**EE315 - Electronics Laboratory**

# Experiment - 1 Simulation
# AC Signal Measurement and Half-Wave Rectifiers

### Objectives

**1.** Review AC signal parameters.
**2.** Review diode response.
**4.** Analyze single diode rectifier circuits.

# Background

You must read the section titled ***Grounding of Test Equipment*** in the ***Electronics Laboratory Beginner's Guide*** before you start the experimental procedure. The voltage measurements in this experiment require the scope setup for differential voltage measurements using two scope probes.

## 1. AC Signals

Nonsinusoidal signals are an important class of signals, and are found in a wide variety of situations. There are many different ways to characterize these signals. This experiment focuses on a number of these.

**Peak-to-peak amplitude** of a periodic waveform is the difference between the lowest and highest amplitudes in the waveform.

**Peak value** of a waveform is the magnitude of the highest (or lowest) point of the waveform with respect to a reference or ground level. If the waveform is symmetric above and below the reference level, then a single peak value is specified. In case a waveform is not symmetric around the reference level, the positive and negative peak values should be measured and specified separately.

**DC (average) value** of a periodic signal is the average value over the period.

**RMS (root-mean-square) value** of a periodic waveform is defined as:

$$V_{rms} = \sqrt{\frac{1}{T}\int_{t=0}^{T} v^2(t)\, dt}$$

where, **square** of the signal is integrated over the waveform period, **T**, and then square **root** of the **mean** value is calculated. RMS value of a signal is useful in power calculations. Power dissipation **P** on a resistor can be calculated in terms of the RMS values

$$P = \frac{V_{rms}^2}{R} = I_{rms}^2 R$$

where, $\mathbf{V_{rms}}$ is the RMS voltage across **R**, and $\mathbf{I_{rms}}$ is the RMS current through **R**. Resistive power calculation based on RMS values will be correct for all AC waveform types combined with any DC level.

## Pulse Waveform Parameters

Pulse waveforms appear in a variety of shapes in common application areas, such as logic circuits, digital communications, and switching power control. Description of pulse waveforms require additional timing parameters:

**On time ($T_{on}$) or high time ($T_{high}$):** Time duration where the signal is at **active level** (mostly with a high amplitude) in a pulse waveform.

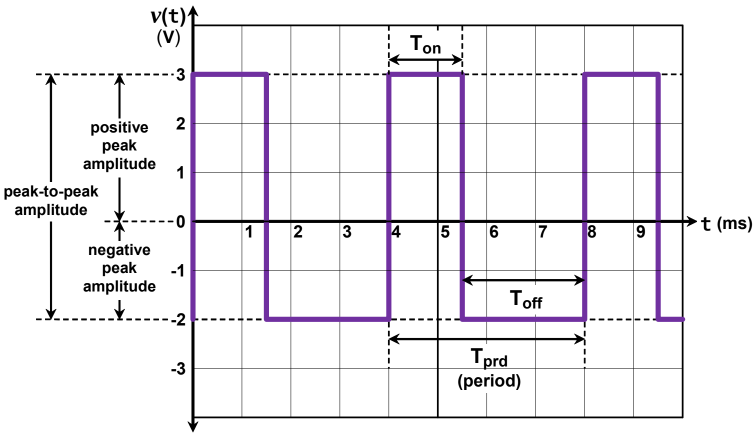
**Off time ($T_{off}$) or low time ($T_{low}$):** Time duration where the signal remains idle at reference level (mostly with a low amplitude) in a pulse waveform.

**Duty cycle (DC):** Ratio of on time to period of a pulse waveform. Duty cycle is usually expressed as a percentage:

$$\textbf{Duty cycle} = \frac{T_{on}}{T_{on} + T_{off}} \times 100\ \% = \frac{T_{on}}{T_{prd}} \times 100\ \%$$

As shown in the following example, if $\mathbf{T_{prd} = 4.0\ ms}$ and $\mathbf{T_{on} = 1.5\ ms}$ for a pulse waveform, then its duty cycle is

$$\textbf{Duty cycle} = \frac{1.5}{4.0} \times 100\ \% = 37.5\ \%.$$



# 2. Diode Circuits

A semiconductor diode is a non-linear device that changes its conductivity depending on the voltage polarity accross its terminals. A diode acts like a good conductor when it is forward biased and it has a large resistance when the voltage polarity is reversed. V-I characteristics (current versus voltage graph) of a diode is given below. A piecewise linear circuit model of the diode is obtained by linearizing the V-I characteristics.

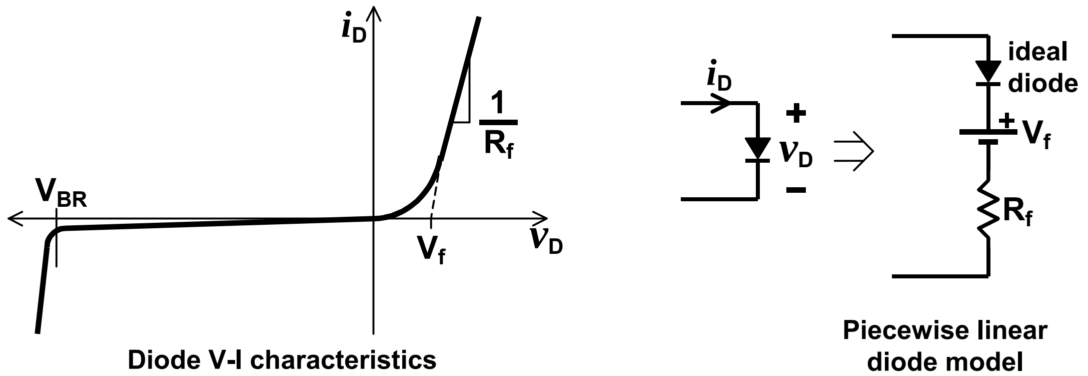



**Figure 2. Diode current versus voltage characteristic curve and piecewise linear model under forward bias.**

Diodes are used in various rectifier and wave shaping circuits. Conventional diodes can be divided into two groups as follows:

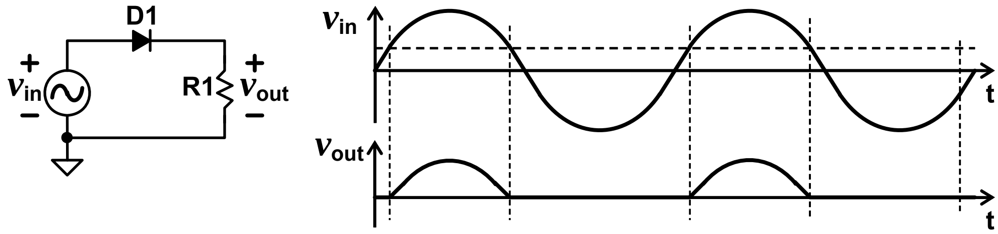
1. Signal diodes
2. Power diodes

**Signal diodes** are generally used in low-voltage and low-current applications where ON/OFF switching speed and a large reverse-to-forward resistance ratio are important. Signal diodes are mostly packaged in small cases, since power dissipation in typical waveshaping circuits is not critical.

**Power diode** applications are generally limited to the various rectifier circuits that convert AC power to DC power. Important characteristics of the power diodes are their ability to dissipate power and withstand large forward currents and large reverse voltages. Package size of power diodes varies dramatically with the current carrying and power dissipation capability.

# Half-Wave Rectifier Circuits

Electrical power in buildings is available in AC because of better transmission and distribution efficiency compared to DC power. All electronic equipment working on DC power require some kind of rectifier to convert AC voltage to DC voltage. The basic half-wave rectifier circuit is shown below. The input signal $v_{in}$, is assumed to be purely sinusoidal AC signal with a time-average value of zero. The output signal $v_{out}$, across the load resistor **R1** will be as shown in the figure, since the diode conducts a significant current only during the positive cycles of the input waveform. The half-wave rectifier circuit produces an output signal that has a fundamental frequency same as the input AC freuency.




In case of an ideal diode, $v_{out}$ does not depend on the load resistor and obviously it is exactly the positive half of $v_{in}$. The rectified output is now a combination of an AC signal and a DC component. Generally, it is the DC part of the rectified signal that is of interest, and the undesired AC component is described as ripple. Furthermore, the pulsating DC output signal can be made steady by means of a smoothing circuit. The quality of a rectifier circuit is measured by its ***ripple factor***, which is defined as

$$\textbf{Ripple factor} = \frac{\textbf{RMS value of AC output}}{\textbf{DC level of output}}$$

# Preliminary Work

**1.** Show that the RMS value $\mathbf{V_{rms}}$ of a sinusoidal voltage waveform $v(t) = V_p \sin(\omega t)$ is given by

$$V_{rms} = \frac{V_p}{\sqrt{2}}$$

where, $\mathbf{V_p}$ is the peak value of the voltage waveform.

**Hint:** Calculate the RMS value as it is defined:

$$V_{rms} = \sqrt{\frac{2}{T}\int_{t=0}^{T/2} V_p^2 \sin^2(\omega t)\, dt} \quad \text{by using} \quad \sin^2(\omega t) = \frac{1 - \cos(2\omega t)}{2}$$

**2.** Show that the total RMS value $\mathbf{V_{\Sigma rms}}$ of a voltage waveform $v(t) = V_{DC} + v_{AC}(t)$ is given by

$$V_{\Sigma rms} = \sqrt{V_{DC}^2 + V_{ACrms}^2}$$

where, $\mathbf{V_{ACrms}}$ is the RMS value of the pure AC component $v_{AC}(t)$.

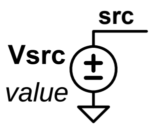
**Hint:** DC (average) value of $v_{AC}(t)$ is zero by definition, and this implies

$$\frac{1}{T}\int_{t=0}^{T} v_{AC}(t)\, dt = 0$$

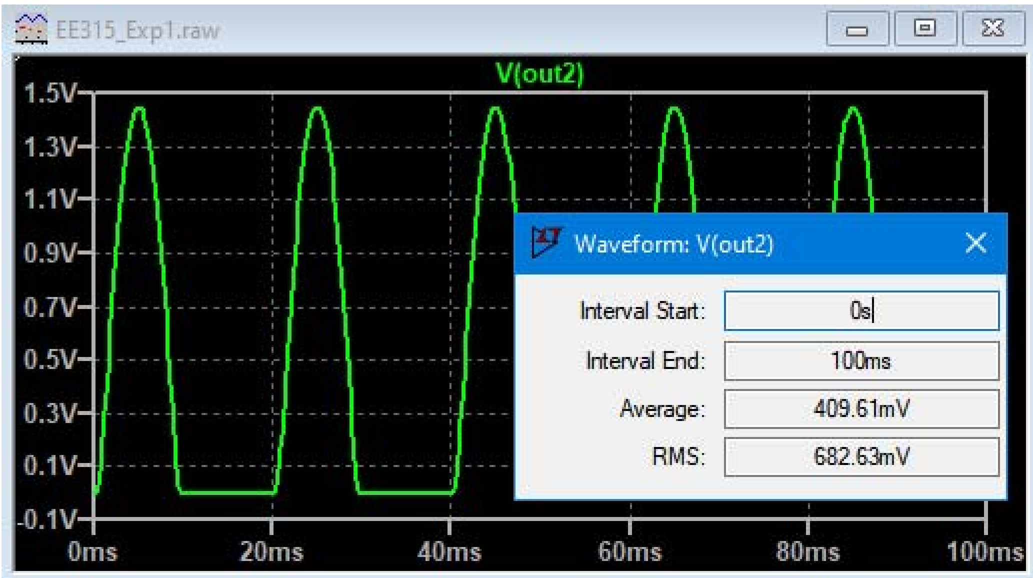
# Procedure

## Part 1. Voltage Measurements

Open a new schematic file in LTspice and place a voltage source with a net label at its positive output as shown below. Set the ***value*** of the voltage source to obtain **1 kHz** sine wave, with an amplitude of **2 V**p-p (peak-to-peak). Enter the SPICE directive "**.tran 5m**" to set the simulation time to **5 ms**.




**LTspice** calculates the DC level (average) and RMS value of a waveform when you left-click on the waveform label while holding down the CTRL key. DC and RMS values are calculated for part of the signal displayed on the waveform waveform window as shown below.




**1.a)** Run the simulation and record the following for the sine wave:

Peak voltage: __________ Peak to-peak voltage: __________

DC voltage: __________ RMS voltage __________

**1.b)** Change the voltage source value to "**`PULSE(-1 1 0 500u 500u 0 1m)`**" to obtain a triangle wave and measure the following:

Peak voltage: __________ Peak to-peak voltage: __________

DC voltage: __________ RMS voltage __________

**1.c)** Change the voltage source value to "**`PULSE(-1 1 0 1u 1u 499u 1m)`**" to obtain a square wave and measure the following:

Peak voltage: __________ Peak to-peak voltage: __________

DC voltage: __________ RMS voltage __________

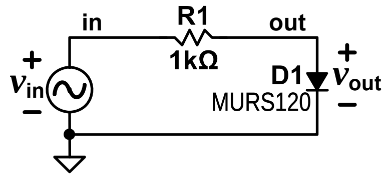
**1.d)** Change the voltage source parameters to obtain a square wave with **4 V**p-p AC amplitude and **1 V** DC offset. Measure the following voltages:

Peak voltage: __________ Peak to-peak voltage: __________

DC voltage: __________ RMS voltage __________

## Part 2. Diode Circuits

Build the following circuit, with a **1 kΩ** resistor and a **MURS120** silicon rectifier diode. Set the voltage source parameters to obtain **1 kHz** triangular waveform with **5 V** peak amplitude. Enter the SPICE directive "**.tran 5m**" to set the simulation time to **5 ms**.



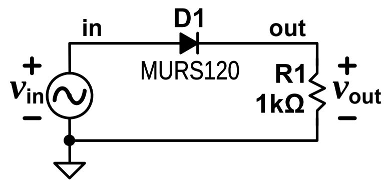

**2.a)** Describe a method to measure the current in this circuit based on voltage measurements only.

Note that an ordinary oscilloscope probe can only measure voltage in practice. In this circuit, ground connectors of scope probes and function generator cable should be attached to the same reference node in the circuit. Oscilloscope should be setup for differential voltage measurements by using two probes.

**2.b)** Measure the circuit current using the current probe of LTspice and compare with the results you obtained based on voltage measurements.

Right-click on the waveform window and select the **Add Traces** option. If a resistor value is entered with the "`ohm`" unit as in the expression "`(V(n1)-V(n2))/1000ohm`" LTspice can display the resultant waveform with the correct phsical unit and scaling.

**3.** Build the following circuit and use the same **1 kHz** triangular waveform as $v_{in}$.




**3.a)** Sketch the $v_{in}$ and $v_{out}$ waveforms.

**3.b)** Estimate turn on voltage of the diode on the $v_{in}$ and $v_{out}$ waveforms.

Right-click on the waveform window and select the **Draw>Line** option. Draw a line that matches the linear region of the $v_{out}$ waveform. Find the $v_{in}$ voltage corresponding to the time instant where the line you draw crosses the **0 V** reference.

**4.** Change input waveform to $v_{in}$ **= 2V sin(2π 1000t)**.

**4.a)** Sketch $v_{out}$ waveform. What is the peak output voltage measured?

**4.b)** Verify that the measured peak output voltage is correct with calculations.

**4.c)** Measure the frequency and period of $v_{in}$ and $v_{out}$.

**5.** Change input waveform to square wave keeping frequency and amplitude settings the same.

**5.a)** Measure the peak voltage and DC component of $v_{out}$.

**5.b)** Calculate the expected average value of $v_{out}$ based on the measurements on $v_{in}$ signal and compare with the DC value measured before.

**5.c)** Measure the frequency and period of $v_{in}$ and $v_{out}$.

# Questions

**Q1.** What is the minimum possible value of the ratio $\mathbf{V_{peak}/V_{rms}}$ for a waveform? Describe a waveform that has the minimum $\mathbf{V_{peak}/V_{rms}}$ ratio.

**Q2.** A function generator has **50 Ω** output source resistance. Draw a modified schematic for the circuit used in step-**3** of the procedure that gives a closer simulation of the actual experiment with a function generator.