Journal of Engineering Research Innovation and Scientific Development 3 (2), (2025), 12-16 https://doi.org/10.61448/jerisd32253

Journal of Engineering Research Innovation and Scientific Development
journal homepage: www.jerisddelsu.com
Print ISSN 2971-6594 Online ISSN 2971-6608

Computer-Aided Transistor Circuit Analysis Package for Single-Stage Power Amplifier Using Impedance and Admittance Parameters

Henry E. Amhenrior[1*] and Davidson U. Akra[2]

[1,2]*Department of Electrical and Electronic Engineering, Faculty of Engineering, Edo State University Uzairue, Edo State, Nigeria*

Corresponding author: henrino2003@yahoo.com (Amhenrior H.E)

Article history: Received: 11-03-25, Revised: 01-05-25, Accepted: 08-05-25, Published: 10-05-25

Abstract

This paper presents a computer-aided transistor circuit analysis package for a single-stage power amplifier using impedance and admittance parameters. This was borne out of the need to speed up the circuit design process by decreasing the time spent during circuit analysis. The methodology involves deriving the impedance and admittance parameter quantities namely voltage gain, current gain, input resistance, output resistance, and power gain from first principles. The softwareconsists of the development of a menu-based Graphical User Interface (GUI) which was developed using Swing, a Java GUI widget toolkit; programming in Java to carry out the mathematical analysis, and circuit simulation which was achieved using JSpice integrated with MULTISIM® for real-time circuit simulation and analysis.The programming was designed to flow as a main program based on the menuwith subroutines which comprises of z-parameter mathematical analysis subroutine, y-parameter mathematical analysis subroutine, impedance equivalent circuit simulation subroutine, admittance equivalent circuit simulation subroutine, and power amplifier circuit simulation subroutine. The results obtained from the studies show that the developed software provides accurate mathematical circuit analysis and performs simulations correctly, quickly, and efficiently. It is therefore recommended that stakeholders in the field such as engineers, educational institutions, and the government should adopt this software for their use as it is reliable and fast in analyzing single-stage transistor power amplifiers.

Keywords: Analysis; Transistor; z-Parameter; y-Parameter; Single-stage Amplifier; Performance quantities

1. Introduction

Single-stage power amplifiers' design and analytical assessment are paramount across various electronic applications, encompassing audio amplification and communication systems. Typically, the performance metrics that define these amplifiers include gain, efficiency, bandwidth, and input/output impedance. Precisely evaluating these parameters is indispensable for optimizing the amplifier's performance, guaranteeing reliability, and adhering to design specifications. Nonetheless, the manual calculation and examination of transistor-based amplifier circuits can prove to be labour-intensive and susceptible to inaccuracies, particularly as the complexity of circuits escalates (Wong et al., 2015).

In this framework, computer-aided tools provide considerable benefits by automating computations, visualizing outcomes, and facilitating swift prototyping. Impedance parameters, which are integral in ascertaining the behavior of transistor circuits, furnish critical insights into the amplifier's input-output characteristics, stability, and power handling capacities. Developing a specialized analysis package concentrating on these parameters can streamline the design process and augment comprehension, especially for single-stage configurations that constitute the foundational elements of more intricate systems.

There has been much interest by researchers in developing tools for students' learning in other to bridge the gap between theoretical perception of electronic courses and practical. In pursuit of this, Pandiev *et al.* (2020) introduced modular laboratory kits for single-transistor amplifiers, focusing on educational applications. The kits allow students to configure and experiment with common-emitter, common-source, common-collector, and common-drain amplifiers using discrete components like BJTs and MOSFETs. Although the paper contributes to educational tools for amplifier analysis, it does not address computer-aided analysis packages for single-stage power amplifiers utilizing impedance and admittance parameters in designing or analyzing amplifiers. Qiuyue *et al.*, (2024) presented a paper that advocated for the use of Multisim software to solve the lack of interest by students in the study of Electricity and Electronic Technology due to lack of balanced mathematical knowledge. Their method blends theory with practical to raise learners' interest in the course. Also, Thain (2022) presented a SPICE model for bipolar junction transistors that enhances amplifier design verification by computing necessary variables from user-defined parameters, thus improving students' confidence in their analysis techniques.

Cuntan et al. (2024) employed a LabVIEW-based simulation tool for designing and analyzing amplifier characteristics in various configurations. The application enables users to evaluate transistor parameters using Bode plots and validated the results using Multisim and experimental setups. This is a valuable educational tool used for practical design scenarios. While the LabVIEW simulation tool offers significant flexibility and precision in evaluating amplifier stages, the study does not focus on single-stage power amplifiers or their design using impedance or admittance parameters. Amhenrior (2018) applied the two-port network model to derive h-parameter equations, including input resistance, forward current gain, reverse voltage gain, and output admittance of a transistor. These parameters were obtained using mathematical tools like partial derivatives, which allowed precise modeling of small-signal amplifier behavior. However, there was no software developed for transistor analysis. Amhenrior & Amhenrior (2018) in their paper presented a mathematical analysis of a Small-Signal Single-Stage Transistor Amplifier using Hybrid Parameters, alongside the development of software for efficient analysis. It employs matrix and determinant methods to derive performance quantities such as input resistance, current and voltage gains, output impedance, and power gain. The software, developed in Visual Basic 6, features a user-friendly GUI and automates calculations. It significantly reduces errors and time in circuit analysis, making it a valuable tool for engineers and educational institutions. It however did not employ impedance and admittance parameters for analysis.

From the foregoing, none have developed a software for analysing a single-stage transistor amplifier utilizing the twin parameters of impedance and admittance to obtain the performance quantity of a transistor, hence the need for this study.

2. Materials and Method

The performance quantities were derived from first principle in this section using, among other mathematical tools, partial differential equations. The results were used to derive the fundamental equations of the z-parameters and y-parameters connecting the fundamental inputs and outputs of currents and voltages of the power amplifier as a two-port network.

2.1 Analysis of the Single-stage Amplifier Using the General Z-Parameter Equivalent Circuit

The general z-parameter equivalent circuit equations are given in Equations 1 and 2.

$$V_1 = z_{11} I_1 + z_{12} I_2 \quad (1)$$

$$V_2 = z_{21} I_2 + z_{22} I_2 \quad (2)$$

For this mathematical examination, the power amplifier's z-parameter equivalent circuit with an applied voltage source, series resistance at the input and output circuits, and a power amplifier two-port system was utilized. This is depicted in the general z-parameter equivalent circuit diagram in Figure 1.

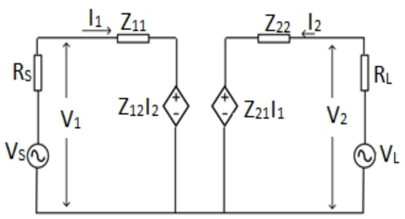

Figure 1: Transistor General Z -Parameter Equivalent Circuit

Input Impedance

By converting Equations 1 and 2 to matrix form we have;

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

From the equivalent circuit of Figure 1:

$$V_2 = V_L - I_2 R_L \quad (3)$$

Substituting Equation 3 into Equation 2

$$V_L - I_2 R_L = z_{21} I_1 + z_{22} I_2$$

But $R_L = z_L$

$$V_L = z_{21} I_1 + (z_{22} + z_L) I_2 \quad (4)$$

From Equations 1 and 4

$$V_1 = z_{11} I_2 + z_{12} I_2$$

$$V_L = z_{21} I_1 + (z_{22} + z_L) I_2$$

We have the matrix; $\begin{bmatrix} V_1 \\ V_L \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} + z_L \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$

$$I_1 = \frac{1}{D} \begin{vmatrix} V_1 & z_{12} \\ V_L & z_{22} + z_L \end{vmatrix}$$

Where, $D = \begin{vmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} + z_L \end{vmatrix} = z_{11} (z_{22} + z_L) - z_{12} z_{21}$

When $V_L = 0$

$$I_1 = \frac{1}{z_{11} (z_{22} + z_L) - z_{12} z_{21}} \begin{vmatrix} V_1 & z_{12} \\ 0 & z_{22} + R_L \end{vmatrix}$$

$$I_1 = \frac{v_1 (z_{22} + z_L)}{z_{11} (z_{22} + z_L) - z_{12} z_{21}} \quad (5)$$

Therefore,

Input resistance $Z_i = \frac{z_{11} (z_{22} + z_L) - z_{12} z_{21}}{(z_{22} + z_L)}$ (6)

Voltage gain

From Equation (3) $V_2 = V_L - I_2 z_L$

When $V_L = 0$; $V_2 = - I_2 z_L$ (7)

From Equations (1) and (7)

$$V_1 = z_{11} I_2 + z_{12} I_2$$

$$V_2 = - I_2 z_L$$

Converting to matrix form;

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ 0 & - z_L \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

$$I_1 = \frac{1}{D} \begin{vmatrix} V_1 & z_{12} \\ V_2 & - z_L \end{vmatrix}$$

Where, $D = \begin{vmatrix} z_{11} & z_{12} \\ 0 & - z_L \end{vmatrix} = - z_L z_{11}$

$$I_1 = \frac{1}{- z_L z_{11}} \begin{vmatrix} V_1 & z_{12} \\ V_2 & - z_L \end{vmatrix} = \frac{- z_L v_1 - z_{12} v_2}{- z_L z_{11}}$$

From Equation 5

$$I_1 = \frac{v_1 (z_{22} + z_L)}{z_{11} (z_{22} + z_L) - z_{12} z_{21}}$$

Then, $\frac{- z_L v_1 - z_{12} v_2}{- z_L z_{11}} = \frac{v_1 (z_{22} + z_L)}{z_{11} (z_{22} + z_L) - z_{12} z_{21}}$

$$A_v = \frac{v_1}{v_2} = \frac{z_L (- 2R_L z_{11} + z_L - z_{12} z_{21})}{z_{12} (-z_{11} z_{22} - z_L z_{11} + z_{12} z_{21})}$$

Current gain

Putting Equations 2 and 3 together when $V_L = 0$,

$$-I_2 z_L = z_{21} I_1 + z_{22} I_1$$

$$A_i = \frac{I_2}{I_1} = \frac{z_{21}}{- (z_L + z_{22})} \quad (8)$$

Output Impedance

From Equation 2, $V_2 = z_{21} I_2 + z_{22} I_2$

Dividing through by I_2, $\frac{V_2}{I_2} = \frac{z_{21} I_2}{I_2} + z_{22}$

and, $\frac{I_2}{I_1} = \frac{z_{21}}{- (z_L + z_{22})}$

$$\frac{V_2}{I_2} = \frac{z_{21} (- (z_L + z_{22}))}{z_{21}} + z_{22} = - z_L$$

Hence; $z_o = - z_L$

Power gain

$$A_p = A_v A_i = \frac{z_L (- 2 z_L z_{11} + z_{12} z_{21})}{z_{12} (-z_{11} z_{22} - z_L z_{11} + z_{12} z_{21})} \times \frac{z_{21}}{- (z_L + z_{22})} \quad (9)$$

The performance quantities of the power amplifier calculated are:

Input impedance $Z_i = \frac{v_1}{i_1} = \frac{z_{11} (z_{22} + R_L) - z_{12} z_{21}}{(z_{22} + R_L)}$

Voltage gain $A_v = \frac{v_2}{v_1} = \frac{R_L (- 2R_L z_{11} + z_{12} z_{21})}{z_{12} (-z_{11} z_{22} - R_L z_{11} + z_{12} z_{21})}$

Current gain $A_i = \frac{i_2}{i_1} = \frac{z_{21}}{-(R_L + z_{22})}$

Output impedance $Z_o = \frac{v_2}{i_1} = R_L$

Power gain $A_p = A_v A_i = \frac{R_L (- 2z_L R_{11} + z_{12} z_{21})}{z_{12} (-z_{11} z_{22} - R_L z_{11} + z_{12} z_{21})} \times \frac{z_{21}}{- (R_L + z_{22})}$

2.2 Analysis of the Single-stage Amplifier Using the General Y-Parameter Equivalent Circuit

The general z-parameter equivalent circuit equations are given in Equations 10 and 11.

$$I_1 = y_{11} V_1 + y_{12} V_2 \quad (10)$$

$$I_1 = y_{21} V_1 + y_{22} V_2 \quad (11)$$

For this mathematical examination, the power amplifier's y-parameter equivalent circuit with an applied voltage source in parallel to the admittance. This is shown in the general y-parameter equivalent circuit diagram in Figure 2.

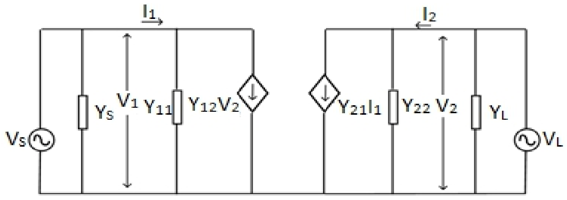

Figure 2: Equivalent circuit for admittance parameter

From the output section of the circuit presented in Figure,

$$v_L = v_2 + \frac{I_2}{y_L} \quad (12)$$

And for the input side, $v_s = v_1 + \frac{I_1}{y_s}$ (13)

Voltage gain

From Equation 12, when $v_L = 0$

$$I_2 = -v_2 y_L \quad (14)$$

Substituting into Equation 11

$$-(y_L + y_{22}) V_2 = y_{21} V_1 \quad (15)$$

$$A_v = \frac{v_2}{v_1} = \frac{-y_{21}}{(y_L + y_{22})} \quad (16)$$

Input admittance

From Equation 10; $I_1 = y_{11} V_1 + y_{12} V_2$

Dividing through by $\frac{I_1}{v_1} = y_{11} + y_{12} \frac{v_2}{v_2}$

From Equation 16,

$$y_{in} = \frac{I_1}{v_1} = y_{11} - \frac{y_{12} y_{21}}{(y_L + y_{22})} \quad (17)$$

Output Admittance

Because of the symmetry of the equivalent circuit in Figure 2 we can write the equation for output impedance is:

$$y_{out} = y_{22} - \frac{y_{12} y_{21}}{(y_s + y_{11})} \quad (18)$$

Power Gain

$$A_p = A_v A_i \quad (19)$$

$$\frac{I_2}{I_1} = \frac{v_2 y_{out}}{v_1 y_{in}}$$

$$A_i = \frac{- y_{21} y_{out}}{(y_L + y_{22}) y_{in}} \quad (20)$$

Substituting into Equation 19

Therefore, $A_p = A_v^2 \frac{y_{out}}{y_{in}} = \left(\frac{-y_{21}}{(y_L + y_{22})}\right)^2 \times \frac{y_{out}}{y_{in}}$ (21)

Summary of the derived Admittance performance quantities

input admittance $y_{in} = y_{11} - \frac{y_{12} y_{21}}{(y_L + y_{22})}$

output admittance $y_{out} = y_{22} - \frac{y_{12} y_{21}}{(y_s + y_{11})}$

voltage gain $A_v = \frac{-y_{21}}{(y_L + y_{22})}$

current gain $A_i = \frac{-y_{21} y_{out}}{(y_L + y_{22}) y_{in}}$

power gain $A_p = \left(\frac{-y_{21}}{(y_L + y_{22})}\right)^2 \times \frac{y_{out}}{y_{in}}$

2.3 Software Package Development

The software aspect of this work involved the development of a Java-based Graphical User Interface (GUI) and programming in language to perform the various circuit analysis and their performance quantities as derived in sections 2.1 and 2.2. The JSPICE, a Java-based simulation tool for conducting was used for the transient analysis of the circuit and real-time circuit simulation was achieved through the utilization of MULTISIM®. The software package was designed and developed into a number of subroutines that handle a single operation each namelyimpedance (z-parameter) mathematical analysis, admittance (y-parameter) mathematical analysis, power amplifier circuit diagram simulation, impedance equivalent circuit diagram simulation, and admittance equivalent circuit diagram simulation

2.3.1 Principle of Operation

Upon launching the application, the graphical user interface (GUI) displays the welcome screen with a button leading to the main menu when clicked on. Figures 3 and 4 show the launch screen and the menu screen respectively.

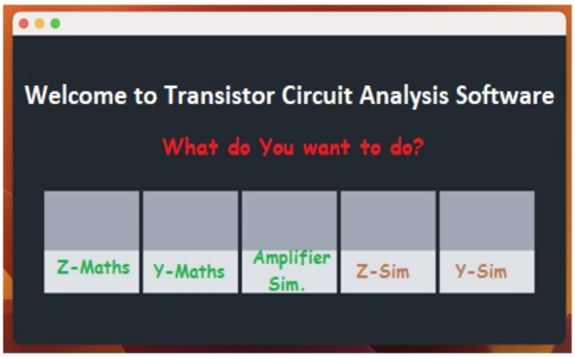

Figure 3: The launch screen of the application

Figure 4. The menu screen of the application

The mathematical operations section of the software package is organized into two divisions, depending on the parameter employed for analysis namely, the impedance z-parameters and analysis and the admittance y-parameters analysis. Each selected operation runs the associated function and prompts the user for the necessary data to complete the operation in a textbox in the GUI. When the required data are entered and the "compute" button is clicked, the software will calculate the corresponding operational parameters based on the provided inputs in the analysis chosen. The results will be displayed on the right-hand side of the form, allowing users to conveniently view and analyze the computed operational values derived from the inputted parameter values. Before moving on to either the simulation of the amplifier circuit diagram, the impedance equivalent circuit, or the admittance equivalent circuit, the results of the mathematical analysis of the z-parameter or the y-parameter namely the performance quantities which include the input resistance, the output impedance, the current gain, the voltage, and the power gain are displayed and stored. These results are then used to simulate the power amplifier circuit diagram in the analysis chosen and the corresponding simulation subroutine will be executed. To exit or close the program the exit button is clicked and the program will be terminated.

The operational flow of the application is shown in the flowchart in Figure 5. Figure 6 and Figure 7 show GUI for the impedance analysis and the operational flowchart for conducting z-parameter and y-parameter mathematical analysis respectively using the software package.

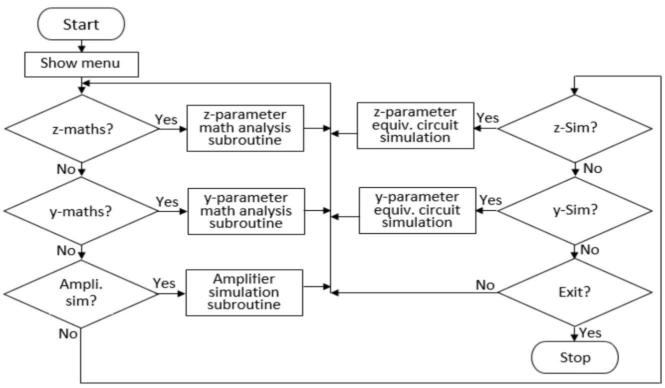

Figure 5: Flowchart showing the general flow of the application

In Figure 5, the software keeps checking to see if any of the menus is selected starting from the first till the last and repeats the process until one is selected and the corresponding subroutine is executed and the results are displayed and the program returns to the main program flow checking for the next subroutine to be selected or the exit button to end the program.

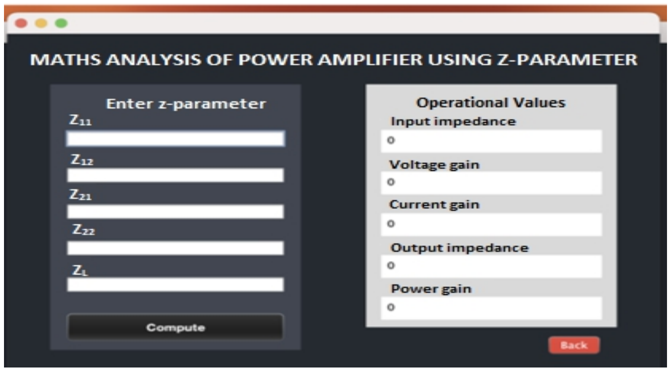

Figure 6: The GUI flow for impedance z-parameter math analysis

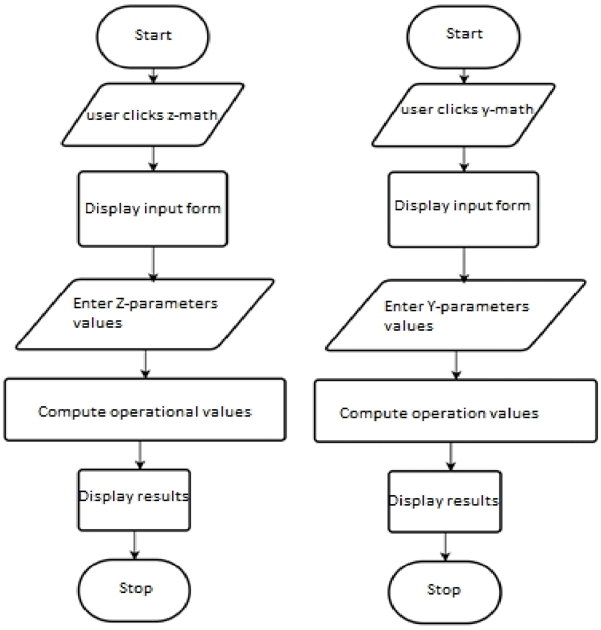

Figure 7: Flowchart showing the flow process for both z-parameter and y-parameter mathematical analysis.

Onceeither of mathematical analysis in z-parameter or y-parameter is selected, the respective subroutine will be entered into and a GUI with textboxes in Figure 6 will be displayed. Once the data are supplied, the subroutine as in the flowchart of Figure 7 will be executed and the respective performance quantities will be computed and displayed.

2.3.2 Simulation with the Software Package

This software can simulate three kinds of circuits, namely power amplifier circuit diagram simulation, impedance equivalent circuit diagram simulation and admittance equivalent circuit diagram simulation in class A, class B, and class AB power amplifiers. Figure 8 shows the flowchart of the simulation process of the software package.

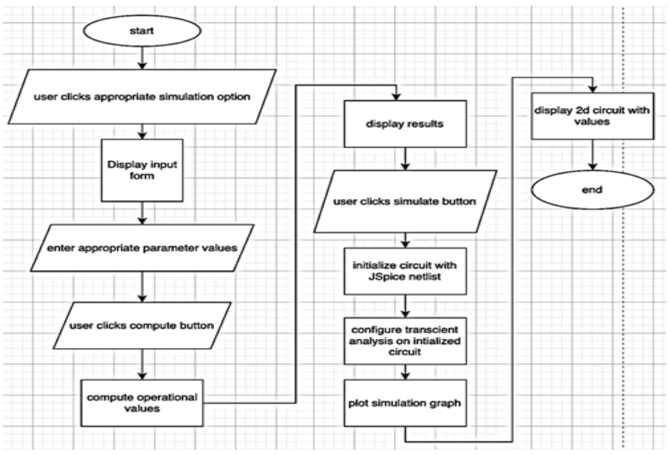

Figure 8: A flowchart showing the simulation process used

In Figure 8, when the simulation option is selected and the data required are supplied, the software will compute the operational values and display the results, and save them. When the simulate button is clicked, Jspice will be initialized and a transient plot of the circuit will be executed and displayed in 2D.

2.4 Test

For the purpose of evaluating the performance of the software, sample problems for impedance and admittance analysis were used as follows

1. Impedance Parameter Analysis

The values for this test were obtained from the data sheet of the LM386N audio power amplifier microchip.

$Z_{11} = 100, Z_{12} = 1, Z_{21} = 100000$ and $Z_{22} = 1000$, with an estimated load resistance of 20 ohms.

The values were inputted into the impedance math analysis section of the software package and the result of the computations are shown in Figure 9 and the simulated equivalent circuit is shown in Figure 10.

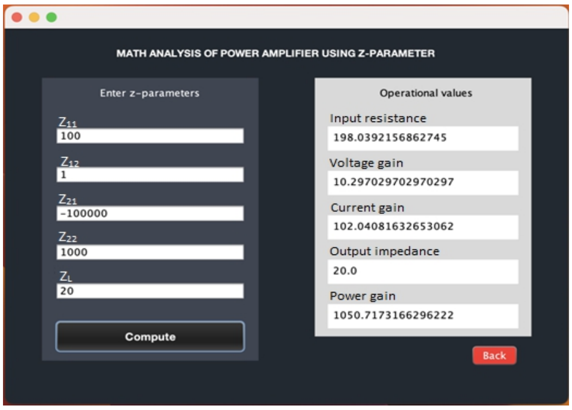

Figure 9: Impedance parameter math analysis with input and result

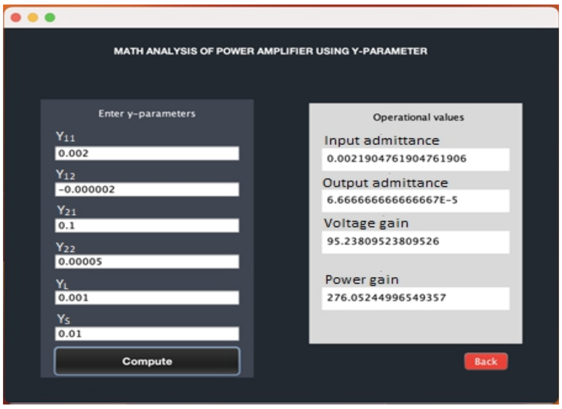

Figure 10: Impedance equivalent circuit in Multisim

2. Admittance Parameter Analysis

The values for this test were obtained from the 8002D SOP8 power amplifier microchip data sheet.

$Y_{11} = 0.002$, $Y_{12} = 0.000002$, $Y_{21} = 0.1$ and $Z_{22} = 0.00005$, with an estimated load resistance of 1000 ohms and source resistance of 100 ohms.

The values were inputted into the admittance mathematical analysis section of the software package and the results of the computation are as shown in Figure 11.The simulated equivalent circuit is shown in Figure 12.

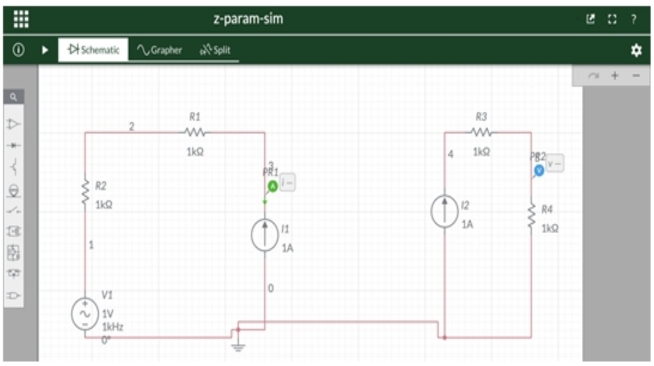

Figure 11: Admittance parameter analysis with input and results

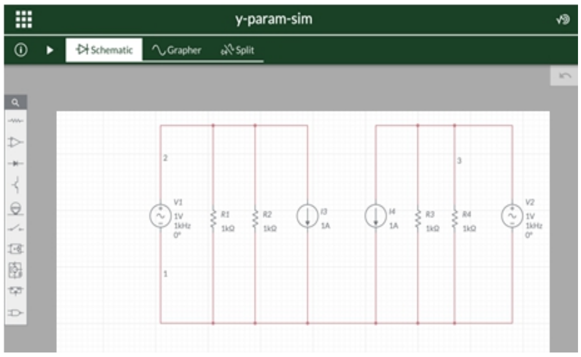

Figure 12: Admittance equivalent circuit in Multisim

3. Transient Analysis Test

To evaluate the application's performance in transient analysis for impedance and admittance parameters, the values for the sampled tests above were used as the test inputs and the transient analysis output of the application after simulating for 2000 milliseconds in each case are shown in Figures 13 and 14.

Figure 13: Transient analysis result for impedance parameter

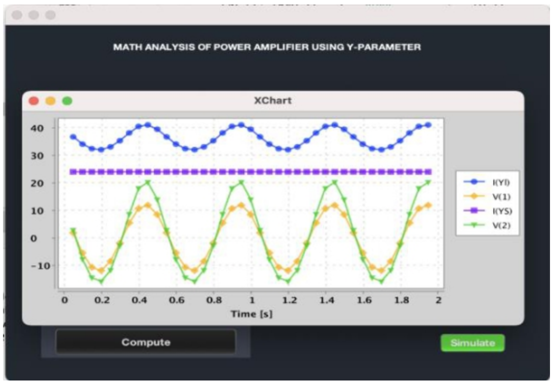

Figure 14: Transient analysis result for Admittance parameter

3. Results and Discussion

The results of the sampled test of the mathematical analysis using impedance and admittance parameters are presented in Figures 9 and 11 respectively, while Figures 10 and 12 show the simulated circuits of the sampled circuits for impedance and admittance using the software.

Figure 13 and 14 show the transient analysis plots. This study introduces a computer-aided analysis package tailored for single-stage power amplifier circuits, utilizing both impedance (Z) and admittance (Y) parameters for effective circuit evaluation. The package combines mathematical modeling with circuit simulation to provide a comprehensive analysis environment. The results obtained demonstrate the accuracy, efficiency, and usefulness of the developed tool.

To validate the effectiveness of the package, mathematical analyses were conducted using impedance and admittance models. The outcomes of these calculations are illustrated in Figure 9 (impedance analysis) and Figure 11 (admittance analysis). These figures show the analytical input data and the corresponding results, confirming that the mathematical framework implemented in the package yields correct and consistent outcomes. Notably, these results align closely with manual analysis, verifying the accuracy of the automated system.

Complementing the analytical approach, circuit simulations were performed using Multisim. The equivalent circuits based on impedance and admittance parameters are shown in Figure 10 and Figure 12, respectively. These simulated diagrams accurately reflect the theoretical designs, allowing real-time observation and interaction. This visual representation strengthens confidence in the design's integrity and helps users better understand the behavior of amplifier circuits modeled through these two-port network parameters.

Further insight into circuit performance is gained through transient analysis, represented in Figures 13 and 14. These figures reveal the time-domain response of the amplifier circuits under test. A clear amplification of both the output voltage and current is observed compared to their input counterparts, confirming the intended operation of the power amplifier. More importantly, the gain characteristics observed in the simulations correspond well with the values obtained in the mathematical analysis, reinforcing the consistency and reliability of the developed system.

Another key observation is the efficiency and responsiveness of the software during simulation runs. The system generated results swiftly, making it practical for both instructional use and early-stage design testing. This responsiveness makes the package especially beneficial for students, educators, and design engineers who require fast feedback during circuit development and analysis.

In summary, the integration of impedance and admittance parameter models into a computer-aided analysis environment has proven effective. It delivers accurate results, mirrors theoretical expectations, and offers fast, responsive simulation—all of which are essential for modern electronic circuit design and analysis workflows.

4. Conclusion

The study carried out analysis of a power amplifier using the impedance and admittance parameters by carefully deriving the performance quantities from the first principle. The amplifier can be simulated and a transient plot analysis base on the performance quantities obtained can be obtained. From the results obtained, these performance quantities namely power gain, voltage gain, and current gain, input and output impedances as well as admittances have been successfully computerized. With the utilization of the circuit analysis package and either impedance or admittance parameters, reliable determination of the operating parameters for class A, B, and AB power amplifiers is now attainable. The challenges associated with manual analysis of power amplifiers have been largely overcome through the implementation of this software package. When provided with accurate input values, the software package ensures precise and error-free analysis of single-stage transistor power amplifiers. Consequently, this computer-aided circuit analysis package for single-stage power amplifiers is regarded as an indispensable tool for design engineers, industrialists, scientists, and educational institutions.

Nomenclature

V_1 - Input voltage
V_2 - Output voltage
I_1 - Input current
I_2 - Output current
z_{11} - Input impedance
z_{12} - Reverse transfer impedance
z_{21} - Forward transfer impedance
z_{22} - Output impedance,
y_{11} - Input admittance
y_{12} - Reverse transfer admittance
y_{21} - Forward transfer admittance
y_{22} - Input impedance
V_L - Load voltage
V_S - Voltage source
R_L - Load resistance
R_S - Series resistance

References

Amhenrior, H.E. (2018). Analysis of Hybrid Parameters of a Single Stage Small Signal Transistor Amplifier Using Two-Port Network. Nanoscience and Nanometrology. 4(1), 9-15. DOI: 10.11648/j.nsnm.20180401.12.

Amhenrior H. E.,Amhenrior S. A. (2018). Mathematical Review and Circuit Analysis Software Development for Small-Signal Single-Stage Transistor Amplifier Using Hybrid Parameter. American Journal of Quantum Chemistry and Molecular Spectroscopy. 2(1), 1-8.doi: 0.11648/J.AJQCMS.20180201.11

Cuntan C., Panoiu C., Panoiu M., Baciu I.,Mezinescu S. (2024). *Using the LabVIEW Simulation Program to Design and Determine the Characteristics of Amplifiers*. Chips. 3, 69–97. doi: 10.3390/chips3020004.

Dan-Abia D., Obot A.,Udofia K. (2019). Design and Analysis of a Multistage Common Emitter Amplifier for Low Frequency Applications. European Journal of Engineering Research and Science. 4(10). doi:10.24018/ejers.2019.4.10.1431.

Pandiev I. M.,Stoimenov E. C. (2020). Development of Single-Transistor Amplifier Modular Laboratory Kits for Electronic Engineering Education. 2020 XI National Conference with International Participation (ELECTRONICA), Sofia, Bulgaria, 1-4.doi: 10.1109/ELECTRONICA50406.2020.9305170.

Qiuyue L., Jizheng S., Jun C. (2024). Application of Multisim in the Course of Electricity and Electronics Technology of Internet of Things Application Technology in Higher Vocational Colleges Frontiers in Educational Research. 7(5), 31-35, doi: 10.25236/FER.2024.070505.

Thain W. (2022). A SPICE Model for Bipolar Junction Transistor Amplifier Design Verification That Uses Specified DC and Small-Signal AC Parameters Simultaneously. Southeast Con 2022. 355-360.doi: 10.1109/SoutheastCon48659.2022.9763891.

Wardhana A. W., Ramadhani Y.,Priswanto (2022). Design and Simulation of a Multistage Common-Emitter, Common-Collector, AC Voltage Amplifier. Journal Nasional TeknikElektro. 11(2).doi:10.25077/jnte.v11n2.1009.2022.

Wong Y. C., Syafeeza A. R., Hamid N. A., (2015). A Transistor Sizing Tool for Optimization of Analog CMOS Circuits: TSOP. International Journal of Engineering and Technology. 7(1), 140-146