

Circuits Theory and Electronic Fundamentals

Integrated Master in Engineering Physics, IST, University of Lisbon

Lab 3: AC/DC Converter

Alexandre Sequeira (96503), Duarte Marques (96523), João Chaves (96540)

May 8th 2021

Contents

1	Introduction	1
2	Theoretical Analysis	2
3	Simulation Analysis	6
4	Conclusion	9

1 Introduction

The objective of this laboratory assignment is to use an AC/DC converter to obtain an output DC signal of value 12 V from an input AC signal of amplitude 230 V and frequency 50 Hz. The envelope detector and voltage regulator circuits were chosen as shown in Figure 1, in which designations have been assigned to each node.

The envelope detector is formed by a full-wave bridge rectifier circuit, with 4 diodes connected to the resistor R_1 , in parallel with the capacitor. On the other hand, the voltage regulator corresponds to the resistance R_2 in series with a positive voltage limiter (17 diodes in series). The values of the resistances and the capacitance and the number of diodes were chosen in order to decrease the monetary cost of the circuit and approximate the output signal to a constant 12 V as much as possible.

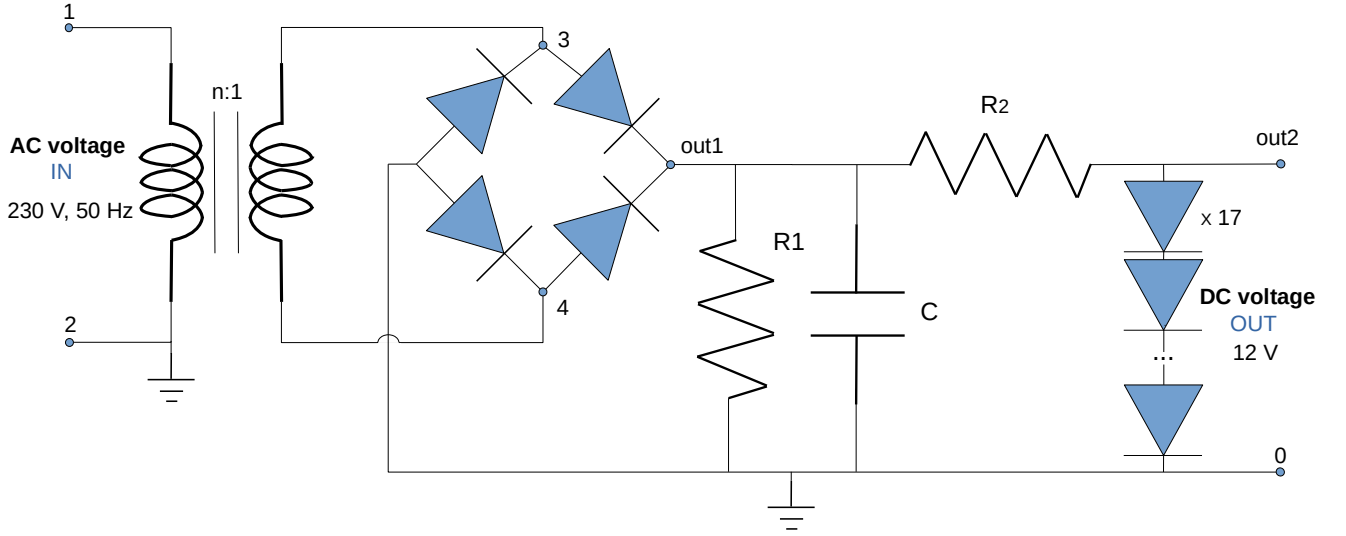


Figure 1: Circuit to be analysed in this laboratory assignment.

2 Theoretical Analysis

The full-wave bridge rectifier circuit, assuming ideal diodes, computes the absolute value of the sinusoidal AC voltage of the second transformer. Therefore, the voltage in resistor R_1 's terminals is given by

$$v_{O_{rectified}}(t) = |v_{t2}| \quad (1)$$

Where v_{t2} is the AC voltage in the secondary, given by $v_{t2} = \frac{v_{t1}}{n}$, assuming that the transformers' wires are coiled according to the conventional way, that is, in order to have a downwards current through the primary and an upwards current through the secondary. The value of n corresponds to the proportion of turns between the primary and the secondary, as shown in Figure 1. Moreover, the capacitor, connected to the rectifier, transforms the rectified wave by attenuating the oscillations. Again, assuming the ideal diode model, after a time t_{OFF} , when the rectified signal's voltage value starts to decrease more abruptly, the current in the capacitor, given by $i_C = C \frac{dv_C}{dt}$, becomes very large, the diode eventually goes off and the capacitor starts discharging. This value can be computed by:

$$t_{OFF} = \frac{1}{\omega} \tan^{-1} \left(\frac{1}{\omega R_1 C} \right) \quad (2)$$

Where $\omega = 2\pi f$ is the angular frequency and $f = 50$ Hz is given. The diode starts conducting again after t_{ON} , in which the rectified signal's voltage equals the discharging capacitor's. While the capacitor is charging and the diodes are conducting, the voltage out of the envelope detector is given by $v_{O_{env}}(t) = v_{O_{rectified}}(t)$ (3). On the other hand, while the capacitor is discharging, it is given by:

$$v_{O_{env}}(t) = V_{t2} \cos(\omega t_{OFF}) e^{-\frac{t-t_{OFF}}{R_1 C}} \quad (4)$$

The value V_{t2} is the amplitude of the voltage v_{t2} in the secondary. It is worth noting that t_{OFF} in the exponential above must be summed half a period, $\frac{T}{2} = \frac{1}{2f}$, every time equation 4 is used again (i.e., every "cycle" of charging-discharging in the capacitor).

Finally, the voltage regulator is used to attenuate the oscillations in the Envelope Detector's output voltage. In this case, the ideal diode model is not used; instead, the diode model with a

voltage source and a resistor is taken into account; a DC analysis and an incremental analysis are made. The diode equation, given by

$$i_D = I_S \left(e^{\frac{v_D}{\eta V_T}} - 1 \right) = I_S \left(e^{\frac{v_{OUT}}{N\eta V_T}} - 1 \right) \quad (5)$$

Returns the value of the current that passes in each diode. The saturation current's value used in this laboratory assignment is $I_S = 1.0 \times 10^{-14}$ A, which is the value used by Ngspice in its default diode model. This model was utilized in Section 3. On the other hand, the thermal voltage is given by $V_T = \frac{kT}{q} \approx 26$ mV at room temperature (considering $k = 1.38064852 \times 10^{-23}$ m² kg s⁻² K, $q = 1.60217662 \times 10^{-19}$ C and $T = 300.15$ K, which is the temperature considered by Ngspice). It was also considered that $\eta = 1$, because that is the value used by Ngspice. Because there are $N = 17$ equal diodes in series, the voltage drop through each of them is given by $v_D = \frac{v_{OUT}}{N}$, where $v_{OUT} = v_{out2}$, thus the relation given in 5.

By applying the Kirchhoff Voltage Law to the rightmost mesh, the following non-linear equation is obtained:

$$v_{OUT} + R_2 I_S \left(e^{\frac{v_{OUT}}{N\eta V_T}} - 1 \right) - v_{O_{env}} = 0 \quad (6)$$

The equation shown above was solved by using Octave and Newton Raphson's iterative method, in which the iterations are given by $x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)}$, with $f(x_n) \equiv f(v_{OUT}) = 0$ given by equation 6. It was decided to solve this system not for every instant, but only for the DC components of the voltages. Therefore, $V_{O_{env}}$ was given by the average of the voltage $v_{O_{env}}$, obtained previously. In this way, the DC component V_{OUT} was obtained. Next, the incremental analysis was taken into account. The incremental resistance of each diode is given by

$$r_d = \frac{\eta V_T}{I_S e^{\frac{v_{OUT}}{N\eta V_T}}} \quad (7)$$

Additionally, the incremental component of the output voltage is given by

$$v_{out}(t) = \frac{Nr_d + R_2}{Nr_d} v_{O_{env}}(t) \quad (8)$$

Where $v_{O_{env}}(t) = v_{O_{env}}(t) - V_{O_{env}}$ was calculated for every instant. The final output voltage, between nodes out2 and 0 of Figure 1, is given by:

$$v_{OUT}(t) = V_{OUT} + v_{out}(t) \quad (9)$$

Having presented the theoretical aspects at hand, it is worth mentioning the values used for the resistances and the capacitance, as well as the value of n , i.e., the quotient between the voltages in the primary and secondary, respectively. These are presented in Table 1, shown below.

Name	Value
R_1	10.75000 k Ω
R_2	5.10979 k Ω
C	5000 μF
V_{T2}	50 V
n	4.6

Table 1: Values used for the resistances R_1 and R_2 , the capacitance C , the amplitude V_{T2} of the secondary's AC voltage and the quotient n .

These were the same values used in the Ngspice simulation, presented in Section 3, in order to properly compare the results obtain by both methods. They were chosen in order to

minimize the ripple and to obtain a final average output voltage as close to 12 V as possible, through the Ngspice simulation (these results will be shown in Tables 3 and 4). At the same time, the values of R_1 , R_2 and C were kept as low as possible, in order to reduce the cost and increase the merit of the work as much as possible (these values are calculated at the end of Section 3. Moreover, we also chose the number of diodes and the voltage in the transformer's secondary in a way that kept this voltage sufficiently smaller than the primary's (so that the transformer decreases the voltage, not increase it). Conjugating all these different factors, the values presented in Table 1 were chosen.

By using equations 3 and 4, the output voltage of the Envelope Detector circuit is plotted below, for 10 periods (with each period given by $T = \frac{1}{2f}$, because the rectified signal has double the frequency of the original sinusoidal voltage). In the same figure, the voltage at the output of the Voltage Regulator circuit is shown. Even though it wasn't requested, the rectified signal is also shown.

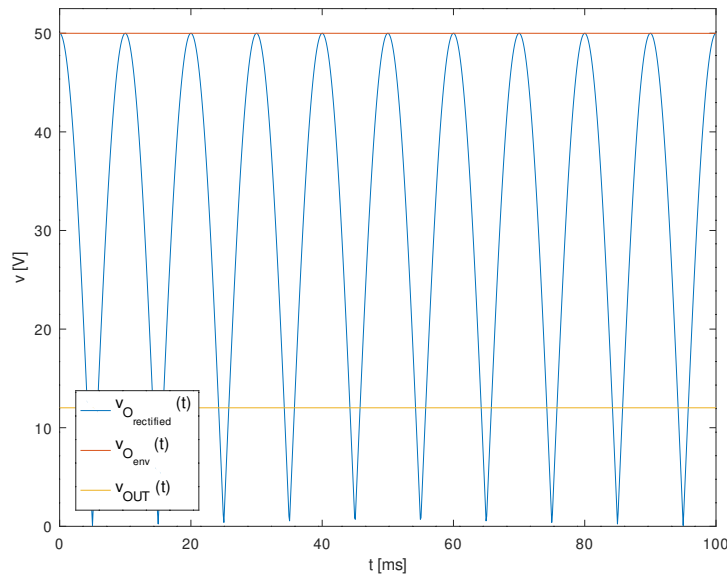
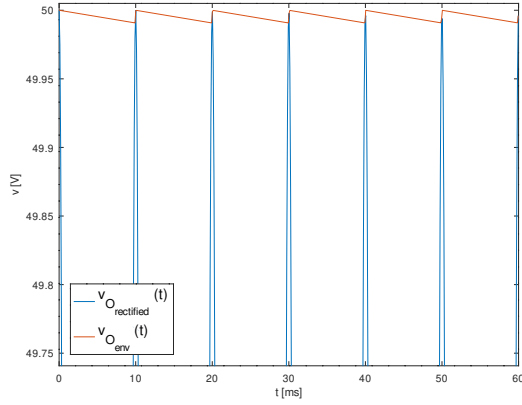
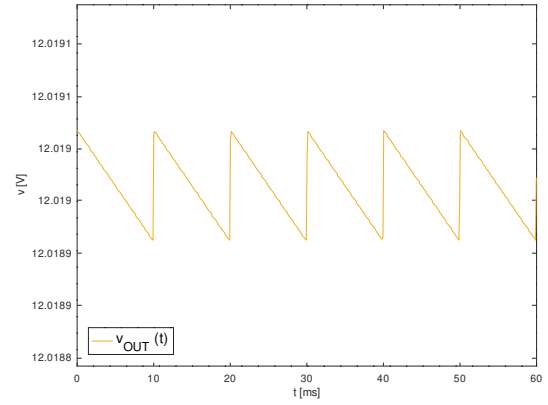


Figure 2: Rectified wave and voltages at the output of the Envelope Detector and Voltage Regulator circuits.

Because the oscillations in the output voltages are very small, it was decided to plot them separately, but for only 6 periods, in order to visualize the signals in more detail. These plots are shown in Figures 3a and 3b. In Figure 4, the value of $v_{OUT} - 12V$ over the 10 initial periods has been plotted.



(a) Detailed plot of the Envelope Detector circuit's output voltage with the rectified signal for 6 periods.



(b) Detailed plot of the Voltage Regulator circuit's output voltage for 6 periods.

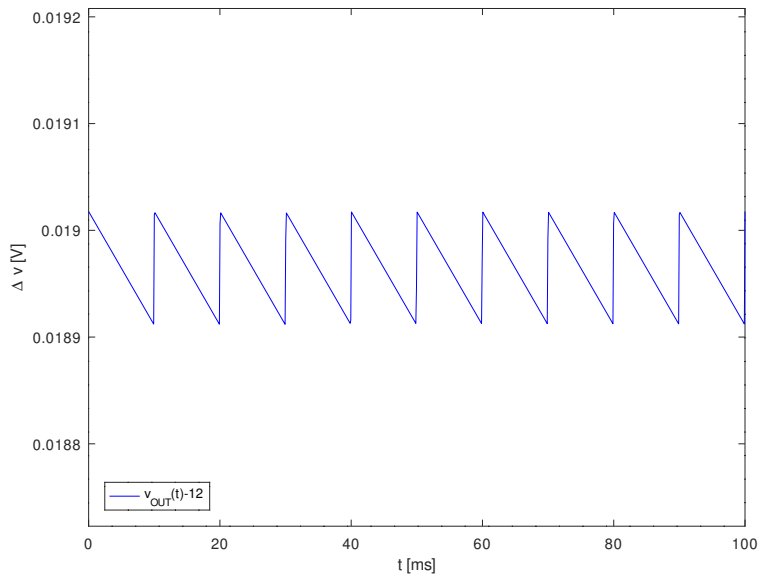


Figure 4: Difference between the final output voltage obtained and the ideal DC signal of 12 V.

Finally, the average final output voltage (output of the Voltage Regulator circuit) was determined, as well as the ripple, given by $ripple(v_{OUT}) = \max(v_{OUT}) - \min(v_{OUT})$. The values obtained are shown in Table 2.

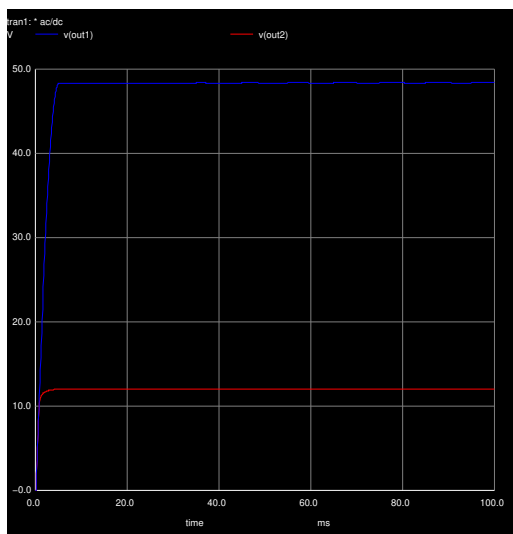
Name	Value [V]
\bar{v}_{OUT}	1.201896e+01
$\max(v_{OUT})$	1.201902e+01
$\min(v_{OUT})$	1.201891e+01
$ripple(v_{OUT})$	1.057158e-04

Table 2: Output voltage's average and ripple.

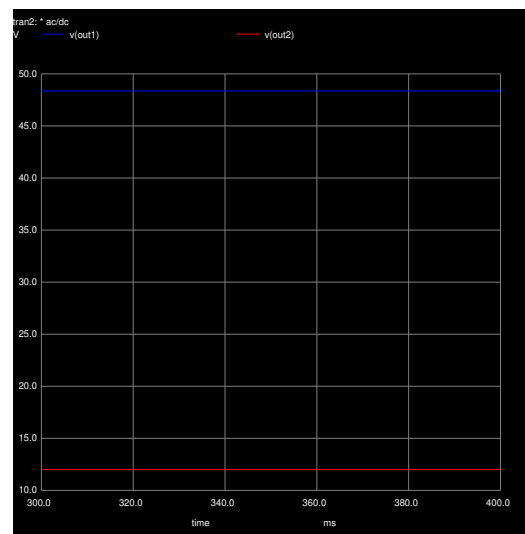
3 Simulation Analysis

In order to simulate this circuit with Ngspice, the ideal transformer model was used. Because the voltage in the primary, v_{T1} , is known, the primary was replaced by a dependant current source and the secondary by a dependant voltage source, in order to have a voltage $v_{T2} = \frac{v_{T1}}{n}$ in the secondary. The values used for R_1 , R_2 , C , V_{T2} and n are those shown in Table 1 of Section 2.

The plot containing the Envelope Detector's output voltage (voltage in node out1) and the Voltage Regulator's output voltage (voltage in node out2) is shown below, for 10 periods.

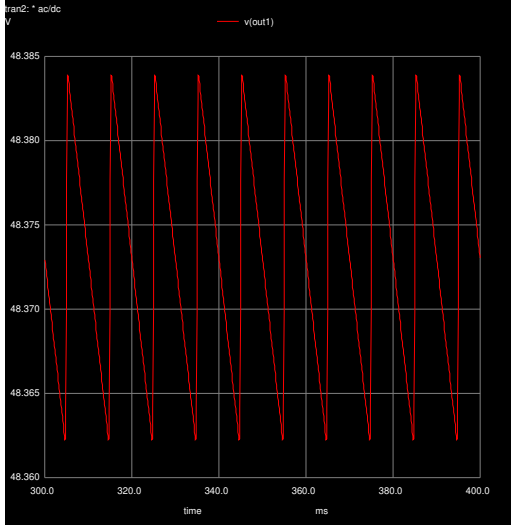


(a) Envelope Detector's output voltage (v(out1)) and Voltage Regulator's output voltage (v(out2)) for the first 10 periods.

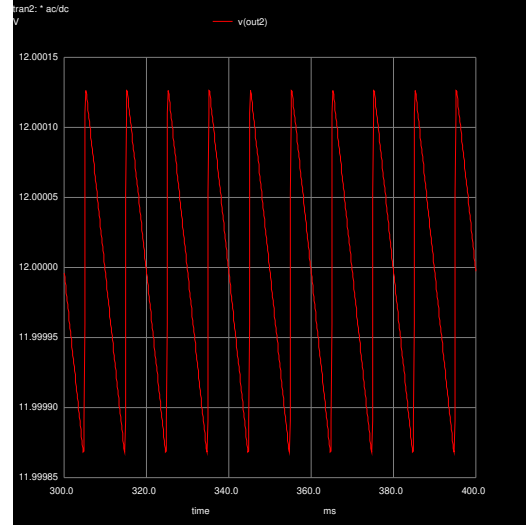


(b) Envelope Detector's output voltage (v(out1)) and Voltage Regulator's output voltage (v(out2)) for 10 later periods.

As we can see in the graphs in Figure 5a, the initial transitory behaviour of the voltages is still clearly present. Therefore, all the values measured in this section were only done in the time interval in which v(out1) and v(out2) are represented in Figure 5b (from 300 ms to 400 ms). Again, as was the case in Section 2, the behaviour of these voltages is not entirely clear from these plots, because the oscillations are very small compared to the y-axis scale. Therefore, separate plots have been made for each voltage, as shown below.



(a) Envelope Detector's output voltage ($v(out1)$) for the time interval [300,400]ms.



(b) Voltage Regulator's output voltage ($v(out2)$) for the time interval [300,400]ms.

We can see that these graphs are very similar to the ones obtained in the Theoretical Analysis. However, it can be seen that, in this case, the maximum values of these output voltages are smaller than those obtained from the theoretical model. In Figure 6a, it can be seen that the maximum is approximately 48.383 V, but, in Section 2, it is approximately 50 V. This may be due to the fact that the diodes were considered ideal in the Envelope Detector. However, Ngspice uses a much more complex diode model. The existence of transitory behaviour may also explain this disparity, as well as the fact that, even though in Figures 3a and 3b the voltages appear to be maximum at every integer multiple of the period, this is not the case in Figures 6a and 6b.

In Figure 7, the fluctuations of the Voltage Regulator's output around 12 V (i.e., $v(out2)-12V$) have been plotted. Comparing these to those in the Theoretical Analysis, there is a clear difference. In the previous section, this final voltage always oscillated above 12V. However, in this case, the signal is clearly much more centered around 12V and the values of $v_{OUT} - 12V$ vary between positive and negative values. It is also clear that the ripple (the difference between the maximum and minimum values of the signal) in the simulation's results is about double of the theoretical analysis' ripple. This may also be explained by the factors talked about above. Additionally, it is also worth mentioning that it was considered, in order to obtain the voltage output from the Envelope Detector, that it was equal to the rectified signal's voltage when this curve's value equaled that of the discharging capacitor's, i.e., no t_{ON} was calculated. Even though this is a very good approximation, slightly more accurate results could be obtained by calculating an actual value for t_{ON} . On the other hand, for the Voltage Regulator analysis, the DC and incremental components of $v_{O_{env}}$ were considered separately and the non-linear equation 6 was solved for the DC component, considering the average of the Envelope Detector's voltage. Different results could have been obtained by solving the nonlinear equation for every time instant. However, that might not have been the case, because the solutions of nonlinear equations are

almost never entirely accurate and might depend on the stop condition, the function's behaviour and the initial values.

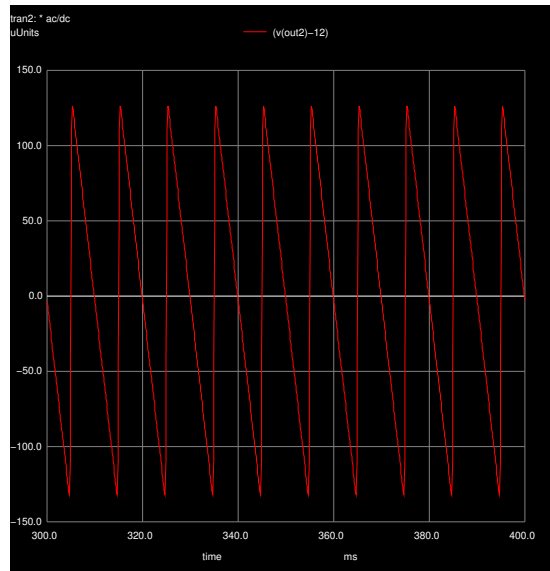


Figure 7: Deviation of output signal from the target DC voltage 12 V over the time interval [300,400]ms.

Now, as in the Theoretical Analysis, the final output's average voltage has been calculated, as well as its ripple, given by $ripple(v_{OUT}) = ripple(v(out2)) = max(v_{OUT}) - min(v_{OUT}) = max(v(out2)) - min(v(out2))$. In Table 3, the results obtained in both cases are shown side by side.

Theoretical		Simulation	
Designation	Value [V]	Designation	Value [V]
\bar{v}_{OUT}	1.201896e+01	\bar{v}_{OUT}	1.200000e+01
$max(v_{OUT})$	1.201902e+01	$max(v_{OUT})$	1.200013e+01
$min(v_{OUT})$	1.201891e+01	$min(v_{OUT})$	1.199987e+01
$ripple(v_{OUT})$	1.057158e-04	$ripple(v_{OUT})$	2.600000e-04

Table 3: Comparison between theoretical and simulation analysis' results.

As we can see, a rather stable 12 V voltage has been obtained in both cases. However, as it already has been discussed, the final voltage in the theoretical model is always bigger than 12V. The simulation analysis' \bar{v}_{OUT} has 7 digits equal to those of the desired output voltage, 12V. However, the value on the left side of Table 3 only has 3 digits in this condition. This is because, as it was discussed in Section 2, different values of R_1 , R_2 , C and n were tried out, in order to get a smaller ripple as possible and an average output voltage as close as possible to 12V in Ngspice, as well as a smaller monetary cost as possible, as already mentioned in the above section. The fact that the theoretical value of \bar{v}_{OUT} is further away from 12V shows

us that the theoretical model considered is clearly different than the one used by Ngspice. It is also subjected to some approximations, as it has been discussed. On the other hand, the ripple obtained in the Theoretical Analysis is about half of the one obtained from the simulation, even though they have the same order of magnitude.

Finally, the total monetary cost and the merit M of the circuit used have been calculated and are shown below in Table 4. These were determined by using the results obtained from Ngspice. The cost is given by $cost = cost\ of\ resistors + cost\ of\ capacitor + cost\ of\ diodes$, in which each $1k\Omega$ in the resistances costs 1 monetary unit (MU), as well as each $1\mu F$ in the capacitance; the cost of each diode is 0.1 MU. On the other hand, the merit M is given by

$$M = \frac{1}{cost \times (ripple(v_{OUT}) + |\bar{v}_{OUT} - 12| + 10^{-6})} \quad (10)$$

Designation	Value [V]
Cost	5.017960e+03
Merit	7.635409e-01

Table 4: Cost and merit obtained for this circuit.

4 Conclusion

In this laboratory assignment, the intended objective has been achieved. The AC/DC converter has been simulated accordingly by using Ngspice and a rather precise output voltage as close to 12V as possible has been obtained. Moreover, a suitable theoretical model was utilized in order to predict the output of the simulated circuit, having obtained a voltage quite close to the one obtained in Ngspice. However, the value obtained in this analysis was slightly further away from the desired 12V DC output voltage. The differences between the results obtained in both analysis may be due to several factors. The theoretical diode model considered is different from the much more complex diode model used by Ngspice. Moreover, the theoretical analysis is subjected to certain approximations and a nonlinear equation must be solved.

A ripple of approximately $10^{-4}V$ was obtained for both cases. A smaller value could have been obtained by using bigger values for R_1 and C , for example. However, that would also lead to a larger monetary cost. By taking into account the different aspects at hand, rather precise results and an acceptable merit M have been obtained.