Applications of Orcad Pspice Software in Designing, Simulation and Analysis of Various Applications of Operational Amplifier

Rajender kumar
SES, BPSMV
Sonipat India
rajender.mtech@gamil.com

Krishan Kumar
SES,BPSMV
Sonipat India
x2_krishan@yahoo.com

Abstract- This paper introduces a generalized method for simulating the various applications of operational amplifier generally termed as op-amps using Orcad PSpice. Now days engineering education is taught using simulation software. The usefulness of this SPICE software in learning various application of op-amp is enormous than proto typing practical laboratory. This PSPICE software’s which are important part of engineering education plays significant role in analyzing and simulating the various applications of op-amp such as adder, substractor, log & antilog amplifier, differentiator, integrator, multivibrator, waveform generators and comparator. We will present some designing, analysis and simulation of specific applications of op-amp using this software’s in detail which plays a vital role in various field of engineering.

Keywords: Operational amplifier, Orcad PSpice, SPICE, software.

I. INTRODUCTION

Operational amplifier in present-day is the most widely used electronics device in field of electronic systems such as communications systems, medical electronics, instrumentation, signal processing and performing many mathematical operations such as addition, subtraction, multiplication, integration and differentiations etc. It consists of various stages namely input stage, intermediate stage gain, level shifting stage and output stage and is fabricated as an integrated circuit. The symbol of typical op-amp is shown in fig.1.

3. Voltage gain is infinite
4. Bandwidth is also infinite
5. Slew rate is infinite
6. Common mode rejection ratio is infinite
7. Input Offset voltage is zero

Fig1. Circuit symbol of op-amps

It has two input terminal named inverting V- and non inverting terminal V+ and one output terminal Vout. Vs+ and Vs- are power supply terminal. The ideal operational amplifier characteristics are:

1. Infinite input impedance
2. Zero output impedance

But physical op-amp is not ideal. A physical op-amp has large voltage gain, high input impedance and small output impedance. The block diagram of op-amp is shown in fig 2.
The IC 741 is widely used general purpose op-amp whose circuit symbol is shown in fig 1. The Op-amp can be used in two configurations namely Open loop configuration and closed loop configuration. The open loop configuration is one in which no feedback in any form is fed to the input from the output whereas as closed loop configuration is in which a fraction of output is fed back to input. A few limitations of open loop configurations are: 1. Clipping of output waveform may occur if output voltage exceeds saturation voltage of op-amp. 2. Open loop gain is not constant 3. Bandwidth is negligibly small. 4. That’s why this configuration is not suitable for AC applications but it finds use in non linear applications such as comparators, square wave generator and astable multivibrator. The closed loop configurations finds its utilization in linear applications such as adder, transconductance amplifier, instrumentation amplifier, integrator, differentiator, log and antilog amplifier.

II. VARIOUS APPLICATIONS OF OP-AMP

Operational amplifiers which have origins in analog computers, are used in many linear, non-linear and frequency-dependent circuits. Characteristics of a circuit using an op-amp are set by external components with little dependence on temperature changes or manufacturing variations in the op-amp itself, which makes op-amps popular building blocks for circuit design. The various applications of op-amp is shown in fig 3.
Computers can be powerful tools if used properly, especially in the realms of science and engineering. Software exists for the simulation of operational amplifier based circuits by computer, and these programs can be very useful in helping circuit designers test ideas before actually building real circuits, saving much time and money. These same programs can be fantastic aids to the beginning student of electronics, allowing the exploration of ideas quickly and easily with no assembly of real circuits required. Of course, there is no substitute for actually building and testing real circuits, but computer simulations certainly assist in the learning process by allowing the student to experiment with changes and see the effects they have on circuits.

Throughout the practical session and beyond classroom, we would be incorporating computer printouts from circuit simulation frequently in order to illustrate important concepts. By observing the results of a computer simulation, a student can gain an intuitive grasp of circuit behavior without the intimidation of abstract mathematical analysis. To simulate various circuits involving op-amps on computer, we make use of different spice software’s like Orcad PSpice, Aim Spice, LTSpice and B2Spice. Both LTSpice and PSPICE are used professionally, and are based on the open source SPICE3 engine, which was developed at the University of California, Berkeley and serves as a foundation for electronic simulation. Simulating a circuit involves creating the circuit by placing parts and connecting with wires, running the simulation for the circuit, and then analyzing the results. In this paper we are present three different applications of op-amps which are simulated using OrCAD PSpice software.

IV. INTRODUCTION OF ORCAD PSPICE SOFTWARE

SPICE which stands for “Simulation Program for Integrated Circuit Emphasis” is general purpose circuit program that simulates electronics circuits. It can perform both basic and advanced analysis like DC sweep, AC sweep, Transient analysis, noise...
analysis and Fourier analysis. It was developed at Berkeley in University of California, USA. There are three platforms shown by figure 4(a-b) for PSpice which depends on SPICE version are as follows:
1. PSpice A/D or OrCAD PSpice A/D (Version 9.1 or above)
2. Pspice Schematics (Version 9.1 and below)
3. OrCAD Capture lite (Version 9.2 & above)

In PSpice A/D, circuit is described by statements and analysis commands which is called a circuit file. The platform for OrCAD Capture lite is similar to Pspice Schematics but with more advance features. In SPICE, A circuit is described to a computer by using circuit file which contains circuit details of components and elements, sources and commands for analysis descriptions and the way in which output is to be presented. A circuit file format for RLC circuit whose pulse response is to be calculated shown by figure 5 which can be read by SPICE is as follows:

```
1 3 2  L1  50uH
R1  2
V1  1 0  pulse(-200 200 0 1ns 1ns 100us 200us)
R1  1 2  2
L1  2 3  50uH
C1  3 0  10uF
```

After writing the circuit file, it is run on PSpice A/D platform; the simulated output is shown in figure 7.

V. DESIGN, ANALYSIS AND SIMULATION AND DISCUSSION
The first circuit which we have design, analyses and simulated is lossy integrator which is shown in fig.
6(a). The integrator is circuit which provides an output voltage proportional to the time integral of the input as given by equation 1.

\[ V_{out} = -\frac{1}{R_1 C_f} \int V_i dt \]  --- Eq. 1

If the input signal is as a square wave, then output will be a triangular wave as shown by fig. 5(b) as per eq.1.

![Lossy Integrator Circuit](image)

Fig. 6(a-b) PSpice Schematics of lossy integrator and its input and output waveform with OrCAD PSpice.

The 2\textsuperscript{nd} application of op-amps is waveform generator. Using op-amps, we can generate square wave and triangular waveform with single circuit shown by 7(a) and its output is shown by 7(b).

![Square and Triangular Waveform Circuit](image)

Fig. 7(a-b) PSpice Schematics of square and triangular and its input and output waveform with OrCAD PSpice

The third and final applications which we will design, analyses and simulate using OrCAD PSpice is narrowband band pass filter. Here we are taking the case of narrowband band pass filter shown in fig. 8(a) and its frequency response is by fig 8(b). The performance parameters such as bandwidth, resonant frequency etc. of this filter can be determined by resistor R1 and two capacitor C2 and C3 using following equations.
B = \frac{0.1591}{RC} \quad \text{Eq. 2}

Fig. 8(a-b) PSpice Schematics narrowband band pass filter and its input and output waveform with OrCAD PSpice

**VI. CONCLUSION**

This paper mainly discusses the various applications of 741 operation amplifier and their simulations using OrCAD PSpice software. We can design, simulate and verify any of applications with required parameters. Because of graphical facilities of OrCAD PSpice, it is useful in enhancing the understanding of various applications of op-amps. We can also use different spice software like LT spice, Top Spice and Spice Opus for better understanding of various applications and compare the results with theoretically calculated result.

**REFERENCES:**

[2] PSPICE A/D online manual

**AUTHOR'S BIOGRAPHIES**

**Rajender Kumar** has done his B.Tech in ECE from MDU, Rohtak and M.Tech in VLSI Design From GJUS&T, Hisar. He has published five papers in national journal and conferences. He is life member of ISTE. His main areas of interest are linear integrated circuits and VLSI Design.

**Krishan Kumar** has done his B.Tech in ECE from MDU, Rohtak and M.Tech in VLSI Design From GJUS&T, Hisar. He has published five papers in national journal and conferences. He is life member of ISTE. His main areas of interest are image processing and VLSI Design.