I J R B A T, Issue (Special-17), June 2021: 235-239

A Double-Blind Peer Reviewed & Refereed Journal

OPEN ACCES

e-ISSN 2347 – 517X

Original Article

INTERNATIONAL JOURNAL OF RESEARCHES IN BIOSCIENCES, AGRICULTURE AND TECHNOLOGY

STUDY OF RC COUPLED AMPLIFIER USING PSPICE AND TOP SPICE

Sanjay K. Tupe

Department of Physics, Kalikadevi Arts, Commerce & science college, Shirur (Ka), Dist.- Beed. 413249. Maharashtra. India.

ABSTRACT:

This paper addresses the performance of RC coupled amplifier using PSpice and Top Spice electronic 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, Simulation.

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].

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.

Theory of RC coupled amplifier: - A transistor amplifier is designed to work in the active region. A single stage amplifier is designed using voltage divider bias. In a voltage divider bias

$$\mathrm{V_B} = \mathrm{V_{CC}}\frac{\mathrm{R_2}}{\mathrm{R_1} + \mathrm{R_2}}$$

Applying Kirchhoff's *Voltage Law* for the output side of amplifier

$$\mathrm{V_{cc}} = \mathrm{I_c R_c} + \mathrm{V_{cE}} + \mathrm{I_E R_E}$$

$$\mathrm{I_E} \cong \mathrm{I_C}$$

$$\mathrm{R_C} = \frac{\mathrm{V_{CC}} - \mathrm{V_{CE}} - \mathrm{V_E}}{\mathrm{I_C}}$$

$$V_E = I_E R_E \cong I_C R_E \quad \therefore \; R_E = \frac{V_E}{I_C}$$

Design of R1, R2, f & CE

For a fluctuation in I_{R1} and I_{R2} there will be small change in I_B. For example, if I_{R1}=21I_B and a 5% change in I_{R1} occurs there will be only $\frac{5}{21}$% change in I_B. Therefore, the circuit will be stable against small changes in R_1 and R_2 due to temperature or tolerance. $\mathrm{R_2} \leq \frac{\beta \mathrm{R_E}}{10}$

$$V_B = V_{BE} + V_E \qquad \therefore$$

$$\mathrm{R_2} = \frac{\beta \mathrm{R_E}}{10}$$

$$V_B(R_1 + R_2) = V_{CC} R_2$$

$$R_1 = V_{CC}\frac{R_2}{V_B} - R_2$$

The coupling capacitor C_{in} along with resistance combinations shown in fig forms a high pass filter whose cut-off frequency; f = $\frac{1}{2\pi RC}$ $\quad \therefore C_E = \frac{1}{2\pi R_E f_{IE}}$

Some tolerances have been made in order to obtain standard values of components. All capacitors are designed to obtain a fair frequency response curve.

Transient Analysis:

In Transient Analysis, we perform the simulation of the circuit and analyze the output voltage with respect to time. For this maximum simulation time and the time limit are set in the corresponding parameters window. Then simulation is performed with 'Display Waveform' option enabled. The output of the amplifier is viewed in the Waveform viewer. The user also has the provision to simulate with print and plot outputs.[7]

CONCLUSION:

When we built the RC coupled amplifier circuit of same configuration in PSpice & Top Spice software's we observed the above results. These are very close to the actual built of circuit results. 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.

REFERENCES:

P. Spice Schematics, Evaluation Version9.1 www.cadence.com

Top SPICE/Win32 version 7.16c by panzer development. www.penzar.com

Muhammad H. Rashid, Introduction to PSpice Using OrCAD for circuits and electronics. Prentice hall of India private limited, New Delhi – 110 001, 2006.

"Comparative study of different spice software's using a stable 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.

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).

http://www.ecircuitcenter.com/About SPICE.htm.

Operational Amplifier - G. B. Clayton

Operational Amplifier and Linear Integrated Circuits - R. A. Gaikwad

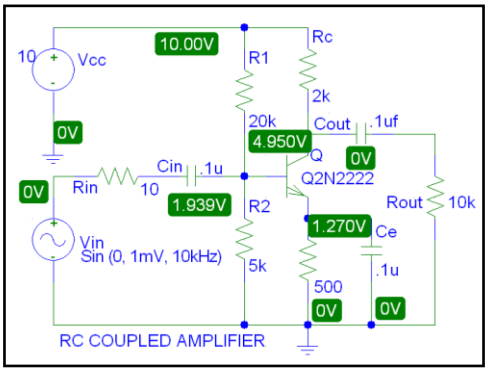
Principles of Electronics - V. K. Mehta

Electronic Principles - A. P. Malvino

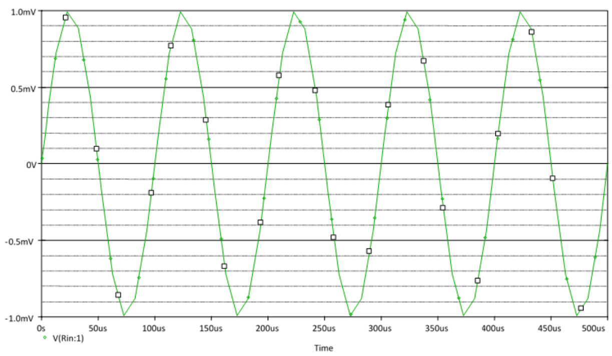
Comparative study of various circuit simulation software's", A.V. Mancharkar, S. K. Tupe, A.S. Jadhav, J.B. Patwardhan and S.H. Behare. 95th Indian Science Congress, Jan 2008, Andhra University, Visakhapatnam.

"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.

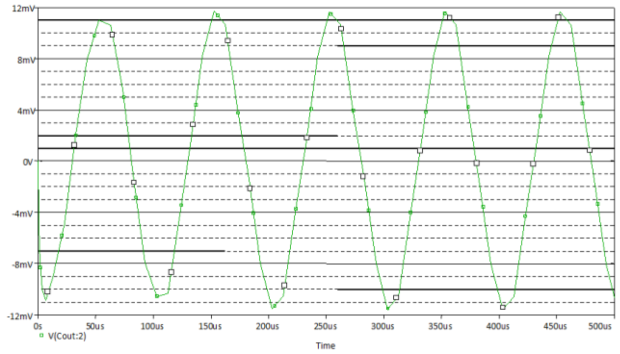
RC Coupled Amplifier htm.



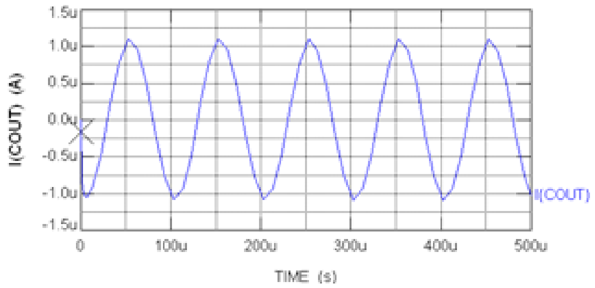
Circuit diagram of the RC coupled amplifier



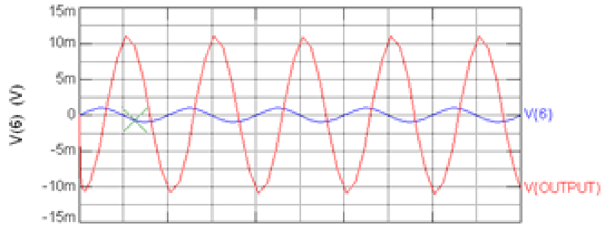
Input signal given to the RC coupled amplifier has amplitude 1 mV. [PSpice Software]



Output signal given to the RC coupled amplifier has amplitude 1 mV. [PSpice Software]



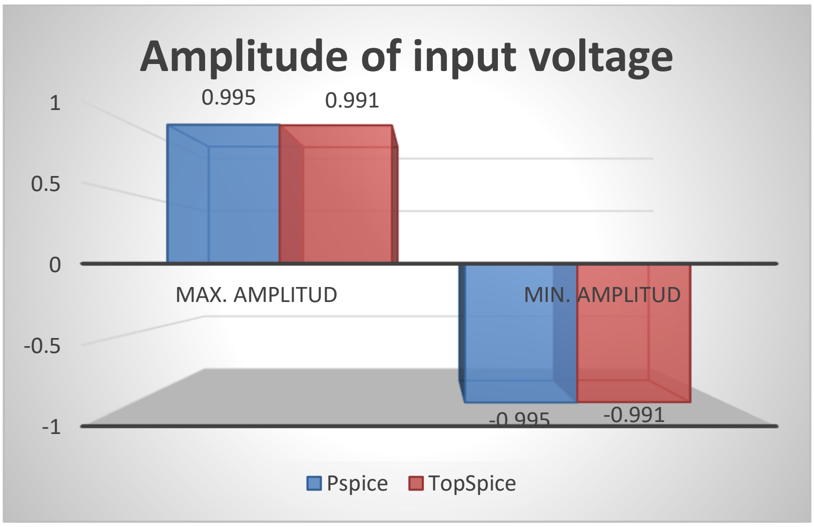
Output signal current given by the RC coupled amplifier has amplitude 12 µA. [Top Spice



Input signal has the amplitude 0.991 mV, Output signal given to the RC coupled amplifier has amplitude 11.071mV. [Top Spice

The following table shows the maximum and minimum input signal amplitudes in mV

Software	Amplitude of the input signal	
	Max. amplitude	Min. Amplitude
P Spice	0.995 mV	-0.995 mV
Top Spice	0.991 mV	-0.991 mV



The following table shows the maximum and minimum signal amplitudes in mV

Software	Amplitude of output signals	
	Max. Amplitude	Min. Amplitude
Pspice	11.674 mV	-10.561 mV
Top Spice	11.071 mV	-10.942 mV