New York Institute of Technology
Laboratory Manual I
EENG-275

Lab Report Format

The lab report must be word processed. Circuit diagrams and graphs can be pasted in from PSPICE, Micro-cap, Electronics Workbench, Mathcad, etc. Lab report must contain:

1. A cover page containing the following information:
    a. Course Number:
    b. Lab Title:
    c. Student's Name:
    d. Lab Partner(s):
    d. Instructor Name:
    e. Date of Submission:

2. A statement of the objectives **"DO NOT COPY FROM THE MANUAL"**. List only the objectives that are addressed in the actual lab.

3. List all equipment used in the experiment:
    a. List the workstation number where you were sitting at.
    b. List model and series number of the equipment used at the workstation.
    c. List all components used in the experiment.
    d. Measure the component values accurately as possible.

4. Explain in your result in full clear detail.

5. Experimental, theoretical and computer simulation results should be placed tabulated next to each other.

6. In your conclusion, discuss your results. Did you verify your calculations? If your results are outside the measuring accuracy of your equipment try to explain any discrepancies. Also, did you satisfy the objective?

7. Answer all the questions in the lab experiment.

8. All equations, circuits, graphs, tables and diagrams must be labeled appropriately (e.g. Figure 1. Voltage Gain A_V). Also, circuit diagrams with critical points must be appropriately labeled.

EENG-275

Experiment # 0

Properly Using Lab Equipment

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Read and use the resistor color code.
2. Use the digital multi-meter as an Ohmmeter, Voltmeter, and Current meter.
3. Set and adjust a DC power supply
4. Become familiar with the use of a breadboard.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-Meter (DMM)
1- 6.8 Ω Resistor
1- 100 Ω Resistor
1- 2.2 kΩ Resistor
1- 33 kΩ Resistor
1- 270 kΩ Resistor
1- 1MΩ Resistor
1- 1 kΩ Potentiometer

III: Breadboard Layout

A solder-less breadboard is the most common type of prototyping circuit board. Prototyping a circuit is the process of creating a model suitable for complete evaluation of its design and performance. This requires the circuit to be designed, built and tested in the laboratory. Theoretical calculations and computer simulation are part of the design process. Once the circuit configuration is determined, the circuit is built on a prototyping board. There are two main types of prototyping circuit boards:

- Solder-less Breadboards
- Perfboard

A circuit built on a breadboard requires neither soldering nor wire wrapping the connections.

Examine the breadboard in your kit. This board will be used throughout the semester. The breadboard has two terminal strips, four bus strips, and three binding posts as shown in Figure 1. Each bus strip has two rows of contacts. Each row is a common point, or node. Bus strips must be used as power and ground.

Each terminal strip has 2 sets of 5 rows. Each column of 5 contacts is a common tie point. Build circuits on the terminal strips by inserting the leads of circuit components into the contact receptacles and making connections with 22 AWG (American Wire Gauge) wire. Larger gauge wire will damage the board. Use the red and black binding posts for power supply connections.

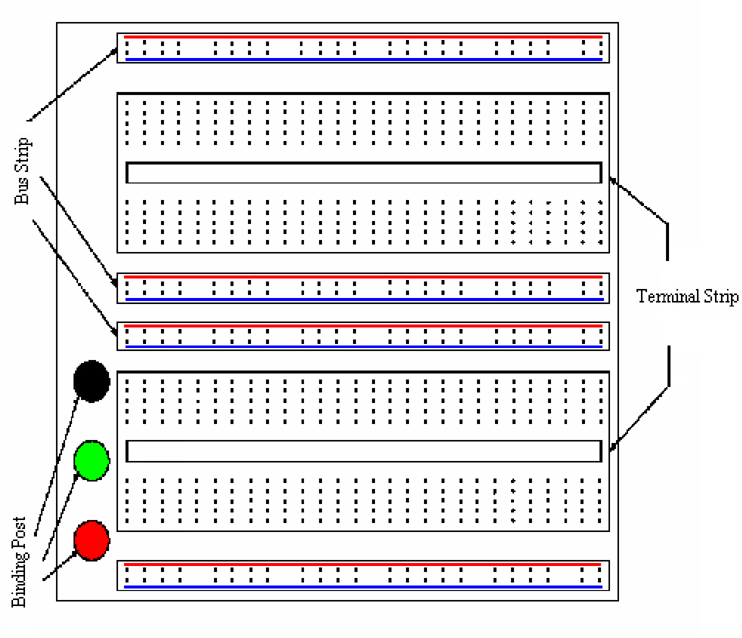


Figure 1: Typical Solder-less Breadboard

IV: Resistance Color Code

Choose any of the resistors and examine it closely with your partner. Notice that there are four color bands. Rotate the resistor so that last band (the color may be gold or silver) is on your right side. Using Figures 2 and 3 associate the resistor's color code with a value in Table 1. For that row in the table, determine the maximum and minimum resistor values, based on the tolerance color code.

Color	1st band	2nd band	Multiplier	Tolerance
Black	0	0	1	
Brown	1	1	10	±1%
Red	2	2	100	±2%
Orange	3	3	1k	
Yellow	4	4	10k	
Green	5	5	100k	±0.5%
Blue	6	6	1M	±0.25%
Violet	7	7	10M	±0.10%
Grey	8	8		±0.05%
White	9	9		
Gold			0.1	±5%
Silver			0.01	±10%

Figure 2: Resistor Color Code

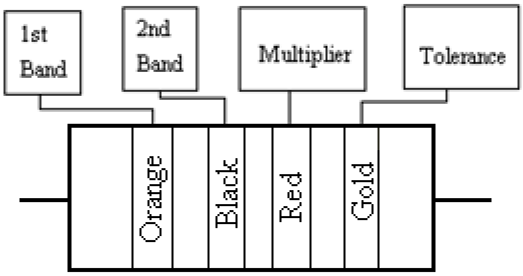


Figure 3: Reading the Color Code

If you put the first and second band together for this resistor, you will get a 3 and a 0.

$$30*(100) = 3000\Omega = 3\ k\Omega \pm 5\%. \qquad (1\text{-}1)$$

Percentage tolerance is used to determine the range of resistance level. This is calculated by taking the tolerance band value and multiplying it by the nominal value.

$$(5\%)*(3\ k\Omega) = 0.15\ k\Omega \qquad (1\text{-}2)$$

The maximum value of the resistance is the sum of the tolerance value and the nominal value.

$$\text{Maximum Value} = 3\ k\Omega + 0.15\ k\Omega = 3.15\ k\Omega \qquad (1\text{-}3)$$

The minimum value of the resistance is the subtraction of the tolerance value and the nominal value.

$$\text{Minimum Value} = 3\ k\Omega - 0.15\ k\Omega = 2.85\ k\Omega \qquad (1\text{-}4)$$

Resistor	Color Bands – Color	Minimum Value	Maximum Value
6.8 Ω			
100 Ω			
2.2 kΩ			
33 kΩ			
270 kΩ			
1 MΩ			

Table 1 Associating a resistor's color code with a value and finding a range

V: Using the Digital Multi-Meter as an Ohmmeter

Fixed resistor

To measure the resistance of a resistor, use a Digital Multi-Meter (DMM).
Connect the resistor into the bread broad
Connect the Black Probe to the COM terminal and Red Probe to the terminal marked with "Ω." Set the meter to "Ω."

Figure 4: Bottom Half Of DMM.

Connect the Black Probe to one end of the resistor now connect the Red Probe to the other end of the resistor.
Set the meter to “Ω” function.

Figure 5: Top Half of DMM.

Now that the meter has given a reading, compare the difference between the measured resistance and the nominal resistance. The percentage difference is calculated by:

$$\text{Percentage\_Error} := \frac{\text{Nominal\_Value} - \text{Measured\_Value}}{\text{Nominal\_Value}} \cdot 100 \qquad (1\text{-}5)$$

Complete Table 2.

Nominal Value	Ohmmeter Reading	Percentage Difference
6.8 Ω		
100 Ω		
2.2 kΩ		
33 kΩ		
270 kΩ		
1 MΩ		

Table 2 Comparing measured resistance with nominal value

Variable resistor

A potentiometer (variable resistor) is a three terminal device that is used primarily to control potential voltage levels.

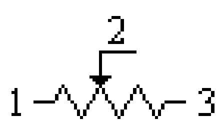


Figure 6: Potentiometer schematic

To determine the maximum value of a potentiometer connect the Red Probe of the ohmmeter to either of outer terminal leads on the potentiometer (terminal 1 or terminal 3 in Figure 6). Now connect the Black Probe to the other remaining outer terminal (terminal 1 or terminal 3 in Figure 6).

Turn the control knob as far as it will go in the clockwise direction and record your answer in the Table 3 below. Now turn the control arm in the counterclockwise direction as far as it will go and record your answer. Now turn the control arm to any position between the two extremes and record the resulting resistances.

Clockwise	Counterclockwise	Any Position

Table 3: Total Resistance of a Potentiometer

To have a desired resistance value, connect the Red Probe of the ohmmeter to either of outer terminal leads on the potentiometer (terminal 1 or terminal 3 in Figure 6). Now connect the Black Probe to the inner terminal also known as the wiper, terminal 2 in Figure 6.

Turn the control knob as far as it will go in the clockwise direction and record your answer in the Table 4 below. Now turn the control arm in the counterclockwise direction as far as it will go and record your answer. Now turn the control arm to any position between the two extremes and record the resulting resistances.

Clockwise	Counterclockwise	Any Position

Table 4: Resistance Between A Wiper Terminal And An Outer Terminal of A Potentiometer

VI: Using the Digital Multi-Meter for DC Voltage Measurements

A voltmeter is a device for measuring voltage. **The voltmeter is placed in parallel with the circuit element whose voltage is to be measured**. Recall that two elements are in parallel when they share the same pair of nodes and hence share the same voltage.

Locate the voltage adjustment knob, current adjustment knob, and the power switch on the power supply. Before you turn on the power supply rotate both the current and voltage adjustment knobs to the left. Turn on the power supply. Note that both readings of current and voltage are at zero. Now rotate the current knob about half way. Now set the voltage adjustment knob for a reading of 0.5 V.

Take the DMM and connect the Black Probe to the COM terminal on the meter and Red Probe to the terminal marked with "V" on the meter. Set the meter to V---- function. The Red Probe is connected to the positive (+) voltage terminal of the power supply and the Black Probe to the negative (-) voltage terminal of the power supply.

Set the power supply to the following voltages as stated in the Table 5 below and record your data.

Power Supply Voltage	DMM Voltage Reading
0.5 V	
2.0 V	
5.0 V	
15 V	
20 V	

Table 5: Voltage Measurements With A DMM.

VII: Using the Digital Multi-Meter for Making Current Measurements

Construct the circuit shown in Figure 7, **BUT DO NOT CONNECT THE SOURCE**. Set the DMM selector switch to A--. Connect the Black Probe to the COM terminal of the DMM and the Red Probe to the DMM terminal marked 300mA.

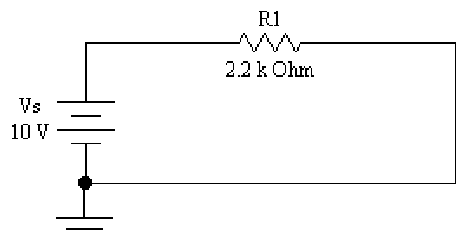


Figure 7: Circuit Where Current Is To Be Measured.

Insert the meter as shown in Figure 8.

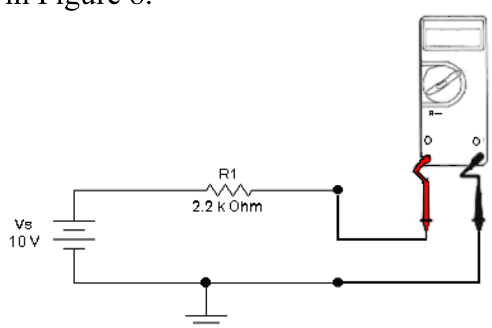


Figure 8: Measure Current Using A DMM.

With your **INSTRUCTORS APPROVIAL** connect the source and read the current value.

Note: If you attempt to read current "ACROSS" any component (as in reading voltage), you will damage the equipment, and cause a dangerous situation! If you're not sure on how to measure current, please call over your instructor.

EENG-275
Experiment # 1
Series Circuit and Parallel Circuits

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Verify equations for computing equivalent resistance.
2. Verify Kirchhoff's Voltage Law.
3. Verify Kirchhoff's Current Law.

II. Material and Equipment

1 - NYIT supplied Lab Kit
1 - Digital Multi-Meter (DMM)
1- 220 Ω Resistor
1- 330 Ω Resistor
1- 470 Ω Resistor
1- 1 kΩ Resistor
1-2.2 kΩ Resistor
1- 2.7 kΩ Resistor
1- 4.7 kΩ Resistor

III. Preparation

1. Calculate I_T, R_T, V_1, V_2, V_3, and V_4 in Figure 1, assuming V_S =12V, R_1= 220 Ω, R_2= 330 Ω, R_3= 470 Ω, and R_4= 1 kΩ. Redraw the circuit showing assigned voltage polarities and assigned current directions. Show all computer simulation for Figure 1.1. Prepare a table that will contain calculated and measured values.
2. Show that all the voltages found in Figure 1.1 satisfy KVL?

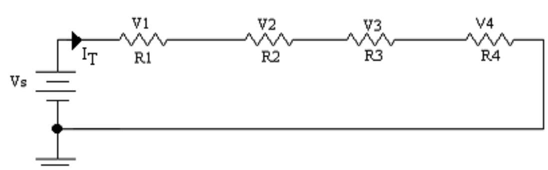


Figure 1.1: Series Circuit for calculations

3. Calculate I_T, I_1, I_2, I_3, and R_T in Figure 1.2, assuming V_S = 12 V, R_1= 1 kΩ, R_2= 2.7 kΩ, R_3= 4.7 kΩ. Redraw the circuit showing assigned voltage polarities and assigned current directions. Show all computer simulation for Figure 1.2. Prepare a table that will contain calculated and measured values.
4. Show that all the currents found in the Figure 1.2 satisfy KCL?

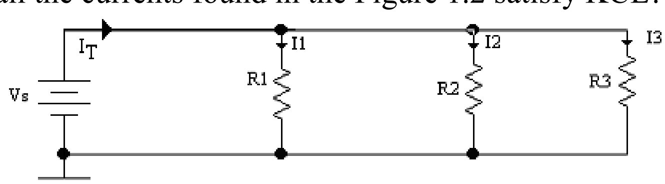


Figure 1.2: Parallel Circuit for calculations

IV. Laboratory Procedure

1. Construct the circuit shown in Figure 1.1. Do not connect the source. Measure R_T with the DMM.
2. Connect the source as shown in Figure 1.1. Measure and tabulate V_1, V_2, V_3, V_4, V_T and I_T.
3. Construct the circuit shown in Figure 1.2. Do not connect the source. Measure R_T with the DMM.
4. Connect the source as shown in Figure 1.2. Measure and tabulate I_1, I_2, I_3, and I_T.

VII. Questions

1. How is a voltmeter connected to measure an unknown voltage?
2. Calculate the percentage differences between the values obtained in the preparation and the actual measured values.
3. Compare experimental and calculated values.
4. Demonstrate the satisfaction of KVL and KCL.
5. When using a DMM to do a current measurement, the implicit assumption is that the meter resistance is 0. What will be the effect on the measurement if the resistance of the meter is not 0.

EEENG-275
Experiment # 2
Series- Parallel Circuits

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Verify voltage divider rule.
2. Verify current divider rule.
3. Analyze series-parallel circuits.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- 1.5 kΩ Resistor
1- 2 kΩ Resistor
1- 2.2 kΩ Resistor
1- 3.3 kΩ Resistor
2- 510 Ω Resistor
2- 680 Ω Resistor
2- 1 kΩ Resistor
3- 3 kΩ Resistor

III. Preparation

1. For the circuit of Figure 2.2 calculate R_T, I_2, I_3, I_4, I_5, I_6, I_S, V_1, V_A, V_5, and V_4. With V_S= 15 V, R_1= 680 Ω, R_2= 680 Ω, R_3= 1 kΩ, R_4= 2.2 kΩ, R_5= 510 Ω, and R6= 510 Ω. Show all computer simulation.
2. For the circuit of Figure 2.3 calculate R_T, I_S, I_1, I_2, and I_3. With V_S= 10 V, R_1= 510 Ω, R_2= 1.5 kΩ, R_3= 1 kΩ, R_4= 1 kΩ, and R_5= 2.2 kΩ. Show all computer simulation.
3. In this voltage divider circuit of Figure 2.1, the parameters are:
    a. Total Input Resistance(R_1+R_2) = 10 kΩ
    b. The open circuit load gain factor $\left(\frac{V_2}{V_1}\right) = 0.2$
    c. Calculate the value of R_1, and R_2,

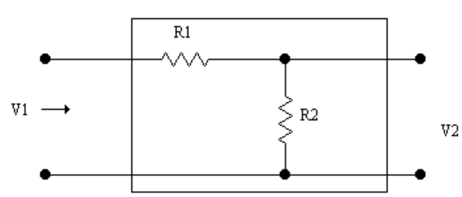


Figure 2.1

IV. Laboratory Procedure

1. Construct Figure 2.2, measure R_T, I_2, I_3, I_4, I_5, I_6, I_S, V_1, V_A, V_5, and V_4 using the DMM.

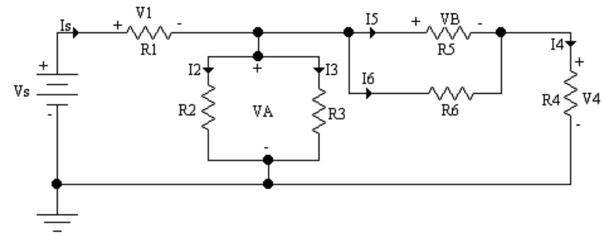


Figure 2.2

2. Construct Figure 2.3, measure R_T, I_S, I_1, I_2, and I_3 using the DMM.

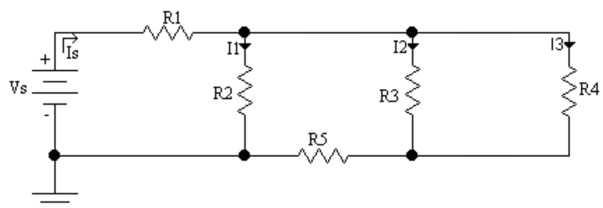


Figure 2.3

3. Build the design of your voltage divider circuit. Verify that the circuit gives the correct gain of 0.2 and input resistance of 10kΩ.

V. Questions

1. State the factors which can account for the percent differences. Consider both components and the measuring instrument
2. Suppose in Figure 2.3, R_3 burns out and makes an open. Will R_T be larger or smaller?
3. In Figure 2.3, what will I_S be if a resistor of the value R_T is placed across V_S and all other resistors remain the same?

EENG-275

Experiment # 3:

Circuit Analysis

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Apply and verify the Node Voltage method.
2. Apply and verify the Mesh Current method.
3. Apply and Verify Thevenin Theorem.
4. Apply and Verify Norton Theorem.
5. Apply and Verify Superposition Theorem.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1 -100 Ω Resistor
1- 510 Ω Resistor
1- 1 kΩ Resistor
1- 1.5 kΩ Resistor
1- 2.2 kΩ Resistor
1- 3.3 k Ω Resistor
1- 4.7 kΩ Resistor
1- 5.1 kΩ Resistor
1- 10 kΩ Resistor
1- 15 kΩ Resistor
2- 6.8 kΩ Resistor

III. Preparation

1. Calculate the value of each voltage and current of Figure 3.1 with V_{S1}= 12 V, V_{S2}= 6 V, R_1= 1 kΩ, R_2= 3.3 kΩ, R_3= 4.7 kΩ. Use the nodal voltage method and the mesh current method to prove your results. Show all computer simulation.
2. Using Thevenin equivalent of the circuit to the left of R_L in Figure 3.2, calculate R_{TH}, V_{TH} and I_L, with V_S= 15 V, R_1= 2.2 kΩ, R_2= 3.3 kΩ, R_3= 4.7 kΩ, R_4= 6.8 kΩ, R_5= 10 kΩ, and R_L= 6.8 kΩ. Show all computer simulation.
3. Using Norton's equivalent of the circuit of Figure 3.3, calculate R_N, I_N, and I_L, with V_{S1} = 12 V, V_{S2} = 10 V, R_1= 2.2 kΩ, R_2= 3.3 kΩ, R_3= 4.7 kΩ, R_4 = 6.8 kΩ, R_5= 10 kΩ, and R_L= 15 kΩ. Show all computer simulation.
4. In the circuit of Figure 3.4, find the response of I_{R1}, I_{R2}, I_{R3}, I_{R4}, V_{R1}, V_{R2}, V_{R3} and V_{R4} using superposition; with V_{S1}= 5 V, V_{S2}= 10 V, R_1= 1 kΩ, R_2= 1.5 kΩ, R_3= 5.1 kΩ, R_4= 510 Ω. Show all computer simulation.

IV. Laboratory Procedure

1. Construct the circuit of Figure 3.1, measure the value of each current and voltage using the DMM.

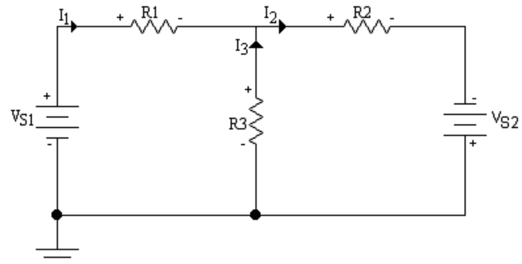


Figure 3.1

2. Construct the circuit of Figure 3.2, measure R_{TH}, V_{TH}, I_L.

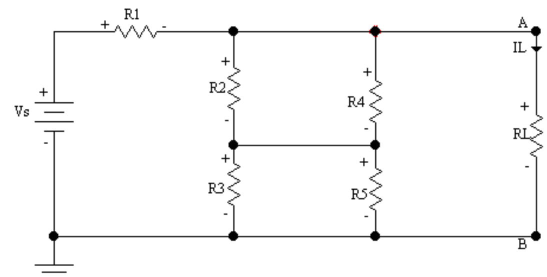


Figure 3.2

3. Construct the circuit of Figure 3.3, measure R_N, I_N, I_L using the DMM.

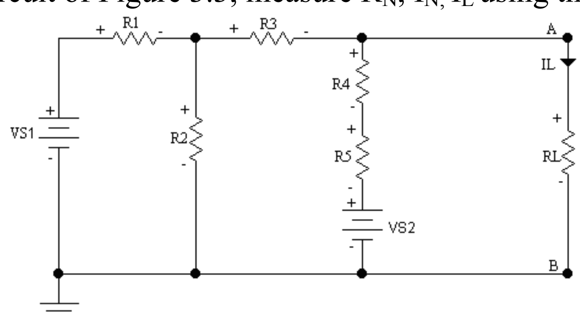


Figure 3.3

4. Construct the circuit of Figure 3.4, measure I_1, I_2, I_3, I_4, V_1, V_2, V_3 and V_4 using the DMM.

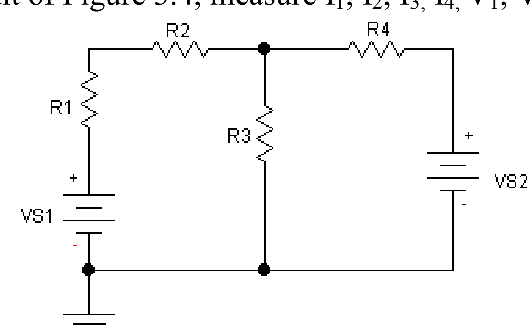


Figure 3.4

V. Questions

1. How does the measured value compare with the theoretical equivalent resistance you found?
2. More than likely, the two values don't agree. What other sources of resistance might be present in your circuit? How might you test for them?
3. Are the two circuits really equivalent? If the circuits were both enclosed in a "black box", would you be able to tell them apart based on your measurements?

EENG-275
Experiment # 4:
Function Generator, Oscilloscope

I. Objectives
Upon completion of this experiment, the student should be able to

1. To become familiar with the use of a function generator.
2. To become familiar with the use of a digital stored oscilloscope.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Function Generator
1- Oscilloscope
1- 51 Ω Resistor
1- 1.5 kΩ Resistor
1- 2 kΩ Resistor
1- 5.1 kΩ Resistor
1- 0.1 μF Capacitor
1- 4.7 uH Inductor

III. Protek 9301 Function Generator
The Protek function generator is a precise low distortion instrument, capable of generating sine, triangular, square, and ramp signals in the 0.1Hz to 32 MHz frequency range. The front panel and the main features are shown in Figure 4.1 and Table 4-1.

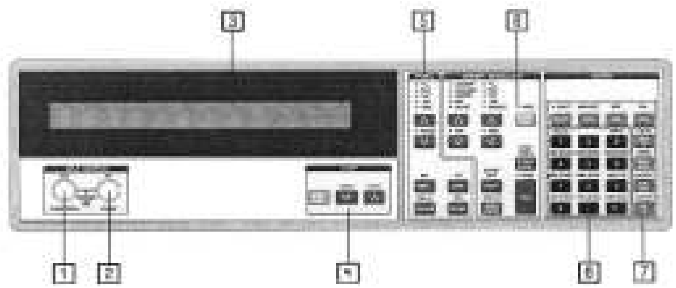


Figure 4.1: Front panel of Protek function generator.

Table 4-1 some features of the Protek function generator.

No.	Name	Feature Description
1	CH1 Function Output	50 Ω square, triangle, sine wave output.
2	CH1 Sync Output	Square wave output for driving digital logic.
3	Parameter Display	Display the signal parameter values set for each channel.
4	Step	The step key permits the operator to increase or decrease the selected parameter's step value.
5	Function Keys	These keys choose the main waveform output. The Function [Up/Down] arrow keys select the output waveform.
6	Entry Keys	The numeric keypad allows for direct entry. The value is entered by pressing one of the Unit keys. A typing error may be corrected by using the CLR key.
7	Units Keys	The Units keys are used to enter numeric values. Simply

		press the key with the desired units to enter the value
8	Sweep/Modulation	These keys control the modulation and sweep capabilities.

When the function generator is powered on the display parameter values that appear are:

Display Description

- OUT FREQUENCY: indicates the frequency selected.
- AMPL: indicates the output voltage amplitude and units.
- OFFSET: indicates the dc offset output voltage.

Operating Procedure

Select the desired frequency by pressing the [**FREQ**] key and typing the value on the keypad. Complete the entry by pressing the appropriate units (**Hz**, **kHz**, or **MHz**). The output frequency is displayed, on the LCD display. Select the output (**FUNC** menu) waveform (square, triangle, or sine) by pressing the **FUNC** Up/Down arrow keys until the desired function LED is lit. The **SWEEP/MODULATE** should be on **LIN/SINGLE**. The amplitude signal level can be set by pressing the [**AMPL**] key and typing in the value on the keypad. Complete the entry by pressing the appropriate units (Vpp, Vrms, or dB).

IV. Tektronix TDS 2024 Oscilloscope

The Tektronix TDS 2024 oscilloscope provides accurate real-time acquisition of signals. It supports features such as automatic measurement, peak detect, storage of four reference waveforms, and autoset. The front panel and the main features of the TDS2024 are shown in Figure 4.2.

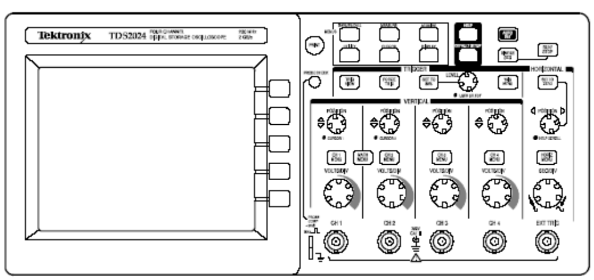


Figure 4.2: Front Panel of TDS 2024.

The **VERTICAL** Control relates to the vertical movement of the scope trace. This oscilloscope has four vertical sections so that it can be display four waveforms simultaneously.

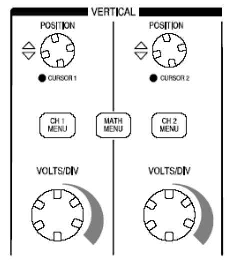


Figure 4.3: **VERTICAL** Control

Table 4-2 **VERTICAL** Control Description.

CH1, CH2, CH3 & CH4 (Menu)	Display the channel input menu selections.
CH1, CH2, CH3 & CH4 (Volts/Div)	Selects calibrated scale factors.
CH1, CH2, CH3 & CH4 (Position)	Positions the waveform vertically.

The **HORIZONTAL** Control relates to the horizontal movement of the scope trace.

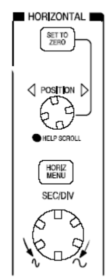


Figure 4.4: **HORIZONTAL** Control

Table 4-3: **HORIZONTAL** Control Description

POSITION	Adjusts the horizontal position for all channels and math waveforms.
SEC/DIV	Selects the horizontal time/div (scale factor) for the waveform.

The control buttons provide features to be used for all channels as listed in Table 4-4.

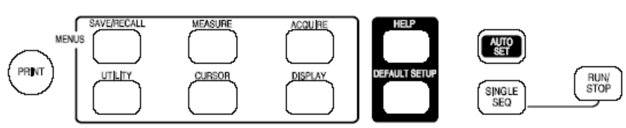


Figure 4.5: Control Buttons

Table 4-4: Control Button Description

SAVE/RECALL	Displays the SAVE/RECALL menu for setups and waveform.
MEASURE	Displays the automated measurements menu.
ACQUIRE	Displays the acquisition menu.
UTILITY	Displays the UTILITY menu.
CURSOR	Displays the CURSOR menu.
DISPLAY	Displays the DISPLAY menu.
AUTOSET	Automatically sets the instruments control to produce a usable display of the input signal.
RUN/STOP	Starts and stops waveform acquisition.

The connectors provide the input for the signal display as shown in Figure 4.6.

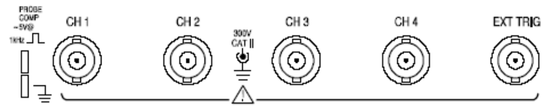


Figure 4.6: Input Connector for waveforms.

In addition to displaying waveforms, the display is filled with many details about the waveform and the oscilloscope control settings.

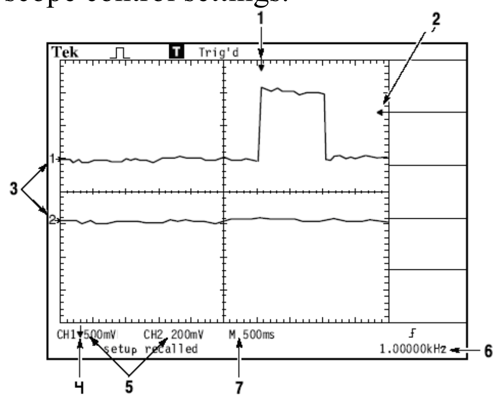


Figure 4.7: Display of Two Waveforms

Table 4-5: Display Description of Oscilloscope

No.	Name	Feature Description
1	Horizontal Trigger Maker.	Marker shows horizontal trigger position.
2	Width Trigger Marker.	Marker shows Edge or Pulse Width trigger level.
3	Channel Indicator	On-screen markers show the ground reference points of the Displayed waveforms.
4	Waveform Indicator	An arrow icon indicates that the waveform is inverted.
5	Vertical Scale	Selected vertical scale factors (Volts/Div).
6	Trigger Frequency.	Readout shows trigger frequency.
7	Time Scale	Readout shows main time base setting.

V. Probe Setting and Connections

Probes are available with various attenuation factors which affect the vertical scale of the signal. The default setting for the Probe is 10X. Be sure that the Attenuation switch on the P2200 probe matches the Probe option in the oscilloscope. Switch settings are 1X and 10X.

Attenuation switch

Note: When the Attenuation switch is set to 1X, the P2200 probe limits the bandwidth of the oscilloscope to 7 MHz. To use the full 200 MHz bandwidth of the oscilloscope, be sure to set the switch to 10X.

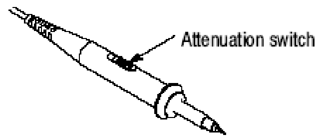


Figure 4.8: Probe Lead Setting

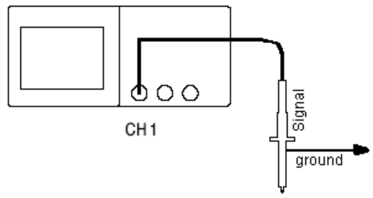


Figure 4.9: Connection of Probe Lead Too Oscilloscope

Figures 4.9 and 4.10 show how the oscilloscope is connected to a circuit.

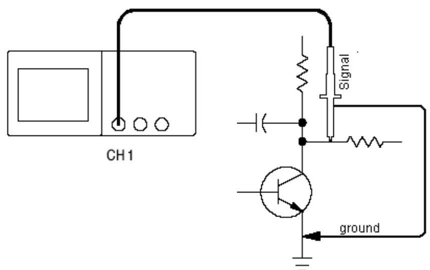


Figure 4.10: Probe Lead Connected to a Circuit

VI. Downloading the Oscilloscopes Display

On the lab computer desktop look for the Tek OpenChoice Desktop icon. Double click on it, and allow the program to load. Once the program has loaded, click on the Select Instrument button. A window will appear. Click on ASRL1::INSTR to select it and click the OK button. Notice that Get Screen icon has been activated. Double click on the Get Screen icon and allow time for the image to appear on the screen. It takes a minute for the image to appear, so be patient. Once the image has appeared, use the **SAVE AS** to save it on to your memory key.

VII. Laboratory Procedure

Display A DC Signal

1. Construct the circuit of Figure 4.11, with V_S of the power supply set to 5 V. Set the **VERTICAL** scale for Channel 1 to 2 (Volts/Div).
2. Download the image seen on the oscilloscope.
3. Explain in detail the image seen on the oscilloscope.

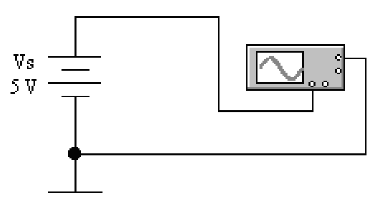


Figure 4.11

4. Construct the circuit of Figure 4.12, with V_S = 12 V, R_1= 6 kΩ, R_2= 2 kΩ. Set the **VERTICAL** scale for Channel 1 to 5 (Volts/Div). Measure V_S using Channel 1and TP1(Test Point 1) using Channel 2 of the oscilloscope.
5. Download the image seen on the oscilloscope.
6. Explain in detail the image seen on the oscilloscope.

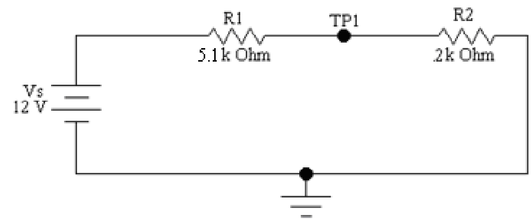


Figure 4.12

Display an AC Signal

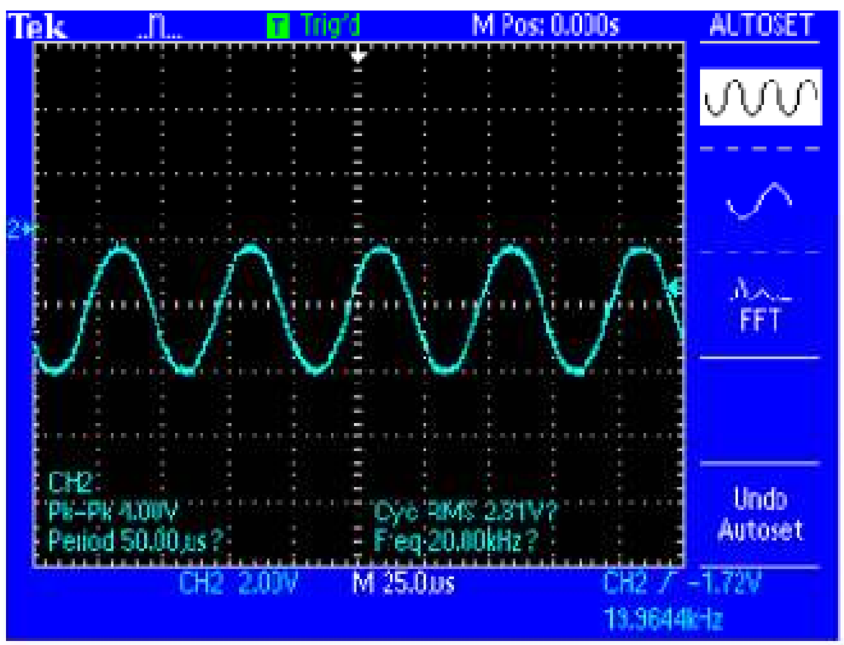


Figure 4.13: Display on scope

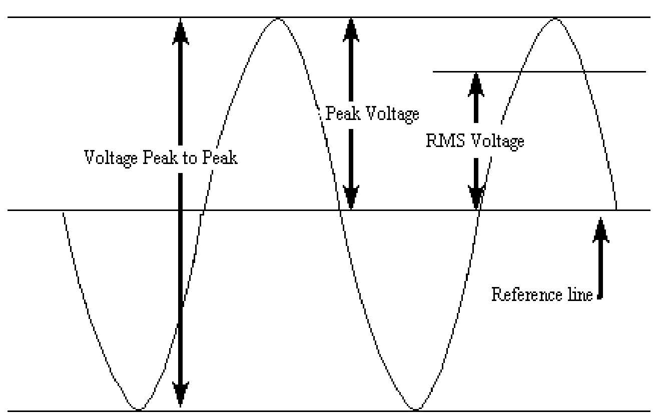


Figure 4.14: Explanation of displayed waveform

1. Using Figure 4.13 and 4.14, answer the following questions:
   a. What is the setting on the Volts/Div?
   b. What is the peak voltage?
   c. What is the voltage peak to peak?
   d. What is the time base of the sec/div?
   e. What is the frequency of the signal (f)?

Taking Cursor Measurement

You can use the cursors to quickly take time and voltage measurements on a waveform.

1. Construct the circuit of Figure 4.15, with the V_S =10 Vpp at a frequency of 20 kHz, R_1= 1.5 kΩ, C_1= 0.1 µF, and L_1 = 4.7 µH.
2. Setup the oscilloscope so that the voltage Vs can be displayed on Channel 1, and the voltage Va can be displayed on Channel 2.

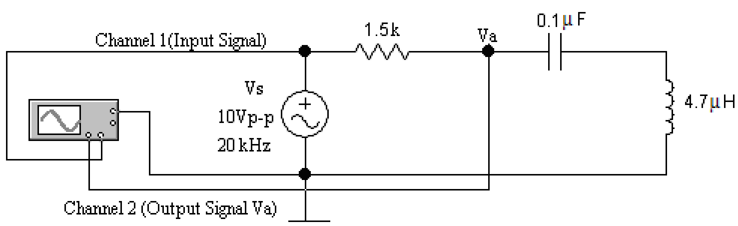


Figure 4.15

Measuring Peak-to-Peak Voltage

3. Push the **CURSOR** button on the oscilloscope to bring the cursor menu.
4. In the **CURSOR** menu push the top option button called Type until it is on the Voltage Mode. Notice that the LED light under both vertical position knobs will turn on to indicate the use of cursor 1 and cursor 2 functions.
5. Turn the cursor 1 **POSITION** knob to place a cursor on the highest peak of the signal.
6. Turn the cursor 2 **POSITION** knob to place a cursor on the lowest peak of the signal.
7. The peak-to-peak voltage is read as [Cursor 2]-[Cursor1] =Delta.
   a. What is the value of cursor 1?
   b. What is the value of cursor 2?
   c. What is the delta value?
8. Download the image seen on the oscilloscope.
9. Explain in detail the image seen on the oscilloscope.
10. Repeat the above process for Channel 2.

Measuring Frequency

1. Push the **CURSOR** button on the oscilloscope to bring up the cursor menu.
2. In the **CURSOR** menu push the top option button call Type until it is on the Time Mode.
3. Turn the cursor 1 **POSITION** knob and place the cursor on the center of the first peak of the waveform.
4. Turn the cursor 2 **POSITION** knob and place the cursor on the center of the second peak of the waveform.
5. The period can be directly read as [Cursor 2]-[Cursor1] = Period.
   1. What is the value of cursor 1?

2. What is the value of cursor 2?
3. What is the period?

6. The frequency is $\dfrac{1}{Period}$. What is the frequency?
7. Download the image seen on the oscilloscope.
8. Explain in detail the image seen on the oscilloscope.
9. Repeat the above process for Channel 2.

Automatic Measurements

It is desired to measure the voltage peak-to-peak at point V_s and V_a. as indicated in Figure 4.15.

1. Push the **MEASURE** button on the oscilloscope to bring up the measure options. There are five options to use for any of the channels to be measured.
2. Push anyone of the five option buttons.
3. Push the Source option button and select the channel you desire to be measured.
4. Push the Type option button and select the parameter to be measured, in this case Pk-Pk.
5. Push the Back option button.
6. Repeat the process to measure frequency.
7. Download one image containing information for Channel 1 and Channel 2 showing the voltage and frequency measurements of both channels.

VIII. Questions

1. Suppose the function generator is set to a 1 MHz sine-wave and the oscilloscope is set to 200nsec/div. Determine the number of squares that one period of the wave will occupy.
2. Repeat question 1 if the function generator is changed to a 1 MHz square wave.
3. What is the dc level of the waveform in Figure 4.18 below?
4. What is the frequency of the waveform in Figure 4.18 below?

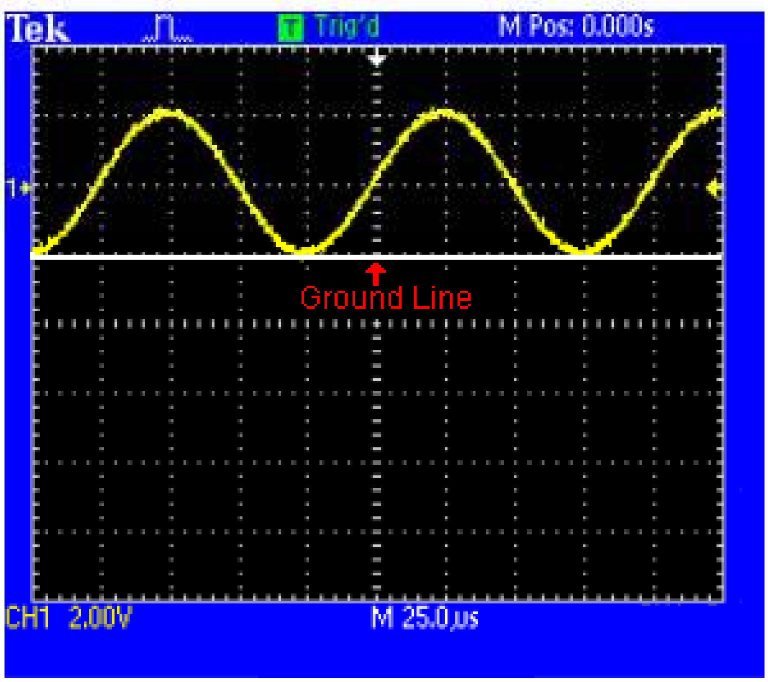


Figure 4.18

EENG-275
Experiment # 5:
Time Domain vs. Frequency Domain

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Calculate and measure the RC Time Constant.
2. Calculate and measure the RL Time Constant.
3. Calculate and measure the bandwidth of an RC and RL circuit.
4. Relate the time and frequency domains.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Function Generator
1- Oscilloscope
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- 51 Ω Resistor
1- 220 Ω Resistor,
1- 3 k Ω Resistor
1- 0.01 μF Capacitor
1- 10mH Inductor

III. Preparation

1. In Figure 5.1, let V_S be a square wave (TTL Function) from 0 to 5 Vp with a frequency of 3 kHz, R_1= 3 kΩ, C_1= 0.01 μF. Calculate τ and sketch the charging and discharging of V_C (t). Also, show a computer simulation.

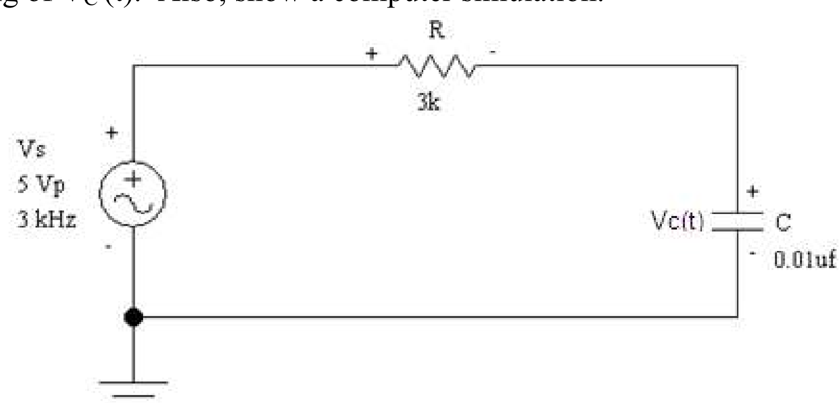


Figure 5.1

2. In Figure 5.2, let V_S be a square wave from -2.5 V to 2.5 V (5 Vpp) with a frequency of 1 kHz, R_1= 220 Ω, L_1= 10mH. Calculate τ and sketch the charging and discharging of V (t). Also, show a computer simulation.

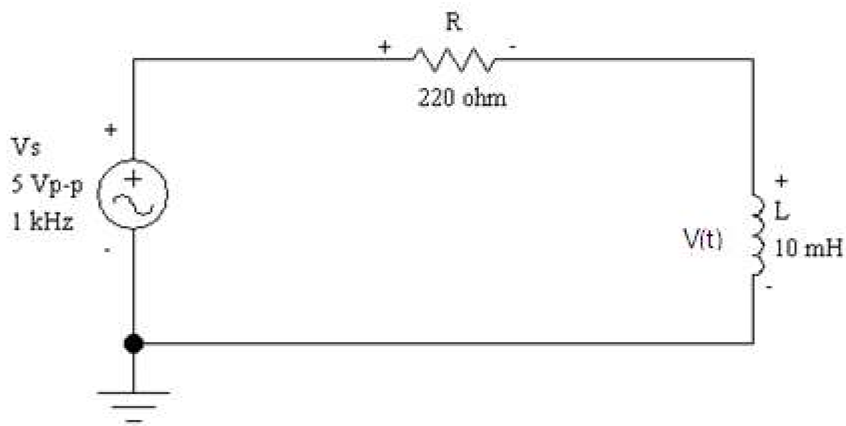


Figure 5.2

IV. Rise Time Measurements

1. Turn the sec/div knob to display the rising edge of the waveform to look like Figure 5.3.
2. Push the cursor button.
3. Push the type option button and select the time.
4. Turn the cursor 1 knob and place it at the point where the waveform crosses the second graticule line below center screen as shown in Figure 5.10. This is the 10% level of the waveform.
5. Turn the cursor 2 knob and place it at the point where the waveform crosses the second graticule line above center screen as shown in figure 5.10. This is the 90% level of the waveform.
6. The delta readout in the Cursor Menu is the rise time of the waveform.

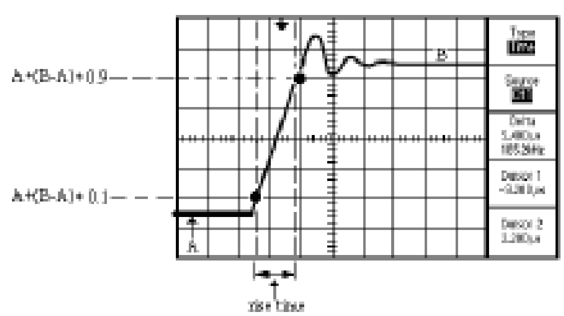


Figure 5.3

Taking a rise time measurement can be done automatically but you must expand the time base enough to make a proper reading.

V. Laboratory Procedure

1. Construct the circuit shown in Figure 5.1. Set up the function generator to produce a square wave (TTL Function) from 0 to 5 Vp with a frequency of 3 kHz.
2. Measure V_S, $V_C(t)$, and download your results.
3. Determine the rise time of $V_C(t)$. **Note that you must expand the time base enough to make a proper reading.**

4. Adjust the frequency, of the function generator, starting at 100 Hz, until the amplitude of the voltage across the capacitor, displayed on the oscilloscope, is 0.707 of the input voltage amplitude. Download your results.
5. Construct the circuit shown in Figure 5.2. Set up the function generator to produce a square wave from -2.5 V to 2.5 V (5 Vpp) with a frequency of 1 kHz.
6. Measure V_S, $V_C(t)$, and download your results.
7. Determine the rise time from the voltage waveform printout. **Note that you must expand the time base enough to make a proper reading.**

EENG-275
Experiment # 6:
Diode Characteristics, Diode Clipper, and Diode Clamper

I. Objectives

Upon completion of this experiment, the student should be able to:

1. To measure voltage, current relationship of a forward-biased pn junction device.
2. To determine currents in a diode circuit.
3. To design and verify a diode clipper circuit.
4. To design and verify a diode clamper circuit.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- Function Generator
1- Oscilloscope
1- 51 Ω Resistor
1- 220 Ω Resistor
1- 680 Ω Resistor
1- 1 kΩ Resistor
1- 1N4001 Diode
2- 100 Ω Resistor

III. Preparation

1. For, Figure 6.1 assume Vs = 10V, R_1= 100 Ω, D_1 =1N4001. Calculate the diode forward current I_F. Show all computer simulation.

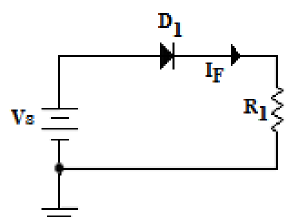


Figure 6.1

2. Calculate v_{OUT} in Figure 6.2, with Vin= 10 Vpp with a frequency of 1 kHz, R_1= 100 Ω, D_1= 1N4001, R_L= 680 Ω. Also, show a computer simulation.

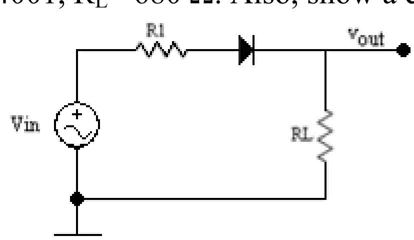


Figure 6.2

3. Calculate the I_D for Figure 6.3, with V_S= 3V, D_1= 1N4001, R_1= 100 Ω, R2 = 220 Ω, R3= 100 Ω. Show all computer simulation.

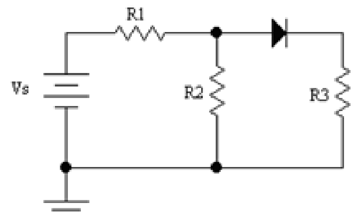


Figure 6.3

4. Design a clipper circuit which limits input signals to +3V and -2V. This means that the output signal should not rise above 3V and should not fall below -2V. Assume V_S has amplitude of 5V and a frequency of 1 kHz, Verify your design by computer simulation on a computer.
5. Design a clamper circuit to clamp the upper limit of the input signal to 0V. Use a 5.1 kΩ resistor, 1N4001diode. Use a square wave input signal of ±1.5V amplitude and a frequency of 1 kHz. Calculate the value of capacitance required. Verify your design by computer simulation on a computer.

IV. Laboratory Procedure

1. Construct the circuit of Figure 6.1, measure the current value I_F.
2. Construct the circuit of Figure 6.2; measure V_{IN} and v_{out} using the oscilloscope.
3. Construct the circuit of Figure 6.3, measure the value of I_D and V_D.
4. Construct the clipper design circuit. Capture the input and output waveforms.
5. Construct your clamper design circuit. Capture the input and output waveforms.

EENG-275
Experiment # 7:
DC Power Supplies

I. Objectives
Upon completion of this experiment, the student should be able to:

1. To analyze a half-wave rectifier circuit
2. To analyze a full-wave rectifier circuit.
3. To observe the operation of a 3-terminal regulator.

II. Material and Equipment
1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- Function Generator
1- Oscilloscope
1- Center Tapped Transformer
1- 10 Ω Resistor
1- 51 Ω Resistor
1- 100 Ω Resistor
1- 1 kΩ Resistor
1- 2.2 kΩ Resistor
1- 10 kΩ Resistor
1- 7805 Voltage Regulator
1- 100 uF Capacitor
1- 220 uF Capacitor
4- 1N4001 Diode

III. Preparation
1. Calculate v_{OUT} for Figure 7.1, with V_{IN}= 10 Vpp sine-wave with a frequency of 60 Hz, R_1= 10 kΩ, D_1= 1N4001. Show all computer simulation.

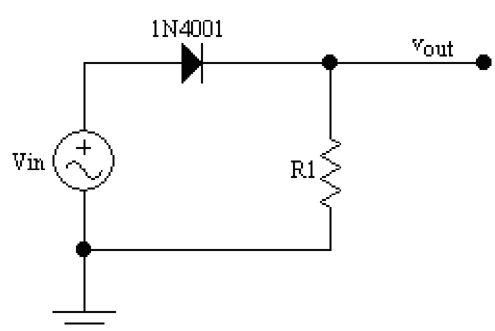


Figure 7.1: Half-Wave Rectifier

2. Calculate v_{OUT} for Figure 7.2, with D_1= 1N4001, D_2= 1N4001, R_L = 100 Ω. Show all computer simulation.

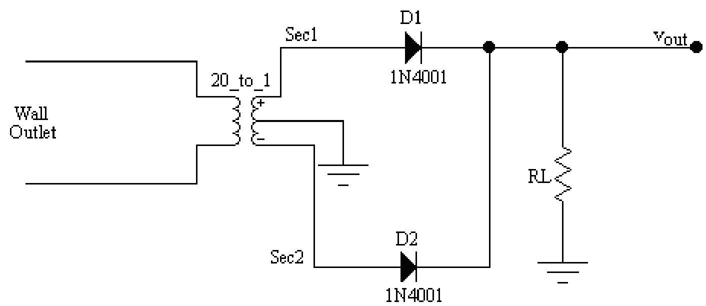


Figure 7.2: Full-Wave Rectifier Circuit

3. Calculate v_{OUT} for Figure 7.3, with D_1= 1N4001, D_2= 1N4001,R_L = 1 kΩ. Show all computer simulation.

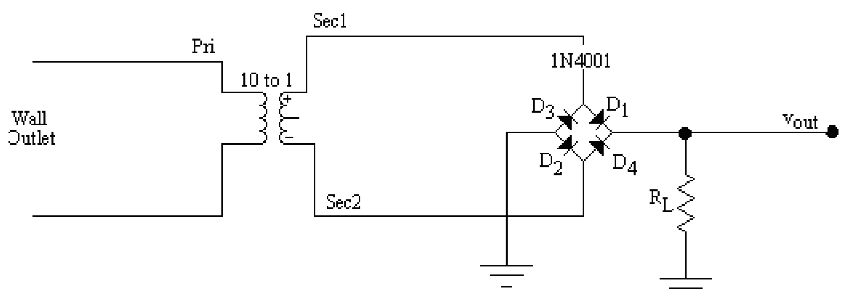


Figure 7.3: Bridge Rectifier Circuit

4. Calculate v_{OUT}, Icharge, and the ripple voltage for Figure 7.4, with R_1= 10 Ω, D_1= D_2 = 1N4001, C_1= 100 µF, R_L = 2.2 kΩ. Also, show a computer simulation.

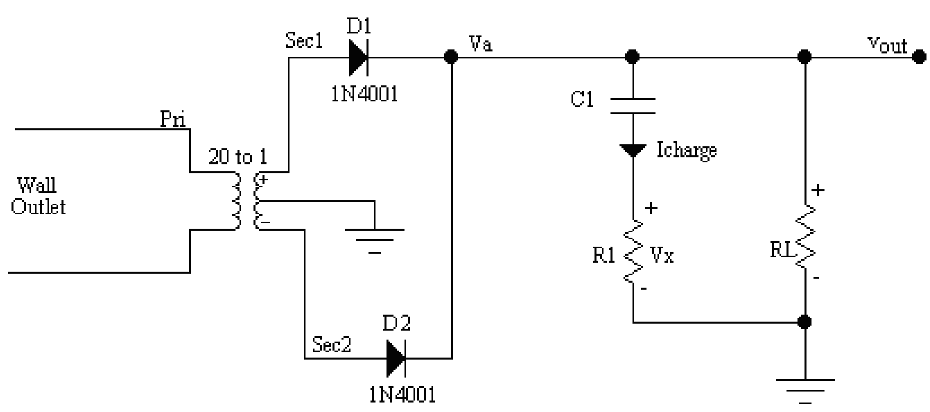


Figure 7.4: Full Wave Rectifier Filter

5. Assume $D_1 = D_2 = D_3 = D_4$ = 1N4001, R_L = 100 Ω, C_1 = 220 µF, for Figure 7.5.
    a. Determine the effective input resistance, Rin, to the 7805 regulator if Iin is 50 mA. Calculate the ripple at V_1.
    b. Using the ripple rejection from the datasheet in Appendix B, estimate the ripple out at terminal 3.

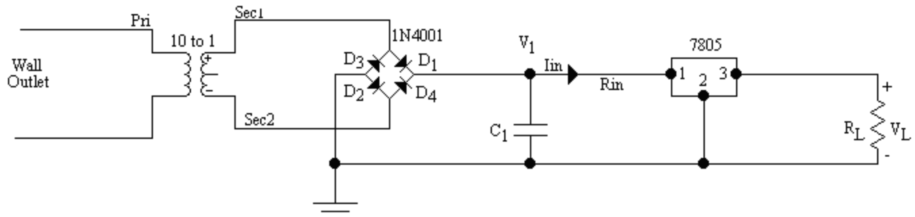


Figure 7.5 Voltage Regulator

Note: In the lab you will measure the dc current going into terminal 1 and recalculate the ripple voltage to see if it agrees with the measurement.

IV. Laboratory Procedure

Note that Appendix B contains the datasheet for the 7805 regulator chip.

1. Construct the circuit of Figure 7.1. Measure V_{IN}, v_{OUT} using the oscilloscope.
2. Construct the circuit of Figure 7.2. Measure Sec1, Sec2, v_{OUT} using the oscilloscope.
3. Construct the circuit of Figure 7.3. Measure Sec1, Sec2, v_{OUT}, using the oscilloscope.
4. Construct the circuit of Figure 7.4. Measure Sec1, Sec2, v_{OUT}, V_X, using the oscilloscope.
5. Construct the circuit of Figure 7.5. Measure Iin using the DMM, calculate Rin. Calculate the ripple voltage at V_1 and compare to the measured ripple at V_1. Measure the ripple at R_L. Decrease C_1 to 100 uF, to increase the ripple. Measure V_L and explain what's happening.

Note: To measure the ripple at pins 1 and 3 of the 7805, be sure to set the coupling of the scope to AC rather than DC as the ripple will be very small.

EENG-275
Experiment # 8:
Bipolar Junction Transistor (BJT) Amplifier

I. Objectives

Upon completion of this experiment, the student should be able to:

1. Determine the dc bias currents and voltages for a single stage BJT amplifier.
2. Determine the small signal (AC) voltage gain of the amplifier.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- Function Generator
1- Oscilloscope
1- CA3046 Transistor Array
1- 51 Ω Resistor
1- 470 Ω Resistor
1- 3.9k Ω Resistor
1- 5.6k Ω Resistor
1- 22 µF Capacitor
2- 10k Ω Resistor
3- 100µF Capacitor
3- 1k Ω Resistor

III. Preparation

1. Draw the equivalent DC circuit for Figure 8.1.
2. Determine the DC load line for Figure 8.1.
3. For the circuit of Figure 8.1, with $V_{CC} = 10V$, $R_1 = 5.6k\ \Omega$, $R_2 = 10k\ \Omega$, $R_C = R_E = 1k\ \Omega$, $R_L = 1k\ \Omega$, $\beta=105$. Calculate the dc current I_{EQ}, V_{BQ}, V_{CEQ}, I_{CQ}, and V_{EQ}.

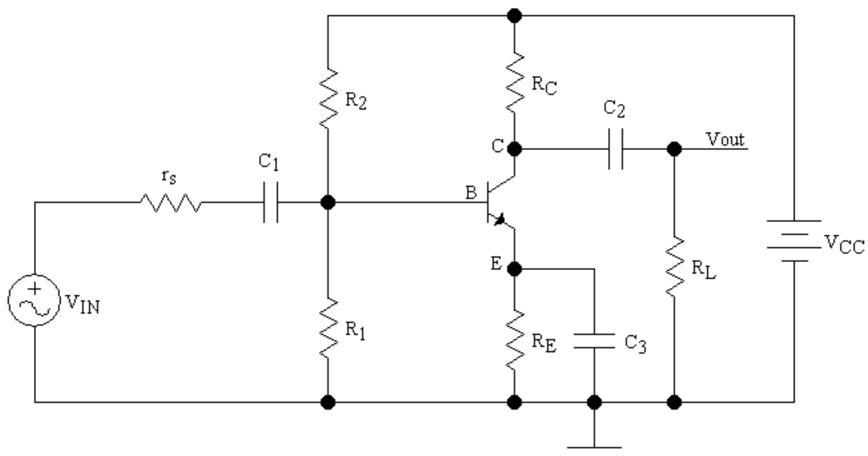


Figure 8.1 Transistor Amplifier Circuit

4. Sketch the AC load line for Figure 8.1.
5. For the circuit of Figure 8.1, with V_{IN} a sinusoid of 5 kHz and V_{CC}=10V, r_S = 10k Ω, R_1= 5.6k Ω, R_2 = 10k Ω, $R_C = R_E = R_L$= 1k Ω, $\beta=105$, $C_1 = C_2 = C_3$= 100 uF, draw the small signal equivalent circuit.
6. Calculate rπ and the voltage gain $Av = \frac{Vout}{Vin}$.
7. Determine the peak amplitude of V_{IN} that makes the output reach its maximum undistorted level. **When you construct the circuit this will give you an estimate of the amplitude to set on the sine wave generator.** Verify your results with a computer simulation.
   For the circuit of Figure 8.1, V_{IN} is a sine wave with a frequency of 5 kHz, V_{CC} = 12V, R_C = 3.9k Ω, R_E = 470 Ω, R_L = 1k Ω, $\beta=105$. Calculate V_{CQ}, V_{BQ}, V_{EQ}, I_{CQ}, I_{BQ}, I_{EQ}, R_1, and R_2 for the maximum possible symmetrical swing. **Hint: First calculate I_{CQ} for maximum symmetrical swing. All the other values can easily be calculated. R_1 and R_2 should be less than 100 kΩ.**
8. Verify your results with a computer simulation that shows the maximum undistorted waveform for Vout.

IV. Laboratory Procedure

Note that Appendix B contains the datasheet for the CA3046 transistor array chip.

1. Construct the circuit of Figure 8.1. Measure the DC values I_{EQ}, V_{BQ}, V_{CEQ}, I_{CQ}, and V_{EQ}.

DO NOT GO TO STEP THREE UNTIL YOUR MEASUREMENTS AGREE WITH STEP TWO OF THE LABORATORY PREPARATION.

2. Construct the circuit of Figure 8.1. Measure the maximum input signal level for an undistorted V_{OUT} for a 5 kHz sine-wave input. Determine the Av experimentally.
3. Compare with calculations and simulation and explain your results in detail.
4. Construct the circuit of Figure 8.1. Measure V_C, V_B, V_E, I_C, I_B and I_E using the DMM.

DO NOT GO TO STEP EIGHT UNTIL YOUR MEASUREMENTS AGREE WITH STEP SEVEN OF THE LABORATORY PREPARATION.

5. Construct Figure 8.1, with the calculated values for R_1, and R_2.
6. Measure the maximum input signal level for an undistorted V_{OUT} for a 5 kHz sine-wave input. Determine the Av experimentally
7. Compare with calculations and simulation and explain your results in detail.

EENG-315
Experiment # 9:
MOSFET Amplifier

I. Objectives

Upon completion of this experiment, the student should be able to:

1. To bias a NMOS transistor.
2. To use a NMOS transistor in a common-source amplifier configuration and to measure its amplification.
3. To study the effect of the source resistor and bypass capacitor on the amplification.

II. Material and Equipment

1- NYIT supplied Lab Kit
1- Digital Multi-meter (DMM)
1- DC Power Supply
1- Function Generator
1- Oscilloscope
1-CD4007 MOS Transistor Array
1-33 nF Capacitor
1-0.1 uF Capacitor
1-1 uF Capacitor
1-10 kΩ Resistor
1-51 kΩ Resistor
1-100 kΩ Potentiometer

III. Preparation

Part I

1. Design the common-source amplifier of Figure 9.1.

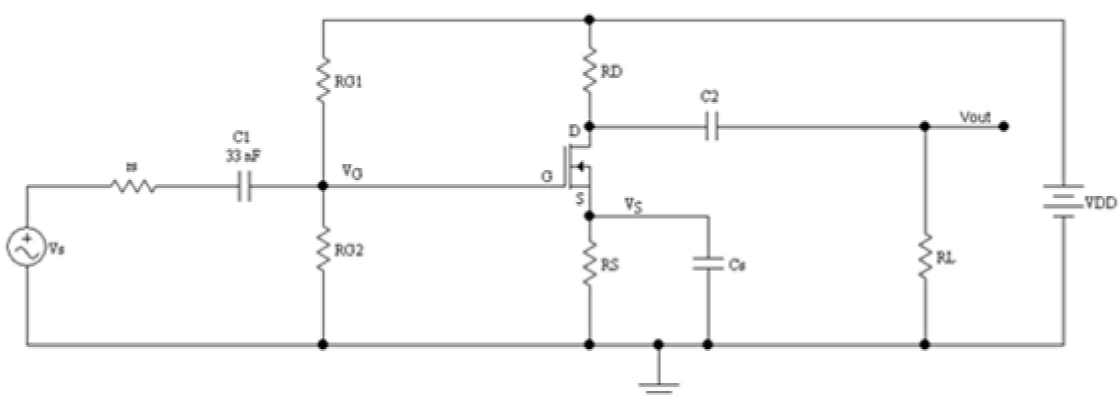


Figure 9.1: Common Source Amplifier

2. The NMOS transistor (CD4007CN array) has the following characteristics:
   a. Vt=1.2
   b. $\frac{k_n' W}{L} = 0.7 \frac{mA}{V^2}$
   c. $\lambda = 0.004 V^{-1}$
   d. $\gamma = 1.9 V^{0.5}$
3. Use Figure 9.3, find V_D.

4. Find the resistor values R_S and R_D.
5. Find the values for V_S and V_G.
6. Find the values of the resistors RG1, and RG2.

Part II

1. Use Figure 9.4, find the value of the trans-conductance g_m and small signal output resistor r_o.
2. Assume that no load resistance R_L is present, find the value of A_{vo}.
3. Use Figure 1 to calculate the voltage gain A_V for the load resistance R_L=10 kΩ.
4. Assume that a source resistance rs = 5 kΩ, what is the effect on the overall amplification G_V?
5. What is the minimum voltage at the source in order to keep the transistor in saturation?
6. After obtaining the **approval of your instructor**, may you continue to the next part of the experiment.

IV. Laboratory Procedure

Note that Appendix C contains the datasheet for the CD4007 MOS Transistor Array chip.

***Precaution*: MOSFET transistors are very susceptible to breakdown due to electrostatic discharge. It is recommended that you always ground yourself before picking up the MOSFET chip. Do not touch any of the pins of the chip.**

1. You will be using the CD4007 MOSFET array. This array contains three NMOS and three PMOS transistors as shown in Figure 9. 2.

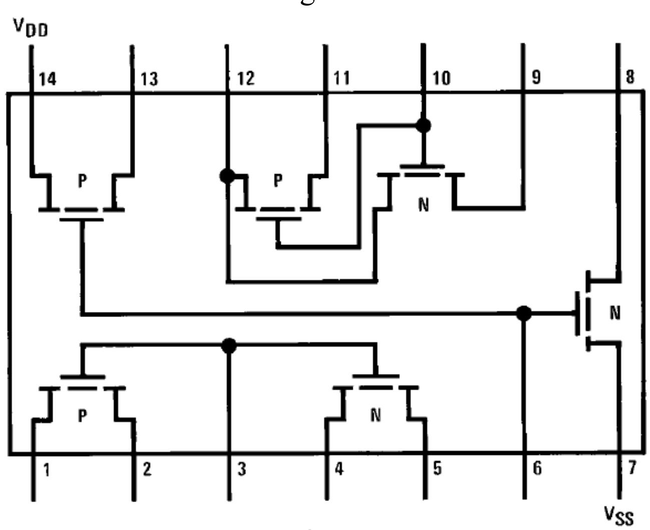


Figure 9.2: CD4007 MOSFET array

2. It should be mentioned that the transistor characteristics of the CD4007 could vary considerably from chip to chip. The transistors may come from a different batch, what can explain why the threshold voltage, the trans-conductance parameter and the output resistance is different from the one used in the hand calculations and the computer simulation.

3. Build the circuit of Figure 3.

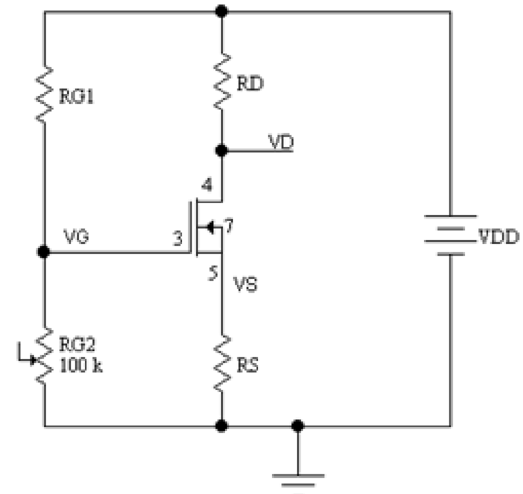


Figure 9.3: Biasing Circuit for Common-Source Amplifier

4. Use the transistor between the pins 3, 4 and 5.

Notice that we have connected the bulk (pin 7) to the source (pin 5) of the NMOS transistor. This can be done since we are only using a single NMOS transistor in the array. The reason for shorting drain-to-bulk is to eliminate the back-gate (body) effect on the threshold voltage. For the biasing resistor R_{G2}, use a100 kΩ potentiometer.

5. Measure the DC voltage at the drain VD. The drain voltage should be position around 9 or 10 V.
6. Measure the gate and source voltages.
7. What is the corresponding drain current I_D?
8. Build the circuit of Figure 9.4.

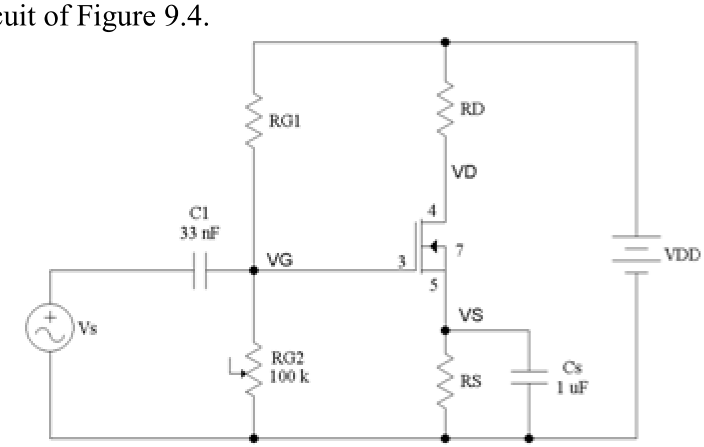


Figure 9.4

9. Connect a sinusoidal input signal of 0.4Vpp at a frequency of 5k Hz.
10. Measure the voltage swing at the drain using the oscilloscope.
11. Display the input voltage VS on Channel 1, and the drain voltage VD on Channel 2.
12. Measure the peak to peak values of the input and output signal. What is the open-circuit amplification voltage gain A_{VO}?

13. Notice the phase relationship. How does it compare to the calculated one? **It should be mentioned that the transistor characteristics can vary considerably from chip to chip. It is not unusual the measured values differ by as much as 10-20% from the specified ones.**
14. Increase the amplitude of the input signal and observe the output signal. When does the output signal start to distort?
15. What is the maximum output voltage swing before considerable distortion occurs?
16. Reduce the input signal so that the output signal looks undistorted.
17. Measure the FFT of the output signal and determine the total harmonic distortion. Write down how many dB the amplitude of the harmonics are below the amplitude of the fundamental frequency.
18. Build the circuit of Figure 9.1.
19. Measure the output voltage Vout and the corresponding amplification A_V.
20. Notice that the DC voltage has been removed from the output voltage. What happened to the amplification?
21. Measure the frequency response of the amplifier with the load resistor connected. Change the frequency of the input signal. Starting from 5 kHz, reduce the frequency till the amplitude of the output has decreased by a factor of 0.707 (or 3dB). Record this 3dB frequency.
22. What is the phase relationship between input and output signal? Next, increase the value of the input frequency till the amplification A_V reduces to 0.707 of its value at 5kHz. Record the high-frequency 3dB point.
23. What is the bandwidth of the amplifier?

Appendix A- LM7805 Voltage Regulator

FAIRCHILD
SEMICONDUCTOR®

May 2006

LM78XX/LM78XXA
3-Terminal 1A Positive Voltage Regulator

Features

- Output Current up to 1A
- Output Voltages of 5, 6, 8, 9, 10, 12, 15, 18, 24
- Thermal Overload Protection
- Short Circuit Protection
- Output Transistor Safe Operating Area Protection

General Description

The LM78XX series of three terminal positive regulators are available in the TO-220 package and with several fixed output voltages, making them useful in a wide range of applications. Each type employs internal current limiting, thermal shut down and safe operating area protection, making it essentially indestructible. If adequate heat sinking is provided, they can deliver over 1A output current. Although designed primarily as fixed voltage regulators, these devices can be used with external components to obtain adjustable voltages and currents.

Block Diagram

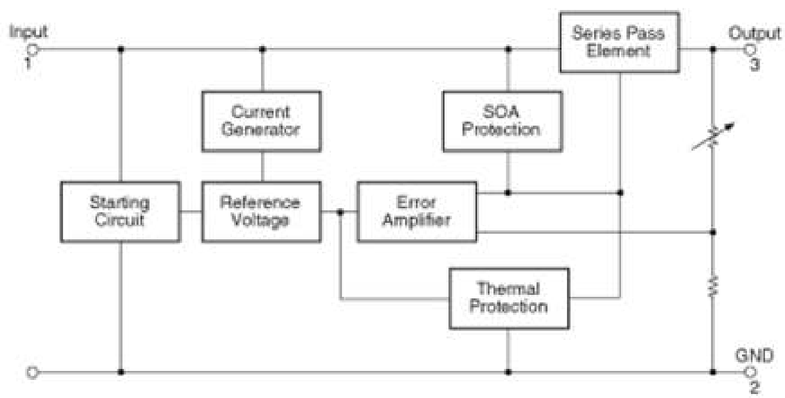


Figure 1.

Pin Assignment

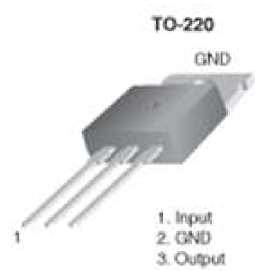


Figure 2.

Absolute Maximum Ratings

Absolute maximum ratings are those values beyond which damage to the device may occur. The datasheet specifications should be met, without exception, to ensure that the system design is reliable over its power supply, temperature, and output/input loading variables. Fairchild does not recommend operation outside datasheet specifications.

Symbol	Parameter		Value	Unit
V_I	Input Voltage	V_O = 5V to 18V	35	V
		V_O = 24V	40	V
$R_{\theta JC}$	Thermal Resistance Junction-Cases (TO-220)		5	°C/W
$R_{\theta JA}$	Thermal Resistance Junction-Air (TO-220)		65	°C/W
T_{OPR}	Operating Temperature Range	LM78xx	-40 to +125	°C
		LM78xxA	0 to +125	
T_{STG}	Storage Temperature Range		-65 to +150	°C

Electrical Characteristics (LM7805)

Refer to the test circuits. -40°C < T_J < 125°C, I_O = 500mA, V_I = 10V, C_I = 0.1μF, unless otherwise specified.

Symbol	Parameter	Conditions		Min.	Typ.	Max.	Unit
V_O	Output Voltage	T_J = +25°C		4.8	5.0	5.2	V
		5mA ≤ I_O ≤ 1A, P_O ≤ 15W, V_I = 7V to 20V		4.75	5.0	5.25	
Regline	Line Regulation(1)	T_J = +25°C	V_O = 7V to 25V	–	4.0	100	mV
			V_I = 8V to 12V	–	1.6	50.0	
Regload	Load Regulation(1)	T_J = +25°C	I_O = 5mA to 1.5A	–	9.0	100	mV
			I_O = 250mA to 750mA	–	4.0	50.0	
I_Q	Quiescent Current	T_J = +25°C		–	5.0	8.0	mA
ΔI_Q	Quiescent Current Change	I_O = 5mA to 1A		–	0.03	0.5	mA
		V_I = 7V to 25V		–	0.3	1.3	
$\Delta V_O/\Delta T$	Output Voltage Drift(2)	I_O = 5mA		–	-0.8	–	mV/°C
V_N	Output Noise Voltage	f = 10Hz to 100kHz, T_A = +25°C		–	42.0	–	μV/V_O
RR	Ripple Rejection(2)	f = 120Hz, V_O = 8V to 18V		62.0	73.0	–	dB
V_{DROP}	Dropout Voltage	I_O = 1A, T_J = +25°C		–	2.0	–	V
r_O	Output Resistance(2)	f = 1kHz		–	15.0	–	mΩ
I_{SC}	Short Circuit Current	V_I = 35V, T_A = +25°C		–	230	–	mA
I_{PK}	Peak Current(2)	T_J = +25°C		–	2.2	–	A

Notes:

1. Load and line regulation are specified at constant junction temperature. Changes in V_O due to heating effects must be taken into account separately. Pulse testing with low duty is used.
2. These parameters, although guaranteed, are not 100% tested in production.

Appendix B- LM3046 Transistor Array

National Semiconductor

July 1999

LM3046
Transistor Array

General Description

The LM3046 consists of five general purpose silicon NPN transistors on a common monolithic substrate. Two of the transistors are internally connected to form a differentially-connected pair. The transistors are well suited to a wide variety of applications in low power system in the DC through VHF range. They may be used as discrete transistors in conventional circuits however, in addition, they provide the very significant inherent integrated circuit advantages of close electrical and thermal matching. The LM3046 is supplied in a 14-lead molded small outline package.

Features

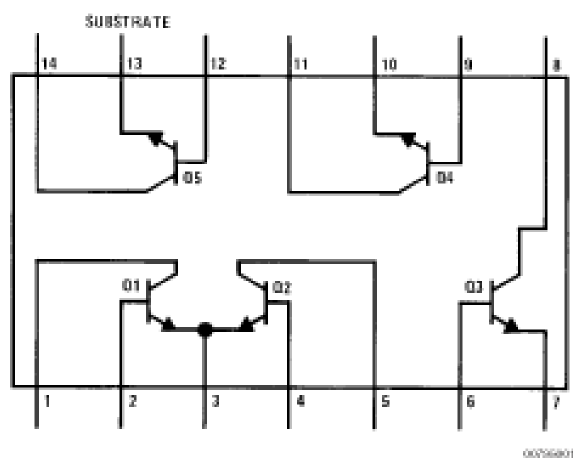
- Two matched pairs of transistors
  V_{BE} matched ±5 mV
  Input offset current 2 μA max at I_C = 1 mA
- Five general purpose monolithic transistors
- Operation from DC to 120 MHz
- Wide operating current range
- Low noise figure: 3.2 dB typ at 1 kHz

Applications

- General use in all types of signal processing systems operating anywhere in the frequency range from DC to VHF
- Custom designed differential amplifiers
- Temperature compensated amplifiers

Schematic and Connection Diagram

Small Outline Package



Top View
Order Number LM3046M
See NS Package Number M14A

Appendix C - CD4007 MOS Transistor Array

National Semiconductor

February 1988

CD4007M/CD4007C Dual Complementary Pair Plus Inverter

General Description

The CD4007M/CD4007C consists of three complementary pairs of N- and P-channel enhancement mode MOS transistors suitable for series/shunt applications. All inputs are protected from static discharge by diode clamps to V_{DD} and V_{SS}.

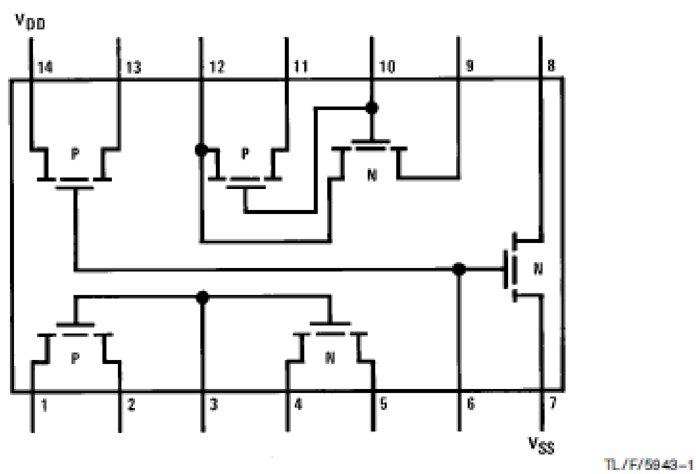
For proper operation the voltages at all pins must be constrained to be between V_{SS} − 0.3V and V_{DD} + 0.3V at all times.

Features

- Wide supply voltage range 3.0V to 15V
- High noise immunity 0.45 V_{CC} (typ.)

Connection Diagram

Dual-In-Line Package



Top View

Note: All P-channel substrates are connected to V_{DD} and all N-channel substrates are connected to V_{SS}.