⏚) with your DMM.

   d. Measure the voltage between the +6-V output jack and the common jack with your DMM.

   e. Compare the results of steps II.B.3.c and II.B.3.d.

   f. Turn the +6-V Voltage knob fully CCW.

4. The ±20-V outputs

   a. Turn the "Tracking Ratio" knob (upper right corner) fully clockwise (CW).

   b. Select the +20-V meter.

c. Set the output voltage to ≈ 12 V using the ±20-V Voltage knob and the front panel meter (use the top meter scale).

d. Select the –20-V meter and note the meter reading. Compare it to the value obtained in step c.

e. Measure the voltage from both the +20-V and the –20-V output jacks to the chassis ground (⏚) jack with your DMM.

f. Measure the voltage from both the +20-V and the –20-V output jacks to the common jack with your DMM.

g. Compare the results of steps II.B.4.e and II.B.4.f.

h. Change the setting of the Tracking Ratio knob. Measure the voltage from both the +20-V and –20-V output jacks to the common jack with your DMM. Observe the effect of the Tracking Ratio knob on the output voltage values.

III. Use of the Agilent E3630A power supply

A. Description

The power supply has three output jacks (+6 V, +20 V, and –20 V), a common (COM) jack, a chassis-ground jack (⊥), and two digital meters on its front panel. The meter on the left measures the output voltage and the meter on the right measures the output current.

B. Operation

1. Press the “Power” switch. Numbers should appear on the digital meters. If the meters remain blank, check the power cord and the laboratory bench power outlet switches.

2. Turn the “+6” and “±20” knobs (upper right corner of front panel) fully counterclockwise (CCW).

3. The +6-V output:

a. Select +6 V with the “+6” switch at the bottom of the front panel. (The meters on the front panel will now show the voltage and the current at the +6-V output jack.)

b. Set the output voltage to ≈ 5 V with the “+6” knob in the upper right corner of the panel and the “Volts” meter.

c. Measure the voltage between the +6-V output jack and the chassis ground jack (⊥) with your DMM.

d. Measure the voltage between the +6-V output jack and the common jack with your DMM.

e. Compare the results of steps III.B.3.c and III.B.3.d.

f. Turn the +6-V Voltage knob fully CCW.

4. The ±20-V outputs

a. Turn the “Tracking ratio” knob (upper right corner) fully clockwise (CW).

b. Select the +20-V meter by pressing the “+20” button at the bottom of the front panel..

c. Set the output voltage to ≈ 12 V using the “±20” knob and the front-panel meter.

d. Select the –20-V meter by pressing the “-20” button. Note the meter reading. Compare it to the value obtained in step c.

e. Measure the voltage from both the +20-V and the –20-V output jacks to the chassis ground (⊥) jack with your DMM.

f. Measure the voltage from both the +20-V and the –20-V output jacks to the common jack with your DMM.

g. Compare the results of steps III.B.4.e and III.B.4.f.

h. Change the setting of the “Tracking ratio” knob. Measure the voltage from both the +20-V and –20-V output jacks to the common jack with your DMM. Observe the effect of the “Tracking ratio” knob on the output voltage values.

THE OSCILLOSCOPE AND FUNCTION GENERATOR

I. Purpose

This chapter will introduce you to the use of the Tektronix Model TDS 210 60-MHz Oscilloscope, the Tektronix Model TDS 220 100-MHz Oscilloscope, the Hitachi V-1100A 100-MHz Oscilloscope, the Wavetek Model 190 20-MHz Function Generator, the Fluke/Philips Model PM 5193 50-MHz Function Generator, and the Tektronix Model CFG253 3-MHz Function Generator. To learn to use these instruments, you must go into the laboratory and observe the effect of each knob and switch. Merely reading this chapter is not sufficient.

II. Getting started

First, display a signal from the function generator on the oscilloscope screen.

A. Oscilloscope

1. Tektronix Model TDS 210/220

a. Turn on the oscilloscope using the power button on the top of the unit. If the oscilloscope does not turn on, check its power cord and the on/off switches for the laboratory bench power outlets.

b. The display should show the Power On Self Test. All of the tests should indicate PASSED. Pressing any button will immediately replace the Power On Self Test display with the regular oscilloscope display. If no button is pressed, the Power On Self Test display will be replaced by the regular oscilloscope display automatically in less than one minute.

c. If the language on the display is ***not*** to your liking:

(1) Press the *Utility* menu button on the upper right half of the front panel. The *Utility* menu will be displayed on the right side of the screen.

(2) Press the button next to the *Language* block (lowest on the screen) until the desired language appears.

d. Configure channel 1 of the oscilloscope as follows:

(1) Press the CH 1 menu button. The menu for channel 1 appears on the right side of the screen. Press the buttons next to the menu items until the menu blocks read as follows:

(a) Coupling: AC.

(b) BW Limit: ON and "20 MHz".

(c) Volts/Div: Coarse.

(d) Probe: 1X.

(2) Turn VOLTS/DIV *knob* (**not** button!) until 5.00V appears in lower left corner of screen next to CH1.

(3) Turn the Vertical Position *knob* for channel 1 until the 1→ on the left side of the screen points to the screen's horizontal center line. The *trace* (solid dark line) should now be a straight line across the center of the screen.

2. Hitachi V-1100A

a. Turn on the oscilloscope with the POWER button in the lower left hand corner of the front panel. If the oscilloscope does not turn on, check its power cord and the on/off switches for the laboratory bench power outlets.

b. Adjust TRIGGER controls:

(1) MODE button – press AUTO.

(2) SOURCE button – press CH1.

(3) COUPLING button – press AC.

c. Sweep time (TIME/DIV) knob to 0.2 ms.

d. VERT MODE controls – depress CH1 and BW LIMIT buttons.

e. AC GND DC switch (directly above the CH1 BNC connector) to AC.

f. Channel 1 sensitivity knob (VOLTS/DIV) to 5.

g. *If there is no trace on the screen but the* A TRIG'D *lamp is on*, perform the following steps:

(1) Depress and hold in the TRACE FINDER button (beneath the display).

(2) Adjust CH1 POSITION knob (above the VERT MODE buttons) until a trace appears on the screen.

(3) *If there is still no trace on the screen, GET HELP!*

h. Adjust the FOCUS knob (just right of POWER button) to obtain sharpest line possible on the screen.

i. Adjust A&B INTENSITY knob (below display screen) to obtain trace of suitable brightness.

B. Function generator

1. Wavetek Model 190

    a. Turn on the function generator by pressing the power switch. The lamp above the frequency dial on the left side of the control panel should light. If it does not, check the power cord and the laboratory bench power outlet switches.

    b. Set the generator output frequency to 1 kHz:

        (1) Set the frequency dial to 1.0.

        (2) Set FREQ MULT switch (to the right of the dial) to 1 k.

    c. Set MODE switch to CONT to produce a continuous signal.

    d. Select a symmetrical triangle wave:

        (1) FUNCTION switch to triangle (⋀⋁).

        (2) SYM button out.

    e. DC OFFSET button out (no DC offset).

    f. Set maximum output level:

        (1) All OUTPUT ATTEN buttons out.

        (2) AMPLITUDE knob fully clockwise (CW).

    g. Display function generator signal on oscilloscope screen

        (1) Connect the FUNC OUT connector on the function generator to the CH 1 input connector on the oscilloscope using a coaxial cable with BNC connectors on both ends. The triangle wave produced by the function generator should appear on the oscilloscope screen.

        (2) Adjust CH 1 VOLTS/DIV, SEC/DIV, and AUTO INTENSITY knobs to produce a display to your liking.

2. Tektronix FG-504

    a. Turn on the function generator by pulling out the power switch on the mainframe at the right edge of the front panel. The lamp next to the frequency dial on the left side of the control panel should light. If it does not, check the power cord and the laboratory bench power outlet switches.

    b. Set the generator output frequency to 1 kHz:

        (1) Set the frequency dial to 1.

(2) Set MULTIPLIER switch (to the right of and below the dial) to 10^3.

c. Press FREE RUN button below the frequency dial to produce a continuous signal.

d. Create a symmetrical triangle wave:

(1) Push triangle (∧∨) button (to the right of the frequency dial).

(2) VAR SYMM button in.

e. OFFSET button (center of AMPL knob above the OUTPUT BNC connector) in (to produce no DC offset).

f. Set maximum output level:

(1) Press 0dB button.

(2) AMPL knob fully clockwise (CW).

g. Display function generator signal on oscilloscope screen

(1) Connect the output on the function generator to the CH 1 input connector on the oscilloscope using a coaxial cable with BNC connectors on both ends. The triangle wave produced by the function generator should appear on the oscilloscope screen.

(2) Adjust CH 1 VOLTS/DIV, SEC/DIV, and AUTO INTENSITY knobs to produce a display to your liking.

3. Fluke/Philips Model PM 5193

a. Turn on the function generator by pressing the top of the power switch in the lower left corner of the front panel. The display and buttons should light up. If they do not, check the power cord and the laboratory bench power outlet switches.

b. Set the generator output frequency to 1 kHz:

(1) Press the FREQUENCY START button. The frequency display (left side of the display) should go blank.

(2) Press the FREQUENCY Hz/kHz button until the green indicator lamp next to “kHz” on the frequency display (left side of the display) is illuminated.

(3) Press 1.0 ENTER on the keypad on the right side of the front panel.

c. Press the MODULATION OFF button.

d. Select a symmetrical triangle wave by pressing the appropriate (/\/) WAVEFORM button.

e. Press the LEVEL Vdc button until its built-in indicator lamp is off (no DC offset).

f. Set the output signal level:

(1) Press the LEVEL Vpp button. The output level display (right side of display) will go blank.

(2) Type 4.0 on the keypad at the right side of the front panel and press the ENTER key to create a 4-V p-p signal.

g. Display function generator signal on oscilloscope screen

(1) Connect the output connector on the function generator to the CH 1 input connector on the oscilloscope using a coaxial cable with BNC connectors on both ends. The triangle wave produced by the function generator should appear on the oscilloscope screen.

(2) Adjust CH 1 VOLTS/DIV, SEC/DIV, and AUTO INTENSITY knobs to produce a display to your liking.

4. Tektronix Model CFG253

a. Turn on the function generator by pressing the POWER switch on the left side of the control panel. The lamp above the POWER switch should light. If it does not, check the power cord and the laboratory bench power outlet switches.

b. Set the generator output frequency to 1 kHz:

(1) Set the FREQUENCY dial (lower right corner of the panel) to 1 (the heavy black line between .9 and 1.2).

(2) Press the 1K RANGE button (top of the panel).

c. Select a symmetrical triangle wave:

(1) Press the triangle (/\/) FUNCTION button.

(2) SYMMETRY button (bottom of the panel) out.

d. DC OFFSET knob (upper left of the panel) in (no DC offset).

e. Set maximum output level:

(1) VOLTS OUT button (bottom center of the panel) out.

(2) AMPLITUDE knob (upper left corner of the panel) fully clockwise

(CW).

f. Display function generator signal on oscilloscope screen

(1) Connect the MAIN connector on the lower left of the function generator to the CH 1 input connector on the oscilloscope using a coaxial cable with BNC connectors on both ends. The triangle wave produced by the function generator should appear on the oscilloscope screen.

(2) Adjust CH 1 VOLTS/DIV, SEC/DIV, and AUTO INTENSITY knobs to produce a display to your liking.

III. Oscilloscope familiarization

A. Tektronix Model TDS 210/220

1. Trigger

a. Function: The trigger circuit of the oscilloscope initiates the horizontal trace. When properly adjusted, it starts the trace at the same point in the periodic input signal each time, so that successive traces lie on top of each other for a clean, easily-interpreted display. The trigger controls operate by sensing the amplitude and slope of the trigger signal, which may be the signal displayed, the 60-Hz power line signal, or an external trigger signal supplied by the oscilloscope user to the EXT TRIG input connector.

b. LEVEL knob: sets the trigger signal level at which the sweep trigger occurs. If the CH 1 or CH 2 signal is selected as the trigger signal, this level is displayed by the arrow (←) on the right side of the screen.

c. TRIGGER MENU button: When pressed this displays the trigger menu on the right side of the screen. The menu features are selected or activated by pressing the buttons adjacent to the menu blocks:

(1) Edge/Video: If Edge is selected, then the oscilloscope triggers off of the rising or falling edge of the signal.

(2) Slope: Pressing the button toggles the trigger between a rising (/) or falling (\ ) slope of the trigger signal.

(3) Source: Selects between the signal on CH 1, CH 2, Ext (a signal provided to the EXT TRIG input), Ext/5, or AC line (the 60-Hz signal powering the oscilloscope).

(4) Mode: Selects between Auto, Norm, and Single. When Single is selected, the oscilloscope will sweep only once (instead of repetitively) whenever the RUN/STOP button is pressed.

(5) Coupling: Selects how the trigger signal is connected to the trigger

circuit — AC (DC level is blocked), DC, Noise Reject, HF Reject, and LF Reject.

Experiment with the trigger controls.

2. Horizontal controls

   a. POSITION knob: moves the trace horizontally on the screen.

   b. HORIZONTAL MENU button: Displays the horizontal menu on the right side of the screen. The menu features are selected or activated by pressing the buttons adjoining the menu blocks. The most important menu item is the one on the bottom — Trig Knob. The block is large enough to include two activation buttons, but only the top button has an effect. Pressing the top button toggles the function of the TRIGGER LEVEL knob (right side of front panel) between controlling the trigger level and controlling the HOLDOFF (the time between horizontal sweeps).

   c. SEC/DIV knob: Selects a calibrated number of seconds for the sweep to cover each horizontal division on the display screen.

Experiment with the horizontal controls.

3. Vertical controls

   a. POSITION knobs adjust the vertical positions of the traces on the screen.

   b. VOLTS/DIV knobs adjust the (calibrated) number of divisions a signal will subtend on the screen. The value(s) selected is (are) displayed in the bottom left corner of the screen.

   c. MENU buttons (CH 1 and CH 2).

      (1) Toggles the trace (CH 1 or CH 2) on and off.

      (2) Displays the CH 1 or CH 2 trace menu on the right side of the screen. The buttons adjoining the menu block activate or select the features described within each block:

         (a) Coupling: Sets how the input signal to the channel (via the CH 1 or CH 2 input connector) is connected to the oscilloscope circuits. Selects between AC (DC level is blocked), DC, and Ground (oscilloscope circuit is disconnected from the input signal and connected to ground; the input signal itself is *not* grounded; flat trace is displayed).

         (b) BW Limit: When selected ON, bandwidth of the oscilloscope is reduced from 60 MHz (TDS 210) or 100 MHz (TDS 220) to 20 MHz (reduces interference from 90-MHz WSUP-FM signal).

(c) Volts/Div: Selects between Coarse or Fine steps when turning the VOLTS/DIV knob.

(d) Probe: Compensates the oscilloscope for probes that attenuate the input signal. 1X, 10X, 100X, and 1000X can be selected. Regular coaxial cables (with BNC connector or clip leads at one end) are 1X probes. High-impedance oscilloscope probes are 10X. UW–P has no 100X or 1000X probes. *This setting must match the probe used or the amplitude displayed on the screen will not be correct!*

(e) Invert: When selected ON, multiplies the input signal by –1 (*i.e.*, flips it upside down).

Demonstration: Push the DC OFFSET button on the Wavetek function generator. Turn the DC OFFSET knob on the function generator and observe the effect on the signal displayed on the oscilloscope screen for AC, GND, and DC coupling.

Experiment with the vertical mode controls.

B. Hitachi V-1100A

1. Trigger

a. Function: The trigger circuit of the oscilloscope initiates the horizontal trace. When properly adjusted, it starts the trace at the same point in the periodic input signal each time, so that successive traces lie on top of each other for a clean, easily-interpreted display. The trigger controls operate by sensing the amplitude and slope of the trigger signal, which may be the signal displayed, the 60-Hz power line signal, or an external trigger signal supplied by the oscilloscope user to the EXT TRIG input connector.

b. VAR HOLDOFF knob: varies the time between successive horizontal sweeps.

c. LEVEL knob: sets the trigger signal level at which the sweep trigger occurs.

d. SLOPE buttons: commands the oscilloscope to trigger on a positive (/) slope (**+** button) or negative (\ ) slope (**–** button) of the trigger signal.

e. A TRIG'D lamp: lights when triggering is achieved.

f. MODE buttons:

(1) AUTO: the trigger level is selected automatically (the oscilloscope will "free run" if no suitable trigger signal is present).

(2) NORM: the trigger level is adjusted manually.

(3) SGL SWEEP: only one horizontal sweep will occur after the oscilloscope is triggered; used when measuring transient events.

g. The SOURCE buttons select the source of the trigger signal:

(1) CH 1, CH 2, CH 3, and CH 4 buttons — the trigger signal is the input signal on the selected channel.

(2) LINE — the trigger signal is the 60-Hz power line signal; the signal from the function generator will not be displayed neatly on the screen because the generator signal and the power line signal are not in phase.

Experiment with the trigger controls.

2. Horizontal controls

a. H POSITION knob (large gray knob): moves the trace horizontally on the screen.

b. TIME/DIV knob: Selects a calibrated number of seconds for the sweep to cover each division on the display screen. The value selected is shown on the bottom right of the screen.

Experiment with the horizontal controls.

3. Vertical controls

a. POSITION knobs adjust the vertical positions of the traces on the screen. When the CH 2 position knob is pulled out, the trace for the CH 2 signal is *inverted*, *i.e.*, multiplied by –1 (flipped upside down).

b. VOLTS/DIV knobs (light gray) adjust the (calibrated) number of divisions a signal will subtend on the screen. The values selected are displayed in the lower left corner of the screen.

c. Small, dark gray knobs on the VOLTS/DIV knobs:

(1) When rotated CCW, cause sensitivity to be reduced by an uncalibrated amount. Turned fully clockwise until click gives calibrated measurements.

(2) When pulled out, cause VOLTS/DIV to be reduced by a factor of 5.

d. The coupling (AC GND DC) switch selects how the input signal is coupled into the oscilloscope:

(1) AC — DC signals are blocked.

(2) GND — the oscilloscope input (*not* the point in the circuit where the oscilloscope probe is connected) is connected to ground.

(3) DC — all signals (DC and AC) are coupled into the oscilloscope.

Demonstration: Push the DC OFFSET button on the Wavetek function generator. Turn the DC OFFSET knob on the function generator and observe the effect on the signal displayed on the oscilloscope screen for AC, GND, and DC coupling.

e. VERT MODE controls:

(1) Channel select buttons (CH 1 and CH 2) selects which input signal will be displayed. When both are pushed in, only one signal — the sum of the two inputs — appears on the screen; if the CH 2 signal is *inverted* (CH 2 POSITION knob pulled out), then the *difference* of the two signals (CH 1 signal *minus* CH 2 signal) is displayed.

(2) ALT CHOP switch selects how two signals are displayed simultaneously:

(a) ALT: The oscilloscope alternates between the two input signals — it has only one electron gun, so it can't display both signals on the same sweep. However, the phase difference between the two signals might not be displayed accurately.

(b) CHOP: The oscilloscope switches between input signals, *i.e.*, it draws a tiny part of one waveform with its electron beam, draws a tiny part of the other, draws a tiny part of the first, etc. The phase difference between the two signals is displayed accurately.

(3) BW LIMIT button: limits the oscilloscope bandwidth to 20 MHz (reduces interference from 90-MHz WSUP-FM).

Experiment with the vertical mode controls.

IV. Function generator familiarization

A. Wavetek Model 190

1. Frequency adjustment

a. Control knob on the left permits coarse frequency adjustments.

b. FREQ MULT (Hz) switch selects the factor by which the value set on the control knob is multiplied to give the frequency.

c. VERNIER/SYM permits fine frequency adjustments; when turned fully CW the output frequency is calibrated to the value set by the control knob on the left.

Experiment with the frequency adjustment controls.

2. MODE switch

   a. CONT — the output signal is produced continuously.

   b. TRIG — 1 cycle of the output waveform is produced when an external trigger signal is applied to the TRIG IN connector, or when the MAN TRIG button is pushed.

   c. GATE — output is produced continuously as long as an external trigger signal is present at the TRIG IN connector, or as long as the MAN TRIG button is depressed.

   Experiment with the MODE switch positions.

3. TRIG LEVEL knob on MODE switch: sets the level of the external trigger signal required when the function generator is in the TRIG or GATE modes.

4. FUNCTION switch

   a. DC — DC OFFSET button must be in; output will be DC, like the output of a power supply, and is adjusted with the DC knob on the FUNCTION switch. (NOTE — DC offset can be supplied with the three output waveforms listed below if the DC OFFSET button is depressed).

   b. Sine wave (∿) — sine wave output is produced.

   c. Triangle wave (⋀⋁) — triangle wave output is produced.

   d. Square wave (⊓⊔) — square wave output is produced.

   Experiment with the effects of the FUNCTION switch.

5. SYM button

   a. Pushing in the button reduces the output frequency to one tenth of the selected value.

   b. VERNIER/SYM knob (on FREQ MULT switch) adjusts the symmetry of the output signal when the SYM button is depressed.

   Experiment with the effect of the SYM button and VERNIER/SYM knob on the output of the function generator.

6. Amplitude controls

   a. OUTPUT ATTEN (dB) buttons — Electrical engineers often express power ratios in decibels, abbreviated dB. The ratio between power levels P_1 and P_2 expressed in dB is defined as

$$\text{ratio (dB)} = 10 \log_{10} \frac{P_2}{P_1}$$

or, since $P = V^2/R$,

$$\text{ratio (dB)} = 10 \log_{10} \frac{V_2^2}{V_1^2} = 20 \log_{10} \frac{V_2}{V_1}$$

Therefore 20 dB corresponds to a voltage ratio of 10, 40 dB to a voltage ratio of 100, 60 dB to 1000, etc. Since ratios in dB are logarithmic, the attenuations produced by the 40 20 10 buttons add (*i.e.*, if both the 10 and the 20 buttons are depressed, the output signal is reduced by 30 dB).

b. AMPLITUDE knob — allows continuous adjustment of the output amplitude.

Experiment with the effects of the amplitude controls.

B. Fluke/Philips PM 5193

1. WAVEFORM buttons select waveforms. At frequencies above 200 kHz, only sine and square waves can be generated.

2. Frequency adjustment

a. To set frequency:

(1) Press FREQUENCY START button. The frequency display (left side) should be blank.

(2) Press FREQUENCY Hz/kHz button until the green light on the display is next to the desired units (Hz or kHz).

(3) Type in the desired frequency (in Hz or kHz) with the keypad on the right side of the front panel, then press the ENTER key.

b. To change the frequency in small steps:

(1) Press the FREQUENCY ΔFREQ button. Frequency display (left side of display) should go blank.

(2) Press the FREQUENCY Hz/kHz key until the green lamp indicates the desired units (Hz or kHz) of the frequency step.

(3) Use the keypad on the right side of the front panel to type in the desired frequency step size. Press the ENTER key when done. The display should now show the desired frequency step size.

(4) To increase or decrease the generator frequency by the selected step size, use the FREQUENCY **+** and **–** buttons. The frequency display (left side of the display) will then shown the *actual* generator frequency instead of the selected step size.

3. Modulation

   a. The OFF button shuts off all modulation. The INT and EXT buttons select either the internal modulation source or an external source. (Input for an external modulating signal is on the *back* panel.)

   b. AM and FM buttons select the type of modulation. If FM modulation is selected, pressing the DEV button allows the peak frequency deviation to be entered with the keypad on the right side of the front panel.

4. Signal LEVEL

   a. Vdc and +/– buttons allow a positive or negative DC level to be added to the output with the keypad on the right side of the front panel.

   b. Vpp, dBm, and Vrms buttons select the units of the signal level that is entered with the keypad on the right side of the front panel and displayed on the right side of the display.

   c. ΔLEVEL button: Pressing this button allows a level step size to be entered with the keypad. The **+** and **–** buttons change the generator output level by the selected step size. The *actual* generator output level (*not* the step size) then appears on the right side of the display.

C. Tektronix Model CFG253

1. Frequency adjustment

   a. FREQUENCY knob on the lower right of the panel adjusts the frequency.

   b. RANGE buttons (top center of panel) select the factor by which the value set on the FREQUENCY knob is multiplied to give the frequency.

   c. If the SYMMETRY VARIABLE button (bottom center of the panel) is depressed, the selected frequency is divided by 10.

   Experiment with the frequency adjustment controls.

2. FUNCTION buttons (upper right corner of the panel)

   a. Sine wave (∿) — sine wave output is produced.

   b. Triangle wave (⋀⋁) — triangle wave output is produced.

   c. Square wave (⊓⊔) — square wave output is produced.

   Experiment with the effects of the FUNCTION buttons.

3. SYMMETRY VARIABLE button (bottom center of the panel)

   a. Pushing in the button reduces the output frequency to one tenth of the selected value.

   b. SYMMETRY knob (upper left of the panel) adjusts the symmetry of the output signal when the SYMMETRY button is depressed.

   Experiment with the effect of the SYMMETRY VARIABLE button and the SYMMETRY knob on the output of the function generator.

4. Amplitude controls

   a. VOLTS OUT button (bottom center of panel) — When pressed, the peak output voltage is limited to 2 V p-p (permits finer adjustment of output level). When this button is out, the peak output signal level is 20 V p-p

   b. AMPLITUDE knob (upper left of the panel) — allows continuous adjustment of the output amplitude.

   Experiment with the effects of the amplitude controls.

If you have difficulty with either the oscilloscope or the function generator, see your instructor. Upperclassmen might also be able to help, but be advised that some of them have acquired bad habits.

USE OF FREQUENCY COUNTER

I. Purpose

This chapter will introduce the student to the use of the Fluke Model 7260A Universal Counter/Timer.

II. Frequency counter description

The Fluke 7260A Counter/Timer has more capability than is required in introductory circuits courses. Most students will use it only to measure the frequency or period of a periodic signal.

III. Frequency counter operation

A. Input control

1. Use channel A.

2. Input protection:

   a. The counter can withstand input signal levels of 250 V rms (> 600 V p-p).

   b. For maximum protection, set ATTEN (attenuator) switch to its highest value (x100). Reduce switch setting only if counter will not trigger.

3. AC DC coupling: AC coupling will usually work best for signals of interest to us.

4. + – switch: determines the input signal slope on which the counting circuits are triggered. Either setting will work for our purposes.

5. FILTER button: set to OUT position.

6. SEP COM switch: set to SEP.

7. TRIGGER LEVEL knob: fully counterclockwise (CCW) to the PRESET position usually works best.

B. Instrument function and accuracy

1. Function slide switch:

   a. FREQ A position: counter measures the frequency f of the signal entering input A of the instrument in either MHz or kHz (see the LED indicator next to the frequency display).

   b. PER A position: counter measures the period $T = 1/f$ of the signal entering input A of the instrument in either seconds, milliseconds, or microseconds (see the LED indicator next to the frequency display).

2. RESOLUTION slide switch: determines the accuracy of the measurement based on the counting time. The accuracy of the least significant digit of the display is given above the slide switch position. The more accurate the setting, the longer the counter takes to update the display (longer time between measurements).

C. Power controls

1. BATTERY button: Out (line power). The counter can operate on its internal battery without being connected to the power line. By selecting line power you save the battery for occasions when line power is unavailable.

2. POWER (green button): push in to energize the counter. The display should light. If it doesn't, check to see if the line cord has been removed from the instrument. If it has, attempt to locate and attach the cord before resorting to battery operation.

D. Familiarization

Connect a signal from the Wavetek function generator to input A of the counter. Experiment with the counter switch settings to get a better understanding of their functions.

USE OF THE NETWORK ANALYZER

I. Purpose

This chapter describes, in excruciating detail, the procedures by which we can measure the frequency response of a two-port network with a network analyzer.

II. Hewlett-Packard 3577A Network Analyzer

A. Network Analyzer Preparation

1. Before turning on the network analyzer, connect the OUTPUT port to the INPUT R port with the two Type N "T" connectors and the Type N barrel connector as shown in Figure 1 below.

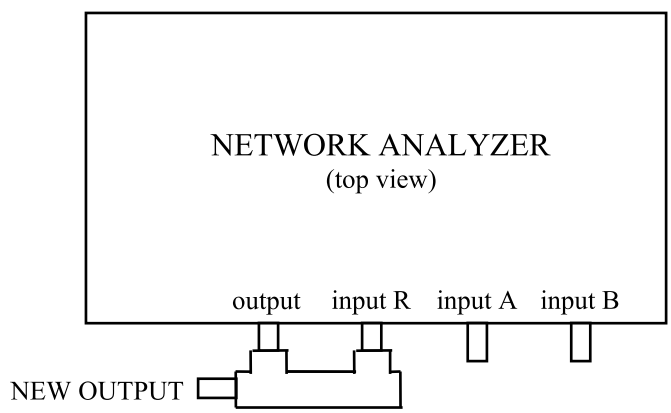

Figure 1. Proper network analyzer configuration for frequency response measurement.

2. Turn on the network analyzer. (The analyzer is plugged into a terminal strip. There is a switch on the end of the terminal strip and a second switch on the analyzer's front panel.)

3. Set the output signal from the analyzer:

   a. Press the SWEEP TYPE button on the analyzer. The sweep menu appears on the right side of the display screen. Select a logarithmic or a linear frequency sweep by pressing the button next to the desired sweep type statement on the menu.

b. Set the range of the frequency sweep as follows:

(1) Push the FREQ button on the analyzer. The frequency menu appears on the screen.

(2) Press the button next to the START FREQ statement on the menu. The START FREQ statement will appear brighter than the other items on the menu.

(3) Enter the desired starting frequency with the keyboard on the analyzer. Frequency units (MHz, kHz, and Hz) will appear on the screen in place of the frequency menu. Select the appropriate units by pressing the button next to the units desired.

(4) The selected start frequency is displayed on the screen and the frequency menu reappears.

(5) Press the button next to the STOP FREQ statement on the menu. The STOP FREQ statement will appear brighter than the other items on the menu.

(6) Select the desired stopping frequency by repeating step II.C.2.c above for the desired stop frequency.

(7) The selected stop frequency is displayed on the screen and the frequency menu reappears.

c. Adjust the analyzer's receiver characteristics as follows:

(1) Press the ATTEN button on the analyzer. The attenuator menu appears on the right side of the screen.

(2) Press the buttons next to the ATTEN R, A, and B statements on the menu until 20 dB is brighter for all three. (This switches 20-dB attenuators into all three network analyzer ports and helps protect the analyzer from excessive signal levels.) If the 20-dB statement(s) is/are already brighter, you don't have to press the button(s).

(3) Set input impedance of network analyzer ports:

(a) Unless you are going to measure the s-parameters of your network (in which case, see the section of this manual on the measurement of s-parameters), always set the input impedance of the R port to 1 MΩ by pressing the the button next to the IMPED R statement until 1 MΩ appears brighter on the screen. (Explanation: Due to the connection illustrated in Figure 1, your circuit sees a source impedance from the network analyzer produced by the network analyzer output port in parallel with port R. The network analyzer output port impedance is always 50 Ω, so if you select the R port impedance to be 50 Ω your circuit will see two 50-Ω impedances in parallel, or 25 Ω, which is probably not what you assumed when you designed your circuit.)

(b) If you want the network analyzer to provide a 50-Ω load impedance for your circuit, press the buttons next to the IMPED A and B statements until 50 Ω appears brighter on the screen.

(c) If you want to prevent the network analyzer from loading the circuit under test, press the buttons next to the IMPED A and B statements until 1 MΩ appears brighter for each item. WARNING: The high input impedances will improperly terminate coaxial test cables, so the 35-pF/foot capacitance of the cable will appear in parallel with the output of your circuit. This will affect the measured frequency response. To minimize this effect, you can use a high impedance oscilloscope probe instead of a coaxial test cable on the output port of your circuit. The high impedance oscilloscope probe attenuates the signal by a factor of 10 (20 dB).

d. Set the amplitude of the test signal from the network analyzer as follows:

(1) Press the AMPTD button on the analyzer. The amplitude menu appears on the right side of the screen. The analyzer output amplitude is displayed on the screen.

(2) Press the button next to the AMPTD statement in the menu. The AMPTD statement will appear brighter than the other menu items.

(3) Enter the desired signal amplitude with the keypad on the analyzer. (NOTE: For maximum accuracy and dynamic range, make the analyzer signal amplitude as large as possible. For passive circuits — i.e., circuits without gain — select approximately 0 dBm. For circuits with gain, select a signal amplitude that will not drive the circuit into nonlinear operation, nor result in a signal from the circuit that will overload the network analyzer. See section II.F.2 below if OVERLOAD ON INPUT R appears on the analyzer screen.) Amplitude units appear on the screen in place of the amplitude menu. Select the units with the button next to the desired units on the screen. The amplitude you selected appears on the screen. The amplitude menu reappears.

4. Configure the analyzer display:

a. Display the magnitude of the measured frequency response on Trace 1.

(1) Press the TRACE 1 button on the analyzer. The indicator lamp above the button will light. Trace 1 will appear brighter on the screen than Trace 2.

(2) Press the DISPLY FCTN button on the analyzer. The function menu will appear on the right side of the screen.

(3) Press the button next to the LOG MAG statement in the menu. The LOG

MAG statement will then appear brighter than the other menu items. Trace 1 will now display the magnitude (in dB) of the measured frequency response.

b. Display the phase of the measured frequency response on Trace 2.

(1) Press the TRACE 2 button on the analyzer. The lamp above the TRACE 1 button goes off and the lamp above the TRACE 2 button lights. Trace 2 now appears brighter on the screen than does Trace 1.

(2) If the function menu is not displayed on the right side of the screen, press the DISPLY FCTN on the analyzer to display the function menu.

(3) Press the button next to the PHASE statement in the menu. The PHASE statement will now appear brighter than the other menu items. Trace 2 will now display the phase of the measured frequency response.

5. Configure the network analyzer to measure voltage gain.

a. To measure voltage at output of circuit referenced to voltage at input of circuit:

(1) Press the INPUT button on the network analyzer. The input menu appears on the right side of the screen.

(2) Press the button next to the A/R (if you are going to connect the output of your circuit to input A of the network analyzer) or the B/R (if you are going to connect the output of your circuit to input B of the network analyzer) statement in the menu. (EXPLANATION: The signal into the R port of the analyzer is the voltage at the input port of the circuit under test. The signal into the A or B port of the network analyzer is the voltage at the output port of the circuit. The ratio A/R or B/R is thus a voltage gain.) The A/R or B/R statement will now appear brighter than the other menu items.

b. To measure the voltage at the output of the circuit referenced to the voltage source inside the network analyzer:

(1) Press the INPUT button on the network analyzer. The input menu appears on the right side of the screen.

(2) Press the button next to the A or B statement in the menu, depending on which port you will connect to the output of your circuit. The A or B statement will now appear brighter than the other menu items. (EXPLANATION: The signal into the B port of the network analyzer is the voltage at the output port of the circuit. The voltage produced by the network analyzer is approximately constant over the selected frequency range. Therefore it is necessary only to divide the measured voltage B by the known network analyzer source voltage to obtain the voltage gain. The network analyzer source voltage is *twice as large* [6 dB higher] as the amplitude selected in section II.C.4 above, because the selected

amplitude is the amplitude across a 50-Ω load impedance. Because of the 2-to-1 voltage divider caused by the 50-Ω source impedance of the network analyzer and the assumed 50-Ω load, the generator amplitude is actually twice as large as the value set in section II.C.4 and displayed on the screen.)

6. Calibrate the network analyzer for voltage gain measurements. NOTE: this calibration is optional. The improved measurement accuracy obtained is usually negligible except at high frequencies.

   a. Connect a coaxial test cable to the new output port shown in Figure 1 above. Connect a second test cable (either a coaxial cable or a high impedance oscilloscope probe *with a ground clip*) to the INPUT B port of the network analyzer.

   b. Connect the cable from the new output port to the cable from the INPUT B port of the analyzer (both the center conductor and the ground must be connected). If the OVERLOAD ON INPUT B message appears on the screen and the red lamp over the INPUT B port is illuminated, the signal amplitude you have selected is too large. Reduce the signal amplitude by following this procedure:

      (1) Press the AMPTD button on the analyzer. The amplitude menu and the signal amplitude will be displayed on the screen.

      (2) Press the button with the ↓ arrow on the analyzer next to the knob repeatedly (this reduces the signal amplitude) until the OVERLOAD ON INPUT B message disappears and the red lamp over the INPUT B port is extinguished. The new signal amplitude is displayed on the screen. This is the largest usable amplitude (which yields the most accurate measurements).

      (3) To insure that the network analyzer has reset from the overload condition, press the button next to the CLEAR TRIP on the network analyzer screen (Amplitude menu).

   c. Press the MEASR CAL button on the network analyzer. The calibration menu appears on the screen.

   d. Trace 1 calibration (the two traces must be calibrated separately).

      (1) Press the TRACE 1 button on the network analyzer.

      (2) Press the button next to the NORMLIZE statement in the menu. Trace 1 is now calibrated.

   e. Trace 2 calibration.

      (1) Press the TRACE 2 button on the analyzer.

      (2) Press the button next to the NORMLIZE statement in the menu. Trace 2

is now calibrated.

f. Calibration check: Both traces should now be straight lines. Trace 1 (magnitude) should be at the top of the graph and the number following the MAG statement at the top of the screen should read approximately 0 dB. Trace 2 should be in the middle of the graph and the number following the PHASE statement (just below the MAG statement) should be approximately zero.

g. Press the INPUT button on the analyzer. The input menu appears on the right side of the screen. The A, B, A/R, or B/R statement is no longer brighter than the rest. Instead, the USER DEF INPUT is brighter. The analyzer is now calibrated to measure voltage gains. The cable losses and the phase shift due to the cable lengths have been taken into account.

B. Measurement of frequency response

1. Connection of the circuit to the network analyzer.

a. NOTE the 25 VDC maximum on all input ports! Do not exceed under *any* circumstances! If your circuit has DC bias potentials on the input and output ports, use coupling capacitors on these ports to block the DC and protect the analyzer.

b. Connect your circuit to the network analyzer as shown in Figure 2 below.

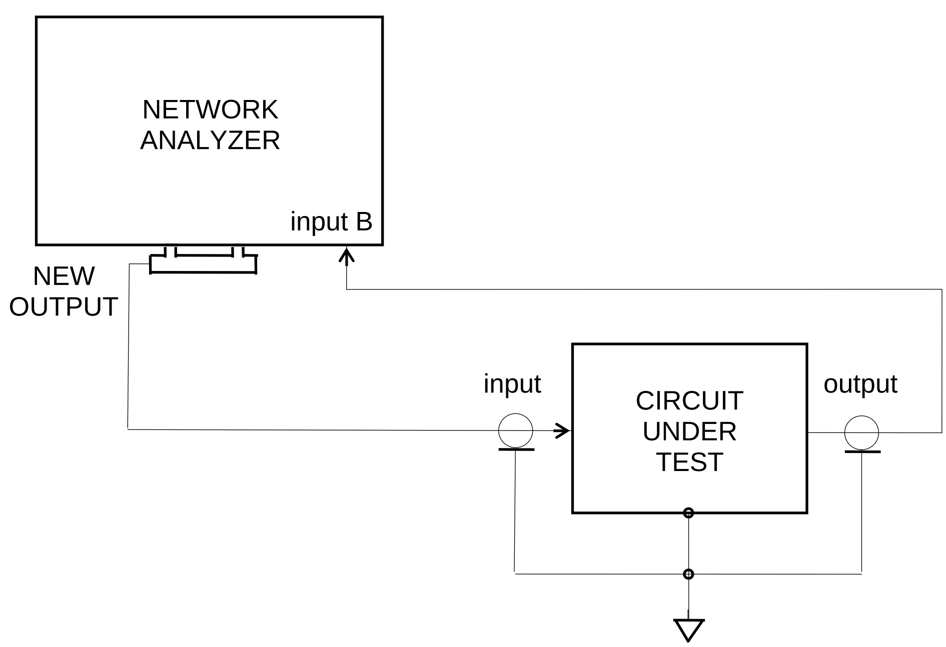

Figure 2. Circuit connection for frequency response measurement.

2. Response measurement.

   a. The measured voltage gain now appears on the screen as a function of frequency.

   b. The marker can be moved to any desired frequency in the measurement range with the knob on the network analyzer, and the magnitude and phase of the frequency response at that frequency can be read from the screen.

C. Plotting results

1. Prepare plotter.

   a. At least one pen must be placed in the plotter's pen carousel. (You can use as many as three pens.) Lift the plastic cover and lift out the carousel to add or remove pens. NOTE: you must provide your own plotter pens.

   b. Turn on the plotter with the switch on the back.

   c. Insert paper.

      (1) Move the lever at the right of the paper tray to the PAPER LOAD position.

      (2) Place the 8.5 x 11-inch paper in the paper tray against the paper guide on the *left* so that the paper is under both (left and right) rollers. The upper edge of the paper should be even with the white line in the upper left corner of the paper tray.

      (3) Move the lever at the right side of the paper tray to the PAPER HOLD position. The rollers will come down onto the paper.

2. To plot results shown on screen:

   a. Press the PLOT button on the network analyzer. The plot menu appears on the network analyzer screen.

   b. Press the button next to the CONFIG PLOT on the screen.

      (1) Select the desired pens for the frequency response traces and the background by pressing the appropriate menu button and entering the plotter carousel pen number with the network analyzer keypad.

      (2) Select the desired plotting speed with the button next to the PN SPEED SLOW FST on the screen. (Slow speed results in a higher-quality plot.)

      (3) Press the button next to RETURN on the network analyzer screen when you are ready to proceed.

c. Press the button next to the description of what you want to plot (usually PLOT ALL) on the network analyzer screen. The plot menu disappears and plotting begins automatically. NOTE: If plotting does not begin, move the lever next to the plotter paper tray to the PAPER LOAD position and back to the PAPER HOLD position.

d. You can stop the plotting at any time by pressing the button next to the ABORT PLOT on the network analyzer display.

D. Network Analyzer shutdown

1. Turn off the network analyzer and the plotter. (For the analyzer in the Electronics Laboratory, this can be done with the switch on the end terminal strip.)

2. Return all test cables to the appropriate wall brackets.

III. Agilent 4395A Network/Spectrum/Impedance Analyzer

A. Network Analyzer Preparation

1. Before turning on the network analyzer, connect the RF OUT port to the R port with the Type N "T" connector and the coaxial cable as shown in Figure 1 below.

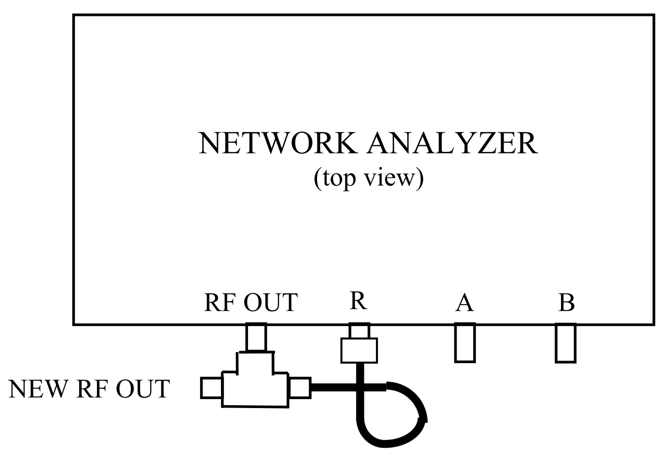

Figure 1. Proper network analyzer configuration for frequency response measurement.

2. Turn on the network analyzer. (The analyzer is plugged into a terminal strip. There is a switch on the end of the terminal strip and a LINE switch in the lower left corner of the analyzer's front panel.)

3. Set the output signal from the analyzer:

   a. Press the **Sweep** button on the analyzer. The sweep menu appears on the right side of the display screen. Select the SWEEP TYPE MENU (bottom button next to the screen) and the Sweep Type Menu will be displayed. Select a logarithmic or a linear frequency sweep by pressing the button next to the desired sweep type statement (either LIN FREQ or LOG FREQ) on the menu.

   b. Set the range of the frequency sweep as follows:

      (1) Push the **Start** button on the analyzer panel. The starting frequency of the sweep appears on the left side of the screen.

      (2) Enter the desired starting frequency with the keyboard on the analyzer, followed by the desired multiplier button (**M** for MHz, **k** for kHz, and ×**1** for Hz).

      (3) The selected start frequency is displayed on the screen.

      (4) Press the **Stop** button on the analyzer panel. The stop frequency of the sweep appears on the left side of the screen.

      (5) Select the desired stopping frequency by repeating step III.A.3.b.(2) above for the desired stop frequency.

      (6) The selected stop frequency is displayed on the screen.

   c. Adjust the analyzer's receiver characteristics as follows:

      (1) Press the **Scale Ref** button on the analyzer panel. A menu appears on the right side of the screen.

      (2) Press the button next to the ATTENUATOR MENU statement on the screen. The attenuator menu is displayed on the right side of the screen.

      (3) Press the buttons next to the ATTEN R, A, and B statements on the menu. The attenuator setting for the selected port is displayed on the left side of the screen. If any port has an attenuator setting less than 20 dB, use the ⇑ button on the analyzer panel to increase the attenuator setting to 20 dB. (This helps protect the analyzer from excessive signal levels.)

      (4) Network analyzer input impedance:

         (a) The network analyzer automatically provides a 50-Ω termination on all three input ports (**R**, **A**, and **B**).

         (b) If a high input impedance is desired (to avoid loading the device under test), ask the instructor to provide an Agilent 41800A active probe from the workshop. Connect the probe's RF connector to the desired input port (**A** or **B**) and the power connector to one of the **PROBE POWER** sockets on the analyzer's front panel. **Do *NOT* insert the tip of the active probe into a protoboard/breadboard hole. The probe tip might break.** (Return the active probe to the instructor when finished making measurements).

d. Set the amplitude of the test signal from the network analyzer as follows:

(1) Press the **Source** button on the analyzer. The source menu appears on the right side of the screen.

(2) Press the button next to the POWER statement on the menu. The analyzer output amplitude is displayed on the screen.

(3) Enter the desired signal amplitude with the keypad on the analyzer, then press the $\times$**1** key. (NOTE: For maximum accuracy and dynamic range, make the analyzer signal amplitude as large as possible. For passive circuits — i.e., circuits without gain — select approximately 0 dBm. For circuits with gain, select a signal amplitude that will not drive the circuit into nonlinear operation, nor result in a signal from the circuit that will overload the network analyzer. See section III.A.6.c below if OVERLOAD ON INPUT R appears on the analyzer screen.)

4. Configure the analyzer display:

a. Display the magnitude and phase responses separately or simultaneously:

(1) Press the **Display** button on the analyzer. The display menu appears on the right side of the screen.

(2) Press the button next to DUAL CHAN on the menu to toggle between **on OFF** (only one channel displayed at a time) and **ON off** (both channels displayed simultaneously).

b. Display the magnitude of the measured frequency response on Channel 1.

(1) Press the **Chan 1** button on the analyzer. The indicator lamp next to the button will light.

(2) Press the **Format** button on the analyzer. The format menu will appear on the right side of the screen.

(3) Press the button next to the LOG MAG statement in the menu. The LOG MAG statement will then be underlined (LOG MAG). Channel 1 will now display the magnitude (in dB) of the measured frequency response.

c. Display the phase of the measured frequency response on Channel 2.

(1) Press the **Chan 2** button on the analyzer. The lamp next the **Chan 1** button goes off and the lamp next to the **Chan 2** button lights.

(2) If the format menu is not displayed on the right side of the screen, press the **Format** button on the analyzer to display the format menu.

(3) Press the button next to the PHASE statement on the menu. The PHASE statement will then be underlined (PHASE). Channel 2 will now display the phase of the measured frequency response.

5. Configure the network analyzer to measure voltage gain.

a. To measure a typical frequency response (voltage at output of circuit referenced to voltage at input of circuit):

(1) Press the **Meas** button on the network analyzer. The measurement menu appears on the right side of the screen.

(2) **NOTE: This step must be performed for each channel separately.** Press the button next to A/R (the output of the circuit connected to input A of the network analyzer) or B/R (the output of the circuit connected to input B of the network analyzer) statement in the menu. (EXPLANATION: The signal into the R port of the analyzer is the voltage at the input port of the circuit under test. The signal into the A or B port of the network analyzer is the voltage at the output port of the circuit. The ratio A/R or B/R is thus a voltage gain.) The A/R or B/R statement will now be underlined.

b. To measure the voltage at the output of the circuit referenced to the voltage source inside the network analyzer:

(1) Press the **Meas** button on the network analyzer. The measurement menu appears on the right side of the screen.

(2) Press the button next to the MORE statement on the menu. Another measurement menu is then displayed on the right side of the screen.

(3) Press the button next to the A or B statement in the menu, depending on which port you will be connected to the output of the circuit. The A or B statement will now be underlined. (EXPLANATION: The signal into the A or B port of the network analyzer is the voltage at the output port of the circuit. The voltage produced by the network analyzer is approximately constant over the selected frequency range. Therefore it is necessary only to divide the measured voltage A or B by the known network analyzer source voltage to obtain the voltage gain.) This step must be performed for each channel separately.

6. Calibrate the network analyzer for voltage gain measurements.

a. **The IF bandwidth of the network analyzer must be set properly or the measured results will be garbage! This step *MUST* be performed!**

(1) Press the **BW / Avg** key on the analyzer panel. The bandwidth and

averaging menu appears on the right side of the screen. The IF BANDWIDTH of the network analyzer is displayed on the left side of the scrren.

(2) If a **linear** frequency sweep is selected, use the ⇓ key on the analyzer panel to reduce the IF BANDWIDTH to a value less than the sweep bandwidth (*e.g.*, if the analyzer is sweeping from 100 to 10 000 Hz, the IF bandwidth must be set to less than 9 900 Hz, the sweep bandwidth). The selected IF BANDWIDTH is probably low enough if the trace on the screen is smooth and without noise.

(3) If a **log** frequency sweep is selected, press the button next to the IF BW statement at the right side of the screen until the line beneath IF BW reads **AUTO man** (not **auto MAN**).

b. Connect a coaxial test cable to the new output port shown in Figure 1 above. Connect a second test cable (either a coaxial cable or a high-impedance active probe) to the **A** or **B** port of the network analyzer. Connect the cable from the new output port to the cable from the **A** or **B** port of the analyzer (both the center conductor and the ground must be connected).

c. If the OVERLOAD ON INPUT A or OVERLOAD ON INPUT B message appears on the screen:

(1) It might be a transient overload. Press any button. If the OVERLOAD message disappears, it was a transient overload. Skip to step III.A.6.d below.

(2) If pressing any button doesn't make the OVERLOAD message disappear, the selected signal amplitude is too large. Reduce the signal amplitude as follows:

(a) Press the **Source** button on the analyzer. The source menu will be displayed on the right side of the screen.

(b) Press the button next to the POWER statement on the right side of the screen. The network-analyzer output power is now displayed on the left side of the screen.

(c) Reduce the output power with the ⇓ arrow on the analyzer (below the knob) repeatedly (this reduces the signal amplitude) or the keypad. The new signal amplitude is displayed on the screen.

d. Press the **Cal** button on the network analyzer. The calibration menu appears on the screen.

e. If N 50Ω does not appear below the CAL KIT statement on the right side of the screen, select the proper calibration kit:

(1) Press the button next to the CAL KIT statement on the screen. The calibration calibration-kit menu appears on the screen.

(2) Press the button next to the N 50Ω statement on the screen. The correct calibration kit is now selected.

(3) Press the button next to the RETURN statement on the screen.

f. Calibration

(1) If both channels are displayed simultaneously, both channels are calibrated simultaneously. If only one channel is displayed at a time, each channel must be calibrated separately.

(2) Press the button next to the CALIBRATE MENU statement on the right side of the screen. The calibrate menu appears on the screen.

(3) Press the button next to the RESPONSE statement on the right side of the screen. The calibrate menu is replaced with the response menu.

(4) Press the button next to the THRU statement on the screen. The statement WAIT -- MEASURING CAL STANDARDS might appear on the right side of the screen if a wide sweep bandwidth has been selected.

(5) When the analyzer has completed its calibration measurements, the THRU statement will be underlined (THRU) and the statement PRESS 'DONE' IF FINISHED WITH CAL appears on the left side of the screen. Press the button next to the DONE statement at the bottom right side of the screen.

(6) If only one channel is displayed on the screen, select the other channel and repeat the above procedure.

g. Calibration check:

(1) Both traces should now be straight lines.

(2) Press the **Marker** button on the analyzer panel. A frequency marker appears on the screen. This marker can be moved to any desired frequency in the measurement range with the knob on the network analyzer, and the magnitude and phase of the frequency response at that frequency can be read from the screen.

(3) When Channel 1 (magnitude) is displayed, the number upper right corner of the magnitude grid should be approximately 0 dB. When Channel 2 (phase) is displayed, the number at the upper right corner of the phase grid should be approximately zero.

(4) There should be a "C" or the word "Cor" at the left side of the screen, indicating that the network analyzer is calibrated.

B. Measurement of frequency response

1. Connection of the circuit to the network analyzer.

a. **NOTE the ±7 V maximum on all input ports!** Do not exceed under ***any*** circumstances! If the circuit being tested has DC bias potentials on the input and output ports, use coupling capacitors on these ports to block the DC and protect the analyzer.

b. Connect the circuit to be tested to the network analyzer as shown in Figure 2 below (use the input — A or B [shown] — for which the analyzer was calibrated).

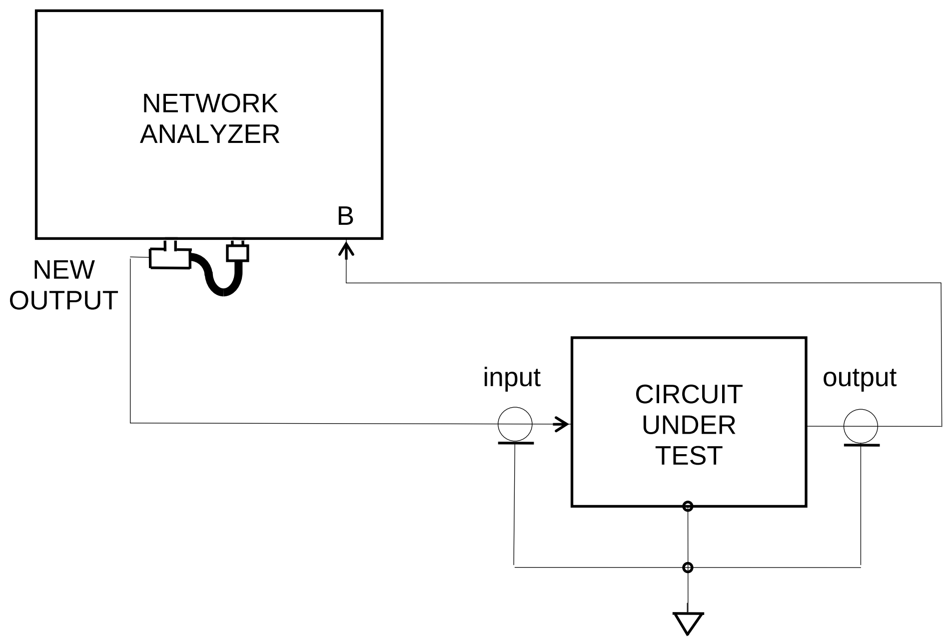

Figure 2. Circuit connection for frequency response measurement.

2. Response measurement.

a. The measured voltage gain now appears on the screen as a function of frequency.

b. Press the **Marker** button on the analyzer panel. A frequency marker appears on the screen. This marker can be moved to any desired frequency in the measurement range with the knob on the network analyzer, and the magnitude and phase of the frequency response at that frequency can be read from the screen.

C. Copying results to a computer file

1. Insert a floppy disk (IBM format) into the disk drive in the network analyzer's front panel.

2. To copy the screen image (including all displayed numbers):

   a. Press the **Save** button on the network analyzer. The save menu appears on the screen.

   b. Press the button next to the GRAPHICS statement on the screen. A new menu appears.

   c. Follow the instructions displayed on the screen to name the file to which the screen image will be saved. Note that the menu buttons can be used to erase letters, etc. When done naming the file, press the button next to the DONE statement. The floppy drive will be activated and the lamp next to the drive will be illuminated while the file is being saved.

   d. The screen image is now saved on the floppy disk as a ***.tif*** file with selected name.

D. Network Analyzer shutdown

1. Turn off the network analyzer.

2. Return all test cables to the appropriate wall brackets.

3. Return the active probe (if used) to the instructor.

IV. Hewlett-Packard 4195A Network Analyzer

A. Network Analyzer Preparation

1. Before turning on the network analyzer, connect the OUTPUT port (either S1 or S2) to the corresponding INPUT port (R1 or R2) with the two Type N "T" connectors and the Type N barrel connector as shown in Figure 1 below.

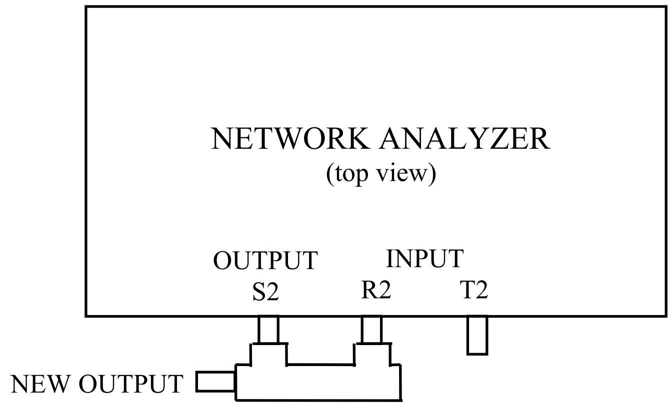

Figure 1. Proper network analyzer configuration for frequency response measurement.

2. Turn on the network analyzer. (The analyzer is plugged into a terminal strip. There is a switch on the end of the terminal strip and a second switch on the analyzer's front panel below the display screen.)

3. Configure the instrument to perform network analysis (and not spectrum or impedance analysis):

   a. Press the CONFIG button on the analyzer panel. The configuration menu appears on the right side of the display screen.

   b. Press the top button next to the screen (next to the NETWORK statement on the menu). The NETWORK statement will be highlighted. The instrument will now function as a network analyzer.

   c. Press the bottom button next to the screen (next to the PORT SELECT statement). The port selection menu appears at the right side of the screen.

   d. Press the button next to T1/R1 or T2/R2 (whichever ports are in use).

   e. The instrument will now function as a network analyzer.

4. Set the output signal from the analyzer:

   a. Press the MENU button on the analyzer. The sweep menu appears on the right side of the display screen.

   b. Select a logarithmic or a linear frequency sweep by pressing the top button next to the screen until **log** or **lin** (whichever is desired) is highlighted.

   c. Set the range of the frequency sweep as follows:

(1) Push the START button on the analyzer. The start frequency is displayed in the lower left corner of the screen.

(2) Enter the desired starting frequency with the keyboard on the analyzer. Select the frequency units (MHz, kHz, and Hz) by pressing the appropriate button to the right of the keypad.

(3) The selected start frequency is displayed in the lower left corner of the screen.

(4) Press the STOP button on the analyzer. The stop frequency is displayed in the lower left corner of the screen..

(6) Select the desired stopping frequency by repeating steps IV.A.4.c.(2)-(3) above for the desired stop frequency.

(7) The selected stop frequency is displayed in the lower left corner of the screen.

d. Adjust the analyzer's input attenuators as follows:

(1) Press the REF ATTEN button above the receiver input (R1 or R2) in use. The input attenuation appears in the lower left corner of the screen.

(2) Use the ⇑ and ⇓ buttons on the analyzer panel to set the input attenuation to 20 dB. (This switches a 20-dB attenuator into the reference port and helps protect the analyzer from excessive signal levels.)

(3) Repeat the two preceding steps with the TEST ATTEN button (above the T1 or T2 port, whichever is in use).

e. Set the amplitude of the test signal from the network analyzer as follows:

(1) Press the AMPLITUDE button on the analyzer (above output connector S1 or S2, whichever is in use). The analyzer's output-signal amplitude appears in the lower left corner of the screen.

(2) Enter the desired signal amplitude with the keypad on the analyzer. (NOTE: For maximum accuracy and dynamic range, make the analyzer signal amplitude as large as possible. For passive circuits — i.e., circuits without gain — select approximately 0 dBm. For circuits with gain, select a signal amplitude that will not drive the circuit into nonlinear operation, nor result in a signal from the circuit that will overload the network analyzer. See section IV.A.5.c below if OVERLOAD ON INPUT R appears on the analyzer screen.) Select the amplitude units with the appropriate button to the right of the keypad.

4. Configure the analyzer display:

a. The network analyzer automatically displays response magnitude on Trace A and response phase on Trace B.

b. Press the DISPLAY button on the analyzer panel. The display menu appears on the right side of the screen.

c. Press the top menu button (next to the **rectan x-A&B** menu statement).

d. Press the menu buttons next to the TRACE A and TRACE B menu statements so that **on** is highlighted.

5. Calibrate the network analyzer.

a. Connect a coaxial test cable to the new output port shown in Figure 1 above.

b. Connect a second test cable to the INPUT (T1 or T2) port of the network analyzer.

(1) If a 50-Ω load impedance is desired, use a 50-Ω coaxial cable (with clip leads or with coaxial connectors at both ends).

(2) If a load impedance greater than 50 Ω is desired, ask the instructor to get an Agilent 41800A active probe from the workshop. Plug the probe's power connector into the PROBE POWER socket on the analyzer's front panel. **Do *NOT* insert the tip of the active probe into a protoboard/breadboard hole. The probe tip might break.**

c. Connect the cable from the new output port to the cable from the INPUT (T1 or T2) port of the analyzer (both the center conductor and the ground must be connected). If the OVERLOAD ON T (1 or 2) INPUT message appears on the screen, the output signal amplitude is too large. Reduce the signal amplitude by following this procedure:

(1) Press the AMPLITUDE button above the S1 or S2 port (whichever is in use). The output-signal amplitude is displayed in the lower left corner of the screen.

(2) Press the button with the ⇓ arrow on the analyzer (below the knob) repeatedly (this reduces the signal amplitude) until the OVERLOAD ON T (1 or 2) INPUT message disappears. The new signal amplitude is displayed on the screen.

d. For logarithmic frequency sweeps, the resolution bandwidth must be adjusted:

(1) Press the RES BW button on the network analyzer. The resolution bandwidth is displayed in the lower left corner of the screen.

(2) Use the ⇓ arrow on the analyzer (below the knob) to reduce the resolution bandwidth until the displayed traces are relatively flat and free of noise. (NOTE: This will lengthen the sweep time.).

e. Press the CAL button on the network analyzer. The calibration menu appears on the right side of the screen.

f. Press the button next to the TRANS CAL statement on the menu. The transmission calibration menu appears on the right side of the screen.

g. Press the button next to the NORMLIZE (THRU) statement on the menu (top button).

h. Calibration check: Both traces should now be straight lines. The number following the T/R statement at the top of the screen should read approximately 0 dB. The number following the θ statement (just below the T/R statement) should be approximately zero degrees.

B. Measurement of frequency response

1. Connection of the circuit to the network analyzer.

a. NOTE the **7 VDC maximum** on all input ports! Do not exceed under *any* circumstances! If the circuit under test has DC bias potentials on the input and output ports, use coupling capacitors on these ports to block the DC and protect the analyzer.

b. Connect your circuit to the network analyzer as shown in Figure 2 below.

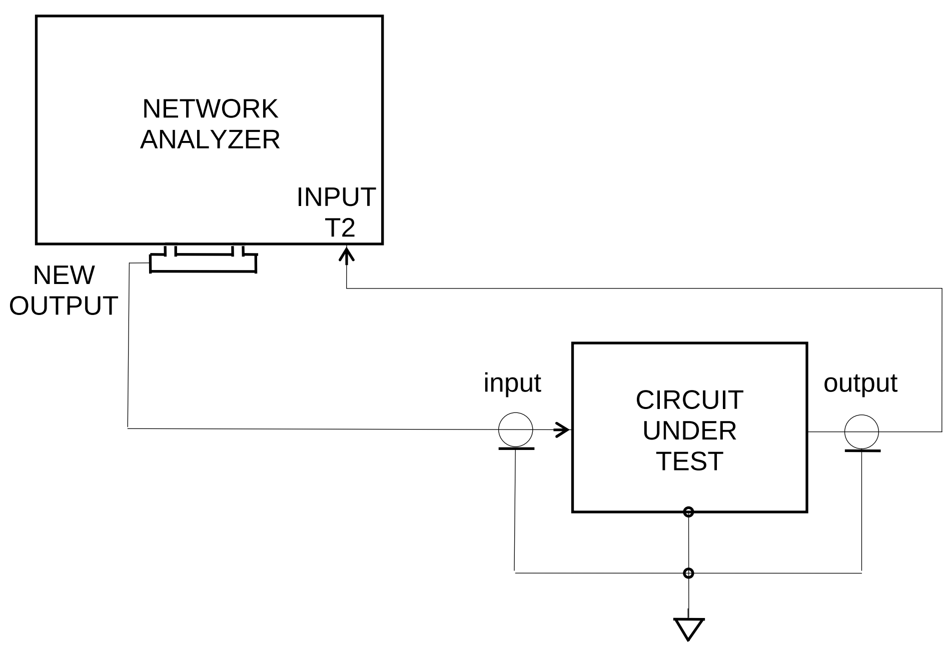

Figure 2. Circuit connection for frequency response measurement.

2. Response measurement.

   a. The measured voltage gain (magnitude on Trace A and phase on Trace B) appears on the screen as a function of frequency.

   b. The marker can be moved to any desired frequency in the measurement range with the knob on the network analyzer, and the magnitude and phase of the frequency response at that frequency can be read from the screen.

C. Plotting results

1. Prepare plotter.

   a. At least one pen must be placed in the plotter's pen carousel. (You can use as many as three pens.) Lift the plastic cover and lift out the carousel to add or remove pens. NOTE: you must provide your own plotter pens.

   b. Turn on the plotter with the switch on the back.

   c. Insert paper.

      (1) Move the lever at the right of the paper tray to the PAPER LOAD position.

      (2) Place the 8.5 x 11-inch paper in the paper tray against the paper guide on the *left* so that the paper is under both (left and right) rollers. The upper edge of the paper should be even with the white line in the upper left corner of the paper tray.

      (3) Move the lever at the right side of the paper tray to the PAPER HOLD position. The rollers will come down onto the paper.

2. To plot results shown on screen:

   a. Press the COPY button on the network analyzer. The copy menu appears on the right side of the network analyzer screen.

   b. Press the button next to the PLOT statement on the menu (bottom button). The plot menu appears on the screen.

   c. Press the button next to the ALL statement on the menu (top button). The network analyzer display will then be plotted.

D. Network Analyzer shutdown

1. Turn off the network analyzer and the plotter.

2. Return all test cables to the appropriate wall brackets.

3. Return the active probe (if used) to the instructor.

USE OF THE SPECTRUM ANALYZER

I. Purpose

This chapter describes how to use the Tektronix 7L5, Agilent E4411B, and Agilent E4403B spectrum analyzers to measure signal spectra.

II. Theory of operation

The operation of the spectrum analyzer can be understood by referring to Figure 1 below. The frequency f_{LO} of a tunable local oscillator is increased linearly at the same rate that the oscilloscope trace sweeps across the screen. This sinusoidal local oscillator signal is used to drive a mixer. The other input to the mixer is the analyzer input signal that has energy at frequency f_{IN} . The mixer output has components at frequencies $f_{LO} - f_{IN}$ and $f_{LO} + f_{IN}$. A bandpass filter with very narrow bandwidth centered on frequency f_{IF} passes only the $f_{LO} - f_{IN}$ signal. Therefore only input signal energy at $f_{IN} = f_{LO} - f_{IF}$ reaches the detector and drives the vertical deflection of the oscilloscope. The user of the spectrum analyzer can select the vertical deflection to be proportional either to the signal voltage (linear display) or to the logarithm of the signal voltage (dB display). The resolution of the spectrum analyzer (the ability to separate two signals that are closely spaced in frequency) can be increased by decreasing the bandwidth of the IF and video filters.

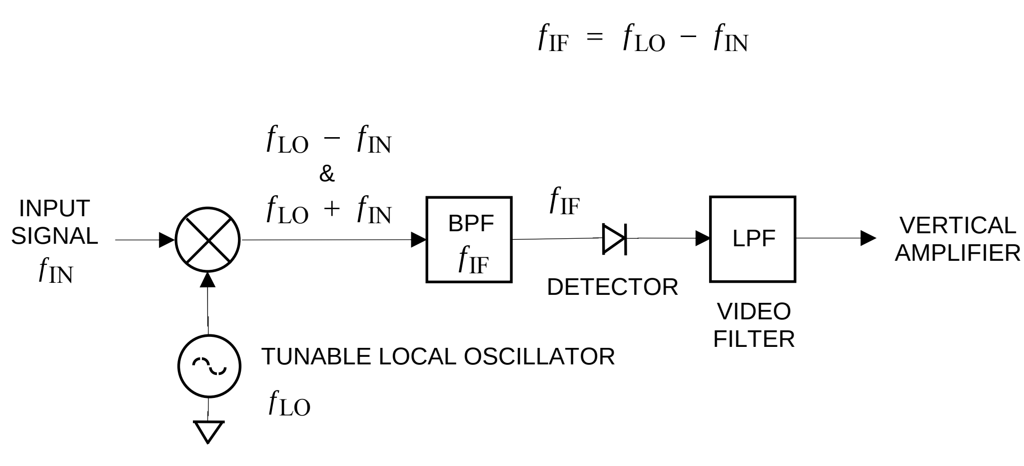

Figure 1. Spectrum analyzer block diagram.

III. Tektronix 7L5

A. Equipment description

The Tektronix 7L5 spectrum analyzer is a plug-in module for the Tektronix 7600-series oscilloscope mainframes.

B. Operation

1. Mainframe configuration.

a. Power on: Pull out the POWER switch on the oscilloscope mainframe to the right of the display screen. The green power indicator lamp above the POWER switch should light. If it does not, check the line cord and the workbench power switches.

b. VERT MODE switches: select either ADD or RIGHT.

c. INTENSITY (concentric with READOUT) and GRAT ILLUM (graticule illumination) knobs: adjust for desired levels.

d. READOUT knob (small, concentric with INTENSITY knob): adjusts the intensity of spectrum analyzer settings that are displayed on the screen; set to desired level.

2. 7L5 control functions.

a. DOT FREQUENCY knob: adjusts the frequency at which the bright dot marker appears on the display screen. The selected frequency for the dot marker is displayed in the upper right corner of the screen. The knob adjusts the dot frequency in 10-kHz steps unless the FINE TUNING button adjacent to the knob is pressed (button lights red when activated), in which case the adjustment steps are 250 Hz.

b. DOT MKR (dot marker) knob (small knob concentric with the large DOT FREQUENCY knob): positions the bright marker dot anywhere on the left half of the display screen.

c. DIGITAL STORAGE buttons:

(1) A: press for normal operation.

(2) B: same results as button A.

(3) SAVE A: Stores trace A.

(4) B – SAVE A: Displays the difference between the signal on trace B and the signal stored when SAVE A was pressed.

(5) MAX HOLD: As the trace sweeps across the screen, the display changes only at the frequencies where the signal amplitude is larger than that already displayed.

d. BASELINE CLIPPER knob: chops off noise at the bottom of the display screen.

e. RESOLUTION knob (large knob concentric with small FREQUENCY

SPAN/DIV knob): adjusts bandwidth of the spectrum analyzer's IF filter. Narrower bandwidth produces finer resolution. COUPLED position: guarantees calibrated measurement for selected FREQUENCY SPAN/DIV and TIME/DIV.

f. FREQUENCY SPAN/DIV (frequency span/division) knob (small knob concentric with large RESOLUTION knob): adjusts the frequency range represented by each horizontal division on the display screen. The selected value is displayed in the lower right corner of the screen. NOTE: the frequency axis of the spectrum analyzer display is always linear, unlike the logarithmic frequency display of which the network analyzer is capable, or the frequency axis of a Bode plot.

g. REFERENCE LEVEL knob (large knob concentric with small VAR knob): adjusts what signal level will reach the top of the display. The value selected is displayed at the top center of the screen. The units of the vertical axis are set with the INPUT switches (see IV.B.14 below). When the REFERENCE LEVEL knob is pulled out, turning it produces 10-dB steps; when pushed in, turning it gives 1-dB steps.

h. VAR (variable reference level) knob (small knob concentric with large REFERENCE LEVEL knob): allows adjustments of less than 1 dB in the reference level, but the display is uncalibrated ("<" symbol is displayed at the top center of the screen) unless the knob is turned counterclockwise until it clicks ("<" symbol vanishes).

i. UNCAL lamp: lights if selected sweep rate (TIME/DIV) is too fast for the selected resolution (">" symbol appears at top center of display screen).

j. LOG/LIN buttons: set the vertical display to 10 dB/div, 2 dB/div, or mV/div. Normally, 10 or 2 dB/div is selected.

k. TRIGGERING controls:

(1) The button pressed will light red.

(a) FREE RUN button: the horizontal sweep is triggered as soon as the previous sweep ends. This gives the fastest possible repetition rate.

(b) INT (internal) button: horizontal sweep is triggered when the input signal reaches a level selected with the LEVEL/SLOPE knob.

(c) LINE button: horizontal sweep is triggered by the power line signal when that signal reaches a level selected with the LEVEL/SLOPE knob.

(2) MODE buttons

(a) NORM (normal): usual mode of operation.

(b) MAN SWP (manual sweep): horizontal sweep is accomplished by manually turning the MAN SWP knob (same as the LEVEL/SLOPE knob, performing a different function when this button is pressed).

(c) SGL SWP/READY (single sweep/ready): produces a single sweep each time button is pressed.

l. TIME/DIV (time/division) knob: sets the sweep speed per horizontal division. If set too high, the UNCAL lamp lights and the “>” symbol appears at the top center of the display screen, in front of the displayed reference level.

m. CALIBRATOR BNC connector: source of a –40 dBV, 500-kHz signal used to calibrate the spectrum analyzer display.

n. INPUT switches:

(1) TERMN Z (terminal impedance): sets the input impedance of the INPUT BNC connector port. 50 Ω, 1 MΩ + 28 pF, or 600 Ω can be selected. In normal operation, either the 50-Ω or the 1-MΩ position is used. WARNING: with the switch in the 1-MΩ position, a standard BNC cable will contribute approximately 35 pF/ft in parallel with the analyzer’s input impedance.

(2) REF (reference level): selects units for vertical axis.

(a) dBm 50 Ω: dB referenced to 1 mW. However, the spectrum analyzer measures voltage, not power, so the analyzer assumes that the voltage is measured across a 50-Ω resistance to convert the measured voltage to power.

(b) dBV: dB referenced to 1 V rms, independent of actual circuit or input resistance level.

(c) dBm 600 Ω: dB referenced to 1 mW, but the analyzer assumes the voltage was measured across a 600-Ω resistance to do the voltage-to-power conversion.

o. INPUT BNC connector: the input signal is connected here. *Note the following limits on input signal amplitude*:

(1) 50-Ω reference level switch selected: 3.5 Vdc or Vrms (+24 dBm) *maximum*.

(2) 1 MΩ/28 pF reference level switch selected:

(a) For ac or pulses with risetimes ≤ 2 V/µs — 15 V p-p *maximum*.

(b) For ac or pulses with risetimes > 2 V/µs — 40 V dc + ac *maximum*.

(3) 600-Ω reference level switch selected: 12 Vdc rms (+24 dBm) *maximum*.

NEVER exceed these limits under any circumstances! Be especially careful of dc inputs. Whenever possible, use a blocking capacitor to strip the dc from input signals.

p. VERT POSITION (vertical position) knob: adjusts the vertical position of the trace on the display screen. Used when calibrating the spectrum analyzer.

q. HORZ POSITION (horizontal position) knob: adjusts the horizontal position of the trace on the display screen. Used when calibrating the spectrum analyzer.

C. Spectrum analyzer calibration

The spectrum analyzer should be calibrated before accurate measurements are attempted.

1. Connect a coaxial cable with BNC connectors on both ends between the CAL BNC connector and the INPUT BNC connector on the spectrum analyzer.

2. Switch and control positions for calibration.

a. INPUT switches.

(1) TERMZ: set to 50 Ω.

(2) REF: set to dBV.

b. Adjust REFERENCE LEVEL knob for –40 dBV (displayed at top center of display screen).

c. LOG/LIN buttons: press 10 dB/div.

d. Adjust DOT FREQUENCY knob to 500 kHz (displayed in upper right corner of the screen).

e. Use DOT MKR to put the bright marker knob in the center of the screen.

f. Set TIME/DIV knob to the AUTO position.

g. SOURCE buttons: press FREE RUN.

h. MODE buttons: press NORM.

i. Set RESOLUTION knob to COUPLED position.

j. Set FREQUENCY SPAN/DIV knob to 1 kHz (displayed in lower right corner of display).

3. Calibration procedure

a. Adjust the VERT POSITION knob so that the peak of the displayed signal is at the top grid line.

b. Adjust the HORZ POSITION knob so that the peak of the displayed signal is in the center of the display (at 500 kHz).

D. Familiarization with spectrum analyzer

1. With no input signal to the spectrum analyzer, set the DOT FREQUENCY to 0 Hz and position the DOT MKR near the center of the spectrum analyzer display screen. Note the large "dc" signal displayed at 0 Hz. Actually, *there is no dc signal present*. The large 0-Hz signal merely indicates the position of 0 Hz. (The minimum input frequency the spectrum analyzer can handle is 5 Hz.) *Avoid applying dc to the INPUT BNC connector!*

2. Connect a function generator output to the spectrum analyzer input with a coaxial cable with BNC connectors at both ends.

3. Adjust the function generator to produce a square wave or a triangular wave.

4. Check the function of each spectrum analyzer control on the displayed spectrum. Use the dot marker to measure the frequency of each harmonic of the function generator output. Measure the amplitude of each harmonic.

IV. Agilent E4411B 9-kHz to 1.5-GHz Spectrum Analyzer and Agilent E4403B 9-kHz to 3-GHz Spectrum Analyzer

A. Operation

1. Power:

a. To turn on the spectrum analyzer, press *On* button (at the lower left-hand corner of the display screen). The display screen should light up. If it does not, check the line cord and the workbench power switches.

b. To turn off the spectrum analyzer, press the *Standby* button (at the lower left-hand corner of the display screen).

2. Frequency range

a. Specified range method

(1) Press FREQUENCY Channel button on the analyzer's front panel. The *Freq*/*Channel* menu appears on the ride side of the display screen.

(2) Press the button next to *Start Freq* (on the menu on the screen). The frequency at the start of the sweep (*i.e.*, the left edge of the screen) is displayed on the screen. Enter the desired start frequency with the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen.

(3) Press the button next to *Stop Freq* (on the menu on the screen). The frequency at the end of the sweep (*i.e.*, the right edge of the screen) is displayed on the screen. Enter the desired start frequency with the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen.

b. Specified center-frequency method

(1) Press FREQUENCY Channel button on the analyzer's front panel. The *Freq*/*Channel* menu appears on the ride side of the display screen.

(a) Press the button next to *Center Freq* (on the menu on the screen). The frequency at the middle of the screen is displayed.

(b) Enter the desired center frequency with the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen.

(2) Press the SPAN X Scale button on the analyzer's front panel. The span menu appears at the right edge of the screen.

(a) Press the button next to *Span* (on the menu on the screen). The frequency span (*i.e.*, the frequency difference between the right and left edges of the screen) is displayed on the screen.

(b) Enter the desired frequency:

(A) With the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen.

(B) With the down (↓) and/or up (↑) arrow buttons on the front panel of the analyzer.

3. Signal amplitude

a. Press the AMPLITUDE Y Scale button on the front panel of the analyzer. The *Amplitude* menu appears on the right edge of the screen.

b. Reference level

(1) Press the button next to *Ref Level*. The reference level (the amplitude of signal at the ***top*** of the display grid) appears on the screen.

(2) Enter the desired reference level:

(a) With the keypad. Select units (dBm, mV, etc.) with the button at the right side of the screen.

(b) With the down (↓) and/or up (↑) arrow buttons on the front panel of the analyzer.

c. Vertical scale

(1) Press the button next to *Scale* / *Div*. The vertical scale is displayed on the screen.

(2) Enter the desired vertical scale:

(a) With the keypad. Select units (dB is the *only* choice) with the button at the right side of the screen.

(b) With the down (↓) and/or up (↑) arrow buttons on the front panel of the analyzer.

d. Log or linear scale:

Press the button next to *Scale Type* (on the menu on the screen) to toggle between a logarithmic or linear vertical scale. (The selected type is underlined.) NOTE: In *almost all cases*, spectra are displayed on a *log* scale.

4. Bandwidth adjustment (refer to section II of this chapter).

a. IF bandwidth:

(1) Press the BW / *Avg* button on the front panel of the spectrum analyzer. The *BW* / *Avg* menu appears on the right side of the screen.

(2) Press the button next to *Res BW* (on the right side of the screen) to toggle between *Auto* (selected automatically by the spectrum analyzer) and *Man* (set by the user). The IF bandwidth (or *resolution* bandwidth — abbreviated *Res* bandwidth) is displayed on the screen.

(3) To set the IF bandwidth manually:

(a) Use the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen. NOTE: Only certain IF bandwidths are available; the spectrum analyzer will select the available bandwidth closest to the one chosen by the user.

(b) Use the down (↓) and/or up (↑) arrow buttons on the front panel of the analyzer.

b. Video bandwidth:

(1) Press the BW / *Avg* button on the front panel of the spectrum analyzer. The *BW / Avg* menu appears on the right side of the screen.

(2) Press the button next to *Video BW* (on the right side of the screen) to toggle between *Auto* (selected automatically by the spectrum analyzer) and *Man* (set by the user). The video bandwidth is displayed on the screen.

(3) To set the video bandwidth manually:

(a) Use the keypad. Select units (Hz, MHz, etc.) with the buttons at the right side of the screen. NOTE: Only certain video bandwidths are available and depend on the IF bandwidth; the spectrum analyzer will select the available bandwidth closest to the one chosen by the user.

(b) Use the down (↓) and/or up (↑) arrow buttons on the front panel of the analyzer.

5. Sweep averaging

Averaging two or more signal sweeps reduces the noise level and helps to stabilize fluctuating signal levels. NOTE: Increasing the number of sweeps being averaged increases the time needed for the spectrum analyzer to update the display.

a. Press the BW / *Avg* button on the front panel of the spectrum analyzer. The *BW / Avg* menu appears on the right side of the screen.

b. Press the button next to *Average* (on the right side of the screen) to turn sweep averaging on and off. The number of consecutive sweeps being averaged together is displayed on the screen.

c. To select the number of sweeps being averaged:

(1) Use the keypad to enter the number of sweeps.

(2) Press the button next to *Enter* (on the right side of the display screen).

6. Markers

a. Displaying a marker

(1) Press the *Marker* → button (on the front panel of the spectrum analyzer in the MARKER section). The *Marker* → menu appears on the right side of the screen and a diamond-shaped marker appears at the center frequency of the display.

(2) Adjust the marker frequency:

(a) Keypad: Enter desired marker frequency. Select units from the menu at the right side of the screen. The marker moves to the selected frequency. The marker frequency and the signal amplitude at that frequency appear on the grid and in the upper right corner of the display.

(b) Control knob: Rotating the control knob (on the front panel of the spectrum analyzer) moves the marker on the screen. The frequency of the marker and the signal amplitude at that frequency appear on the grid and in the upper right corner of the display.

(3) Moving marker to a signal peak:

(a) Press the *Peak Search* button (on the front panel of the spectrum analyzer in the MARKER section). The *Peak Search* menu appears at the right side of the screen and the marker moves to the peak of the largest signal on the screen. The marker frequency and signal amplitude are displayed in the grid area of the screen and in the upper right corner of the screen.

(b) Press the *Marker →* button again. Press the button next to the *Mkr → CF* in the *Marker →* menu. The center frequency of the display will change to the marker frequency; the marker (and the signal it is marking) will be in the center of the screen.

b. Displaying additional markers

(1) Press the *Marker* button (on the front panel of the spectrum analyzer in the MARKER section). The *Marker* menu appears at the right side of the display.

(2) Press the button next to *Select Marker* in the *Marker* menu until the desired marker number is underlined.

(3) Press the *Peak Search* button to display the *Peak Search* menu on the right side of the screen.

(4) Press the button next to *Next Peak* in the *Peak Search* menu. The selected marker number appears on the next highest signal peak on the screen. The frequency of the marker and the associated signal amplitude are displayed in the grid area and in the upper right corner of the screen.

c. Deleting markers

(1) To turn off only certain markers:

(a) Press the *Marker* button on the front panel of the spectrum analyzer to display the *Marker* menu on the screen.

(b) Press the button next to *Select Marker* until the marker to be turned off is underlined.

(c) Press the button next to *Off* in the *Marker* menu to turn off the selected marker.

(2) To turn off all markers:

(a) Press the *Marker* button on the front panel of the spectrum analyzer to display the *Marker* menu on the screen.

(b) If there is only one marker to delete, press the button next to *Off* on the *Marker* menu.

(c) If there is more than one marker to delete, press the button next to *More* on the *Marker* menu until *2 of 2* is displayed in the lower right corner of the screen. Press the button next to *Marker All Off* to delete all markers.

B. Screen shots

1. Insert an *MS-DOS formatted* (Mac, Unix, and Linux won't work) 1.44-MB floppy disk (sorry, but the design of the Agilent E4411B and E4403B analyzers predates USB) in the disk drive on the right side of the spectrum analyzer's front panel.

2. Press the *File* button on the front panel of the spectrum analyzer. The *File* menu appears at the right edge of the screen.

3. Press the button next to *Save* on the *File* menu. The *File* menu is replaced on the screen by the *Save* menu.

4. Press the button next to *Format*. The *Save* menu is replaced by the *Format* menu.

5. Press the button next to *Bitmap*. The *Format* menu is replaced by the *Save* menu. Under the word *Format* in the *Save* menu will be the word *Bitmap*.

6. Press the button next to *Save Now*. The screen shot will be saved as a **.**GIF file with the name SCREN*xyz*.GIF, where *xyz* is a 3-digit number.

MEASUREMENT OF S-PARAMETERS

I. Description of *s*-parameters

You never know when you might be mugged, assaulted, or otherwise accosted by an *s*-parameter, so you should be able to recognize an *s*-parameter when you see one, the better to describe your assailant to the police.

The *s*- in *s*-parameter stands for scattering. Unlike other two-port network parameters, which relate the voltages and currents at the network ports, *s*-parameters relate voltage *waves* incident on and reflected from the ports. The concept of incident and reflected waves is shown in Figure 1 below.

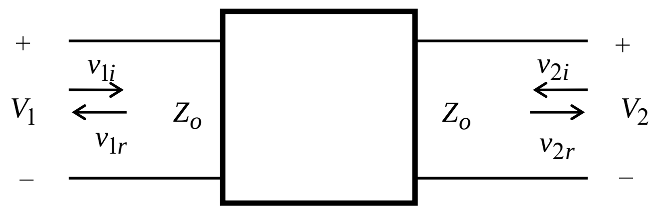

Figure 1. Voltage waves incident on and reflected from a two-port network.

In Figure 1, v_{1i} is the voltage wave incident on (into) port 1, v_{1r} is the voltage wave reflected from (out of) port 1, v_{2i} is the voltage wave incident on (into) port 2, and v_{2r} is the voltage wave reflected from (out of) port 2. These voltage waves are related to each other by the *s*-parameters as follows:

$$v_{1r} = s_{11}v_{1i} + s_{12}v_{2i}$$

$$v_{2r} = s_{21}v_{1i} + s_{22}v_{2i} \quad .$$

The voltage waves are considered to be "scattered" by the network, hence the name scattering parameters. From the above equations, we can define the *s*-parameters in terms of the incident and reflected voltage waves as:

$$s_{11} = \left.\frac{v_{1r}}{v_{1i}}\right|_{v_{2i} = 0} = \text{input port reflection coefficient}$$

$$s_{12} = \left.\frac{v_{1r}}{v_{2i}}\right|_{v_{1i} = 0} = \text{reverse voltage gain}$$

$$s_{21} = \left.\frac{v_{2r}}{v_{1i}}\right|_{v_{2i} = 0} = \text{forward voltage gain}$$

$$s_{22} = \left.\frac{v_{2r}}{v_{2i}}\right|_{v_{1i} = 0} = \text{output port reflection coefficient}$$

To create the $v_{ni} = 0$ conditions given in the above equations, we must terminate both ports of the network in R_o, the characteristic impedance of our measuring system. This eliminates further reflections of our voltage waves; unwanted reflections would degrade the accuracy of the measurements.

From the above equations, we see that the scattering parameter s_{mn} is the voltage wave out of port m due to the voltage wave into port n when both ports are terminated in R_o.

II. *S*-parameter measurement

We will now describe, in sickening detail, the procedures by which we can measure the *s*-parameters of a two-port network with an Agilent 4395A network analyzer.

A. Network Analyzer Preparation:

1. Before turning on the network analyzer, connect the 50-Ω Signal Divider from the HP35676A Reflection-Transmission Test Kit (available in the laboratory next to the network analyzer or from your instructor) to the network analyzer. Connect the *R* port on the signal divider to the *R* port on the network analyzer and the *A* port on the signal divider to the *A* port on the analyzer with the two short cables in the test kit. (The cables have type N male-to-male connectors on both ends.) Connect the RF OUTPUT port on the analyzer to the RF INPUT port on the signal divider with the long cable in the test kit. This signal divider separates the forward and reflected voltage waves and directs them to the proper port of the network analyzer. The incident wave goes into the *R* (Reference) port and the reflected wave goes in the *A* port.

2. Turn on the network analyzer with the pushbutton LINE switch on the analyzer's front panel.

3. Set the output signal from the analyzer by following this procedure:

   a. Sweep type:

      (1) Press the SWEEP button on the analyzer. The sweep menu appears on the right side of the display screen.

      (2) Press the button next to SWEEP TYPE MENU at the bottom right of the display.

      (3) Select a linear frequency sweep by pressing the button next to the LIN

FREQ statement on the menu.

b. Set the range of the frequency sweep as follows:

(1) Push the START button on the analyzer. The start frequency appears on the screen.

(2) Use the keypad on the analyzer to enter the desired starting frequency and units (G = GHz, M = MHz, k = kHz, and x1 = Hz). The selected start frequency is displayed on the screen.

(3) Press the STOP button on the analyzer. The stop frequency appears on the screen.

(4) Use the keypad on the analyzer to enter the desired stopping frequency and units (G = GHz, M = MHz, k = kHz, and x1 = Hz). The selected stop frequency is displayed on the screen.

c. Set the amplitude of the test signal from the network analyzer as follows:

(1) Press the SOURCE button on the analyzer. The source power amplitude is displayed on the screen.

(2) Set the source power with one of these methods:

(a) Use the keypad to enter the desired source power in dBm, and then press the x1 button. (The source power in mV is displayed beneath the source power in dBm.)

(b) Use the up (↑) and down (↓) arrows on the network analyzer to adjust the source power in 1-dB steps.

For maximum accuracy and dynamic range, make the analyzer signal amplitude as large as possible. (NOTE from the "INS. LOSS 10 dB NOM." statement on the 50-Ω signal divider that the signal divider attenuates the power by 10 dB.) For passive circuits — *i.e.*, circuits without gain — select approximately +5 dBm. For circuits with gain, select an amplitude that will not drive the circuit into nonlinear operation, nor result in a signal from the circuit that will overload the network analyzer.

B. Configure the analyzer display.

1. For a rectangular display:

a. Create a split display:

(1) Press the *Display* button in the MEASUREMENT section on the analyzer. The display menu appears on the right side of the screen.

(2) Press the button next to DUAL CHAN (at the top of the display menu on the screen) so that ON is capitalized and *off* is lower case. The screen should now show two graphs, one above the other.

b. Display the magnitude of the measured s-parameter on Channel 1.

(1) Press the *Chan* 1 button in the ACTIVE CHANNEL section on the analyzer. The indicator lamp to the left of the button will light. NOTE the CH1 designation at the top left-hand corner of the upper graph on the screen.

(3) Select a log or linear scale:

(a) For a logarithmic (*i.e.*, dB) scale, press the button next to LOG MAG (at the top of the menu on the screen) LOG MAG will be underlined.

(b) For a linear scale, press the button next to MORE (near the bottom of the on-screen menu). The *Format* menu changes. Press the button next to LIN MAG (at the top of the menu). LIN MAG will be underlined.

c. Display the phase of the measured s-parameter on Channel 2.

(1) Press the *Chan* 2 button in the ACTIVE CHANNEL section on the analyzer. The indicator lamp to the right of the button will light. Note the CH2 designation at the top left-hand corner of the lower graph on the screen.

(2) Press the *Format* button in the MEASUREMENT section on the analyzer. The format menu will appear on the right side of the screen.

(3) Press the button next to PHASE on the on-screen menu (second from the top). PHASE will be underlined.

2. For a polar display (the network analyzer will display the magnitude and phase angle of the measured *s*-parameters above the polar display):

a. If *two* graphs are shown on the screen, turn off the dual-channel display.

(1) Press the *Display* button in the MEASUREMENT section of the network analyzer. The display menu appears on the right side of the screen.

(2) Press the button next to DUAL CHAN (top of the on-screen menu) so that *on* is in lower case and OFF is capitalized.

b. Select Channel 1 by pressing the *Chan* 1 button in the ACTIVE CHANNEL section of the analyzer.

c. Press the *Format* button in the MEASUREMENT section of the analyzer. The format menu will appear on the right side of the screen.

d. Press the button next to the POLAR CHART statement in the menu. CHART will be underlined. A polar plot appears on the network analyzer display.

3. For a Smith Chart display (the network analyzer will display the impedance corresponding to a measured reflection coefficient [s_{11} or s_{22}]):

a. If *two* graphs are shown on the screen, turn off the dual-channel display.

(1) Press the *Display* button in the MEASUREMENT section of the network analyzer. The display menu appears on the right side of the screen.

(2) Press the button next to DUAL CHAN (top of the on-screen menu) so that *on* is in lower case and OFF is capitalized.

b. Select Channel 1 by pressing the *Chan* 1 button in the ACTIVE CHANNEL section of the analyzer.

c. Press the *Format* button in the MEASUREMENT section of the analyzer. The format menu will appear on the right side of the screen.

d. Press the button next to the SMITH CHART statement in the menu. CHART will be underlined. A Smith chart appears on the network analyzer display.

C. Measurement of s_{11} and s_{22}.

1. Configure the network analyzer to measure a voltage reflection coefficient.

a. Rectangular display:

(1) Press the *Chan* 1 button in the ACTIVE CHANNEL section of the analyzer panel. The lamp to the left of the button should light.

(2) Press the *Meas* button in the MEASUREMENT section of the network analyzer panel. The measurement menu appears on the right side of the screen.

(3) Press the button next to S PARAMETERS (near the bottom of the on-screen menu). The measurement menu is replaced by the *s*-parameter menu on the screen.

(4) Press the button next to the S11 [A/R] statement in the *s*-parameter menu. S11 [A/R] will be underlined.

(5) Press the *Chan* 2 button in the ACTIVE CHANNEL section of the analyzer panel. The lamp to the right of the button should light.

(6) Repeat steps II.C.1.a.(1)(3) and (4).

b. Polar or Smith Chart display:

(1) Press the *Meas* button in the MEASUREMENT section of the network analyzer panel. The *s*-parameter menu appears on the right side of the screen.

(2) Press the button next to the S11 [A/R] statement in the *s*-parameter menu. S11 [A/R] will be underlined.

The network analyzer is now configured to measure s_{11} and s_{22}.

2. Calibration. The network analyzer must be calibrated for the type of measurement to be performed.

a. Connect a **50-Ω** test cable to the TEST 50 Ω port on the signal divider. Use a cable that will work well over the selected frequency range.

b. Select the calibration kit:

(1) Press the *Cal* button in the MEASUREMENT section on the analyzer panel. Several options are listed on the right side of the screen.

(2) Press the button next to CAL KIT. The list of options is replaced by a list of possible calibration kits.

(3) If the 50-Ω test cable has clip leads on one end, press the button next to USER KIT. USER KIT will be underlined.

(4) If the 50-Ω test cable has coaxial connectors at both ends, press the button next to the connector (*e.g.*, N 50Ω) that will attached to the device under test. The selected calibration kit will be underlined.

c. Rectangular display

(1) Each channel must be calibrated separately. Calibrate Channel 1 first. Press the *Chan* 1 button on the network analyzer.

(2) Press the *Cal* button in the MEASUREMENT section of the analyzer panel. A list of options appears at the right side of the screen.

(3) Press the button next to CALIBRATE MENU. The list of options at the right side of the screen is replaced by the calibration menu.

(4) Press the button next to S11 1-PORT. The calibration menu on the screen is replaced by a list of calibration standards.

(5) One by one, place the appropriate calibration standard (open, short, or load) at the end of the 50-Ω test cable and press the button next to that standard. (If the has clip leads on one end, the LOAD standard can be a 51-Ω resistor, OPEN means leave the clip leads unconnected, and

SHORT means connect the clip leads together.) When all standards have been done, press the button next to DONE: 1-PORT CAL. (Instructions to do these steps appear on the screen.)

(6) Calibrate Channel 2. Press the *Chan* 2 button on the network analyzer panel, then repeat steps II.C.2.c.(2)-(5).

d. Polar or Smith Chart display

(1) Press the *Cal* button in the MEASUREMENT section of the analyzer panel. A list of options appears at the right side of the screen.

(2) Press the button next to CALIBRATE MENU. The list of options at the right side of the screen is replaced by the calibration menu.

(3) Press the button next to S11 1-PORT. The calibration menu on the screen is replaced by a list of calibration standards.

(4) One by one, place the appropriate calibration standard (open, short, or load) at the end of the 50-Ω test cable and press the button next to that standard. (If the has clip leads on one end, the LOAD standard can be a 51-Ω resistor, OPEN means leave the clip leads unconnected, and SHORT means connect the clip leads together.) When all standards have been done, press the button next to DONE: 1-PORT CAL. (Instructions to do these steps appear on the screen.)

e. Check for a good calibration:

(1) With the end of the test cable open circuited, the network analyzer display should show a magnitude of approximately 1.0 (or 0 dB) and a phase of approximately 0° over the entire frequency range.

(2) With the end of the test cable shorted, the analyzer display should show a magnitude of approximately 1.0 (or 0 dB) and a phase of approximately ±180° over the entire frequency range.

(3) If you do not obtain these test results, repeat the calibration. First, however, inspect the 50-Ω test cable. There are many defective cables in the laboratory. If in doubt, replace the cable before recalibrating.

3. Measure s_{11}.

a. Connect port 1 of the circuit you wish to test to the 50-Ω test cable on the network analyzer as shown in Figure 2 below. Put a 50-Ω load (a 51- or 47-Ω resistor) on port 2 of the circuit in order to properly terminate the port for *s*-parameter measurement. (You could use the B port of the network analyzer to provide the 50-Ω termination, but by providing your own termination the accuracy of the measurement will not be degraded by the characteristics of the cable to the B port.) The 50-Ω impedance of the TEST 50 Ω port of the signal divider provides the proper termination for port 1 of the circuit under test.

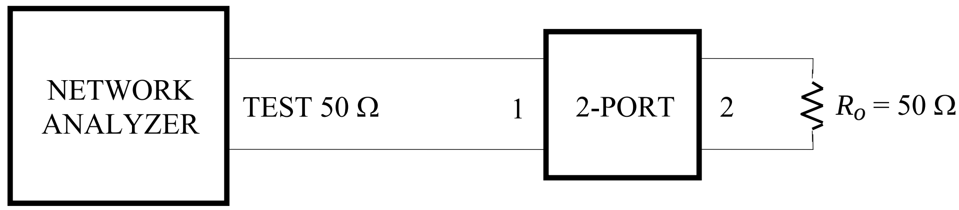

Figure 2. Circuit connection for s_{11} measurement.

b. The measured value of s_{11} appears on the screen as a function of frequency.

(1) For a rectangular display, the value of s_{11} at any frequency can be read directly from the displayed graphs.

(2) For rectangular, polar, or Smith-Chart displays, push the *Marker* button in the MARKER section of the network analyzer panel to place a marker on the screen. The knob can now be used to move the marker to any frequency in the measurement range.

(3) Use the knob to put the marker on a frequency of interest. (Marker frequency is displayed on the screen following the statement MARKER.)

(a) Rectangular plots: $|s_{11}|$ at the marker frequency appears just above the top graph; *ang* s_{11} at the marker frequency appears just above the lower graph.

(b) Polar plots: The magnitude and angle of s_{11} at the marker frequency are displayed in the upper right-hand corner of the screen.

(c) Smith Chart plots: The real and imaginary parts of the impedance corresponding to the measured s_{11} at the marker frequency are displayed in the upper right-hand corner of the screen.

4. Measure s_{22}.

(1) Connect port 2 of your circuit to the cable from the signal divider as shown in Figure 3 below.

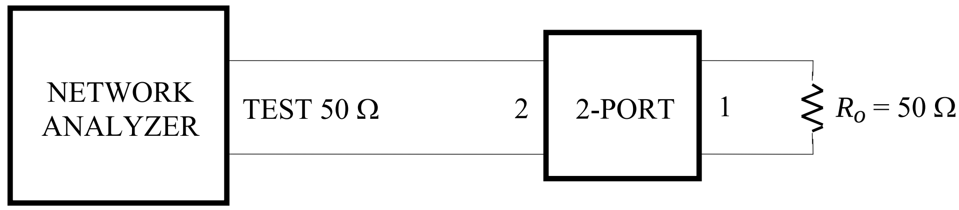

Figure 3. Circuit connection for s_{22} measurement.

(2) Remove the 50-Ω termination (the 47- or 51-Ω resistor) from port 2 of your circuit and put it on port 1 to terminate that port properly. The signal divider now provides the proper termination for port 2, the port under test.

(3) The measured value of s_{22} appears on the screen as a function of frequency. The value of s_{22} at any frequency can be read directly from the displayed graph (rectangular display), or steps II.C.3.b. above can be repeated.

D. Measurement of s_{21} and s_{12}.

1. Configure the network analyzer:

a. Rectangular display:

(1) Press the *Chan* 1 button in the ACTIVE CHANNEL section of the analyzer panel. The lamp to the left of the button should light.

(2) Press the *Meas* button in the MEASUREMENT section of the network analyzer panel. The measurement menu appears on the right side of the screen.

(3) Press the button next to S PARAMETERS (near the bottom of the on-screen menu). The measurement menu is replaced by the *s*-parameter menu on the screen.

(4) Press the button next to the S21 [B/R] statement in the *s*-parameter menu. <u>S21 [B/R]</u> will be underlined.

(5) Press the *Chan* 2 button in the ACTIVE CHANNEL section of the analyzer panel. The lamp to the right of the button should light.

(6) Repeat steps II.D.1.a.(1), (3), and (4).

b. Polar plot:

(1) Press the *Meas* button in the MEASUREMENT section of the network

analyzer panel. The measurement menu appears on the right side of the screen.

(2) Press the button next to S PARAMETERS (near the bottom of the on-screen menu). The measurement menu is replaced by the *s*-parameter menu on the screen.

(3) Press the button next to the S21 [B/R] statement in the *s*-parameter menu. S21 [B/R] will be underlined.

The network analyzer is now configured to measure s_{21} and s_{12}.

2. Calibration. The network analyzer must now be calibrated for voltage gain measurements.

a. Connect a **50-Ω** test cable to the TEST 50 Ω port on the signal divider. Use a cable that will work well over the selected frequency range. Connect another **50-Ω** test cable (similar to the one connected to the TEST 50 Ω port of the signal divider) to the B 50 Ω port of the network analyzer.

b. Select the calibration kit:

(1) Press the *Cal* button in the MEASUREMENT section on the analyzer panel. Several options are listed on the right side of the screen.

(2) Press the button next to CAL KIT. The list of options is replaced by a list of possible calibration kits.

(3) If the 50-Ω test cable has clip leads on one end, press the button next to USER KIT. USER KIT will be underlined.

(4) If the 50-Ω test cable has coaxial connectors at both ends, press the button next to the connector (*e.g.*, N 50Ω) that will attached to the device under test. The selected calibration kit will be underlined.

c. Rectangular display

(1) Each channel must be calibrated separately. Calibrate Channel 1 first. Press the *Chan* 1 button on the network analyzer.

(2) Press the *Cal* button in the MEASUREMENT section of the analyzer panel. A list of options appears at the right side of the screen.

(3) Press the button next to CALIBRATE MENU. The list of options at the right side of the screen is replaced by the calibration menu.

(4) Press the button next to RESPONSE. The calibration menu on the screen is replaced by a list of SHORT OPEN THRU.

(5) One by one, place the appropriate calibration standard (open, short, or

thru) at the end of the 50-Ω test cable *from the 50Ω SIGNAL DIVIDER* and press the button next to that standard. (If the test cables have clip leads on one end, SHORT means connect the clip leads on the cable from the 50Ω SIGNAL DIVIDER together, OPEN means leave the clip leads on the cable from the 50Ω SIGNAL DIVIDER unconnected, and THRU means connect the red clips of both test cables together and the black clips of both test cables together.) When all standards have been done, press the button next to DONE: RESPONSE. (Instructions to do these steps appear on the screen.)

(6) Calibrate Channel 2. Press the *Chan* 2 button on the network analyzer panel, then repeat steps II.d.2.c.(2)-(5).

d. Polar display

(1) Press the *Cal* button in the MEASUREMENT section of the analyzer panel. A list of options appears at the right side of the screen.

(2) Press the button next to CALIBRATE MENU. The list of options at the right side of the screen is replaced by the calibration menu.

(3) Press the button next to RESPONSE. The calibration menu on the screen is replaced by a list of SHORT OPEN THRU.

(4) One by one, place the appropriate calibration standard (open, short, or thru) at the end of the 50-Ω test cable *from the 50Ω SIGNAL DIVIDER* and press the button next to that standard. (If the test cables have clip leads on one end, SHORT means connect the clip leads on the cable from the 50Ω SIGNAL DIVIDER together, OPEN means leave the clip leads on the cable from the 50Ω SIGNAL DIVIDER unconnected, and THRU means connect the red clips of both test cables together and the black clips of both test cables together.) When all standards have been done, press the button next to DONE: RESPONSE. (Instructions to do these steps appear on the screen.)

e. Check for a good calibration:

(1) With the end of the test cable open circuited, the network analyzer display should show a magnitude of approximately 0 over the entire frequency range.

(2) With the end of the test cable from the TEST 50 Ω shorted, the analyzer display should show a magnitude of approximately 0 over the entire frequency range.

(3) With the two test cables connected together, the analyzer display should show a magnitude of approximately 1.0 and a phase of approximately 0° over the entire frequency range.

(4) If you do not obtain these test results, repeat the calibration. First,

however, inspect the 50-Ω test cable. There are many defective cables in the laboratory. If in doubt, replace the cable before recalibrating.

3. Measurement of s_{21}.

   a. The network analyzer provides the proper 50-Ω termination for both ports of your network. A 50-Ω termination (the 47- or 51-Ω resistor) is not required.

   b. Connect port 1 of your circuit to the cable from the TEST 50 Ω port of the signal divider, and connect port 2 of your circuit to the INPUT B port of the analyzer as shown in Figure 4 below.

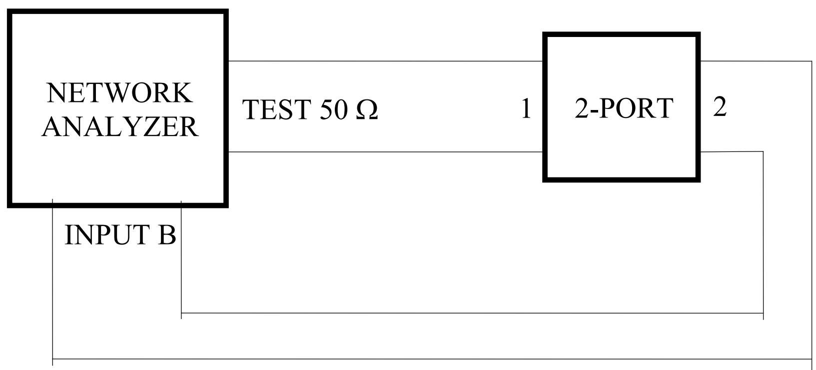

Figure 4. Circuit connection for s_{21} measurement.

   c. s_{21} now appears on the screen as a function of frequency. As above, the marker can be moved to any desired frequency in the measurement range with the knob, and the magnitude and phase of s_{21} at that frequency can be read from the screen.

4. Measurement of s_{12}.

   a. The network analyzer provides the proper 50-Ω termination for both ports of your network. A 50-Ω termination (the 47- or 51-Ω resistor) is not required.

   b. Connect port 2 of your circuit to the cable from the TEST 50 Ω port of the signal divider, and connect port 1 of your circuit to the INPUT B port of the analyzer as shown in Figure 5 below.

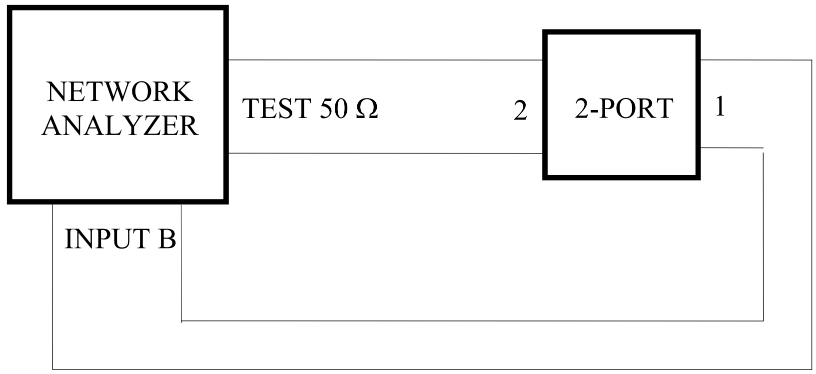

Figure 5. Circuit connection for s_{12} measurement.

c. s_{12} now appears on the screen as a function of frequency. As above, the marker can be moved to any desired frequency in the measurement range with the knob, and the magnitude and phase of s_{12} at that frequency can be read from the screen.

III. Measurement of 2-terminal impedances

The network analyzer can be used to measure the impedance of two-terminal elements such as resistors, capacitors, and inductors. Measure with a Smith Chart display as described in Section II.C. above. (NOTE that a two-terminal element—*e.g.*, an inductor—does *not* have a second port that must be terminated with a matched load.) The real and imaginary parts of the impedance will be displayed in the upper right corner of the screen.

PC BOARD FABRICATION

I. Purpose

This chapter provides the information needed to fabricate simple printed circuit (PC) boards. Some of the circuits you will need to construct during the course of your engineering education will have to be built on PC boards instead of protoboards. Simple PC boards are easily fabricated in the laboratory.

II. PC Board Basics

A. Conventional PC board.

The conventional PC board is shown in Fig. 1 below. The leads of the components pass through holes drilled in the board. The leads are soldered to the circuit conductors on the top and/or bottom of the board. The holes in commercially-produced boards have copper plated through them to provide additional soldering surface. This increases the mechanical strength of the solder joint. We lack the equipment to plate copper through the holes, but we produce usable PC boards without this added step.

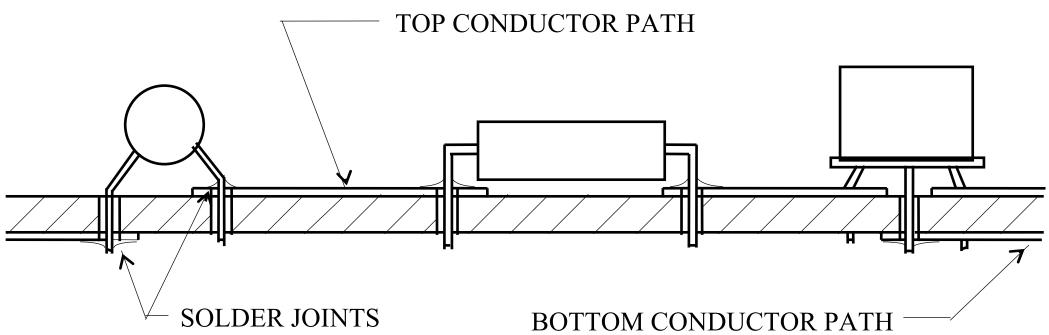

Figure. 1. A conventional PC board (edge view).

B. Surface mount PC board.

A PC board technique that is gaining favor is the surface mount PC board, shown in Fig. 2 below. A surface mount PC board uses holes drilled through the board only to connect circuit conductors on opposite sides of the board. Surface mount PC boards allow placing more components in the same space because "surface mount devices" (SMD) with small or no leads are used. However, conventional electrical components can also be used, as shown in Fig. 2. Surface mount solder bonds are more easily broken than conventional through-the-board solder bonds.

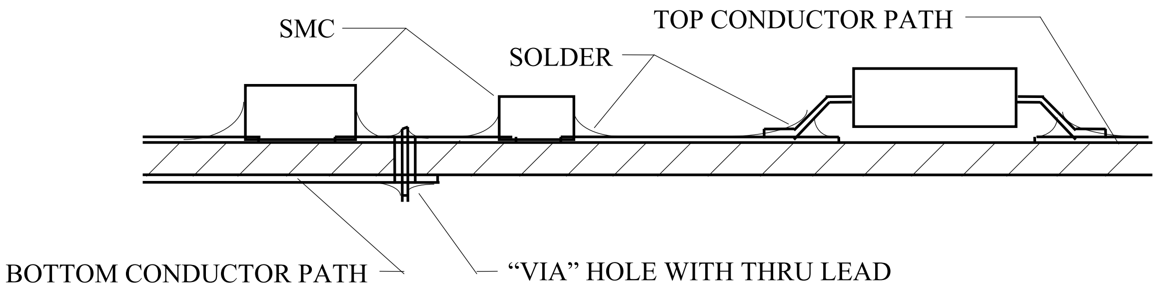

Fig. 2. A surface mount PC board (edge view).

C. PC board layers.

1. Single-sided PC boards.

Single-sided boards have conductor metallization on only one side.

2. Double-sided PC boards.

Double-sided boards have conductor metallization on both sides. Double-sided boards are necessary if the circuit pattern involves crossovers. Also, one side of a double-sided board can be left unetched to provide a large ground plane. Large ground planes significantly improve the operation of high-frequency circuits.

3. Multilayer PC boards.

Complex digital circuits require multilayer boards, which are fabricated by bonding several double-sided boards on top of each other. The fabrication of multilayer boards requires precision alignment of the individual boards.

III. Fabrication Procedure

A. Etching PC boards.

1. Required material:

a. Blank PC board.

A blank PC board has a solid sheet of copper on one or both sides of the board. The *unwanted* copper is etched or milled away, leaving the desired circuit pattern.

Blank PC boards, both single- and double-sided, can be obtained from Radio Shack or EE Department laboratory technician. If the board is to be etched, the blank board should be cut to the size of the finished board before the circuit pattern is etched. The tools in the Circuits Laboratory can be used to cut the board to the required size.

b. Board cleaner.

You will etch better PC boards if you start with a *clean* piece of blank board. Good PC board cleaners include:

(1) Acetone.

(2) A commercially-available copper cleaner (for cleaning copper kitchen utensils).

(3) 400-grit sandpaper.

c. Masking material.

A blank PC board has a sheet of copper conductor covering at least one side of the board. Circuit paths, component mounting pads, and ground planes must be created by removing the minimum amount of copper from the board. (Copper is a toxic metal and the disposal of copper etched from PC boards can pose environmental problems.) Leave as much copper as possible connected to the ground paths and power supply lines to minimize the resistance and inductance of these circuit paths.

The copper that will remain on the board must be masked to protect it from the etchant. Materials that can be used for masking include:

(1) Almost any kind of tape.

(2) Any oil-based ink. A good, permanent laundry marker works. WARNING: good results are hard to obtain when using ink.

(3) Fingernail polish (really!). Sally Hanson's "Tough as Nails" works well. It can be removed with acetone.

(4) PC board masking tape (cut to shape for holes, runners, IC's, etc.) available from Radio Shack.

d. Etchant.

Unwanted copper is removed from the board with an etchant. Commonly used etchants include:

(1) Ferric chloride (available from Radio Shack; one bottle will etch a 10 x 10-in. PC board).

(2) Ammonium persulfate.

Etchants work better at higher temperatures — 110° F works well.

e. Etching tank.

An etching tank in which the PC board and etchant can be placed is required. The tank must be made (or at least lined) with a material that will not react with the etchant. A glass or plastic cake pan works well.

2. Fabrication steps. NOTE: Follow the safety guidelines in section III.C below.

a. Select either a single- or a double-sided configuration.

b. Determine the position of each component.

c. Drill all necessary holes.

d. Clean the board.

e. Mask the circuit pattern.

f. Etch the board.

g. Solder components into position.

h. Test the completed circuit.

B. Milling PC boards.

1. Required material:

Blank PC board.

A blank PC board has a solid sheet of copper on one or both sides of the board. The *unwanted* copper is etched or milled away, leaving the desired circuit pattern.

Blank PC boards, both single- and double-sided, can be obtained from Radio Shack. If the board is to be etched, the blank board should be cut to the size of the finished board before the circuit pattern is etched. The tools in the Circuits Laboratory can be used to cut the board to the required size.

2. Fabrication steps.

a. Create either a *detailed* drawing including all dimensions, hole sizes, etc., or an AutoCAD *.dxf* file of the desired circuit pattern.

b. Give the drawing or *.dxf* file and blank PC board to the EE Department laboratory technician. He will mill the PC board for you.

B. Fabrication Safety.

Heed the warning signs on the power tools in the Circuits Laboratory and *WEAR EYE PROTECTION* when using them. If you don't have safety glasses, the eye goggles you used in the chemistry laboratory can be used. Be careful when soldering — hot solder flux can boil, causing flux bubbles to burst and spray hot flux and small pieces of molten solder around your work area.

Don’t try to etch a larger piece of PC board material than is needed for your final circuit; that only wastes material. However, if your completed PC board will be small and difficult to handle, don’t cut your board to its final size before you have drilled all necessary holes in the board. When drilling holes in a very small board, your hands will get uncomfortably close to moving parts of the drill.

LABORATORY SAFETY

I. Purpose

Engineering students are preparing for careers in which they will have to supervise technical work involving varying degrees of electrical and mechanical hazard. Each engineering student should give serious thought to the subject of accident prevention and prepare hmself/herself mentally to recognize and avoid dangerous situations and to cope with emergencies. This self training is essential since there is no set of safety rules that apply to all situations.

II. Fire safety

A. KEEP CALM.

Your example is important — panic is contagious.

B. EXTINGUISH FIRE OR EVACUATE.

1. If the fire is small (such as a wastebasket), use the nearest fire extinguisher according to the directions on the fire extinguisher nameplate. NOTE: Once used, the extinguisher needs to be serviced, even if it is not empty. Report it to the Physical Plant at 342-1155.

2. If in doubt, EVACUATE THE BUILDING AND CALL THE FIRE DEPARTMENT.

a. Activate the building fire alarm (red boxes in corridors).

b. Implement the emergency evacuation plan for the building.

c. Phone the FIRE DEPARTMENT (9-911 from a University telephone, 911 from a pay telephone). Report the exact location of the fire.

d. If safe to do so:

(1) Check all areas for people.

(2) Notify CAMPUS POLICE.

(3) Close doors and windows.

(4) Turn on lights.

e. Meet fire fighters and police staff at the building entrance to provide additional information and assistance.

III. Tornado safety

A. KEEP CALM.

Your example is important — panic is contagious.

B. SEEK SHELTER.

1. Follow instructions posted next to classroom and laboratory doors.

2. Stay together and move to lowest level, interior hallways or designated shelter areas.

3. Avoid areas with large expanses of glass.

IV. Electrical Safety

A. Dangerous current levels:

1. Voltage is not a reliable indication of danger. IT IS THE CURRENT THAT KILLS.

2. Electrical resistance of the human body:

a. The resistance of the body varies depending on the skin condition (dry, wet, sweaty, etc.) at the point of contact. Therefore it is almost impossible to predict how much current will be produced in the body by a given voltage.

b. Skin resistance can vary from 1000 Ω for wet skin to over 500 000 Ω for dry skin.

c. When the skin is broken, the body presents no more than 500 Ω of resistance.

3. Current path through the body:

The path of the current through the body has much to do with the shock's danger. A current from finger to elbow through the arm may produce only a painful shock, but the same amount of current from one hand to the other or from hand to foot might prove fatal.

4. Current amplitude effects:

a. The intensity and danger of an electrical shock increases with the amount of current.

b. A painful shock can result from 10 mA.

c. 20 mA might be sufficient to cause muscle paralysis.

d. As the current approaches 100 mA breathing stops.

e. The current range of 100 to 2000 mA is particularly dangerous because it is almost certain to result in lethal ventricular fibrillation (uncontrolled twitching of the walls of the heart's ventricles).

f. Above 2000 mA, the muscular contractions are so severe that the heart is forcibly clamped during the shock. This clamping of the heart protects the heart from going into ventricular fibrillation, so the victim's chances for survival are improved. However, there will be other effects (such as severe burns) as the level of current increases.

B. In case of electrical shock:

1. It is important to free the victim from the current as quickly as can be done *safely*.

2. Do not touch the victim until the electric power is turned off — you can not help a person by becoming a second victim.

C. Rules for working with electrical equipment and circuits:

1. When wiring a circuit, always connect the power source as the last step. When disassembling a circuit, disconnect the power source first.

2. Do not touch any electrical equipment when your hands are wet or sweaty.

3. All electric circuits — even unenergized circuits — are potentially dangerous. For example, a large capacitor may contain charge long after it is disconnected if it does not have a discharging resistor.

4. Adjustments in energized circuits should be made with extreme caution. When working with live electrical circuits use only one hand if possible; keep the other hand away from the circuit.

5. Keep watch chains or bands, rings, and other metallic objects out of contact with equipment and circuits.

6. Avoid exposing your eyes to electric arcs from any source. Electric arcs are powerful generators of ultraviolet radiation which can cause serious eye damage or blindness.

7. Haste causes many accidents. Work deliberately and carefully. Check your work as your proceed. Good planning before constructing the circuit will promote safety.

8. If you suspect faulty equipment, notify your instructor and report the fault on the EE Department's Web page (click on *Equipment Repair Form* on the left side of the page). Do not attempt to perform any maintenance yourself.

V. Mechanical safety.

A. *NEVER* remove any safety shields or safety interlocks from the machinery.

B. Eye protection:

Wear appropriate eye protection (safety goggles, safety glasses) when working with any equipment that can produce dust, splinters, slivers, shards, etc.

C. Clothing:

Do not wear loose-fitting clothing (e.g., ties, balloon sleeves, etc.) when working with rotating machinery (lathes, milling machines, circular saws, band saws, drill presses, etc.). Such clothing items could become caught in the equipment and drag parts of the body into contact with the machinery.

D. Keep hands well clear of moving machinery. When working with small pieces of stock, use jigs to support the work piece so that hands can be kept at a safe distance from the machinery.

E. Haste causes many accidents. Work deliberately and carefully. Check your work as you proceed. Good planning before beginning work will promote safety.

F. If you suspect faulty equipment, notify your instructor. Do not attempt to perform any maintenance yourself.