

A SIMULATION OF SINGLE STAGE BJT AMPLIFIER USING LTSPICE

***Mohammad Nizam Ibrahim, Zainal Hisham Che Soh, Irni Hamiza Hamzah, Ali Othman**

Faculty of Electrical Engineering
Universiti Teknologi MARA (UiTM) Pulau Pinang
13500 Permatang Pauh, Pulau Pinang, Malaysia

*Corresponding author's email: mnizam@ppinang.uitm.edu.my

Abstract

Students and academicians at universities around the world use a variety of simulation tools to measure important characteristics of electrical and electronic circuits. Most students, however, cannot afford to own the simulation tools because those can be very expensive. Without having any simulation tools, a learning process can be hindered since the students are required to complete a number of homeworks, assignments, and pre-laboratory tasks outside their classrooms. This paper presents a popular and free of charge simulation tool for electrical and electronic circuits known as LTSpice. To demonstrate the capability of the tool, a basic single stage common emitter Bipolar Junction Transistor (BJT) amplifier is constructed and simulated by using the LTSpice. There are four important characteristics of the amplifier: emitter current, voltage gain, input impedance, and output impedance are measured and then compared with theoretical calculation. The comparison results show a high accuracy is achieved for all the amplifier characteristics. The measured emitter current and output impedance achieved 100% accuracy while the voltage gain and input impedance achieved 99.1% and 99.7% accuracy, respectively. Therefore, this paper provides an encouragement for the students and academicians to own and use the LTSpice simulation tool to simulate electrical and electronic circuits in their learning and teaching processes.

Keywords: LTSpice simulation; time domain analysis; learning process; teaching tool

1.0 INTRODUCTION

Nowadays, there are a variety of electrical and electronic simulation tools available in the markets for electrical and electronic engineering courses. Students and academicians use the simulation tools commonly to observe and measure electrical characteristics mainly on current and voltage signals. They also use the tools to understand and investigate the behavior of electrical and electronic circuits. The usage of the simulation tools would help the users to do an experiment easily with different component values and apply 'what-if' analysis (Hall, 2000) and (Shackleford, 1995). In semiconductor industry, engineers use extensively the simulation tools because the circuit simulation process is one of important stages in analogue electronics design (Baker, 2011). The results of simulation are highly valuable since they can be used as a reference for modification and verification of the design circuit in order to meet the design requirements.

The simulation tools can be categorized into two types: non-commercial and commercial. The non-commercial simulation tools are commonly free and open-source program but their capabilities are limited. In contrast, the commercial simulation tools provide unlimited capabilities but their price can be

very expensive. Some commercial simulation developers, such as OrCAD software (2014), support the learning process of electrical and electronic students by providing them with a student version simulation tool. However, the student version works with limited capabilities. Other commercial developers might provide a student version simulation tool that has unlimited capabilities but the functionality of the tool is time-limited.

The use of the simulation tools becomes a part of educational environments and a complement to most of laboratory works (Pisarski, 1999). Therefore, universities across the world provide their academicians and students with simulation tools particularly with the commercial tools. However, the number of commercial tools being purchased and fully licensed was limited since the tools are very expensive. Thus, students might need to share the simulation tools among them during the laboratory sessions. Moreover, the tools might be available only at specific laboratories. The limited number of available simulation tools and the time constraint may prevent students to enhance their skills and understanding the electrical and electronic courses. This is because enhancement of student skills in any simulation tools requires a lot of hand-on practise on the tool.

LTSpice is one of the promising simulation tools for simulation of electrical and electronic circuits. It is a free of charge tool with a high performance provided by Linear Technology Corporation (LTC, 2014) which is one of the industry leaders in producing analog and digital integrated circuits. The tool is based on Simulation Program for Integrated Circuits Emphasis (SPICE). It is worth to note that the SPICE-based program is a well known as the industry standard program in electrical and electronic engineering. Although the LTSpice is a free tool, the user can simulate the circuits without limitation of the circuit sizes, complexity and components counts. The LTSpice contains a comprehensive library which the user may select a variety of LTC components. If needed, the user may also import third party models from a number of semiconductor manufacturers such as the National Semiconductor and the Texas Instruments.

Adopting the LTSpice as a simulation tool can lead to many advantages. The user-friendly LTSpice can be easily used to observe and verify, numerically and graphically, all the magnitude of current or voltage of electrical and electronic circuits. The most basic electrical and electronic engineering courses taught at universities, particularly at diploma level, require students to only verify (without fabricating) the simple and basic circuit. Since the LTSpice is a free tool, the students may install the program in their laptop and then perform the circuit verification independently at their convenient time and place. For advanced electrical and electronic engineering courses whereby students may require to design electrical and electronic circuit, the LTSpice can be used to design a real-world circuit using LTC components or other third party models from different semiconductor manufacturers.

The aim of this paper is to demonstrate and suggest the use of LTSpice as a simulation tool for simulation of electrical and electronic circuit. For instance, a basic common emitter Bipolar Junction Transistor (BJT) amplifier circuit is constructed in LTSpice and four important characteristics like emitter current, voltage gain, input impedance, and output impedance are measured. The measured characteristics are compared against theoretically calculation.

2.0 COMMON EMITTER AMPLIFIER

Bipolar Junction Transistor (BJT) is a device with three terminals known as emitter, base, and collector. The BJT can be categorized into two types which known as NPN and PNP (Anderson & Anderson, 2005). For NPN, the emitter and collector of the BJT are made from n-type material while its base is made from p-type material. In contrast, the emitter and the collector of PNP are made from p-type material while its base is made from n-type material. In either type, a common emitter amplifier is configured by passing an input signal to the base terminal while measuring an output signal at the

collector terminal (Boylestad & Nashelsky, 2006). For the purpose of theoretical calculation, we would consider NPN type of BJT. Figure 1 shows an example of NPN type of BJT that configured as a common emitter amplifier. In this configuration, the input signal is voltage input (V_{in}) while the output signal is voltage output (V_{out}). The emitter terminal is common to both the input and output voltages and therefore it is known as a common emitter.

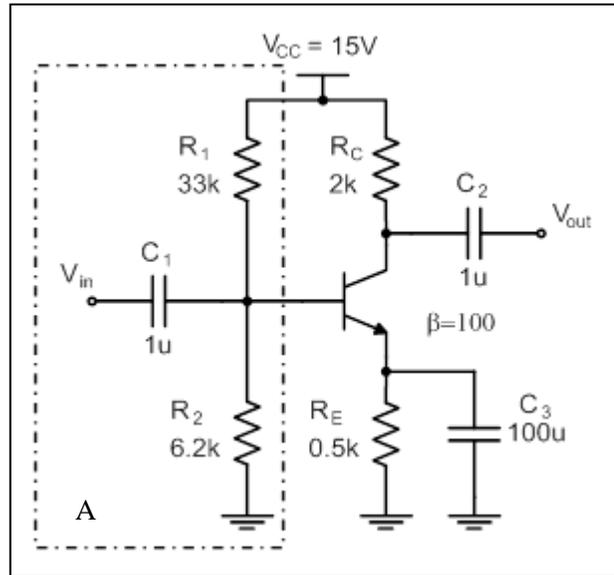


Figure 1 BJT common emitter amplifier

One important parameter for an analysis of BJT is a dc beta (β) in which its value is commonly in the range of $50 < \beta < 300$. The value of β varies since it depends on BJT fabrication techniques and process tolerances (Neaman, 2010). To illustrate theoretical calculation of common emitter amplifier, it is assumed that the $\beta = 100$, $V_{BE} = 0.7V$ and input voltage $V_{in}(t) = 10mV\sin(2\pi ft)$ with the frequency, $f = 10kHz$.

The block A (dash-dot box) of Figure 1 can be simplified into thevenin equivalent circuit. Thevenin equivalent circuit consists of voltage source (V_{TH}) and series resistance (R_{TH}) (Alexander & Sadiku, 2008).

$$V_{TH} = \frac{R_2}{(R_1 + R_2)} V_{CC} = \frac{6.2k}{(33k + 6.2k)} 15 = 2.37V$$

$$R_{TH} = R_1 // R_2 = 33k // 6.2k = 5.22k\Omega$$

With the thevenin equivalent circuit parameters (V_{TH}, R_{TH}) are replaced on the block A, the emitter current (I_E) can be calculated as:

$$I_E = \frac{V_{TH} - V_{BE}}{\left(\frac{R_{TH}}{\beta + 1} + R_E\right)} = \frac{(2.37 - 0.7)}{\left(\frac{5.22k}{101} + 0.5k\right)} = 2.85mA$$

Further, the emitter resistance of the BJT (r_e) and base-emitter internal resistance (r_π) can be calculated as following (Boylestad & Nashelsky, 2006):

$$r_e = \frac{26mV}{I_E} = \frac{26m}{2.85m} = 9.12\Omega$$

$$r_\pi = (\beta + 1)r_e = (101)(9.12) = 921.3\Omega$$

The theoretical calculation of voltage gain (A_V) for the common emitter amplifier can be estimated as:

$$A_V = -\frac{R_C}{r_e} = -\frac{2k}{9.12} = 219.2V/V$$

The input impedance (Z_i) and output impedance (Z_o) of the common emitter amplifier are given by:

$$Z_i = R_{TH}/r_\pi = 5.22k/921.3 = 783.1\Omega$$

$$Z_o = R_C = 2k\Omega$$

3.0 SIMULATION AND DISCUSSION

A common emitter BJT amplifier constructed in LTSpice is shown in Figure 2. The model of BJT is NPN type 2N2222. Since we have assumed the value of dc beta $\beta = 100$ in theoretical calculation, we need to write a statement in SPICE directive. In this case, we use ‘.model 2N2222 NPN(bf=100)’ statement that changes the default value of the dc beta $\beta = 100$.

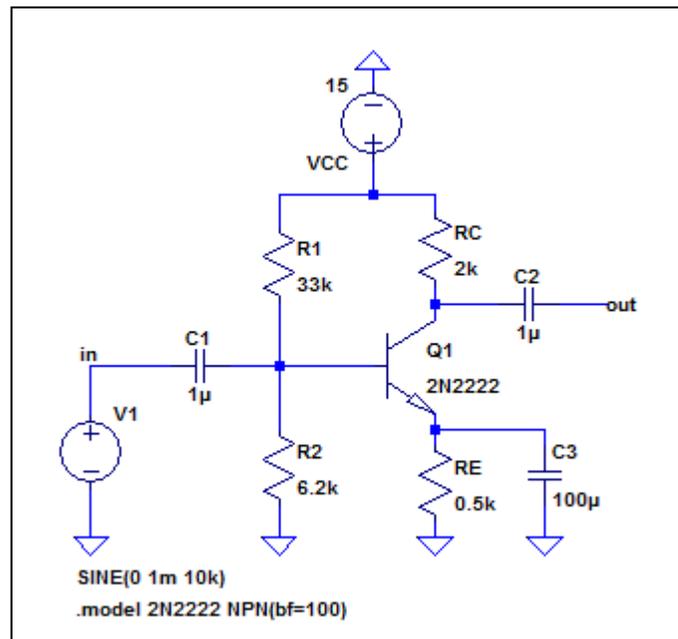


Figure 2 BJT common emitter amplifier in LTSpice

First, LTSpice DC operating function is applied to the BJT common emitter amplifier circuit. Figure 3 shows a snapshot of the emitter current value measured by a current probe. The result shows that the emitter current value $I_E = 2.85mA$.

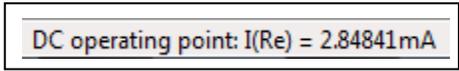


Figure 3 Snapshot of emitter current value in LTSpice

Second, LTSpice transient analysis function is applied to the circuit in order to visualize graphically both the input and output voltages (V_{in}, V_{out}). Figure 4 shows both the input voltage (top) and output voltage (bottom).

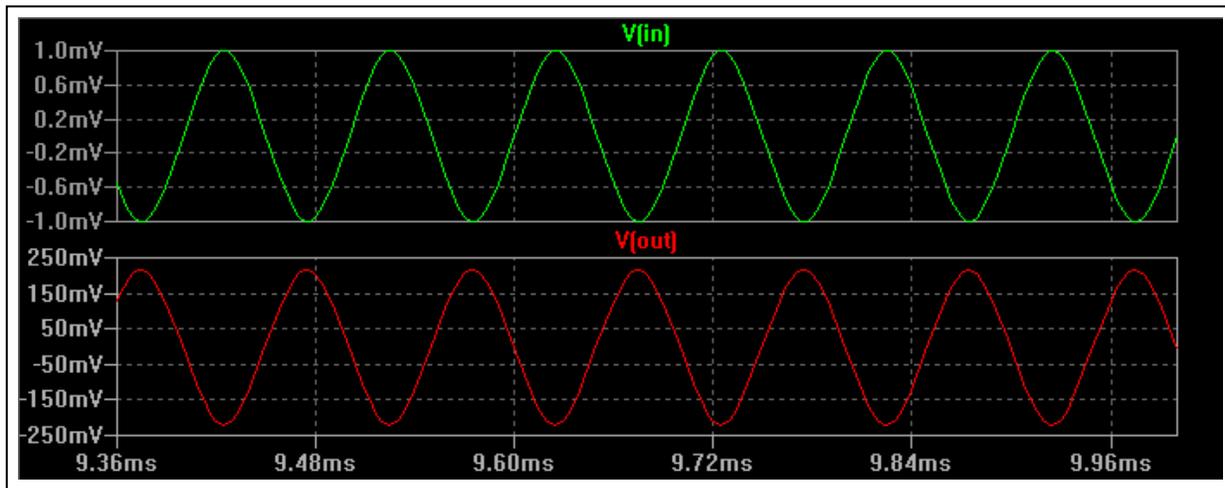


Figure 4 Input voltage (V_{in}) and output voltage (V_{out})

Further, Figure 5 shows details of measurements on maximum peak and minimum peak for the input and output voltages using LTSpice waveform viewer tool. The result shows that the peak-peak output voltage is $V_{out(pp)} = -434.4mV$ whereas the peak-peak input voltage is $V_{in(pp)} = 1.99mV$. Therefore, the voltage gain is $A_V = \frac{V_{out(pp)}}{V_{in(pp)}} = -217 V/V$.

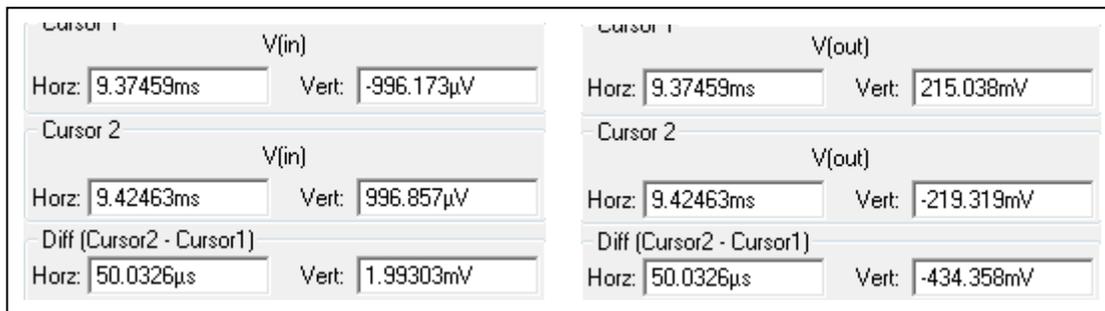


Figure 5 Detail measurements of input voltage (V_{in}) and output voltage (V_{out})

Finally, LTSpice AC sweep analysis is applied to the circuit in order to measure the input and output impedances (Z_{in}, Z_{out}). To measure the input impedance, the voltage source $V1$ in Figure 2 is replaced by equivalent 1 ampere AC current source with a large resistor across this current source as shown by Figure 6. Then AC sweep analysis is applied for frequencies ranging from 1Hz to 100MHz.

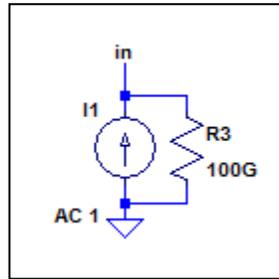


Figure 6 Equivalent current source used for measuring input and output impedance

Figure 7 shows the plot of $V_{in}/I1$ versus the range of sweep frequency. Using the LTSpice waveform viewer tool as shown by Figure 8, the details result of the input impedance $Z_i = 780.6\Omega$ which is measured at the frequency of 10kHz.

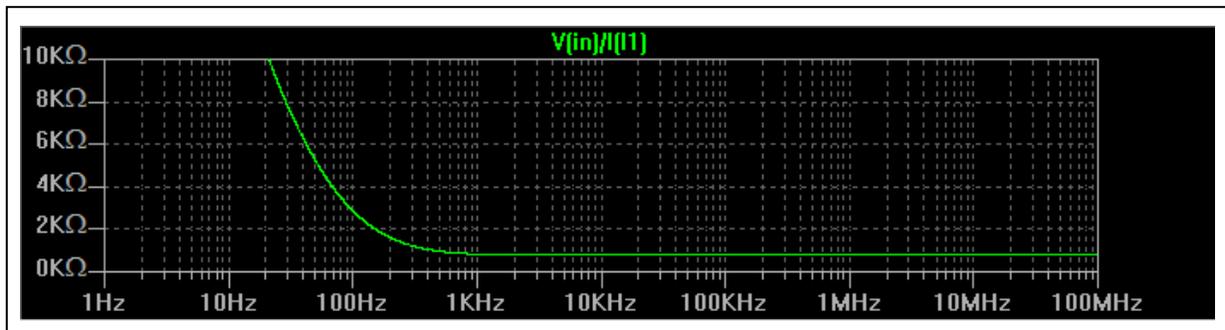


Figure 7 AC sweep of input impedance



Figure 8 Detail measurements of input impedance (Z_i)

To measure the output impedance the voltage source $V1$ is replaced by short circuit while the current source of Figure 6 is connected at output node. The AC sweep analysis is applied for frequencies ranging from 1Hz to 100MHz.

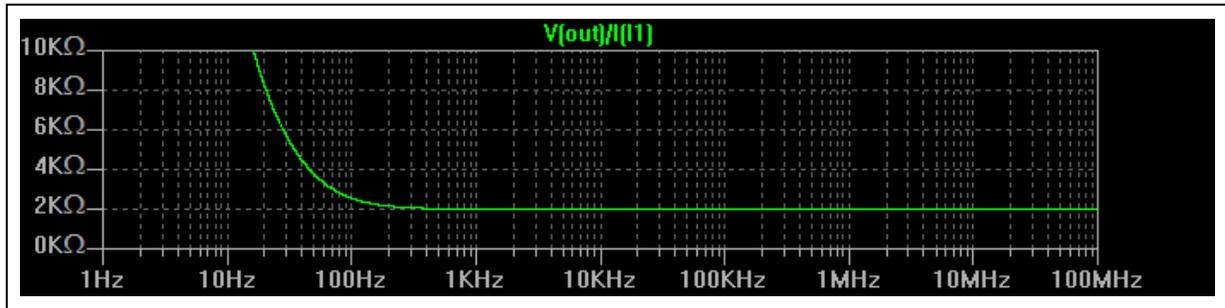


Figure 9 AC sweep of output impedance

Figure 9 shows the plot of $V_{out}/I1$ versus the range of frequency. The result shows that, at frequency of 10kHz, the output impedance $Z_{out} = 2k\Omega$.

Table 1 summarizes the results for the simulation in LTSpice and theoretically calculation on emitter current, voltage gain, input- and output-impedances of the BJT common emitter amplifier. Further, the accuracy results on the simulation as compared to theoretically calculation is calculated as follows:

$$Accuracy (\%) = \frac{LTSpice \text{ Simulation}}{\text{Theoretically calculation}} \times 100\%$$

It shows that the results of simulation are of a high accuracy as compared to the theoretically calculation.

Table 1 Summary of simulation in LTSpice, theoretically calculation and accuracy

Characteristics	LTSpice simulation	Theoretically calculation	Accuracy (%)
Emitter current (I_E)	2.85mA	2.85mA	100
Voltage gain (A_v)	-217V/V	-219V/V	99.1
Input impedance (Z_i)	780.6Ω	783.1Ω	99.7
Output impedance (Z_{out})	2kΩ	2kΩ	100

4.0 CONCLUSION

A common emitter Bipolar Junction Transistor (BJT) amplifier has been constructed and simulated by a free LTSpice simulation tool. The simulation results on the four basic characteristics of the BJT amplifier which are emitter current, voltage gain, input impedance, and output impedance were observed graphically and numerically. The simulation results were compared against theoretically calculation. The comparison results show a high accuracy is achieved for all of the four basic characteristics of the BJT amplifier. Thus, the LTSpice is a good simulation tool for electrical and electronic students to simulate, verify and investigate electrical and electronic circuits. Future ongoing research is to show a simulation of other types of amplifier and rectifier using the LTSpice.

References

- Alexander, C. K. & Sadiku, M. (2008). *Fundamentals of Electric Circuit, Fourth Edition*. Mc Graw Hill. ISBN: 978-0-07-352955-4.
- Anderson, B. L. & Anderson, R. L. (2005). *Fundamentals of Semiconductor Devices, International Edition*. Mc Graw Hill. ISBN: 0-07-236977-9
- Baker, R. J. (2011). *CMOS Circuit Design, Layout, and Simulation*. Wiley-IEEE Press. ISBN: 978-1-118-03823-9.
- Boylestad, R. L. & Nashelsky, L. (2006). *Electronic Devices and Circuit Theory*. Prentice Hall. ISBN: 0-13-197408-4.
- Hall, T. (2000). Using simulation software for electronics engineering technology laboratory instruction. *2000 American Society of Engineering Education Annual Conference*, 3547, St. Louis, MO.
- Linear Technology Corporation (2014). Retrieved from <http://www.linear.com/ltspace>.
- Neaman, D. A. (2010). *Microelectronics: Circuit Analysis and Design, Fourth Edition*. Mc Graw Hill ISBN: 978-007-128947-4.
- OrCAD (2014). Retrieved from <http://www.orcad.com/buy/orcad-educational-program>
- Pisarski, S. (1999). Impact of simulation software in the engineering technology curriculum. *1999 American Society of Engineering Education Annual Conference*, 2548, Charlotte, NC.
- Shackleford, R. A. (1995). , Learning about real components. *IEE Colloquium on Computer Based Learning in Electronic Education*.