CHAPTER – I

INTRODUCTION TO FILTERS AND SIMULATION

CHAPTER – I

INTRODUCTION TO ACTIVE FILTERS AND SIMULATION SOFTWARE

1.1 INTRODUCTION

1.2 CLASSIFICATION OF FILTERS

1.3 ACTIVE FILTER

1.4 SIMULATION

1.5 SIMULATION SOFTWARE PSPICE

1.6 NEED OF SIMULATION

1.7 REQUIREMENT OF SIMULATION

1.8 LIMITATIONS OF PSPICE

CHAPTER –I

INTRODUCTION TO ACTIVE FILTERS AND SIMULATION SOFTWARE

1.1 INTRODUCTION :-

Audio signal processing provides a multitude of uses for active filters. Through simple filter functions Op. Amp can perform the precise response shaping and adjustments required in high quality audio system like equalized preamplifiers, active tone control and graphic equilizers. Filters are also useful for the simulation of the human vocal tract. For electronic and pop music effects voltage controlled filters or wah- wah filters are used. In rectifier circuit the output is pulsating type dc. To reject the ripples of ac mains from the rectified output the filter circuits are used. Filters are used not only to eliminate certain frequency components but also to emphasize or deemphasize them. Filters are used in virtually every phase of electronics and listing of all possible applications would be a long task.

1.2 CLASSIFICATION OF FILTER :-

A filter is often a frequency selective circuit that passes a specified band of frequencies and blocks or attenuates signals of frequencies outside this band. Filters are classified in number of ways.

1) Analog or digital
2) Passive or Active
3) Audio (AF) or Radio frequency (RF)

Digital filters process analog signals using digital techniques whereas analog filters are designed to process analog signals. Depending on the type of elements used in their construction, filters may be classified as passive or active. Elements used in passive filters are resistors, capacitors and inductors. Active filters on the other hand employ transistors or Op. Amp in addition to resistors and capacitors. Depending on the range of frequencies in which filters used are of two types of filters, audio frequency filter (20Hz – 20 kHz) & radio frequency filter (Above 20 kHz). The most common filters exhibit frequency dependent gain. The filter affects not only amplitude but also phase. The way in which a filter behaved with frequency is called frequency response. This response is further divided into magnitude and phase response. On the basis of magnitude response filters are classified as lowpass, highpass, band pass and band reject filter. Their idealized magnitude response and processing of incoming signals are shown in fig. 1.1. Because of their squared edges these responses are said to be the brickwall type.

i) The low pass response is characterized by frequency f_0 called the cutoff frequency such that $|H| = 1$ for $f < f_0$ and $|H| = 0$ for $f > F_0$, where H is the gain of the circuit , indicating that input sinusoids with frequency less than f_0 go through the filter with unchanged amplitude. While those with frequency greater than f_0 undergo complete attenuation.

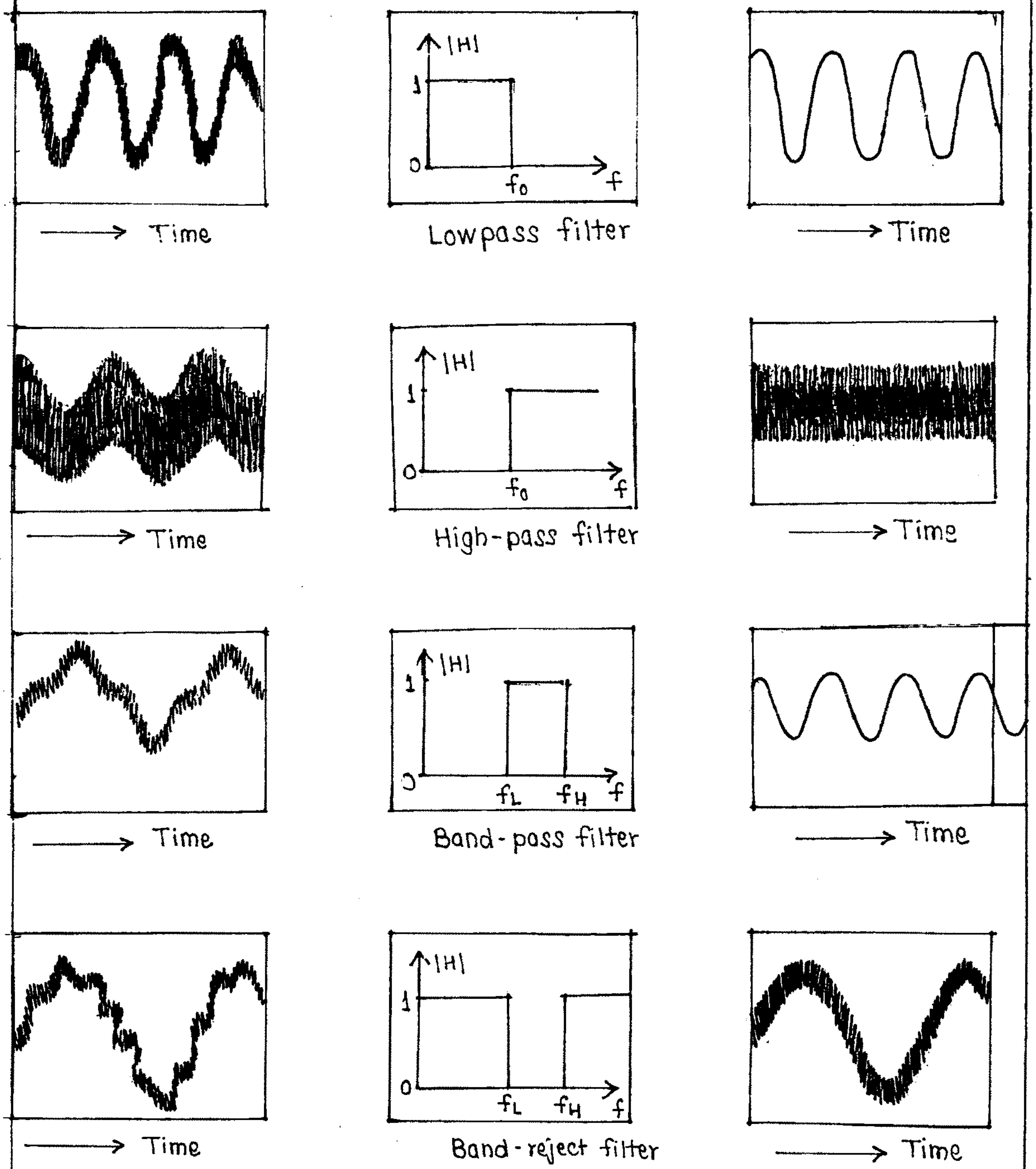

Fig 1·1

Idealized responses of four filter types in the time domain

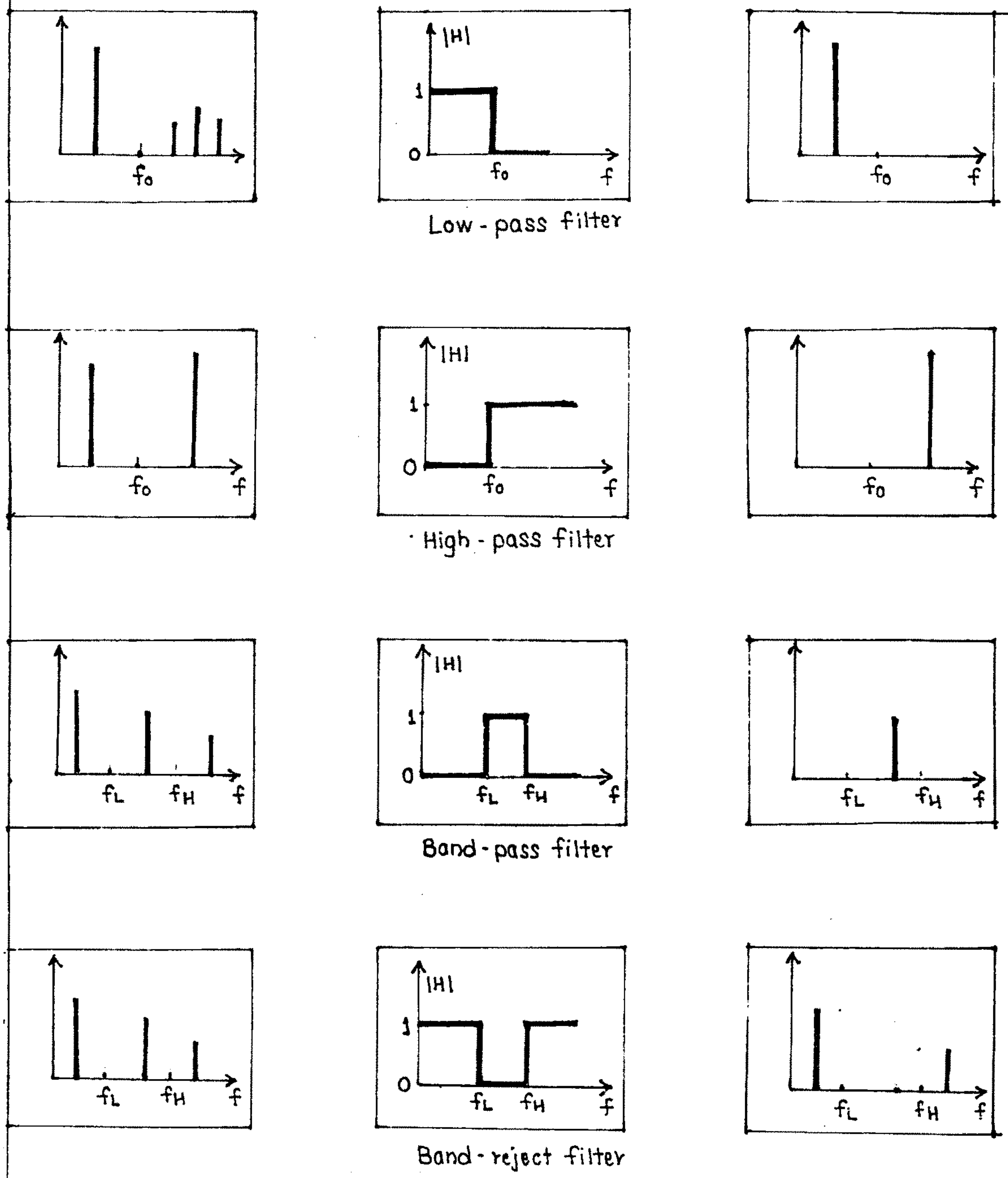

Fig 1·2

Idealized responses of the four filter types in the frequency domain

ii) The high pass response is complementary to the low pass response Input sinusoids with frequency greater than cut off frequency f_0 emerge from the filter with unchanged amplitude while those with frequency less than f_0 undergo complete attenuation.

iii) The band pass response is characterized by a frequency band $f_L < f < f_H$ such that input sinusoids with frequency within the band emerge unchanged while those with frequency outside the band are attenuated. The band reject response is complementary to the band pass response, as it blocks out the frequency component within the band $f_L < f < f_H$ while passing all the others.

The filter may be better understood in terms of its effect on the spectrum of the input signal, that is in frequency domain. This viewpoint is illustrated in fig. 1.2

1.3 ACTIVE FILTERS :-

Filters constructed using passive components like R, L and C are called passive filters. Some of the limitations of the passive filters are namely the size of the inductor at low frequencies, the necessity of buffers or isolation amplifier to prevent loading while cascading sections of filters and the need of an external amplifier to adjust the required gain.

Active filters consist of active devices like Op. Amp., transistor etc. It eliminates the inductor. Elimination of inductor is a great advantage. Since their performance is the least ideal of basic circuit elements. It is bulky heavy and expensive. Beyond the frequency reach of the operational amplifier inductor take over. So high frequency filters are still implemented with passive RLC components. Here inductor size and weight are more manageable since inductor and capacitor values decreases as operating frequency range is increased.

Low cost monolithic Operational Amplifier made active filters feasible. The departure of actual Op. Amp from ideality at high frequencies restricts active filter application below MHz range (audio and voice band range)

1.4 SIMULATION :-

Simulation allows designer to design to be modified and analyzed without effort, expense and time of building a prototype. Computer simulation is a powerful supplement to traditional design techniques. Simulation is an aid in initial design development during the bread boarding phase and during debugging and diagnostic phase. Simulation allows designer to test initial design theories before circuit design begins.

For many circuits breadboarding is impossible because of excessive circuit complexity, layout specific parasitic effects or as in the case of integrated circuits. For such circuits due to simulation one can investigate the circuit behaviour before building a working prototype.

With simulation designer can effectively predict the performance of a circuit by changing one or more circuit variables. Due to all above reason computer simulation is playing an increasingly important role in electronic circuit design.

1.5 SIMULATION SOFTWARE :-

In 1968 Ron Rohrer a junior faculty member at the university of California, Berkely designed a nonlinear circuit simulator called CANCER (Computer Analysis of Nonlinear circuits Excluding Radiation). CANCER simulator includes resistor, capacitors, inductors and two types of nonlinear device junction diodes and BJT. The lifetime of CANCER program was limited because the routine solution could handle no more than 400 components and or 100 circuit nodes.

During the early 1970, Larry Nagel continued improving the CANCER program. In 1971 this improved version, named SPICE 1 (Simulation program with Integrated circuit Emphasis) was released SPICE 1 quickly become an industry standard simulation tool. SPICE 1 offered several improvements over CANCER. JEFT and MOSFET devices were added. It also offered a new approach to modeling called macromodeling.

With macromodel, designer could describe portion of a circuit in the form of relocatable circuit templates (Subcircuits). During this time the rapid development of IC industry explored the work on SPICE. Many IC problems could not be examined with traditional design techniques.

Because of this computer simulation of integrated circuits proved to be an invaluable design tool.

The next release of program came in 1975 with the introduction of SPICE 2 SPICE 2 offer new formulation for voltage defined elements. In SPICE 2 accuracy and speed of transient analysis was improved. From 1975 to 1983, Berkeley continued improving and upgrading the SPICE 2 program. In 1983 SPICE 2 version G6 was released. It was the last FORTRAN version of SPICE.

With increasing use of Unix based workstations, Berkeley made decision to rewrite the SPICE 2 program in C. The new C version of program was known as SPICE 3. SPICE 3 includes improved device models, voltage and current controlled switches, pole zero analysis and graphical postprocessor for viewing simulation results. The SPICE 3 written in modular C code is easier to modify than the SPICE 2 written in FORTRAN. It demonstrates superior convergence characteristics. There are several vendors offered versions of SPICE.

Meta Software	-	HSPICE
Intu Soft	-	IS_SPICE
Spectrum Software's	-	MICRO-CAP
Microsims	-	PSPICE

1.6 NEED OF SIMULATION :-

Simulation does not replace the breadboard, but simulation complements the breadboard. Simulation may reveal many things not readily learned in the lab. There are various reasons for simulating circuit design.

i) Verify design theories -

Simulation offers the ability to quickly test circuit design theories before a single wire is soldered. Verifying the theories of design can be done at different levels behavioural models, macromodels and circuit component models. Behavioural model express circuit blocks in mathematical relationship. Behavioral blocks are easy to construct and simulate hundreds or thousands of time faster than circuit or component level. Macromodels express circuit blocks in the form of simplified equivalent circuits. Macromodel circuit element may include real circuit components and ideal circuit components. Macromodel simulates slower than behavioral model but much faster than circuit or component level model. Macromodel express circuit blocks in the form of simplified equivalent circuit. But because of their simplified representation, macromodel does not give the accuracy of a transistor level simulation. Components level simulation is a simulation of the components in a circuit. This type of simulation offers the most accurate analysis but requires a long run time. A component level simulation represents lowest level of the simulation.

ii) Circuit performance and yield analysis tool :-

Simulation allows the designer to quickly test a circuit over a variety of conditions including temperature variations; element value variations and power supply variations. The ability to alter circuit parameters offers designer a fast, efficient means of testing the circuit operations under variety of operating conditions. If the yield of a circuit depends on the component value variations, simulation may be useful in estimating circuit yields.

Multiple simulation is possible using circuit simulator. It offers user a fast, efficient means to test a circuit performance characteristic as one or more component value change SPICE package now offer Monte Carlo and worst case analysis. Monte Carlo & Worst case analysis automatically vary circuit components as multiple simulations are performed. These analysis enhance a users ability to measure circuit performance over a number of changing circuit parameters.

iii) Simulation allows a designer to examine the circuit without the risk of damage to the circuit or the designer.

iv) As a failure analysis tool, circuit simulation has limits. The most obvious limit is the lack of ability to predict layout dependent parasitic behaviors.

1.7 REQUIREMENT OF SIMULATION :-

Simulation requires a text editor to create the input files and a simulation program.

A minimum machine recommendation is 386 PC. Accurate simulation requires accurate component models and accurate device model parameters. One important feature of SPICE is a library of common electrical components. Model of BJT, transistor, diode operational Amplifier and thousands of other devices can be stored in a library and called into wherever necessary from eval.lib file. Designer can develop his own device models.

Without a good simulator, accurate device models and good user skill, simulation results are simply imaginary numbers.

1.8 LIMITATIONS OF PSPICE :-

Circuit simulator Pspice has the following limitations.

i) Student version of Pspice is restricted to circuit with 10 transistor, professional DOS version has a limit of 200 BJT, 150 MOFET.

ii) Program is not interactive that is circuit can not be analyzed for various component values without editing the program statements.

iii) Pspice does not support iterative method of solution i.e. if the elements of circuit are specified the output can be predicted, on the other hand if the output is specified Pspice can not be used to synthesize the circuit elements.

iv) Input impedance can not be determined directly without running the probe, the graphical coprocessor.

v) Distortion analysis is not available in Pspice.

vi) Output impedance of circuit can not be printed or plotted graphically.