Faculty of Engineering and Technology

Electrical and Computer Engineering

ENEE3309 Communication Systems

2023-2024-1st Semester Project - Phase 2

Prepared by: Rasha Daoud 1210382

Nadia Thaer 1210021

Lama Khateeb 1213515

Sarah Hassouneh 1210068

Instructor: Dr. Qadri Mayyala

Section: 4

Date: 18 Jan 2024

Table of Contents

Table Of Figures:

Circuit Design:

In this phase we had to design a hardware implementation of the circuit we designed in phase 1. We used Autodesk TinkerCad to implement the circuit and simulate it on a bread board.

The circuit we mainly designed in phase 1(Modulator 1) on Pspice is shown below (Figure 1):

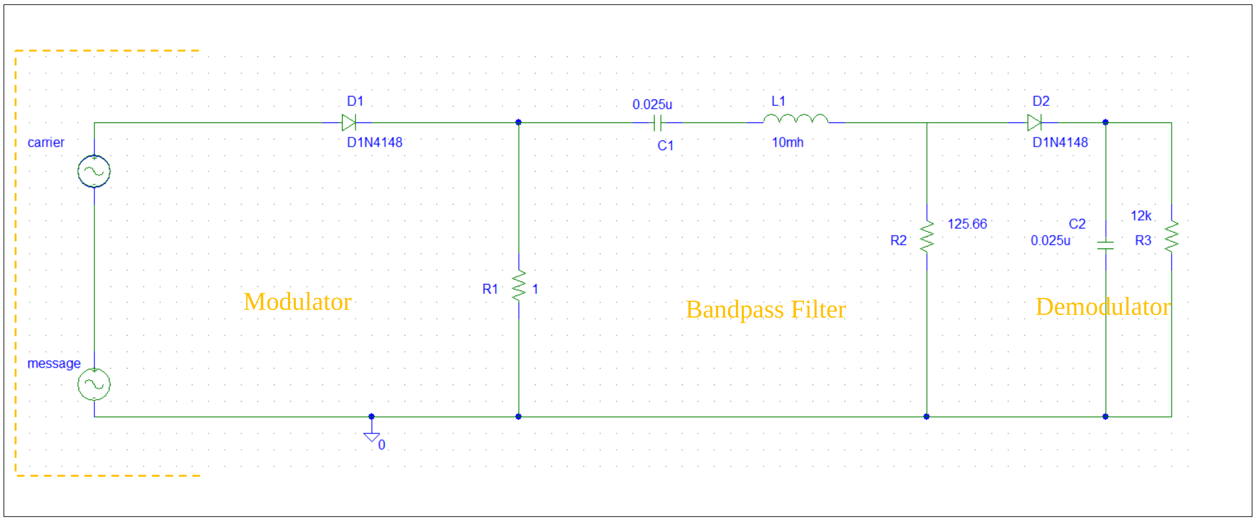


Figure 1: Circuit in Pspice

The circuit we designed on the bread board is shown (Figure 2), the circuit we designed is similar to the one we designed in Pspice in phase 1; we used the same values that were calculated in phase1. However, we had to adjust the value of **R1**, originally it was 1Ω and we had to increase it to 20Ω. We noticed after trial and error, setting R1 to 20Ω gave us the most accurate simulation results, especially on oscilloscope 3 that showed us the signal (message + carrier) after passing through the diode (see Figure 3). This can be justified for two causes; first, larger resistors offer better stability and reliability in terms of performance especially in applications where precision and consistency are essential. Second, in sensitive circuits, smaller resistors may contribute more to thermal noise because of the power that get lost in the resistor. Using a larger resistor with a higher resistance value can help minimize noise, especially in high-impedance circuits (like in our case where there is an impedance generated from the capacitor and inductor). When we had R1 set as 1Ω we noticed that the carrier and message signal on oscilloscopes

(1,2,3) shown in (Figure 3) had problems as if the circuit was shorted between R1 terminals, and the diode result was not accurate.

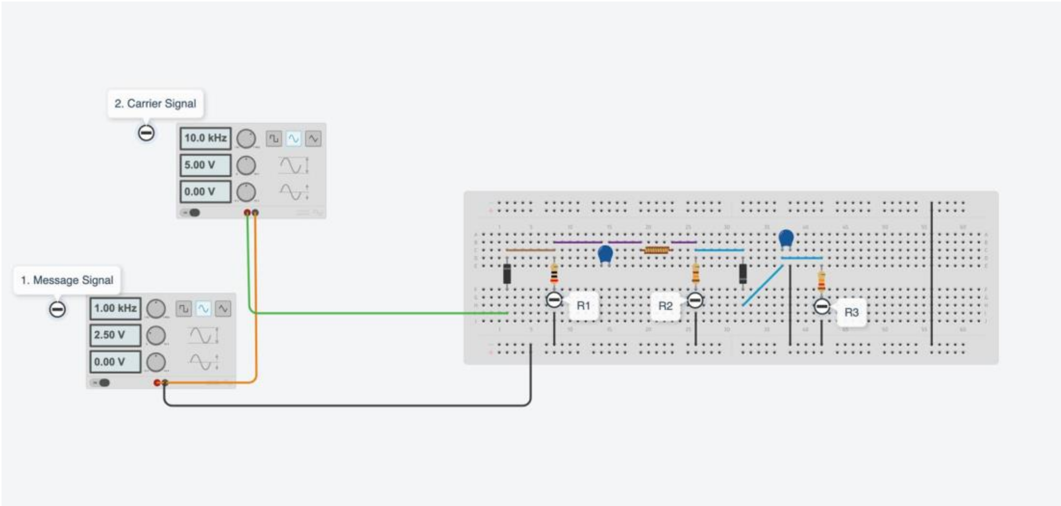


Figure 2: Circuit on breadboard

We connected the components (see Appendix 1 for components details) and then used different oscilloscopes to monitor results and signal throughout the stages of the modulator and demodulator as follows (Figure 3):

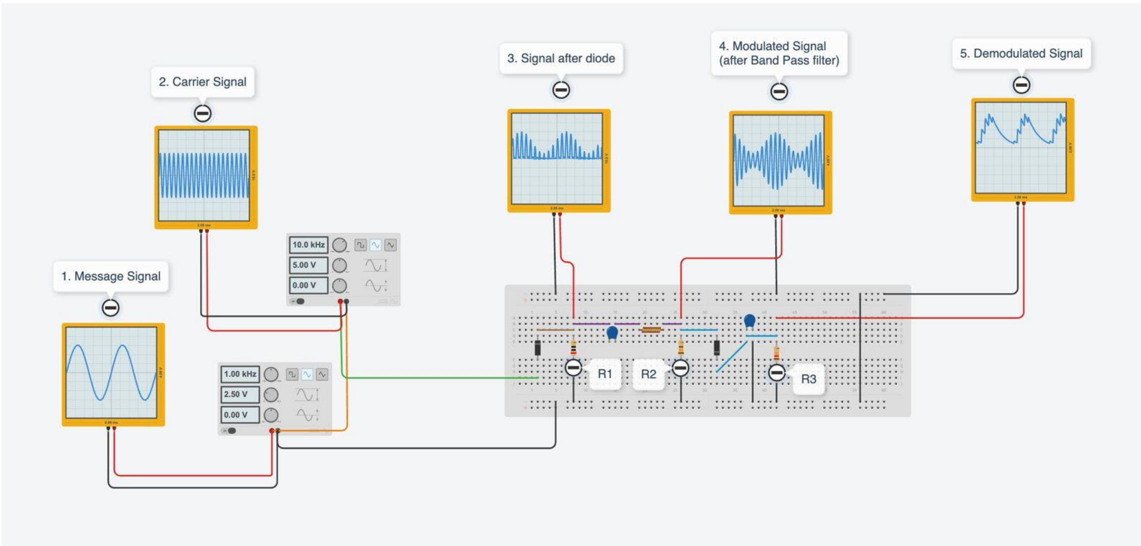


Figure 3: Breadboard Circuit after Simulation[1]

[1] *a larger picture is shown in Appendix 1*

The circuit on TinkerCad can be accessed through this link here. Please note that you can simulate the circuit directly through this link, **but please wait for the circuit to stabilize which could take up to a minute or two .**

In the upcoming section we will compare the results we achieved on the breadboard circuit on each oscilloscope with the results we achieved while building circuit on Pspice.

Results & Comparisons :

In this part we will compare each output on the oscilloscopes that we obtained from the bread board circuit on TinkerCad with the figures we obtained in the Pspice simulation.

1. Comparative Analysis of Message and Carrier Signal (Oscilloscope 1 & 2 results)

The message signal (modulating signal) and the carrier signal (modulated signal) are the signals that we have identified by determining their amplitude and frequency in this form:

$m(t) = Am \cos(2\pi f m t)$.

$c(t) = Ac \cos(2\pi f c t)$.

Considering that the frequency of the carrier is larger than the frequency of the message by around 10 times more, and we can notice this difference in figures below. (Figure 4) shows the message signal obtained from the simulation on Pspice. (Figure 5) shows the carrier signal obtained from the simulation on Pspice. We can observe the general shape of the two signals, which take the form of sinusoidal signals.

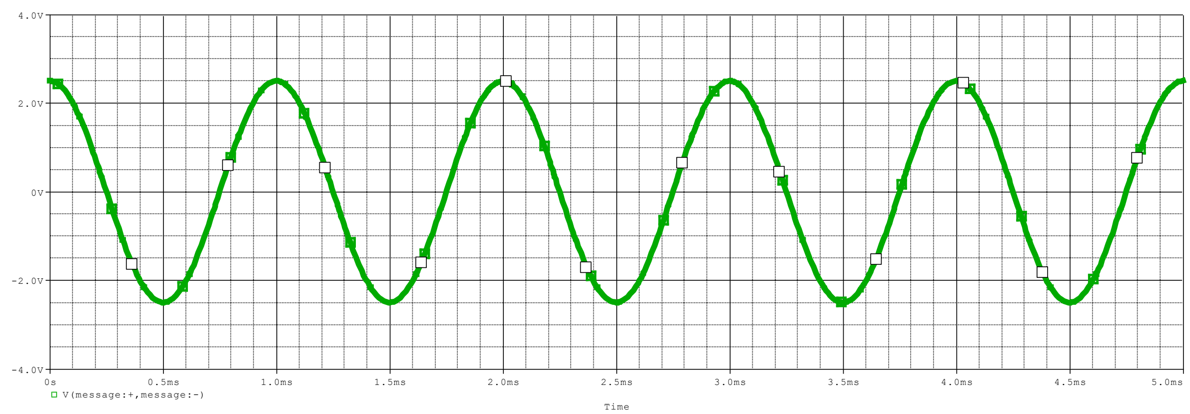


Figure 4: Simulation of Message signal on Pspice

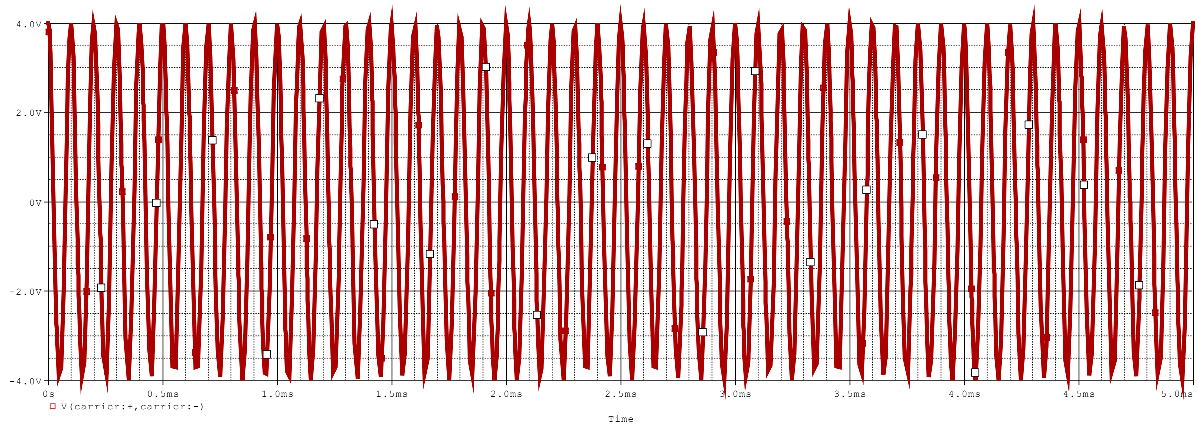


Figure 5: Simulation of Carrier signal on Pspice

Figure 6 below is the outputs on oscilloscope 1 and 2 on TinkerCad tool which is represent the message and the carrier signal respectively.

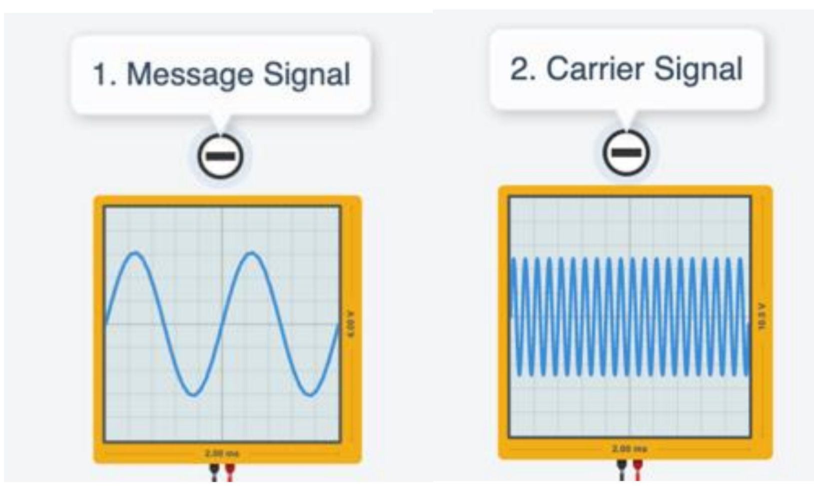


Figure 6: Message and Carrier signal on TinkerCad.

We can notice that the shape of the signal is very similar to the sinusoidal signal coming from Pspice. It is clear that there is a difference between the frequency of the message and the carrier, and this is evident from the small period of the carrier compared to the period of the message.

An important note is that the function generator in TinkerCad in designed to generate sine wave not a cosine, however we know that the relationship between the sine and cosine is a phase shift of 90 degrees. So, we can actually replace the cosine function with a sine function. As long as we maintain a zero-phase shift between the message and the carrier (by using both sine or cosine waves) we will get the same results and the analysis won't differ. In amplitude modulation (AM), the important factor is the relationship between the message signal and the carrier signal, rather than the specific choice of sine or cosine functions.

2. Comparative Analysis of Signal After Diode (Oscilloscope 3 results)

From the Phase 1 the signal after the diode is represent by the voltage on R1 the voltage outcomes from a clipper circuit as in (Figure 7):

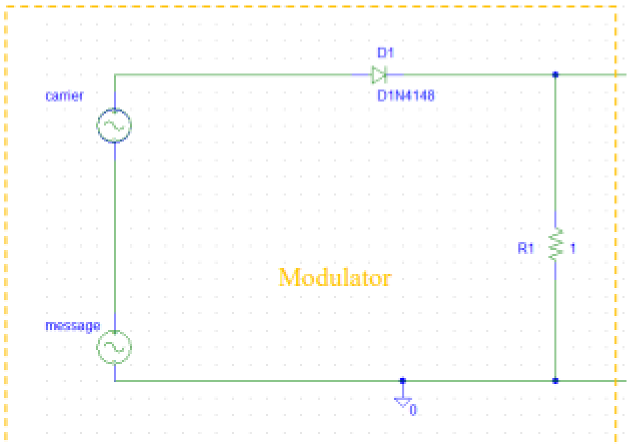


Figure 7: The Clipper Circuit

Then the Signal After the Diode across R1 as simulated in Pspice is shown in (Figure 8):

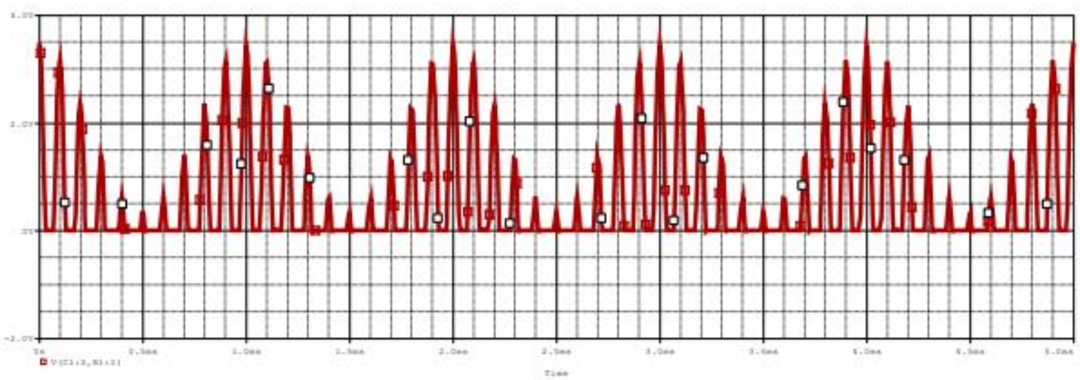


Figure 8: Voltage on R1

Signal as simulated in TinkerCad is shown in (Figure9), note that the signal will show on Oscilloscope 3:

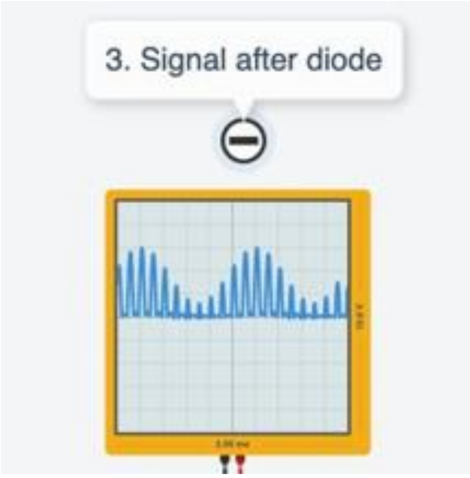


Figure 9: Result on TinkerCad

The graphical representations above showcase the voltage on R1 obtained from PSpice and TinkerCad simulations after the application of the Diode the close alignment between the two graphs reaffirms the consistent behavior of the circuit in both simulation tools.

3. Comparative Analysis of Band Pass Filter Effects (Oscilloscope 4 results)

In the course of simulating the circuit using both PSpice and TinkerCad, a thorough examination of the voltage on R2 values (post-band pass filtering) was conducted. The outcomes revealed a remarkable consistency in the voltage on R2 values between the two simulations, implying that the band pass filter had a parallel impact on the circuit's resistance in both instances. Nevertheless, it's crucial to recognize the possibility of slight variations in R2 values due to distinctions in simulation environments and the specific components employed in each tool.

Comparison Results:

Figure 10 illustrates the voltage on R2 values obtained from the PSpice simulation, while Figure 11 showcases the corresponding results from the TinkerCad simulation.

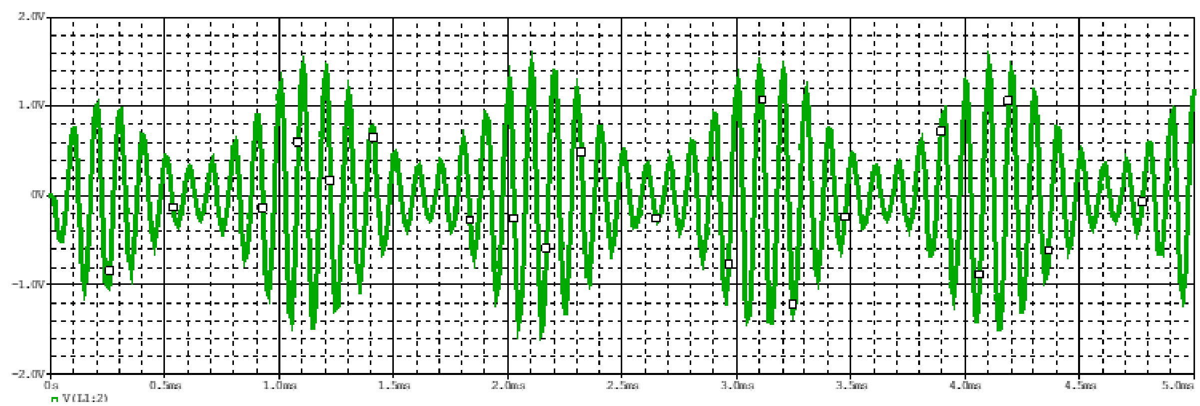


Figure 10: PSpice Simulation Result

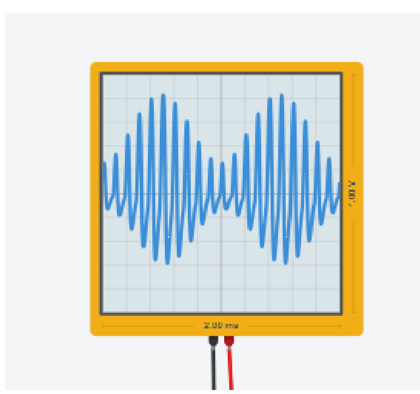


Figure 11: TinkerCad Simulation Result

The graphical representations above showcase the voltage on R2 obtained from PSpice and TinkerCad simulations after the application of the band pass filter. The close alignment between the two graphs reaffirms the consistent behavior of the circuit in both simulation tools. Any marginal variations observed could be attributed to inherent differences in simulation methodologies and component models.

These visual aids enhance the understanding of the comparative analysis and provide a clear illustration of the R2 values in each simulation, facilitating a comprehensive assessment of the circuit's behavior.

4. Comparative Analysis of Demodulated Signal (Oscilloscope 5 results)

The demodulated signal is the signal we get in the last stage after using the demodulator, which in this case in an envelope detector. (Figure 12) shows the signal that we obtained from the simulation on Pspice in phase1, we can notice the general shape of the signal, that follows the path (detects the envelope) of the modulated signal.

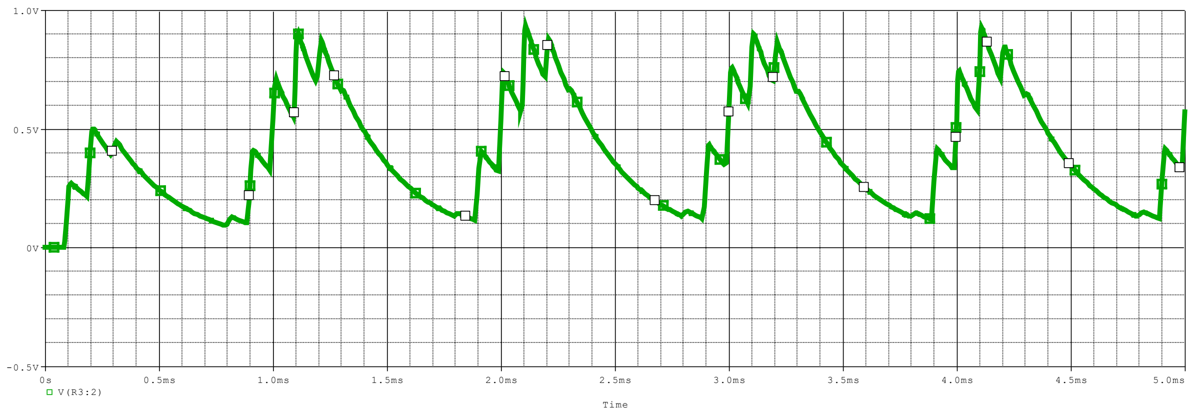


Figure 12: Demodulator output (Message after envelope detection)

Now if we compare the results, Figure 13 shows the output on oscilloscope 5 which is the demodulated signal. We can notice that the shape of the signal is very similar to the signal from Pspice. It is clear that it has those peaks and the points where it drops (capacitor discharge) then goes back up again (capacitor charges).

In both cases we notice that the demodulated signal is smaller in amplitude than original.

Because the outputs are very similar and are consistent with our simulation in Pspice we can conclude that our original design is correct and that the bread board circuit implementation is also correct.

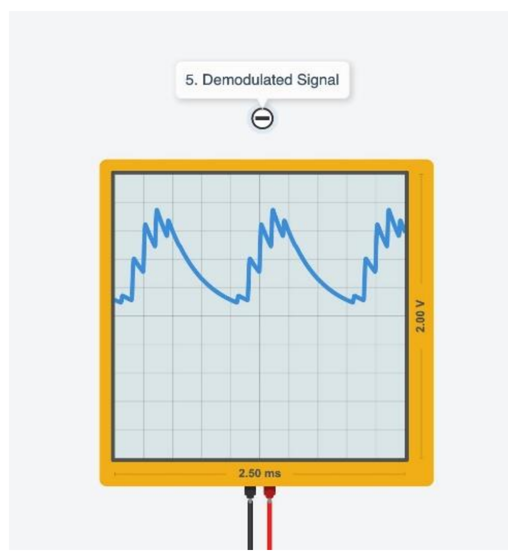
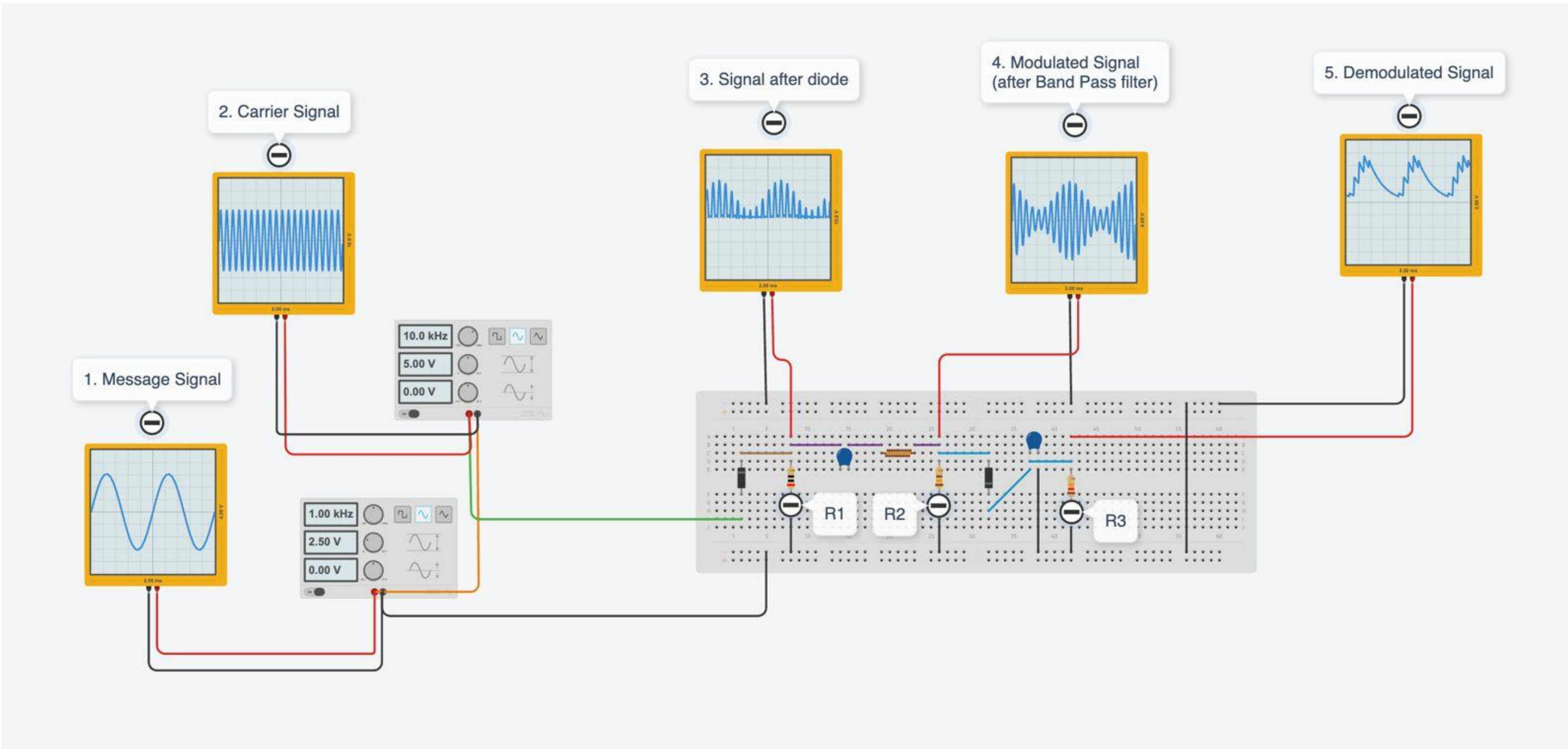


Figure 13: Demodulated Signal in TinkerCad

Appendix 1:

Breadboard Circuit after Simulation Enlarged:



Components Used in TinkerCad simulation:

Name	Quantity	Component
RR1	1	20 Ω Resistor
FUNCcarrier	1	10000 Hz, 5 V, 0 V, Sine Function Generator
D1 D2	2	Diode
FUNCmessage	1	1000 Hz, 2.5 V, 0 V, Sine Function Generator
C1 C2	2	25 nF Capacitor
L1	1	10 mH Inductor
RR2	1	125.66 Ω Resistor
RR3	1	12 kΩ Resistor