International Journal of Scientific Research in Science, Engineering and Technology
Print ISSN: 2395-1990 | Online ISSN : 2394-4099 (www.ijsrset.com)

Study of RC Coupled Amplifier Using B2 Spice, TINA and Circuit Maker Software's

Dr. Sanjay K. Tupe[1]

[1]Assistant Professor, Department of Physics & In charge Principal, Kalikadevi Arts, Commerce & science college, Shirur (Ka), Dist.- Beed. 413249. Maharashtra, India

Article Info
Volume 9, Issue 5
Page Number : 81-84

Publication Issue :
July-August-2021

Article History
Accepted : 02 July 2021
Published: 25 July, 2021

ABSTRACT

This paper addresses the performance of RC coupled amplifier using B2 Spice, TINA and Circuit Makerelectronic circuit simulation software's. Traditionally electronic circuit design was verified by building prototypes, subjecting the circuit to the various stimuli and then measuring its response using appropriate laboratory equipment's. Prototype building is somewhat time consuming. But produces practical experience from which we judge the manufacturability of the design. Computer programs that simulate the performance of an electronic circuit provide a simple cost-effective means of confirming the intended operation prior to circuit construction and verifying new ideas that could led to improve the circuit performance.

Keywords: - RC Coupled amplifier, Amplification, Transient Analysis, Smoke Analysis Simulation.

I. INTRODUCTION

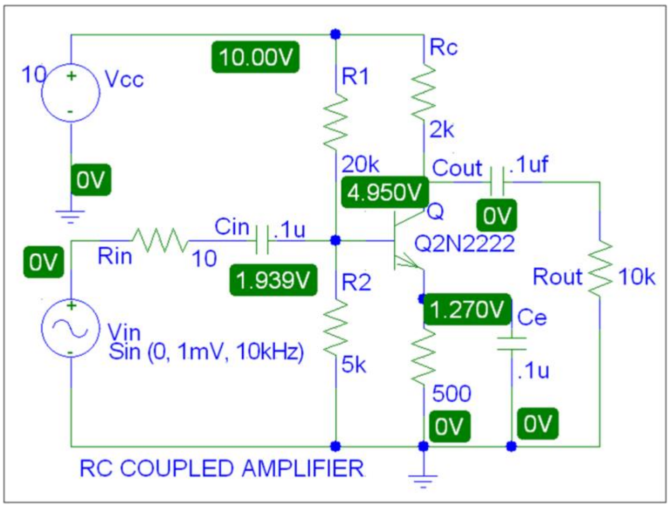
The evolution of electronics technology almost in to every facet because of low cost, reliability and ease of interface [1]. The electronic industry is getting progressively more and more efficiently more at new products in wide range and verity of circuits in service of human being. We also saw the more and more products coming in to the market in shorter time [2]. Hence low-cost circuit design, with an accurate, linear and faster testing techniques are addressed. A verity of electronic components PSpice commercially available which plays an important role in design development of accurate circuit design performance and optimum reliability [3].

II. SIMULATION

Electronic simulation of circuit function is now a common practice in the design of individual circuit and the complete systems. The most of the circuit designer can simulate, and design the circuit and develop it as early as they can and hence in market [4]. Spice software models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is versatile programmed and is widely used both in Industries and Universities. [5]. The circuit performance and its reliability in any circuits for to minimize the failure can be tested. To meet the required standard of the circuits and hence quality instruments, the circuit analysis is performed. In case of any failure or problem on can easily redesign it by modifying the very same circuit in a few minutes using highly sophisticated simulation tools [6].

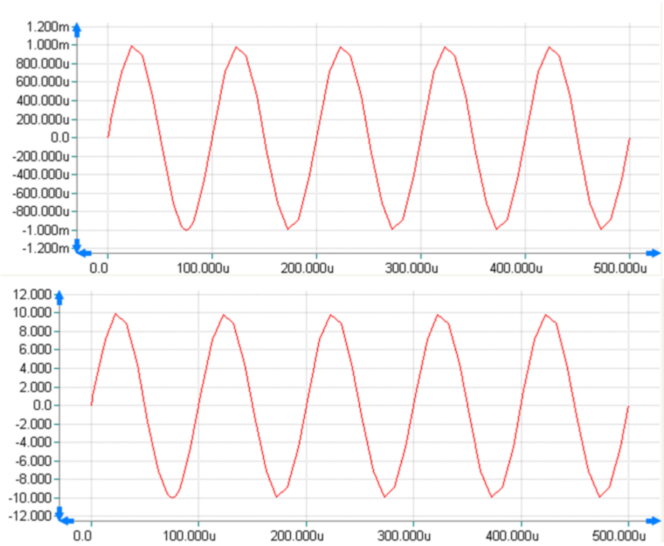
The role of spice software's is very vital in all fields of engineering and industries for the design and built the electronic circuits. Today many groups around the world are investigating advanced software capable of responding a wide verity of components. Recent years have witnessed the excellent progress in the field of spice software. These improve the ability of users to integrate different types of electronic circuits in to their systems or applications. The spice software would have more capability to design and built electronic circuits in wider range of applications.

In case of classroom / laboratories study teaching the spice, experiments will be tried for example in the design of RC coupled amplifier. Here various software's can be come to our reuse and the effect can be easily demonstrated by changing various capacitors so also can be done in case of other circuits of amplifiers and oscillators even for modulation studies.

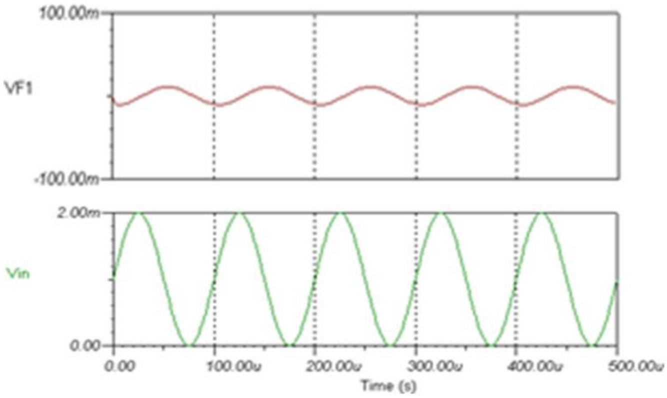


Circuit diagram of the RC coupled amplifier

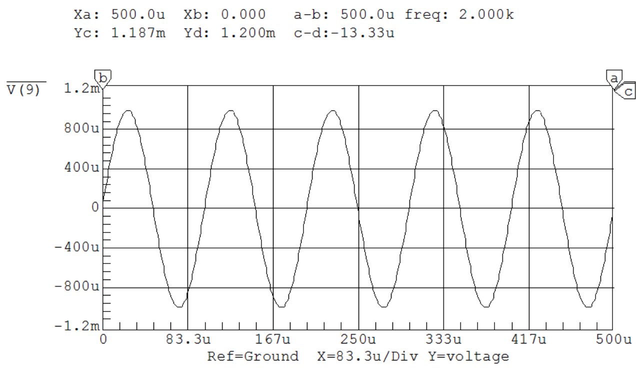
III. GRAPHICAL OUTPUTS

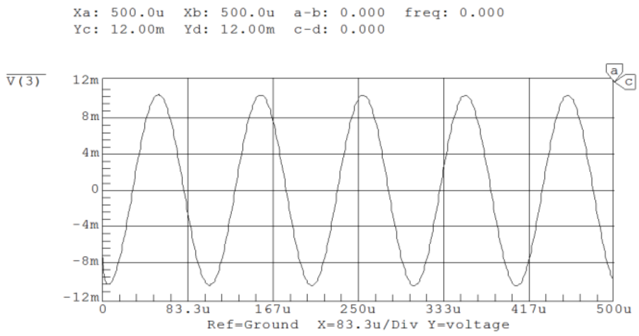


The fig shows input and output signals of the RC coupled amplifier by using B2 Spice Software.



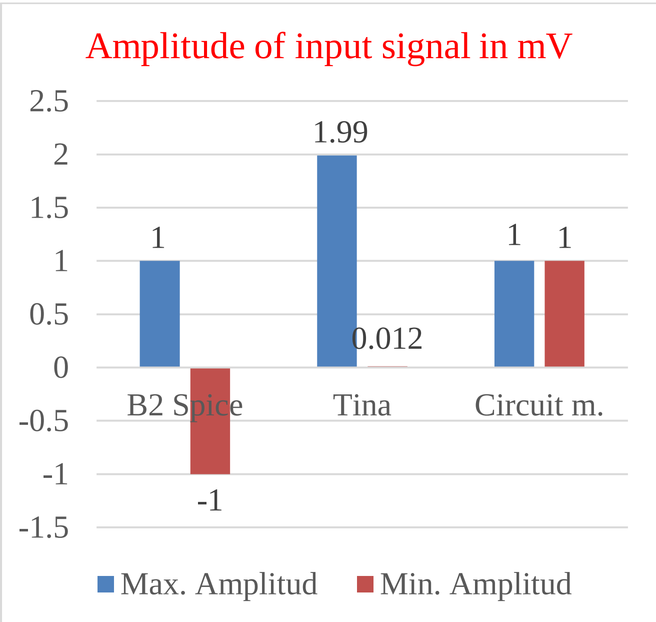
The fig shows input and output signals of the RC coupled amplifier by using TINA Software.



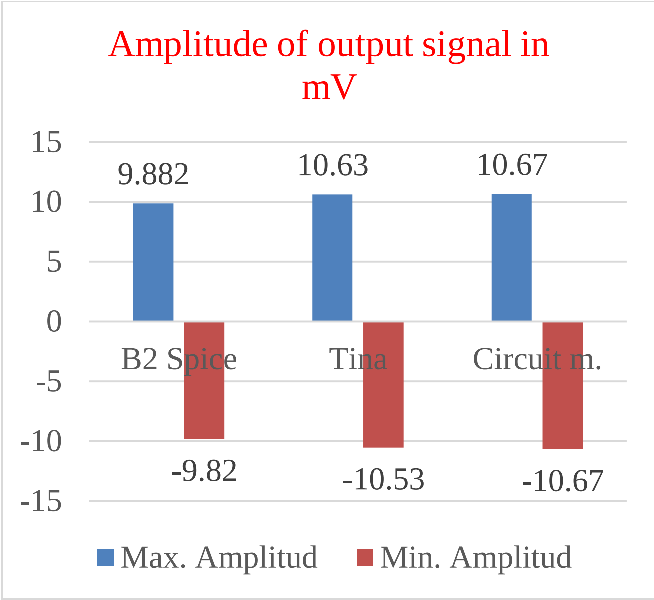


The fig shows input and output signals of the RC coupled amplifier by using Circuit Maker Software.

Software	Amplitude of input signal	
	Max. Amplitude	Min. Amplitude
B2 Spice	1 mV	-1 mV
Tina	1.99 mV	0.012 mV
Circuit m.	1 mV	1 mV



Software	Output Signal Amplitude	
	Max. Amplitude	Min. Amplitude
B2 Spice	9.882 mV	-9.82 mV
Tina	10.63 mV	-10.53 mV
Circuit m.	10.67 mV	-10.67 mV



Spice helps to determine which components are over-stressed using Smoke analyses or observed yields using Monte Carlo analysis are help to prevent board failures. Advanced simulation performance technology save time, improves reliability, and speeds convergence on larger designs.

IV. CONCLUSION

When we built the RC coupled amplifier circuit of same configuration in B2 Spice, TINA and Circuit Maker software's we observed the above results. These are very close to the actual built of circuit results. But in TINA and Circuit Maker, We can't observe the proper input signal wave. Means these software's are useful for to build and design the number of electronic circuits for the human welfare, are the time sever as well useful for the virtual education in pandemics condition and fulfils the anytime anywhere lab requirements.

V. REFERENCES

[1]. PSpice Schematics, Evaluation Version9.1 www.cadence.com
[2]. TopSPICE/Win32 version 7.16c by panzer development. www.penzar.com

[3]. B2 Spice A/D 5.2.3, Beige Bag Software www.beigebage.com info@ beigebage.com
[4]. TINATM for Windows, The Complete Electronics Lab version 6.00.008SFS.
[5]. CircuitMaker V6.2C Protel Technology, Inc. 5252N Edgewood Dr Ste175 Provo UT84604 USA.
[6]. Muhammad H. Rashid, Introduction to PSpice Using OrCAD for circuits and electronics. Prentice hall of India private limited, New Delhi – 110 001, 2006.
[7]. “Comparative study of different spice software’s using astable multivibrator in different spice software’s”, Sanjay K. Tupe, Sayyad S.B. and S.H. Behare. International Journal of Recent Trends in Engineering, CEE NOV 2009.
[8]. Maheshwari L. K. and Anand M.M.S., Laboratory Experiments and PSPICE Simulation’s in Analog Electronics., Prentice Hall of India Pvt. Ltd., New Delhi, (2007).
[9]. http://www.ecircuitcenter.com/About SPICE.htm.
[10]. Operational Amplifier - G. B. Clayton
[11]. Principles of Electronics - V. K. Mehta
[12]. Electronic Principles - A. P. Malvino
[13]. Comparative study of various circuit simulation software’s”, A.V.Mancharkar, S.K. Tupe, A.S. Jadhav, J.B. Patwardhan and S.H. Behare. 95th Indian Science Congress, Jan 2008, Andhra University, Visakhapatnam.
[14]. “Comparative study of various circuit simulation software’s by using triangular waveform generator circuit”, S.K. Tupe, B. Deshmukh, and A.V. Mancharkar. Presented 97th Indian Science Congress, Jan 2010, to be held at Thiruanantpurum.
[15]. RC Coupled Amplifier htm.