EECS 211: Electrical Engineering II

Laboratory Manual

By: Prof. Gabriel M. Rebeiz
The University of Michigan
EECS Dept.
Ann Arbor, Michigan

Introduction

This laboratory manual combines six experiments designed to introduce you to the world of electronics and electrical signals. We will concentrate on applications to audio circuits. Please approach the labs with curiosity and a "what if ...?" approach. Ask questions and try other circuits if you wish (keeping always safety in mind).

Laboratory Safety:

The voltages and currents encountered in your experiments are not dangerous. You will, however, be working with laboratory equipment connected to the 120 V, 60 Hz AC voltage. This voltage <u>can</u> be lethal under certain conditions. It is therefore extremely important that you observe safe laboratory procedures at all times and follow the instructions carefully.

- <u>Immediately</u> report all dangerous conditions (stripped 120 V, AC lines; sparks in laboratory equipment; loose wall socket; ...) to your laboratory instructor. <u>Report any condition even if you are not sure of the hazardous level</u>.

- For fire, medical, police, notify your laboratory instructor or call 911.

- No student is permitted in the lab alone at <u>any</u> time. The student must be accompanied at all times by a laboratory partner and/or the laboratory instructor. This rule is observed for regular and make-up lab sessions.

- If any equipment breaks, please report it immediately to the instructor.

- Any student who knowingly imposes a dangerous electrical situation on himself, other students or the lab instructor (ex.: stripping 110V lines; electrocuting himself or other students, ...) will be denied access to the laboratory and will receive a failing grade in the class (F).

- Remember: Safety, Safety, Safety! When in doubt, ask your instructor and do not touch it!

Honor Code:

The Honor Code applies to all laboratory work, which means the pre-lab preparation, the experiments and the final lab report. You are expected to include the following Honor Code statement on all your work:

"I have neither given nor received aid on this report, nor have I concealed any violation of the Honor Code."

Collaboration Policy:

You are encouraged to work with your laboratory partner and/or with a group of friends on the pre-lab, lab-reports and homework problems. However, you <u>cannot</u> <u>copy</u> the work of anybody else (enrolled in the class or not). <u>You must write your own work without looking at anybody's work</u>. You are allowed to help (or receive help from) your colleagues in the form of discussing the work, explaining the problem and even telling (or learning) in detail how to solve a problem (verbally with few sketches). <u>But you cannot copy the final (or near final) work of anybody.</u>

Grading Policy:

Laboratory	20%
CAD Assignments	10%
Homeworks	10%
Quiz I	15%
Quiz II	15%
Final Exam	30%

The grading policy of each lab is:

Pre-Lab Report	25%	(on time)
	0%	(late)
Data Taken in Lab	25%	
Post Lab Reports	50%	(on time)
	25% Penalty	(one day late)
	50% Penalty	(two days late)
	100% Penalty	(three or more days late)

<u>IMPORTANT RULE</u>

Any student missing a lab (not present in the lab) with no proper or reasonable excuse will get a "0" grade on that specific lab and will have his/her final letter grade reduced by a full letter (for ex.: $A^- \rightarrow B^-$). Any student missing two labs with no proper excuse will automatically get a failing grade (F) on the <u>entire</u> course.

Laboratory Procedures:

This laboratory manual combines six experiments designed to introduce you to the world of electronics and electrical signals. We will concentrate on applications to audio circuits. Please approach the labs with curiosity and a "what if ...?" approach. Ask questions and try other circuits if you wish (keeping always safety in mind).

The experiments are designed to be performed within the 3-hour laboratory time. During each laboratory period, you will be expected to carry out one experiment, to record the experimental data, to make a few computations to determine how it agrees with expectations, perhaps plot some graphs, and then answer some questions concerning the experimental work. The

laboratory instructor will then sign on your notebook <u>before</u> you leave the lab. You will then prepare the final lab report which is due one week after your lab session. To successfully complete the experiments in one lab period, you must come prepared to the laboratory. You should read the experiment in advance and answer the pre-lab questions. The pre-lab counts for 25% of the lab grade and must be turned in at the <u>beginning</u> of the lab. Also having the following:

1. Your notebook (numbered pages).

2. An INK pen (color is a plus).

3. A calculator capable of *sin*, *cos*, *log*, *Ln*, e^x functions.

4. A ruler!

All engineers use exclusively ink and lab reports with numbered pages. This is extremely important in companies where patent issues and prior work or knowledge of work may be challenged in a court of law at anytime. So, it is good for you to start learning the correct way to handle data. If you make a mistake, neatly cross it out and continue on. If a graph is incorrect, cross it out too with a large X, put a line and continue. <u>No</u> points will be deducted for a crossed out material, as long as it is neat. However, lab reports with hash marks all over the place, data presented in haphazard fashion and general sloppiness will suffer as much as 50% penalty (in severe cases) on the final grade.

The following guidelines will result in a good lab report:

1. Always put a <u>Unit</u> (V, A, Hz, ...) after every input or answer. <u>Underline</u> or circle your answer.

2. Always <u>label the axis</u> on a graph and put the units too.

3. When using an equation, write it out first in analytical form (Ex.: $R_p = R_1R_2/(R_1 + R_2)$) and then substitute the data.

4. Sketch a circuit diagram neatly and label the component values.

5. Sketch an experimental diagram and label the equipment used.

6. Comment on the agreement or lack of agreement with the calculated values. Also comment if the measured results are unexpectantly too high or too low. It is rather o.k. to have done an unsuccessful experiment (I do it all the time) but it is NOT o.k. to accept the data as it comes without thinking about it, questioning if it makes sense, and knowing finally what went wrong!

    No papers can be attached to the lab notebook. Everything has to be cut and glued to the lab notebook.

Laboratory Experiments:

The laboratory procedures will be as follows:

1. **<u>Pre-Lab</u>:** Due the date of the laboratory. <u>Each</u> student submits his/her own copy of the pre-lab and signs the Honor Code.

2. **<u>Experiment</u>:** The student works with a partner and they both take the data in <u>separate</u> notebooks. All laboratory data will be written in INK on a laboratory notebook. The lab notebook must have numbered pages and be the kind to which pages cannot be added. The lab instructor will look at the data and <u>sign</u> on your notebook at the end of the experiment.

3. **<u>Laboratory Report</u>:** A short lab report is due exactly one week later in the lab. Each student will write his/her own lab report on the notebook and sign the Honor Code.

3

<u>Laboratory #1</u>: Low-Pass Filters; Step Response vs. Q

Laboratory #2:   Active Bandpass Filters; Frequency Shift Keying Communications; Step Response vs. Q

Laboratory #3:   Diodes and Bridge Rectifiers; Power Supplies

Laboratory #4:   Diode Small-Signal Measurements; Amplitude Modulation; Envelope (AM) Diode Detectors

Laboratory #5:   LEDs, Phototransistors and an AM Optical Link

Laboratory #6:   Frequency Modulation (FM); Generation and Detection of FM Signals; FM Optical Link

Knowledge Level Needed for Lab Experiments:

Experiment No. 1:   Active Low-Pass Filter

Pre-lab/lab:
- MatLab
- S-domain, Bode plots, poles and zeroes
- First order circuits in time domain
- An introduction to second order circuits
- Input and output impedances of networks

Lab-Report and CAD Assignment:
- Second order circuits in time and frequency domain
- Complex Poles (ω_0, Q)
- Introduction to Spice (Mentor Accusim).

Experiment No. 2:   Active Band-Pass Filters

Pre-lab/lab:
- Second order circuits in time and frequency domain
- Complex Poles (ω_0, Q)
- Concept of frequency shift keying (FSK)

Lab Report and CAD Assignment:
- Better knowledge of Spice (Mentor Accusim)
- Complex input and output impedances of circuits

Experiment No. 3:   Full-Wave Diode Rectifiers

Pre-lab/lab:
- Diode operation in large signal mode (rectification)
- Full-wave (bridge) rectifier operation
- Single ended and center-tapped transformers
- Linear regulators (efficiency, ripple, etc..)

Lab Report and CAD Assignment:
- Defining a diode model in Spice
- Use of Spice with non-linear components (diodes)

Experiment No. 4: Linear and Non-Linear Diode Analysis, AM Modulation/Detection

Pre-lab/lab:
- Small signal resistance of diodes in linear region
- Operation of diodes in square-law regime

4

- Amplitude modulation (sidebands, modulation depth)
- Envelope (AM) detectors

Lab Report:
- No new material needed

Experiment No. 5: AM Optical Link

Pre-lab/lab:
- LED operation
- Phototransistor operation (responsivity, dark current, ...)

Lab Report:
- No new material needed

Experiment No. 6: FM Optical Link

Pre-lab/lab:
- FM modulation (sidebands, maximum deviation, modulation index)
- FM generation using voltage-controlled-oscillators (VCO)
- FM detection using a slope detector

Lab Report:
- No new material needed

The Agilent 3631A Power Supply
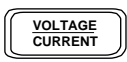
The Agilent 3631A has three power supplies, a +6 V supply capable of delivering 5A, and two supplies of +25 and –25 V capable of delivering 1A each. The $\perp$ (ground) output is the reference ground and is connected to the ground of the building. This is why it is important to connect the COM (common) terminal of the ±25 V supplies, and the (–) terminal of the +6 V supply to the $\perp$ (ground) reference.

Looking now at the control keys:

1. The $\boxed{\substack{\text{Output} \\ \text{On/Off}}}$ key turns the output ON or OFF.

2. **To Set the Output Voltage:**
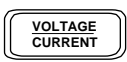
a. Press the $\boxed{\text{+6 V}}$, $\boxed{\text{+25 V}}$, $\boxed{\text{–25 V}}$ keys to select the power supply to be set.

b. Press $\boxed{\substack{\text{VOLTAGE} \\ \text{CURRENT}}}$ key so that the Volt Display is active.

c. Use the circular control knob to set the output voltage. Use the arrow keys for selecting the resolution.

3. **To Set the Maximum Output Current:**
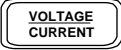
a. The $\boxed{\substack{\text{Display} \\ \text{Limit}}}$ key lets you select the maximum current that the power supply can deliver (up to 5A for the 6V and 1A for the ±25V supplies). This is basically your current protection feature.

b. Press the $\boxed{\substack{\text{VOLTAGE} \\ \text{CURRENT}}}$ key so that the Current Display is active.

c. Use the circular control knob and the resolution keys to set this limit (if needed).

5

4. **To Read the Output Voltage or Output Current:**

a. The ⌈VOLTAGE/CURRENT⌉ key also shows the output voltage and the output current of the power supply.

b. To measure the output current of the supply, make sure that the ⌈Display Limit⌉ key is not active.

The Front Panel at a Glance
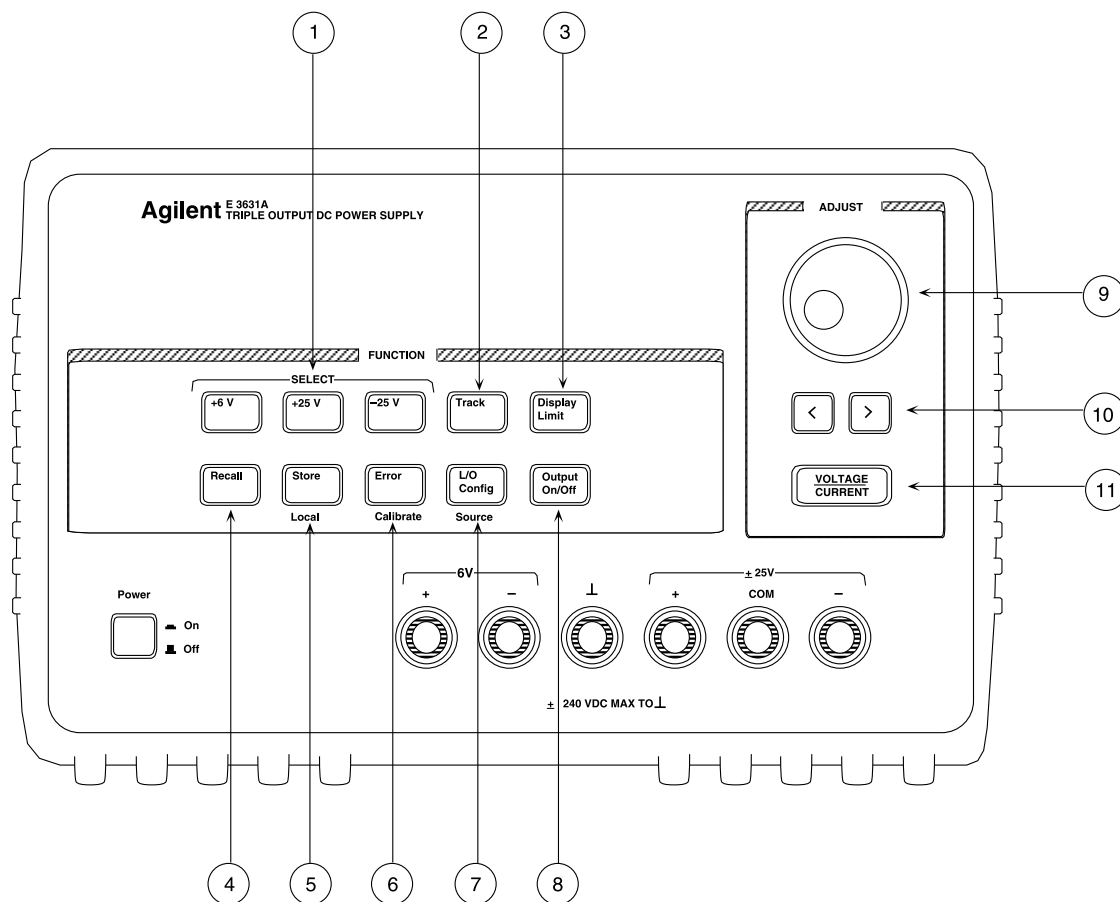
1 Meter and adjust selection keys	7 I/O Configuration / Secure key
2 Tracking enable/disable key	8 Output On/Off key
3 Display limit key	9 Control know
4 Recall operating state key	10 Resolution selection keys
5 Store operating state / Local key	11 Voltage/current adjust selection key
6 Error / Calibrate key	

The Agilent 33120A Function Generator

The Agilent 33120A generates a whole range of waveforms with precise control on the frequency amplitude, waveform shape, offset voltage, etc. We will use only a small portion of its capabilities in EECS 210, but the Agilent 33120A will "shine" in EECS 211 with its AM, FM, FSK, etc. modulation features.

1. **To Choose a Waveform:**

    Simply press the AC waveform keys, $\boxed{\sim}$ for a sinewave,$\boxed{\sqcap}$ for a square wave, $\boxed{\sim}$ for a triangular wave or $\boxed{\diagup}$ for a sawtooth wave.

2. **To Set the Frequency:**

    a. Press the $\boxed{\text{Freq}}$ key.

    b. Enter the frequency either by using the control (circular) knob and the horizontal (or vertical) arrows for choosing the significant digits. You can also <u>enter</u> the frequency numerically:

    b1. Press $\boxed{\text{Enter Number}}$

    b2. Enter the digits. For example 1•249

    b3. Enter the unit. For example kHz (or Hz or MHz)

    The sine and square waves can be set from 0.1 mHz to 15 MHz.
    The triangular and sawtooth waves can be set from 0.1 mHz to 100 KHz.

3. **To Set the Amplitude:**

    a. Press the $\boxed{\text{Ampl}}$ key.

    b. Enter the amplitude by either using the control (circular) knob and the horizontal or vertical arrows for choosing the significant digits. You can also <u>enter</u> the amplitude numerically:

    b1. Press $\boxed{\text{Enter Number}}$

    b2. Enter the digits. For example 2.4

    b3. Enter the unit. For example Vpp (or Vrms or dBm)

    b4. You can also use the $\boxed{\text{Shift}}$ key to enter values in mV.

    The amplitude can be set from 100 mVppk to 20 Vppk into an open circuit (50 mVppk to 10 Vppk into a 50 Ω load).

    IMPORTANT: The firmware inside the Agilent33120A assumes that you have placed a 50 Ω load across its output, in accordance with good RF engineering practice. It then calculates the output voltage by dividing its internal open-circuit voltage by two, and displays the answer. If the circuit that you connect to the output of the Agilent33120A does NOT have an equivalent Thevenin impedance of 50 Ω, then the

7

voltage displayed will be inaccurate. For a high impedance load, you can set the display to "HI-Z", and the function generator will display the correct number. Do this by hitting:

    (Blue shift + MENU)

    right arrow to SYS MENU

    down arrow to OUT TERM

The right/left arrows or the knob now toggle you between 50 Ω and HIGH Z. Set it to HIGH Z and hit enter. The display will now indicate the front panel voltage for a high impedance load.

4.   **To Set a DC Offset:**

The [Offset] key allows you to add a DC offset voltage to any AC waveform. Normally, the offset is set to zero volts and the AC waveform has an average value of zero. However, in some cases, it is good to offset the AC waveform to simulate DC and AC inputs to a circuit.

a.   Press the [Offset] key.

b.   Enter the value by either using the control (circular) know or using the [Enter Number] key, entering the number and choosing the units (V or mV).

c.   You can also choose a positive or negative offset using the ± (green) key in the numerical entry.

5.   **To Set the Duty Cycle:**

The [Shift] % Duty key allows you to set the duty cycle of a square waveform only. It is easiest to select the value using the control knob (or using the [Enter Number] menu).

The Front Panel at a Glance
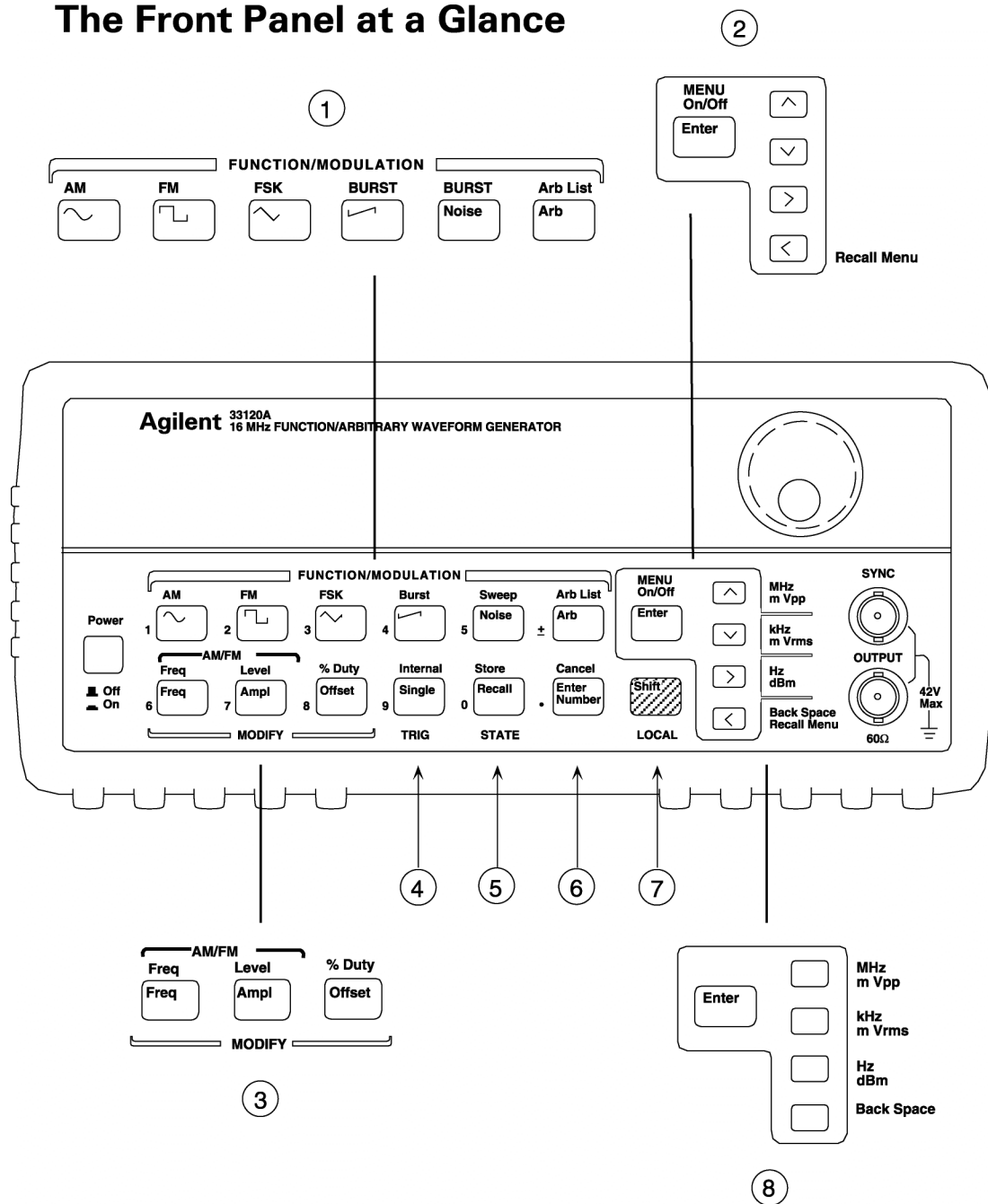
|	
1 Function / Modulation keys	5 Recall / Store instrument state key
2 Menu operation keys	6 Enter Number key
3 Waveform modify keys	7 Shift / Local key
4 Single / Internal Trigger key	8 Enter Number "units" keys
(Burst and Sweep only)	

The Agilent 34401A Multimeter

The Agilent 34401A multimeter is a high precision, 6 $^1/_2$ digit multimeter, capable of measuring V, I, resistance, freq, time, etc. for DC and AC signals. Throughout the lab experiments, we will only use a small portion of its capabilities, but you are encouraged to learn all what you can about it.

1. **Measuring Voltages:** Voltages are measured in <u>parallel</u> with a circuit. The $\boxed{\text{DC V}}$ (or the $\boxed{\text{AC V}}$) key should be selected. The example below shows how to measure V_B (V_B is positive):
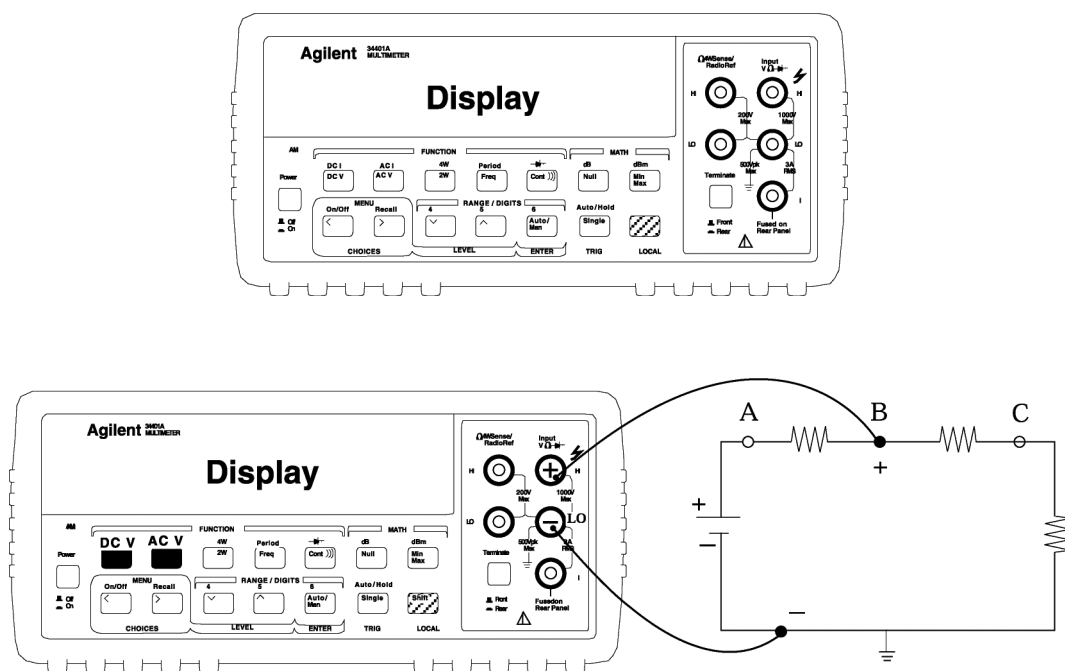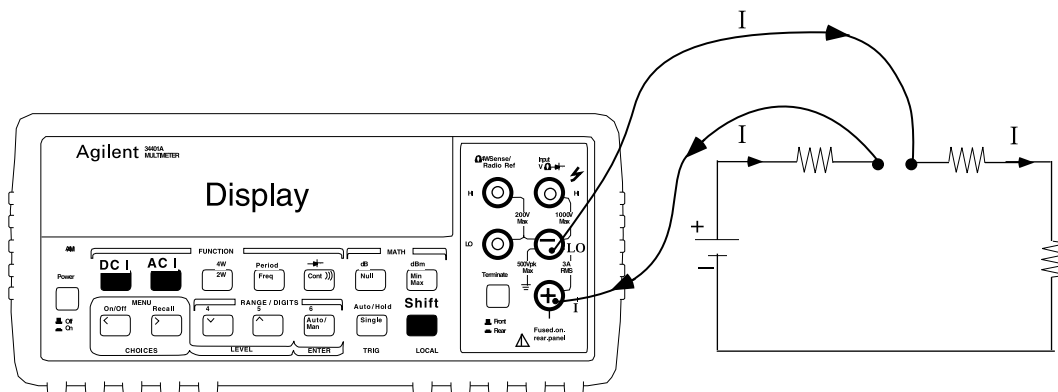

The example below shows how to measure V_{BC} (V_{BC} is positive):
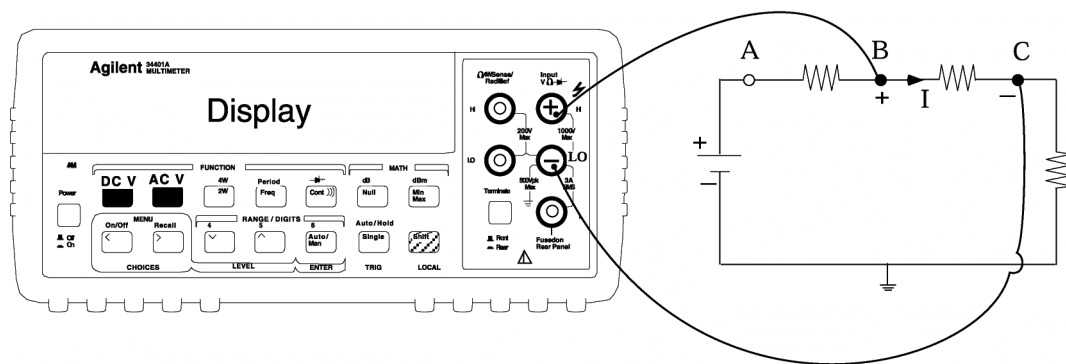
The input resistance of the voltmeter is 10 MΩ. This means that it will not load the circuit and result in a large measurement error if the resistances are 300 kΩ or smaller (see lab report assignment).

2. **Measuring Currents:** Currents are measured in <u>series</u> with a circuit. Basically, the current passes by the multimeter so that it can be measured. The [DC V | DC I] or [AC V | AC I] keys should be selected using the blue [Shift] key. The example below shows how to measure I (I is positive):


The input resistance of the ammeter is 0.1 Ω for the 1A and 3A ranges, and 5 Ω for the 10 mA and 100 mA ranges. This means that it will not load the circuit and result in a large measurement even for most resistance values (100 kΩ to 2 Ω).

In order to measure a current, you need to break the circuit and re-route it to the multimeter. This is not possible in finished I.C. boards and is the main reason why technicians/engineers mostly measure voltages and calculate I using V = IR.

3. **Measuring Resistances:** In order to measure a resistance, the multimeter imposes a small voltage (mV level) on the resistor and measures the resulting current. The resistance is then R = V/I. Two wire (2w) resistance measurements are easy: Connect the resistor as shown below and press the [2W] button.
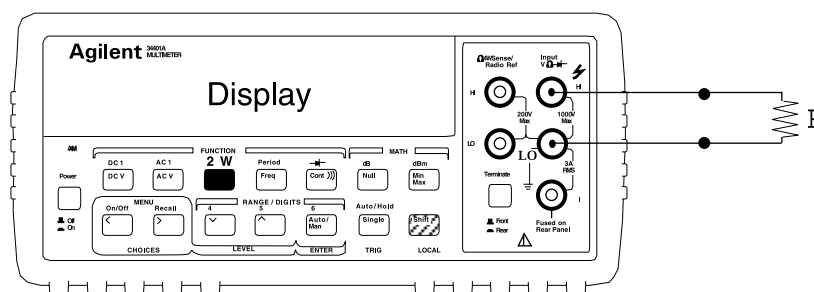
4. **Measuring AC Signals:** The multimeter can also measure the RMS values of AC waveforms (voltage or current, depending on the connection). The AC waveform can be sinewave, square-wave, triangular ... anything, since the multimeter only measures its RMS value (root mean square). The multimeter will also determine its frequency ([Freq]) and its period ([Shift], [Period Freq]).

The RMS voltages measurement is accurate up to 300 KHz, and the RMS current measurement is accurate to 5 KHz. The frequency and period are accurate up to 300 KHz but can go to 1 MHz with Vppk > 600 mV.

5. **Measuring Continuity** $\boxed{\text{Cont })))}$: Many times, it is necessary to check if a node is connected to another node (via a short-circuit). For example, you may want to follow a ground node in a complicated circuit. You can connect the two nodes to the (V, Ω) and (LO) inputs (just like voltage or resistance measurements) and the multimeter will beep if there is a short circuit.

6. **Setting the Resolution:** The resolution keys are set using the Range/Digit area. You can select 4, 5, 6 resolution digits, or simply Auto (for automatic selection).

7. **Single/Auto/Hold Measurements:** In noisy or time-varying measurements, the display is constantly changing and it may be helpful to take a single measurement or hold a measurement until it is deleted. This key does the job. Use the Auto setting to set the multimeter back into its normal operation.

The Front Panel at a Glance
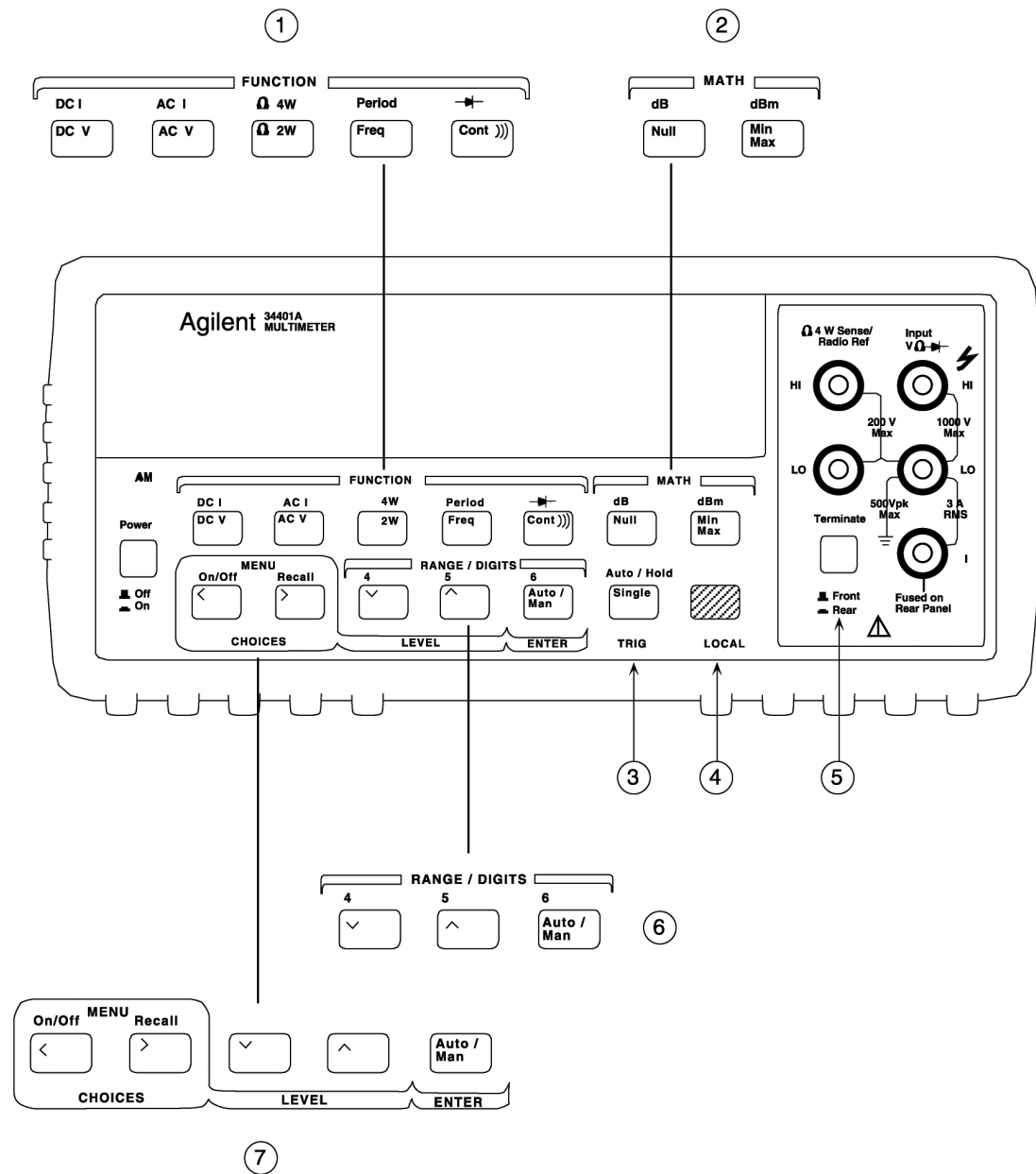
1 Measurement Function keys
2 Math Operation keys
3 Single Trigger / Autotrigger / Reading Hold key
4 Shift / Local key
5 Front / Rear Input Terminal Switch
6 Range / Number of Digits Displayed keys
7 Menu Operation keys

The Agilent 54645A Oscilloscope

1.0   Time Domain Measurements:

Equipment: Agilent 33120A Waveform Generator
                        Agilent 54645A Oscilloscope

1. Connect the output of the Agilent 33120A waveform generator to the Agilent scope using a coaxial cable.

2. Set the waveform generator to deliver a sinewave at 1 KHz and 2 Vppk. Also, set the offset voltage to be zero (see p. 4).

3. If you do not see a waveform on the scope, press the [Auto-scale] key. This key resets a large portion of the scope settings and displays the waveform. (Do not use this key often. You can develop a very bad habit and never learn how to use well a scope!)

4. Turn the VERTICAL Volts/Div knob of Channel 1 and see how you can expand or compress the waveform depending on your selection. You will find that you can easily "saturate" a scope and you should never do this. The waveform must always be within the display area.

    Turn the Position knob to see how you can move the center of the waveform up and down. Now, center the waveform and choose a 500 mV/div setting.

5. Turn the HORIZONTAL Time/Div knob and see how you can expand or compress the waveform depending on your timebase selection. If you choose a 200 µs/div setting, the 1 KHz (/msec period) sinewave "looks" expanded. If you choose a 2 msec/div setting, the 1 KHz sinewave "looks" compressed. Choose a 500 µs/div setting.

    Turn the Delay knob to the right to see how you can delay the triggering time. Look at the dark small arrows (on top and bottom of the screen). These define the actual trigger point. Return the delay back to 0.00 s (align the arrows).

6. Trigger Selection: The scope needs a signal to trigger its sampling circuitry. The triggering signal could be derived from the signal itself or from external or internal references.

    a. Press the [Source] key under the TRIGGER section. You will find on the bottom of the screen:

   Ch.1   Ch. 2   External   Line

        Press Ch. 2: Since you have no input to Channel 2, you will lose your lock and the signal will not be stable on the screen.

        Press Line: The scope is now triggering on the 60 Hz AC line voltage. Since it is not locked to the 1 KHz waveform, the signal will not be stable on the screen.

        Press External: The scope triggers on an external signal provided by the external trigger input. Since you have no input, the signal will not be stable on the screen. You can connect the SYNC output of the Agilent 33120A waveform generator to the External Trigger input. Your signal will then be stable on the screen. Do it if you wish.

Press <u>Ch. 1</u>: The scope is triggering back on the input signal and the signal is stable on the screen. (Remove the external trigger if you have connected it.)

7. <u>Slope/Glitch Triggering</u>: A scope can trigger on the rising edge or falling edge of a waveform. Look at the left side of the screen. You will find two small dark arrows, on top and bottom of the screen. These arrows define the triggering plane.

 a. Press the [Slope Glitch] key and choose the rising edge key (see the screen). Look at the waveform at the "arrow" reference plane. Note that you are triggering at the rising slope of the sinewave.

 b. Choose the falling edge and note that you are triggering at the falling edge of the sinewave.

 c. Return to rising edge trigger mode (we do not use TV or Glitch triggering).

8. <u>Mode/Coupling Triggering</u>: A scope needs a certain voltage level to trigger. Normally, this is set automatically, but in certain cases, you want to control this level so that you do not trigger on low level signals or noise.

 a. Press the [Mode Coupling] key and look at the bottom of the screen. The <u>Auto Level</u> is highlighted.

 b. Press the <u>Auto</u> option on the screen, and turn the <u>Level</u> knob under the TRIGGER section. Look at the waveform. As the triggering level is raised (or lowered), the scope triggers a bit late (or earlier) so as to align the set level with the triggering reference plane. When the level is above the waveform peak, the scope does not trigger anymore and you lose lock.

 c. Press the <u>Normal</u> option on the screen and repeat. When the level is above the waveform peak, you loose the trigger and the waveform freezes. The scope is not running anymore.

 d. Press back the <u>Auto Level</u> option.

 e. The <u>Coupling</u> key, AC or DC, means that the signal is either AC or DC coupled. If it is AC coupled, then the scope will not show the DC offset of the signal.

 f. The <u>Reject</u> key introduces low-pass filter with a corner freq. of 50 KHz to reject all noise above 50 KHz, or a high-pass filter with a corner freq of 50 KHz to reject all noise below 50 KHz. We rarely use this key.

9. <u>Voltage Measurements</u>: Look at the top of the scope under the <u>Measure</u> section and press the [Voltage] key. Now look at the bottom of the screen.

 a. Make sure that you are on <u>Source</u> 1 (for Channel 1).

 b. ❏ Press (measure) <u>Vpp</u>, <u>Vavg</u>, and <u>Vrms</u> and write these values in your notebook.

 c. Press <u>Clear Meas</u> and then <u>Next Menu</u>.

 d. Press <u>Vmax</u>, <u>Vmin</u>, <u>Vtop</u>, <u>Vbase</u>. Do not write them in your notebook.

10. <u>Time Measurements</u>: Press now the [Time] key and look at the bottom of the screen.

a. Make sure that you are on Source 1 (for Channel 1 signals).

b. ❏ Press Freq, Period, Duty and write these values on your notebook.

c. Press Clear Meas and then Next Menu.

d. Press +Width (positive part of waveform), –Width, Risetime (defined at 10% to 90% of the waveform).

11. Cursor Measurements: Press the ⌷Cursors key and look at the bottom of the screen.

a. You have two cursors, V1 and V2 and you can read any voltage on these cursors. Also, you can read the difference between V1 and V2 (ΔV). Read Vpeak and Vppk using the cursors.

b. Same for time measurements with t1, t2 and Δt.

Note that if you want to read the Degrees (or phase of a signal), you need to calibrate first. Put a cursor at a zero crossing (t1) and put the other cursor one period away Δt = 1 ms). Press the Deg key and the Set 360° key. You have now calibrated the phase.

2.0   Square-Wave Risetime Measurement:

1. Set the Agilent 33120A to give 10 KHz square-wave with 2 Vppk.

2. On the Scope, choose a 500 mV/div setting for Vertical, and a 20.0 μs/div for Horizontal.

3. ❏ Go to the Measure section, press the ⌷Time key, then go to the Next Menu and press the Risetime key. Write it on your lab notebook.

4. Choose now a 50 ns/div timebase. Notice how the waveform is still triggered underneath the arrows and you only see the rising edge of the square wave. Draw the waveform in your lab notebook. (You will see some oscillations called "ringing" and you will study this is EECS 211.)

5. ❏ Measure the risetime and write it in your lab notebook.

3.0   Frequency Domain Measurements:

1. Set the Agilent 33120A to give a 10 KHz sinewave with $V_{ppk} = 2V$. Connect it to Channel 1 of the scope.

2. Choose the vertical settings at 0.5 V/div and a horizontal setting (timebase) at 500 μs/div. You will see a lot of sinewaves on the screen.

Entering Math Mode:

3. Press the Math Mode ⌷± button.

4. At the bottom of the screen, you will see Function 1 and Function 2.

Function 1 does addition (+), subtraction (–) or multiplication (*) on signals of Channels 1 & 2.

Function 2 does integration (∫dt), differentiation (dv/dt) or a Fast Fourier Transform (FFT) on signals of Channels 1 or 2, or of the resulting waveform of Function 1 (F1).

5. Press Function 2: ON and Menu

    Operand: 1, 2, F1 (Choose the operand to be Channel 1)

    Operation: FFT, ∫dt, dv/dt (Choose FFT).

 | | | |
    |---|---|---|
    | Units/div | 10 dB | (Sets the units in dB for the vertical lines) |
    | Ref Level | 0.00 dBV | (Sets the top horizontal line in dBV) |
    | FFT Menu | | (Goes into the FFT menu) |

6. Press FFT Menu, you will see at the bottom of the screen:

Center Freq	48.8 KHz	(Shows the center freq. Controlled by timebase (horizontal) settings)
Freq Scan	97.7 KHz	(Shows the freq. scan. Controlled by timebase (horizontal) settings).
Window	Hanning	(Can be selected between Hanning, flat-top, Exponential and Rectangular. Always choose Hanning.)
Autoscale FFT		(Moves 0 Hz to left and autoscales the vertical settings. I do not like it.)

7. Press Channel 1 button ⬜ 1 twice to get rid of the time domain representation.

    > (To view Channel 1 in the time domain, press ⬜ 1 at any time.)

8. Press Math Mode button ⬜ ± to get back into the FFT menu at the bottom of the screen.

    You have in front of you a clear representation of a sinewave signal in frequency domain. Notice the single peak at 10 KHz (the rest is sampling noise).

9. ❑   Press ⬜ Cursors. Using V_1, V_2 and f_1, f_2, measure the amplitude of the sinewave (in dBV) at 10 KHz and the average amplitude of the noise. Write them in your lab notebook.

10a. ❑   Change the amplitude of the sinewave to 100 mVppk and using the Cursors mode, measure the voltage in dBV. Write it down on your notebook.

     Return the amplitude back to 2 Vppk.

10b. ❑   Using the knob on the function generator, vary the frequency in 1 KHz steps to 100 KHz and see the peak moving on the screen. Notice what happens above 100 KHz. The signal returns to the screen and moves backward!! This is not correct and is due to the sampling circuity/firmwave of the scope. THEREFORE, ALWAYS BE SURE THAT YOUR FREQUENCY SPAN (AT THE TOP OF THE SCREEN) is LARGER THAN YOUR SIGNAL!

     Return the sinewave frequency to 10 KHz.

11. ❑ Now comes the interesting part: Press the <u>Square-Wave</u> button (10 KHz, Vppk = 2V) and measure the amplitude of each harmonic ($3f_O$, $5f_O$, $7f_O$ and $9f_O$) and write them in your lab notebook.

12. Press back the sinewave button. Now play with the timebase settings and see how you change the center frequency and frequency span of the display. Always choose a setting which puts your fundamental and harmonics frequency within the set frequency span!

> To view Channel 1 in the time domain, press [1] at any time.
>
> To view Channel 1 in the frequency domain, press [±] at any time.

Agilent 54645A Oscilloscope Front Panel