HO CHI MINH CITY UNIVERSITY OF TECHNOLOGY

OFFICE FOR INTERNATIONAL STUDY PROGRAM

SEMESTER: 251

LAB 2 REPORT

INSTRUCTOR:

MR. PHAM CONG THAI

Student Name	ID
Trần Lê Tuấn Anh	2452089
Ngô Đức Anh (Leader)	2452054
Nguyễn Trường An	2452020

Class ID: CC04

Ho Chi Minh city, October 16, 2025

3.1 Complete diode model

According to the above simulation circuit (Figure 1.2), determine the voltage across the resistor and the diode (VR, VD), and also the current I in the circuit with **two different values of R_1**, including **220 Ohm** and **1.5K Ohm**. It is assumed that the complete diode model is used to analyse, having the forward voltage at **0.7V** and the internal resistor equal to **50 milliohms**.

Finally, the simulation on PSpice is run to double check with theory calculation. Brief explanations concerning the difference between theory and simulations can be provided in the report.

3.1.1 Theory calculation

Notes:

Explanations, formulas, and equations are expected rather than only results.

According to KVL, we have: $-V_{source} + V_D + I_F.R + I_F.r' = 0$

We have $V_D = 0{,}7\ V$ so:

When $R = 220\Omega$, $V_R = I_F.R = \frac{V_{source} - V_D}{R + r\prime}.R = \frac{5 - 0{,}7}{220 + 50}.200 = 3{,}503(V)$

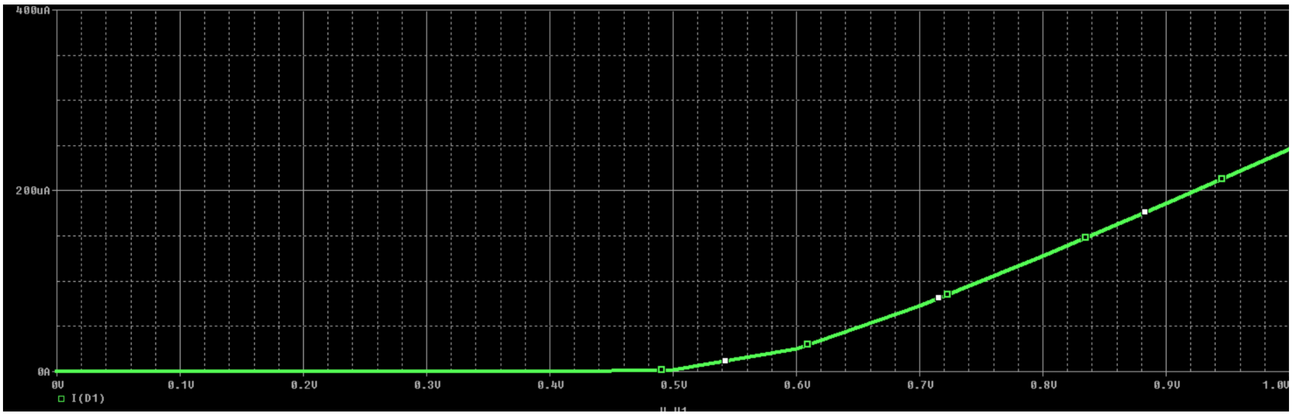
And when $R = 1{,}5k\Omega$, $V_R = I_F.R = \frac{V_{source} - V_D}{R + r\prime}.R = \frac{5 - 0{,}7}{1500 + 50}.200 = 4{,}161(V)$

3.1.2 PSpice Simulation

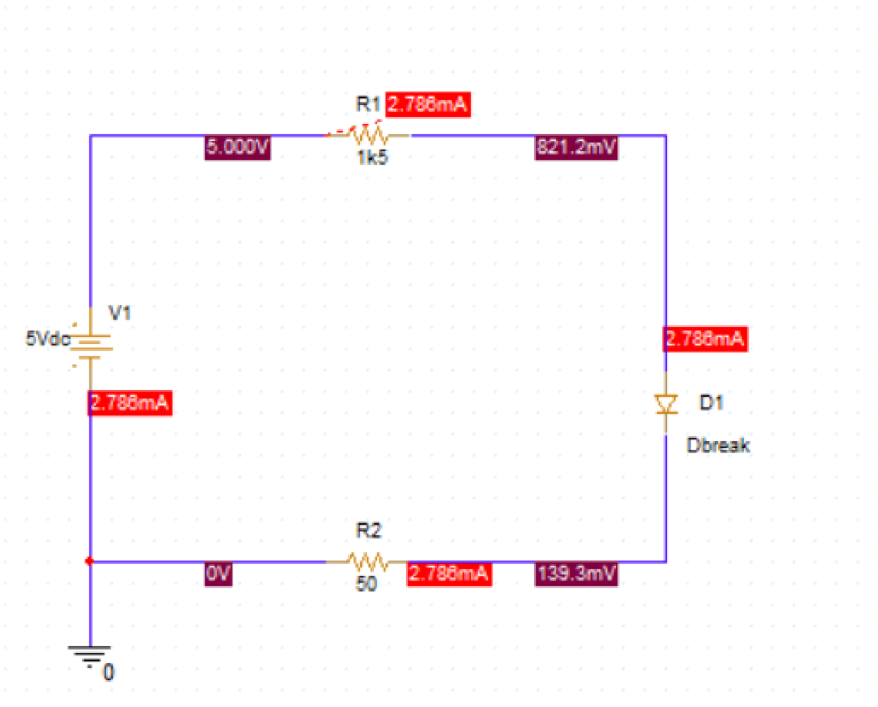
Set the simulation profile to bias-point. Moreover, enable both **Enable Voltage Bias Display** and **Enable Current Bias Display** to show the simulation results.

Students are supposed to capture the screen in PSPice and present in this report.

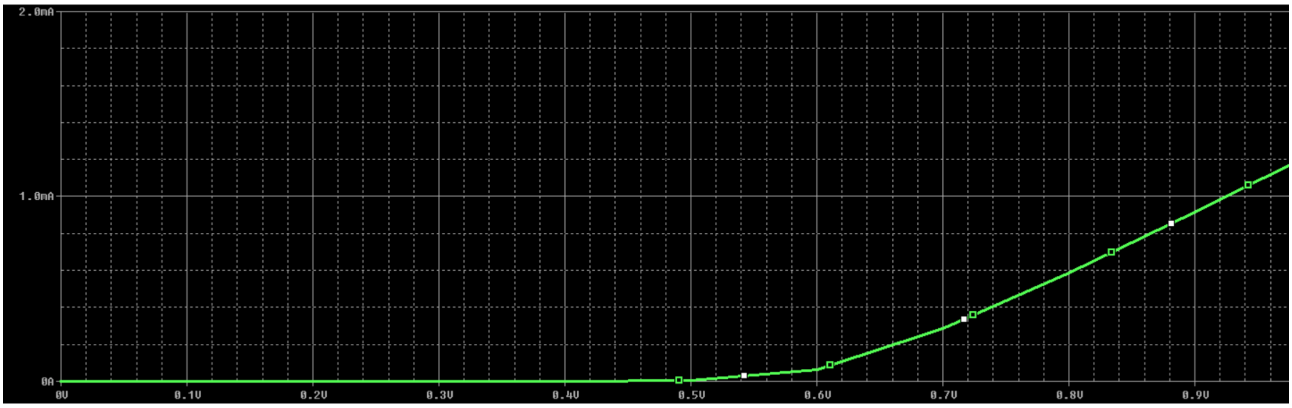
Simulation results (images): Your image goes here



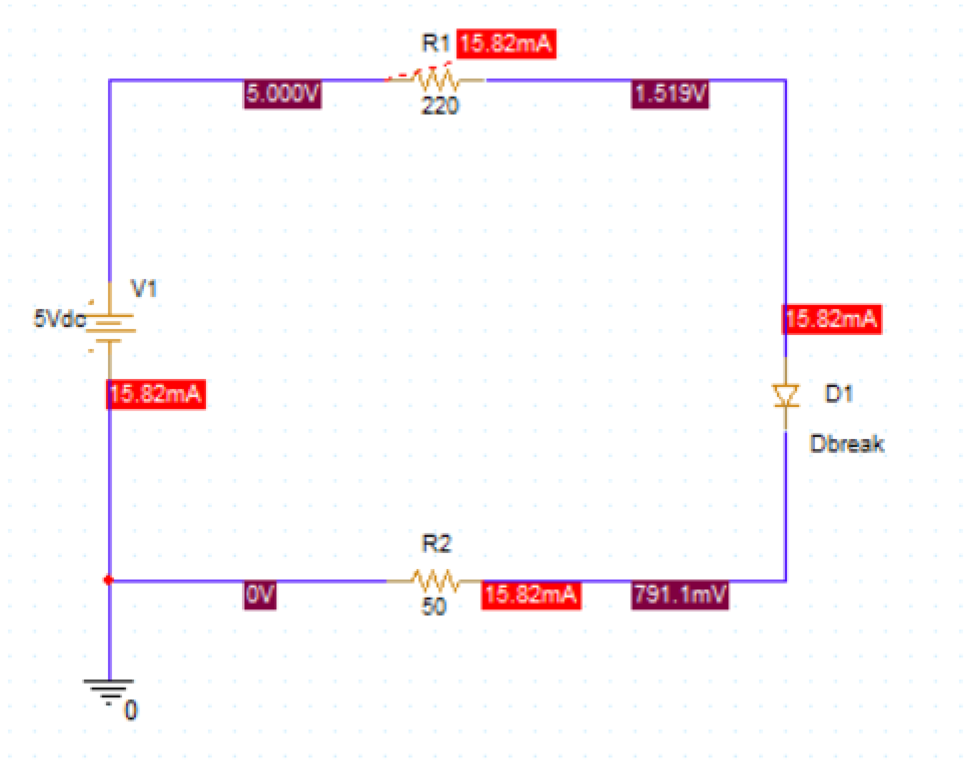
The V-I characteristic when R=1500kΩ.



The bias point simulation when R=1500kΩ



The V-I characteristic when R=220kΩ.



The bias point simulation when R=220kΩ

3.1.3 Comparison

In this section, the theory calculations and PSpice simulations are summarized in the table below to compare the difference. Students are supposed to fill all information in the table.

	Theory			PSpice		
	V_R	V_D	I	V_R	V	I
R = 220 Ohm	3.503 (V)	0.7 (V)	15.92 (mA)	3.481(V)	0.7279 (V)	15.82 (mA)
R=1.5K Ohm	4.2434 (V)	0.7 (V)	2.774 (mA)	4.1788 (V)	0.6827 (V)	2.786 (mA)

According to the above Exercise Results, give some comments about observation (between calculation results and simulation results): The values obtained by theory's calculation and Pspice' simulation are really close to each other. Thus, we can consider both are exactly the same.

3.2 Diode in a series

Similar to the previous exercise, determine the value of the voltage V_{D1}, V_{D2}, V_{D3} and the current I for the give circuit. Then, simulate again the circuit using PSpice. However, in this case, the practical diode model is used with the forward voltage is 0.7223V.

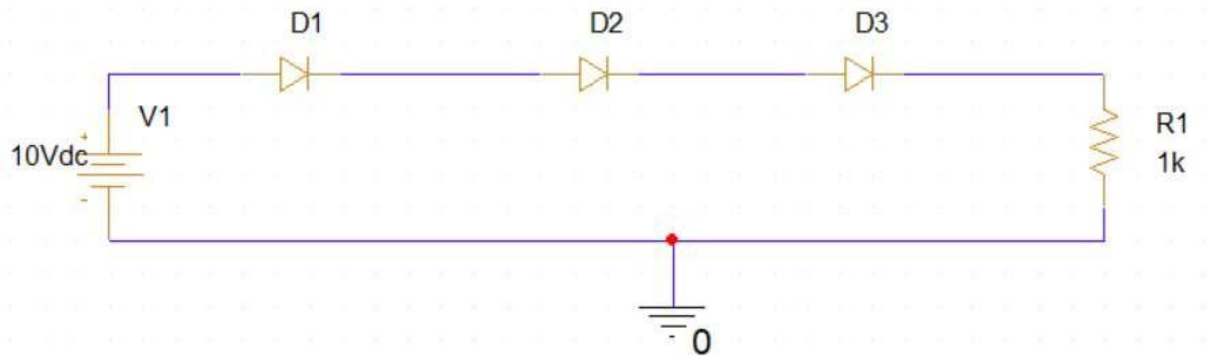


Figure 1.15: Find the voltage and the current in the given circuit

3.2.1 Theory Calculation

Notes:

Explanations, formulas, and equations are expected rather than only results.

According to the properties of practical diode model, we have:

$V_{D1} = 0{,}7223V$

$V_{D2} = 0{,}7223V$

$V_{D3} = 0{,}7223V$

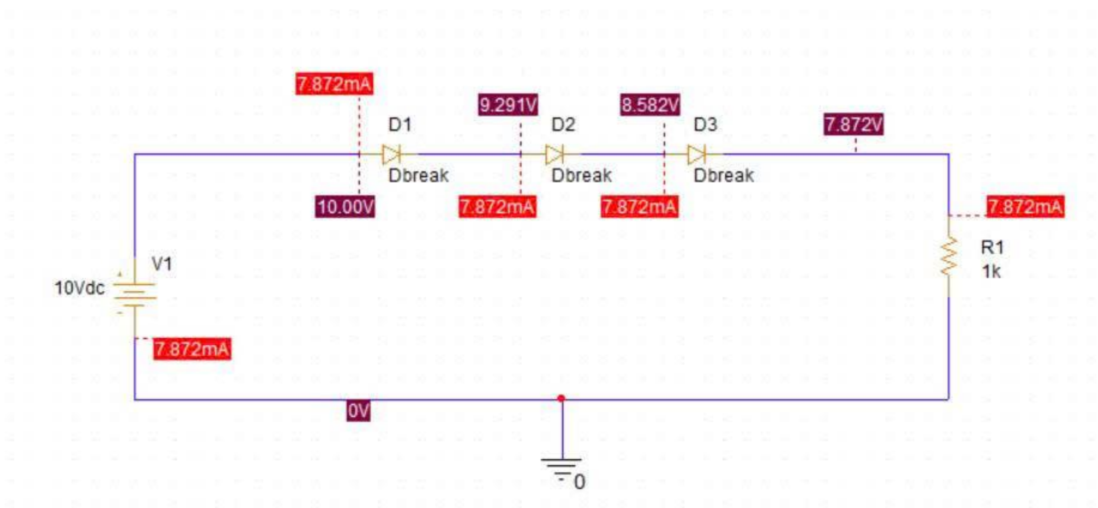
$V_{R1} = V_{source} - 3V_0 = 10 - 3.0{,}7223 = 7{,}8331(V)$

Formula to calculate I:$I = \frac{V_{R_1}}{R_1} = 7{,}8331.10^{-3}(A)$

3.2.2 PSpice Simulation

Set the simulation profile to bias-point. Moreover, enable both **Enable Voltage Bias Display** and **Enable Current Bias Display** to show the simulation results. Students are supposed to capture the screen in PSPice and present it in this report.

Simulation results (images):



The bias point simulation.

3.2.3 Comparison

	V_{D1}	V_{D2}	V_{D3}	V_{R1}	I
Calculation	0.7223 (V)	0.7223 (V)	0.7223 (V)	7.83 (V)	7.8 (mA)
PSpice	0.709 (V)	0.709 (V)	0.729 (V)	7.827 (V)	7.872 (mA)

The circuit in this exercise is a simple solution to design a power supply by leveraging a voltage drop of a diode. For example, a SIM module used to send SMS messages has a good voltage supply at 4.3V. In this case, a diode is connected from a 5V supply (which is a very popular voltage) and then, connected to the module SIM. Not only used to protect the module to avoid reverse current, a diode is a low-cost solution to generate 4.3V power supply for the SIM module.

3.3 Circuit Analysis with Diode

In PSpice, some exercise in the lesson can be simulated to confirm the results. Although it is not exactly the same values (e.g. voltage and current), the simulation in PSpice is a tool to check your solution. An example simulation circuit having diode and resistors is depicted as follows:
Students are proposed to analyse this circuit using practical diode model, with the dropdown voltage is around 0.7V. Then the simulation is run on PSpice to double check with your results.

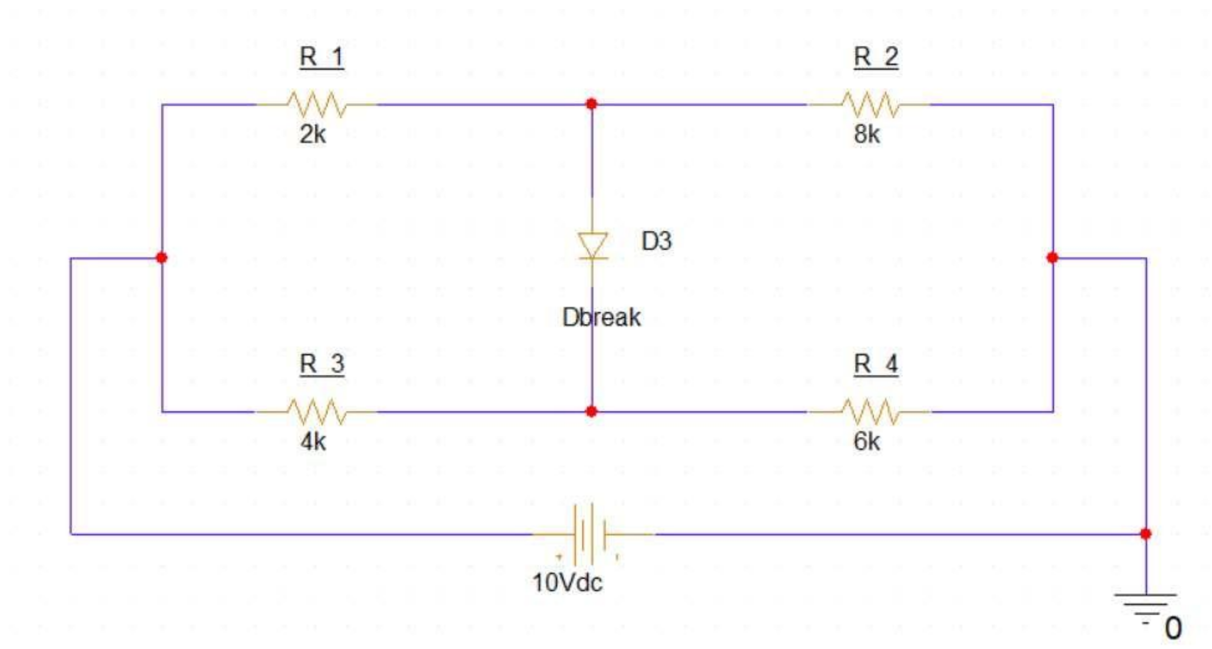


Figure 1.16: Circuit analysis with diode

3.3.1 Theory calculation

It is assumed that the voltage at anode and cathode of the diode V_1 and V_2.
It is assumed that the diode is in forward bias mode.
According to the practical diode mode: $V_1 - V_2 = 0.7V$
Students are proposed to construct the equations to determine the current across all the resistors.

Node labeling and Known voltages

- Left outer node = +10V (battery positive)
- Right outer node = 0V (ground)
- Top middle node (junction of R_1 and R_2) → call its voltage $\mathbf{V_1}$
- Bottom middle node (junction of R_3 and R_4) → call its voltage $\mathbf{V_2}$
- Diode D_3 between the two middle nodes; forward condition assumed, so: $\mathbf{V_1 - V_2 = 0.7V}$

Take current positive leaving each node. Let I_D be diode current leaving node V_1 toward V_2 (so positive I_D means current flows top → bottom).

KCL at node V_1:

$$\frac{V_1 - 10}{R_1} + \frac{V_1 - 0}{R_2} + I_D = 0$$

KCL at node V_2:

$$\frac{V_2 - 10}{R_3} + \frac{V_2 - 0}{R_4} - I_D = 0$$

Diode relation:

$$V_1 - V_2 = 0.7$$

Solve three equations we have:

$$V_1 = 7.480\,V$$

$V_2 = 6.780\,V$

$I_D = 0.000325\,A = 0.325\,mA$

$$I_{R1} = \frac{10 - V_1}{R_1} = \frac{10 - 7.480}{2000} = \frac{2520}{2000} = 1.260\,mA$$

$$I_{R2} = \frac{V_1 - 0}{R_2} = \frac{7.480}{8000} = 0.935\,mA$$

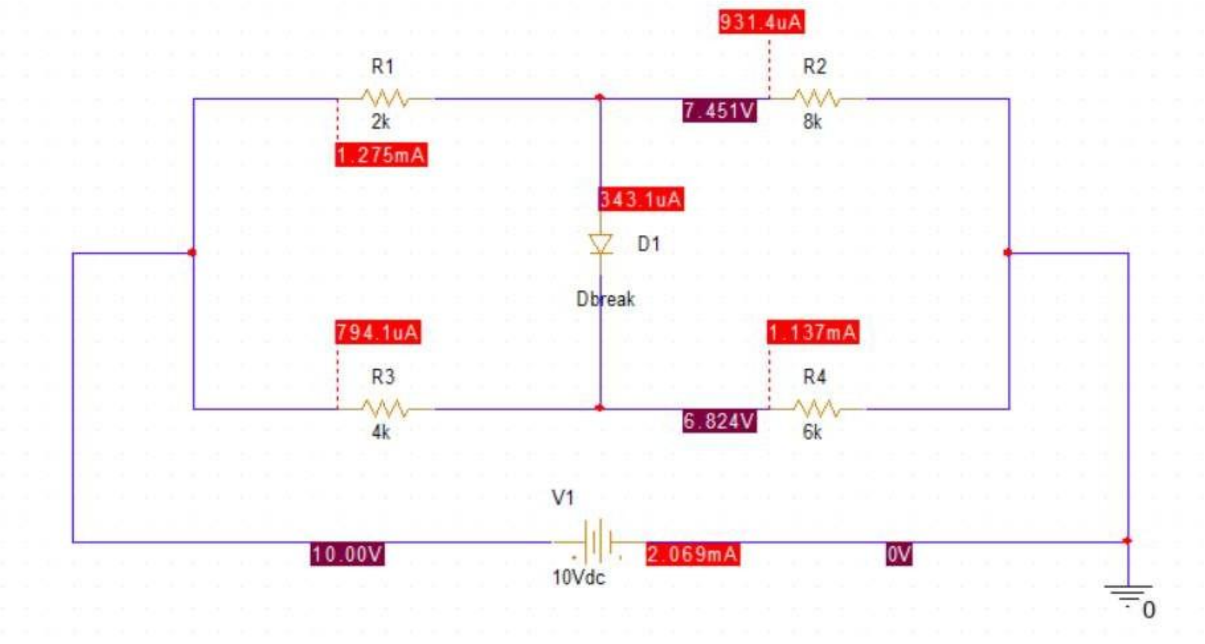
$$I_{R3} = \frac{10 - V_2}{R_3} = \frac{10 - 6.780}{4000} = \frac{3220}{4000} = 0.805\,mA$$

$$I_{R4} = \frac{V_2 - 0}{R_4} = \frac{6.780}{6000} = 1.130\,mA$$

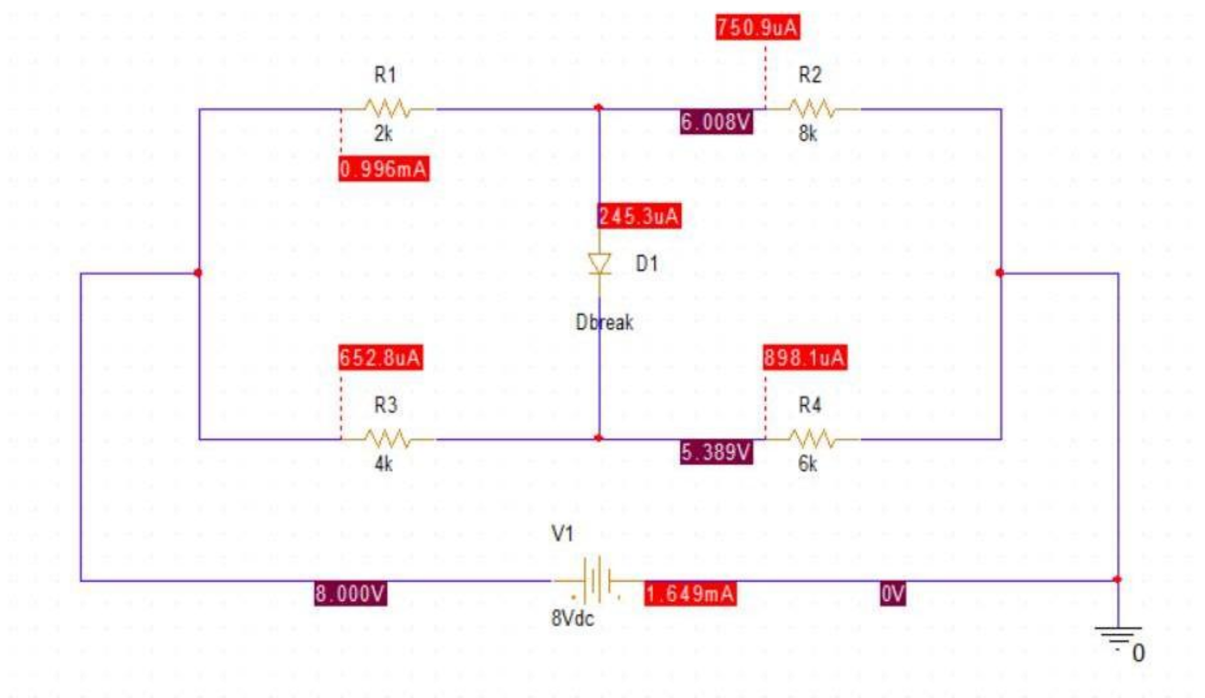
Solve similar for $V = 8V$ and $V = 12V$, we have:

3.3.2 PSpice simulation

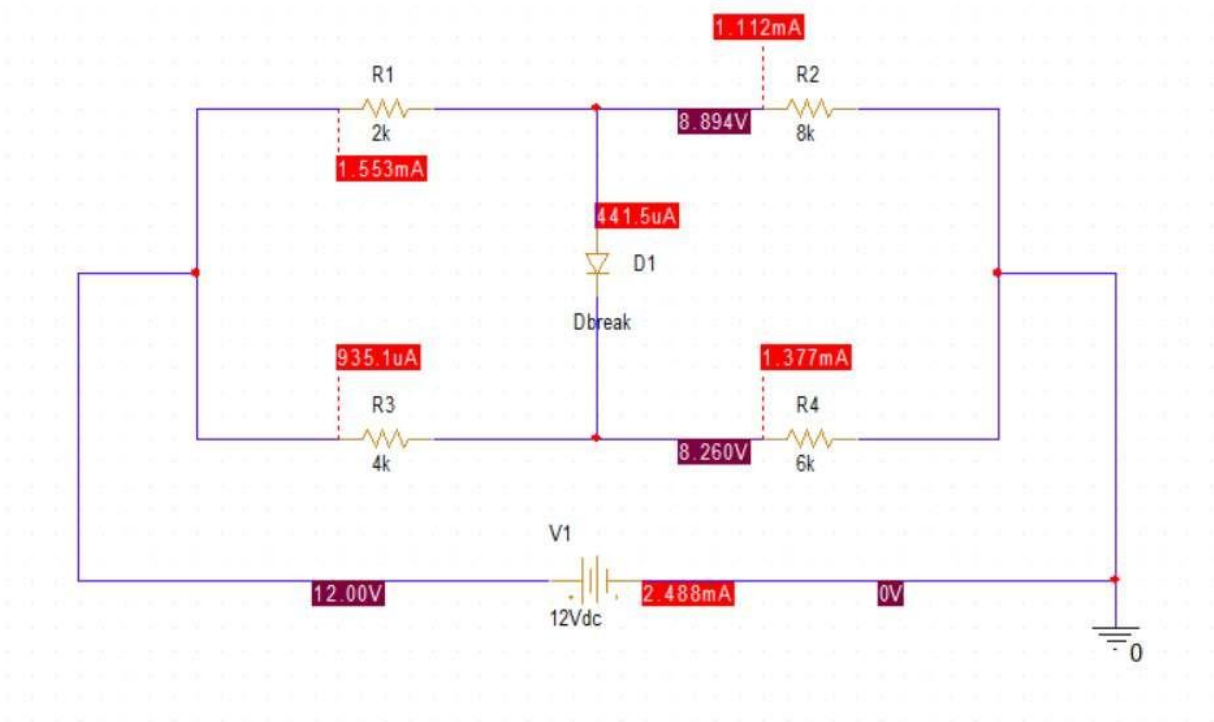
The bias point profile is used to run the simulation in this exercise. Students are proposed to capture the screen on PSpice showing the current and the voltage in the circuit.



The bias point simulation when V=10(V).



The bias point simulation when V=8(V)



The bias point simulation when V=12(V)

3.3.3 Comparison

Students are supposed to summarized the results from both theory calculations and PSpice simulations and fill in the table below.

	Theory Calculation						PSpice Simulation					
	I_{R1}	I_{R2}	I_{R3}	I_{R4}	V_1	V_2	I_{R1}	I_{R2}	I_{R3}	I_{R4}	V_1	V_2
V=8V	0.98 (mA)	755 (μA)	665 (μA)	890 (μA)	6.04 (V)	5.34 (V)	0.996 (mA)	750.9 (μA)	652.8 (μA)	898.1 (μA)	6.008 (V)	5.389 (V)
V=12V	1.54 (mA)	1.115 (mA)	945 (μA)	1.37 (mA)	8.92 (V)	8.22 (V)	1.553 (mA)	1.112 (mA)	935.1 (μA)	1.337 (mA)	8.894 (V)	8.26 (V)

3.4 Clamper Diode Circuit

The circuits in the figure below are known as clampers or DC restorers. The simulation on PSpice is also shown in the figure below. These circuits clamp a peak of a waveform to a specific DC level (e.g.

0.7V). Students are supposed to implement the circuit in PSPice to verify their results from theory calculation

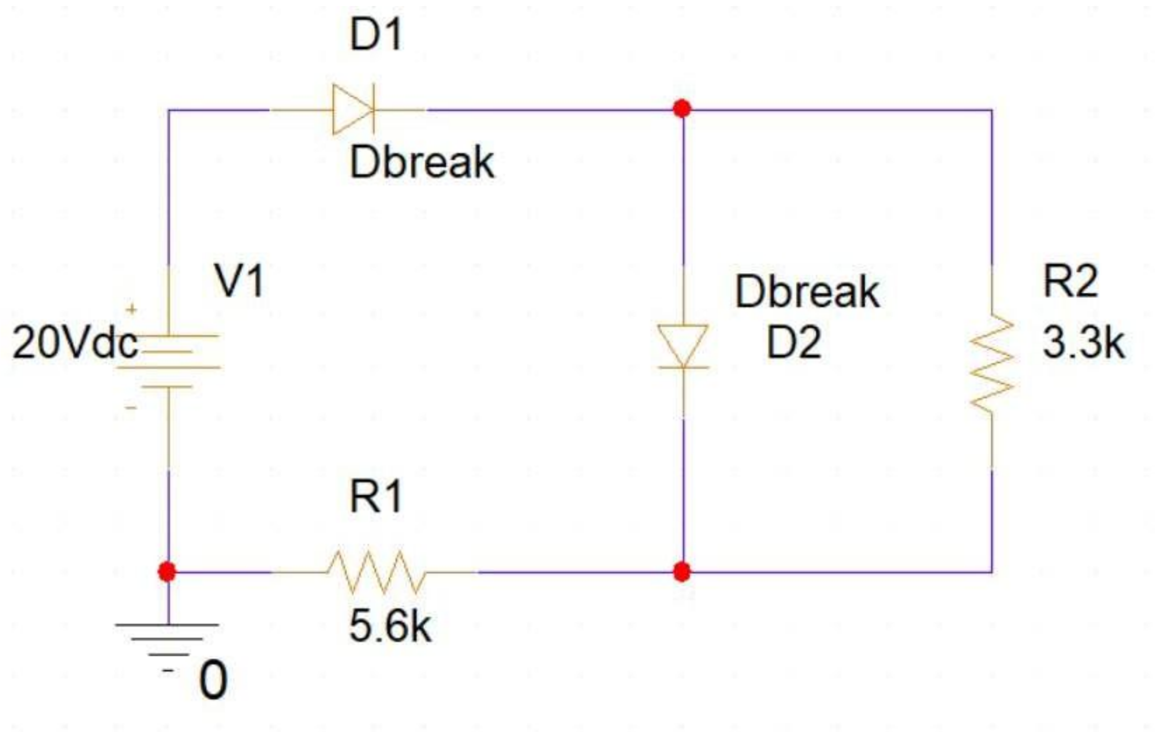


Figure 1.17: Clamper circuit using diode

3.4.1 Theory calculation

Define the voltage:

V_T =top node (Junction of D_1, D_2, top of circult)

V_B =bottom node (junction of R1, bottom of R2)

We have:

• Current through R_1(to ground):

$$I_{R1} = \frac{V_B - 0}{R_1} = \frac{V_B}{R_1}$$

• Current through R_2(from top node to bottom node):

$$I_{R2} = \frac{V_T - V_B}{R_2}$$

• Current through diode D_2(downwards from top to bottom) is I_{D2}.

By KCL at bottom node:

$I_{R2} + I_{D2} = I_{R1} \Rightarrow I_{D2} = I_{R1} - I_{R2}$

Diode voltage relation (practical diode model):

If $D_1$is ON (forward): $V_T = V_1 - V_F$

If $D_1$is OFF: $V_T$is determined by the circuit

If $D_2$is ON (forward): $V_T - V_B = V_F$

If $D_2$is OFF: $I_{D2} = 0$and $V_T - V_B < V_F$
Check whether D_1 conducts
Assume D_1 is ON so $V_t = V_1 - V\_F = 20 - 0.7 = 19.3$ V
Check D_2
Try first assuming D_2 OFF. Then $I\_D_2 = 0$
$\Longrightarrow I_{R2} = I_{R1}$

$$\frac{V_T - V_B}{R_2} = \frac{V_B}{R_1} \Rightarrow V_B = \frac{V_T/R_2}{1/R_1 + 1/R_2}$$

With $V_T = 19.3$ V, $R_1 = 5600\ \Omega$, $R_2 = 3300\ \Omega$
$\Longrightarrow V_B \approx 12.1444$ V
Then $V_T - V_B \approx 19.3 - 12.1444 = 7.156$ V, which is much larger than $V\_F = 0.7$ V. That violates the assumption D_2 OFF. So D_2 must be ON.
D_1 ON and D_2 ON
If both diodes are ON (practical model), then:
$V_T = V_1 - V_F = 19.3$ V
$V_T - V_B = V_F \Longrightarrow V_B = V_T - V_F = 19.3 - 0.7 = 18.6$ V

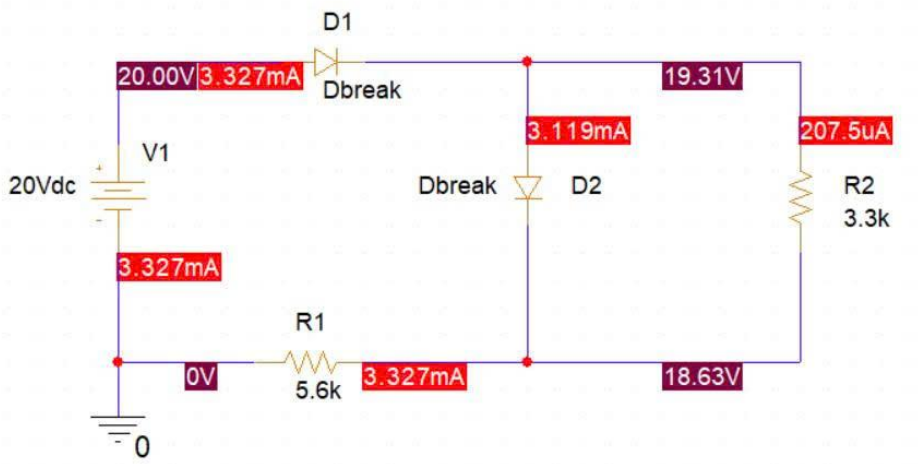
$$I_{R1} = \frac{V_B}{R_1} = \frac{18.6}{5600} \approx 0.0033214A = 3.3214mA$$

$$I_{R2} = \frac{V_T - V_B}{R_2} = \frac{0.7}{3300} \approx 0.000212A = 0.2121mA$$

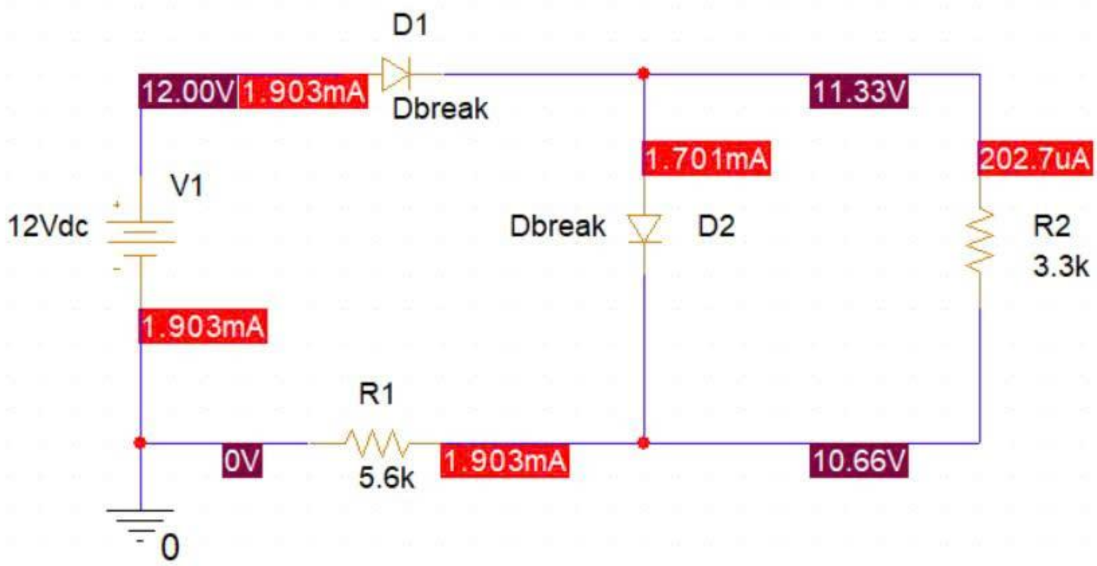
$I_{D2} = I_{R1} - I_{R2} = 3.3214 - 0.212 = 3.1093mA$
The same for $V_1 = 12V$
We have:
$I_{R1} = 1.8929$ mA
$I_{R2} = 0.2121$ mA
$I_{D2} = 1.6807$ mA
$V_{R2} = 0.7$ V

3.4.2 PSpice simulation

The bias point simulation is run in PSpice. Capture your screen with voltage and current are enabled in the results.



The bias point simulation when V=20(V)



The bias point simulation when V=12(V)

3.4.3 Comparison

Students are supposed to summarized the results from both theory calculations and PSpice simulations and fill in the table below.

	Theory calculation				PSpice simulation			
	IR1	IR2	ID2	VR2	IR1	IR2	ID2	VR2
V = 12V	1.89 (mA)	0.212 (mA)	1.678 (mA)	0.6996 (V)	1.903 (mA)	0.2027 (mA)	1.701 (mA)	0.67 (V)
V = 20V	3.32 (mA)	0.212 (mA)	3.108 (mA)	0.6996 (V)	3.327 (mA)	0.2075 (mA)	3.119 (mA)	0.68 (V)

What can be concluded from this table?

When the D2 diode is active, it will maintain a fixed voltage between the anode and cathode. So in any circumstances, the changes in voltage of source does not affect the value
of IR2 and VR2

3.5 Power switching circuit

The corruption of main power can be crucial in many different situations. For instance, when power is abruptly lost, you might want to save some backup data on a micro-controller. Such use cases need some form of automatic switching circuit to a secondary power source, such as a battery.

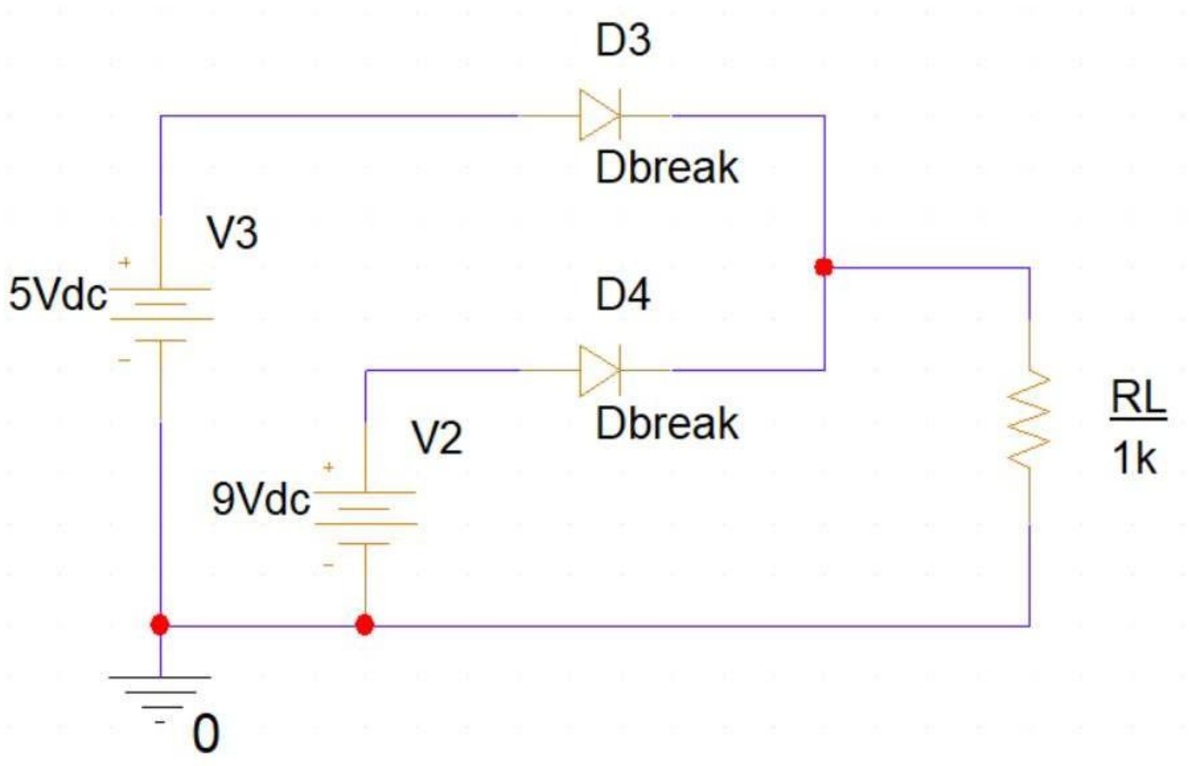


Figure 1.18: Power switching circuit

The simplest solution to this problem is simply to add a diode on each voltage source, as shown in the figure above. In this circuit, a 5V power supply can be used as a backup battery. Meanwhile, 9V is the main power supply for the system, which is demonstrated by a load resistor.

However, the main issue with this naive approach is that the voltage drop (forward voltage) across the diode might be too high for the system. Thanks to Schottky diode, this can be mitigated by using extremely low forward voltage one, which can be found on the market (e.g. Schottky diode having drop-down voltage around 250mV at 1A).

3.5.1 Theory calculation

In this part, it is assumed that the practical diode model is used. Present your equations to calculate the current I_{D3}, I_{D4}, I_{RL} and the voltage V_{RL} in three different cases: only 5V, only 9V and both 5V and 9V for the power supply

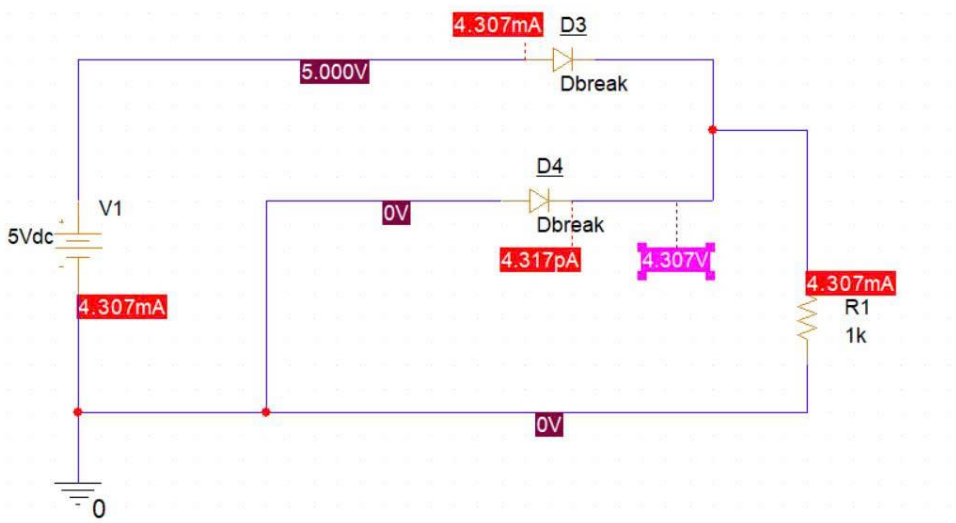
For the case only 5V source, since the 9V source is disconnected from the circuit, there is no current through D4 so D4 is disconnected from the circuit

$$I_{D_3} = \frac{V_{source} - V_{D_3}}{R_L} = \frac{5 - 0,7}{1000} = 4,3(mA)$$
$I_{D_4} = 0A$
$I_{RL} = I_{D_3} = 4,3mA$
For the case only 9V source, since the 5V source is disconnected from the circuit, there is no current through D3 so D3 is disconnected from the circuit
$I_{D_3} = 0A$
$$I_{D_4} = \frac{V_{source} - V_{D_4}}{R_L} = \frac{9 - 0,7}{1000} = 8,3(mA)$$
$I_{RL} = I_{D_3} = 8,3mA$

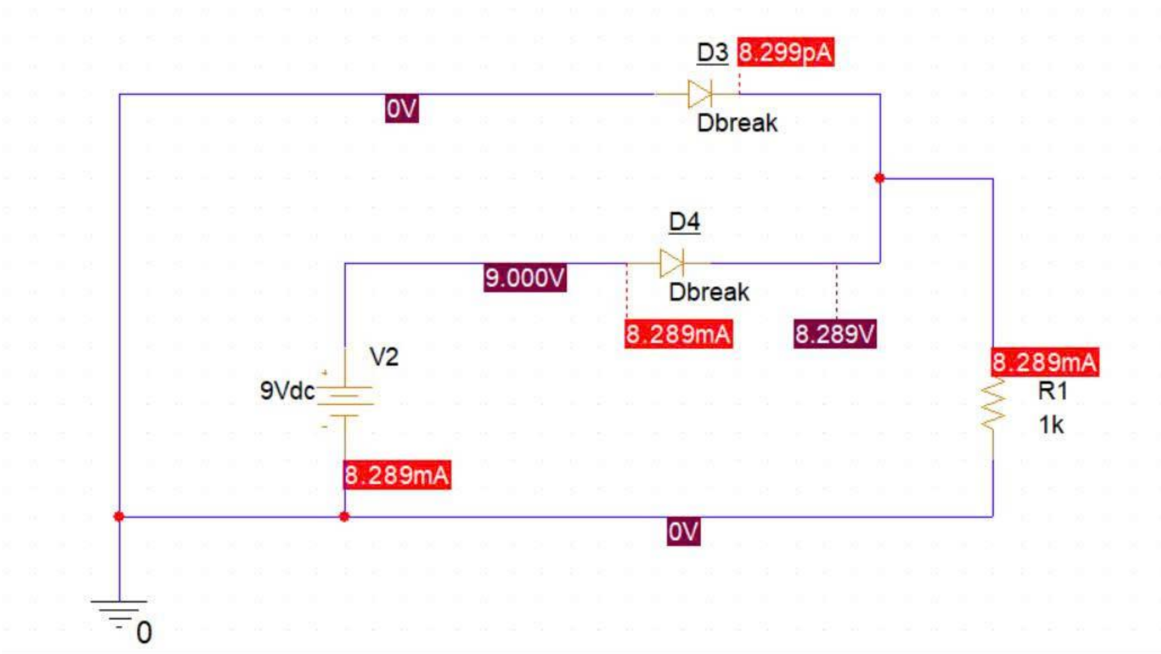
When both sources are connected, if the 9V source still work without any abrupt loss, the 5V source is still a back up so it will not be active. Then there is no current run through D_3. So the values are the same with case 9V only

3.5.2 PSpice simulation

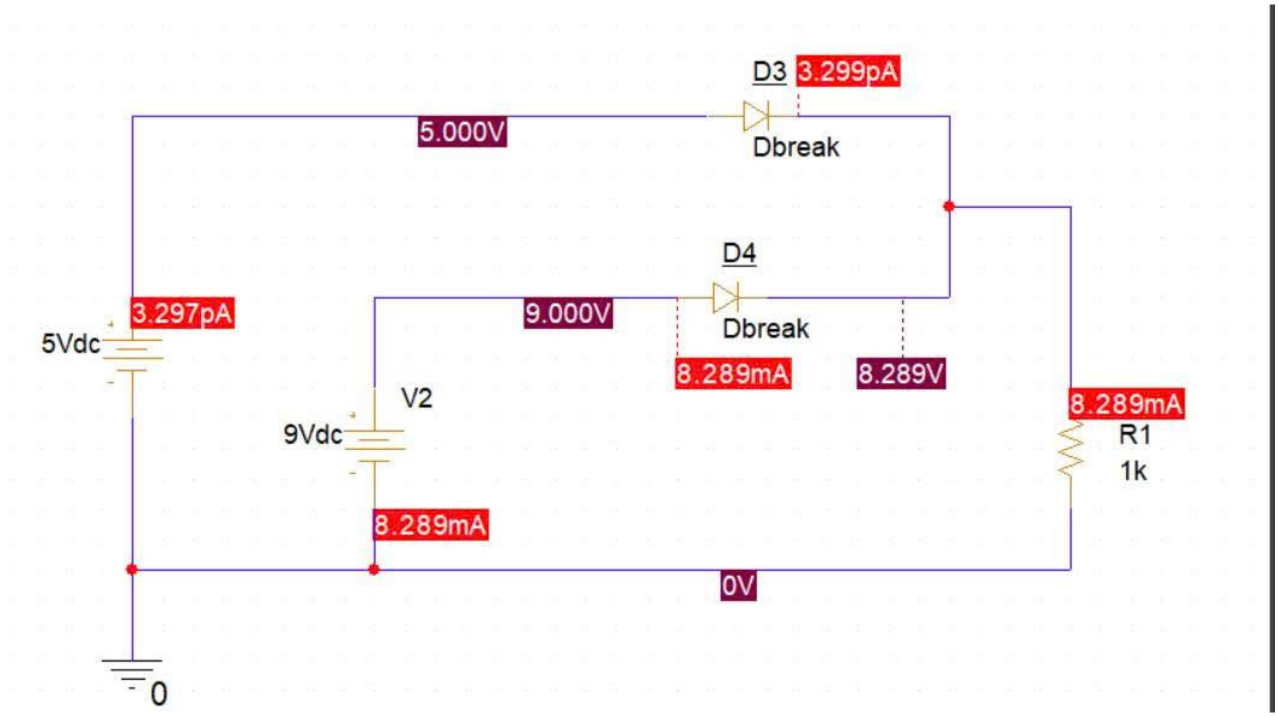
The bias point simulation is run in PSpice with both power sources are enabled. Capture
your screen with voltage and current are enabled in the result



The bias point simulation when there is only 5V source



The bias point simulation when there is only 9V source



The bias point simulation when there are both sources

3.5.3 Comparison

Students are supposed to summarized the results from both theory calculations and PSpice simulations and fill in the table below. Both power sources are connected to the circuit.

Theory calculation				PSpice simulation			
ID1	ID2	IRL	VRL	ID1	ID2	IRL	VRL
0 (A)	8.3 (mA)	8.3 (mA)	8.3 (V)	3.297 (pA)	8.289 (mA)	8.289 (mA)	8.289 (V)

The last feature of the diodes in this circuit is to protect the reverse current in case both the power sources are switch on together. This use case is very popular when a microcontroller platform is programmed and powered by an USB port and also equipped with an adapter for external power supply.

3.6 Half-wave Rectifier

In this exercise, an alternating source is used to generate a half-wave rectifier output using a diode. The schematic of the simulation is given bellow:

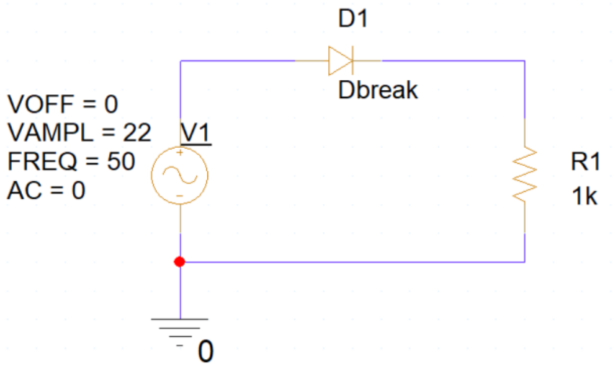


Figure 1.19: Half-wave Rectifier with Voltage Sin Source

Step 1: Create the schematic. The new component in this schematic is the VSIN - Sine Voltage Source, which can be found in the Favorites list. This component has four different parameters are required to configure, as follows:

- VOFF: Offset voltage of the source
- VAMPL: Amplifier voltage of the source
- FREQ: Frequency of the alternative current
- AC: The source type (having value 0 or 1), to switch between VAC and VSIN. In VSIN source, the frequency can be modified.

After setting these parameters are set to the values shown in the figure above, add a voltage probe to track the output by clicking on the Voltage/Level Marker on the PSpice Toolbar, as follows:

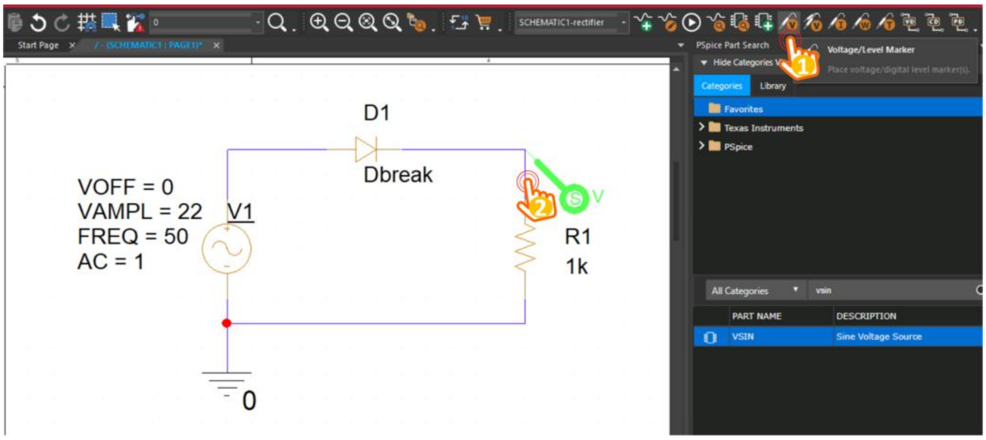


Figure 1.20: Add a voltage marker for simulation tracing

Step 2: Create a new simulation profile. In order to generate the output signal of the voltage probe, the analysis type is set to **Time Domain (Transient)**. Moreover, due to the frequency of the power source is 50Hz, the **simulation time is set to 100ms**, to depict 10 cycles of the output signal. Finally, the resolution is set to 0.1ms in the **Maximum Step size**. The configuration windows for this simulation profile is presented as follows:

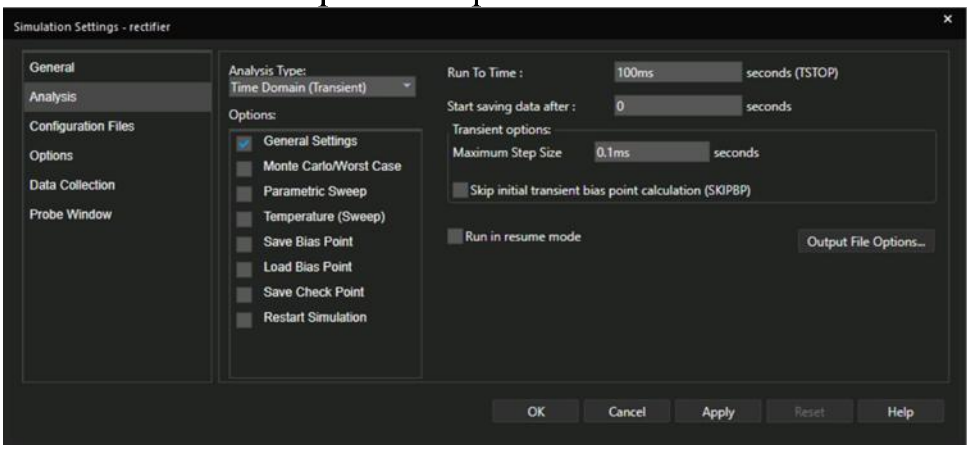


Figure 1.21: Create Time Domain simulation profile

Step 3: Run the simulation and observe the results Finally, run the simulation and the output on the simulation windows should be presented in the figure bellow:

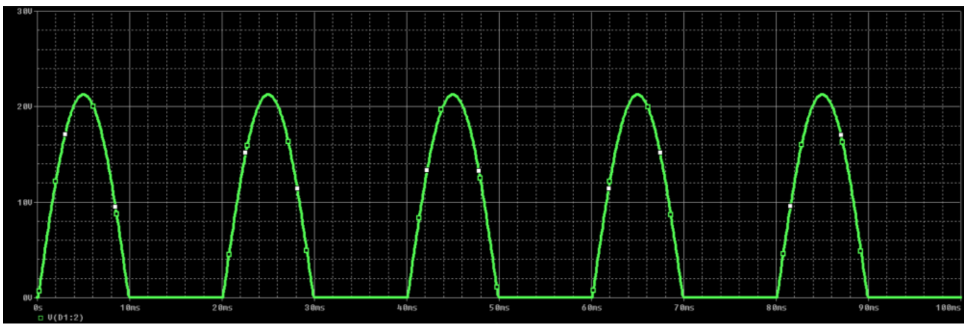


Figure 1.22: Wave form for the half-ware rectifier circuit

3.6.1 Theory calculation

Notes:

Explanations, formulas, and equations are expected rather than only results.

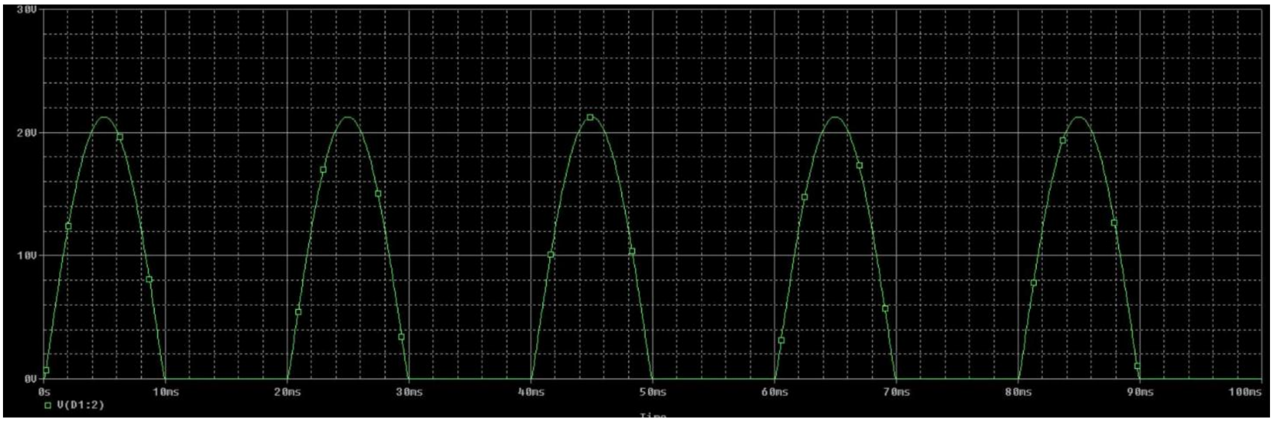
Approximation: diodes have $V_f = 0.78V$

$V_s(t) = V_p \sin(\omega t)$, with $V_p = 22V, f = 50Hz$

Maximum value of V_{R1}:
When the diode conducts at the positive peak,
$V_{R1,\text{max}} = V_p - V_f = 22 - 0.78 = 21.22V$
Minimum value of V_{R1}:
During the negative half-cycle, and also whenever the source amplitude is less than the diode forward drop, the diode is reverse-biased, so the load voltage is zero (assuming ideal diode leakage ≈ 0):
$V_{R1,\text{min}} = 0V$
Duration for a cycle of V_{R1}:
Period of the source (one cycle):
$$T = \frac{1}{f} = \frac{1}{50} = 0.02s = 20.0ms$$
Conduction condition and conduction duration per cycle (pulse width):
Diode conducts when $V_p \sin\,\theta \geq V_f$.
Let $\theta = \omega t$,
$$\sin\,\theta > \frac{V_f}{V_p}$$
We have:
$$\frac{V_f}{V_p} = \frac{0.78}{22} = \frac{39}{1100}$$
Turn-on angle:
$$\theta_{on} = \arcsin\left(\frac{V_f}{V_p}\right) \approx 0.035462\,\text{rad} \approx 2.031822°$$
Diode stays on from $\theta_{on}$to $\pi - \theta_{on}$.
The conduction angle is:
$$\theta_{cond} = \pi - 2\theta_{on} \approx \pi - 2(0.035462) = 3.070669 \approx 175.936°$$
Convert conduction angle to time.
Angular frequency $\omega = 2\pi f$.
Conduction half-duration:
$$t_{cond} = \frac{\theta_{cond}}{\omega} = \frac{3.070669}{2\pi \times 50} \approx 0.009774425\,s \approx 9.774\,\text{ms}$$

3.6.2 Pspice simulation

Export your simulation results to Notepad for instance, and find the minimum and the maximum point of the output voltage, then fill your answer to the section bellow:



The tracking point can also be used directly on the simulation output windows, by the Toggle Cursor option on the toolbar. A Probe Cursor window is opened to update a tracking point.

3.7 Full-wave Rectifier

The following circuit is known as a full-wave bridge diode rectifier. Given that the transformer has the ratio N1/N2 = 10. Write the voltage difference equation VAB and VC D. After that, perform a time-domain (transient) analysis to check the equation you've written

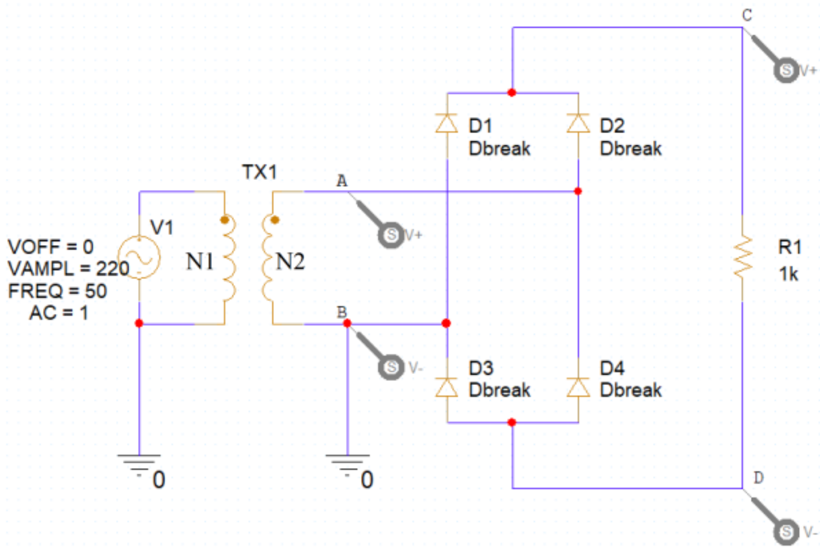


Figure 1.23: Full-wave bridge rectifier

3.7.1 Theory calculation

Notes:

Explanations, formulas, and equations are expected rather than only results.

Approximation: Diodes have $V_f = 0.7V$

Transformer ratio:

$$\frac{N_1}{N_2} = 10$$

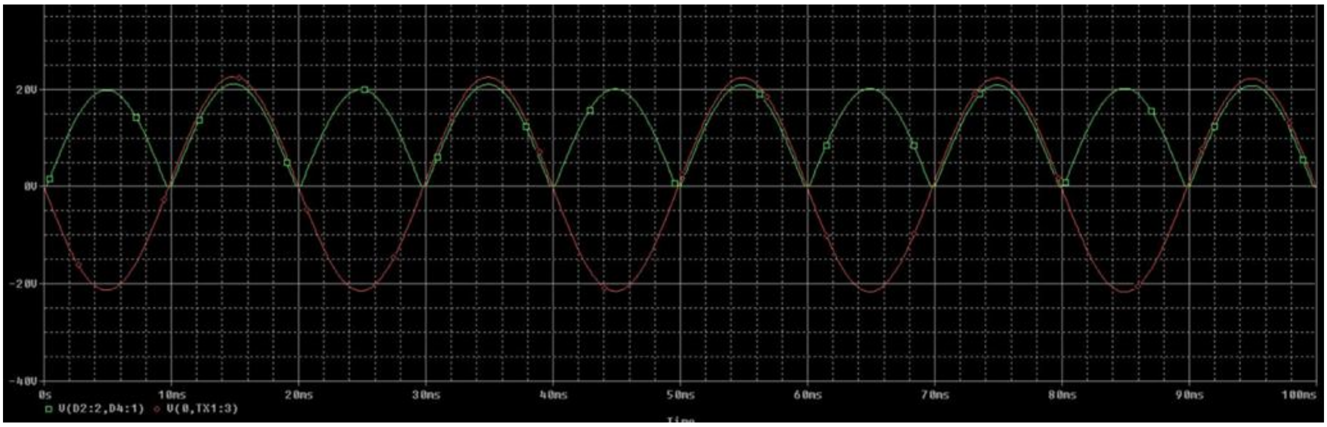
$$V_{primary} = 220\ V = V_{p(primary)} = V_{peak}$$

Therefore, secondary peak voltage:

$$V_{p(secondary)} = \frac{220}{10} = 22\ V_{peak}$$

Diode forward drop: $V_F = 0.7V$
Voltage across secondary winding (transformer output):
$V_{AB} = 22\sin\left(100\pi t\right)(\mathrm{V})$
Full-wave rectified output voltage (across load R_L):
Each conducting path has two diodes in series,
so total drop $= 2V_F = 1.4V$
Therefore, the peak output:
$V_{CD(peak)} = 22 - 1.4 = 20.6V$
$V_{CD} = \mid V_{AB} \mid -1.4 = \mid 22\sin\left(100\pi t\right) \mid -1.4$

3.7.2 Stimulation



The sinusoidal waveform of the voltage difference V_{AB} has the period:

$$T = \frac{1}{f} = \frac{1}{50\mathrm{Hz}} = 0.02(s)$$

The required time to perform the transient analysis in 10 periods of the waveform V_{AB}:
$10T = 10 \times 0.02 = 0.2(s)$

If we want the sampling rate to be ten times higher than the frequency of the sinusoidal voltage difference V_{AB}, the time interval between two consecutive sampling time points should be:

- Sampling frequency $f_s = 10 \times f = 10 \times 50 = 500\ \mathrm{Hz}$
- Time interval (time step) $\Delta t = \frac{1}{f_s} = \frac{1}{500} = 0.002(s) = 2\ \mathrm{ms}$

$$\Delta t = 2\ \mathrm{ms}$$

3.8 Zener Diode as a Regulator

The Zener diode has a well-defined reverse-breakdown voltage, at which it starts conducting current, and continues operating continuously in the reverse-bias mode without getting damaged. Additionally, the voltage drop across the diode remains constant over a wide range of voltages, a feature that makes Zener diodes suitable for use in voltage regulation.

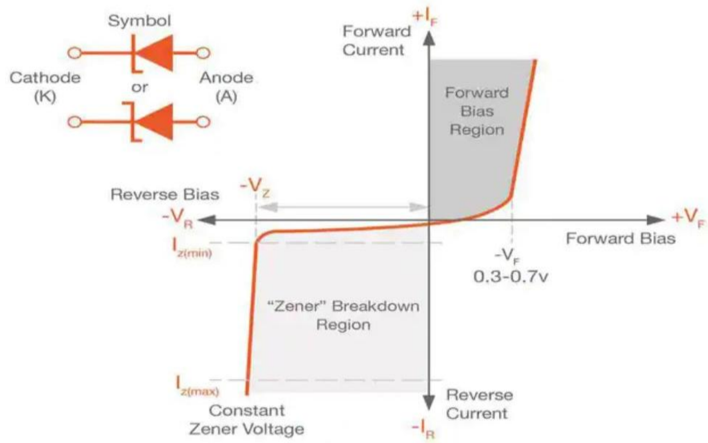


Figure 1.27: Electrical characteristic of Zener diode

In this exercise, a Zener diode is used to design a voltage regular circuit. The schematic in this exercise is given following:

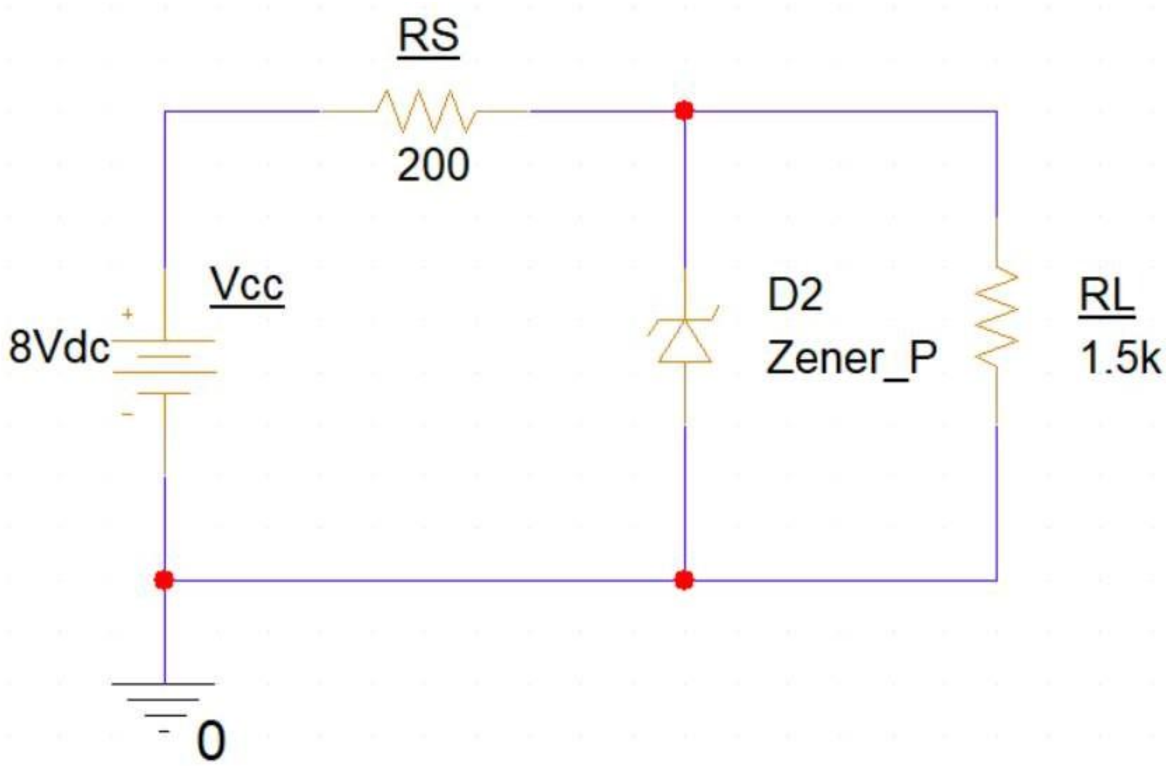


Figure 1.28: Voltage regulator using Zener diode

The Zener component in the circuit can be found in the Favourites list by search the keyword **Zener**. The full name of the component used in the circuit above is **Zener_P-Zener Diode (parameterized)**. The default Zener voltage of this component is VZ = 5V. However, this value can be changed in the properties of the component (right click and select Edit Properties) for other simulations.

For theory calculation, students are supposed to provide equations for these values

When the Zener diode is in reverse breakdown (regulating mode):

$V_L = V_Z = 5V$

Then:

(a) Load current:

$$I_L = \frac{V_L}{R_L} = \frac{V_Z}{R_L}$$

(b) Series resistor current:

$$I_S = \frac{V_{CC} - V_Z}{R_S}$$

(c) Zener current:

$I_Z = I_S - I_L$

(Because total current through $R_S$splits between $R_L$and Zener)

(d) Power dissipated in R_S:

$P_{R_S} = I_S^2 R_S$

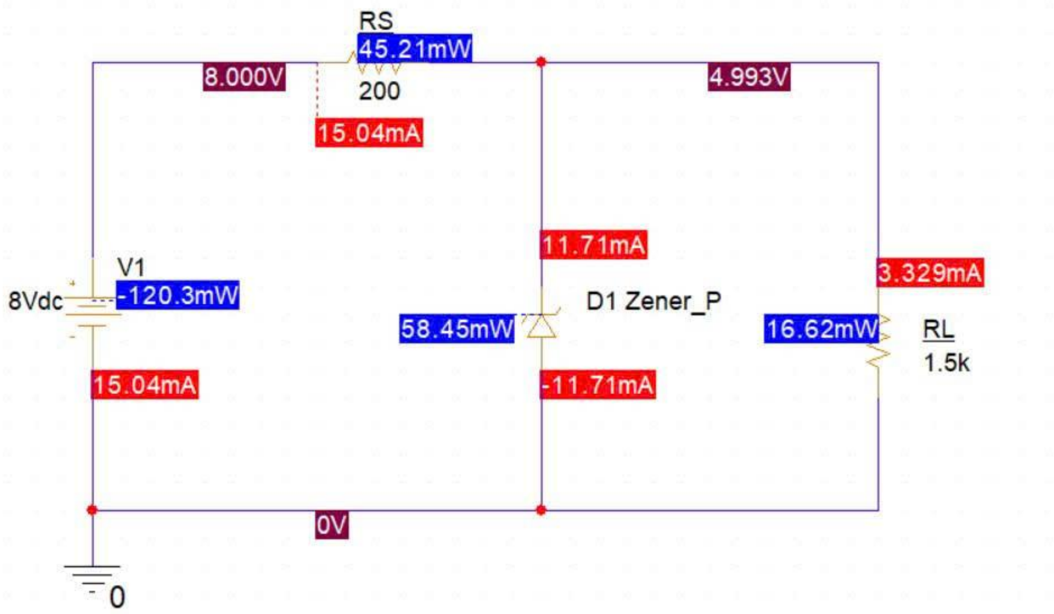
(e) Power dissipated in Zener diode:

$P_Z = V_Z I_Z$

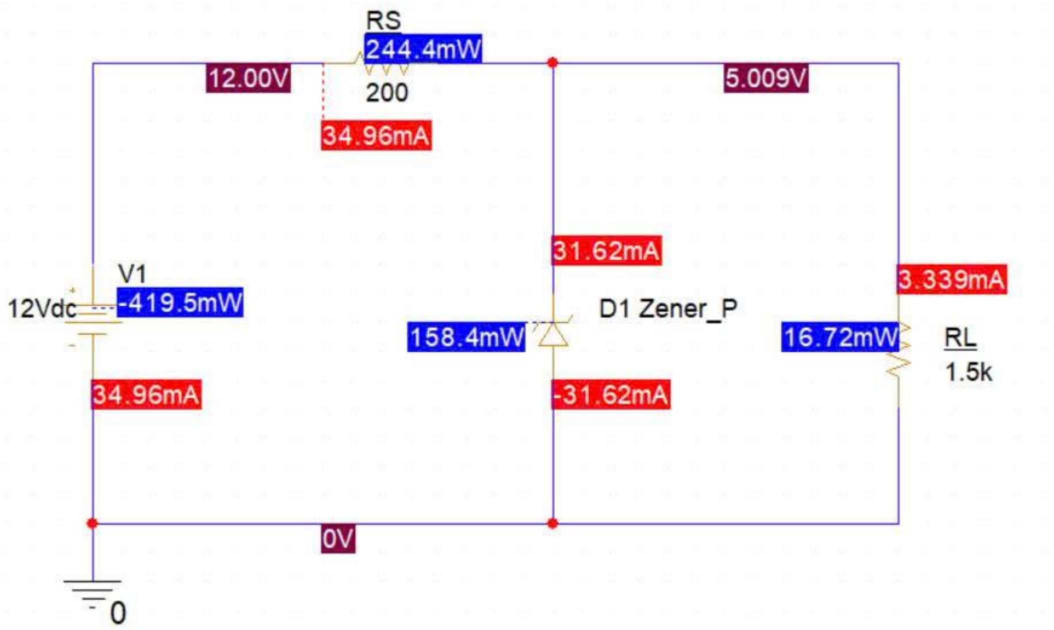
Then, perform the calculation for the Zener diode voltage regulator with two different input voltage, including 8V and 12V power supply. Finally, run the simulations in PSpice (in Bias Point simulation profile) to confirm with the theory calculation.

The results are summarized in the table below.

	Theory Calculation						PSpice Simulation					
	I_S	I_L	I_Z	V_L	P_{RS}	P_z	I_S	I_L	I_Z	V_L	P_{RS}	P_z
Vcc=8V	15 (mA)	3.33 (mA)	11.67 (mA)	4.995 (V)	45 (mW)	58.35 (mW)	15.04 (mA)	3.329 (mA)	11.71 (mA)	4.993 (V)	45.21 (mW)	58.45 (mW)
Vcc=12V	35 (mA)	3.33 (mA)	31.67 (mA)	4.995 (V)	245 (mW)	158.35 (mW)	34.96 (mA)	3.329 (mA)	31.62 (mA)	5.009 (V)	244.4 (mW)	158.4 (mW)



The bias point simulation when V=8(V)

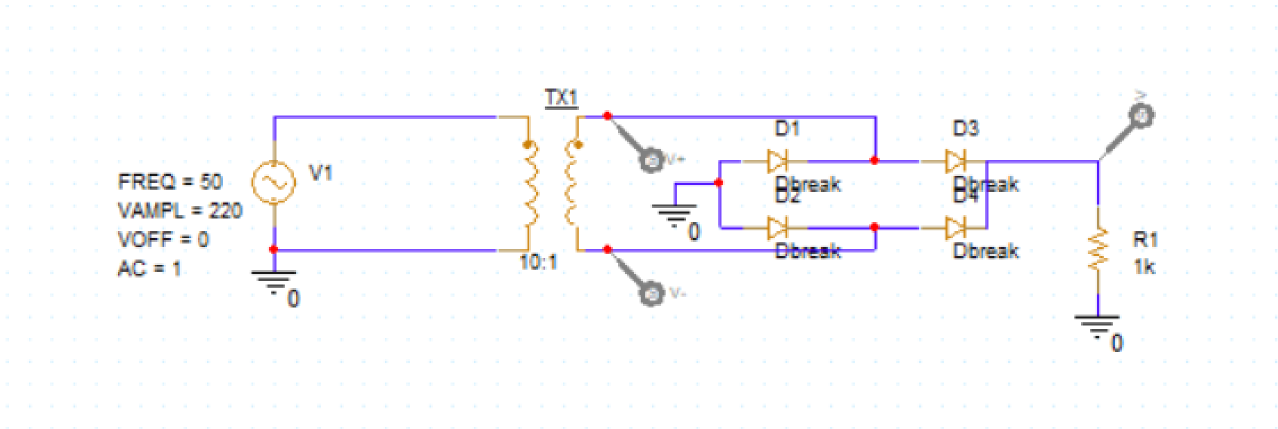


The bias point simulation when V=12(V)

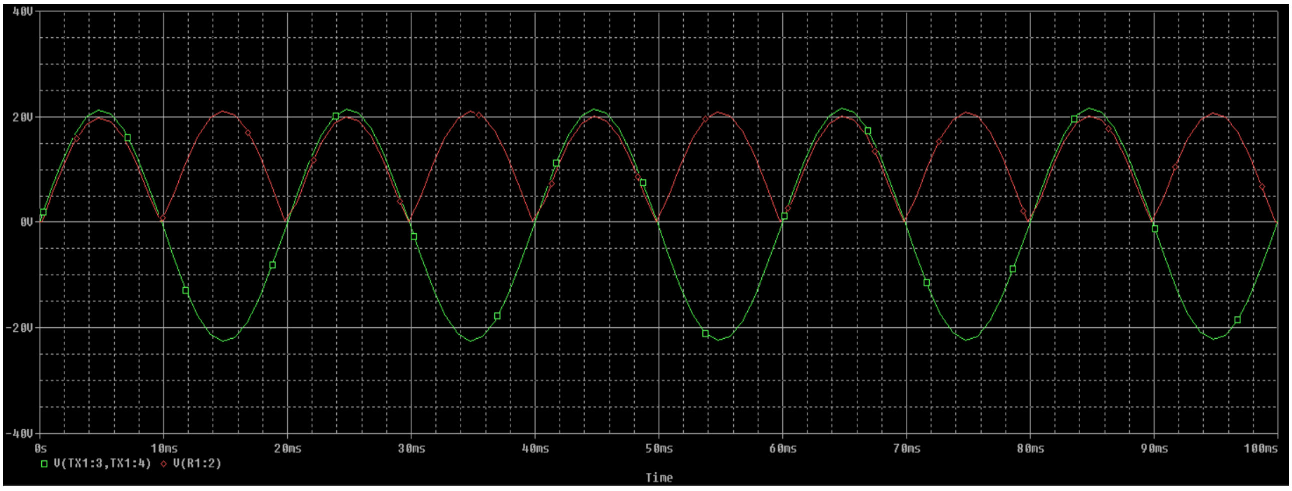
3.9 AC/DC Power Circuit Application

In this exercise, we are building step by step an AC to DC voltage source transformation circuit. Students perform a time-domain simulation and write out comments and explanations for each step.

- **Step 1:** The rectified voltage without any filtering or being regulated.



Stimulation result:

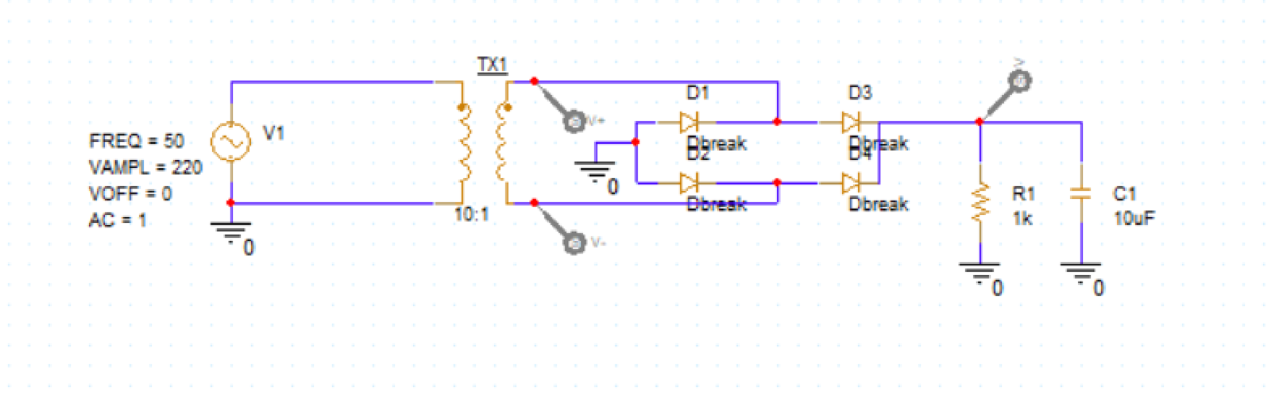


Any comments or explanations:

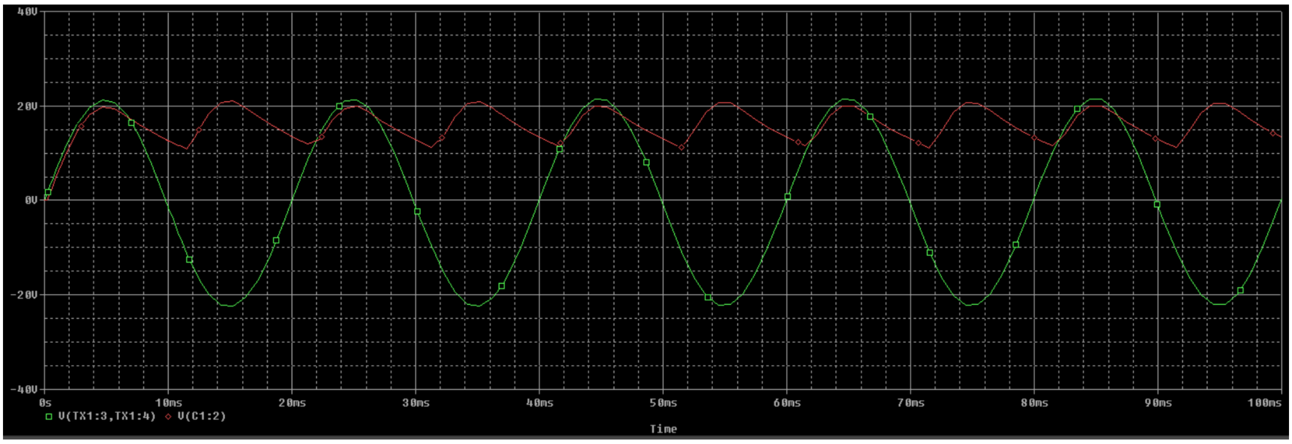
According to the simulation results, the output voltage across the load resistor R1 is a pulsating DC waveform.
The input (green line) is a sinusoidal AC voltage from the transformer secondary, while the output (red line) shows only positive half-cycles of the AC waveform.

This happens because the diode bridge (D1–D4) allows current to pass only when the input polarity is suitable, rectifying both positive and negative halves of the AC input into positive voltage pulses.
However, since there is no capacitor or regulator yet, the voltage still fluctuates between zero and the peak value, rather than being a stable DC level.

- **Step 2**: Rectified voltage regulated with a 10µF capacitor.



Simulation result:

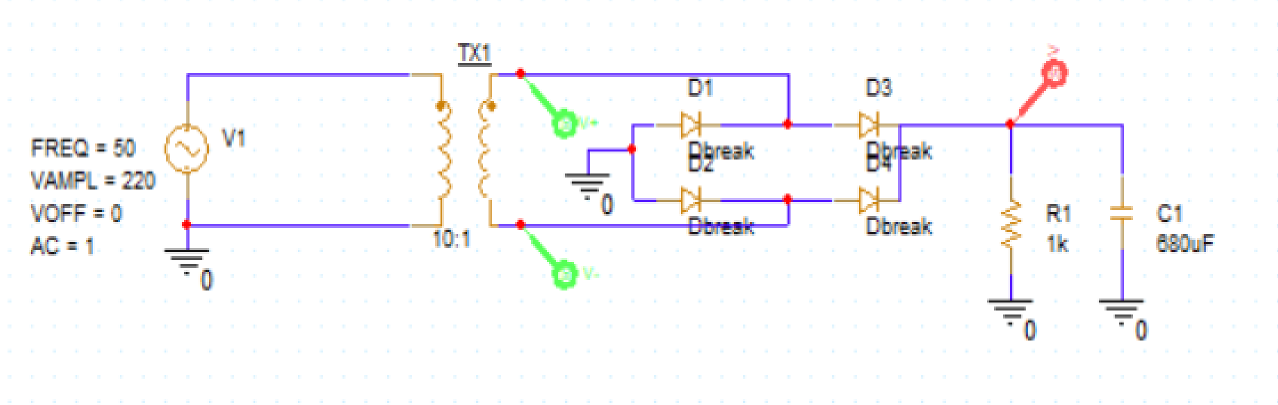


Any comments or explanations:

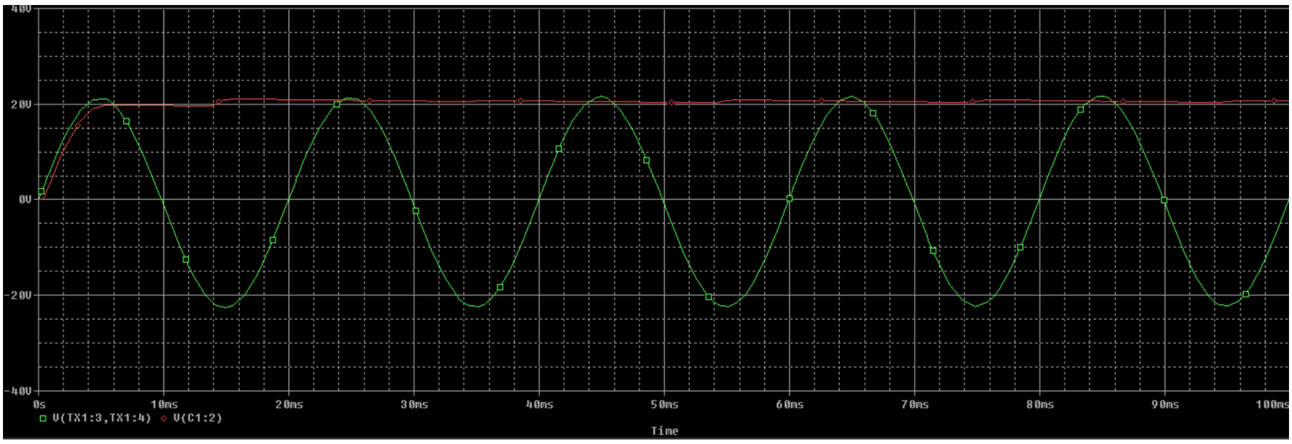
From the simulation, the output voltage across the load resistor R1 becomes much smoother compared to Step 1. The capacitor C1 = 10 µF acts as a filter, storing energy during the peak of each rectified pulse and releasing it when the input voltage drops. This reduces the voltage ripple and helps maintain a more constant DC output.

Although the waveform still shows some small ripples, it is clearly closer to a steady DC voltage than before.

- **Step 3**: Replace the 10µF capacitor with a 680µF one and re-run the simulation, recognize the change in the result and explain.



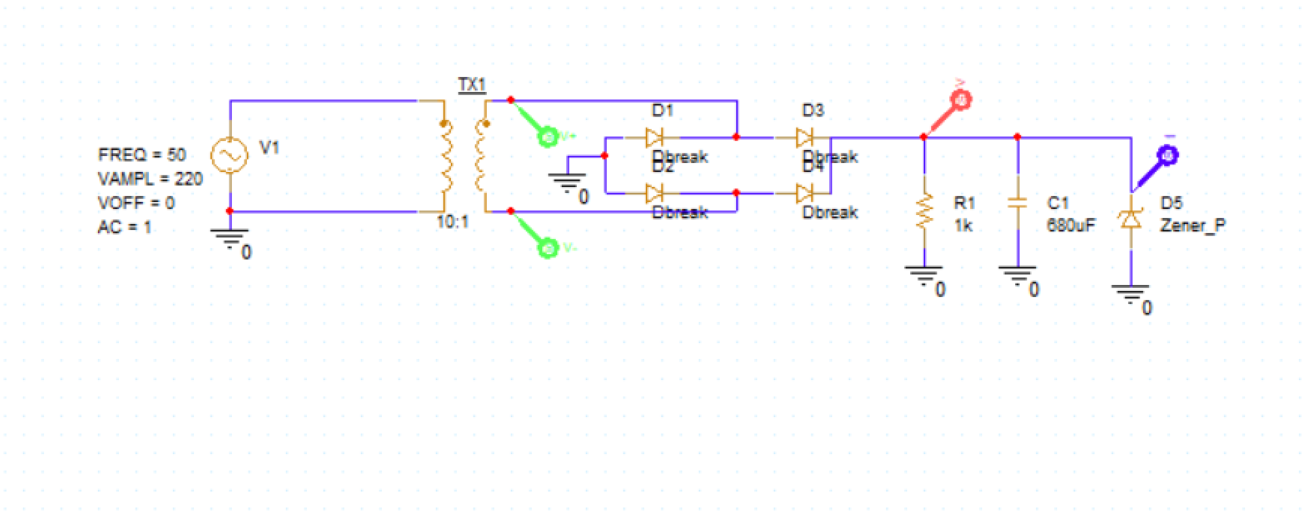
Stimulation result:



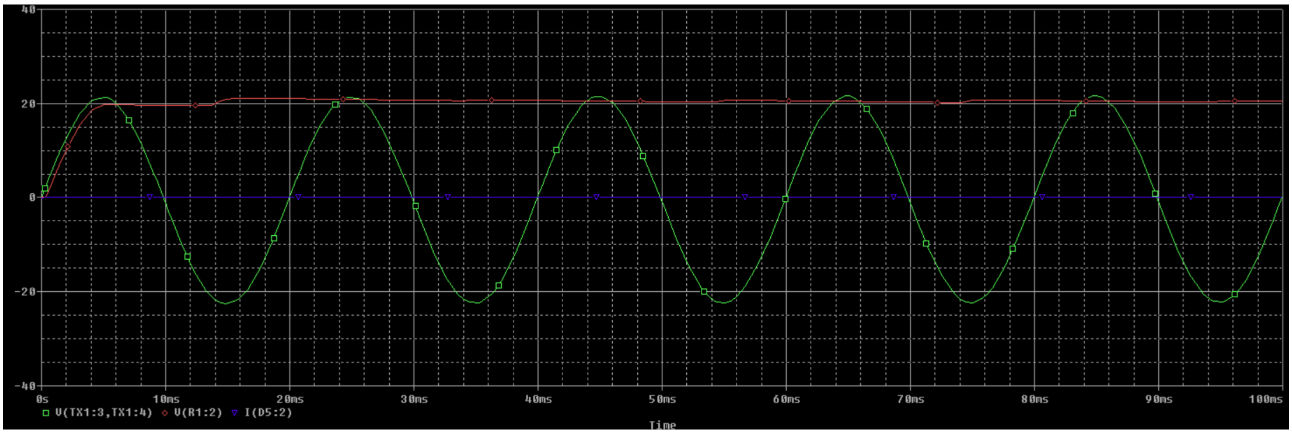
Any comments or explanation: After replacing the 10 µF capacitor with a 680 µF one, the output voltage becomes much smoother and almost constant. The larger capacitor can store more charge and supply current to the load when the input voltage decreases, so the voltage drop between peaks is smaller. As a result, the ripple voltage is reduced, and the output appears nearly flat like a steady DC signal.

This shows that increasing the capacitance improves the filtering effect and gives a more stable DC output, although it also causes larger charging current pulses through the diodes at the peaks of the input waveform.

- **Step 4**: Add a zener diode as in Figure 1.31 with the zener voltage properties set to 22 volts then simulate the circuit and comment or explain the result.



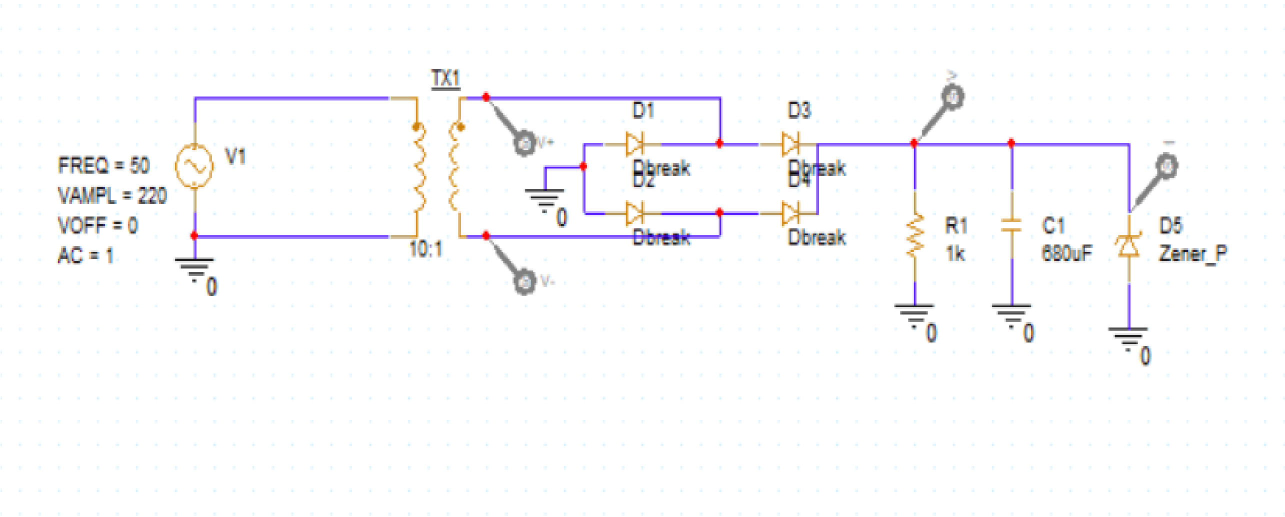
Stimulation result:



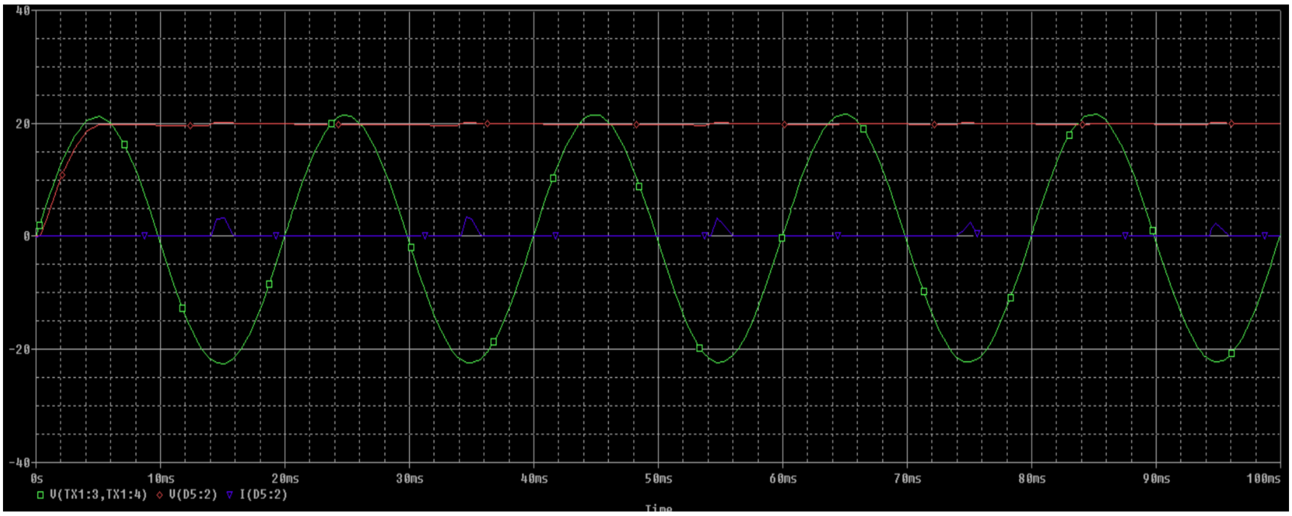
Any comments or explanations: After adding a zener diode rated at 22 volts, the output voltage is limited to around 22 V. The zener diode is connected in reverse bias and begins to conduct once the voltage exceeds its breakdown voltage, keeping the output voltage nearly constant at about 22 V.

This means that even if the rectified and filtered voltage from the previous step tends to rise above 22 V, the zener diode clamps it to this level, providing voltage regulation. As a result, the output becomes a stable DC voltage of approximately 22 V with very small ripple.

• **Step 5:** Change the zener voltage properties of the zener diode to 20 voltage and then re-run the simulation. Comment and explain any changes in the result.



Simulation result:



Any comments or explanations: When the zener voltage is changed from 22 V to 20 V, the output voltage decreases slightly and becomes regulated at about 20 V. The waveform is almost flat, showing a stable DC level with very little ripple. The zener diode conducts whenever the voltage tries to rise above 20 V, limiting the output to this level.

The small pulses that appear in the zener current waveform indicate the moments when the zener is conducting to clamp the voltage. When the voltage falls below 20 V, the current through the zener returns to zero. This confirms that the zener diode is properly regulating the voltage at around 20 V.

3.10 Exercise 8: AC/DC Power Circuit Application with LM2596_5P0_TRANS

Figure 1.32 describes an incomplete Texas Instrument LM2596 - 5.0 Switching Power Supply circuit. It lacks a Zener diode voltage regulator and an inductor reducing the voltage variation. At first, let perform a time-domain (transient) simulation with this incomplete circuit and figure out the problem with the output voltage (the voltage marker at R1).

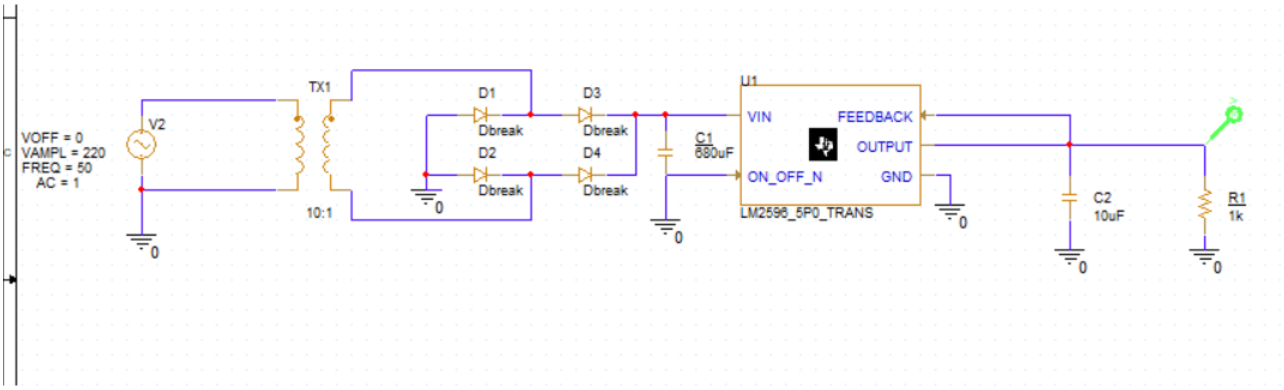
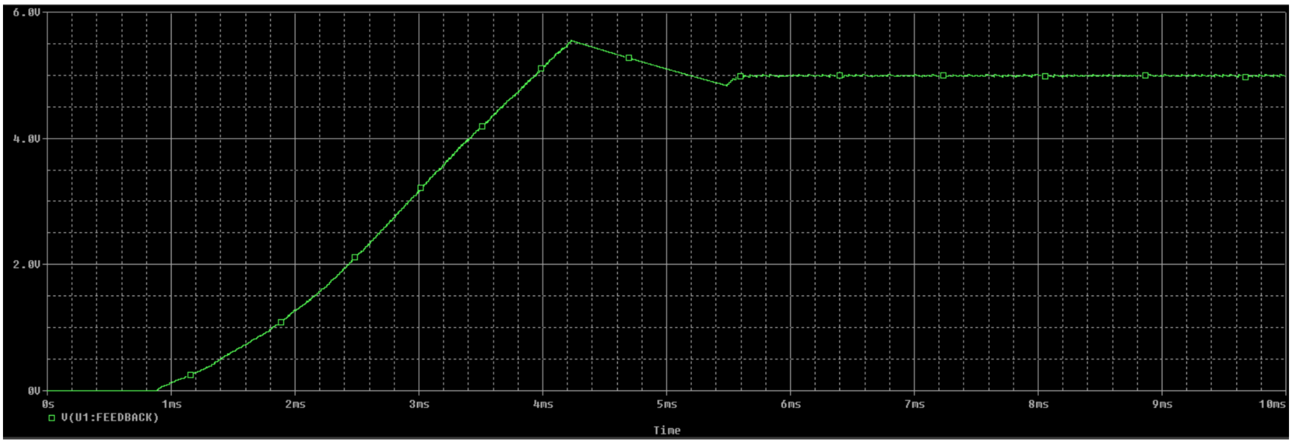


Figure : Incomplete switching power supply circuit

Simulation result(s):



Any comments or explanations: From the simulation, the output voltage gradually rises to about 5 volts and then becomes almost steady. However, small variations and instability can still be seen. This happens because the circuit is incomplete and lacks key components such as a zener diode voltage regulator and an inductor for filtering.

Without these components, the LM2596 cannot fully regulate and smooth the

DC output. The voltage shows slight ripples and may not stay constant under load changes. The zener diode and inductor are normally used to stabilize and filter the voltage, keeping it constant and clean for the output.

In summary, the problem with this incomplete circuit is that the output voltage is not well-regulated and contains minor ripple, meaning additional regulation and filtering components are needed for proper operation.

Next, add an inductor 33µH to the circuit as shown in Figure 1.35 then re-run the simulation and explain any improvements.

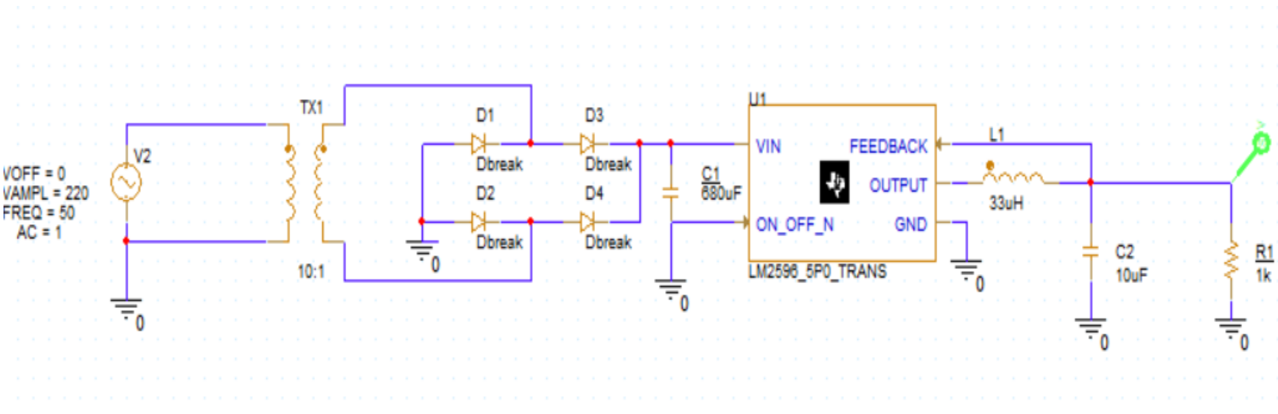
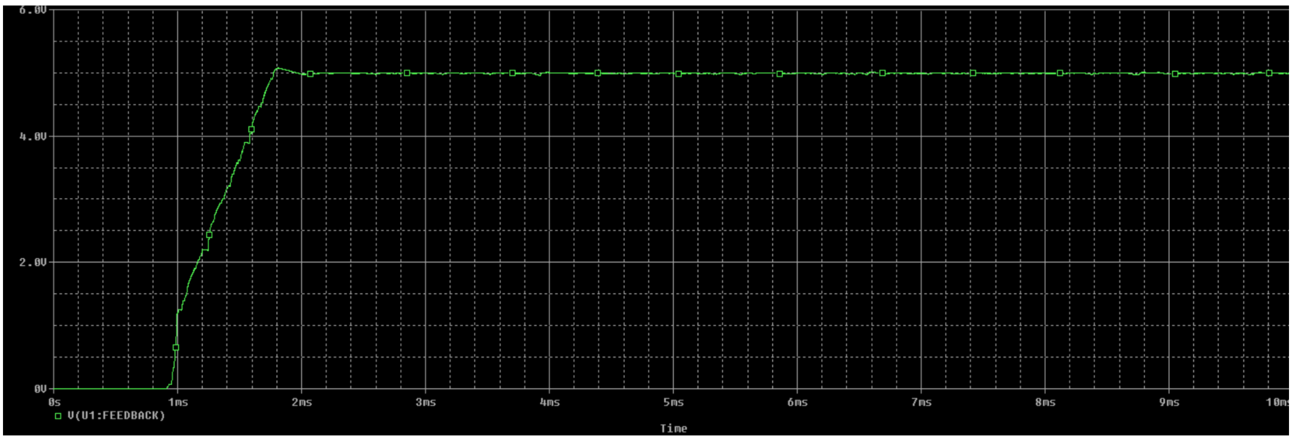


Figure: A 33µH inductor added to the circuit

Simulation result(s):



Any comments or explanations: After adding the 33 µH inductor, the output voltage quickly rises to about 5 V and remains very stable. The inductor smooths the current and, together with the capacitor, filters out voltage ripple. Compared with the previous circuit, the voltage is steadier and cleaner, showing that the inductor improves output stability and reduces ripple**.**

Continue, add a 5V Zener diode to the circuit as shown in Figure 1.36, change the capacitor to 220µF, add a current marker to the Zener diode, re-run the simulation and explain the role of the Zener diode in the circuit.

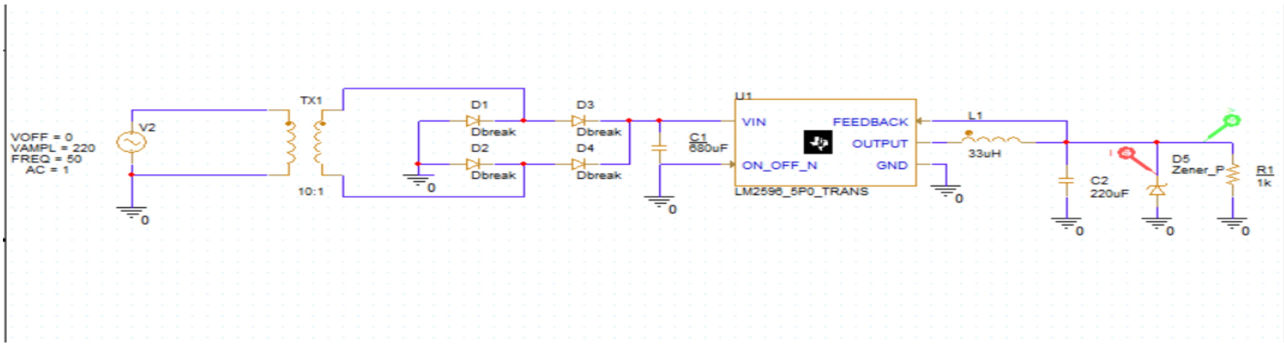
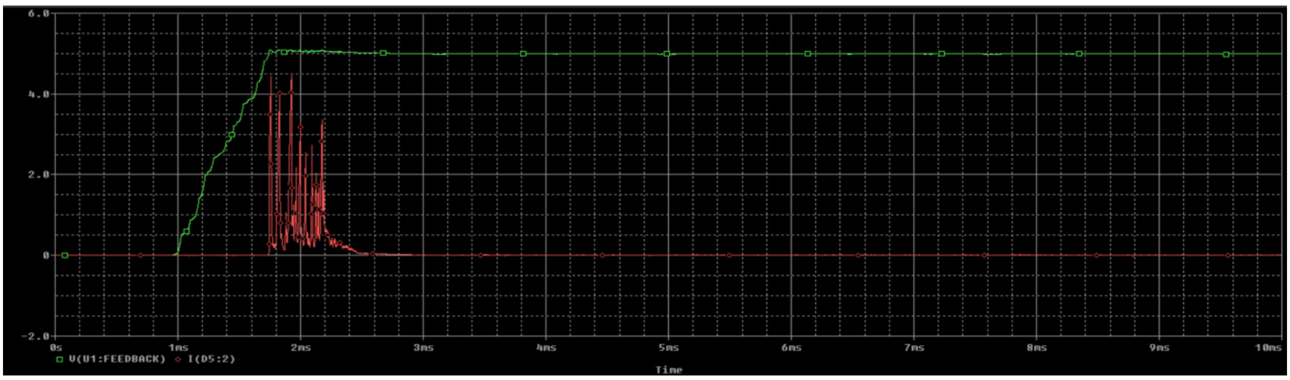


Figure: A 5V Zener diode added to the circuit

Simulation result(s):



Any comments or explanations: After adding the 5 V Zener diode and changing the capacitor to 220 µF, the output voltage remains steady at around 5 V. The current waveform through the Zener shows short pulses during startup, indicating that the Zener conducts briefly when the voltage exceeds 5 V, then stops once regulation is established.

The Zener diode's role is to keep the output voltage fixed at 5 V by conducting excess current whenever the voltage rises above its breakdown voltage. This provides overvoltage protection and helps maintain a stable DC output.