EEE205L

Electronic Circuits I Laboratory

Lab Report

Section: 05

Group No:**04**

Experiment no: 01 (Software)

Name of the experiment: *Experimental Study on the Forward Bias Behavior of a PN Junction Diode and the Reverse Breakdown of a Zener Diode (PSPICE Simulation)*

Prepared by:

Name: Tanzeel Ahmed **ID: 24321367** **Signature**

Tanzeel

All Group members:

Sl.	Name	ID	Signature
1.	**Esrar Ul Hossain Rafin**	**24121187**	*Esrar*
2.	**Tasmiya Tahsin Roza**	**24121339**	*Tasniya*

Date of Submission: 26/08/2025

Objective:

This experiment is done to understand how the electric current and voltage behave in a silicon p-n junction and zener diode in a simulation environment.

Theoretical Background:

A diode is a two-terminal component that allows current to pass through in one direction but not the other. In figure 1.0, we can see the terminals: the anode, through which current is allowed to pass, and the cathode, through which it is blocked. A diode is made using two different materials and consists of a p-n junction, formed by joining p-type material (rich in holes, or positive charge carriers) and n-type material (rich in electrons, or negative charge carriers).

When the positive terminal of a battery is connected to the p-side and the negative terminal to the n-side, the diode becomes forward-biased and allows current to flow. In contrast, when the connections are reversed, the diode is reverse-biased and prevents current from passing. A diode is therefore commonly used to gain control over signals.

A special type of diode is the Zener diode. Unlike a regular diode, a Zener diode not only allows current to flow in the forward direction but is also designed to conduct in the reverse direction when the applied voltage reaches a specific value known as the Zener breakdown voltage. This property makes Zener diodes very useful for voltage regulation, as they can maintain a stable output voltage despite variations in the input voltage or load conditions.

PSPICE is one of the most commonly used circuit simulation tools. It allows users to design electronic circuits on a computer, apply test signals, and visualize the circuit response through graphs and plots. By simulating p-n junction and zener diode circuits in PSPICE, we can understand its characteristics easily in multiple scenarios.

Problem Statement-1 (P-N Junction diode)

To observe and analyze the I-V characteristics of a silicon p-n junction diode under forward and reverse bias conditions.

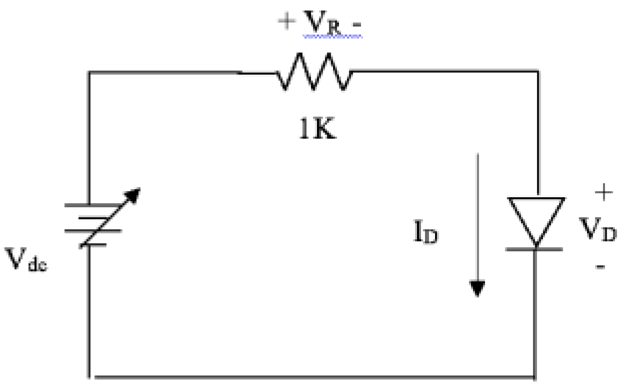
List of equipments:

1. Suitable computer or laptop that meets the minimum requirements
2. ORCAD Pspice Schematics software

List of equipments required in simulation:

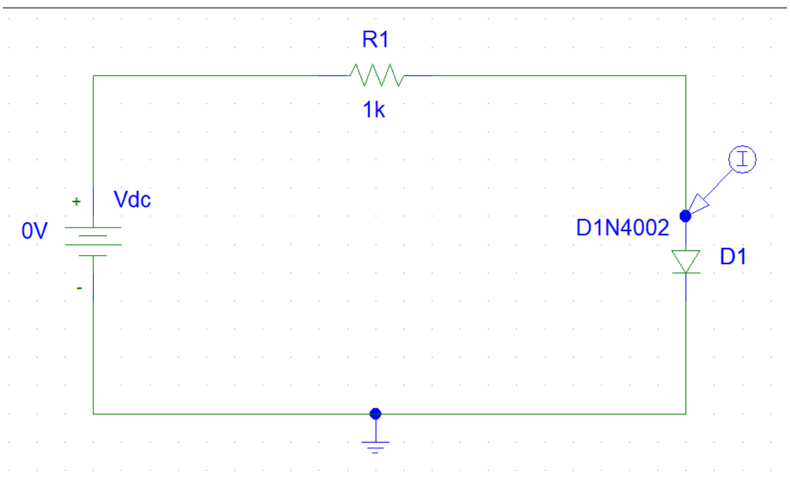
1. Resistor (R) - 1K-ohm - 1x
2. p-n junction diode D1N4002 - 1x
3. DC Voltage source (VDC) Vs =1x
4. Current level marker

Circuit diagram (from manual):



Circuit diagram for diode characteristics

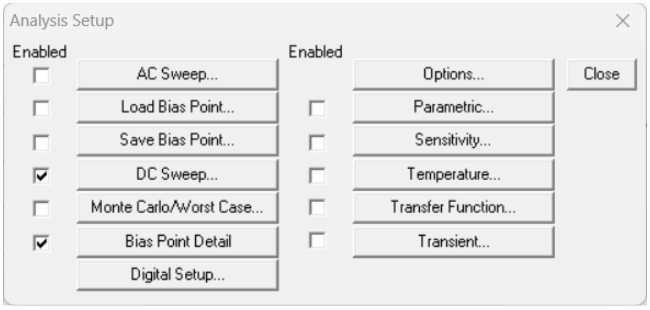
PSPICE Schematic diagram:



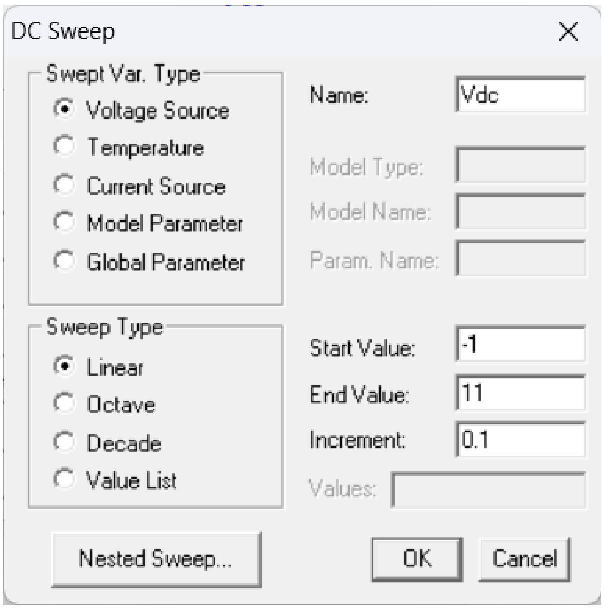
Circuit diagram for diode characteristics in simulation

PSPICE Simulation settings:

Analysis setup menu:

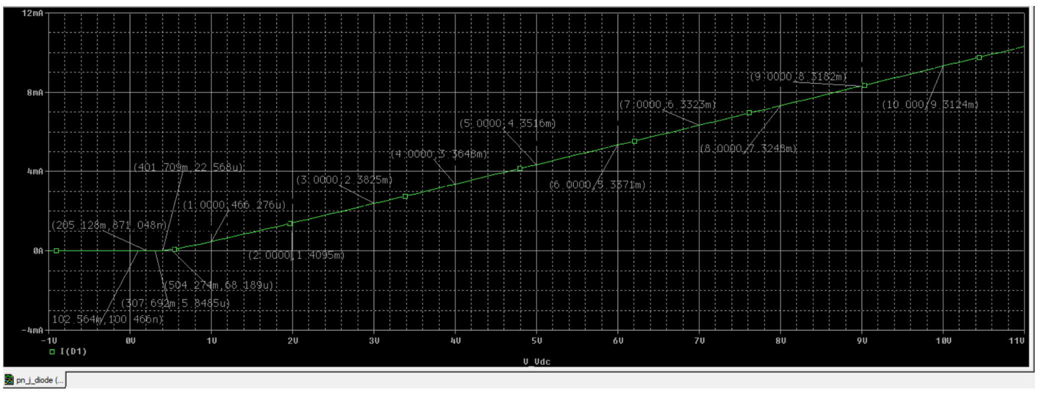


DC Sweep:

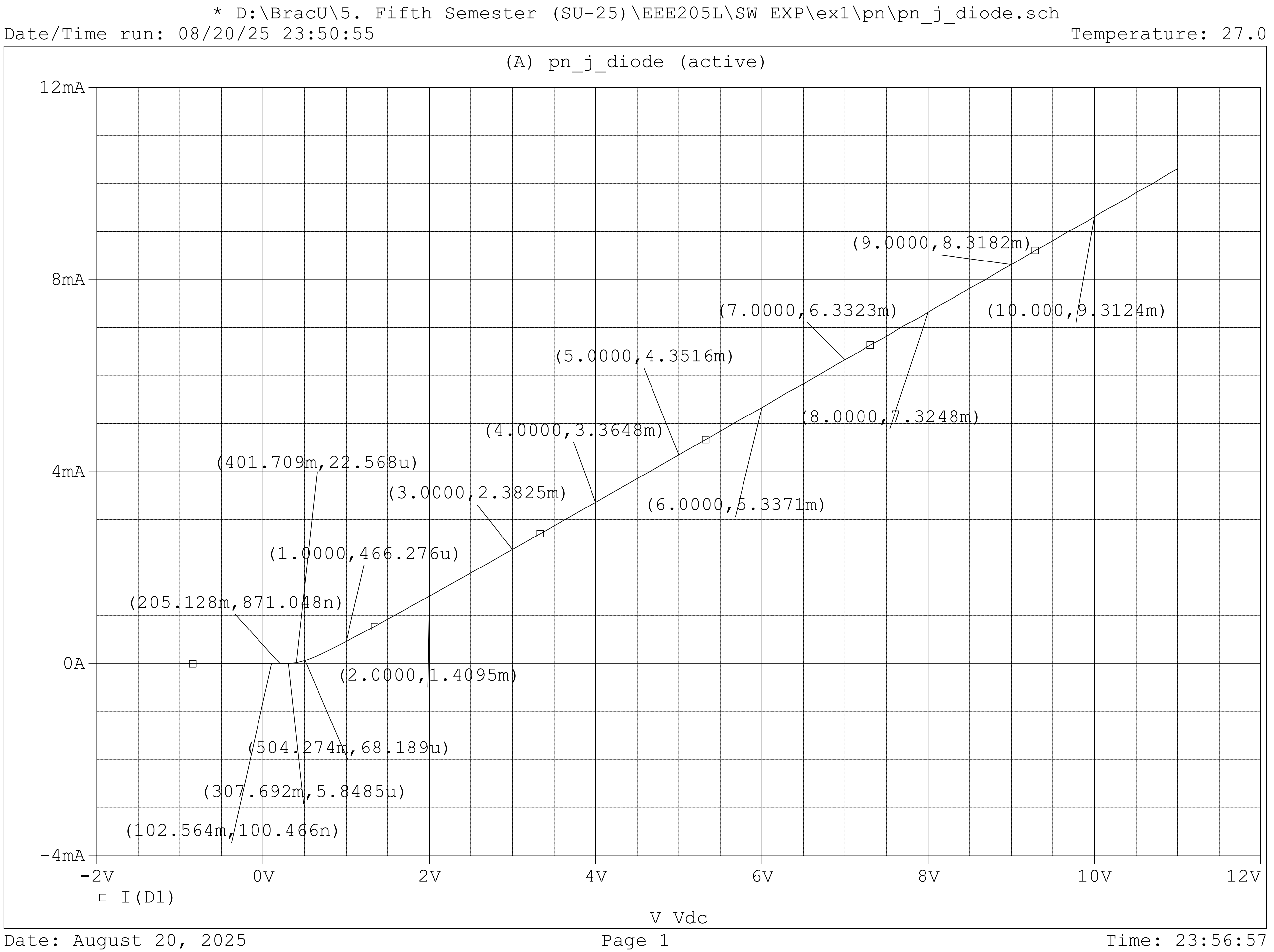


Name: Vdc, Start Value -1, End Value 11, Increment 0.1

Output waveforms:



Diode I-V characteristics curve with points marked



Diode I-V plot simulated in Pspice schematics

Data table (Simulated):

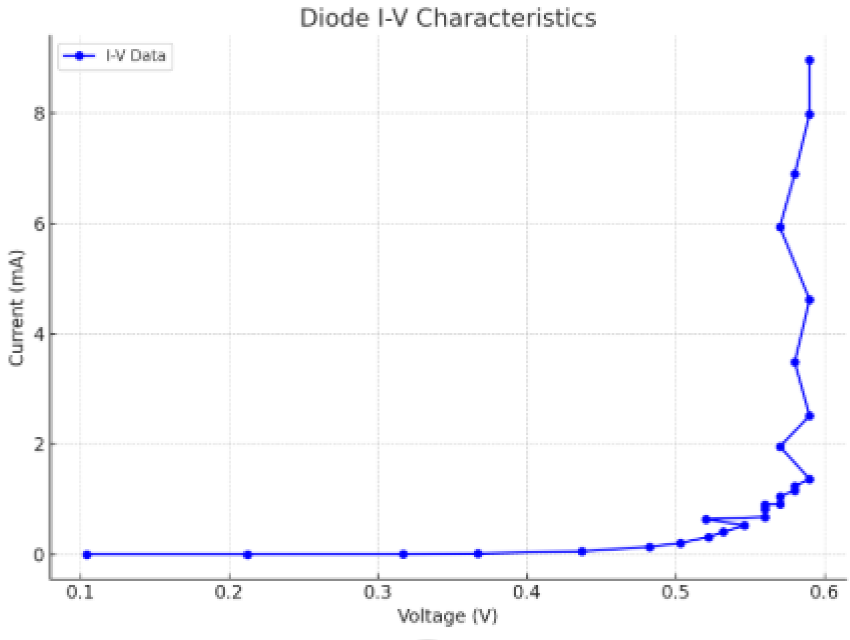
Number of Observation	Vdc (V)	I_D (nA-mA)
1	0.1	100.466nA
2	0.2	871.48nA
3	0.3	5.8485uA
4	0.4	22.568uA
5	0.5	68.189uA
6	1	466.27uA
7	2	1.4095mA
8	3	2.3825mA
9	4	3.3648mA
10	5	4.3516mA
11	6	5.3371mA
12	7	6.3323mA
13	8	7.3248mA
14	9	8.3182mA
15	10	9.3124mA

Data table (From hardware experiment):

No. of Observation	Vdc (V) theoretical	VD (V)	VR (V)	ID = VR/R (mA)
1	0.1	0.104	0.0	0.0000
2	0.2	0.212	0.0	0.0000
3	0.3	0.317	0.002	0.0020
4	0.4	0.367	0.015	0.0151

5	0.5	0.437	0.056	0.0567
6	0.6	0.482	0.133	0.1347
7	0.7	0.503	0.203	0.2056
8	0.8	0.522	0.311	0.315
9	0.9	0.532	0.400	0.405
10	1.0	0.546	0.519	0.525
11	1.1	0.520	0.63	0.638
12	1.2	0.560	0.67	0.678
13	1.3	0.560	0.81	0.82
14	1.4	0.560	0.89	0.902
15	1.5	0.570	0.90	0.911
16	1.6	0.570	1.04	1.053
17	1.7	0.580	1.14	1.155
18	1.8	0.580	1.22	1.236
19	2.0	0.590	1.35	1.36
20	3.0	0.570	1.93	1.955
21	4.0	0.590	2.48	2.513
22	5.0	0.580	3.44	3.485
23	6.0	0.590	4.56	4.62
24	7.0	0.570	5.86	5.937
25	8.0	0.580	6.81	6.899
26	9.0	0.590	7.88	7.98
27	10.0	0.590	8.85	8.96

Hardware experiment plot:



Diode I-V characteristics plot from hardware report

Comparison:

In the simulated graph, we can see that the diode started to show current in the uA range from 0.3 volts which is 5.8485uA. We are considering this as the activation voltage because the previous values of current in 0.1V and 0.2V are in the nA range which is really small to consider. The diode retains uA currents until we reach 2 volts where it goes into mA range which is 1.4095mA. After that the current increases gradually as the voltage is increased. We stopped the simulation at 10V where the current was 9.3124mA.

In the hardware experiment table, we can see that the diode started to show current 0.3 volts which is 0.0020mA. We are considering this as the activation voltage because the previous values of 0.1 and 0.2 are 0mA The diode retains this range until we reach 1.6 volts where it goes into 1.053mA. After that the current increases gradually as the voltage is increased. We stopped the experiment at 10V where the current was 8.96mA.

So, the threshold voltage is the same. Current amount difference at 0.3V = (5.8485uA - 0.0020mA) = 0.003848mA

Current difference at peak 10V = (9.3124 - 8.96)mA = 0.352mA

Discussion:

The circuit was constructed according to the diagram. In the hardware experiment we used multiple values but in simulation, voltage level 0.6 to 0.9 was excluded because of really low current value. The diode (D1N4002) was forward biased. It was also connected in series with a resistor. After a certain amount of time, we can see in both cases that current increased exponentially but voltage was almost constant which created the curve. There is a small amount of difference in the experimental and simulation current because in the lab we usually have external factors like heat which contributes to some amount of loss. The simulation still matches the values of hardware experiment closely and the graph also stays consistent according to theoretical values as well.

Problem Statement-2 (ZENER diode)

To observe and analyze the characteristics of a zener diode.

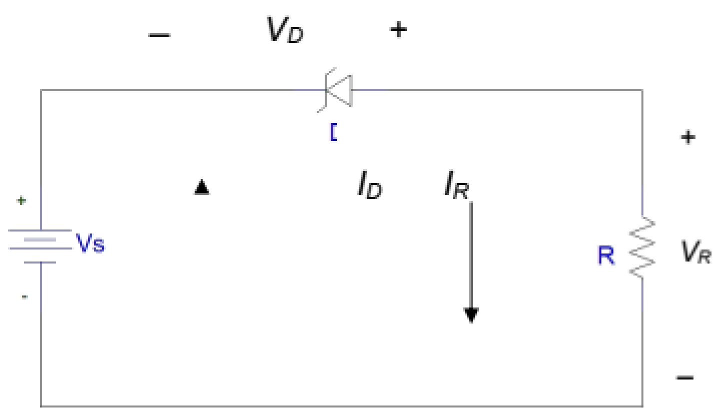
List of equipments:

1. Suitable computer or laptop that meets the minimum requirements
2. ORCAD Pspice Schematics software

List of equipments required in simulation:

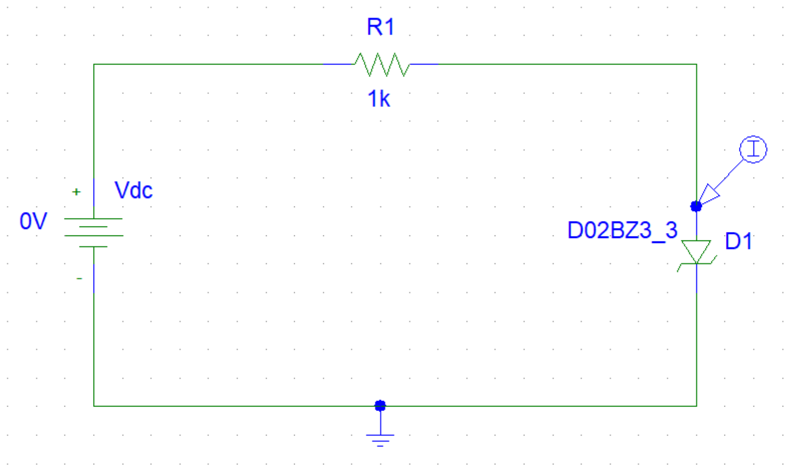
1. Resistor (R) - 1K-ohm - 1x
2. Zener diode (D02BZ3_3) - 1x
3. DC Voltage source (VDC) Vs =1x
4. Current level marker

Circuit diagram (from manual):



Circuit diagram for zener diode characteristics

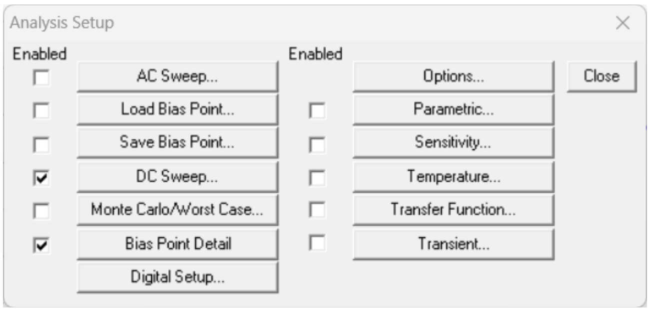
PSPICE Schematic diagram:



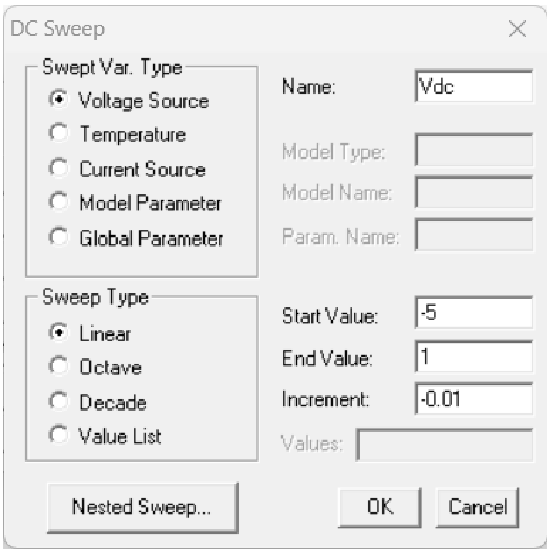
Circuit diagram for zener diode characteristics in simulation

PSPICE Simulation settings:

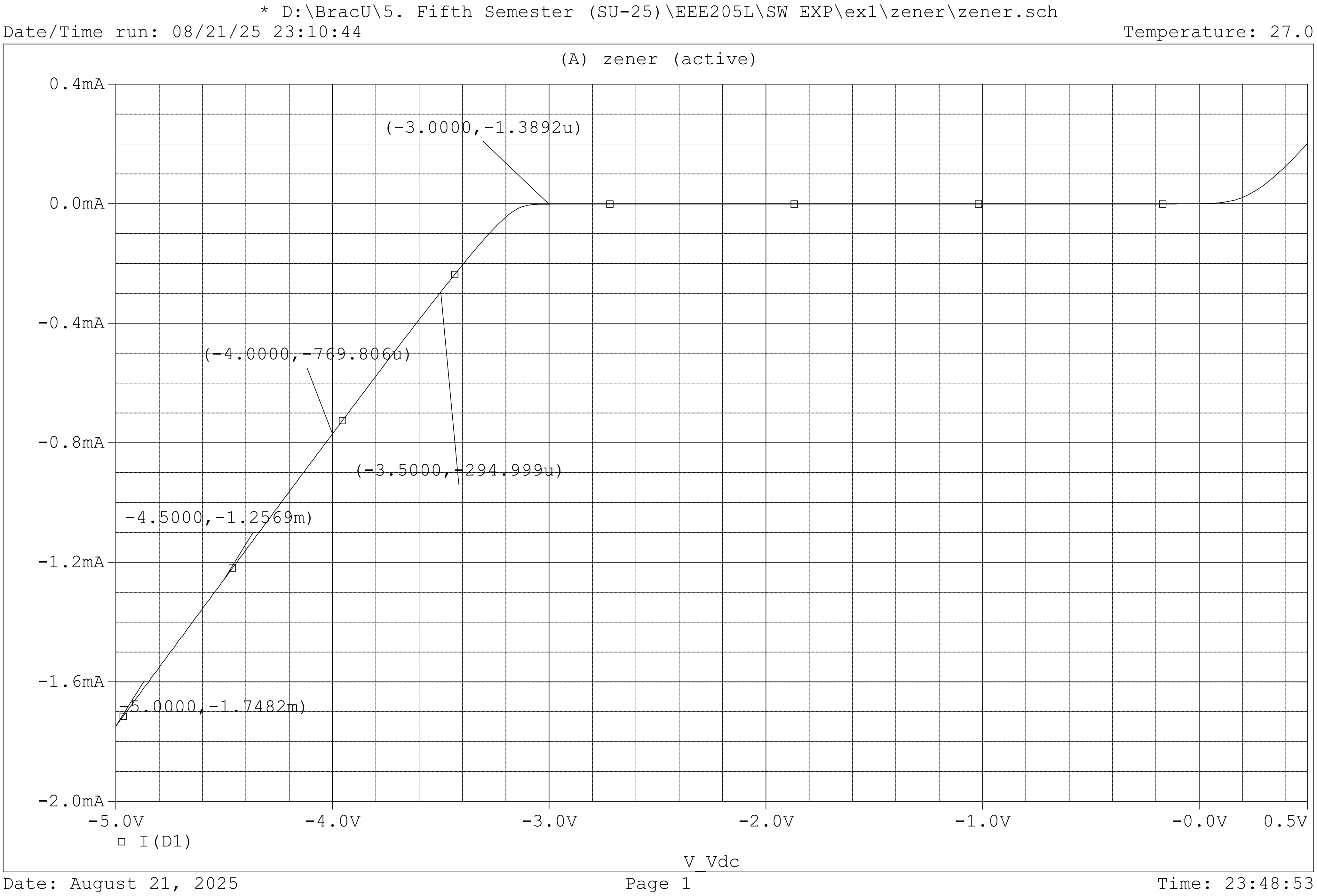
Analysis setup menu:



DC Sweep:

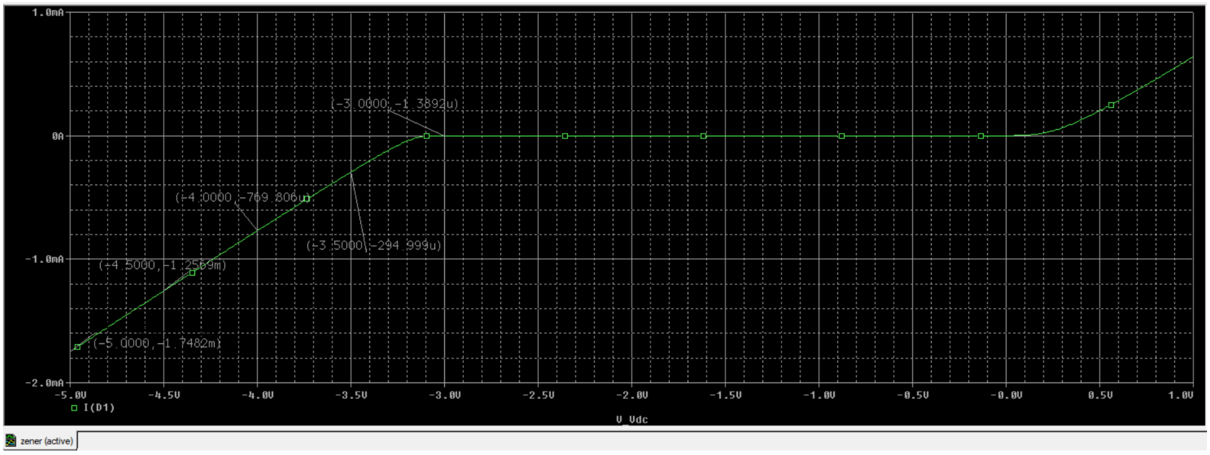


Name: Vdc, Start Value -5, End Value 1, Increment -0.01



ZENER Diode plot simulated in Pspice schematics

Output waveforms:



Zener diode I-V characteristics curve with points marked

Data table (Simulated):

Number of Observation	Vdc (V)	I_D
1	-3	-1.3892uA
2	-3.5	-294.99uA
3	-4	-769.806uA
4	-4.5	-1.2569mA
5	5	-1.7482mA

Data table (From hardware experiment):

The hardware experiment for zener diode was not conducted in EEE205-L section 5. For this reason, theoretical values are used as substitutes for the missing data table.

Case Calculations (using $V_Z = 3.3V$, $R = 1k\Omega$)

1. $V_{dc} = -3.0V$
   $|V_{dc}| < V_Z$ → No breakdown →

$$I \approx 0\,A$$

2. $V_{dc} = -3.5V$
   Breakdown occurs →

$$I = \frac{3.5 - 3.3}{1000} = 0.0002\,A = 0.2\,mA$$

3. $V_{dc} = -4.0V$

$$I = \frac{4.0 - 3.3}{1000} = 0.0007\,A = 0.7\,mA$$

4. $V_{dc} = -4.5V$

$$I = \frac{4.5 - 3.3}{1000} = 0.0012\,A = 1.2\,mA$$

5. $V_{dc} = -5.0V$

$$I = \frac{5.0 - 3.3}{1000} = 0.0017\,A = 1.7\,mA$$

If we define the diode current positive from anode to cathode (the usual device convention), then a Zener in reverse breakdown conducts from cathode to anode, so I_D will be negative.

Number of Observation	Vdc (V)	I_D (Theoretical)
1	-3	0mA
2	-3.5	-0.2mA
3	-4	-0.7mA
4	-4.5	-1.2mA
5	5	-1.7mA

Comparison:

In the simulated graph, we can see that the zener diode started to show current in the uA range from -3 volts which is -1.3892uA. The zener diode retains uA currents until -4.5 volts where it changes into -1.2596mA. After that the negative current increases gradually as the voltage is increased. We stopped the simulation at -5V where the current was -1.7482mA.

In the hardware experiment table, since we used theoretical values, nA is too small to consider as usable negative current so at -3 volts the current is determined 0mA. At -3.5volts the zener diode activates and gives us -0.2mA. It goes into -1.2mA when the voltage is -4.5 and at last during -5V it reaches -1.7mA.

Since the simulated values and theoretical-experimental values are identical, there is no comparison.

Discussion:

The circuit was constructed according to the diagram. Since there is no reference hardware experiment we only used voltage values from -3.0 to -5.0 since the output waveform shows a flatline when tracing positive values from -3.0. The zener diode (D02BZ3_3) was connected in series with a resistor. After a certain amount of time, we can see in both cases that negative current increases exponentially. The simulation accurately matches the values of theoretical calculations and the graph also stays consistent according to theoretical values as well.

To conclude, both simulations were successful and we were able to achieve a basic understanding of how P-N junction and Zener diodes work in various electronic circuits.

Drive links (.sch files):

P-N Junction

Zener

EEE205L

Electronic Circuits I Laboratory

Lab Report

Section: 05

Group No:**04**

Experiment no: 02 (Software)

Name of the experiment: *Implementation of Half-wave/Full-wave rectifier circuits (PSPICE Simulation)*

Prepared by:

Name: Tanzeel Ahmed **ID: 24321367** **Signature**

Tanzeel

All Group members:

Sl.	**Name**	**ID**	**Signature**
1.	**Esrar Ul Hossain Rafin**	**24121187**	*Esrar*
2.	**Tasmiya Tahsin Roza**	**24121339**	*Tasniya*

Date of Submission: 26/08/2025

Objective:

This experiment is done to learn how to construct half and full wave diode rectifier and understand how they work in a circuit by simulation using PSPICE.

Theoretical Background:

Rectification is the process of converting an alternating current (AC) signal, which alternates in polarity, into a direct current (DC) signal, which has a single polarity. Diode rectifier circuits use the unidirectional conduction property of diodes to achieve this conversion. When an AC voltage is applied to a diode, the diode conducts only during the half-cycles where it is forward-biased, blocking current during the reverse-biased half-cycles. There are two categories of rectifier circuits:

1. Half wave rectifier: This uses a single diode to convert only one half of an AC signal into a pulsating DC output.
2. Full wave rectifier: This uses multiple diodes to convert both halves of an AC signal into a continuous pulsating DC output.

In modern circuit analysis, simulation is widely used to predict and study the behavior of electronic circuits before actual implementation. Simulation provides a time efficient way to test different circuit configurations and verify theoretical calculations.

PSPICE is one of the most commonly used circuit simulation tools. It allows users to design electronic circuits on a computer, apply test signals, and visualize the circuit response through graphs and plots. By simulating diode rectifiers in PSPICE, we can clearly observe the input and output waveforms, understand the rectification process without much hassle.

Problem Statement-1 (Half-wave rectifier)

To observe and analyze how one half of an AC signal transforms into DC output in Half-wave rectification.

List of equipments:

1. A computer or laptop that meets the minimum requirements
2. ORCAD Pspice Schematics software

List of equipments required in simulation:

1. Resistor (R_L) - 1K-ohm - 1x
2. VSIN source - 1x

3. p-n junction diode D1N4002 - 1x
4. Capacitors (C) 1uF and 47uF
5. Voltage level markers

Circuit diagram (from manual):

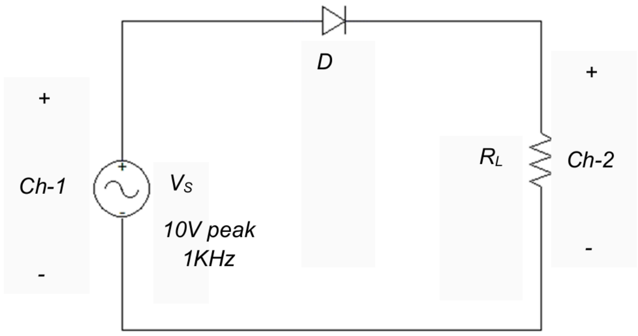


Figure 1: Circuit diagram for half-wave rectifier

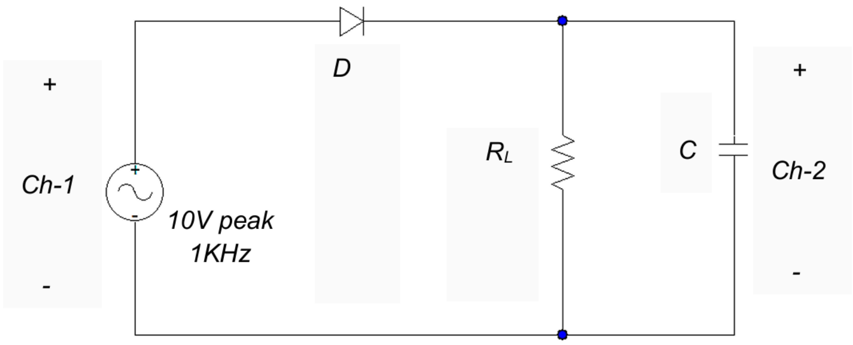
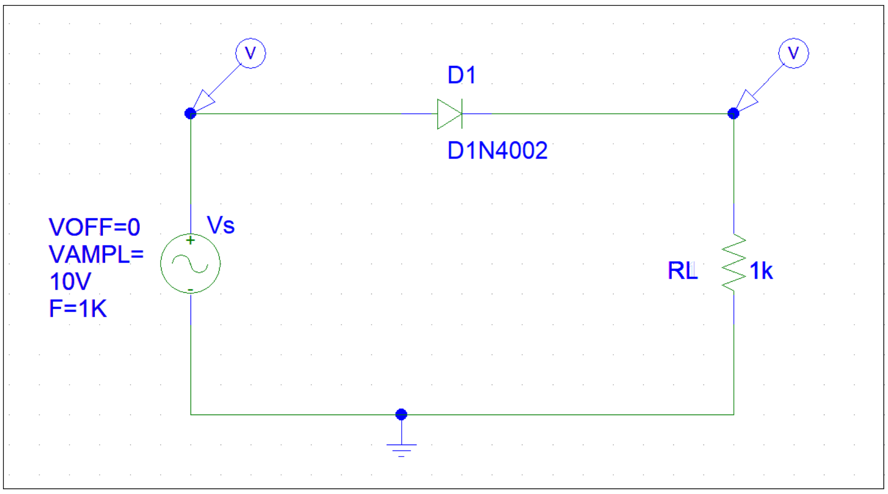
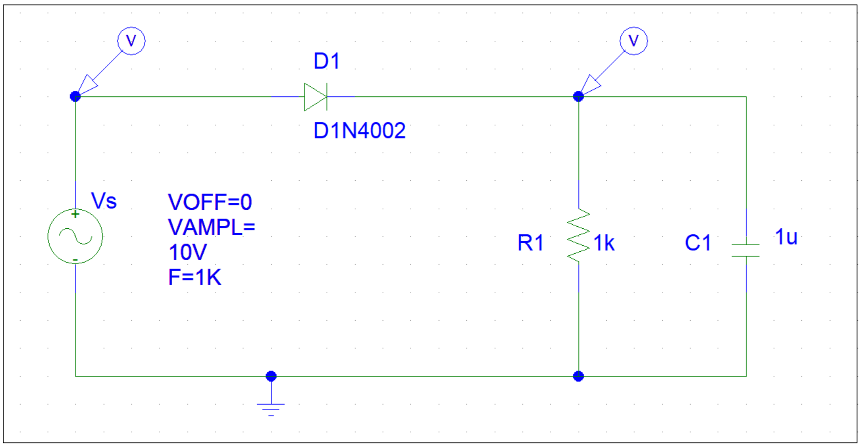


Figure 2: Half wave rectifier with filter

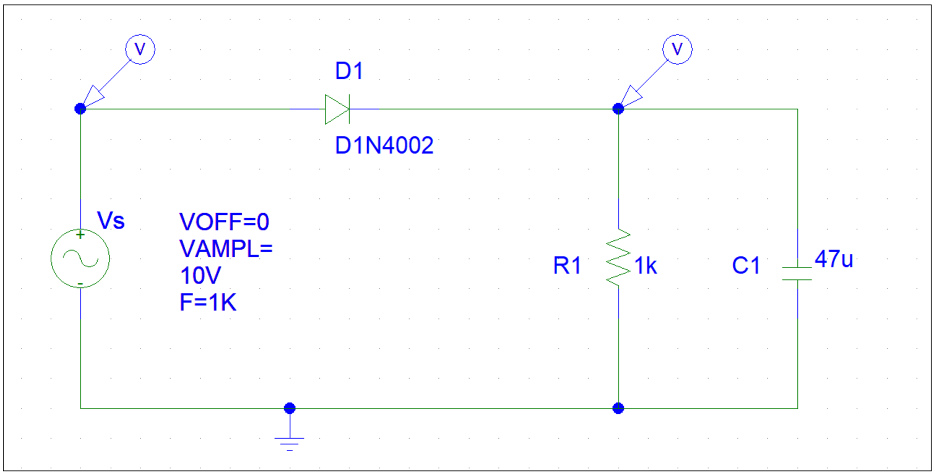
PSPICE Schematic diagram:



Half-wave rectifier without filter in simulation



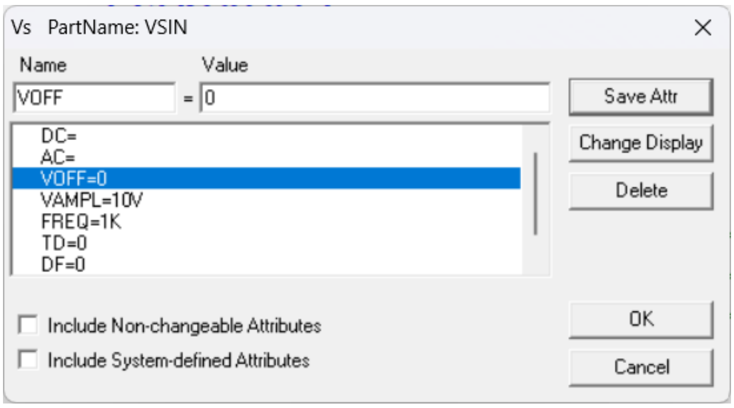
Half-wave rectifier with 1uF capacitor simulation



Half-wave rectifier with 47uF capacitor simulation

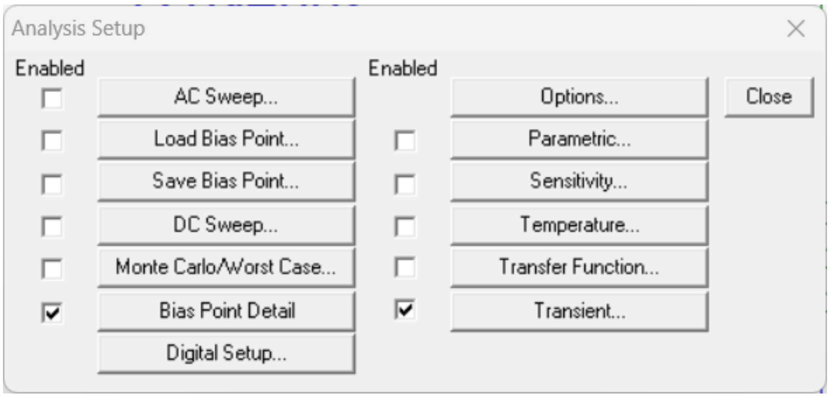
PSPICE Simulation settings:

VSIN (Vs):

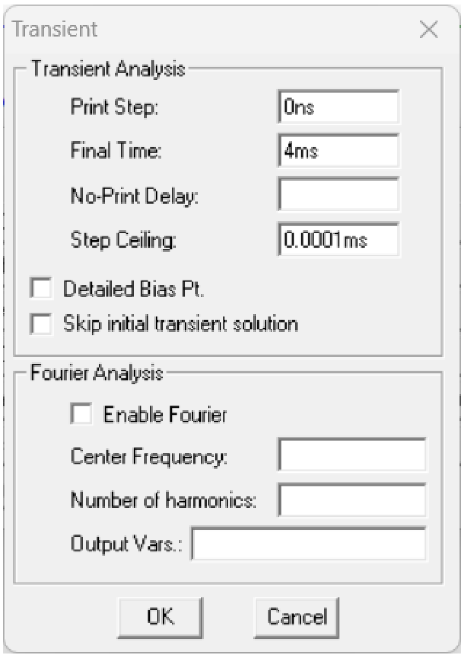


VOFF=0, VAMPL=10V, FREQ=1K

Analysis setup menu:

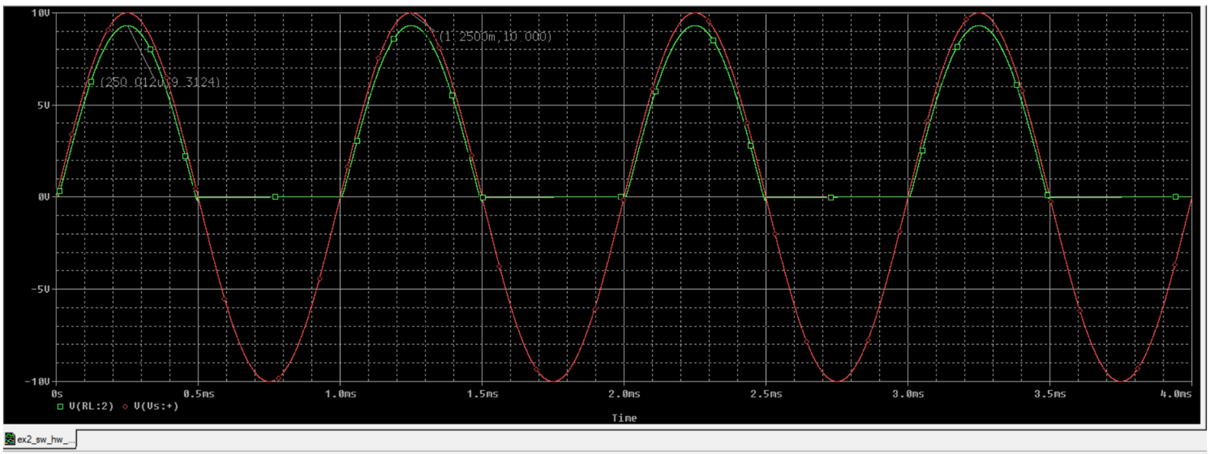


Transient:

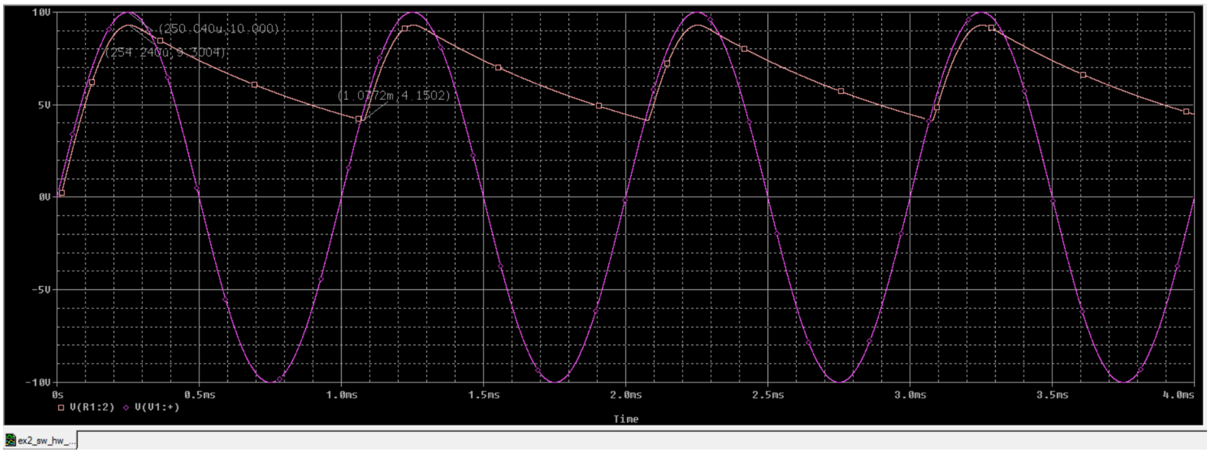


Print Step: 0ns, Final Time: 4ms, Step Ceiling: 0.0001ms

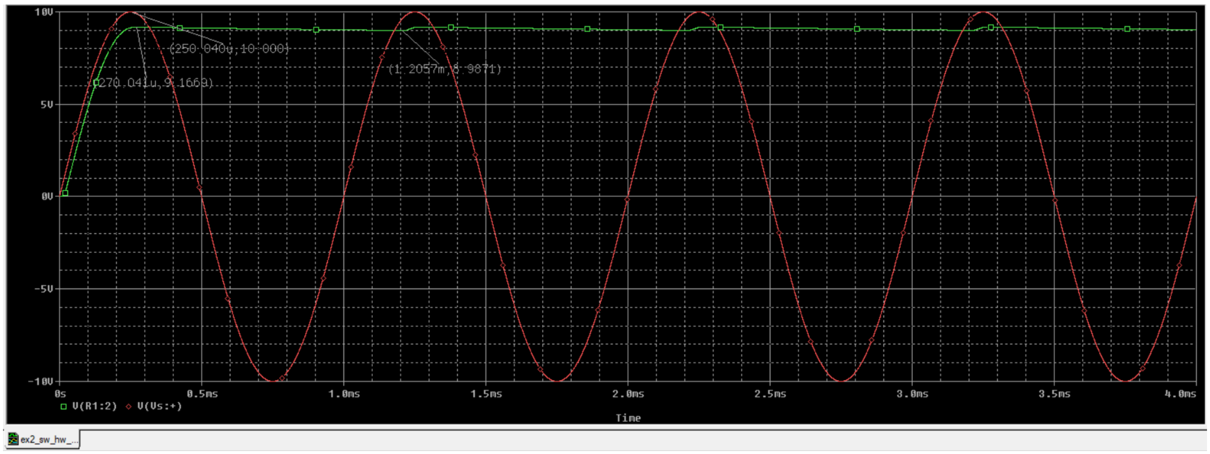
Output waveforms:



Half-wave rectification waveform without capacitor



Half-wave rectification waveform with 1uF capacitor



Half-wave rectification waveform with 47uF capacitor

Data table (Simulated):

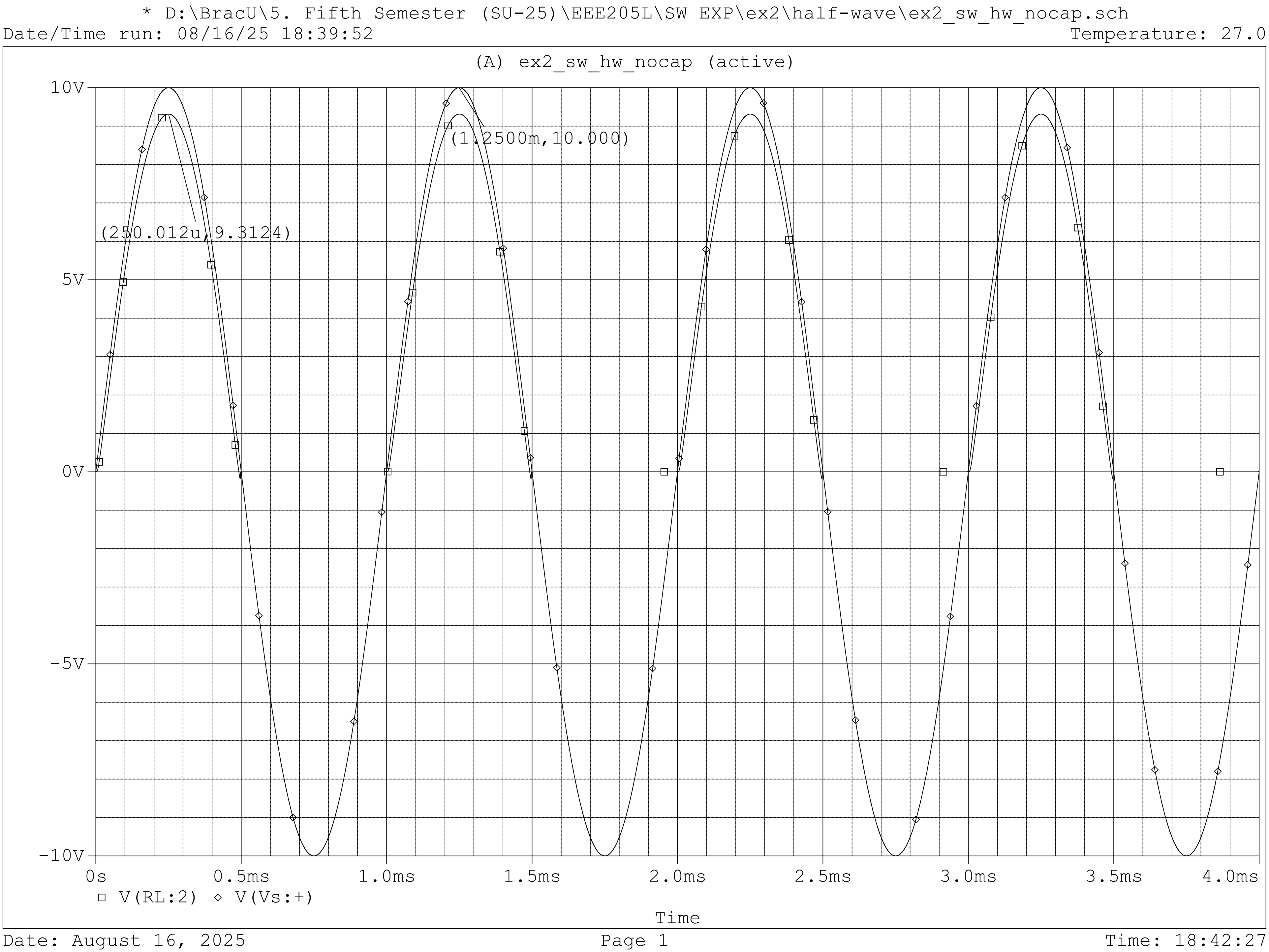
VSIN = 10V max
RL = 1k-ohm

Without capacitor:

Vp = 9.3124V (Voltage across resistor)

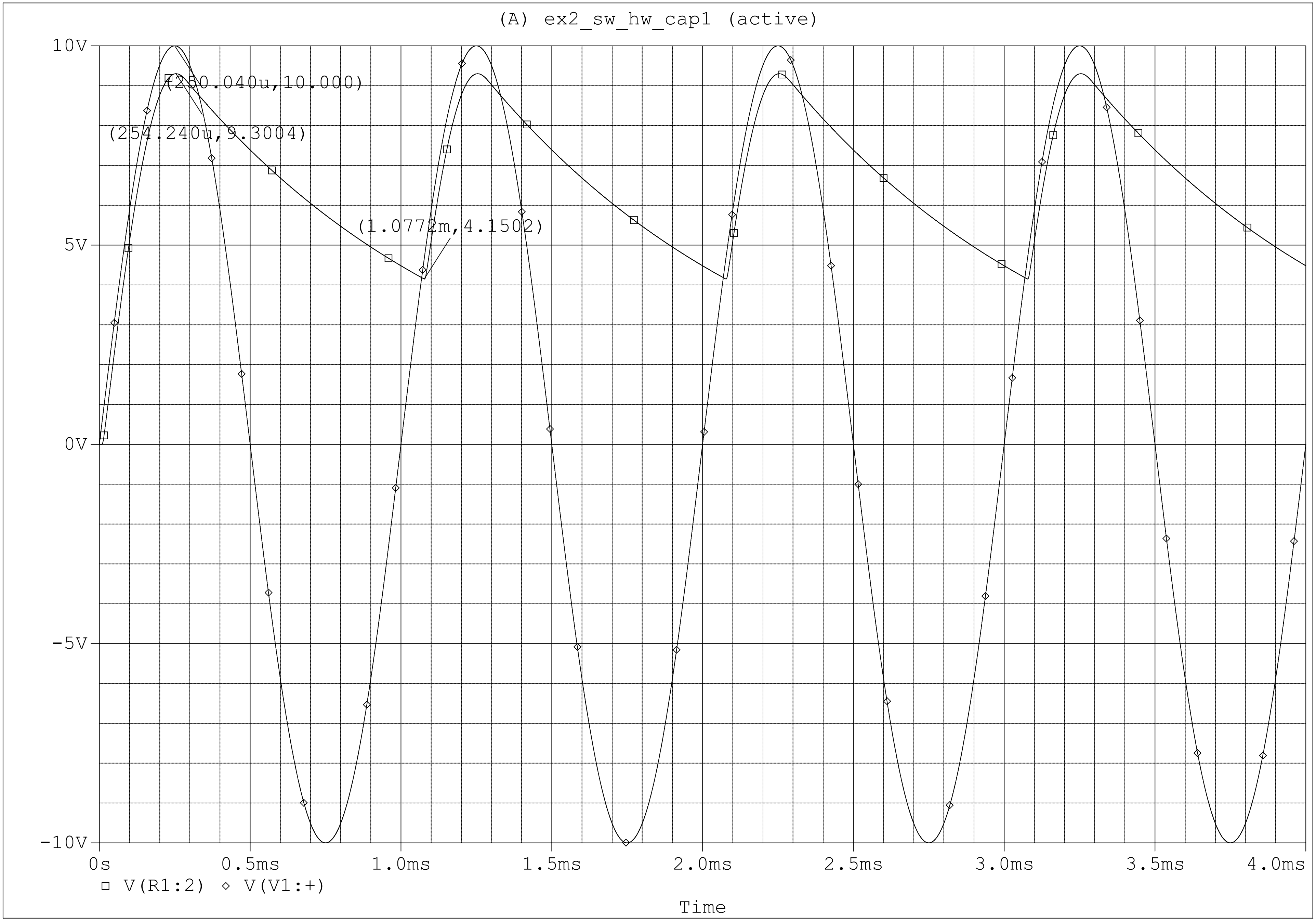
With capacitor:

Capacitance (uF)	Ripple voltage, $V_{R(p-p)}$ (Volts)
1	(9.3004-4.1502) = 5.1502
47	(9.1669-8.9871) = 0.1798



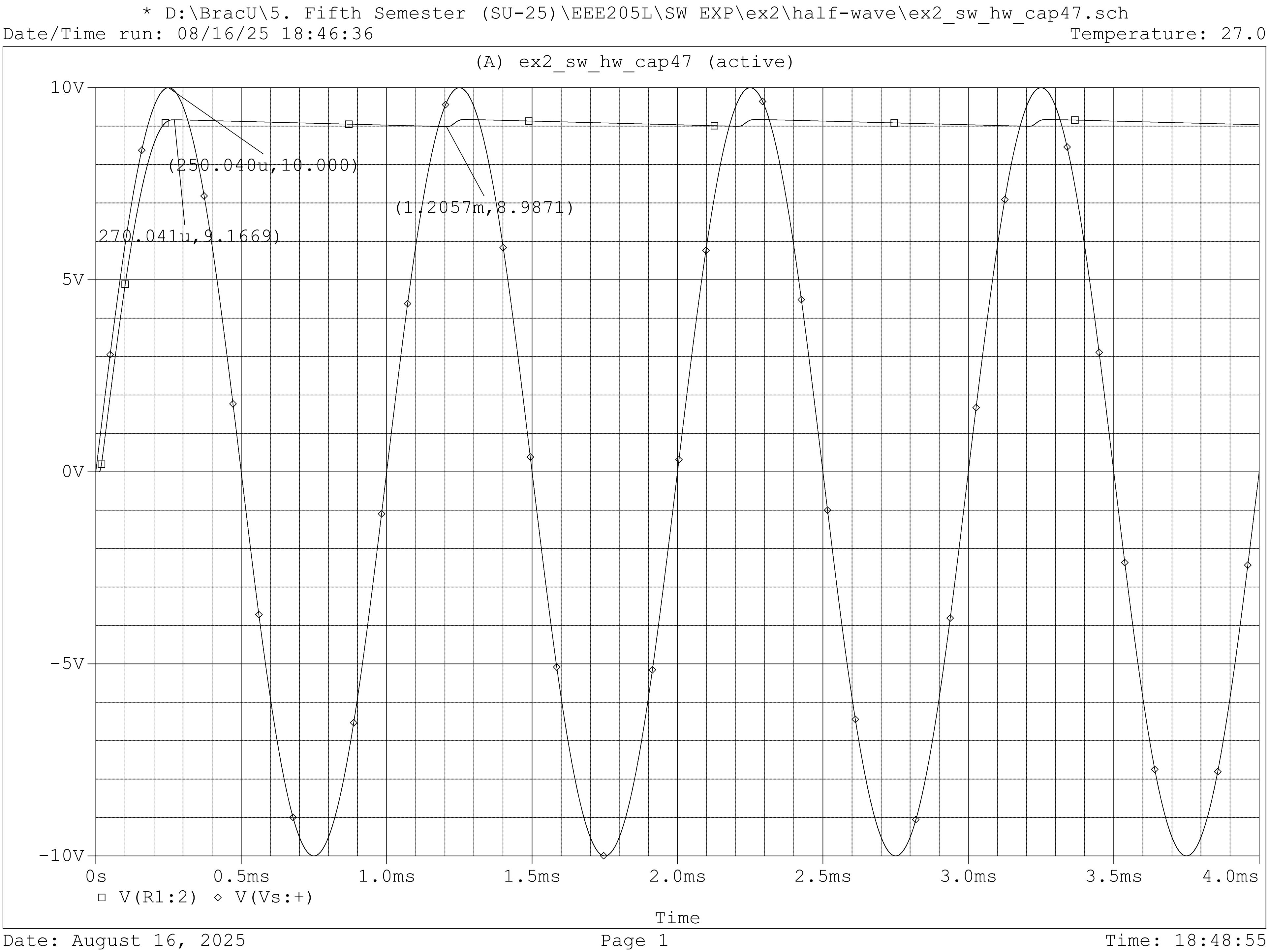
Half-wave rectification without filter

* D:\BracU\5. Fifth Semester (SU-25)\EEE205L\SW EXP\ex2\ex2_sw_hw_cap1.sch
Date/Time run: 08/16/25 18:43:15 Temperature: 27.0



Date: August 16, 2025 Page 1 Time: 18:46:07

Half-wave rectification with 1uF capacitor



Half-wave rectification with 47uF capacitor

Data table (From hardware experiment):

RL= 0.986k-ohm

V_{DO}= 0.86V

Without capacitor:

Vp= 9.20V | V_{DC}=2.82V | V_{RMS}= 3.49

With capacitor:

Capacitance (uF)	Ripple voltage, $V_{R(p-p)}$ (Volts)	V_{DC}	V_{RMS}
1	5.40	6.03V	1.40V
47	0.169	6.92V	0.09V

Comparison:

Simulated voltage through resistor: 9.3124V
Hardware experiment value: 9.20V
Difference: (9.3124-9.20) = 0.112V

Simulated ripple voltage for 1uF: 5.1502V
Hardware experiment value: 5.40V
Difference: (5.40-5.1502) = 0.25V

Simulated ripple voltage for 47uF: 0.1798V
Hardware experiment value: 0.169V
Difference: (0.1798-0.169) = 0.01V

Discussion:

In this simulation, we completed a previously done hardware experiment on a Half-wave rectifier circuit. Half-wave rectifiers only pass the positive half of an AC signal. During the positive half-cycle, the diode becomes forward-biased and conducts, allowing the capacitor to charge up to the peak voltage minus the voltage drop through a diode. Part number D1N4002 has been used for that. During the negative half-cycle, the diode is reverse-biased and blocks current, so the capacitor discharges through the load resistor. This results in a pulsating DC output with significant ripple, especially when using a small capacitor like 1 μF. A larger capacitor, such as 47 μF, reduces this ripple by holding the charge longer between cycles. Comparing the simulated results with hardware values, we can see a small difference which is within the acceptable range of errors. The simulated graphs stay consistent with theoretical and hardware models as well.

Problem Statement-2 (Full-wave bridge rectifier)

To observe and analyze how both sides of an AC signal can transform into DC output in Full-wave bridge rectification using multiple diodes simulated in PSPICE.

List of equipments required:

1. Resistor (R_L) - 1K-ohm - 1x
2. VSIN source - 1x
3. p-n junction diode D1N4002 - 4x
4. Capacitors (C) 1uF and 47uF
5. Voltage level markers
6. Voltage differential markers

Circuit diagram (from manual):

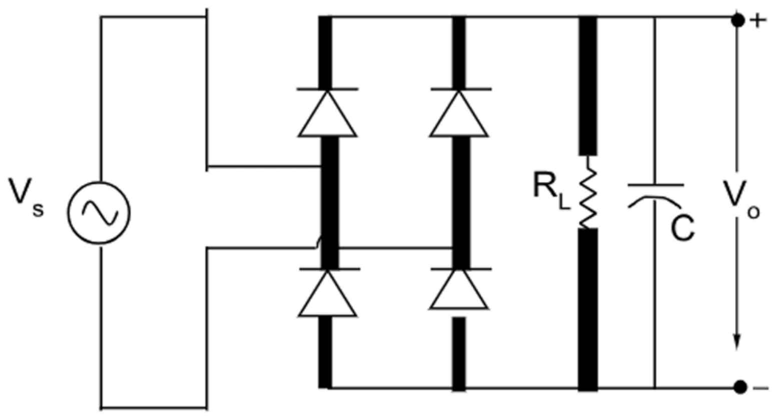
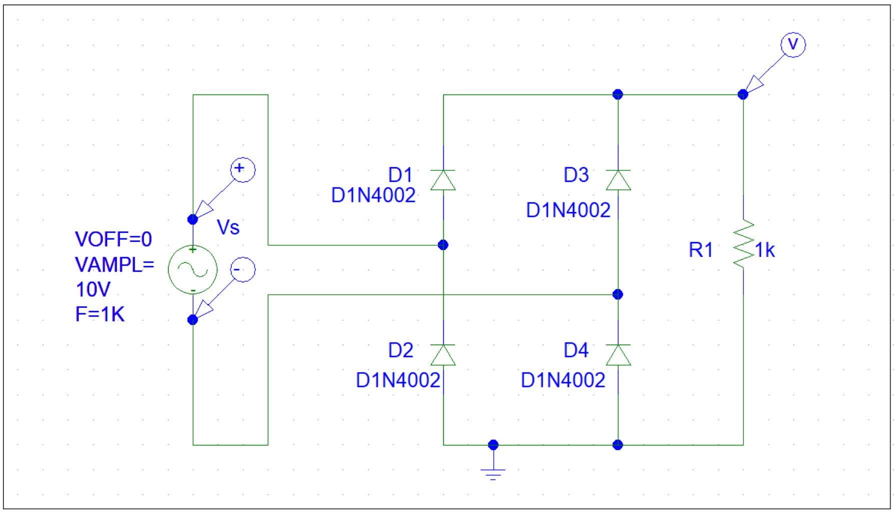
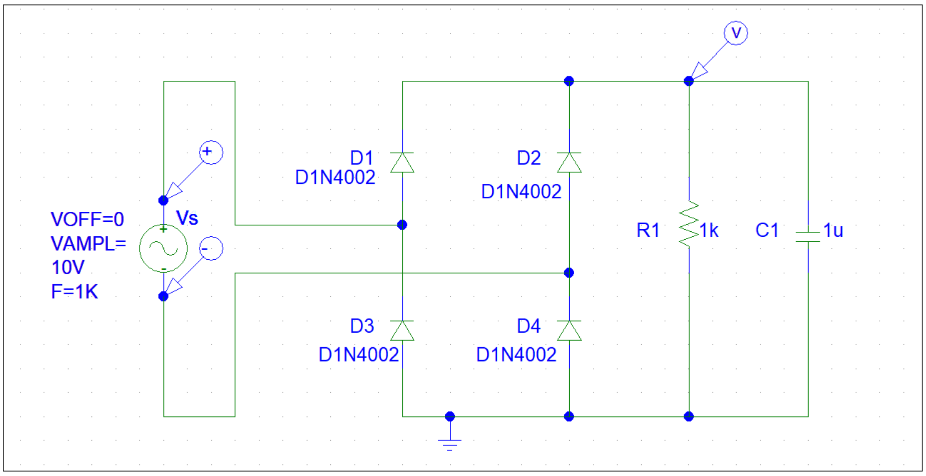


Figure: Circuit diagram for Full-wave bridge rectifier

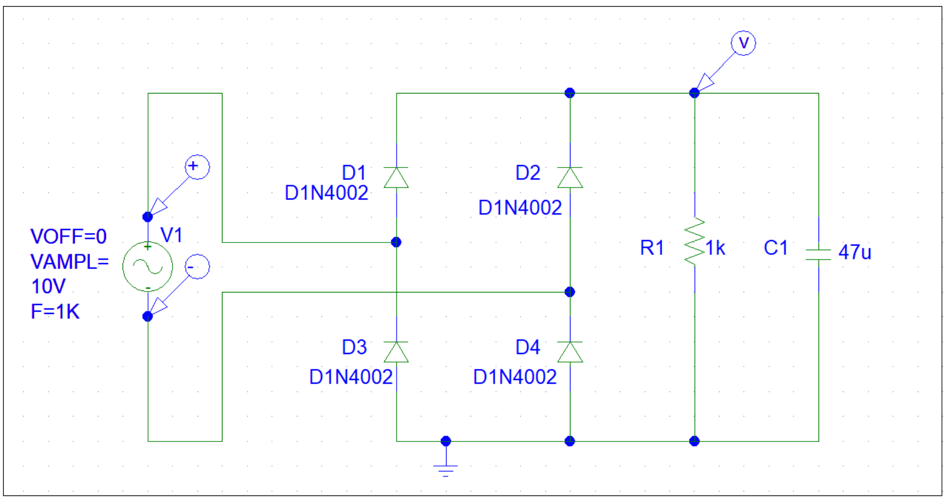
PSPICE Schematic diagram:



Full-wave bridge rectifier without filter in simulation



Full-wave bridge rectifier with 1uF capacitor in simulation

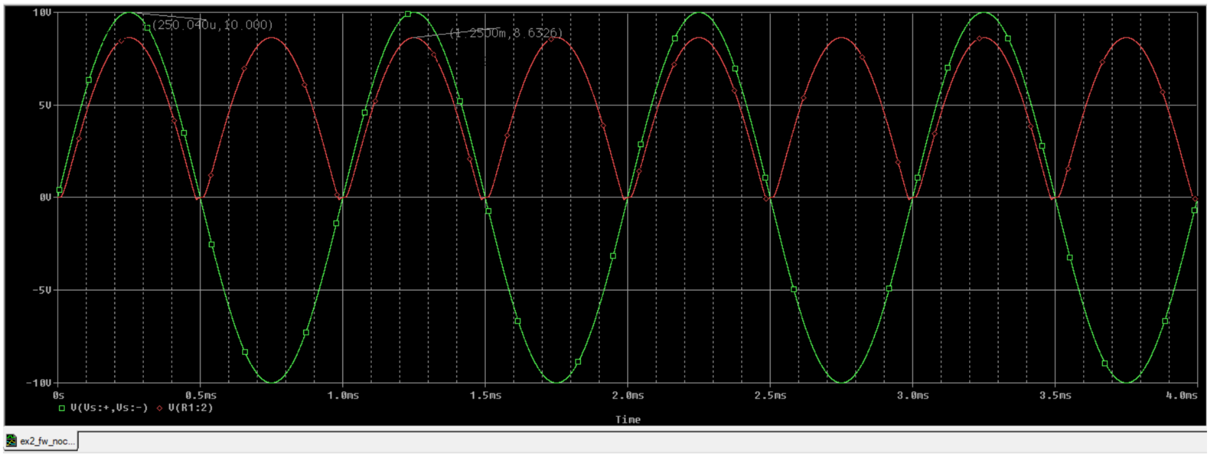


Full-wave bridge rectifier with 47uF capacitor in simulation

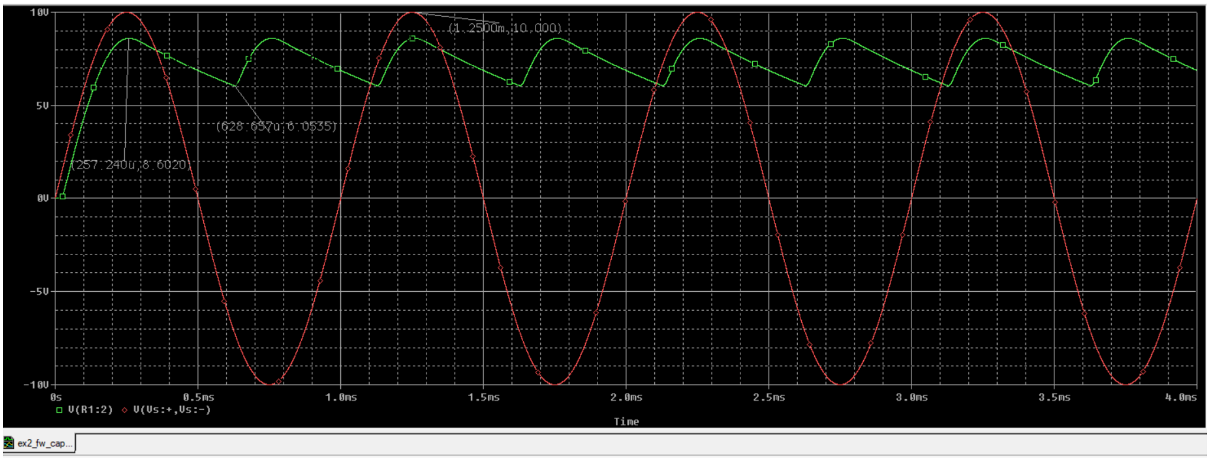
PSPICE Simulation settings:

VSIN, Setup analysis and transient simulation settings are the same as Half-wave simulation.

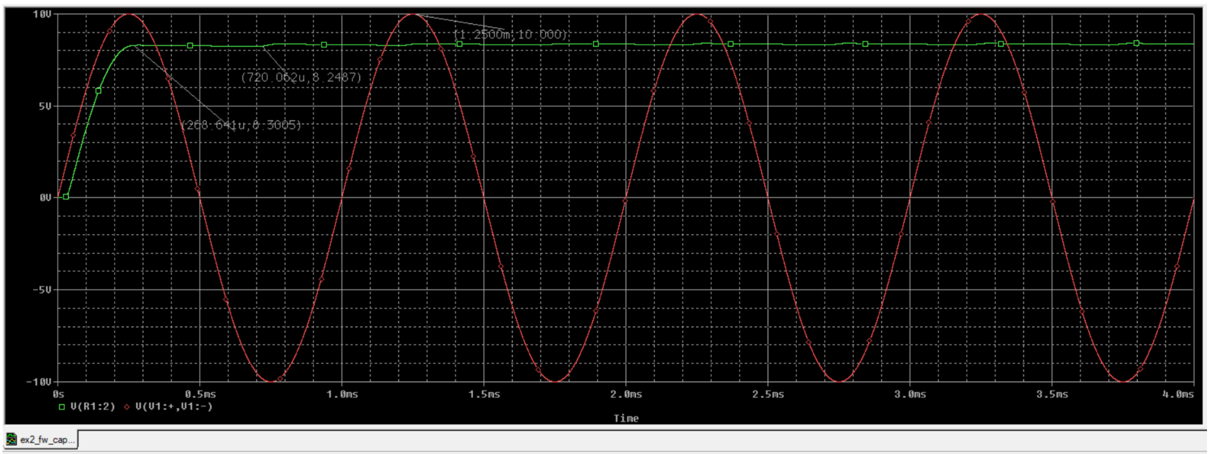
Output waveforms:



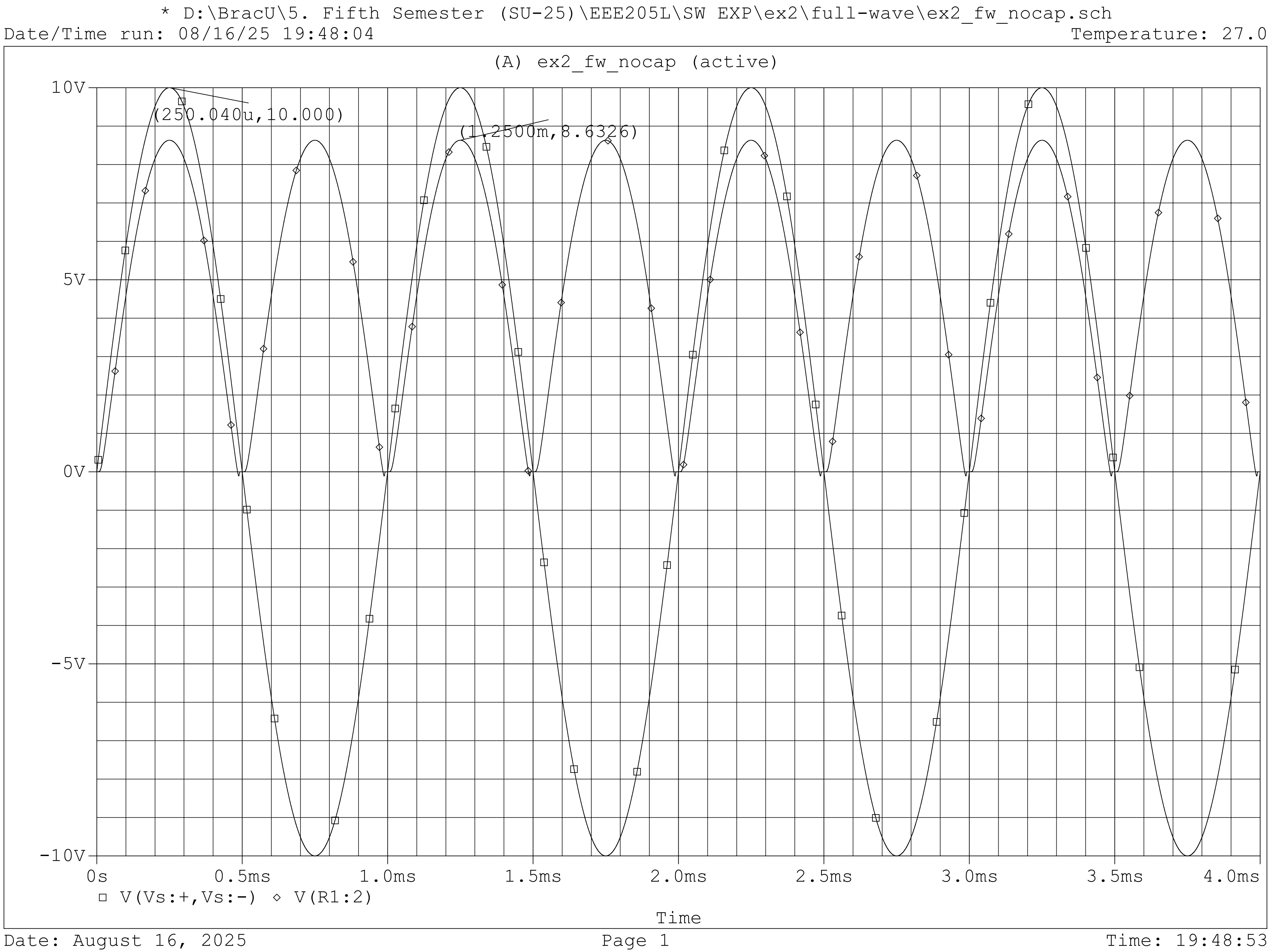
Full-wave rectification waveform without filter



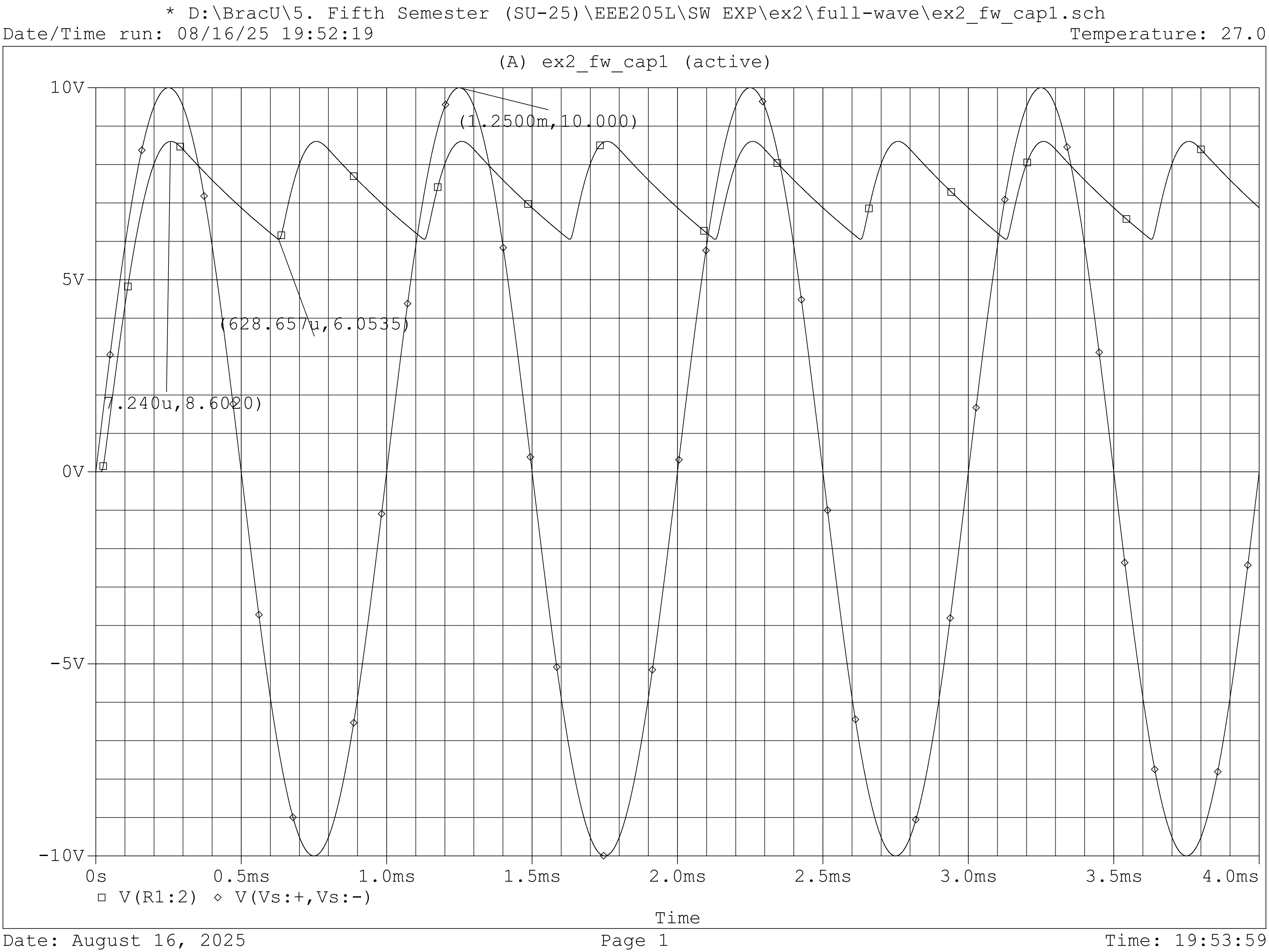
Full-wave rectification waveform with 1uF capacitor



Full-wave rectification waveform with 47uF capacitor

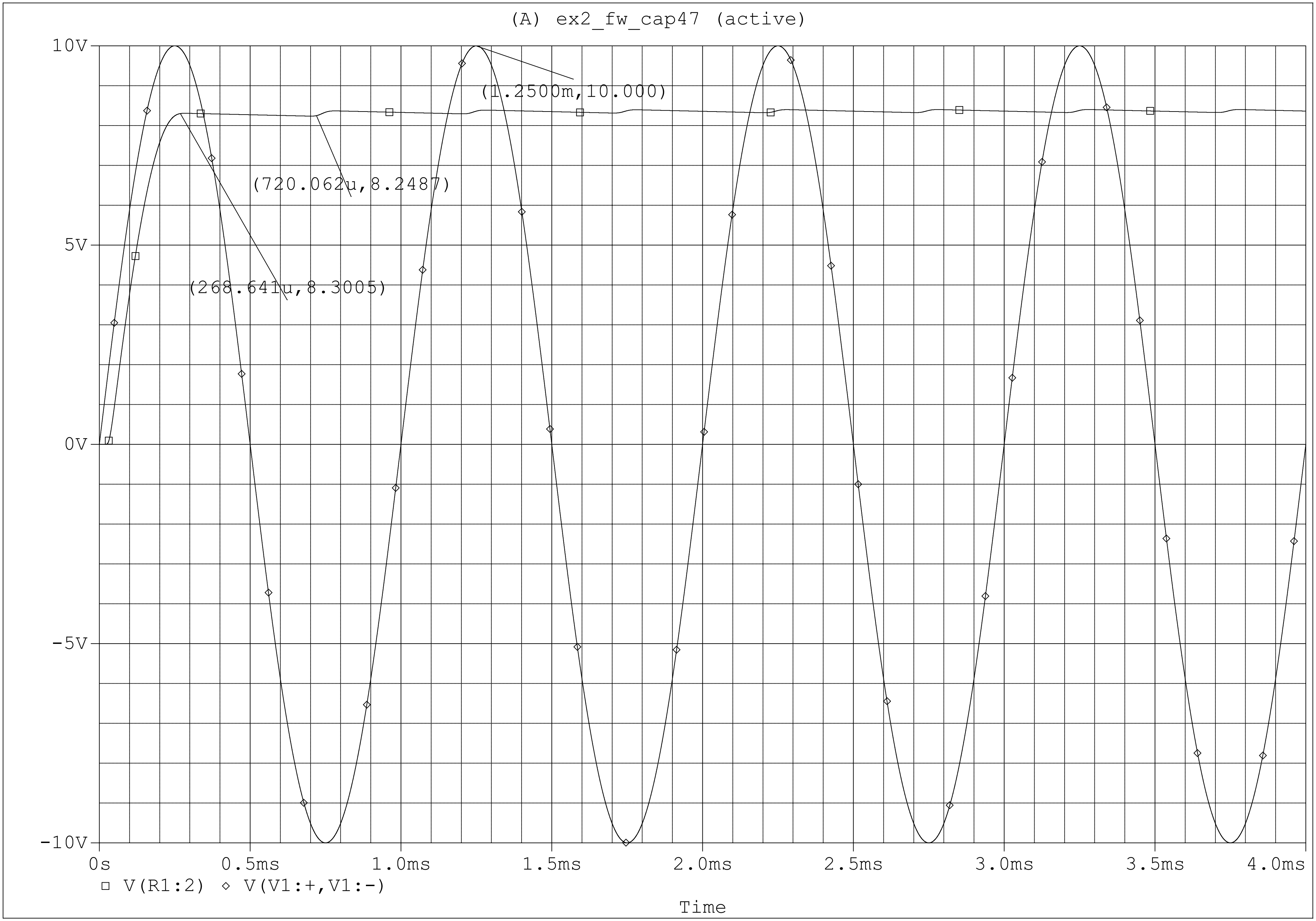


Full-wave rectifier simulation without filter



Full-wave rectifier simulation with 1uF capacitor

* D:\BracU\5. Fifth Semester (SU-25)\EEE205L\SW EXP\ex2\full-wave\ex2_fw_cap47.sch
Date/Time run: 08/16/25 19:54:18 Temperature: 27.0



Date: August 16, 2025 Page 1 Time: 19:58:07

Full-wave rectifier simulation with 47uF capacitor

Data table (Simulated):

VSIN = 10V max
RL = 1k-ohm

Without capacitor:

Vp = 8.6326V (Voltage across resistor)

With capacitor:

Capacitance (uF)	Ripple voltage, $V_{R(p-p)}$ (Volts)
1	(8.6020-6.0535) = 2.5485
47	(8.3005-8.2487) = 0.0518

Data table (From hardware experiment):

RL= 0.986k-ohm

V_{DO}= 0.86V

Without capacitor:

Vp= 8.88V | V_{DC}=5.03V | V_{RMS}= 2.68

With capacitor:

Capacitance (uF)	Ripple voltage, $V_{R(p-p)}$ (Volts)	V_{DC}	V_{RMS}
1	2.68	6.86V	0.66V
47	0.84	7.17V	0.02V

Comparison:

Simulated voltage through resistor: 8.6326V
Hardware experiment value: 8.88V
Difference: (8.88-8.63) = 0.25V

Simulated ripple voltage for 1uF: 2.5485V

Hardware experiment value: 2.68V
Difference: (2.68-2.54) = 0.14V

Simulated ripple voltage for 47uF: 0.0518V
Hardware experiment value: 0.84V
Difference: (0.57-0.0518) = 0.7882V

Discussion:

In the full-wave bridge rectifier experiment, four diodes (D1N4002) are arranged to convert both halves of the AC cycle into a unidirectional output. This means the capacitor is charged twice per cycle, during both the positive and negative halves. As a result, the capacitor discharges for a shorter time, producing a smoother output with lower ripple voltage. Using a larger capacitor, such as 47uF further improves filtering. Comparing the simulated results with hardware values, we can see a small difference which is negligible but we are accepting the software result as superior because the simulation environment is more accurate. The simulated graphs also stay consistent with theoretical and hardware models as well.

To conclude, both simulations were successful. By doing these, we were able to understand the properties of diodes and rectification more precisely. Theoretical predictions closely matched with measured results from the graph which also support the accuracy of formulas used in calculation. The experiment helped us to learn the importance of different rectifiers and filters in various electronic devices.

Drive link (.sch files):

Half-wave rectifier

Full-wave rectifier

EEE205L

Electronic Circuits I Laboratory

Lab Report

Section: 05

Group No:**04**

Experiment no: 03 (Software)

Name of the experiment: *Implementation of Clipper and Clamper network (PSPICE Simulation)*

Prepared by:

Name: Tanzeel Ahmed **ID: 24321367** **Signature**

Tanzeel

All Group members:

Sl.	Name	ID	Signature
1.	**Esrar Ul Hossain Rafin**	**24121187**	*Esrar*
2.	**Tasmiya Tahsin Roza**	**24121339**	*Tasniya*

Date of Submission: 26/08/2025

Objective:

To observe how clipper and clamper circuits modify the shape of input waveforms in PSPICE software

Theoretical Background:

Clipper circuits cut or limit a portion of the input voltage signal. For example, in a half-wave rectifier, the negative part of the signal is clipped. Clippers can be built using diodes in different arrangements. In a series clipper, the diode is in line with the signal path, and in a parallel clipper, it is connected across the load. If we use a biased diode (a diode with a battery in series), we can clip the voltage at levels other than 0.7V.

Clamper circuits, on the other hand, shift the entire signal up or down by adding a DC level. A positive clamper moves the signal upward, while a negative clamper shifts it downward. These circuits use a capacitor and a diode to perform the shifting. The size of the capacitor plays an important role in maintaining the waveform shape.

In modern circuit analysis, simulation is widely used to predict and study the behavior of electronic circuits before actual implementation. Simulation provides a time efficient way to test different circuit configurations and verify theoretical calculations.

PSPICE is one of the most commonly used circuit simulation tools. It allows users to design electronic circuits on a computer, apply test signals, and visualize the circuit response through graphs and plots. By simulating clipper and clamper circuits in PSPICE, we can understand its characteristics without any kind of practical lossl

Problem Statement-1 (Clipper Circuits)

To construct and analyze positive and negative clipper circuits using diodes.

List of equipments:

1. Suitable computer or laptop that meets the minimum requirements
2. ORCAD Pspice Schematics software

List of equipments required in simulation:

1. Resistor (R) - 10K-ohm - 1x
2. VSIN source - 1x

3. p-n junction diode D1N4002 - 1x
4. DC Voltage source (VDC) Vs =1x
5. Voltage level markers

Circuit diagram (from manual):

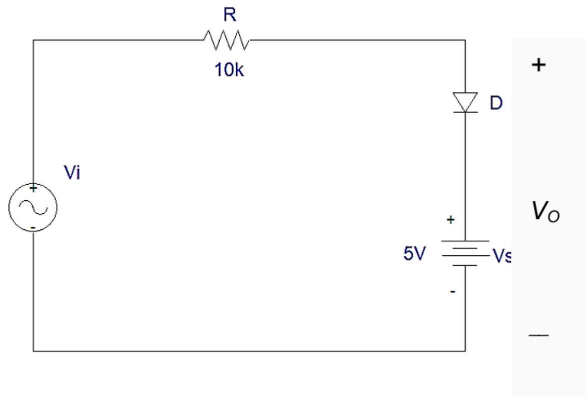


Figure: Positive clipper circuit

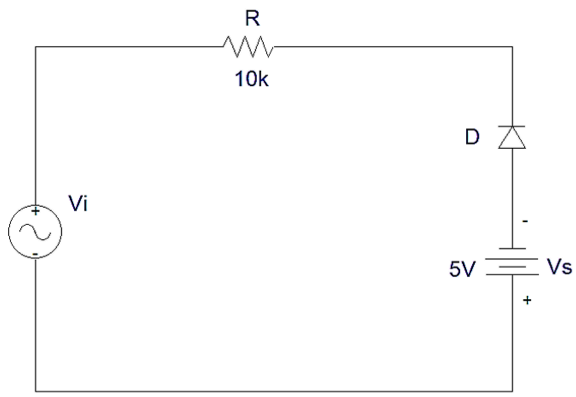


Figure: Negative clipper circuit

PSPICE Schematic diagram:

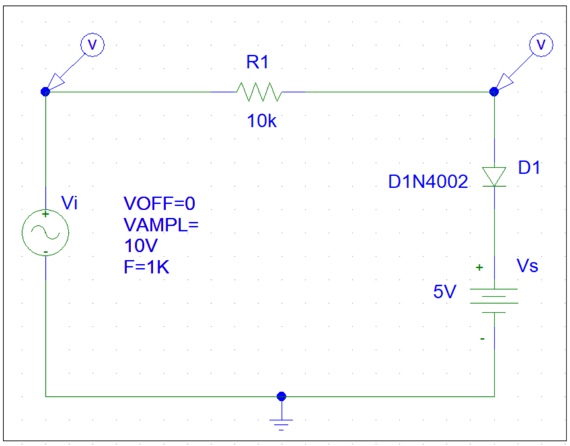


Figure: Positive clipper circuit in simulation

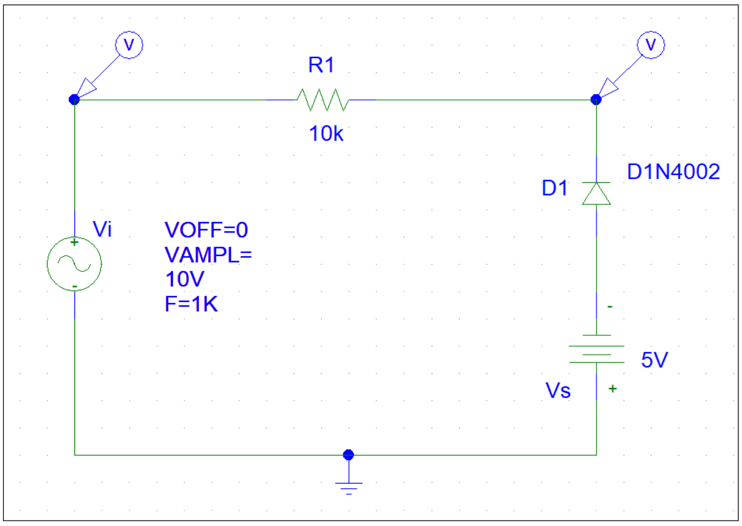
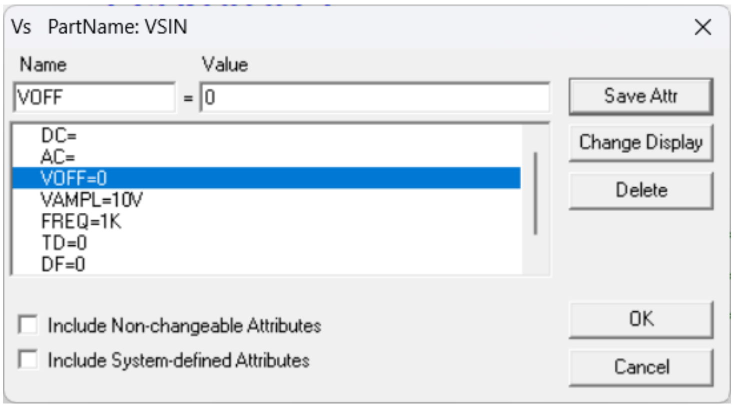


Figure: Negative clipper circuit in simulation

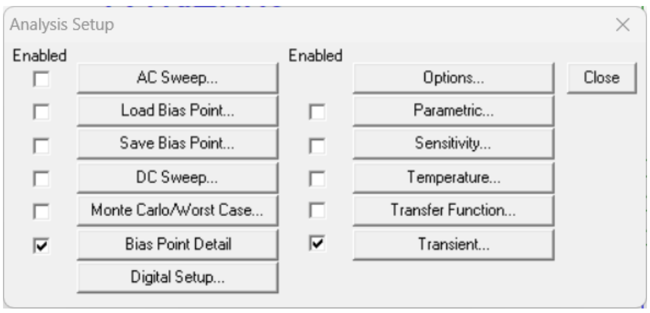
PSPICE Simulation settings:

VSIN (Vs):

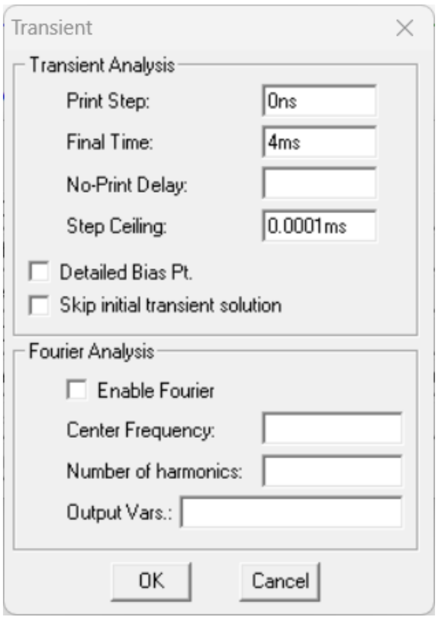


VOFF=0, VAMPL=10V, FREQ=1K

Analysis setup menu:



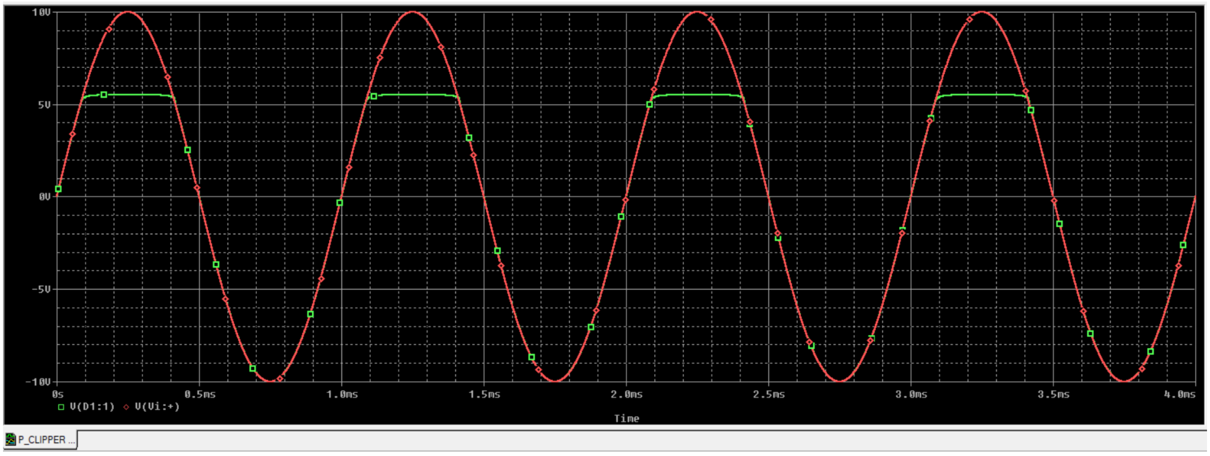
Transient:



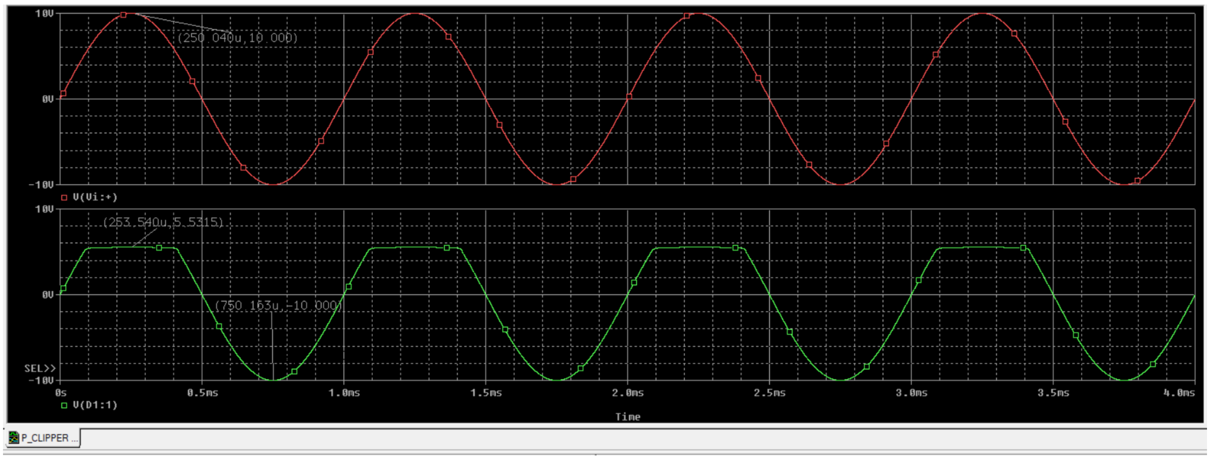
Print Step: 0ns, Final Time: 4ms, Step Ceiling: 0.0001ms

Output waveforms:

<u>Positive Clipper</u>:

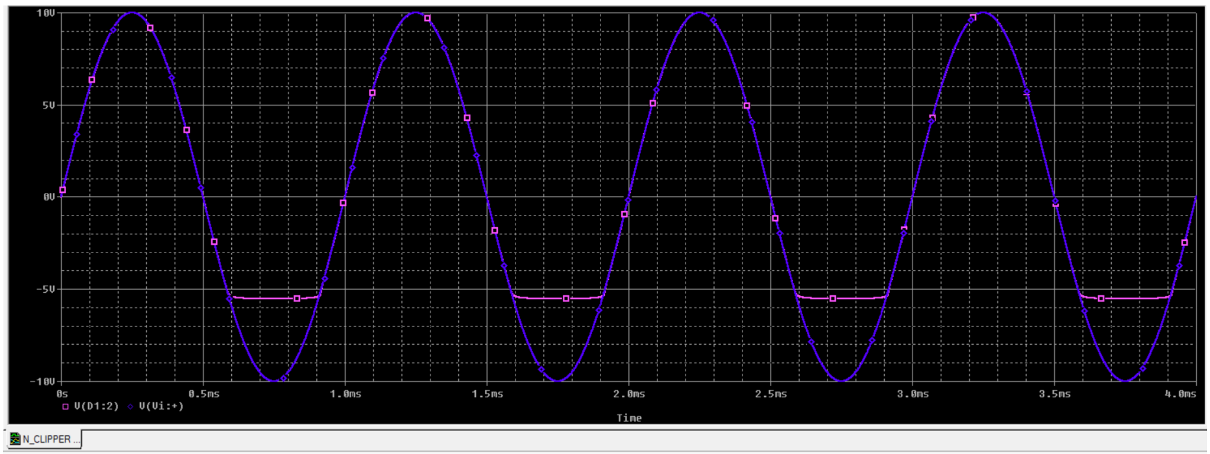


Combined plot of positive clipper

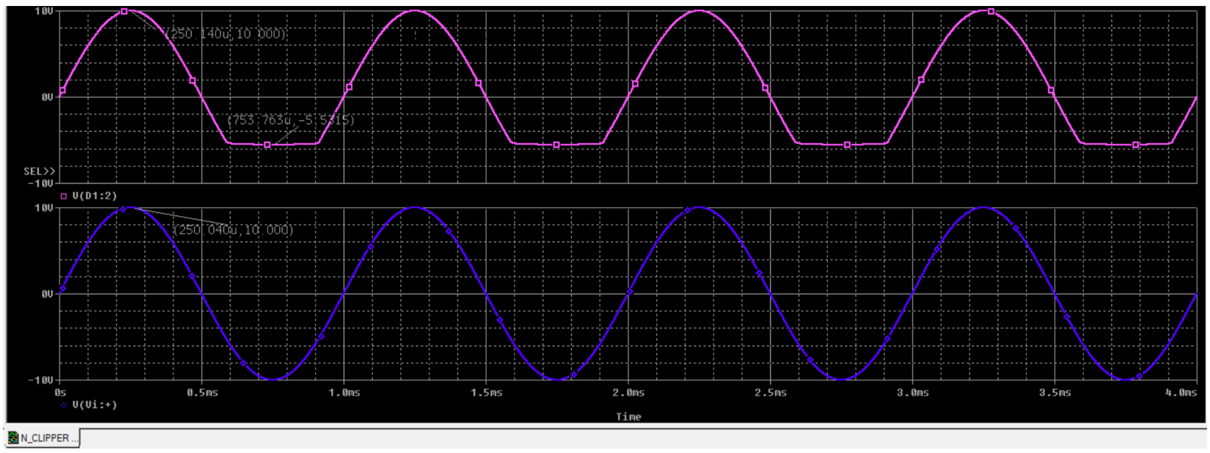


Separated plots of positive clipper for easier value extraction

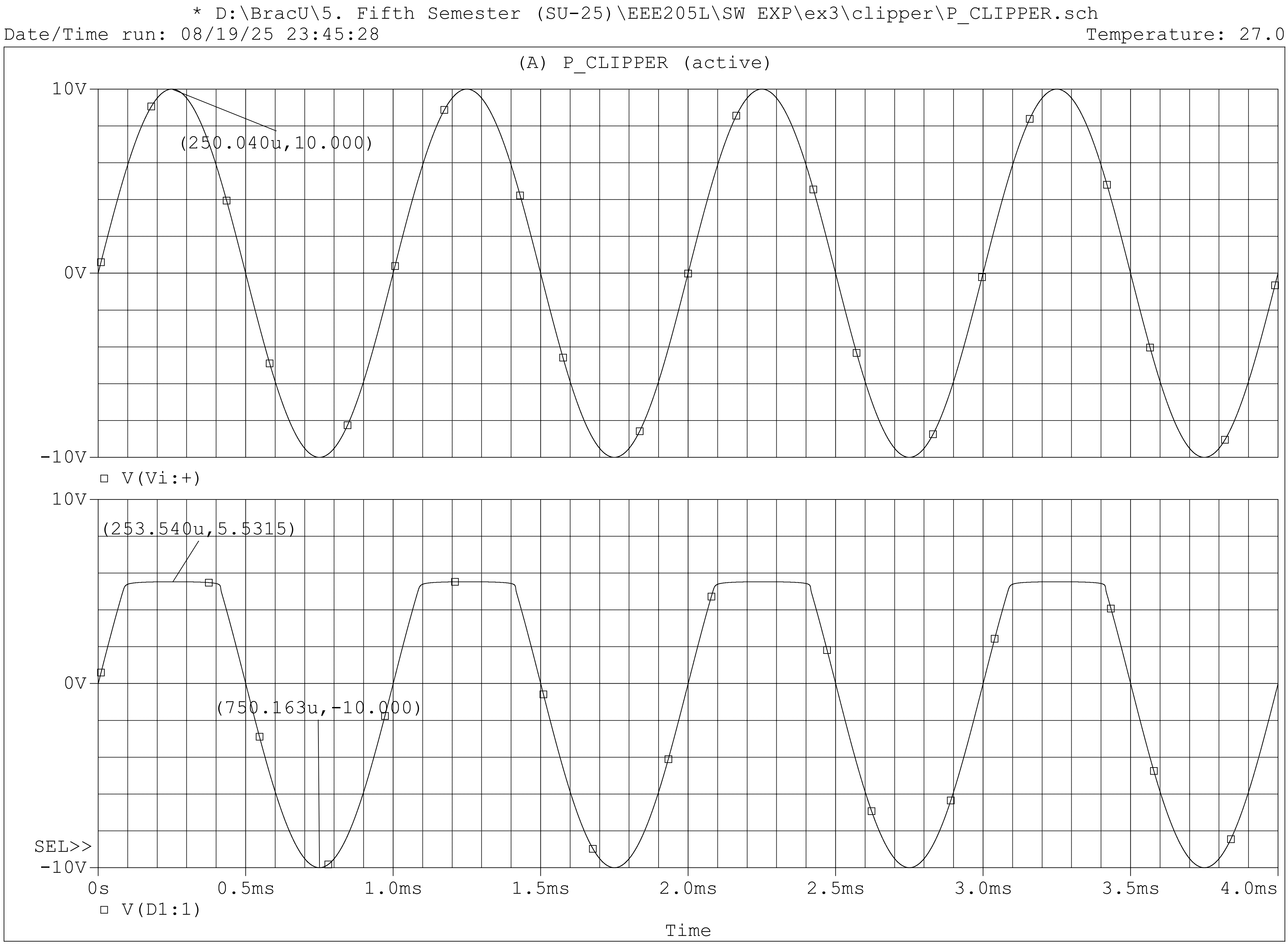
<u>*Negative Clipper:*</u>



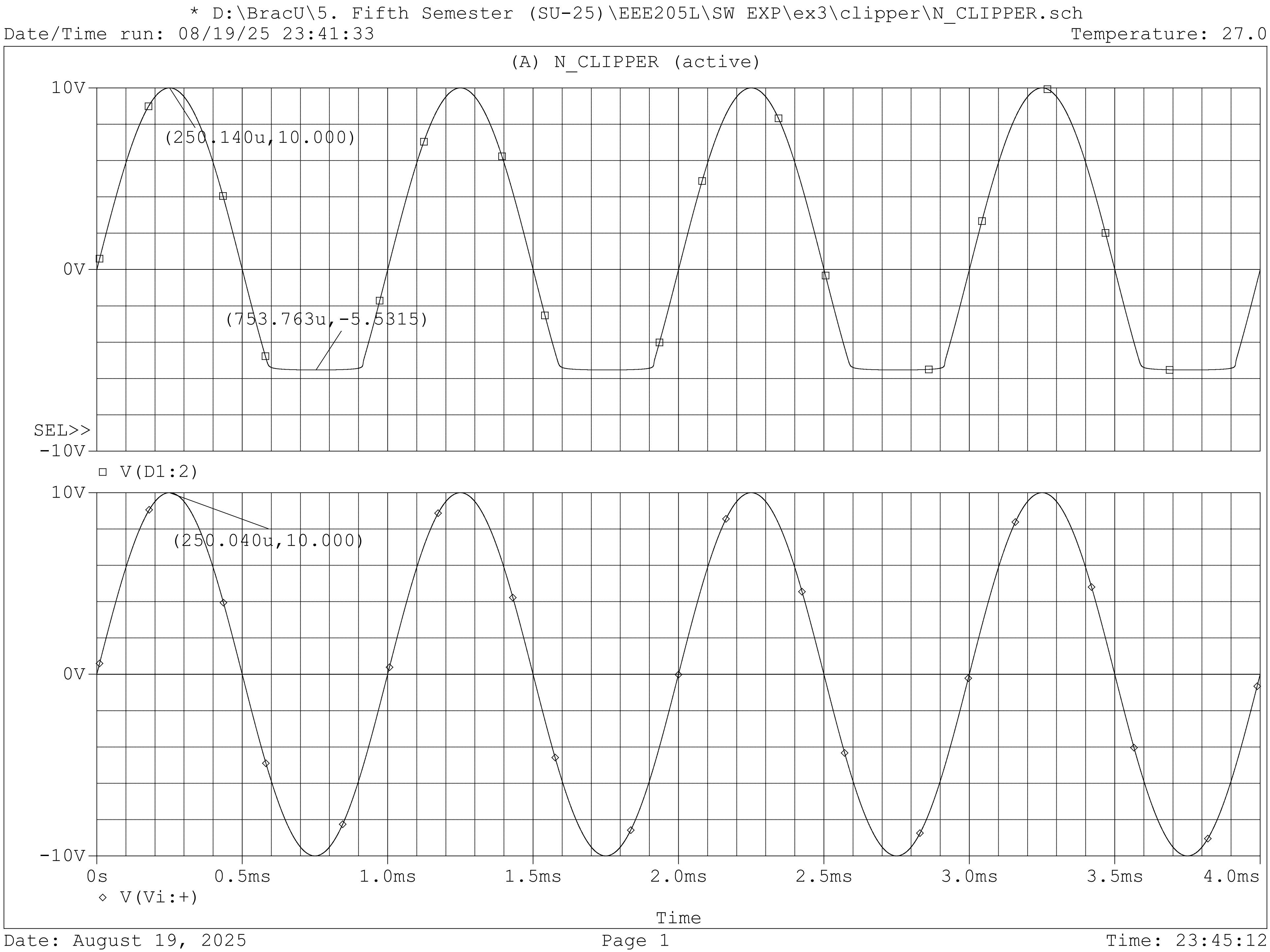
Combined plot of negative clipper



Separated plots of negative clipper for easier value extraction



Positive clipper waveform simulated in Pspice



Negative clipper waveform simulated in Pspice

Data table (Simulated):

Circuit type	Measured clipping voltage (V)
Positive clipper	5.5315
Negative clipper	-5.5315

Data table (From hardware experiment):

Circuit type	Measured clipping voltage (V)
Positive clipper	5.60
Negative clipper	-5.60

Comparison:

Simulated positive clipping voltage = 5.5315V
Hardware tested positive clipping voltage = 5.60V
Difference: (5.60-5.5315) = 0.0685

Simulated negative clipping voltage = -5.5315V
Hardware tested negative clipping voltage = -5.60V
Difference: |(-5.60-5.5315)| = 0.0685V

Discussion:

In the positive clipper circuit, all of the components are connected in series. The main task of a clipper circuit is to clip the incoming AC voltage at a certain level. The diode (D1N4002) was forward biased with the positive point of the DC voltage source. AC 10V amplitude was applied and from the plot markings we can see that it was clipped at 5.5315 volts. In the negative clipper circuit, the diode and voltage DC voltage source are flipped which clips at -5.5315 volts. When compared to hardware results, both circuits only show a difference of 0.0685 volts which is really low and within the acceptable range. The simulated graphs also stay consistent with the hardware plots.

Problem Statement-2 (Clamper Circuits)

To observe how a clamper circuit shifts a waveform up or down using a diode and capacitor.

List of equipments:

1. Suitable computer or laptop that meets the minimum requirements
2. ORCAD Pspice Schematics software

List of equipments required in simulation:

1. Resistor (R) - 220K-ohm - 1x
2. VPULSE source = 1x
3. P-n junction diode D1N4002 - 1x
4. DC voltage source (VDC) Vs - 1x
5. Capacitor (C) - 1uF - 1x
6. Voltage level markers

Circuit diagram (from manual):

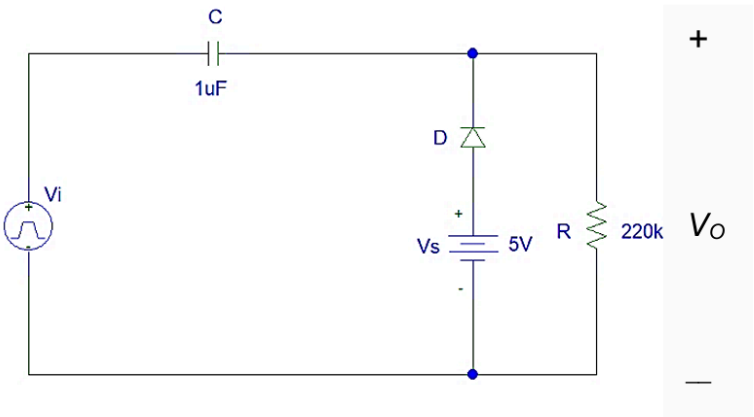


Figure: Positive clamper circuit

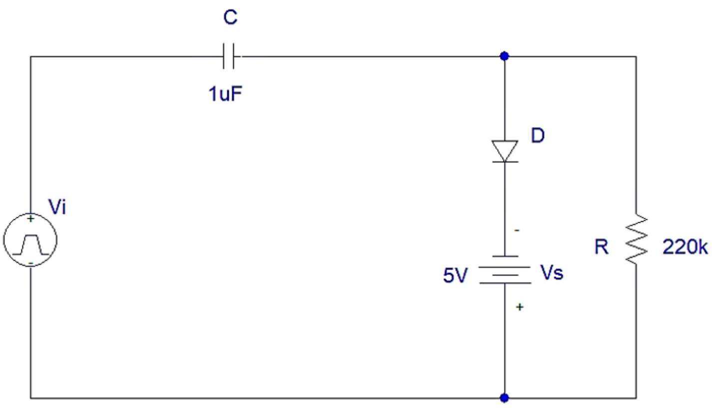


Figure: Negative clamper circuit

PSPICE Schematic diagram:

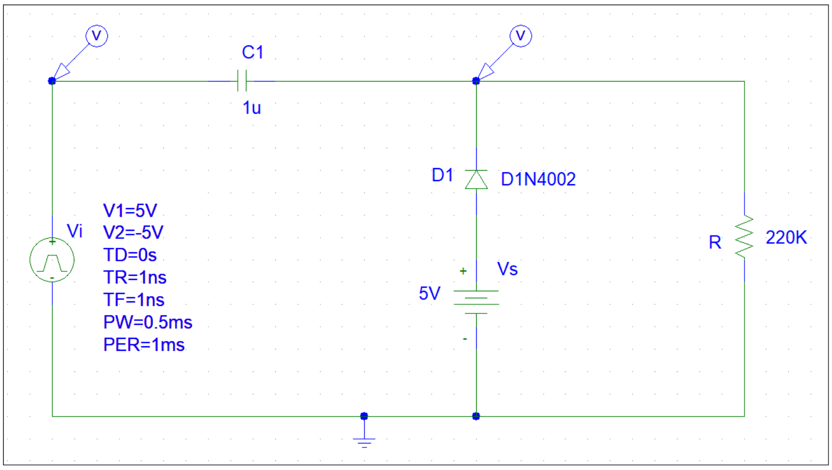


Figure: Positive clamper circuit in simulation

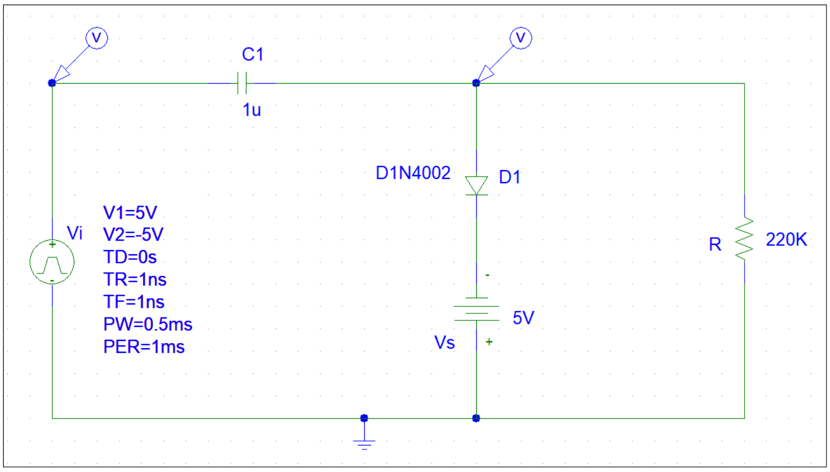
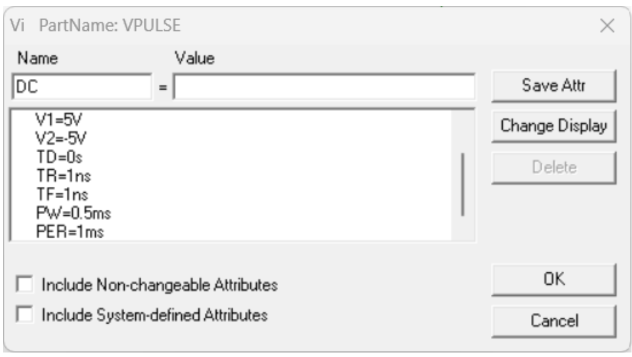


Figure: Negative clamper circuit in simulation

PSPICE Simulation settings:

Setup analysis and transient simulation settings are the same as clipper circuit simulation.

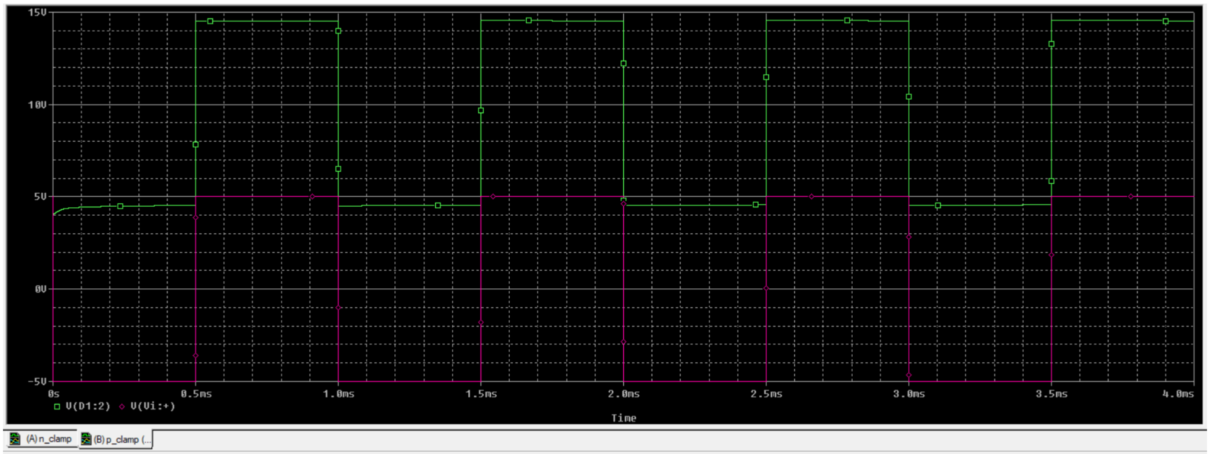
VPULSE settings:



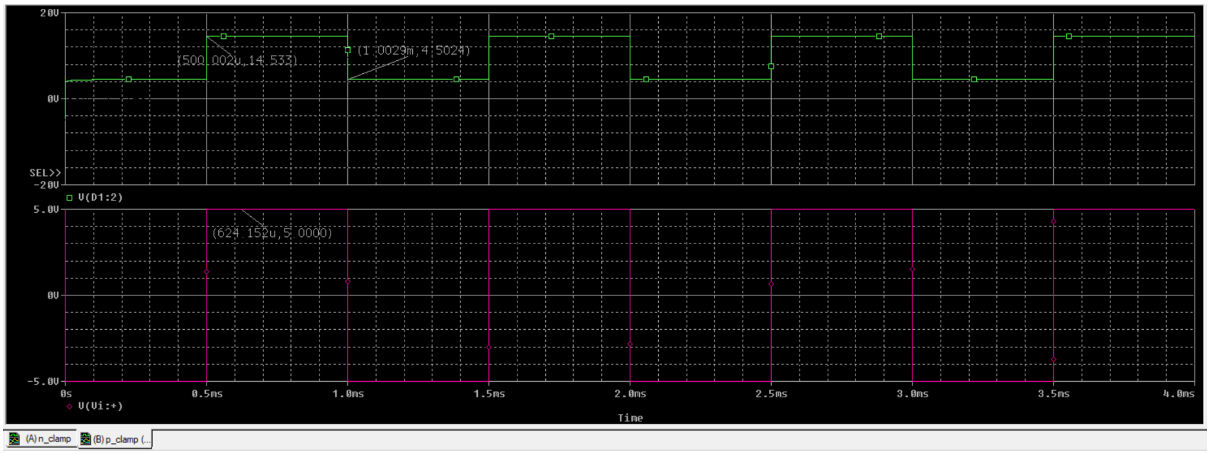
V1=5V, V2=-5V, TD=0s, TR=1ns, TF=1ns, PW=0.5ms, PER=1ms

Output waveforms:

Positive Clamper:

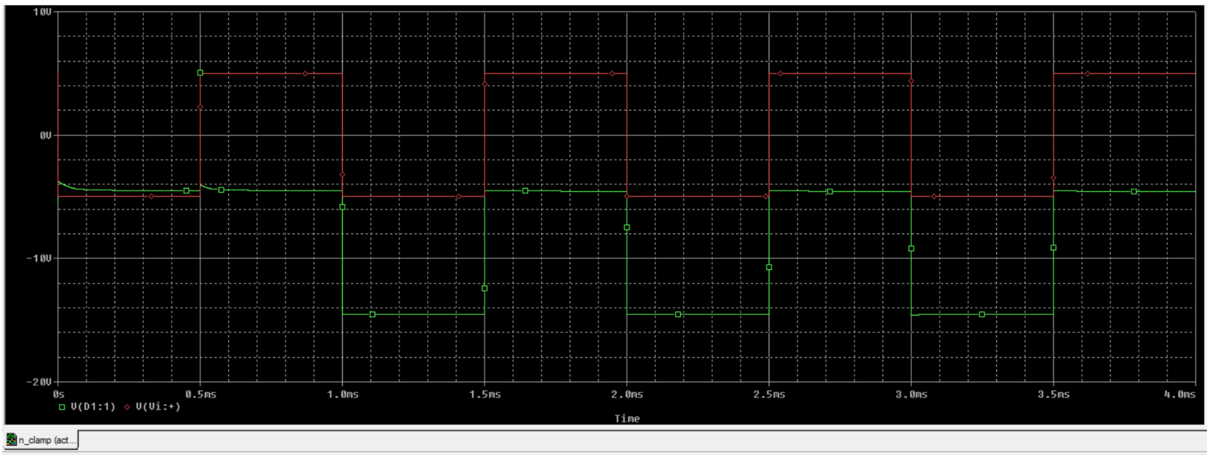


Combined plot of positive clamper

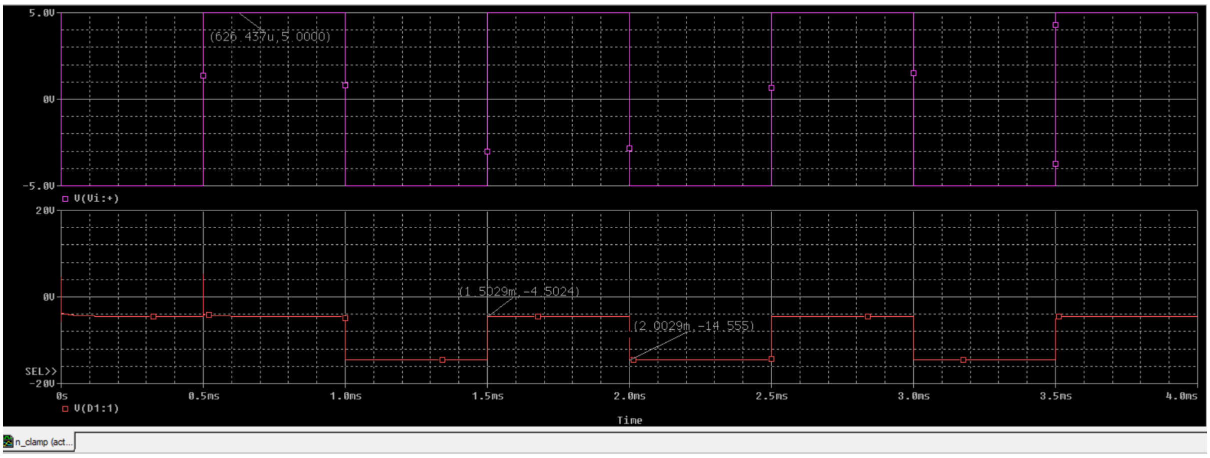


Separated plots of positive clamper for easier value extraction

Negative Clamper:



Combined plot of negative clamper



Separated plots of negative clamper for easier value extraction

Data table (Simulated):

Circuit type	Measured clamping voltage difference (V)
Positive clamper	10.0306
Negative clamper	-10.0526

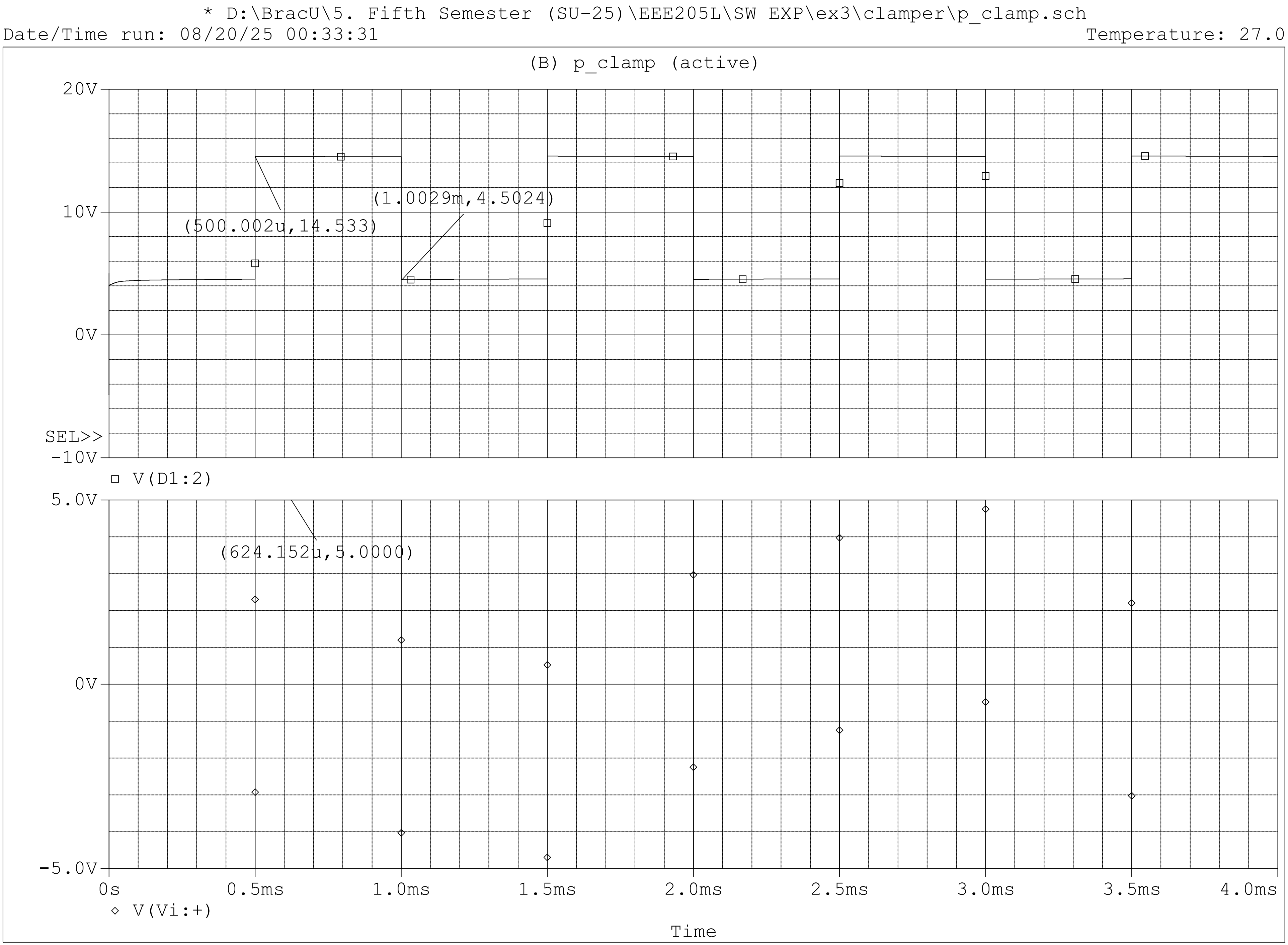
Data table (From hardware experiment):

Circuit type	Measured clamping voltage difference (V)
Positive clamper	9.30
Negative clamper	-9.50

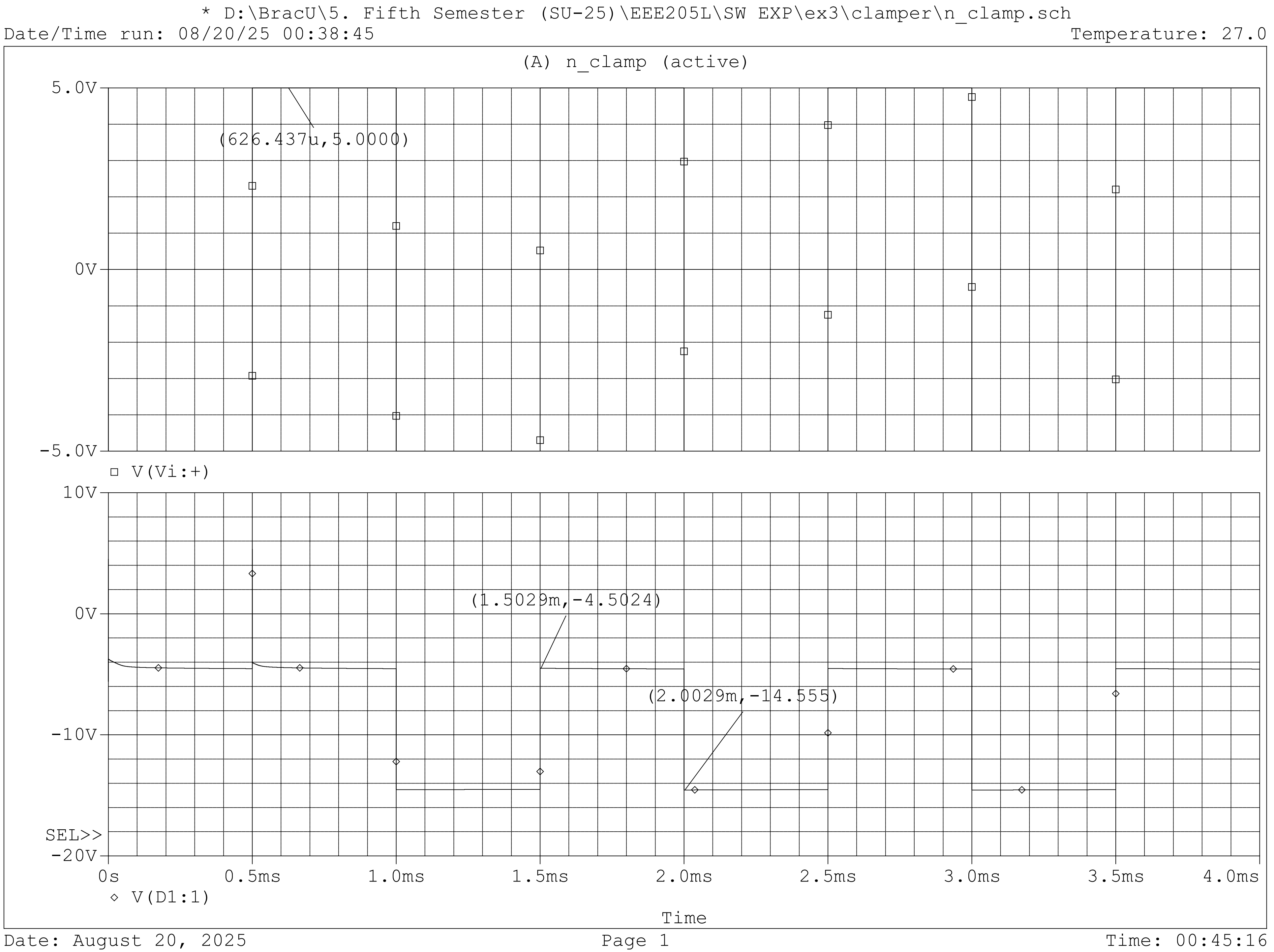
Comparison:

Simulated positive clamping voltage = 10.0306V
Hardware tested positive clamping voltage = 9.30V
Difference: (10.0306-9.30) = 0.7306

Simulated negative clamping voltage = -10.0526V
Hardware tested negative clamping voltage = -9.50V
Difference: |(10.0526-9.50)| = 0.5526V



Positive clamper waveform simulated in Pspice



Negative clamper waveform simulated in Pspice

Discussion:

In the positive clamper circuit, all of the components are connected in series-parallel. The main task of a clamper circuit is to clamp and shift any given waveform up or down without changing its shape. The diode (D1N4002) was forward biased according to the given diagrams. Square wave was used for simulation which was positively clamped with a minimum voltage of 4.5024 volts and maximum 14.533 volts. The connected capacitor charges during one part of the cycle and retains the voltage to shift. In the negative clamper circuit, the waveform was negatively clamped with a minimum voltage of -4.5024 volts and maximum -14.555 volts. When compared to hardware results, both circuits only show a difference of 0.07306 volts for positive and 0.5526 volts for negative which is really low and within the acceptable range. The simulated graphs also stay consistent with the hardware plots.

Overall, the experiment was successful and we gained a better understanding of how clipping and clamping circuits works for various electronic devices.

Drive links (.sch files):

Clipper circuits

Clamper circuits