CONESCAPANHONDURAS2025paper72.pdf

Institute of Electrical and Electronics Engineers (IEEE)

Document Details

Submission ID

trn:oid:::14348:477778909

Submission Date

Jul 31, 2025, 11:20 PM CST

Download Date

Aug 12, 2025, 2:46 PM CST

File Name

CONESCAPANHONDURAS2025paper72.pdf

File Size

521.9 KB

6 Pages

3,605 Words

17,538 Characters

15% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

Match Groups

- **28** Not Cited or Quoted 10%
  Matches with neither in-text citation nor quotation marks
- **0** Missing Quotations 0%
  Matches that are still very similar to source material
- **7** Missing Citation 5%
  Matches that have quotation marks, but no in-text citation
- **0** Cited and Quoted 0%
  Matches with in-text citation present, but no quotation marks

Top Sources

11% Internet sources
14% Publications
1% Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

No suspicious text manipulations found.

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A Flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

Match Groups

- **28** Not Cited or Quoted 10%
  Matches with neither in-text citation nor quotation marks
- **0** Missing Quotations 0%
  Matches that are still very similar to source material
- **7** Missing Citation 5%
  Matches that have quotation marks, but no in-text citation
- **0** Cited and Quoted 0%
  Matches with in-text citation present, but no quotation marks

Top Sources

11% Internet sources
14% Publications
1% Submitted works (Student Papers)

Top Sources

The sources with the highest number of matches within the submission. Overlapping sources will not be displayed.

1	Publication	
	Martin Calasan. "Diode-resistance circuit – Novel exact and approximate solution...	1%
2	Internet	
	d197for5662m48.cloudfront.net	1%
3	Publication	
	Vipin Kumar Singh, Bhim Singh. "Sensorless Control Using Single Current Sensor ...	<1%
4	Internet	
	jaree.scu.ac.ir	<1%
5	Internet	
	journals.pan.pl	<1%
6	Internet	
	www.shastriinstitute.org	<1%
7	Publication	
	Charles A. Gross, Thaddeus A. Roppel. "Fundamentals of Electrical Engineering", C...	<1%
8	Internet	
	library.acadlore.com	<1%
9	Internet	
	www.cityu.edu.hk	<1%
10	Internet	
	link.altmetric.com	<1%

11 Publication

Kaiyi Cheng, Donghao Li, Mingjiang Deng, Xin Li, Guohua Fang. "Application of Im... <1%

12 Internet

exaly.com <1%

13 Internet

www.coursehero.com <1%

14 Internet

repository.tudelft.nl <1%

15 Publication

Vivek Bhatt, Ashish Ranjan. "Active implementation of Chua's diode employing M... <1%

16 Internet

avesis.yildiz.edu.tr <1%

17 Publication

Manuel Cardona, Fernando E. Serrano. "Autoregressive Model for Identification o... <1%

18 Internet

scriptiebank.be <1%

19 Publication

Yie-Tone Chen, D.Y. Chen. "Small-signal modeling of magnetic amplifier post regu... <1%

20 Internet

alumni-portal.sasin.edu <1%

21 Publication

Álvaro Rubinos-Sicre, Francisco D. Freijedo, Thiwanka Wijekoon, Miroljub Bakic. "... <1%

22 Publication

Jingyuan Zhang, Xu Yan, Haorui Luo, Yongxin Guo. "An Efficient Transistor-Model-... <1%

23 Publication

William de Jesus Kremes, Paulo Junior Silva Costa, Carlos Henrique Illa Font, Telle... <1%

24 Publication

John Bird. "Higher Engineering Mathematics", Routledge, 2019 <1%

25 Publication

Ming Rao, Haiming Qiu. "Process Control Engineering - A Textbook for Chemical, ... <1%

---

26 Publication

Xuelian Gao, Hongbo Pang, Chensong Li. "Robust finite-time almost passivity and... <1%

Numerical Simulation of Electrical and Electronic Circuits by GNU Octave

***Abstract*—The electrical simulation of transients in a resistive-inductive-capacitive RLC circuit is presented as an illustration on how to implement the control package of the GNU Octave software. Then half-wave and full-wave rectifier are simulated to observe the performance of these circuits. Ideal and approximated models of the diodes rectifier are implemented to observe the results obtained by the GNU Octave software. Finally the simulation of a common base amplifier is presented by implementing the GNU Octave software.**

***Index Terms*—Transient simulation, nonlinear element simulation, diodes, BJT transistors, rectifiers, amplifiers**

I. Introduction

For this reason it is important to consider the following references. So for example, in [1], [2] circuit simulation is very important taking into consideration a multi-grid system and a full bridge resonant converter then in [3], [4] a transient simulation of a hyper chaotic system and fault current simulation are presented respectively.

Meanwhile circuits with diodes are important to be simulated numerically as appears in references like [5], [6] a synchronous rectifier and a novel diode rectifier are presented respectively. Meanwhile in [7], [8] other results regarding the simulation of diode rectifiers are presented.

Then the numerical simulation of transistors are presented in references like [9], [10] in which an RF-amplifier and a Chua circuit are simulated. Finally in references like [11], [12] there are other results in which circuits with transistors are simulated.

In this paper are evinced the advantages of simulating electrical and electronic circuits with GNU Octave. First the transient simulation of electrical circuits is presented and the the electronic simulation is evinced. It is important to take into consideration that there are important advantages in simulating circuits by a mathematical programming language such as GNU Octave and its control toolbox. The main reason is that simulation can be carry out with electrical, mechanical, hydraulical, neumatical systems. The user can select an appropriate numerical differential equation solver in order to obtain the best results and by mixing, not only with other types of components as explained before, even by mixing analog and digital circuitry. Besides, it is suggested to implement GNU Octave due to it is fast, reliable and a lightweight programming language. It can be implemented in embedded hardware, such as microprocessor or microcontrollers, to simulate electrical and electronic circuits.

By implementing efficiently the GNU Octave control toolbox, the electronics and electrical circuits can be simulated in very few lines of code. Something that is an advantage when there is necessary to implement efifcient code to simulate these circuits. Besides, another advantage, is that using the control toolbox of GNU Octave, this code can be used with other GNU Octave toolbox, such for example, with the signal processing or statistical toolbox.

One of the obvious advantage, is that the GNU Octave software can be implemented to solve circuits with open software, something that is advantageous in under developed countries, in which buying licenses of operative systems and commercial simulation software is expensive. For these reasons is important to simulate circuits in GNU Octave, so in this way the circuits are solved using different GNU Octave differential equation software

II. Problem Formulation

One of the main characteristics of the control toolbox of GNU Octave is that the Laplace transform must be implemented to obtain the transfer functions of electrical and electronic systems. Before starting to simulate circuits in the GNU Octave software, is important to considerate the mathematical model in the form of ordinary differential equation, most general, by implementing the mesh equation of an electrical or electronics circuit. The direct and inverse Laplace transform play an important role to solve this kind of circuits, remember that the control toolbox of GNU Octave implements the transfer functions to test different kinds of response inputs. The theoretical fundamentals for this research study are the following:

Consider the Laplace transform of the function $f(t)$ and its respective inverse Laplace transform:

$$
\begin{aligned}
F(s) &= \int_0^\infty f(t)e^{-st}dt \\
f(t) &= \frac{1}{2\pi j}\oint_{\partial z} F(s)e^{st}ds \qquad (1)
\end{aligned}
$$

In which $s = \omega j$. The current of an approximated version of the diode are given by $I = Isat(exp((V_{in}/(\eta V_t)) - 1)$ in which I_{sat} is the saturation current, η is 1 for silicon and 2 for germanium and V_{in} is the input voltage. Then the transfer function for the common base amplifier is given by [13].

III. Methodology

The methodology is by implementing GNU Octave 7.3.0 with the control package of the respective software. As explained before, the equivalent circuits are obtained in order to simulate the following circuits:

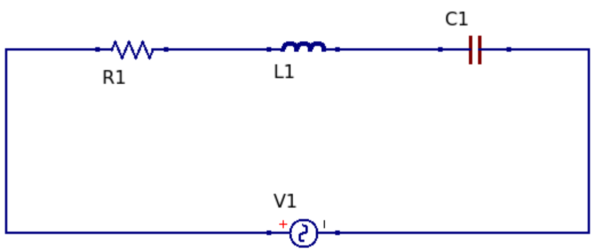


Fig. 1. RLC circuit diagram

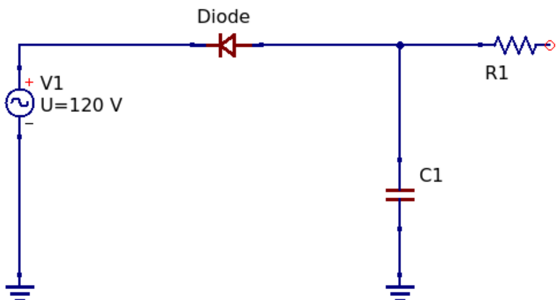


Fig. 2. Half wave rectifier circuit diagram

- An RLC circuit.
- Half wave rectifier.
- Full wave rectifier.
- Common base amplifier.

IV. Simulation of Electrical Circuits by GNU Octave

Firstly, as it obvious, the first circuits that must be simulated in GNU Octave are basically electrical circuits. The transients in these circuits can be simulated in several ways, one by implementing an ordinary differential equation solver or ODE solver as it commonly known. The other way is by representing the electric circuit in transfer function form. So for example, as it is known, circuit Fig. 1 is represented as an integral differential equation in the following form:

$$v(t) = i(t)R_1 + \frac{1}{C_1}\int_0^t i(\tau)d\tau + L_1\frac{di(t)}{dt} \tag{2}$$

in which $v(t)$ is an AC voltage source, $i(t)$ is the circuit current, R_1 is the resistance, C_1 is the capacitance and L_1 is the inductance. so by taking the Laplace transform of the previous equation yields:

$$\mathcal{L}\{v(t)\} = \mathcal{L}\{i(t)R_1\} + \mathcal{L}\left\{\frac{1}{C_1}\int_0^t i(\tau)d\tau\right\} + \mathcal{L}\left\{L_1\frac{di(t)}{dt}\right\} \tag{3}$$

obtaining:

$$V(s) = I(s)R_1 + \frac{1}{C_1 s}I(s) + L_1 sI(s) \tag{4}$$

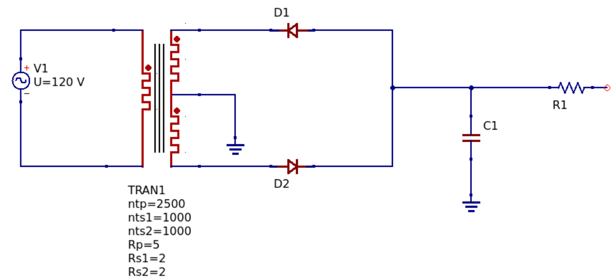


Fig. 3. Full wave rectifier circuit diagram

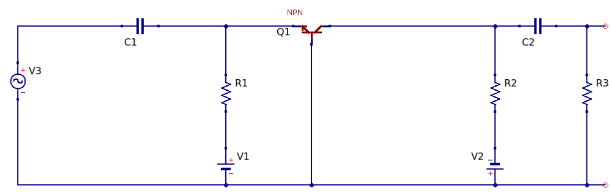


Fig. 4. Common base amplifier BJT circuit diagram

so the transfer function of this circuit is given by:

$$\frac{I(s)}{V(s)} = \frac{1}{R_1 + \frac{1}{C_1 s} + L_1 s} \tag{5}$$

Re-organizing the previous equation yields:

$$\frac{I(s)}{V(s)} = \frac{s}{\frac{1}{C_1} + R_1 s + L_1 s^2} \tag{6}$$

The type of circuit simulated in GNU Octave is basically a resistor-inductor-capacitor circuit. This circuit is depicted in Fig. 1 and it is solved numerically with the help of the GNU Octave control package in the code listed below:

```
pkg load control
L=1e-3; %%% Inductance
R=100000; %%%% Resistance
C=1e-3; %%%% Capacitance
s=tf('s'); %% S variable
Xc=(1/(C*s)); %% Capacitive reactance
Xl=L*s; %%%%%% Inductive reactance
Xt=R+Xc+Xl; %%% Total impedance
G=1/Xt; %%%%%%%%%%%
[I,t]=step(G); %%%
figure
plot(t,I)
xlabel('Time (s)')
ylabel('Current (A)')
print current_transitory.png
h=0.1;
tf=100;
timek=0:h:tf;
V=cos(0.1*timek);
G=1/Xt;
I=lsim(G,V,timek);
```

```
figure
plot(timek,I)
xlabel('Time (s)')
ylabel('Circuit current (A)')
h=0.1;
tf=100;
timek=0:h:tf;
V=cos(0.1*timek).*rand(1,length(timek))*20;
G=1/Xt;
I=lsim(G,V,timek);
```

A. *Simulation of Transients in Electric Circuits*

In Fig. 5 and Fig. 6 the circuit voltage and the filtered capacitor voltage are shown.

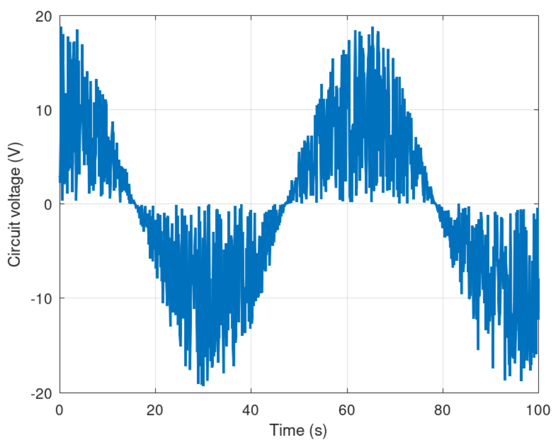


Fig. 5. Circuit voltage in a RLC circuit

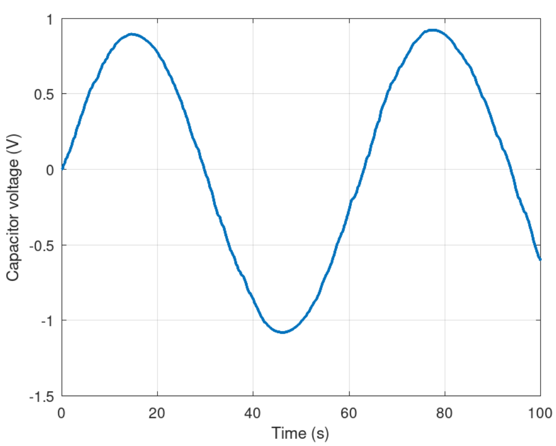


Fig. 6. Capacitor voltage in a RLC circuit

V. Simulation of Electronics Circuits in GNU Octave

As explained in the previous section, not only the simulation of electrical circuits can be ahcieved efficiently in GNU Octave. For these ends, the simulation of electronic circuits is simulated in GNU Octave. The main reason, is that the control toolbox of GNU Octave offers a simple and reliable mathematical programming language solution to solve electronic circuits of any kind. For example the following circuits in the form of transfer function can be simulated with the control toolbox of GNU Octave:

- Circuits with diodes.
- Circuits with bipolar junction transistors BJT.
- Circuits with field effect transistor FET.
- Circuits with complementary metal oxide field effect transistors MOSFET.
- Circuits with operational amplifiers.
- Among others.

The results evinced in this paper can be extended also to simulate not only analog components, even it can be extended to simulate digital circuitry taking into consideration phenomenoms such as delays. This research study can be extended to simulate very large scale integration circuits, offering an alternative to VHDL or VERILOG language. Besides other componenents like tunnel diode, varicap diode SCR's, TRIAC's and DIAC's among other thiristors can be cimulated efficiently with the control toolbox of GNU Octave.

Circuits in which analog electronic components are implemented, are considered in this paper due to the importance of these circuits because they are fundamental to be implemented in power supplies, audio amplifiers among other kinds of circuits. For these reasons, in this section the simulation of half wave, full wave rectifiers and a common base amplifier are shown.

A. *Simulation of a Half Wave Rectifier with Low Pass Circuit*

The half wave rectifier is shown in Fig. 2 meanwhile the output and filtered voltages are shown in Fig. 7.

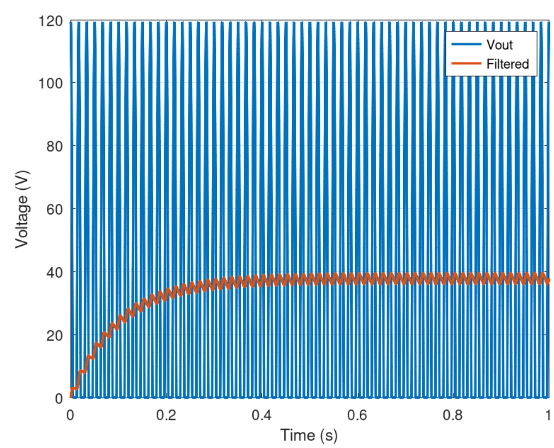


Fig. 7. Output voltage of a half wave rectifier

In Fig. 7 it is noticed how the input AC voltage is rectified in order to produce the DC voltage obtained by the low pass filter used to rectify and elliminate the ripple generated by the diode circuit. With this simulation more complex rectifier circuits can be implemented, such as for example, a bridge rectifier considering an AC transformer.

B. Simulation of a Full Wave Rectifier with Low Pass Filter

Meanwhile a full wave rectifier is shown in Fig. 3. Then in Fig. 8 is shown the filtered voltage and current. The following code is listed to solve this problem:

```
    pkg load control;
clc
clear all
F=60;
omega=2*pi*F;
A=60;
%%%% diodo
Isat=5e-6; %
T=50; %%
eta=2; %%
Vt=T/11.586;
R=1e4;
R2=1e4;
C=1e-6;
s=tf('s');
filtro=(1)/((R*C*s)+1);
tf=0.1;
h=0.0001;
timek=0:h:tf;
Vin1=zeros(1,length(timek));
Vin2=zeros(1,length(timek));
Vout=zeros(1,length(timek));
Iout=zeros(1,length(timek));
for k=1:length(timek)
    Vin1(k)=A*cos(timek(k)*omega);
    Vin2(k)=-A*cos(timek(k)*omega);
    Iout(k)=Isat*(exp((Vin1(k))/(eta*Vt))-1)
    +Isat*(exp((Vin2(k))/(eta*Vt))-1);
endfor
Vout=Iout*R2;
[Ir, t, x]= lsim(filtro,Iout,timek);
Voutr=Ir*R;
```

In Fig. 8 can be noticed the rectified voltages of an AC voltage source in order to be converted to DC voltage. The fullwave rectifier produce the required rectified voltage in which, similar to the half wave rectifier, a low pass filter is implemented in order to suppress the ripple which is found in the circuit. As explained before, these results can be extended to simulate bridge rectifiers or even better more complex rectification circuits such as a switched power supply. Due to the importance to obtain DC voltage converted from AC voltages nowadays, the simulation in GNU Octave is extremely important because the control toolbox of GNU Octave is simple to implement, is reliable and something important is cheaper.

C. Simulation of a Bipolar Junction Transistor Amplifier

The simulation of bipolar junction transistors BJT is important because they are implemented specifically in audio

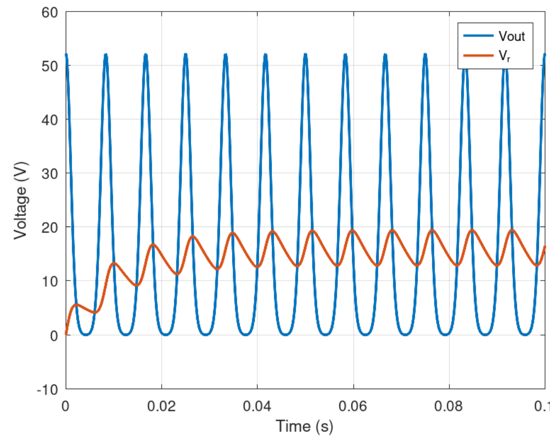


Fig. 8. Voltage of the full wave rectifier

amplifiers, the types of amplifiers that are frequently found are the common emmitter and common base amplifiers. Of course there are more configurations for BJT transistors, and as it is known the amplification characteristics in AC depends on the parameters of the selected BJT transistor. For these reasons, the simulation of a common base amplifier with bipolar transistor in shown in Fig. 4. Meanwhile the results of the amplified voltage is shown in Fig. 9 and the bode plot in Fig. 10. The following code is listed to solve this problem:

```
hfb=1700;
hrb=0.05;
hib=80;
hob=10;
R2=10000;
R3=10000;
pkg load control;
s=tf('s');
F=2000; %% 2000 Hz
omega=2*pi*F;
v1=10*(omega)/(s^2 + omega^2);
v3=(-hfb*v1)
v3=v3//(hib*(hob+(1/R2)+(1/R3))-hrb*hfb);
t0=0;
h=0.00001;
tf=2e-3;
timek=t0:h:tf;
A=2e-3;
u=A*sin(omega*timek);
bode(v3)
grid on
print bode_amplifier.png
v3=(-hfb*u);
v3=v3//(hib*(hob+(1/R2)+(1/R3))-hrb*hfb)
```

In Fig. 9 is shown the input voltage generated by an AC voltage source and the amplified voltage in the output of the amplifier circuit. It is verified that the voltage in red of the output of the common base amplifier is greater that the input

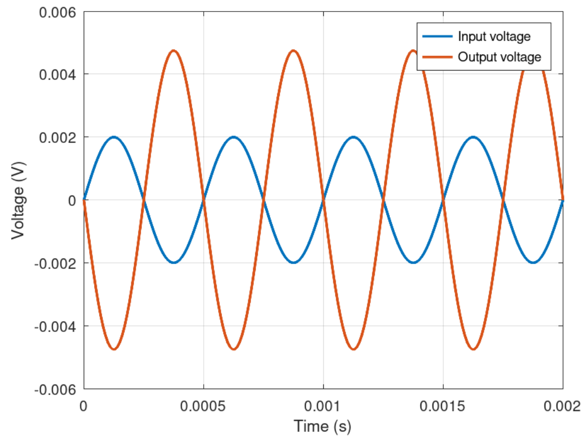
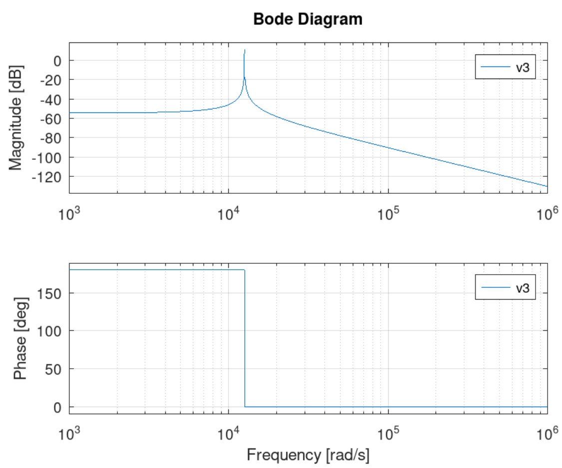


Fig. 9. Voltages of the common base BJT amplifier

voltage that is shown in blue. With this fact it is verified that the output voltage has a phase shift something that is commonly found in BJT amplifiers.



19 Fig. 10. Bode plot of the common base amplifier

19 Finally, in Fig. 10 the bode plot of the common base amplifier is evinced showing that resonance peak is basically found along with the phase shifting as evinced in the phase plot. With these results is confirmed how the amplifier can yield the desired amplification with the designer performance. By tuning and selecting the components of this common base amplifier,this circuit can be implemented as an audio amplifier or even better, for more sophisticated implementation such as a biosignal amplifier. The results obtained in this research study are important to be extended in the simulation of common emmiter amplifiers, push-pull amplifiers among others. The results obtained by simulating implementing the control toolbox of the GNU Octave software can be extended also if the BJT transistors can be implemented in oscillators. The Colpitts oscillator can be implemented by using the GNU octave software, and morder complex circuits involving BJT transistors can be implemented for other applications such as biomedical or industrial applications.

VI. Discussion

In this investigation was explored the full capacities of the control toolbox of the GNU Octave software. Considering the capabilities of this GNU Octave software the simulation of electronic circuits is prove to be effective and sometimes it is not necessary to simulate circuitry in graphical simulation circuit software such as MULTISIM or Pspice. As explained there are several advantages to simulate in the GNU Octave software such as the GNU Octave software is light, fast and reliable, and that nowadays it can be implemented not only in PC, because it can be implemented also in embbeded hardware such as microcontrollers or microprocessors. So it is not necessary to implement sometimes an FPGA's to simulate this circuits. To this means, cheaper hardware such as ARDUINO can be implemented to simulate or implement hardware. This is advantageous specially in under developed countries in which it is difficult to find expensive hardware and software. Many embedded systems hardware is available nowadays to achieve this end.

It is verified, that a half wave rectifier with low pass filter can be simulated efficiently with the control toolbox of GNU Octave. The diode semiconductor model used in this simulation is an approximated model. In this way the half wave circut simulation is more realistic, being useful for the simulation of circuits in which a high fidelity is necessary. One of the advantages of simulating a diode circuit in GNU Octave is that other characteristics such as the diode characteristics changes according to the sorrounding temperature. In this way is advantageous to simulate the electronic circuits in GNU Octave, because there are plenty of conditions sorrounding the characteristic of diodes, considering also the passive elements connected as auxiliary circuits.

Besides the circuit simulation of a full wave rectifier is presented in this paper. The results show that the full wave rectifier is simulated satisfactorily. In this way, the results obtained by implementing the control toolbox of GNU Octave prove that the results approaches a real experimental setup for the full wave rectifier. Besides, these results can be extended to simulate more complex circuitry, in specific, a commutated power supply, something that is wishable from the practical point of view.

Finally, it is verified in this paper, that a common base amlifier is implemented and simulated with the control toolbox of GNU Octave. It is verified that the optimal results are obtained by simulating circuits in GNU OCtave. It is verified that the input voltage is amplified satisfactorily and besides that, the implementation in GNU Octave code allow us to do a complete frequency analysis of the common base amplifier. These results can be extended in the future to simulate circuitry with other components such as FET, MOSFETS and oscillators. It is important that these results imply that other components such as operational amplifiers to design amplifiers or analog filters.

18

VII. Conclusions

In this paper it is verified how electrical and electronic circuits are solved by implementing GNU Octave. It is verified that with the use of the control package it is easy to solve electrical and electronic circuits. Besides it is proved that one of the advantages of the control toolbox of GNU Octave, is that it allows to simulate some conditions such as temperature effects in the diodes, transistors, FET, SCR's, etc and other conditions that are difficult to simulate in graphical circuits simulators such as Pspice and Multisim. Besides, another advantage of simulating circuitry usign the GNU Octave software is that the circuits can be combined easilly with other types of components such as mechanical, hydraulic, neumatic etc.

Acknowledgment

17 Many thanks to the institute of energy research IIE at the Universidad Nacional Autónoma de Honduras and Universidad Don Bosco El Salvador.

References

2 [1] M. Liaqat, T. Alsuwian, A. A. Amin, M. Adnan, and A. Zulfiqar, "Transient stability enhancement in renewable energy integrated multi-microgrids: A comprehensive and critical analysis," *Measurement and Control (United Kingdom)*, vol. 57, no. 2, p. 187 – 207, 2024.

8 [2] Z. Li, Y. Wei, S. Zhang, and J. Liu, "A simple optimal trajectory and thermal balance control strategy for full-bridge resonant converter with wide voltage range application," *IEEE Transactions on Power Electronics*, vol. 39, no. 2, p. 1990 – 1996, 2024.

5 [3] X. Zhang, J. Xu, and A. J. Moshayedi, "Design and fpga implementation of a hyperchaotic conservative circuit with initial offset-boosting and transient transition behavior based on memcapacitor," *Chaos, Solitons and Fractals*, vol. 179, 2024.

4 [4] W. Yang, C. Tu, F. Xiao, and Q. Guo, "A combined regulation method of transient power angle stability control and fault current suppression for vsg," *International Journal of Electrical Power and Energy Systems*, vol. 156, 2024.

12 [5] T. Huang, Y. Chen, B. Zhang, and D. Qiu, "Shared external driver for vhf converter with a synchronous rectifier based on matching network parameters optimization," *IEEE Transactions on Industrial Electronics*, vol. 71, no. 2, p. 1481 – 1491, 2024.

1 [6] T. Abbas, N. Khan, and M. S. Afzal, "A novel diode rectifier with a series connected chopper for reduced input current thd," *Iranian Journal of Science and Technology - Transactions of Electrical Engineering*, 2024.

3 [7] S. Lu, H. Ma, J. Yi, Z. Chen, X. Li, J. Xu, and Q. Feng, "A novel single-stage ac/dc converter integrated interleaved dcm boost pfc and llc dc-dc with reduced bus voltage," *IEEE Transactions on Industrial Electronics*, vol. 71, no. 4, p. 3525 – 3536, 2024.

21 [8] A. Rubinos-Sicre, F. D. Freijedo, T. Wijekoon, and M. Bakic, "Accurate and efficient simulation models for power electronics transformers based on series-connected active unity power factor rectifier modules," *IEEE Transactions on Industrial Electronics*, vol. 71, no. 5, p. 5116 – 5125, 2024.

11 [9] K. Kumar, S. Kumar, and B. K. Kanaujia, "Analytical modelling of ultra-small group delay variation of ultra-broadband rf power amplifier
16 using nsga-ii algorithm," *International Journal of Numerical Modelling: Electronic Networks, Devices and Fields*, vol. 37, no. 2, 2024.

15 [10] V. Bhatt, A. Ranjan, and M. Joshi, "Ccccta-based chua's circuit for chaotic oscillation," *Circuits, Systems, and Signal Processing*, 2024.

6 [11] P. Kumar, R. K. Ranjan, and S.-M. Kang, "A memristor emulation in 180-nm cmos process for spiking signal generation and chaos application," *IEEE Transactions on Circuits and Systems I: Regular Papers*, p. 1–14, 2024.

22 [12] J. Zhang, X. Yan, H. Luo, and Y. Guo, "An efficient transistor-model-assisted layout synthesis approach using improved implicit space mapping for high-performance mmic pas," *IEEE Transactions on Circuits and Systems II: Express Briefs*, p. 1–1, 2024.

[13] W. L. Brogan, *Modern control theory (3rd ed.)*. USA: Prentice-Hall, Inc., 1991.