

# **DESIGN AND ANALYSIS OF DIFFERENT ORDERS OF ACTIVE-RC BUTTERWORTH FILTER**

*Thesis report submitted towards the partial fulfillment of  
requirements for the award of the degree of*

**Master of Technology (VLSI Design & CAD)**

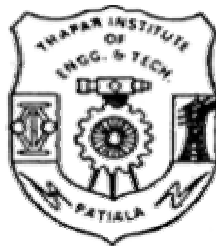
Submitted by

**Nimisha Saini  
Roll No 6040411**

Under the Guidance of

**Mrs. Alpana Agarwal  
Asth. Professor, ECED**

**Mr. Rajesh Khanna  
Asth. Professor, ECED**



**Department Of Electronics and Communication Engineering  
THAPAR INSTITUTE OF ENGINEERING & TECHNOLOGY,  
(Deemed University),  
PATIALA – 147004, INDIA.**

**JUNE, 2006**

## ***Declaration***

I hereby certify that the work which is being presented in the thesis entitled, **“Design and Analysis of Different Orders of Active-RC Butterworth Filter”** in partial fulfillment of the requirements for the award of degree of M.Tech. (VLSI Design And CAD) at Electronics and Communication Department of Thapar Institute of Engineering and Technology (Deemed University), Patiala, is an authentic record of my own work carried out under the supervision of Mrs. Alpana Agarwal, Assistant Professor, ECED (Guide) and Mr. Rajesh Khanna, Assistant Professor, ECED (Co-guide).

The matter presented in this thesis has not been submitted in any other University/ Institute for the award of any degree.

Date: -----

**Nimisha Saini**  
**Roll. No. 6040411**

It is certified that the above statement made by the student is correct to the best of my knowledge and belief.

***Alpana Agarwal***  
***Khanna***

**Guide**

***Rajesh***

**Co-guide**

**Counter Signed By:**

**Head**  
**Electronics and Communication**  
**Engineering Department,**  
**T.I.E.T., Patiala.**

**Dean of Academic Affairs,**  
**T.I.E.T., Patiala.**

## **ACKNOWLEDGEMENT**

To discover, analyze and to present something new is to venture on an untrodden path towards an unexplored destination is an arduous adventure unless one gets a true torchbearer to show the way. I would have never succeeded in completing my task without the cooperation, encouragement and help provided to me by various people. This enlightening guidance, I found in my revered guide Mrs. Alpana Agarwal, Assistant Professor, Electronics & Communication Engineering Department, Thapar Institute of Engineering & Technology (Deemed University), Patiala, without whose patronization it was never possible to give final shape to this thesis. I wish to express my deep gratitude towards her for providing individual guidance and support throughout the thesis work.

I express my heartfelt gratitude towards my co-guide Mr. Rajesh Khanna, Assistant Professor, Electronics and Communication Department, TIET (Deemed University) for his valuable guidance, encouragement, constant involvement, inspiration and the enthusiasm with which he solved my difficulties.

I shall be failing in my duties if I do not express my deep sense of gratitude towards Dr. R. S. Kaler, Professor & Head of the Department, Electronics & Communication Engineering Department and Dr. A.K.Chatterjee , P.G. Coordinator, Electronics and Communication Engineering Department.

I am also thankful to Mr. Sanjay Batish and Mrs. Sushma Jain for extending their help in the VLSI Laboratory.

I would also like to thank all the staff members and my co-students who were always there at the need of the hour and provided with all the help and facilities, which I required for the completion of my thesis. I am also thankful to the authors whose works I have consulted and quoted in this work.

My greatest thanks are to all who wished me success especially my parents. Above all I render my gratitude to the Almighty who bestowed self-confidence, ability and strength in me to complete this work for not letting me down at the time of crisis and showing me the silver lining in the dark clouds.

**Nimisha Saini**

## ABSTRACT

Recently, developments in wireless communication have motivated many design challenges. As a critical block of wireless devices, baseband filters operating with kilohertz to a few megahertz cut-off frequencies significantly contribute to the overall power consumption of receivers. One basic circuit used in these systems is a filter.

All the filters are designed in a different way depending upon the application for which they are being designed. Filters can be classified on the basis of response characteristics, components used, mathematical approximations or digital and analog filters. On the basis of mathematical approximation one type i.e. Butterworth filter is the most popular and widely used filter. Active-RC Butterworth filter is a basic filter that is widely used for anti-aliasing and for channel filtering. For lower order of filters this filter is very commonly used.

In Active-RC filter, an active device has to be chosen for a desirable response of the filter. There is a range of devices like Operational Amplifier, Difference Differential Amplifier, Operational Transconductor Amplifier etc. can be used to design an Active-RC filter. After extensive study, of all the choices Operational Transconductor Amplifier is chosen to design an active-RC Butterworth filter. Amongst all the topologies of OTA (Operational Transconductor Amplifier), on the basis of literature survey, Telescopic OTA, is chosen. The design procedure for a Single stage Telescopic OTA is developed using design equations. The circuit designed is then simulated on Tanner EDA tool. The simulated results are validating the theoretical values.

The Active-RC Butterworth filter design steps are developed, beginning right from the selection of active device to the final filter response. The result of the 6<sup>th</sup> order active-RC Butterworth filter schematic is then compared with the layout drawn on Tanner EDA tool. The passband frequency response achieved after simulation well-justifies that the filter may be used for video frequency applications.

With the increase in filter order many parameters change. Few of them are calculated and observed by designing different orders of filters and analyzing them. Analysis is done in terms of change in roll-off rates, passband frequency achieved and the power dissipation. The results of the analysis are in accordance with the literature.

# Table Of Contents

<b>Declaration.....</b>	<b>i</b>
<b>Acknowledgements.....</b>	<b>ii</b>
<b>Abstract.....</b>	<b>iii</b>
<b>List Of Figures.....</b>	<b>viii</b>
<b>List Of Tables.....</b>	<b>xii</b>

## **Chapter 1**

### **INTRODUCTION.....1-4**

1.1 Objective of the thesis work.....	1
1.2 Abbreviations and Colour schemes.....	1
1.3 Contribution and Organization of thesis.....	3
1.4 Software and Tool used.....	4

### *Chapter 2 DIFFERENT METHODS FOR DESIGNING ANALOG CMOS FILTERS.....5-14*

2.1 Introduction.....	5
2.2 Filter design using OTA.....	5
2.2.1 CMOS Implementation of Transconductor.....	7
2.3 DDA Based Fully Differential Sallen Key filter.....	9
2.4 Active-RC Filters.....	11
2.4.1 Channel Filtering: using Butterworth Filter.....	12
2.5 Comparison.....	14

## **Chapter 3 FILTER THEORY AND BUTTERWORTH FILTER.....15-37**

3.1 Introduction.....	15
3.2 Response Categories of Filters.....	15
3.2.1 The Bode and Phase plots.....	17
3.2.2 Representation of Poles and Zeros on graph.....	19

3.3 Filter Types and Approximations.....	20
3.4 Analog Vs Digital Filters.....	24
3.5 Different Approaches of Implementing Filters.....	25
3.6 Butterworth Filter.....	27
3.6.1 Butterworth Filter Properties.....	28
3.6.2 First Order Low Pass Butterworth Filter.....	31
3.6.3 First Order High Pass Butterworth Filter.....	32
3.6.4 Second Order Filters.....	34
3.6.5 Second Order Low Pass Butterworth Filter.....	35
3.6.6 Second Order High Pass Butterworth Filter.....	36
<b>Chapter 4 INTRODUCTION TO TANNER EDA TOOL.....</b>	<b>38-</b>
<b>48</b>	
4.1 Introduction.....	38
4.2 S-EDIT (Schematic Edit).....	38
4.3 T-EDIT (Simulation Edit).....	39
4.3.1 DC Operating point Analysis.....	40
4.3.2 DC Transfer Analysis.....	41
4.3.3 Transient Analysis.....	42
4.3.4 AC Analysis.....	42
4.3.5 Noise Analysis.....	43
4.4 W-EDIT (Waveform Edit).....	44
4.5 L-EDIT (Layout Edit).....	45
4.5.1 L-Edit: An Integrated Circuit Layout Tool.....	45
<b>Chapter 5 TELESCOPIC OTA: DESIGN, SIMULATION AND SYNTHESIS....</b>	<b>.....49-76</b>
5.1 Introduction.....	49
5.2 Operational Amplifier.....	49
5.2.1 Performance Parameters.....	50
5.2.2 Opamp Topologies.....	51

5.3 OTA (Operational Transconductance Amplifier).....	53
5.4 Telescopic OTA.....	55
5.4.1 Frequency Compensation.....	58
5.4.2 Differential Stage.....	59
<b>5.4.3 Biasing circuit for</b>	
<b>OTA.....</b>	<b>60</b>
5.5 Design of Telescopic OTA.....	62
5.5.1 Steps to design Telescopic OTA.....	62
5.5.2 Design of Telescopic OTA with defined specifications.....	64
5.5.3 Simulation results of OTA.....	67
5.5.4 Analysis of the Simulation Results.....	74
5.5.5 Physical Design of Telescopic OTA.....	74
<b>Chapter 6 ACTIVE-RC BUTTERWORTH FILTERS: DESIGN AND</b>	
<b>ANALYSIS.....</b>	<b>77-103</b>
6.1 Introduction.....	77
6.2 Active-RC Butterworth Filter.....	77
6.3 Design steps to design Active-RC Butterworth filter.....	79
6.4 Design of 6 <sup>th</sup> order Active- RC Butterworth filter.....	82
6.5 Simulation results of the 6 <sup>th</sup> order Butterworth filter.....	87
6.5.1 Analysis of the simulation results.....	90
6.5.2 Physical design of the 6 <sup>th</sup> order Active-RC Butterworth filter.....	90
6.6 Analysis of different orders of Active-RC Butterworth Filter.....	93
<b>Chapter 7 CONCLUSION AND FUTURE</b>	
<b>SCOPE.....</b>	<b>104-105</b>
7.1 Conclusion.....	104
7.2 Future Scope.....	105

<b>REFERENCES.....</b>	<b>106-108</b>
<b>APPENDIX.....</b>	<b>109-113</b>

## List Of Figures

<b>Figure No.</b>	<b>Title of Figure</b>	<b>Page No.</b>
2.1	4 <sup>th</sup> order Butterworth LPF using OTA integrators	7
2.2	The CMOS implementation of the transconductor	8
2.3	Block diagram of DDA	9
2.4	DDA based fully Differential Sallen-Key Filter	10
2.5	5 <sup>th</sup> order low pass filter	11
2.6	Frequency response of the 5 <sup>th</sup> order filter	11
2.7	3 <sup>rd</sup> order Butterworth filter	13
3.1	Filter responses	16
3.2	Filter responses on Bode plots	17
3.3	Showing the center frequency in a band pass response	19
3.4	Showing pole-zero diagram for the example	20
3.5	Response of a 4 <sup>th</sup> order Butterworth lowpass (upper curve) to a square wave input (lower curve)	21
3.6	Response of a 4 <sup>th</sup> order Bessel low-pass (upper curve) to a square wave input (lower curve)	22
3.7	Amplitude response curves for Bessel filters of various orders	22
3.8	Step responses for Bessel low-pass filters	23
3.9	Example of an elliptic low-pass amplitude response	24
3.10	First Order Low Pass Butterworth Filter	31
3.11	First Order High Pass Butterworth Filter	33
3.12	Second Order Low Pass Responses	34
3.13	Second Order Low Pass Butterworth Filter	35
3.14	Second Order High Pass Butterworth Filter	36
3.15	Frequency Response of Second Order High Pass Filter	37
5.1(a)	Telescopic opamp	52

<b>Figure No.</b>	<b>Title of Figure</b>	<b>Page No.</b>
5.1(b)	Folded cascode opamp	52
5.1(c)	Two stage opamp	53
5.2	Ideal OTA	54
5.3	A conventional telescopic opamp	56
5.4	Telescopic opamp single ended output with constant current tail	58
5.5	Differential input pair telescopic OTA	59
5.6(a)	Biasing circuit with diode connected load	61
5.6(b)	Biasing circuit with gate grounded load	61
5.7	Schematic of the Telescopic OTA with constant current source at the tail	68
5.8	Frequency response of the Telescopic OTA with constant current source at its tail	69
5.9	Phase response of the Telescopic OTA with constant current source at its tail	70
5.10	Schematic of the Telescopic OTA with bias circuit at the tail	71
5.11	Frequency response of the Telescopic OTA with bias circuit at its tail	72
5.12	Phase response of the Telescopic OTA with bias circuit at its tail	73
5.13	Layout of Telescopic OTA	75
5.14	Frequency response of layout of Telescopic OTA	76
6.1	1 <sup>st</sup> order opamp based Active-RC filter	78
6.2	Low pass filter response	79
6.3	Graphical construction for determining the poles of a Butterworth filter of order N	81

<b>Figure No.</b>	<b>Title of Figure</b>	<b>Page No.</b>
6.4(a)	OTA symbol	83
6.4(b)	Equivalent circuit of ideal OTA	83
6.5	Schematic of the 6 <sup>th</sup> order Active-RC Butterworth filter	88
6.6	Frequency response of the 6 <sup>th</sup> order low-pass Active-RC Butterworth filter	89
6.7	Layout of the 6 <sup>th</sup> order Active-RC Butterworth filter	91
6.8	Frequency response of the 6 <sup>th</sup> order Active-RC Butterworth Filter	92
6.9(a)	Schematic of the 1 <sup>st</sup> order Active-RC Butterworth filter	94
6.9(b)	Frequency response of the 1 <sup>st</sup> order Active-RC Butterworth Filter	94
6.10(a)	Schematic of the 2 <sup>nd</sup> order Active-RC Butterworth filter	95
6.10(b)	Frequency response of the 2 <sup>nd</sup> order Active-RC Butterworth Filter	95
6.11(a)	Schematic of the 2 <sup>nd</sup> order Active-RC Butterworth filter	96
6.11(b)	Frequency response of the 2 <sup>nd</sup> order Active-RC Butterworth Filter	96
6.12(a)	Schematic of the 3 <sup>rd</sup> order Active-RC Butterworth filter	97
6.12(b)	Frequency response of the 2 <sup>nd</sup> order Active-RC Butterworth Filter	97
6.13(a)	Schematic of the 3 <sup>rd</sup> order Active-RC Butterworth filter	98
6.13(b)	Frequency response of the 3 <sup>rd</sup> order Active-RC Butterworth Filter	98
6.14	Comparison of roll-off rates of simulated and theoretical values	100

<b>Figure No.</b>	<b>Title of Figure</b>	<b>Page No.</b>
6.15	Comparison of roll-off rates of simulated and theoretical values	101
6.16	Increase in power dissipation with increase in filter order	102
6.17	Variation in simulated values from calculated values	103

## List Of Tables

<b>Table No.</b>	<b>Title of table</b>	<b>Page No.</b>
1.1	Colour scheme used for layout design	3
5.1	Comparison of performance of various opamp topologies	53
5.2	Calculated Aspect ratios for the telescopic OTA	67
5.3	Node name correspondence of fig 5.3 with fig 5.7	68
6.1	Ratios of $R_F/R_1$ for different stages in different orders of filters	93
6.2	Change in filter parameters with increase in filter order	99

# Chapter 1

## INTRODUCTION

### 1.1 Objective of the thesis work

In recent years, the broadband communication systems, such as digital video, wireless telephony and computer networking has been a strong driving force for the IC Technology. Filters constitute a small part of a complete communication system. So, they need to be low in power, small in size and they must not limit the performance of the overall system.

Active RC filters are used for anti-aliasing and for channel filtering in many communication systems. This thesis is a step forward to this rapidly growing communication systems field.

The objective of the work was to develop the design steps for an active-RC Butterworth filter, for this to choose and design an active device and then to analyze that how increase in filter order affects different parameters like roll-off rate, power dissipation and passband frequency.

### 1.2 Abbreviations and colour schemes


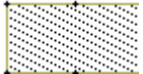

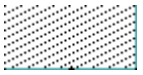





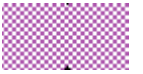

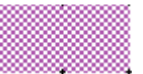
The abbreviations used in this report are listed below followed by the colour scheme used while drawing layout in Table 1.1:

AC	Alternating Current
CAD	Computer Aided Design
CMOS	Complementary Metal Oxide Semiconductor
dB	Decibel
DC	Direct Current
DDA	Difference Differential Amplifier
Dec	Decade
EDA	Electronic Design Automation

Hz	Hertz
IC	Integrated Circuit
LPF	Low Pass Filter
MHz	Mega Hertz
NMOS	N-channel Metal Oxide Semiconductor
Op-Amp	Operational Amplifier
OTA	Operational Transconductance Amplifier
PMOS	P-channel Metal Oxide Semiconductor
SPICE	Simulation Program with Integrated Circuit Emphasis
UGB	Unity Gain Bandwidth
W	Width of transistor
L	Length of transistor

Table 1.1 Colour scheme used for layout design

Colour scheme	Layer	Colour scheme	<b>Layer</b>
---------------	-------	---------------	--------------

	Poly		N Well
	Active		N Select
	Metal1		P Select
	Metal2		Via
	Poly Contact		Overglass
	Active Contact		Pad Comment

### 1.3 Contribution and Organization of Thesis

Main contributions of this thesis are:

- Study of different methods to design an analog CMOS filter.
- Design, simulation and synthesis of Telescopic OTA.
- Design of 6<sup>th</sup> order Active-RC Butterworth filter.
- Analysis of Active-RC Butterworth filters in terms of roll-off rates, passband frequency and power dissipation with increase of filter order.

The thesis report is organized into eight chapters. The chapterwise detail is given below:

Chapter 1: In this chapter, the objective of the thesis work is presented along with the abbreviations used in the report and the colour scheme followed during the layout.

Chapter 2: This chapter is based on the literature survey done for this thesis.

Chapter 3: In this chapter, basic theory of filters and its types are discussed.

Chapter 4: In this chapter, brief introduction to the Tanner EDA tool is presented.

Chapter 5: This chapter discusses the design steps for the design of Telescopic OTA with results of the simulations.

Chapter 6: This chapter discusses the design steps for the design of Active-RC Butterworth filter with results of simulations for 6<sup>th</sup> order Active-RC Butterworth filter and analysis of different orders of Active-RC Butterworth filter.

Chapter 7: This chapter presents the conclusion of the work done and future scope.

## **1.4 Operating System and EDA Tools used**

The operating system and EDA Tools used in thesis are as follows:

Operating System	: Windows XP
Tool	: Tanner EDA Tool
	S-Edit
	T-Spice Pro
	W-Edit
	L-Edit Pro

## Chapter 2

# DIFFERENT METHODS FOR DESIGNING ANALOG CMOS FILTERS

### 2.1 Introduction

Recently, developments in wireless communication and biomedical equipments have motivated many design challenges specifically in low voltage and low power integrated circuits. As a critical block of portable wireless devices, baseband filters operating with kilohertz to a few megahertz cutoff frequencies significantly contribute the overall power consumption of receivers. One basic circuit used in these equipments is filters. Filters can be classified on the basis of mathematical approximations, frequency response, elements used in circuit (active and passive) etc.

All the filters are designed differently depending on application for which they are designed. Since, filters generally constitute only a small fraction of a complete communication system, they need to be low in power and small in size to accommodate the integration of extensive signal-processing functions on the same integrated circuit (IC). To achieve the desired characteristics, filter can be designed by various methods.

On the basis of mathematical approximations, the filters are of four types- Butterworth, Bessel, Chebyshev and Elliptic filter. The first and probably best-known filter approximation is the Butterworth or maximally flat response because of its smooth band pass with no ripple. Due to this feature these filters are highly used in communication filters [19].

### 2.2 Filter Design using OTA

Filter design using OTA (Operational Transconductance Amplifier) is the most common method. Butterworth, Chebyshev, Bessel and Elliptic filters can be designed easily and

efficiently using OTAs. An OTA is a voltage controlled current source (VCCS), more specifically the term “operational” comes from the fact that it takes the difference of two voltages as the input for the current conversion. In addition to the voltage control characteristics, the OTA based circuits show promise for high-frequency applications where conventional opamp based circuits become bandwidth limited.

*High-speed filters are increasingly based on an Operational Transconductance Amplifiers (OTAs) Capacitor ( $G_m$ -C) approach, because of their simple circuitry and improved frequency response. However, the nonidealities of the transconductor limit overall filter performance. Several linearization techniques [16, 29] (one of the techniques utilizes triode-biased transconductors exploiting an active-cascode scheme which is similar to conventional cascode circuits except that the current of the triode transistor is sensed through another path rather than the cascode transistor, allowing for regulation of the drain voltage by negative feedback.) have been developed to enhance the linearity of the transconductor, but precise cancellation of nonlinearities is limited by matching accuracy.*

The performance of OTA-C filters depends on: (i) the OTA circuit, which is the main noise and distortion contributor in the filter, and (ii) the OTA-C filter structure. The later is a fourth-order Butterworth, realized by cascading two biquadratic sections. Each biquad is implemented with 4 OTAs and 2 capacitors. The fully differential transconductor- capacitor filter is shown in fig. 2.1. All transconductors are identical. The designed stop- band attenuation is 60 dB at 80 MHz for a 10 MHz cutoff frequency [7].

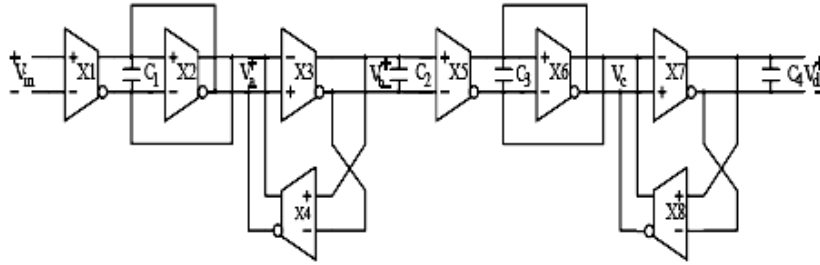


Figure 2.1. 4th-order Butterworth LPF using OTA integrators:  $C_1 = 6.40\text{pF}$ ,  $C_2 = 3.84\text{pF}$ ,  $C_3=2.40\text{pF}$  and  $C_4 = 9.60\text{pF}$ .

### 2.2.1 CMOS Implementation of Transconductor

The differential transconductance circuit is shown in fig. 2.2. It comprises a source-coupled pair with poly-silicon degeneration resistors and a cross-coupled high-impedance load. The linearity is enhanced by the degenerated resistors at the cost of a smaller effective transconductance,  $G_m$ , and a smaller tunable  $G_m$  range. Transistors M4, M5, M6 and M7 are matched. M8 and M9 operate in the triode region, acting as degeneration resistors to provide different operating points at nodes  $v_{pg}$  and  $v_{ng}$ . M4 and M7 act like a pair of positive resistors  $R_+$ , while M5 and M6 function as negative resistors  $R_-$ . The output impedance of the transconductor depends on the parallel combination of  $R_+$  and  $R_-$ , the values of which are controlled by voltages  $v_{cp}$  and  $v_{cn}$  respectively. As a result, the output impedance, and therefore, the Q of the integrator, can be maximized with proper combinations of  $v_{cp}$  and  $v_{cn}$ . With this transconductor, a tunable integrator for very-high-frequency integrated filters can be realized by adjusting the voltage  $v_{ba}$ , which controls the tail current, and thus  $G_m$ . Good high-speed properties stem from the absence of internal high impedance nodes, which pushes non-dominant poles to the gigahertz ranges [6].



Increasing demand for high-speed data services in recent years has led to the evolution of Digital Subscriber Line (DSL) technology. It is necessary for DSL networks to design spectral shaping filters, that shape the transmit signal such that it complies with the time and frequency templates allowed in the network. For this application, Butterworth filter is designed using OTAs so as to decrease the power consumption [8].

### 2.3 DDA Based Fully-Differential Sallen-Key filter

In recent years the portable communication market has been a strong driving force for the IC technology. With the emergence of the third-generation (3G) mobile systems, high-bit-rate multimedia services are turning into reality. One concern with wideband CDMA receiver is the power consumption. With up to 20 MHz bandwidth, the power of baseband filter will be huge if proper power optimization techniques are not taken. The portable applications and the desire to extend the battery lifetime had made the low power design a must. Another concern is to reduce noise figure.

After it was presented by Edward Sackinger and Walter Guggenbuhl in 1987 [8], the CMOS Differential Difference Amplifier (DDA) has gained wide applications in analog signal processing. As shown in fig. 2.3, the DDA is actually an extension to the concept of the opamp. The main difference is that DDA has four inputs ( $V_{pp}$ ,  $V_{pn}$ ,  $V_{nn}$  and  $V_{np}$ ) and the output of DDA can be written as:

$$V_o = V_{op} - V_{on} = A_o [(V_{pp} - V_{pn} - (V_{np} - V_{nn})] \dots\dots\dots(2.1)$$

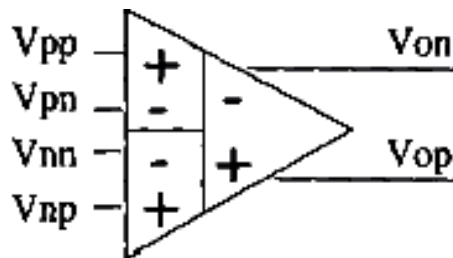


Figure 2.3. Block diagram of DDA

When a negative feedback is introduced, i.e., to  $V_{pn}$  and/or  $V_{np}$ , which appear in (2.1) with a negative sign, the following expression is obtained:

$$V_{pp} - V_{pn} = V_{np} - V_{nn} \text{ with } A_o \rightarrow \infty \dots\dots\dots(2.2)$$

In practice,  $A_o$  is always finite, so there is a difference between the two differential voltages. Therefore, the open-loop gain should be as large as possible in order to achieve high-performance operation.

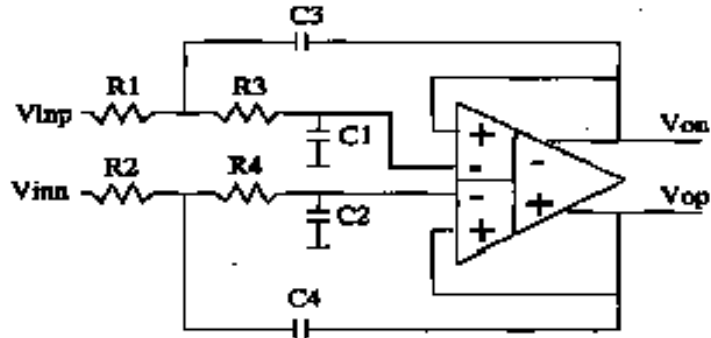


Figure 2.4. DDA based fully Differential Sallen-Key Filter

The DDA can be used to implement fully-differential Sallen- Key filter as shown in fig. 2.4 (where  $R_1 = R_2$ ,  $R_3 = R_4$ ,  $C_1 = C_2$ ,  $C_3 = C_4$ ). Assume  $A_o$  is large enough, then from (2.2) we have  $V_{pn} - V_{nn} = V_{pp} - V_{np} = V_{on} - V_{op}$ , and after some math manipulation, the transfer function of the filter can be derived as:

$$H(s) = (V_{on} - V_{op}) / (V_{inp} - V_{inn}) = \omega_0^2 / (s^2 + s\omega_0/Q + \omega_0^2) \dots\dots\dots(2.3)$$

where  $\omega_0 = \sqrt{1/R_1R_3C_1C_3}$  is the undamped natural frequency,

$Q = (R_1R_2C_1C_3) / [(R_1 + R_3)C_1]$  is the quality factor, which defines the sharpness at which the peak of the magnitude response occurs. When  $R_1 = R_3$ , this unity-gain filter has the advantage of low sensitivity to component value when compared to other Sallen-Key filter structures.

The fifth-order Butterworth low-pass filter is implemented by cascading a first-order RC network and two second-order sections, as shown in fig. 2.5. The DDA based Sallen-Key filter is used in 5<sup>th</sup> order Butterworth filter as a block, and the two second-order sections are cascaded in an order set by their Q-factors.

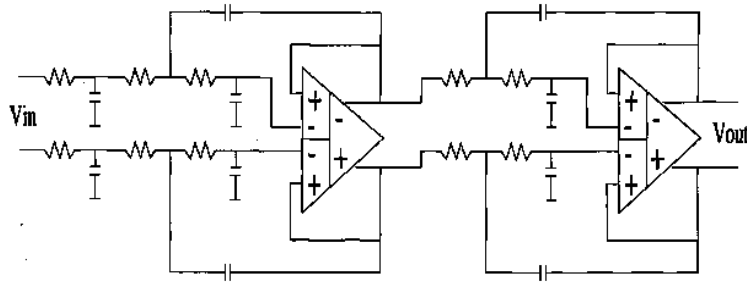


Figure 2.5. 5<sup>th</sup> order low pass filter

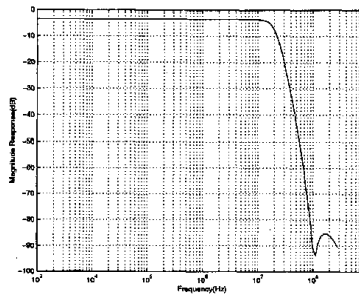


Figure 2.6 Frequency response of the 5<sup>th</sup> order filter

The filter is designed at 0.5  $\mu\text{m}$  with 19 MHz cut-off frequency with a power consumption of 4.5 mW [5].

## 2.4 Active-RC filters

A configurable W-CDMA Baseband Down-Link (BDL) channel suitable to be used with both direct conversion (DIR) and superheterodyne (SHT) architectures has been integrated in a double-poly 0.6  $\mu\text{m}$  CMOS technology. The BDL is an analog signal processor consisting of calibrated active-RC filters and over sampled 6-bit analog-to-digital pipeline converters, which sufficiently suppresses the adjacent channel and other interferers before digital signal processing on an external DSP. The designed BDL channel achieves 38.4 dB SNDR, 43 dB SFDR in DIR mode and 38.3 dB SNDR, 42.8 dB SFDR in SHT mode with an overall power consumption of only 49 mW/58 mW (DIR/SHT) from a single 2.8 V power supply. A low power consumption mode allows to

further reduce these values down to 41 mW/50 mW (DIR/SHT) without significant detrimental effects to BDL performance [17].

A Baseband Down-Link (BDL) channel, integrated in an analog baseband codec suitable for W-CDMA applications, is presented. The BDL architecture can be configured to work with both superheterodyne architectures (SHT) with no baseband filtering in the RF IC and direct conversion architectures (DIR) having all the baseband filtering already in the RF IC. The BDL channel is fully-differential (to increase the immunity to interference and suppress even order distortion) and consists of analog signal-processing circuitry, including internally calibrated active-RC filters and over sampled pipeline Analog-to-Digital Converters (ADCs). The programmable active-RC Butterworth filters perform the channel selection filtering and fit the input dynamic range of the ADCs, which are 6-bit 1.5 bit/stage pipeline structures with digital error correction and interstate gain [11].

#### **2.4.1 Channel Filtering: using Butterworth Filter**

The channel selection filtering is performed in the analog domain in order to use high-speed but only medium resolution ADCs. If channel selection is performed in the digital domain, high-resolution ADCs are required, which leads to large power consumption and large die size. The former approach is thus preferable since power consumption is a key concern in battery-operated products (in particular for W-CDMA applications). The channel filter, which is composed of cascaded 3<sup>rd</sup> order and 1<sup>st</sup> order active-RC Butterworth structures, can be configured for the two operation modes: SHT and DIR.

In the SHT mode, the filtering requirements are fulfilled by a 3<sup>rd</sup> order Butterworth filter with cut-off frequency at 1.92 MHz (fig. 2.7). An 18 dB DC gain is distributed in the filter to suppress the ADC quantization noise and to fit its input dynamic range (2.05 V<sub>pp</sub>). The measured integrated input-referred noise in SHT mode is 180 μV RMS. The in-band group-delay is 121ns with a group-delay deviation of 50 ns.

In the DIR mode, the 3<sup>rd</sup> order Butterworth filter is bypassed and sent off in order to save power, and only a simple 1<sup>st</sup> order smoothing filter with a cut-off frequency of 7.68 MHz

( $F_S/2$ , where  $F_S$  is the channel master clock) is present in front of the ADC. The programmable DC-gain of this filter can be set to 0dB or 5.8 dB depending on the full-scale ( $F_S$ ) of the input signal ( $2.05 V_{pp}/1.05 V_{pp}$ ). Thus, a calibration system becomes necessary to restore the nominal frequency response. This has been achieved by replacing the filter's capacitors by binary weighted capacitors arrays, which can be programmed by a 5-bit digital code generated on-chip by a FSM implementing a novel algorithm, which allows to restore the nominal frequency to  $\pm 3\%$  accuracy in less than  $5\mu s$  covering the whole RC-product variation range (patent pending). The opamp used in both the filters is a two-stage Miller compensated structure with NMOS input stage and NMOS output stage and has a nominal GBW of 80 MHz and a DC gain of 73 dB. Its current consumption is close to 530  $\mu A$  [17].

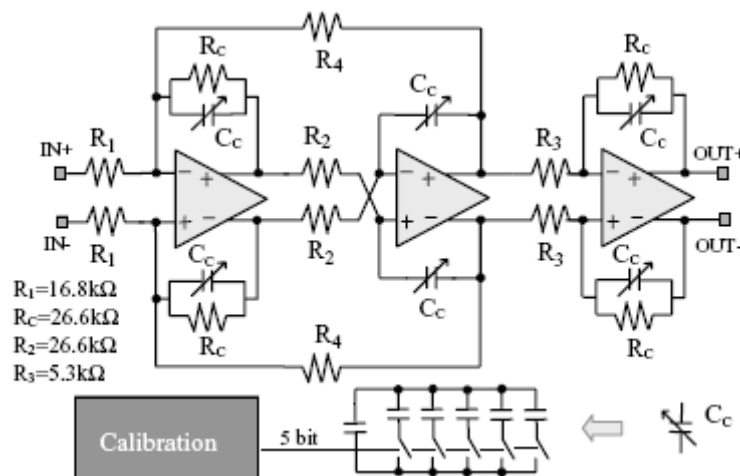


Figure 2.7 3rd order Butterworth filter

The advantages of the active-RC technique in filter's design are the insensitivity to parasitics, the excellent linearity and the large dynamic range. The main drawback of this approach are the large variations in the absolute value of the filter time constants, which can typically vary over a range of  $\pm 50\%$  from nominal values in CMOS technologies [11].

## 2.5 Comparison

Filter design using DDA, OTA and active-RC have been discussed in previous sections. Conventional OTAs do not achieve high dynamic range. Also, OTA-C filters are expected to lose more dynamic range as well as tuning range when power-supply voltage levels are further reduced. If two OTA-C filters are designed, one for low powers while the other was designed for low noise. The low-power filter exhibits a low dynamic range while the low-noise filter exhibits improved dynamic range but consumes excessive power.

If a 2<sup>nd</sup> order fully differential Sallen – Key filter is designed using OTAs then it uses two OTAs and an RC network. Whereas, one DDA along with RC network completes the same design of 2<sup>nd</sup> order Sallen-Key filter. This structure results in 50% power saving with the reduction in number of components and area as well.

In comparison to DDA and OTA based filter, opamp based Active-RC Butterworth filters given reasonably flat passband. It exhibits insensitivity to parasitics, excellent linearity and large dynamic range.

## Chapter 3

# FILTER THEORY AND BUTTERWORTH FILTER

### 3.1 Introduction

Filters of some sort are essential to the operation of most electronic circuits. In circuit theory, a filter is an electrical network that alters the amplitude and/or phase characteristics of a signal with respect to frequency. Ideally, a filter will not add new frequencies to the input signal, nor will it change the component frequencies of that signal, but it will change the relative amplitudes of the various frequency components and/or their phase relationships [27]. Filters are often used in electronic systems to emphasize signals in certain frequency ranges and reject signals in other frequency ranges. Such a filter has a gain that is dependent on signal frequency. A filter's pass band is the range of frequencies over which it will pass an incoming signal. Signal frequencies lying outside the pass band are attenuated. So, *a filter is defined as the electric networks that pass or allow unattenuated transmission of electric signal within certain frequency range and stops or disallows transmission of electric signal outside this range.*

Depending on various measures the filters can be classified in different categories (e.g. response, mathematical approximations, implementation and design approaches) [25].

### 3.2 Response categories of filters

Filters fall into one of the following response categories, based on the overall shape of their pass band.

*Low-pass filters* pass low-frequency signals while blocking high-frequency signals. The pass band ranges from DC (0 Hz) to a corner frequency  $F_C$ .

*High-pass filters* pass high-frequency signals while blocking DC and low-frequency signals. The pass band ranges from a corner frequency ( $F_C$ ) to infinity.

*Band-pass* filters pass only signals between two given frequencies, blocking lower and higher signals. The pass band lies between two frequencies,  $F_L$  and  $F_H$ . Signals between DC and  $F_L$  are blocked, as are signals from  $F_H$  to infinity. The pass band of these filters is often characterized as having a bandwidth that is symmetric around a center frequency.

*Band-stop filters or notch filters* block signals occurring between two given frequencies,  $F_L$  and  $F_H$ . The pass band is split into a low side (DC to  $F_L$ ) and a high side ( $F_H$  to infinity) [26].

*All pass or phase-shift filters* have no effect on the amplitude of the signal at different frequencies. Instead, its function is to change the phase of the signal without affecting its amplitude [26].

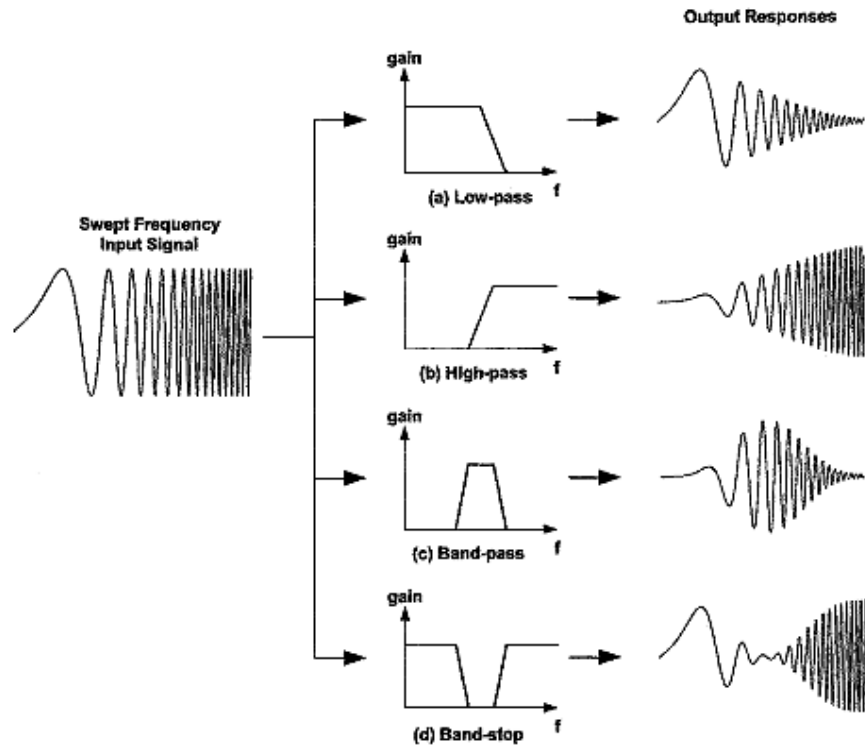


Figure 3.1. Filter responses.

### 3.2.1 The bode and phase plots

*Bode plots* describe the behavior of a filter by relating the magnitude of the filter's response (gain) to its frequency. An example of this type of plot is shown in fig. 3.2.

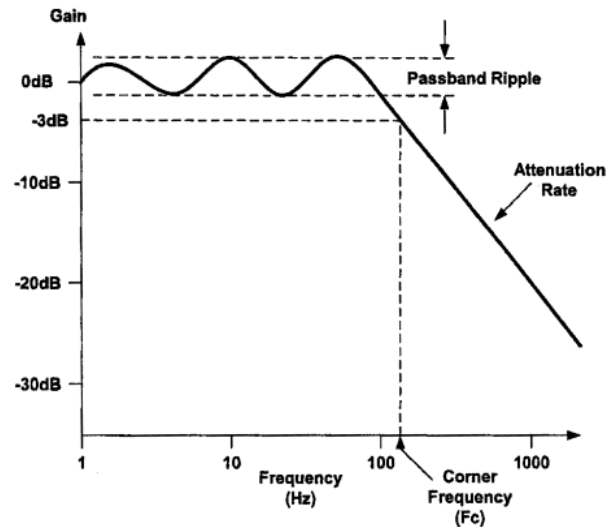


Figure 3.2. Filter responses on Bode plots.

The key feature of this graph is that both axes have logarithmic scales. The horizontal axis represents frequency, measured in hertz, and the vertical axis is measured in decibels.

Decibels are a logarithmic measure of power, where an increase of 10 dB represents a 10-fold increase in power. Because power in an electrical signal is related to the square of voltage, a factor of 10 increases in the voltage of a signal is represented by an increment of 20 dB. *The advantage of drawing a filter's response curve on a Bode plot is that it provides an easy way to describe the filter's response over several decades of frequency and several orders of magnitude* [26]. The decibels of gain of a filter relate to the ratio between input and output voltages:

$$dB = 20 \log \left[ \frac{V_{out}}{V_{in}} \right] \dots \dots \dots (3.1)$$

In addition to affecting the amplitude of a signal, a filter can also cause changes in the phase of signal components as follows:

- If you run sinusoidal signals of different frequencies through this filter, not only does the filter attenuate the higher frequency signals, but it also shifts their phase [25].

- The phase-shift plot describes the shift of sinusoidal signals. A filter's phase response is also important in the case of nonsinusoidal signals because phase shifts distort the waveforms. This is because nonsinusoidal signals can be viewed as combinations of sinusoidal signals of varying frequencies. Shifting the phase of these components relative to one another will change the shape of the overall waveform [26].

The precise shape of a band-pass filter's amplitude response curve will depend on the particular network, but any 2<sup>nd</sup> order band-pass response will have a peak value at the filter's center frequency. The center frequency is equal to the geometric mean of the -3 dB frequencies:

$$F_c = \sqrt{F_L F_H} \dots\dots\dots(3.2)$$

where  $F_c$  is the center frequency  $F_L$  is the lower -3 dB frequency  $F_H$  is the higher -3dB frequency Another quantity used to describe the performance of a filter is the filter's "Q". This is a measure of the "sharpness" of the amplitude response. The Q of a band-pass filter is the ratio of the center frequency to the difference between the -3dB frequencies (also known as the -3dB bandwidth) [26]. Therefore:

$$Q = \frac{F_c}{F_H - F_L} \dots\dots\dots(3.3)$$

See Figure 3.3.

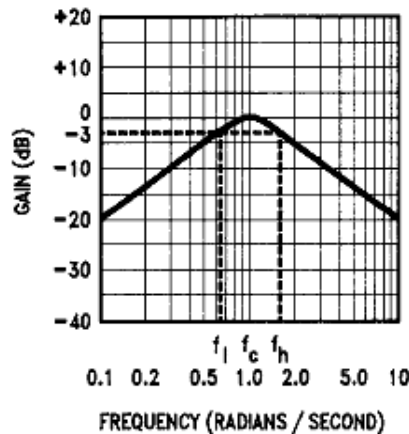


Figure 3.3 Showing the center frequency in a band pass response.

### 3.2.2 Representation of Poles and Zeros on Graph

The way of writing a filter's transfer function is to factor the polynomials in the numerator and denominator so that they take the form:

$$H(s) = H_0 \frac{(s - z_0)(s - z_1)(s - z_2) \dots (s - z_n)}{(s - p_0)(s - p_1)(s - p_2) \dots (s - p_n)} \dots \dots \dots (3.4)$$

The roots of the numerator,  $z_0, z_1, z_2, \dots, z_n$  are known as zeros, and the roots of the denominator,  $p_0, p_1, \dots, p_n$  are called poles.  $z_i$  and  $p_i$  are in general complex numbers, i.e.,  $R + jI$ , where  $R$  is the real part and  $I$  is the imaginary part. All of the poles and zeros will be either real roots (with no imaginary part) or complex conjugate pairs [27].

For example, the 2nd-order band pass network function can be factored to give:

$$H(s) = \frac{s}{(s + 0.5 + j\frac{\sqrt{3}}{2})(s + 0.5 - j\frac{\sqrt{3}}{2})} \dots \dots \dots (3.5)$$

The factored form of a network function can be depicted graphically in a pole-zero diagrams. Fig. 3.4 is the pole zero diagram for equation. The diagram shows the zero at the origin and the two poles, one at

$$s = -0.5 - j\sqrt{3}/2, \text{ and one at } s = -0.5 + j\sqrt{3}/2$$

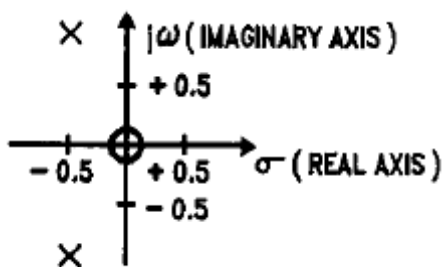


Figure 3.4. Showing pole-zero diagram for the example

### 3.3 Filter types and approximations

Filters with acceptable amplitude response curves may differ in terms of such characteristics as transient response, pass band and stop band flatness, and complexity. Fortunately for the circuit designer, a great deal of work has already been done in this area, and a number of standard filter characteristics have already been defined. These usually provide sufficient flexibility to solve the majority of filtering problems [27]. The "classic" filter functions were developed by mathematicians (most bear their inventors' names), and each was designed to optimize some filter property. The most widely used of these are discussed below.

**1. Butterworth:** The first and probably best-known filter approximation is the Butterworth or maximally-flat response. It exhibits a nearly flat pass band with no ripple. The roll off is smooth and monotonic, with a low-pass or high-pass roll off rate of 20 dB/decade (6 dB/octave) for every pole. Thus, a 5th-order Butterworth low-pass filter would have an attenuation rate of 100 dB for every factor of ten increases in frequency beyond the cutoff frequency [19].

**2 Chebyshev:** Another approximation to the ideal filter is the Chebyshev or equal ripple response. As the latter name implies, this sort of filter will have ripple in the pass band amplitude response. The amount of pass band ripple is one of the parameters used in specifying a Chebyshev filter. The Chebyshev characteristic has a steeper roll off near the cutoff frequency when compared to the Butterworth, but at the expense of monotonicity in the pass band and poorer transient response [12,27].

**3 Bessel:** All filters exhibit phase shift that varies with frequency. This is an expected and normal characteristic of filters, but in certain instances it can present problems. If the phase increases linearly with frequency, its effect is simply to delay the output signal by a constant time period. However, if the phase shift is not directly proportional to frequency, components of the input signal at one frequency will appear at the output shifted in phase (or time) with respect to other frequencies. The overall effect is to distort non-sinusoidal wave shapes, as illustrated in fig. 3.5 for a square wave passed through a Butterworth low-pass filter. The resulting waveform exhibits ringing and overshoot because the

square wave's component frequencies are shifted in time with respect to each other so that the resulting waveform is very different from the input square wave.

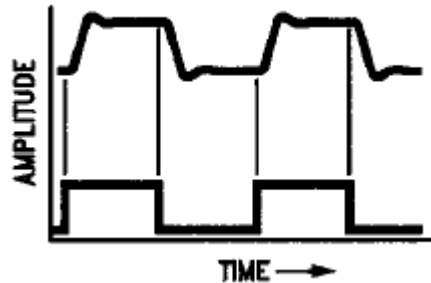


Figure 3.5 Response of a 4<sup>th</sup> order Butterworth lowpass (upper curve) to a square wave input (lower curve).

When the avoidance of this phenomenon is important, a Bessel or Thompson filter may be useful. The Bessel characteristic exhibits approximately linear phase shift with frequency, so its action within the pass band simulates a delay line with a low-pass characteristic. The higher the filter order, the more linear the Bessel's phase response. Fig. 3.6 shows the square-wave response of a Bessel low-pass filter. Note the lack of ringing and overshoot. Except for the "rounding off" of the square wave due to the attenuation of high-frequency harmonics, the wave shape is preserved.

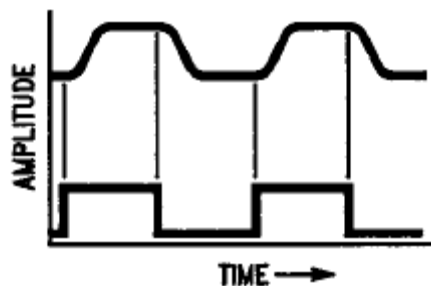


Figure 3.6 Response of a 4<sup>th</sup> order Bessel low-pass (upper curve) to a square wave input (lower curve).

The amplitude response of the Bessel filter is monotonic and smooth, but the Bessel filter's cutoff characteristic is quite gradual compared to either the Butterworth or

Chebyshev as can be seen from the Bessel low-pass amplitude response curves in fig. 3.7. Bessel step responses are plotted in fig.1.8 for orders ranging from 2 to 10 [27].

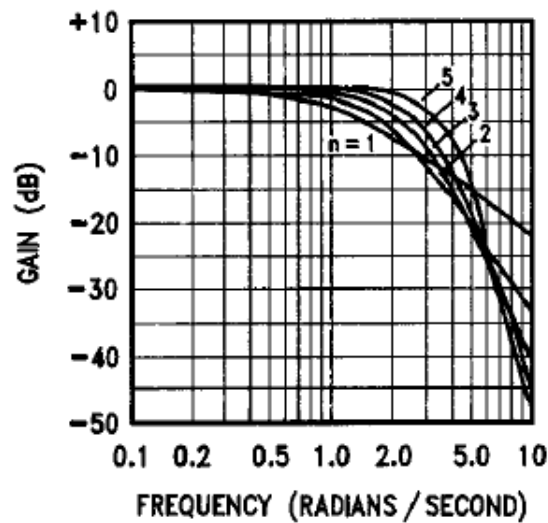


Figure 3.7. Amplitude response curves for Bessel filters of various orders.

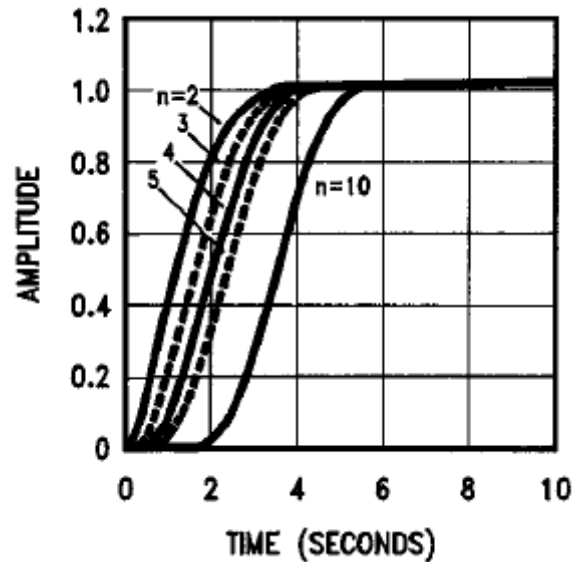


Figure 3.8 Step responses for Bessel low-pass filters.

4 **Elliptic:** The cutoff slope of an elliptic filter is steeper than that of a Butterworth, Chebyshev, or Bessel, but the amplitude response has ripple in both the pass band and the stop band, and the phase response is very non-linear. However, if the primary concern is to pass frequencies falling within a certain frequency band and reject frequencies outside

that band, regardless of phase shifts or ringing, the elliptic response will perform that function with the lowest-order filter. The elliptic function gives a sharp cutoff by adding notches in the stop band. These cause the transfer function to drop to zero at one or more frequencies in the stop band. Ripple is also introduced in the pass band. An elliptic filter function can be specified by three parameters (again excluding gain and cutoff frequency): pass band ripple, stop band attenuation, and filter order  $n$ . Because of the greater complexity of the elliptic filter, determination of coefficients is normally done with the aid of a computer [27, 28]. See fig. 3.9.

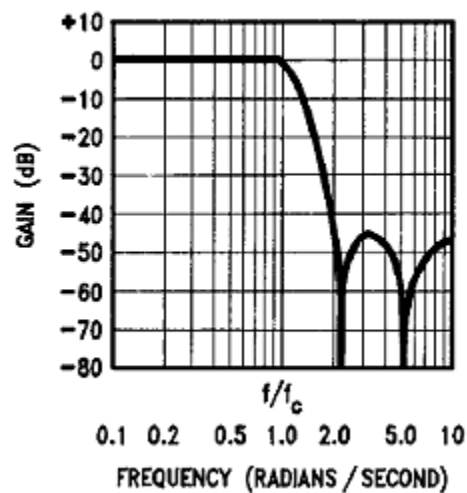


Figure 3.9  
Example of an elliptic low-pass amplitude response.

## 3.4 Analog Vs Digital Filters

### 1. Analog Filters

Analog filters are usually implemented with electronic circuits, making use of three fundamental components: resistors, capacitors and inductors. By arranging these components in a variety of configurations, it is possible to customize filter performance to very specific needs as follows:

- a) Operational amplifiers are commonly used to increase the performance of these filters [27].
- b) It is important to note that these filters are commonly used in “signal conditioning stages” before any digitization takes place. This may involve the removal of unwanted noise or the reduction of bandwidth to simplify further signal analysis or processing. The most notable application of this kind is low-pass filtering for anti-aliasing purposes [25].
- c) The performance of analog filters is directly related to the quality of the components used and the circuit design. Things such as component tolerances, power consumption, design techniques and often the physical size components all play important roles in establishing the practical limits of analog filters [27].

## 2. Digital Filters

The digitization of electric signals into sequences of numbers permits the complete manipulation of these signals to occur mathematically. Voltage signals that are expressed as numbers can easily be scaled through scalar multiplication or offset by adding constants; they can be rectified by using the absolute value operator or modulated with other signals through multiplication. The digital realm provides unbounded opportunities for condition and processing of the signal. This branch of science is known as *digital signal processing* [12].

## 3.5 Different Approaches of Implementing Filters

**1. Passive Filters:** The filters used for the earlier examples were all made up of passive components: resistors, capacitors, and inductors, so they are referred to as passive filters. A passive filter is simply a filter that uses no amplifying elements (transistors, operational amplifiers, etc.). In this respect, it is the simplest (in terms of the number of necessary components) implementation of a given transfer function. Passive filters have other advantages as well:

- a) Because they have no active components, passive filters require no power supplies.

- b) Since they are not restricted by the bandwidth limitations of op amps, they can work well at very high frequencies.
- c) They can be used in applications involving larger current or voltage levels than can be handled by active devices.
- d) Passive filters also generate little noise when compared with circuits using active gain elements. The noise that they produce is simply the thermal noise from the resistive components, and, with careful design, the amplitude of this noise can be very low [19, 27].

Passive filters have some important disadvantages in certain applications, however.

- a) Since they use no active elements, they cannot provide signal gain.
- b) Input impedances can be lower than desirable, and output impedances can be higher the optimum for some applications, so buffer amplifiers may be needed. Inductors are necessary for the synthesis of most useful passive filter characteristics.
- c) These can be prohibitively expensive if high accuracy (1% or 2%, for example), small physical size, or large values are required.
- d) Standard values of inductors are not very closely spaced, and it is difficult to find an off-the-shelf unit within 10% of any arbitrary value, so adjustable inductors are often used. Tuning these to the required values is time-consuming and expensive when producing large quantities of filters.
- e) Furthermore, complex passive filters (higher than 2<sup>nd</sup> order) can be difficult and time-consuming to design [19, 27].

**2. Active Filters:** Active filters use amplifying elements, especially op amps, with resistors and capacitors in their feedback loops, to synthesize the desired filter characteristics. These filters have advantages as follows:

- a) Active filters can have high input impedance, low output impedance, and virtually any arbitrary gain.
- b) They are also usually easier to design than passive filters.

- c) Possibly their most important attribute is that they lack inductors, thereby reducing the problems associated with those components.
- d) Still, the problems of accuracy and value spacing also affect capacitors, although to a lesser degree.
- e) Performance at high frequencies is limited by the gain-bandwidth product of the amplifying elements, but within the amplifier's operating frequency range, the op amp-based active filter can achieve very good accuracy, provided that low-tolerance resistors and capacitors are used [19].

Some of the disadvantages of the filters are as follows:

- a) Active filters will generate noise due to the amplifying circuitry, but this can be minimized by the use of low-noise amplifiers and careful circuit design [27].

**3. Switched-capacitor filters:** These are clocked, sampled-data systems; the input signal is sampled at a high rate and is processed on a discrete-time, rather than continuous, basis. This is a fundamental difference between switched-capacitor filters and conventional active and passive filters, which are also referred to as "continuous time" filters. The operation of switched-capacitor filters is based on the ability of on-chip capacitors and MOS switches to simulate resistors. The values of these on-chip capacitors can be closely matched to other capacitors on the IC, resulting in integrated filters whose cutoff frequencies are proportional to, and determined only by, the external clock frequency. Now, these integrated filters are nearly always based on state-variable active filter topologies, so they are also active filters, but normal terminology reserves the name "active filter" for filters built using non-switched, or continuous, active filter techniques. The primary weakness of switched-capacitor filters is that they have more noise at their outputs both random noise and clock feed through - than standard active filter circuits [27].

### 3.6 Butterworth Filter

Active filters are mainly used in communication and signal processing circuits. They are also employed in a wide range of applications such as entertainment, medical electronics, etc. Most commonly used active filters:

1. Low pass filters
2. High pass filters
3. Band pass filters
4. Band reject filters.

Each of these filters can be built using op-amp as the active element and resistors and capacitors as the passive elements (frequency selective part). Better filter performance is obtained by employing op-amps with higher slew rates and higher gain-bandwidths. The filtering behavior of the circuit is best represented by the frequency response characteristics of the circuit, which shows the variation of the filter circuit gain with respect to operating frequency [24].

There are many filter approximations that have been developed for the filter design. One of the better used is the Butterworth approximation. Butterworth filter is a type of electronic filter characterized by having a maximally flat magnitude response, i.e., no amplitude ripple in the passband. [Contrast with Chebyshev] This circuit is based upon Butterworth functions (or Butterworth polynomials). [For the mathematically inclined, these polynomials represent a specialized solution to a general Mac Laurin series based upon Taylor series expansion. The filter was named after S. Butterworth, a British engineer who first described this response in his paper "On the Theory of Filter Amplifiers," Wireless Engineer, vol. 7, 1930, pp. 536-541. Eleven years later, V.D. Landon coined the phrase maximally flat in his paper "Cascade Amplifiers with Maximal Flatness," RCA Review, vol. 5, 1941, pp. 347-362 [10].

### 3.6.1 Butterworth filter properties

Important properties of Butterworth filter (these properties are applicable for all other filters with different responses):

1. **Filter Order:** The order of a filter is important for several reasons. It is directly related to the number of components in the filter, and therefore to its cost, its

physical size, and the complexity of the design task. Therefore, higher-order filters are more expensive, take up more space, and are more difficult to design. The primary advantage of a higher order filter is that it will have a steeper roll off slope than a similar lower-order filter [27].

2. **Ultimate Roll off Rate:** The roll off of *Butterworth filter* is smooth and monotonic. Usually expressed as the amount of attenuation in dB for a given ratio of frequencies. The most common units are "dB/octave" and "dB/decade". While the ultimate roll off rate will be 20 dB/decade for every filter pole in the case of a low-pass or high-pass filter and 40 dB/decade for every pair of poles for a band pass filter, some filters will have steeper attenuation slopes near the cutoff frequency than others of the same order [19].
3. **Attenuation Rate near the Cutoff Frequency:** If filter is intended to reject a signal very close in frequency to a signal that must be passed, a sharp cutoff characteristic is desirable between those two frequencies [27].
4. **Transient Response:** Curves of amplitude response show how a filter reacts to steady-state sinusoidal input signals. Since a real filter will have far more complex signals applied to its input terminals, it is often of interest to know how it will behave under transient conditions. An input signal consisting of a step function provides a good indication of this [27].
5. **Monotonicity:** A filter has a monotonic amplitude response if its gain slope never changes sign. In other words, if the gain always increases with increasing frequency or always decreases with increasing frequency. Obviously, this can happen only in the case of a low-pass or high-pass filter. A band pass or notch filter can be monotonic on either side of the center frequency, however [27].
6. **Passband Ripple:** *Butterworth filters* have a very flat pass band with no ripples. If a filter is not monotonic within its passband, the transfer function within the

passband will exhibit one or more "bumps". These bumps are known as "ripple". Some systems don't necessarily require monotonicity, but do require that the passband ripple be limited to some maximum value (usually 1 dB or less). Although bandpass and notch filters do not have monotonic transfer functions, they can be free of ripple within their passbands [19, 27].

7. **Stopband Ripple:** Some filter responses also have ripple in the stopbands. We are normally unconcerned about the amount of ripple in the stopband, as long as the signal to be rejected is sufficiently attenuated. Given that the "ideal" filter amplitude response curves are not physically realizable, we must choose an acceptable approximation to the ideal response. The word "acceptable" may have different meanings in different situations. The acceptability of a filter design will depend on many interrelated factors, including the amplitude response characteristics, transient response, the physical size of the circuit and the cost of implementing the design [19, 27].
  
8. **Pass Band Gain:** With active filters, it is possible to achieve a pass band gain higher than 1. Most active filters employ an amplifier, which determines the pass band gain of the filter. Filters with a flat pass-band gain are commonly used, and such a response is provided by *Butterworth filters*. Another class of filters called Chebyshev filters, provide a ripple (or overshoots in) pass-band gain [19].

### 3.6.2 First Order Low Pass Butterworth Filter

Fig. 3.10 shows a first-order low-pass filter that uses an RC network for filtering.

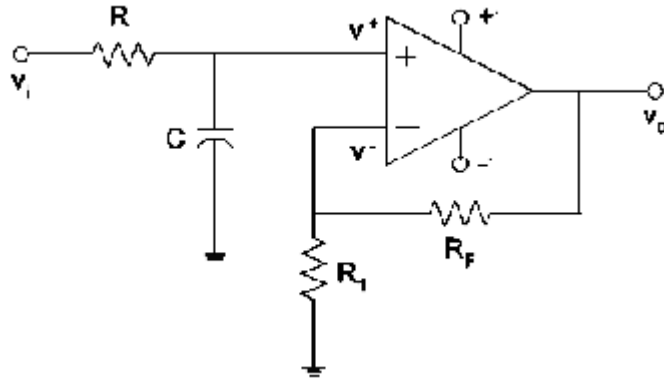


Figure 3.10. First Order Low Pass Butterworth Filter

### Derivation of Transfer Function

The RC network behaves as a voltage divider supplied by  $v_i$ , and hence the voltage at the non-inverting terminal of the op-amp is given as [24]:

$$v^+ = \frac{-jX_c}{R - jX_c} v_i \dots\dots\dots(3.6)$$

The above equation reduces to:

$$v^+ = \frac{v_i}{1 + j2\pi fRC} \dots\dots\dots(3.7)$$

We know that the output of an op-amp non-inverting amplifier is given by:

$$v_o = \left(1 + \frac{R_F}{R_1}\right)v^+ \dots\dots\dots(3.8)$$

Substituting for  $v^+$  from the previous equation,

$$v_o = \left(1 + \frac{R_F}{R_1}\right)\left(\frac{1}{1 + j2\pi fRC}\right)v_i \dots\dots\dots(3.9)$$

$$\frac{v_o}{v_i} = \frac{A_F}{1 + j\left(\frac{f}{f_H}\right)} \dots\dots\dots(3.10)$$

Where  $f_H = \frac{1}{2\pi RC}$

$f_H$  = high cut off frequency of the filter

$A_F = 1 + \frac{R_F}{R_1}$  = pass-band gain of the filter

$f$  is the frequency of the input signal

The gain magnitude and phase angle equations for the filter can be obtained as

$$\left| \frac{V_0}{V_i} \right| = \frac{A_F}{\sqrt{1 + (f/f_H)^2}} \dots\dots\dots(3.11)$$

and

$$\phi = -\tan^{-1} \left( \frac{f}{f_H} \right) \dots\dots\dots(3.12)$$

The operation of the low-pass filter can be verified from the gain magnitude equation [19]:

1. At very low frequencies that is  $f < f_H$ ,  $\left| \frac{V_0}{V_i} \right| \cong A_F$ .
2. At cut-off frequency, that is  $f = f_H$ ,  $\left| \frac{V_0}{V_i} \right| = \frac{A_F}{\sqrt{2}}$ .
3. At higher frequencies that is  $f > f_H$ ,  $\left| \frac{V_0}{V_i} \right| < A_F$ .

### 3.6.3 First Order High Pass Butterworth Filter

Fig. 3.11 shows a first-order high-pass filter that uses an RC network for filtering. The circuit is formed by interchanging the positions of R and C from the low pass filter circuit.

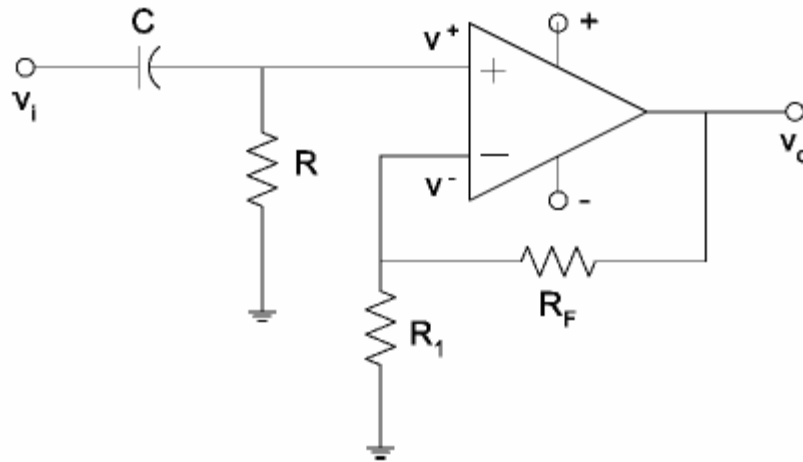


Figure 3.11. First Order High Pass Butterworth Filter

### Derivation of transfer function

Assuming that RC network behaves as a voltage divider

$$v^+ = \frac{R}{R - jX_C} v_i \dots\dots\dots(3.13)$$

The equation 3.13 reduces to

$$v^+ = \frac{j2\pi fRC}{1 + j2\pi fRC} v_i \dots\dots\dots(3.14)$$

The output of the non-inverting amplifier is

$$v_o = \left(1 + \frac{R_F}{R_1}\right) v^+ \dots\dots\dots(3.15)$$

Substituting v+ from the previous equation 2.10

$$v_o = \left(1 + \frac{R_F}{R_1}\right) \left(\frac{j2\pi fRC}{1 + j2\pi fRC}\right) v_i \dots\dots\dots(3.16)$$

$$\frac{v_o}{v_i} = A_F \left(\frac{j(f/f_L)}{1 + j(f/f_L)}\right) \dots\dots\dots(3.17)$$

Where  $f_L = \frac{1}{2\pi RC}$

$f_L$  = low cut-off frequency of the filter

$A_F$  = pass band gain of the filter

$f$  is the frequency of the input signal

The gain magnitude of the high pass filter is given by the equation [10], [19]

$$\left| \frac{V_o}{V_i} \right| = \frac{A_F(f/f_L)}{\sqrt{1 + (f/f_L)^2}} \dots \dots \dots (3.18)$$

### 3.6.4 Second Order Filters

Second order filters provide -40 dB/decade roll-off in the stop-band, and hence perform better frequency selection than the first order type. With second order, and higher-order filters, we can obtain interesting frequency responses. Consider the two frequency responses shown in fig 3.12.

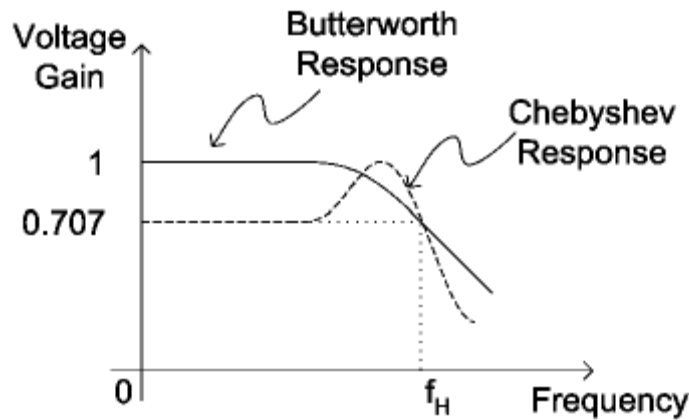


Figure 3.12. Second Order Low Pass Responses

Butterworth filters give us a reasonably flat gain in the pass-band, whereas the Chebyshev filters show a ripple or overshoot in the frequency response. The trade-off is that at the cut-off frequency, Chebyshev filter shows a higher roll-off rate. These frequency response types are determined the damping factor of the filter circuits.

**Damping Factor:** The damping factor (DF) of an active filter circuit determines which response characteristics the filter exhibits –whether, Butterworth or Chebyshev or others [24]. The damping factor is determined by the negative feedback circuit and is defined by the following equation:

$$DF = 2 - \frac{R_F}{R_1} \dots\dots\dots(1.19)$$

To achieve a second-order Butterworth response, for example, the damping factor must be 1.414. Therefore, to implement this damping factor, the feedback resistor ratio must be

$$\begin{aligned} \frac{R_F}{R_1} = 2 - DF = 2 - 1.414 = 0.586 \dots\dots\dots(1.20) \\ R_F = 0.586R_1 \end{aligned}$$

Hence, for a second-order Butterworth response,  $R_F = 0.586 R_1$ .

### 3.6.5 Second Order Low Pass Butterworth Filter

A second order low-pass filter provides a  $-40$  dB/decade roll-off rate in the stop-band [24]. The first order low-pass filter can be converted into a second-order type simply by using an additional RC network (as seen in fig. 3.13).

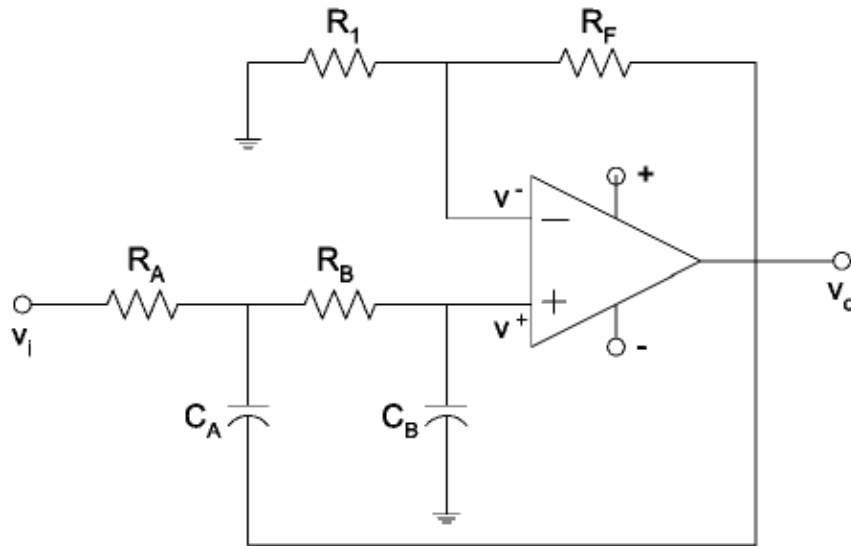


Figure 3.13. Second Order Low Pass Butterworth Filter

The gain of the filter is set by  $R_1$  and  $R_F$ , while the high cut-off frequency ( $f_H$ ) is set by  $R_A$ ,  $C_A$ ,  $R_B$  and  $C_B$  as follows:

$$f_H = \frac{1}{2\pi\sqrt{R_A R_B C_A C_B}} \dots\dots\dots(3.21)$$

The voltage gain magnitude equation is given as:

$$\left| \frac{v_0}{v_i} \right| = \frac{A_F}{\sqrt{1 + (f/f_H)^4}} \dots\dots\dots(3.22)$$

### 3.6.6 Second Order High Pass Butterworth Filter

The second order high pass filter can be formed from the low-pass filter simply by interchanging the frequency determining resistors and capacitors, as shown in fig. 3.14.

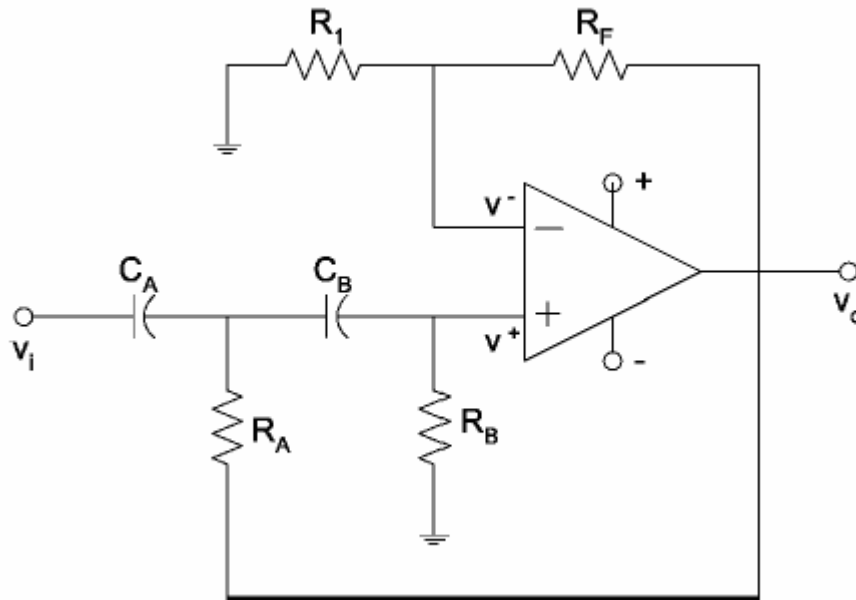


Figure 3.14. Second Order High Pass Butterworth Filter

The low cut-off frequency is given as:

$$f_L = \frac{1}{\sqrt{2\pi R_A R_B C_A C_B}} \dots\dots\dots(3.23)$$

The voltage gain magnitude equation is given as:

$$\left| \frac{v_0}{v_i} \right| = \frac{A_F}{\sqrt{1 + (f_L/f)^4}} \dots\dots\dots(3.24)$$

As in the case of low pass type,  $A_F$  should be 1.586 to ensure Butterworth response. In other words,  $R_F = 0.586R_L$ . As second order low pass and high pass filters are the same circuits except that the positions of resistors and capacitors are interchanged, the design procedure for the high-pass filter is same as that for the low-pass filter [24]. The frequency response of the second order high pass Butterworth filter is as shown in fig. 3.15.

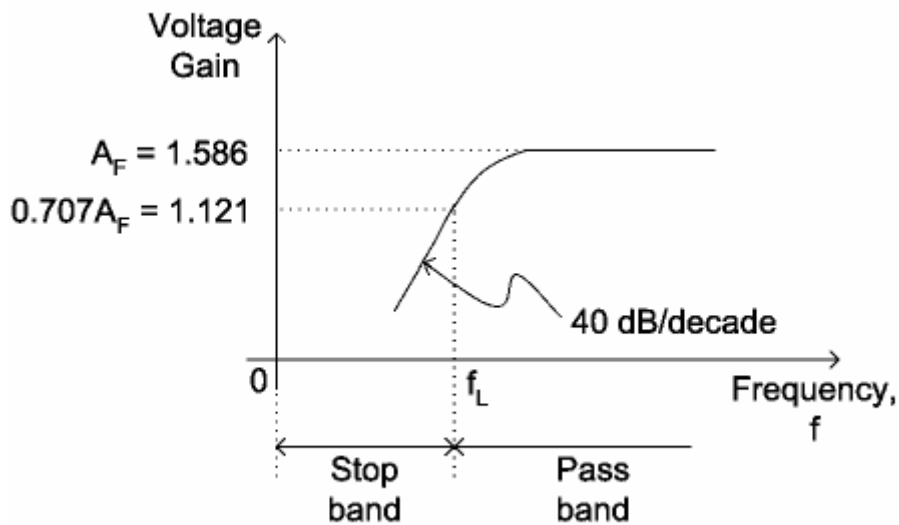


Figure 3.15. Frequency Response of Second Order High Pass Filter

# Chapter 4

## INTRODUCTION TO TANNER EDA TOOL

### 4.1 Introduction

Tanner Tool is a SPICE Computer Analysis Programmed for Analog Integrated circuits. Tanner Tool consists of the following Engine Machines:

1. S-EDIT (Schematic Edit)
2. T-EDIT (Simulation Edit)
3. W-EDIT (Waveforms Edit)
4. L-EDIT (Layout Edit)

Using these Engine tools, SPICE Programmes provides facility to the user to design and simulate new ideas in analog integrated circuits before going to the time consuming and costly process of chip fabrication.

### 4.2 S-EDIT (*Schematic Edit*)

S-Edit is hierarchy of files, modules and pages. It introduces symbol and schematic modes. S-Edit provides the facility of:

1. Beginning a design
2. Viewing, drawing and editing of objects
3. Design connectivity
4. Properties, net lists and simulations
5. Instance and browser schematic and symbol mode

In S-Edit, the available components from the library can be selected to make the schematic of the desired circuit. It explains the design process in detail in terms of file

module operation and module [23]. Effective schematic design requires a working knowledge of the S-Edit design files consist of modules. A module is a functional unit of design such as a transistor, a gate and an amplifier. Modules contain two components:

- a) Primitives – Geometrical objects created with drawing tools.
- b) Instances – References to other modules in file. The instanced module is the original.

Two viewing modes of the S-Edit are:

- a) Schematic mode – This mode helps in creating or viewing a schematic.
- b) Symbol mode – It represents symbol of a larger functional unit such as operational amplifier.

### **4.3 T-EDIT (Simulation Edit)**

The heart of T-Spice operation is the output file (also known as the circuit description, the net list and the input deck). This is a plain text file that contains the device statement and simulation commands, drawn from the SPICE circuit description language with which T-Spice constructs a model of the circuit to be simulated. Input files can be created and modified with any text editor.

T-Spice is a tool used for simulation of the circuit. It provides the facility of

- a) Design Simulation
- b) Simulation commands
- c) Device Statements
- d) User-defined External Models
- e) Small Signal and noise models

T-spice uses Kirchoff's Current Law (KCL) to solve circuit problems. To T-Spice, a circuit is a set of devices attached to the nodes. The voltage at all nodes represents the circuit state. T-Spice solves for a set of node voltage that satisfied KCL (implying that sum of currents flowing into each node is zero).

In order to evaluate whether a set of node voltages is a solution, T-Spice computers and sums all the current flowing out of each device into nodes connected to it (its terminals).

The relationship between the voltages at device terminals and the currents through the

terminal is determined by the device model for a resistor of resistance  $R$  is  
 $I = \Delta V/R$

Where  $\Delta V$  represents the voltage difference across the device.

### 4.3.1 DC Operating point Analysis

DC operating point analysis finds a circuit's steady-state condition, obtained (in principle) after the input voltages have been applied for an infinite amount of time. The **.include** command causes T-Spice to read in the contents of the model file `m12_125.md` for the evaluation of transistors `m1n` and `m1p`. This file (which must be in the same directory as `invert1.sp`) consists of two **.model** commands, describing two MOSFET models called `nmos` and `pmos`:

```
.model nmos nmos
+ Level=2 Ld=0.0u Tox=225.00E-10
+ Nsub=1.066E+16 Vto=0.622490 Kp=6.326640E-05
+ Gamma=0.639243 Phi=0.31 Uo=1215.74
+ Uexp=4.612355E-2 Ucrit=174667 Delta=0.0
+ Vmax=177269 Xj=0.9u Lambda=0.0
+ Nfs=4.55168E+12 Neff=4.68830 Nss=3.00E+10
+ Tpg=1.000 Rsh=60 Cgso=2.89E-10
+ Cgdo=2.89E-10 Cj=3.27E-04 Mj=1.067
+ Cjsw=1.74E-10 Mjsw=0.195

. model pmos pmos
+ Level=2 Ld=0.03000u Tox=225.000E-10
+ Nsub=6.575441E+16 Vto=-0.63025 Kp=2.635440E-05
+ Gamma=0.618101 Phi=0.541111 Uo=361.941
+ Uexp=8.886957E-02 Ucrit=637449 Delta=0.0
+ Vmax=63253.3 Xj=0.112799u Lambda=0.0
+ Nfs=1.668437E+11 Neff=0.64354 Nss=3.00E+10
+ Tpg=-1.000 Rsh=150 Cgso=3.35E-10
```

+ Cgdo=3.35E-10 Cj=4.75E-04 Mj=0.341

+ Cjsw=2.23E-10 Mjsw=0.307

m12\_125.md assigns values to various Level 2 MOSFET model parameters for both  $n$ - and  $p$ -type devices. When read by the input file, these parameters are used to evaluate Level 2 MOSFET model equations, and the results are used to construct internal tables of current and charge values. Values read or interpolated from these tables are used in the computations called for by the simulation. Two transistors, m1n and m1p, are defined in invert1.sp. These are MOSFETs, as indicated by the key letter m, which begins their names. Following each transistor name are the names of its terminals. The required order of terminal names is: drain–gate–source–bulk. Then the model name (nmos or pmos in this example), and physical characteristics such as length and width, is specified. The **.op** command performs a DC operating point calculation and writes the results to the file specified in the Simulate > Start Simulation dialog. The output file lists the DC operating point information for the circuit described by the input file.

### 4.3.2 DC Transfer Analysis

DC transfer analysis is used to study the voltage or current at one set of points in a circuit as a function of the voltage or current at another set of points. This is done by *sweeping* the source variables over specified ranges, and recording the output. The **.dc** command, indicating transfer analysis, is followed by a list of sources to be swept, and the voltage ranges across which the sweeps are to take place.

For example, for inverter with dc input  $V_{in}$  and output out,  $V_{in}$  will be swept from 0 to 3 volts in 0.02 volt increments, and  $V_{dd}$  will be swept from 2 to 4 volts in 0.5 volt increments. The transfer analysis will be performed as follows:  $V_{dd}$  will be set at 2 volts and  $V_{in}$  will be swept over its specified range;  $V_{dd}$  will then be incremented to 2.5 volts and  $V_{in}$  will be reswept over its range; and so on, until  $V_{dd}$  reaches the upper limit of its range. The **.dc** command ignores the values assigned to the voltage sources  $V_{dd}$  and  $V_{in}$  in the voltage source statements, but they must still be declared in those statements. The results for nodes in and out are reported by the **.print** dc command to the specified destination.

### 4.3.3 Transient Analysis

Transient analysis provides information on how circuit elements vary with time. The basic T-Spice command for transient analysis has three *modes*. In the *default* mode, the DC operating point is computed, and T-Spice uses this as the starting point for the transient simulation.

```
.tran 2n 600n  
.print tran in out
```

For the commands shown above, The **.tran** command specifies the characteristics of the transient analysis to be performed: it will last for 600 nanoseconds, with time steps no larger than 2 nanoseconds.

### 4.3.4 AC Analysis

AC analysis characterizes the circuit's behavior dependence on small-signal input frequency. It involves three steps: (1) calculating the DC operating point; (2) linearizing the circuit; and (3) solving the linearized circuit for each frequency.

```
Vin1 in1 GND 2  
Vdd Vdd GND 5.0  
vbias vbias GND 0.8  
vdiff in2 in1 -0.0007 AC 1 90  
.ac DEC 5 1 100MEG  
.print ac vdb(out)  
.print ac vp(out)  
.acmodel opamp1m.out {*}
```

For the commands shown above, three voltage sources (besides  $V_{dd}$ ) are defined. `vdiff` sets the DC voltage difference between nodes `in2` and `in1` to  $-0.0007$  volts; its AC magnitude is 1 volt and its AC phase is 90 degrees.

- `Vin1` sets node `in1` to 2 volts, relative to GND.

- `vbias` sets node `vbias` to 0.8 volts, relative to GND.

The `.ac` command performs an AC analysis. Following the `.ac` keyword is information concerning the frequencies to be swept during the analysis. In this case, the frequency is swept logarithmically, by decades (DEC); 5 data points are to be included per decade; the starting frequency is 1 Hz and the ending frequency is 100 MHz.

The two `.print` commands write the voltage magnitude (in decibels) and phase (in degrees), respectively, for the node `out` to the specified file. The `.acmodel` command writes the small-signal model parameters and operating point voltages and currents for all circuit devices (indicated by the wildcard symbol `*`) to the file `opamp1m.out`.

This example will generate two output files: `opamp1.out`, specified by the Simulate > Start Simulation command, and `opamp1m.out`, specified by the `.acmodel` command.

### 4.3.5 Noise Analysis

Real circuits, of course, are never immune from small, “random” fluctuations in voltage and current levels. In T-Spice, the influence of noise in a circuit can be simulated and reported in conjunction with AC analysis. The purpose of noise analysis is to compute the effect of the noise associated with various circuit devices on an output voltage or voltages as a function of frequency. Noise analysis is performed in *conjunction* with AC analysis; if the `.ac` command is missing, then the

`.noise` command is ignored. With the `.ac` command present, the `.noise` command causes noise analysis to be performed at the same frequencies: starting at 1 Hz, ending at 100 MHz, 5 data points per decade. The `.noise` command takes two arguments: the *output* at which the effects of noise are to be computed, and the *input* at which the noise can be considered to be concentrated for the purposes of estimating the equivalent noise spectral density [22].

## 4.4 W- EDIT (Waveform Edit)

The ability to visualize the complex numerical data resulting from VLSI circuit simulation is critical to testing, understanding and improving these circuits. W-Edit is a waveform viewer that provides ease of use, power and speed in flexible environment designed for graphical data representation. The advantages of W-Edit include;

- a) Tight integration with T-Spice, Tanner EDA's circuit level simulator. W-Edit can chart data generated by T-Spice directly, without modification of the output text data files. The data can also be charted dynamically as it is produced during the simulation.
- b) Chart can automatically configure for the type of data being presented.
- c) A data is treated by W-Edit as a unit called a trace. Multiple traces from different output files can be viewed simultaneously in single or several windows; traces can be copied and moved between charts and windows. Trace arithmetic can be performed on existed tracing to create new ones.
- d) Chart views can be panned back and forth and zoomed in and out, including specifying the exact X-Y co-ordinate range.
- e) Properties of axes, traces, rides, charts, text and colors can be customized.

Numerical data is input to W-Edit in the form of plain or binary text files. Header and comment information supplied by T-Spice is used for automatic chart configuration. Run time update of results is made possible by linking W-Edit to a running simulation in T-Spice. W-Edit saves data with chart, trace, axis and environment settings in files with the WDB (W-Edit Database) [22].

#### **4.5 L-EDIT (Layout Edit)**

It is a tool that represents the masks that are used to fabricate an integrated circuit. It describes the layout design in terms of files, cells and mask primitives. On the layout level the component parameters are totally different from schematic level. So, it provides the facility to the user to analyze the response of circuit before forwarding it to the time consuming and costly process of fabrication. There are rules for designing layout diagram

of a schematic circuit using which user can compare the output response with the expected one [22].

#### **4.5.1 L-Edit: An Integrated Circuit Layout Tool**

In L-Edit layers are associated with masks used in fabrication process. Different layers can be conveniently represented by different colors and patterns. L-Edit describes a layout design in terms of files, cells, instances and mask primitives. One may load as many files as desired into memory. A file may be composed of any number of sets. These cells may be hierarchically related, as in a typical design, or they may be independent, as in a “library” file. Cells may contain any number or combination of mask primitives and instances of other cells.

##### **Cells: The Basic Building Blocks**

The basic building block of the integrated circuit design in L-edit is a cell.

Design layout occurs within cells. A cell can:

- ...Contain part or all of entire design.
- ...Be referenced in other cells as a sub-cell, or instance.
- ...Be made up entirely of instances of other cells.
- ...Contain original drawn objects, or primitives.
- ...Be made up entirely of primitives or a combination of primitives and instances of other cells.

##### **Hierarchy**

L-Edit supports fully hierarchical mask design. Cells may contain instances of other cells. An instance is a reference to a cell; should you edit the instanced cell, the change is reflected in all the instances of that cell. Instances simplify the process of updating a design, and also reduce data within the instanced cell instead, only a reference to the instanced cell is stored, along with the information on the position of instance and on how the instance may be rotated or mirrored. There is no preset limit to the size or complexity

of hierarchy. Cells may contain instances of other cells that in turn contain instances of other cells, to an arbitrary number of levels (subject only to hardware constraints).

L-Edit does not use a “separated” hierarchy: instances and primitives may coexist in the same cell at any level in the hierarchy. Design files are self-contained. The “pointer” to a cell contained in an instance always points to a cell within the same design file. When cells are copied from one file to another, L-Edit automatically copies across any cells that are instanced by the copied cell, to maintain the self-contained nature of the destination file.

## **Design Rules**

Manufacturing constraints can be defined in L-Edit as design rules. Layout can be checked against these design rules.

## **Design Features**

L-Edit is a full-custom mask editor. Manual layout can be accomplished more quickly because of L-Edit’s intuitive user interface. In addition, one can construct special structures to utilize a technology without, worrying about problems caused by automatic transformations. Phototransistors, guard bars, vertical and horizontal bipolar transistors, static structures and Schottky diodes, for example, are as easy to design in CMOS-bulk technology as are conventional MOS transistors.

## **Floor plans**

L-Edit is a manual floor-planning tool. One has the choice of displaying in outline, identified only by name, or as fully fleshed-out mask geometry. When he displays his design in outline, he can manipulate the arrangement of the cells in the design quickly and easily to achieve the desired floor plan.

One can manipulate instances at any level in the hierarchy, with insides hidden or displayed, using the same graphical move/select operations or rotation/mirror commands that he use on primitive mask geometry.

## **Memory Limits**

In L-Edit, one can make your design files as large as one like, given available RAM and disk space.

## **Hard Copy**

L-Edit provides the capability to print hard copy of the design. A multistage option allows very large plots to be printed to a specific scale on multiple 8 ½ x 11 inch page. An L-Edit macro is available to support large-format, high-resolution, color plotting on inkjet plotters.

## **Variable Grid**

L-Edit's grid options support lambda-based design as well as micron based and mil-based design.

## **Error Recovery**

L-Edit's error-trapping mechanism catches system errors and in most cases provides a mean to recover without losing or damaging data.

## **L-Edit Module**

...L-Edit<sup>TM</sup> : a layout editor

...L-Edit  $\square$  Extract<sup>TM</sup>: a layout extractor

...L-Edit  $\square$  DRC<sup>TM</sup>: a design rule checker

L-Edit is a full featured, high performance, interactive, graphical mask layout editor. L-Edit generates layouts quickly and easily, supports fully hierarchical design, and allows an unlimited number of layers, cells, and level of hierarchy. It includes all major drawing primitives and supports 90°, 45°, and all-angle drawing modes.

**L-Edit** □ **Extract creates SPICE-** compatible circuit netlists from L-Edit layouts. It can recognize active and passive devices, sub circuits and the most common device parameters, including resistance, capacitance, device length, width, and area, and device source and drain area.

**L-Edit** □ **DRC features user-** programmable rules and handles minimum width, exact width, minimum space, minimum surround, non-exist, overlap, and extension rules. It can handle full chip and region-only DRC. DRC offers Error Browser and Object browser functions for quickly and easily cycling through rule-checking errors [22].

# Chapter 5

## TELESCOPIC OTA: DESIGN, SIMULATION AND SYNTHESIS

### **5.1 Introduction**

*This chapter starts with the general theory of operational amplifiers and OTA (operational transconductance amplifier). Thereafter, a detailed study of Telescopic OTA is done. The chapter presents the design steps for Telescopic OTA along with the design. Simulation results of the schematic and layout of Telescopic OTA are shown in the end.*

### **5.2 Operational Amplifier**

Operational amplifiers are an integral part of many analog and mixed and mixed-signal systems. Opamps with vastly different levels of complexity are used to realize functions ranging from dc bias generation to high-speed amplification or filtering. The design of opamp continues to pose a challenge as the supply voltage and transistor channel lengths scale down with each generation of CMOS technologies.

Operational Amplifier is defined as “high gain differential amplifier”. By “high”, we mean a value that is adequate for the application, typically in the range of  $10^1$  to  $10^5$ . since opamps are usually employed to implement a feedback system, there open loop gain is chosen according to the precision required of the closed-loop circuit [18]. An ideal opamp has infinite differential voltage gain, infinite input resistance, and zero output resistance. In reality opamp only approaches these values. The term buffered and unbuffered is used to distinguish between high output resistance (Operational Transconductance Amplifier) and low output resistance amplifiers (voltage operational amplifiers) [9].

### 5.2.1 Performance Parameters

The optimal selection of opamp to be used in a particular application is often the key factor, which determines the success or failure of the circuit. There is a wide variety of op-amps available, from those requiring only 1 volt supply with bias currents of the 10 - 15 amp range, to those that will output hundreds of volts at tens of amps. There are many more parameters of opamp, which would be taken into consideration.

1. **Gain** – the open loop gain of an opamp determines the precision of the feedback system employing the opamp. Trading with speed and output voltage swings, the minimum required gain must be known. A high open loop gain may also be necessary to suppress linearity.
2. **Bandwidth** – The speed performance of the opamp is described by small and large signal parameters. The small signal analysis determines the frequency response sketched by a set of poles and zeroes. Since we have to ensure stability, one of the poles must be dominant. The amplitude bode diagram will display a 20 dB/decade roll-off until the gain reaches 0dB. The frequency at which the gain becomes 0dB is called gain bandwidth. With a constant roll-off 20 dB/decade of the achieved unity gain frequency equal to the product of gain and bandwidth. The 3dB frequency may also be specified to allow easier prediction of the closed-loop frequency response.
3. **Common mode rejection ratio** – The common mode rejection ratio (CMRR) of an operational amplifier is the ratio between the differential gain and the common mode gain.

$$\text{CMRR} = 20\log_{10}(A_D/A_C) \quad \text{in decibel,}$$

Typical CMRR values for CMOS amplifiers are in the range 60dB to 80 dB.

The CMRR is an important requirement.

4. **Power supply rejection ratio** – The PSRR is defined as the product of the ratio of the change in supply voltage to the change in output voltage of the opamp caused by the change in the power supply and the open-loop gain of the opamp. An ideal opamp would have an infinite PSRR.
5. **Output swing** – this is the maximum swing of the output node without producing a significant degradation of opamp performance. Since we have to leave some room for the operation of the devices connected between the output nodes and the supply nodes, the output swing is only a fraction of  $(V_{dd} - V_{ss})$ . Typically it ranges between 60% and 80% of  $(V_{dd} - V_{ss})$ . Within the output swing range the response of the opamp should conform to given specifications and in particular the harmonic distortion should remain below the required level.
6. **Slew rate** – Opamps used in feedback circuits exhibit a large signal behavior called “slewing”. This is the maximum achievable time derivative of the output voltage. It is measured using the opamp in the open loop or the unity gain configuration. A large input step voltage fully imbalances the input differential stage and brings the opamp output response into the slewing conditions. The positive slew rate can be different from the negative slew rate depending on the specific design. Typical values range between 40 and 80 V/ $\mu$ s, but micro-power circuits can show much lower figures [18].
7. **Power consumption** – This is the power consumed under stand by conditions. the power used in the presence of a large signal can significantly exceed the one required in the quiescent conditions. Moreover, the consumed power depends on the speed specifications. Typically, higher bandwidth leads to higher power consumption. Low power operation is a very important quality factor. A key design task is to achieve the minimum power consumption for a given required speed [3].

### 5.2.2 Opamp Topologies

A telescopic OTA can be used in many applications like filter design because of its simplicity over other designs, allowing for higher-speed operation. In a folded-cascode

design, there is an input differential pair and two separate current branches for the differential output. The input currents are mirrored with a cascoded configuration to produce the output currents. The telescopic architecture puts both the input differential pair and the output on the same two current branches. This approach eliminates the noise problems caused by the current mirrors and also leads to a more direct signal path, which allows for higher speed. Another advantage of the telescopic architecture is that it uses half the bias current of a folded-cascode design because it has two fewer branches for current. Fig 5.1 (a), (b) and (c) shows the three conventional topologies. Third topology is two-stage opamp in which an additional stage is used. First stage is for high gain and second stage is for high swing. To obtain higher gain, first stage can incorporate cascode devices. An opamp used in a hearing aid must operate with supply voltages as low as 0.9V while delivering single-ended output swings as large as 0.5 V. In such cases, two-stage opamp is used [13].

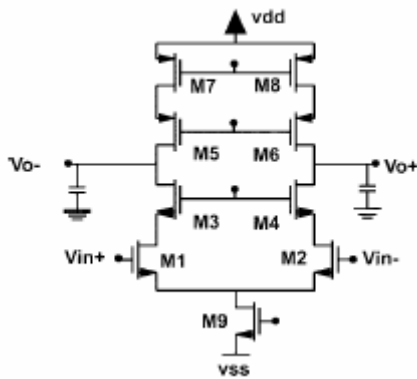


Figure 5.1. (a) Telescopic opamp

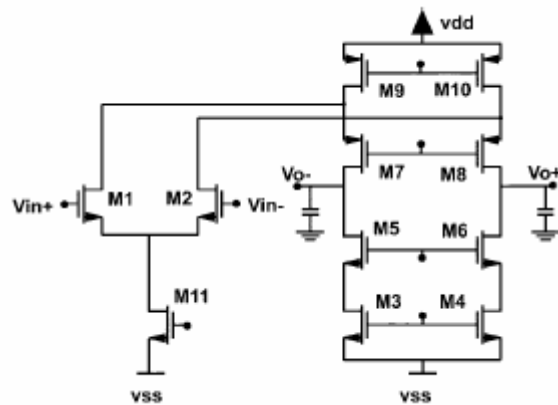


Figure 5.1. (b) Folded cascode opamp

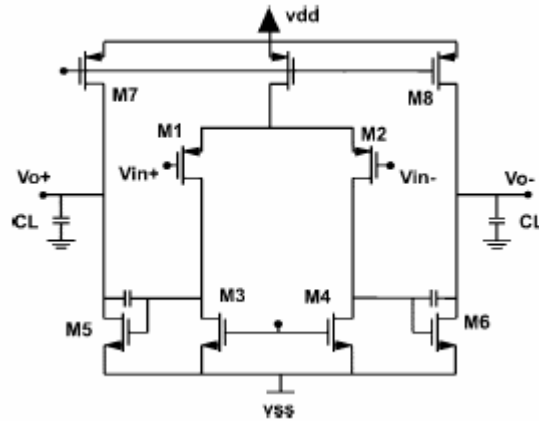


Figure 5.1. (c) Two stage opamp

The above three topologies have been compared in the table shown below. The clear comparison between the three makes it easy to evince why Telescopic has been used for the application of filter [3].

Table 5.1. Comparison of performance of various opamp topologies.

	Gain	Speed	Power Dissipation	Output swing	Noise
Telescopic	Medium	Highest	Low	Medium	Low
Folded cascode	Medium	High	Medium	Medium	Medium
Two stage	High	Low	Medium	Highest	Low

### 5.3 OTA (Operational Transconductance Amplifier)

An OTA is a voltage controlled current source, more specifically the term “operational” comes from the fact that it takes the difference of two voltages as the input for the current conversion. The ideal transfer characteristic is therefore

$$I_{Out} = g_m(V_{In+} - V_{In-}) \dots\dots\dots(5.1)$$

or, by taking the pre-computed difference as the input,

$$I_{\text{Out}} = g_m V_{\text{In}} \dots\dots\dots(5.2)$$

with the ideally constant transconductance  $g_m$  as the proportionality factor between the two. In reality the transconductance is also a function of the input differential voltage and dependent on temperature.

The term “transconductance” comes about because the ratio of the output current over the input voltage,  $g_m$ , has the unit of a conductance if looked at “across the amplifier”. The proportional factor of output vs. input for an amplifier with current input and voltage output has the unit of a resistance and such an amplifier is called a transresistance amplifier.

An ideal OTA has two voltage inputs with infinite impedance (i.e. there is no input current), as shown in fig. 5.2. The common mode input range is also infinite, while the differential signal between these two inputs is used to control an ideal current source (i.e. the output current does not depend on the output voltage) that functions as an output. The proportionality factor between output current and input differential voltage is called transconductance.

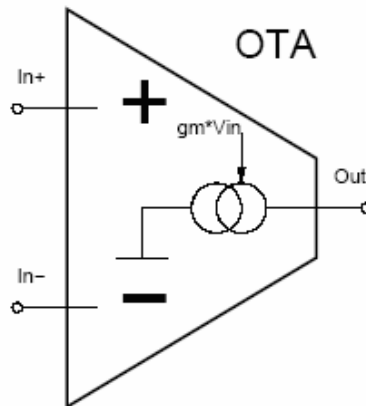


Figure 5.2. Ideal OTA

Any real OTA will thus have circuitry to process the input voltages with low input current over a wide common mode input range, to produce an internal representation of the input differential voltage and to provide a current to the output that is relatively independent of the output voltage. Since an OTA can be used without feedback, the maximum output current and with it the transconductance can often be adjusted [1].

The simplest version of the single stage OTA is the telescopic architecture. The term buffered and unbuffered is used to distinguish between high output resistance (Operational Transconductance Amplifier) and low output resistance amplifiers (voltage operational amplifiers).

## 5.4 Telescopic OTA

The gain that can be achieved by a single stage is around 40 dB. Thus, in order to achieve 80dB or so it is necessary to use a cascade of two stages. However, two stages bring about two poles one close to the other and this requires compensation network, besides increasing the global complexity, reduces the design flexibility.

A cascade with cascode load permits us to achieve high gain without the disadvantage of having two poles one close to each other. Therefore the use of cascode based OTA is an interesting solution alternative to the two stages OTA [3, 9,18].

The simplest version of the single stage OTA is the telescopic architecture, shown in fig. 3.3, the input differential pair injects the signal currents into common gate stages. Then, the circuit achieves the differential to single ended conversion with a cascode current mirror. The transistors are placed one on the top of the other to create a sort of Telescopic composition (this led to the circuit name as telescopic opamp). The small signal resistance at the output node is quite high, it is the parallel connection of two cascode configurations. Such a high resistance benefits the small signal gain without limiting the circuit functionality when we require an OTA function. By inspection of the circuit one finds the low frequency small signal differential gain is proportional to the square of the product of a transistor transconductance and the output resistance.

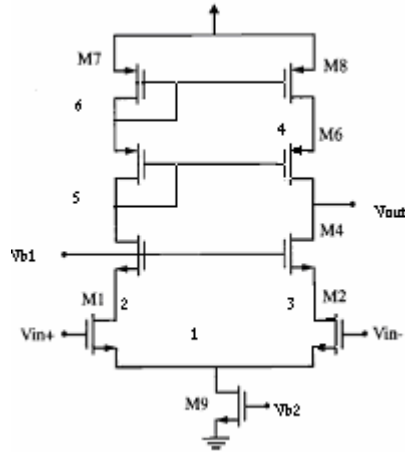


Figure 5.3. A conventional telescopic opamp

Thus, as expected the telescopic cascode achieves a gain similar to the one of the two stages architecture. Moreover, by inspection of the circuit, all the nodes, excluding the output, shows a pretty low small signal resistance. Node 1 is an equivalent ground for differential signals: node 2, 3 and 4 are source terminals of transistors M3, M4 and M6, respectively.

Assuming the capacitance is affecting the given nodes due to parasitic contributions, the resulting time constants are much smaller than the one associated to the output node can easily become the dominant pole of the circuit. Since the circuit shows one high impedance node only, it is not possible to exploit the Miller effect to procure pole splitting. A possible capacitance loading the output node permits to make dominant the related pole and ensure stability [9, 18].

The telescopic configuration uses only one bias current. It flows through the differential input stage, the common baser stage and differential to single ended converter.

The triode limit of M6 establishes the maximum allowed output voltage. By inspection of the circuit it is given by

$$V_{out,max} = V_{DD} - V_{GS7} - V_{GS5} - V_{GS6} - V_{SAT,P} = V_{DD} - V_{Th,P} - 2V_{SAT,P} \dots\dots(5.3)$$

For typical situations it is 1V or more below the positive supply voltage.

The lower boundary of the output voltage depends the triode limit of M4 that, in turn depends on  $V_{b1}$ .

$$V_{out,min} = V_{b1} - V_{GS4} + V_{SAT,4} = V_{b1} - V_{Th,n} \dots\dots\dots(5.4)$$

Normally, the designer broadens the output swing by keeping low  $V_{b1}$ .

However, the value of  $V_{b1}$  affects the minimum level of the input common mode voltage.

$$V_{in,cm} \leq V_{b1} - V_{GS4} - V_{sat,2} + V_{GS2} = V_{b1} - 2V_{SAT,n} \dots \dots \dots (5.5)$$

In turn the input common mode voltage should allow M9 to be in saturation region.

$$V_{in,cm} > V_{SAT,9} + V_{GS2} \dots \dots \dots (5.6)$$

Therefore, we can achieve an optimum negative swing ( $3V_{sat,n}$  above ground) keeping the input common mode voltage as low as  $V_{SAT,9} + V_{GS2}$  approximately equal to  $V_{Th,n} + 2V_{SAT}$ . Assuming a symmetrical output swing around  $V_{out,max}$  and  $V_{out,min}$  the output common mode voltage becomes

$$V_{out,cm} = V_{DD} + V_{b1} - V_{Th,n} - V_{Th,p} - 2V_{SAT,p} \dots \dots \dots (5.7)$$

This for a typical design is a bit higher than  $V_{b1}$ . Thus, the output common mode voltage is different (higher) than the input common mode voltage. This in some applications is a limit.

A telescopic cascode opamp typically has a higher frequency capability and consumes less power than other topologies. The disadvantage of a telescopic opamp is severely limited output swing. In a conventional telescopic opamp shown in fig. 5.3, all transistors are biased in the saturation region. Transistors M1-M2, M7-M8 and the tail current source M9 must have at least  $V_{DS,SAT}$  to offer a good common-mode rejection (CMRR). To allow for process and temperature variations, a small safety margin  $V_{margin}$  is often added to their  $V_{DS}$  to ensure saturation. The maximum differential output swing of a telescopic opamp is shown to be  $2V_{sup} - 10V_{DS,SAT} - 6V_{margin}$ , where  $V_{sup}$  is the supply voltage. With typical  $V_{DS,SAT}$  of 200 mV and  $V_{margin}$  of 100mV, the output swing is limited to  $2V_{sup} - 2.6V$ . In a 3 V supply system; this represents a 45% reduction of available output swing [13]. Although a telescopic opamp without the tail current source improves the differential swing by  $2V_{DS,SAT} + 2V_{margin}$  (600 mV), the CMRR is compromised. Moreover, performance parameters (such as unity-gain frequency) of an opamp with no tail or with a tail transistor in the linear region are sensitive to input common-mode and supply voltage variation which is undesirable in most analog systems [15].

### 5.4.1 Frequency Compensation

The need for compensation arises because  $|\beta H|$  does not drop to unity well before  $\angle\beta H$  reaches  $-180^\circ$ . Stability can be achieved by:

1. Minimizing the overall phase shift, thus pushing the phase crossover out.
2. Dropping the gain, thereby pushing the gain crossover in.

The first approach requires that we attempt to minimize the number of poles in the signal path by proper design. Since, each additional stage contributes at least one pole, this means the number of stages must be minimized, a remedy that yields low voltage gain and/or limited output swing. The second approach, on the other hand, retains the low frequency gain and the output swings but it reduces the bandwidth by forcing the gain to fall at lower frequencies.

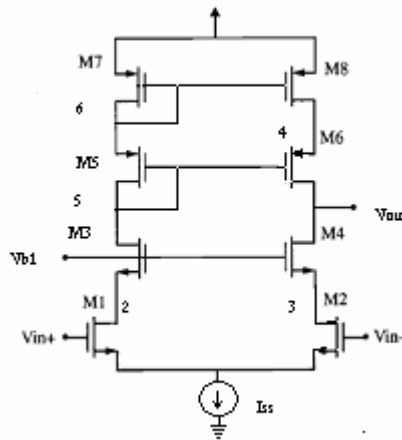


Figure 5.4 Telescopic opamp single ended output with constant current tail

In a signal path number of poles can be identified: path 1 contains a high frequency pole at 2, mirror pole at 5, mirror pole at 6 and high frequency pole at 4; path 2 contains high frequency pole at 3. the output resistance of the opamp is much higher than the small-signal resistances seen at the other nodes in the circuit, it is expected even with a moderate load capacitance, the output pole,  $\omega_{p,out}$ , is closest to the origin. Called the “dominant pole”,  $\omega_{p,out}$  usually sets the open loop 3dB bandwidth. After the dominant pole the first “non dominant” pole will be at node 6. This is because the total capacitance

is higher than at the other nodes. Nodes 4 and (2 or 3) have equal small-signal resistances to ground but node 4 suffer from much more capacitance. Thus, node 4 contributes the next nondominant pole. The poles at nodes 2 and 3 are nearly equal and their corresponding terms in the transfer functions of path 1 and path 2 can be factored out. Thus, they count as one pole rather than two. Active current mirror exhibits zeroes so two zeroes will also be contributed located at twice the mirror pole frequency.

The loop gain is dropped such that the gain crossover point moves towards the origin. To accomplish this, lower the frequency of the dominant pole by increasing the load capacitance. The key point is that the phase contribution of the dominant pole in the vicinity of the gain or the phase crossover points is close to 90° and relatively independent of the location of the pole [3].

### 5.4.2 Differential Stage

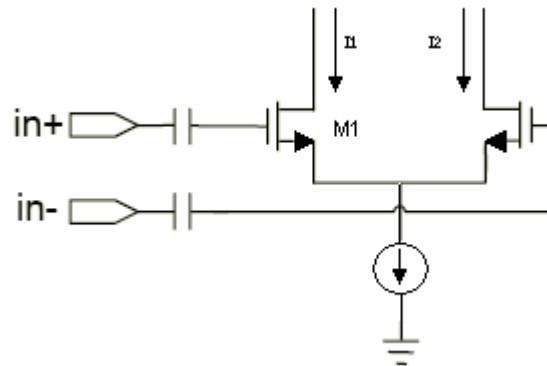


Figure 5.5. Differential input pair telescopic OTA

A differential stage is widely used as the input stage of the opamp. Fig. 5.5 shows its CMOS configuration. It is made of two transistors with their source common, fed by current source. The transistors may either be n-channel or p-channel and they are matched to each other. If the two transistors are in saturation region, we can write

$$I_1 = \frac{\mu C_{ox}}{2} \left( \frac{W}{L} \right)_1 (V_{GS} - V_{Th})^2 \dots\dots\dots(5.8)$$

$$I_2 = \frac{\mu C_{ox}}{2} \left( \frac{W}{L} \right)_2 (V_{GS} - V_{Th})^2 \dots\dots\dots(5.9)$$

where  $(W/L)_1$  and  $(W/L)_2$  are exactly equal, the transistors being matched. Moreover, in the above equations the output conductance has been neglected. The input signals can be expressed as

$$V_{GS1} = V_{GS0} + V_{in}/2 \dots\dots\dots(5.10)$$

$$V_{GS1} = V_{GS0} - V_{in}/2 \dots\dots\dots(5.11)$$

where  $V_{GS0}$  is the common mode component and  $V_{in}$  is a differential signal. Since the bias current can be expressed as

$$I_{ss} = I_1 + I_2 = \frac{\mu C_{OX}}{2} \left( \frac{W}{L} \right)_1 (V_{GS1} - V_{Th})^2 \dots\dots\dots(5.12)$$

*Therefore, in the differential stage, like in the case of the inverter with active load, the transconductance gain increases with the square root of the bias current [9].*

### **5.4.3 Biasing circuit for OTA**

The amplifier stage needs various dc bias voltages and currents for their proper operation. An essential step in the design of an amplifier using MOS transistor is the establishment of an appropriate DC operating point for the circuit. This is step known as biasing or bias design. An appropriate DC operating point or bias point is characterized by a stable and predictable DC drain current  $I_d$  and a DC drain source voltage,  $V_{DS}$  that ensures operation in the saturation mode for all expected signal levels [2].

It is important to place the DC operating point of an amplifier well within the saturation region of the MOSFET in order to maximize the output voltage swing of the amplifier while maintaining linear operation.

There are various biasing circuits available each having some positive and negative points associated with them. Current mirror circuits are found suitable for providing constant biasing few of them are listed below:

- a) Cascode current mirror
- b) Wilson current mirror
- c) Widlar current source

Some simplified biasing circuits are shown here. One of these will be used in the Telescopic OTA circuit later in this chapter. As shown in fig. 5.3 there is a requirement of a constant voltage,  $V_{bias}$  to be applied at the gate of M9 this bias can be generated by various configurations. The configurations used in this thesis are shown in fig. 5.6 (a) and (b). The configurations are chosen because of its simplicity, fewer transistors used and less voltage headroom.

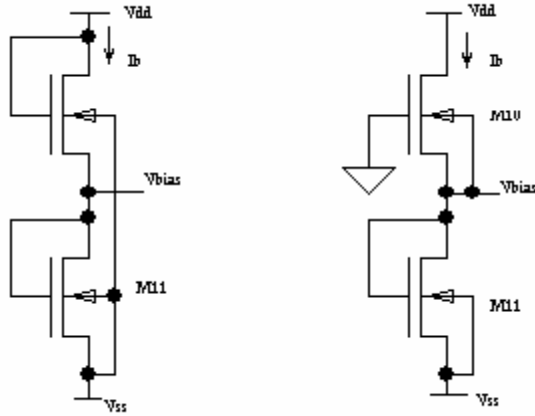


Figure 5.6. (a) Biasing circuit with diode connected load (b) Biasing circuit with gate grounded load

Choosing a current  $I_b$ , the aspect ratios of M9 and M10 for fig. 5.6(b) can be calculated as follows

$$V_{GS10} = V_{dd} - V_{bias} \dots \dots \dots (5.13)$$

$$V_{GS11} = V_{bias} - V_{ss} \dots \dots \dots (5.14)$$

Aspect ratios of the transistors can be calculated as follows:

$$\left(\frac{W}{L}\right)_{10} = \frac{2I_b}{\mu C_{ox}(V_{GS10} - V_{Th,n})} \dots \dots \dots (5.15)$$

$$\left(\frac{W}{L}\right)_{11} = \frac{2I_b}{\mu C_{ox}(V_{GS11} - V_{Th,n})} \dots \dots \dots (5.16)$$

Similarly, choosing a current of the bias chain  $I_b$ , the aspect ratios of M9 and M10 can be calculated for fig. 5.6(b)

$$As, V_{GS10} = 0 - V_{bias} \dots \dots \dots (5.17)$$

$$V_{GS11} = V_{bias} - V_{ss} \dots \dots \dots (5.18)$$

Therefore, aspect ratios of the transistors can be calculated as follows:

$$\left(\frac{W}{L}\right)_{10} = \frac{2I_b}{\mu C_{ox}(V_{GS10} - V_{Th, n})} \dots\dots\dots(5.19)$$

$$\left(\frac{W}{L}\right)_{11} = \frac{2I_b}{\mu C_{ox}(V_{GS11} - V_{Th, n})} \dots\dots\dots(5.20)$$

## 5.5 Design of Telescopic OTA

The design of Telescopic OTA has been shown in steps so as to understand the procedure better. The steps has been defined first and then explained with the help of the example. The transistor names are same as shown in fig. 5.3.

### 5.5.1 Steps to design Telescopic OTA

1. First step is to define the specifications of the Telescopic OTA. The design specification includes Slew rate, Load capacitance, Power dissipation, Supply voltages, DC gain and unity gain bandwidth.
2. To get an estimate of the bias current assume GBW by the dominant pole.

$$2\pi f_T = \frac{2I_{SS}}{(V_{GS} - V_{Th})} \times \frac{1}{C_L} \dots\dots\dots(5.21)$$

where  $I_{SS}$  is the tail current.

3. Distribute the overdrive voltages among the transistors in one leg (of cascaded transistors). Same is done for the other leg.
4. Design tail transistor M9 and calculate W/L of this transistor by using the current equation for the condition of saturation. The equation is

$$I_{SS} = \frac{\mu C_{ox}}{2} \left( \frac{W}{L} \right)_9 (V_{GS} - V_{Th})^2 \dots\dots\dots(5.22)$$

5. Calculate the bias  $V_{b2}$  of transistor M9 using this equation

$$V_{b2} = V_{GS9} - V_{Th} \dots\dots\dots(5.23)$$

6. The differential pair of the circuit, by assuming both of them to be working in saturation mode. W/L of the transistors can be calculated using current equation for the condition of saturation.

$$I_{SS} = \mu C_{ox} \left( \frac{W}{L} \right)_1 (V_{GS} - V_{Th})^2 \dots\dots\dots(5.24)$$

7. Design the bias circuit for the tail transistor. Determine  $V_{b2}$  and then compute the W/L of the two transistors using the current equation for the condition of saturation. The current is mirrored from the cascode arrangement to the tail transistors.

$$V_1 = V_{b2} - V_{Th} \dots\dots\dots(5.25)$$

8. Design the cascode current mirror stage where there are four PMOS transistors which are identical and the current passing through them is same as the drain and gate current passing through them is same as the drain and gate are tied to each other. W/L can be calculated by the current equation as follows

$$I_{SS} = \mu C_{ox} \left( \frac{W}{L} \right)_{5,6,7,8} (V_{GS} - V_{Th})^2 \dots\dots\dots(5.26)$$

where  $V_{GS} = V_{DD} - 3V_{Th,p}$ .

**5.5.2 Design of Telescopic OTA with defined specifications**

- 1. Slew rate = 1000 V/  $\mu$ s
- Load capacitance,  $C_L = 0.5$  pF
- Power dissipation < 3 mW
- Supply voltages =  $\pm 2.5V$

DC gain = 70 dB  
 UGB = 500 MHz

2. After calculating and taking some margin tail current we get  $I_{ss} = 500 \mu\text{A}$
3. Now, distributing the overdrive voltages as 1 V for the tail transistor, 200 mV for the differential pair (four NMOS transistors) and 250 mV for the cascode current mirror stage where there are four PMOS transistors.
4. Design tail transistor M9 and calculate W/L of this transistor by using the current equation for the condition of saturation. The equation is

$$500 \times 10^{-6} = \frac{6.33 \times 10^{-5}}{2} \left(\frac{W}{L}\right)_9 (1)^2 \dots\dots\dots(5.27)$$

$$\left(\frac{W}{L}\right)_9 = 15$$

$$W = 30\mu$$

$$L = 2\mu$$

L is taken as  $2\mu$ . Accordingly, W is calculated. L can be taken  $5 \mu$ ,  $10 \mu$  or any nominal value.

5. Calculate the bias  $V_{b2}$  of transistor M9 using this equation

$$V_{GS9} - V_{Th} = 1 \dots\dots\dots(5.28)$$

$$V_{b2} = -0.9\text{V}$$

6. The differential pair of the circuit, by assuming both of them to be working in saturation mode. W/L of the transistors can be calculated using current equation for the condition of saturation.

$$500 \times 10^{-6} = 6.33 \times 10^{-5} \left(\frac{W}{L}\right)_{1,2} (0.2)^2 \dots\dots\dots(5.29)$$

$$\left(\frac{W}{L}\right)_{1,2} = 197.5$$

$$W = 395 \mu$$

$$L = 2 \mu$$

Considering the same conditions for the pair M3 and M4.

$$\left(\frac{W}{L}\right)_{3,4} = 197.5$$

$$W = 395 \mu$$

$$L = 2 \mu$$

7. Design the bias circuit for the tail transistor. Compute the W/L of the two transistors using the current equation for the condition of saturation. The current is mirrored from the cascode arrangement to the tail transistors. The calculations are

$$500 \times 10^{-6} = \frac{6.33 \times 10^{-5}}{2} \left(\frac{W}{L}\right)_{11} (-0.9 + 2.5 - 0.6)^2 \dots (5.30)$$

$$\left(\frac{W}{L}\right)_{11} = 15.79$$

$$W = 32 \mu$$

$$L = 2 \mu$$

Considering the body effect for M10,  $V_{Th}$  is calculated using the equation

$$V_{Th} = V_{T0} + \gamma \left( \sqrt{V_{SB} - |2\phi_F|} - \sqrt{|2\phi_F|} \right) \dots \dots \dots (5.31)$$

$$V_{Th} = 0.622 + 0.6392 \left( \sqrt{-0.9 + 2.5 - 0.31} - \sqrt{0.31} \right) \dots \dots \dots (5.32)$$

$$V_{Th} = 0.99 \text{ V}$$

$$500 \times 10^{-6} = \frac{6.33 \times 10^{-5}}{2} \left(\frac{W}{L}\right)_{10} (2.5 + 0.9 - 0.99)^2 \dots \dots \dots (5.33)$$

$$\left(\frac{W}{L}\right)_{10} = 2.72$$

$$W = 6 \mu$$

$$L = 2 \mu$$

8. Design the cascode current mirror stage where there are four PMOS transistors which are identical and the current passing through them is same as the drain and gate current passing through them is same as the drain and gate are tied to each other. W/L can be calculated by the current equation as follows

$$500 \times 10^{-6} = 2.64 \times 10^{-5} \left( \frac{W}{L} \right)_{5,6,7,8} (0.25)^2 \dots\dots\dots(5.34)$$

$$\left( \frac{W}{L} \right)_{5,6,7,8} = 303$$

$$W = 606 \mu$$

$$L = 2 \mu$$

Table 5.2 Calculated Aspect ratios for the telescopic OTA

Transistor name	W/L (in $\mu\text{m}$ )
M1	395/2
M2	395/2
M3	395/2
M4	395/2
M5	606/2
M6	606/2

M7	606/2
M8	606/2
M9	30/2
M10	6/2
M11	32/2

### 5.5.3 Simulation results of OTA

This section discusses the results obtained by the schematic level simulations. The schematic drawn in S-Edit module of Tanner EDA Tool and the simulations are done in T-Spice module of Tanner EDA Tool.

The DC gain of the Telescopic OTA has been simulated in T-Spice, Level 2 by applying differential input. The schematic of the Telescopic OTA is shown in fig. 5.7, a constant current source is used and the frequency response curve and phase response is shown in fig 5.8 and fig. 5.9. This simulation with constant current source at the tail is done to verify the response when simulated with transistors at the tail later on.

The node names correspondence of the schematic to fig 5.3 is shown in Table 5.3:

Table 5.3 Node name correspondence of fig 5.3 with fig 5.7

Fig. 3.3	1	2	3	4	5	6	7
Fig. 3.7	N3	N1	N2	P2	P3	P1	V <sub>out</sub>

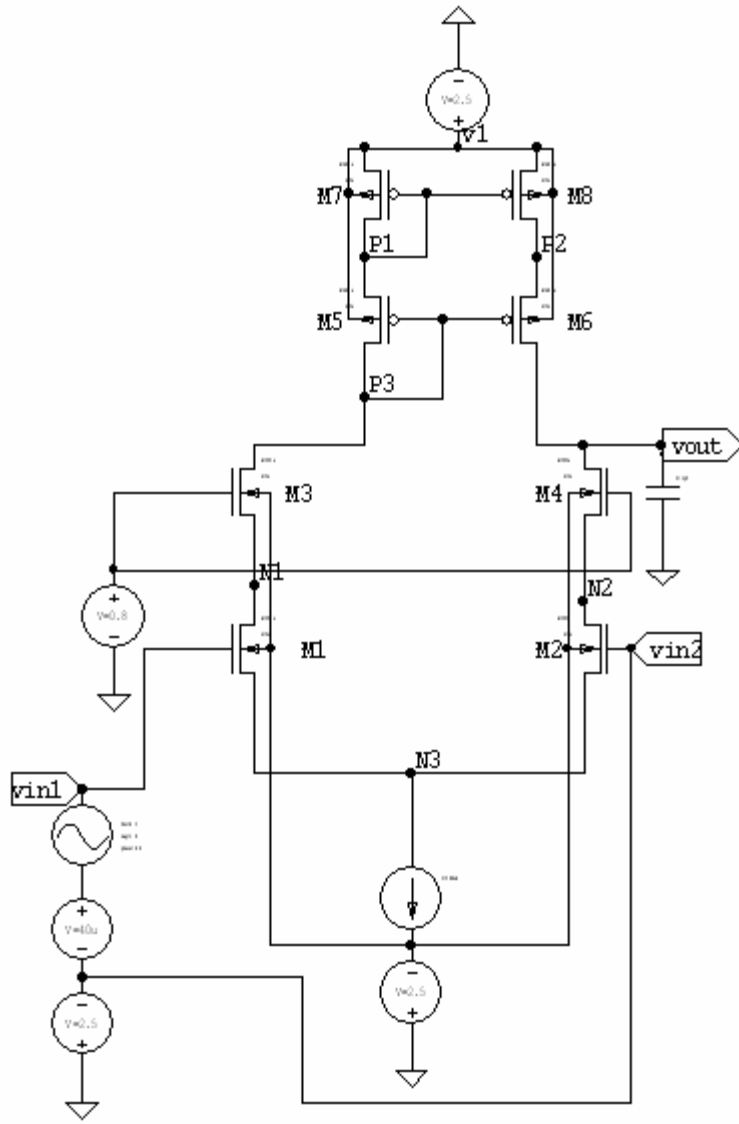


Figure 5.7. Schematic of the Telescopic OTA with constant current source at the tail.

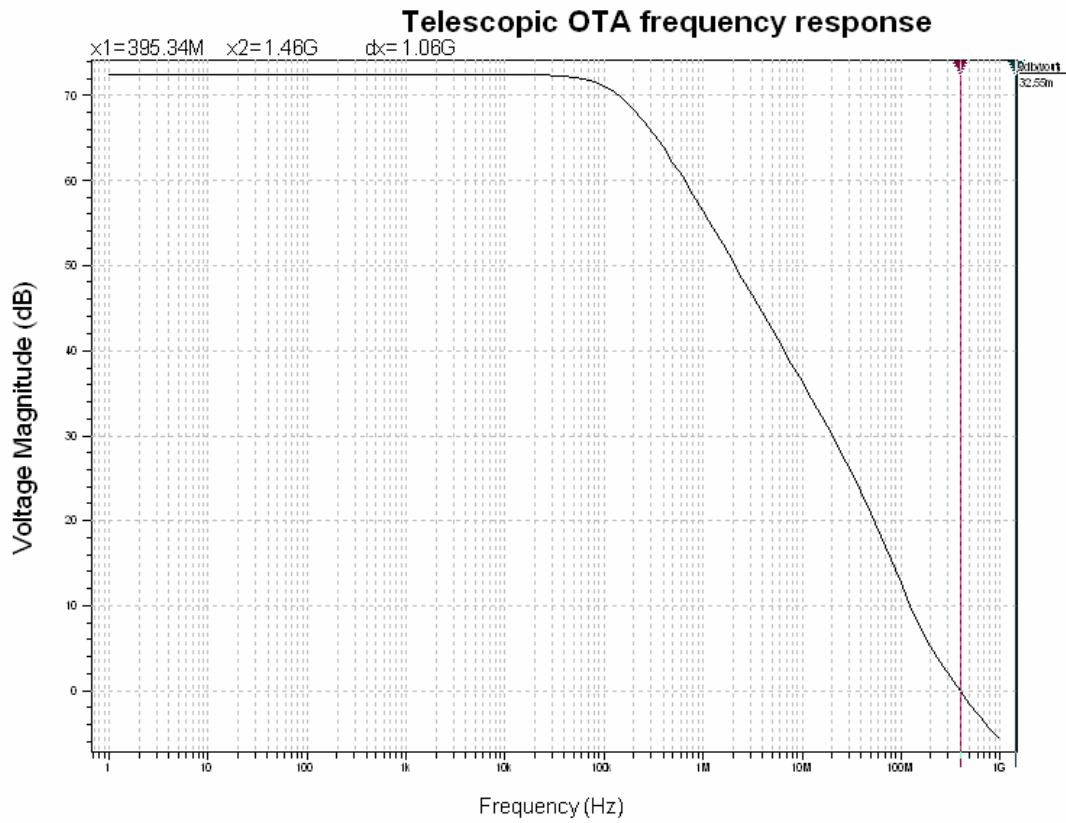


Figure 5.8. Frequency response of the Telescopic OTA with constant current source at its tail

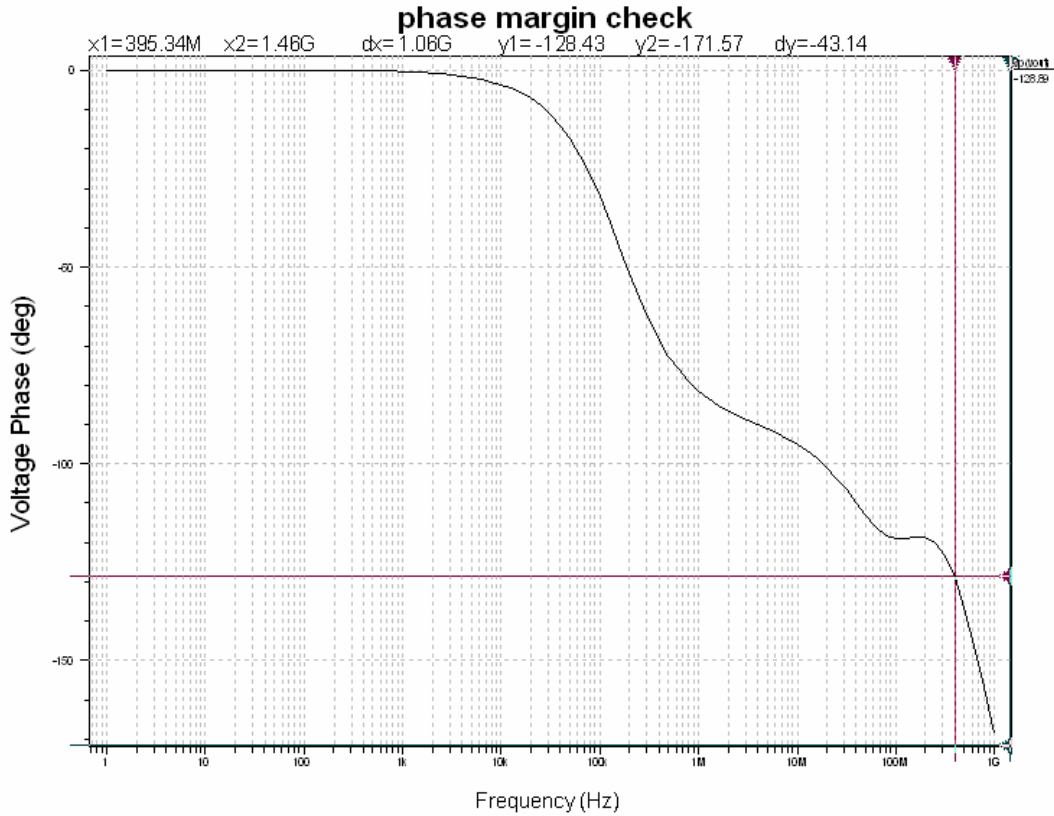


Figure 5.9 Phase response of the Telescopic OTA with constant current source at its tail

The simulation results are as follows:

- DC Gain : 72.5 dB
- UGB : 395.34 MHz
- Phase Margin : 52°

The Telescopic OTA is again simulated with schematic, fig. 5.10, with bias circuit as shown in fig 5.6(a). The results are well justified as the response matches the response of the OTA with constant current source. This means the circuit is generating same current as the constant current source. The frequency response and phase response is shown in fig. 5.11 and fig. 5.12 respectively.

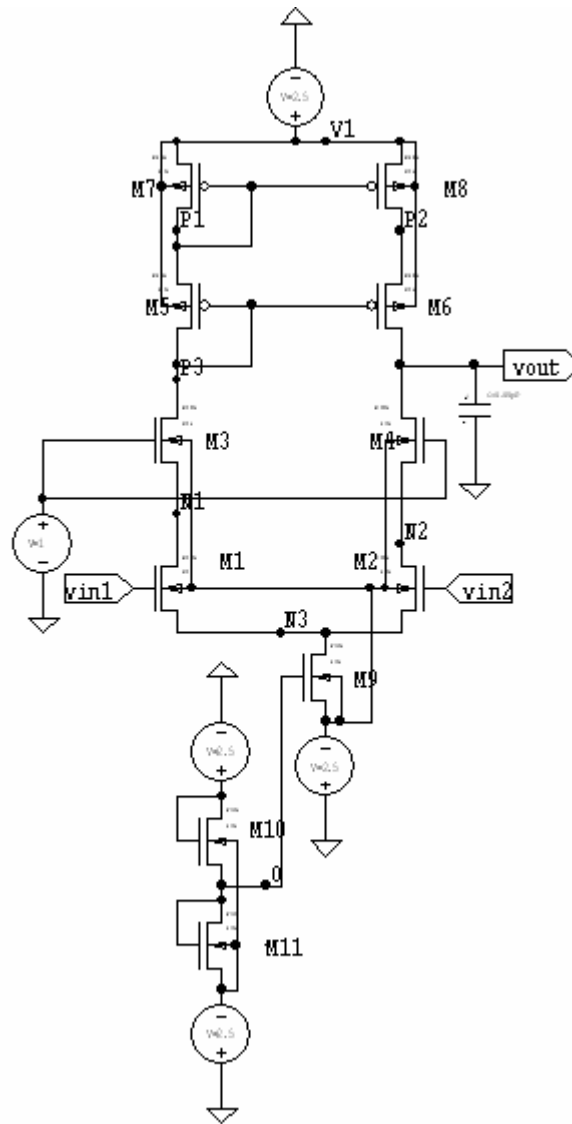


Figure 5.10. Schematic of the Telescopic OTA with bias circuit at the tail.

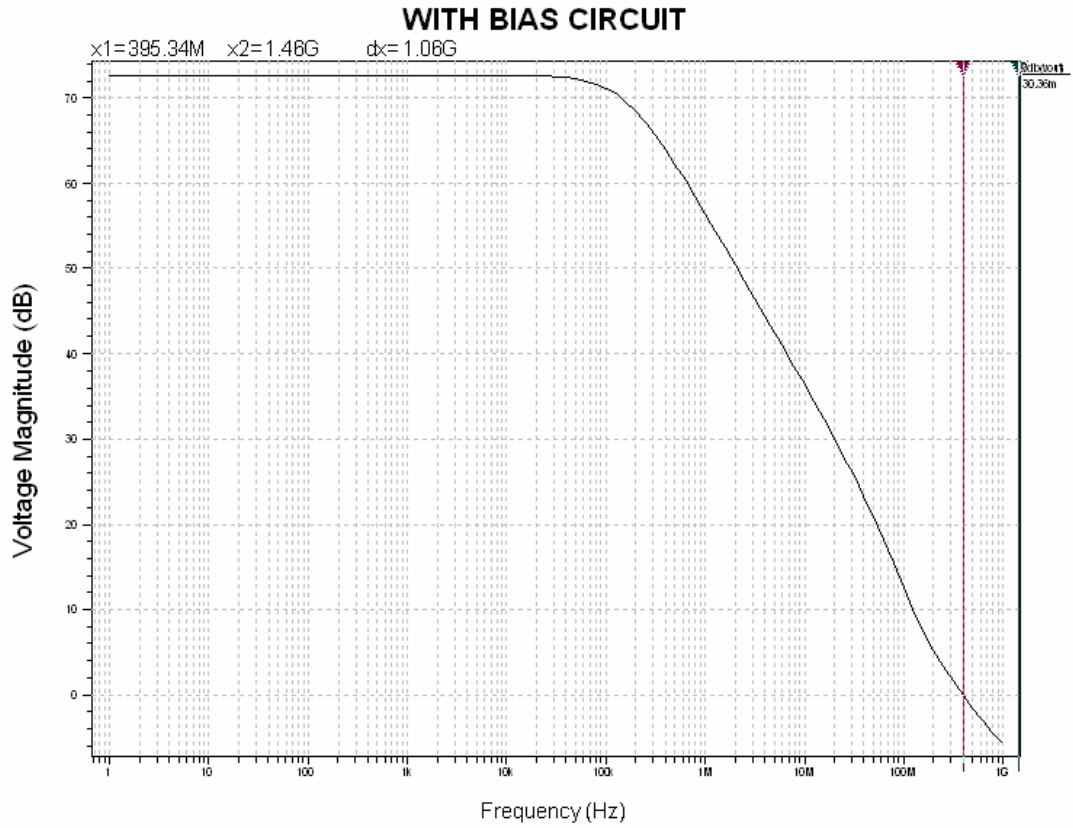


Figure 5.11. Frequency response of the Telescopic OTA with bias circuit at its tail

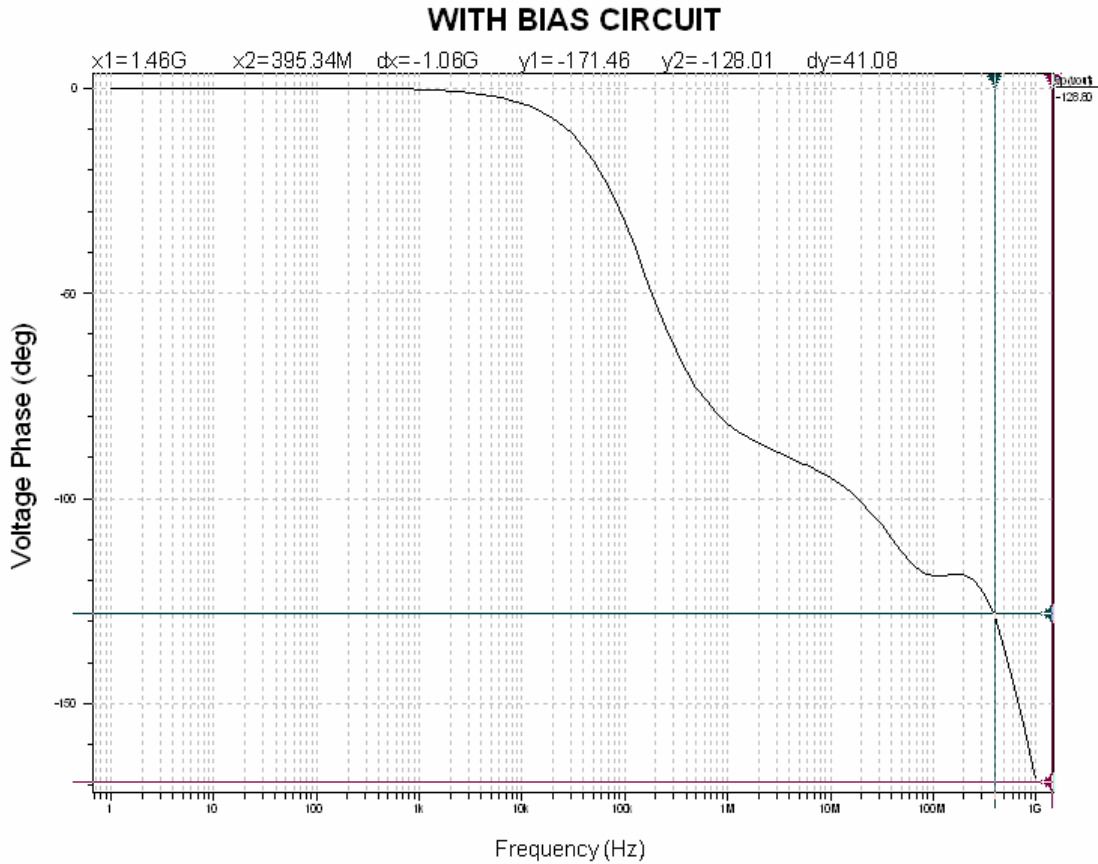


Figure 5.12 Phase response of the Telescopic OTA with bias circuit at its tail

The simulation results of the Telescopic OTA with bias circuit at the tail are as follows:

DC Gain : 72.5 dB  
 UGB : 395.34 MHz  
 Phase Margin : 52°

Thus, the results are in compliance with the previous results.

### 5.5.4 Analysis of the Simulation Results

During simulations, fine-tuning of the transistor dimensions has been done to meet the desired specifications.

1. It is observed that by reducing the  $C_L$  further leads to increase in UGB. But it leads to system instability as shown by phase margin.

2. The input offset voltage has been adjusted to keep transistors in saturation region.
3. The response curves of the Telescopic OTA with constant current source at the tail in place of transistor M9 and with transistor M9 with its gate fed in by bias circuit, are giving same results. This proves that the bias applied at the gate of the tail transistor M9 leads to  $I_{SS} = 500 \mu\text{A}$ .
4. As slew rate decreases with increase in  $C_L$ , the UGB also decreases.

### **5.5.5 Physical Design of Telescopic OTA**

The physical design of OTA is shown in fig. 5.13. The layout is drawn using L-Edit (Tanner Tool). While designing layout special analog layout methods like fingering, centroid geometry etc have been used to take care of matching issues and other related problems. Interdigitization and common centroid techniques are used for matched transistors and fingered structures are used for large sized transistors. The UGB of the frequency response of the layout of Telescopic OTA shows a fall because of the high capacitance appearing at the load.

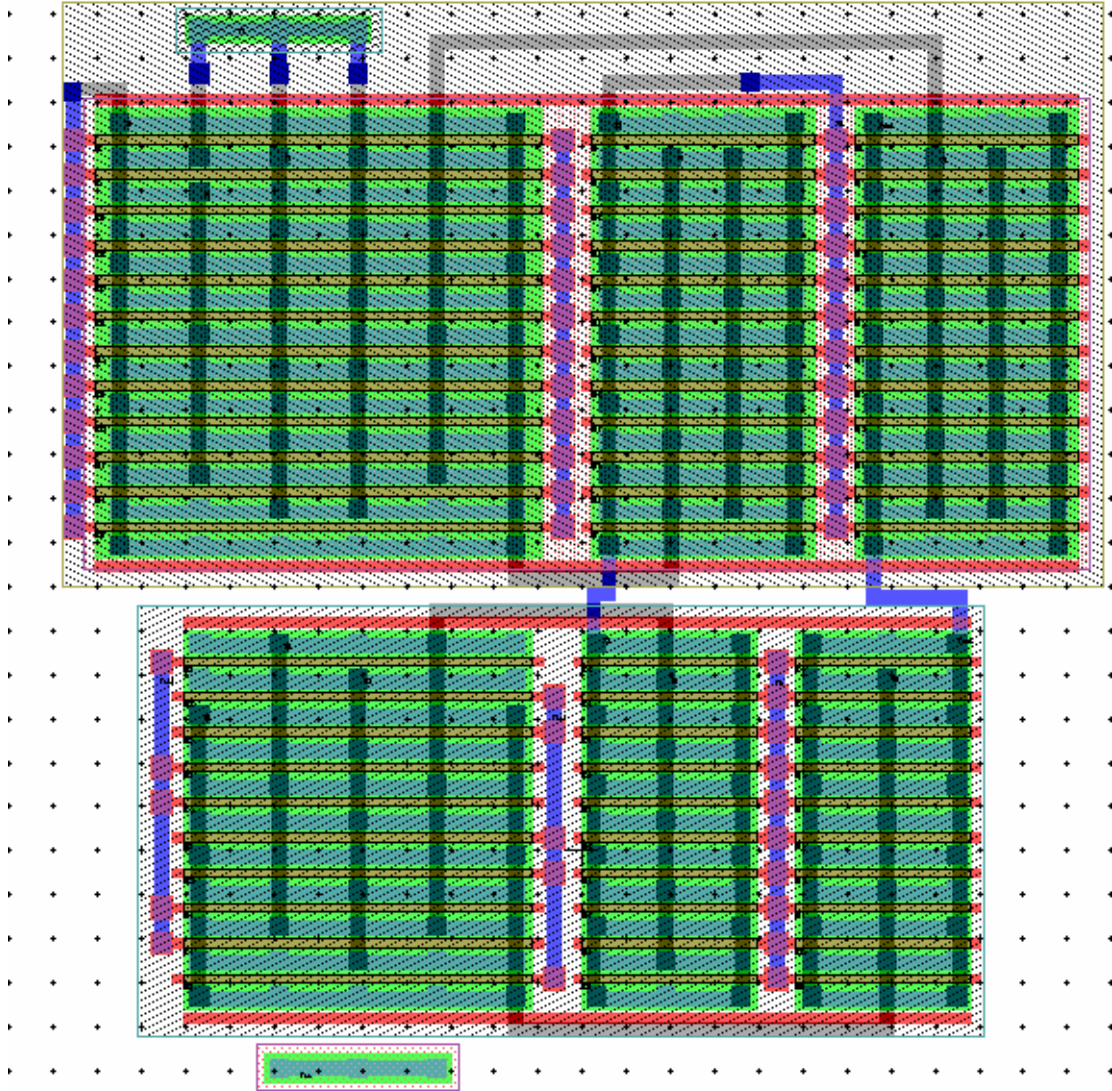


Figure 5.13. Layout of Telescopic OTA

### Telescopic OTA layout response

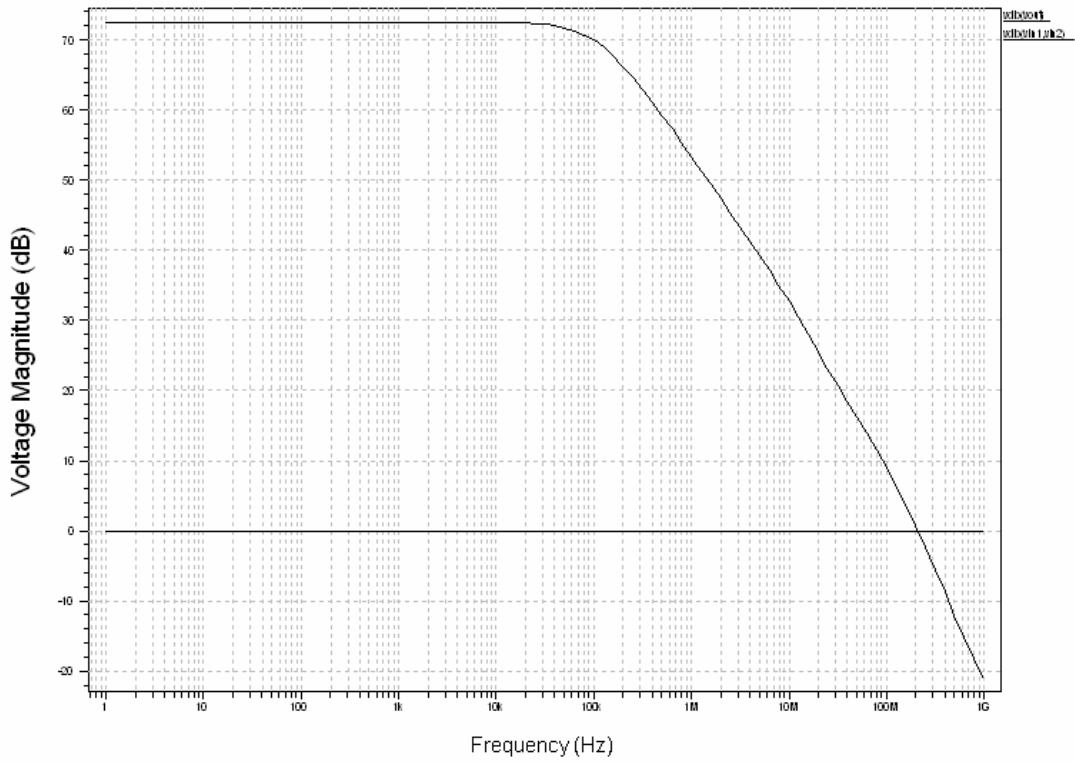


Figure 5.14. Frequency response of layout of Telescopic OTA

# **Chapter 6**

## **ACTIVE-RC BUTTERWORTH FILTERS: DESIGN AND ANALYSIS**

### **6.1 Introduction**

This chapter throws light on the reason behind study on Active –RC Butterworth filter. The design steps for the design of Active-RC Butterworth filter are discussed later in this chapter. Further, design of 6<sup>th</sup> order active-RC Butterworth filter is explained with simulations and results. The layout of the filter is also made and simulated. The different order of filters are designed, simulated and analyzed in terms of roll-off rates, passband frequency and power dissipation.

### **6.2 Active-RC Butterworth Filter**

The oldest technology of realizing filters makes use of inductors and capacitors, and the resulting circuits are called passive LC filters. Such filters work well at high frequencies; however, in low frequency applications (DC to 100 kHz) the required inductors are large and physically bulky, and their characteristics are quite non-ideal. Furthermore, such inductors are impossible to fabricate in monolithic form and are compatible with any of the modern techniques for assembling electronic systems. Therefore, there has been considerable interest in finding filter realizations that do not require inductors. Of the various possible types of inductorless filters, Active-RC filters and switched capacitor filters are most common. Later in this chapter Active-RC filter design has been discussed. Active-RC filters utilize opamps together with resistors and capacitors and are fabricated using discrete, hybrid thick-film, or hybrid thin-film technology. However for large volume production, such technologies do not yield the economies achieved by monolithic IC fabrication [2].

The active-RC filters can be used in channel selection filtering in W-CDMA applications [17]. Active-RC filter is often selected as anti-aliasing filter due to its

high linearity and low power, among which Sallen-key filter is popular because it is easy to analyze and design [14]. So, an active-RC filter may contain an OTA plus RC components. The simplest Active-RC Butterworth filter (1<sup>st</sup> order) is shown in fig. 6.1. The Active-RC Butterworth filter has range of advantages when used for lower order of the filter:

1. Active filters have excellent linearity
2. These filters have low power dissipation
3. These filters are easy to design and analyze.
4. The response of these filters is insensitive to parasitics.
5. Active-RC Butterworth filters have large Dynamic range.

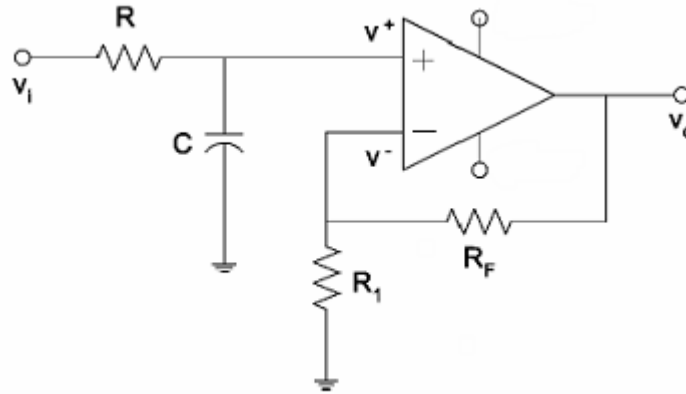


Figure 6.1. 1<sup>st</sup> order opamp based Active-RC filter

### 6.3 Design steps to design Active-RC Butterworth filter

The steps have been made to design an active-RC Butterworth filter. These are as follows:

1. An active device is chosen for the filter. It may be an opamp, OTA or DDA (Differential Difference Amplifier).

2. The specifications for the design of active device are laid down keeping an emphasis on the UGB, power dissipation and the supply voltages used. The design steps for OTA have been mentioned in Chapter 5.
3. Design the active device in accordance with the specifications. Simulate the schematic on the simulation tool and check its performance.
4. Lay down specifications for filter. The filter design process begins with the filter user specifying the transmission characteristics required for the filter. The transmission of a low pass filter is specified by four parameters:
  - a) The passband edge  $\omega_s$ .
  - b) The maximum allowed variation in passband transmission  $A_{max}$ .
  - c) The stopband edge  $\omega_p$ .
  - d) The minimum required stopband attenuation,  $A_{min}$ .

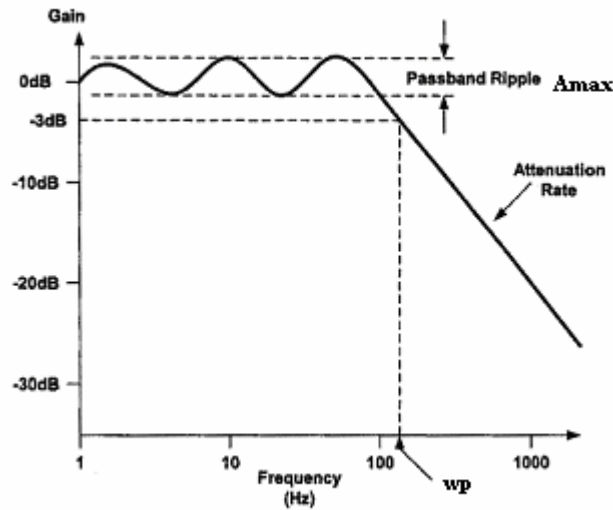


Figure 6.2. Low pass filter response

5. Calculate for which order of filter does the design specifications suit. For the Butterworth filter the order can be calculated if the four parameters mentioned above are given. Transfer function of the Butterworth filter can be calculated subsequently.

The Butterworth low-pass filter exhibits a monotonically decreasing transmission zeros at  $\omega = \infty$ , making it an all pole filter. The magnitude function for an Nth order Butterworth filter with a passband edge  $\omega_p$  is given by

$$|T(j\omega)| = \frac{1}{\sqrt{1 + \varepsilon^2 \left(\frac{\omega}{\omega_p}\right)^{2N}}} \dots\dots\dots(6.1)$$

At  $\omega = \omega_p$

$$|T(j\omega_p)| = \frac{1}{\sqrt{1 + \varepsilon^2}} \dots\dots\dots(6.2)$$

Thus, the parameter  $\varepsilon$  determines the maximum variation in passband transmission,  $A_{\max}$ , according to

$$A_{\max} = 20 \log \sqrt{1 + \varepsilon^2} \dots\dots\dots(6.3)$$

The determination of order as per the specifications and the transfer function can be in following steps:

a) Determination of  $\varepsilon$ .

$$\varepsilon = \sqrt{10^{A_{\max}/10} - 1} \dots\dots\dots(6.4)$$

It can be seen that the first  $2N-1$  derivatives of  $|T|$  relative to  $\omega$  are zero at  $\omega=0$ . This property makes the Butterworth response very flat near  $\omega=0$  and gives the response the name maximally flat response. The degree of passband flatness increases as the order  $N$  is increased.

b) Determine the filter order,  $N$ .

At the edge of the stopband  $\omega = \omega_s$ , the attenuation of the Butterworth filter is given by

$$A(\omega_s) = -20 \log \left[ \frac{1}{\sqrt{1 + \varepsilon^2 (\omega_s / \omega_p)^{2N}}} \right] = 10 \log [1 + \varepsilon^2 (\omega_s / \omega_p)^{2N}] \dots\dots\dots(6.5)$$

This equation can be used to determine the filter order required, which is the lowest integer value of  $N$  that yields  $A(\omega_s) \geq A_{\min}$ .

d) Determine the  $N$  natural modes.

The natural modes of  $N$ th order Butterworth filter can be determined from the graphical construction. The graphical modes lie on a circle of radius  $\omega_p(1/\varepsilon)^{1/N}$

and are spaced by equal angles of  $\pi/N$ , with the first mode at an angle  $\pi/2N$  from the  $+j\omega$  axis, as shown in fig. 6.3. Since the natural modes all have equal radial distance from the origin they all have the same frequency  $\omega_0 = \omega_p(1/\varepsilon)^{1/N}$ .



**1. Selection of the active device.** For the Active-RC filter design first step is to select the active component. For the filter. DDA, OTA and opamps are the few options. OTA has been chosen as the active device for this design.

The conventional operational amplifier (op amp) is used as the active device in the vast majority of the active filter literature. For design purposes, the assumption that the op amp is ideal  $A_V = \infty$ ,  $R_{in} = \infty$ ,  $R_0 = 0$  is generally made, and large amounts of feedback are used to make the filter gain essentially independent of the gain of the op amp. A host of practical filter designs have evolved following this approach. It has also become apparent, however, that operational amplifier limitations preclude the use of these filters at high frequencies, attempts to integrate these filters have been unsuccessful (with the exception of a few non-demanding applications), and convenient voltage or current control schemes for externally adjusting the filter characteristics do not exist [20].

But OTAs offer improvements in design simplicity and programmability when compared to op amp based structures as well as reduced component count. Many of the basic OTA based structures use only OTAs and capacitors and, hence, are attractive for integration. Component count of these structures is often very low (e.g., second-order biquadratic filters can be constructed with two OTAs and two capacitors) when compared to VCVS designs. Convenient internal or external voltage or current control of filter characteristics is attainable with these designs.

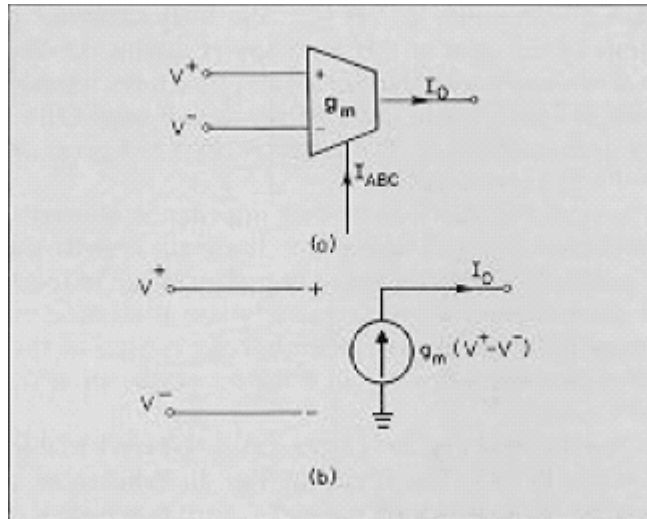


Figure 6.4 (a) OTA Symbol. (b) Equivalent circuit of ideal OTA.

The symbol used for the OTA is shown in fig. 6.4(a), along with the ideal small signal equivalent circuit. The transconductance gain,  $g_m$ , is assumed proportional to  $I_{ABC}$ . The proportionality constant  $h$  is dependent upon temperature, device geometry, and the process [5].

$$g_m = hI_{ABC} \dots\dots\dots(6.7)$$

The output current is given by

$$I_o = g_m(V^+ - V^-) \dots\dots\dots(6.8)$$

As shown in the model, the input and output impedances in the model assume ideal values of infinity. Current control of the transconductance gain can be directly obtained with control of  $I_{ABC}$ . Since techniques abound for creating a current proportional to a given voltage, voltage control of the OTA gain can also be attained through the  $I_{ABC}$  input [20].

So, OTAs are better candidates for the design of Active-RC filter as compared to opamps. DDA is also formed of two single ended OTAs.

Next, matter to be thought is which OTA should be used amongst all the topologies of the OTAs. As discussed in Chapter 5 with a comparison table (Table 5.1), telescopic OTA is a good candidate for the application of filter design because of following reasons:

- a) Noise is low in Telescopic OTA.
- b) Power dissipation is least.
- c) Speed is highest
- d) Gain is medium.

From the above points, it can be seen as a filter should have low noise, low power dissipation and high speed. Medium gain is enough for the filter design as emphasis on getting better passband is made [3].

**2. Specifications of the OTA.** The specifications for the design of active device are laid down keeping an emphasis on the UGB, power dissipation and the supply voltages used. The OTA design has been discussed in full in Chapter 5. same steps are to be followed with same specifications.

**3. Simulations of the schematic.** The design was simulated using Tanner EDA tool. The results are, DC gain = 72.5dB, UGB = 395.34 MHz and Phase margin = 52deg, as shown in Chapter 5.

**4. Specifications of filter.** The specifications have been laid down as follows:

- a) The passband edge  $\omega_s = 600\text{MHz}$ .
- b) The maximum allowed variation in passband transmission  $A_{\text{max}} = 1\text{dB}$ .
- c) The stopband edge  $\omega_p = 15\text{MHz}$ .
- d) The minimum required stopband attenuation,  $A_{\text{min}} = 180\text{dB}$ .
- e) DC Gain = 7.724

**5. Calculations to know the order of the filter.**

a) Determination of  $\epsilon$ .

$$\epsilon = \sqrt{10^{1/10} - 1} \dots\dots\dots(6.9)$$

$$\epsilon = 0.5088$$

b) Determine the filter order, N.

At the edge of the stopband  $\omega = \omega_p$ , the attenuation of the Butterworth filter is given by

$$180 = 10 \log[1 + 0.5088^2 (600/15)^{2N}] \dots\dots\dots(6.10)$$

$$(40)^{2N} = 1.965 \times 10^{18}$$

For N=5,

$$(40)^{2N} = 1.048 \times 10^{16}$$

This gives,  $A(\omega_s) = 157.27\text{dB}$

For  $N=6$

$$(40)^{2N} = 1.677 \times 10^{19}$$

This gives,  $A(\omega_s) = 189\text{dB}$

$N=6$  is the lowest integer value of  $N$  that yields  $A(\omega_s) \geq A_{\min}$ . So, filter order should be 6.

c) Determine the 6 natural modes.

The natural modes of 6<sup>th</sup> order Butterworth filter can be determined from the graphical construction. The graphical modes lie on a circle of radius  $\omega_p(1/\epsilon)^{1/N}$

and are spaced by equal angles of  $\pi/N = 30^\circ$ , with the first mode at an angle  $\pi/2N = 15^\circ$  from the  $+j\omega$  axis. Since the natural modes all have equal radial distance from the origin they all have the same frequency,

$$\omega_0 = \omega_p(1/\epsilon)^{1/N} = 16.78 \times 10^6 \text{ rad/sec} \dots \dots \dots (6.11)$$

Now, the poles are calculated as

$$p_1 = (\cos 105^\circ + j\sin 105^\circ) \times 16.78 \times 10^6 = (-4.34 + j16.2) \times 10^6$$

$$p_2 = (\cos 135^\circ + j\sin 135^\circ) \times 16.78 \times 10^6 = (-11.86 + j11.86) \times 10^6$$

$$p_3 = (\cos 165^\circ + j\sin 165^\circ) \times 16.78 \times 10^6 = (-16.2 + j4.34) \times 10^6$$

$$p_4 = (\cos 195^\circ + j\sin 195^\circ) \times 16.78 \times 10^6 = (-16.2 - j4.34) \times 10^6$$

$$p_5 = (\cos 225^\circ + j\sin 225^\circ) \times 16.78 \times 10^6 = (-11.86 - j11.86) \times 10^6$$

$$p_6 = (\cos 255^\circ + j\sin 255^\circ) \times 16.78 \times 10^6 = (-4.34 - j16.2) \times 10^6$$

So, the transfer function can now be found. The transfer function of the filter is as follows

$$T(s) = 1.72 \times 10^{44} \left[ \left( \frac{1}{(s^2 - js32.4 \times 10^6 - 281.27 \times 10^{12})} \right) \right]$$

$$\left[ \left( \frac{1}{(s^2 - js23.72 \times 10^6 - 281.31 \times 10^{12})} \right) \cdot \left( \frac{1}{s^2 - js8.68 \times 10^6 - 281.27 \times 10^{12}} \right) \right] \dots \dots \dots (6.12)$$

**6. Calculate the values of feedback resistors.** As per calculations the 6<sup>th</sup> order Butterworth filter suits the specifications defined.

For this, three 2nd order filters should be used. So, there are three gain stages to be defined. On the basis of these gain stages  $R_F/R_1$  have been calculated.

The passband gain for the three stages is 1.152, 2.235 and 3 for the subsequent stages. Thus,  $R_F/R_1$  for the three stages are calculated as follows:

$$A_F = 1 + \frac{R_F}{R_1} = \text{pass-band gain of the filter}$$

$$R_{F1}/R_{11} = 0.152$$

$$\text{For } R_{11} = 2\text{k}\Omega, R_{F1} = 304\Omega$$

$$R_{F2}/R_{12} = 1.235$$

$$\text{For } R_{12} = 4\text{k}\Omega, R_{F2} = 4.940\text{k}\Omega$$

$$R_{F3}/R_{13} = 2$$

$$\text{For } R_{13} = 3\text{k}\Omega, R_{F3} = 6\text{k}\Omega$$

- 7. Calculate the RC components.** Taking the value of capacitor as  $C = 0.005 \text{ pF}$ . The value for resistor is  $10 \text{ k}\Omega$ . There is a marginal difference taken between the desired passband frequency and calculated. This is because of the active component reduces the passband frequency.

## 6.5 Simulation results of the 6<sup>th</sup> order Butterworth filter

This section discusses the results obtained by the schematic level simulations. The schematic drawn in S-Edit module of Tanner EDA Tool and the simulations are done in T-Spice module of Tanner EDA Tool.

The schematic of the 6<sup>th</sup> order Butterworth filter is shown in fig. 6.5. The schematic shows three stages of 2<sup>nd</sup> order filters. The frequency response of the 6<sup>th</sup> order Butterworth filter is shown in fig. 6.6.

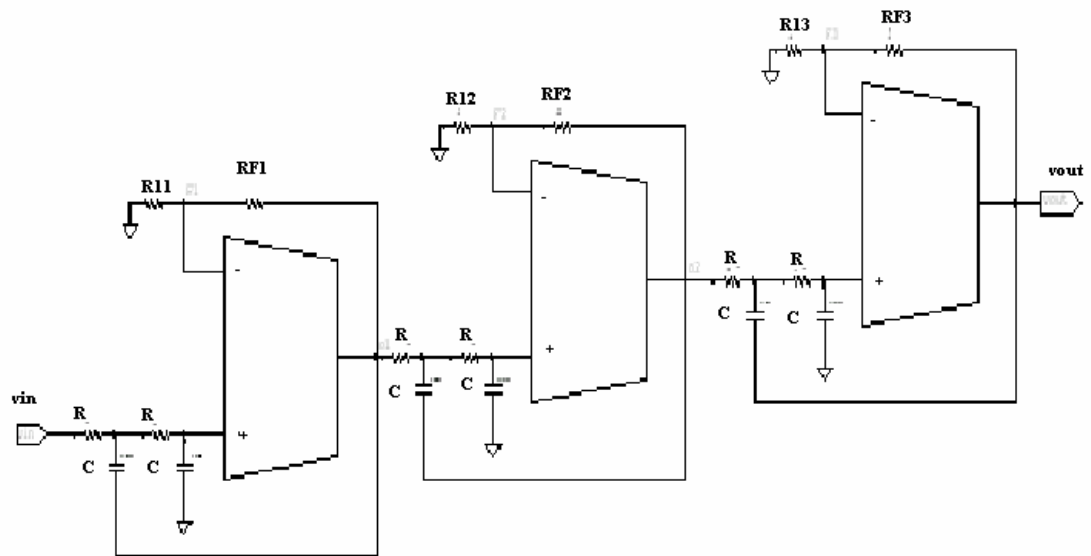


Figure 6.5. Schematic of the 6<sup>th</sup> order Active-RC Butterworth filter

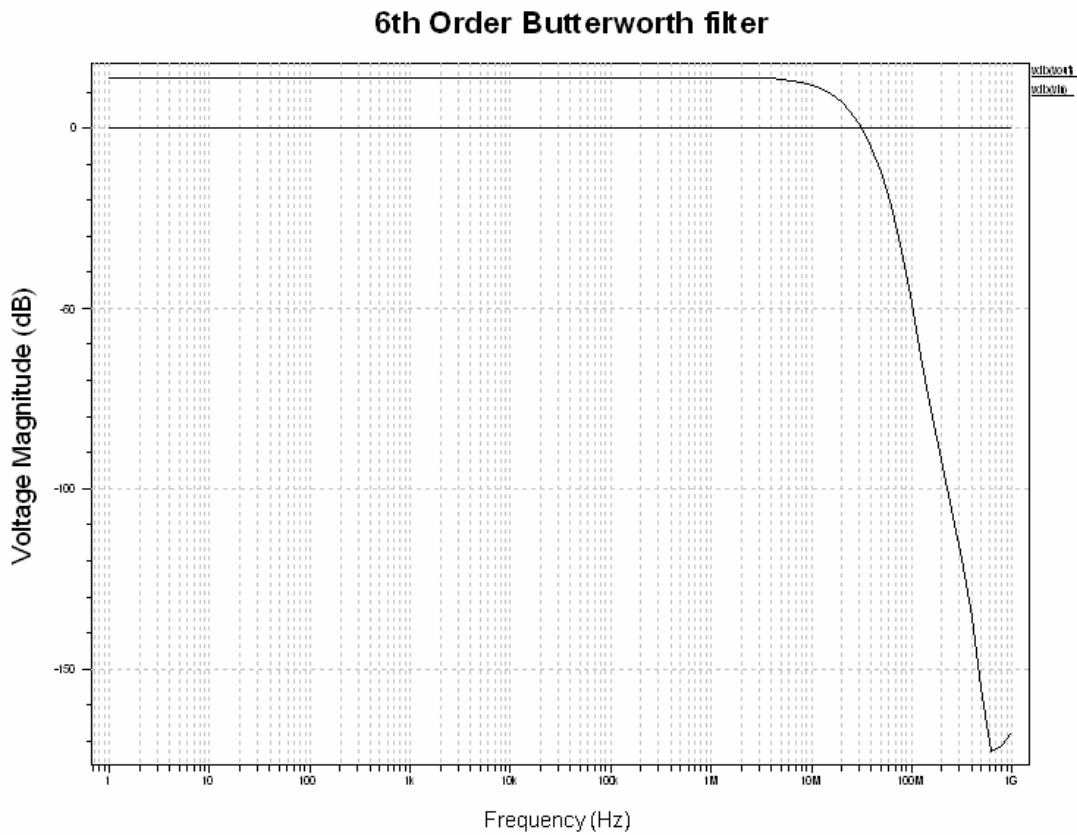


Figure 6.6. Frequency response of the 6<sup>th</sup> order low-pass Active-RC Butterworth filter

The simulation results are as follows:

DC Gain = 13.94 dB

Passband frequency,  $\omega_p = 12.63$  MHz

Roll-off rate =  $-140$  dB/dec

### 6.5.1 Analysis of the simulation results

During simulations some components were varied to see the effects on the frequency response.

1. There is a difference between the calculated values and the simulated results.
2. No zero has been observed in the frequency response. Zero is at infinite frequency in case of Butterworth filter. So, the response has been justified.
3. When the  $R_F/R_1$  has been defined as a ratio. If the values of  $R_F$  and  $R_1$  were varied in same ratio but with different values it affects the gain with little variation.
4. Decreasing the value of RC shows an increase in the passband frequency.
5. Change in values of RC does not affect the DC gain. RC governs the passband frequency and  $R_F/R_1$  decides the passband gain.

### 6.5.2 Physical design of the 6<sup>th</sup> order Active-RC Butterworth filter

The physical design of the 6<sup>th</sup> order Active-RC Butterworth filter is shown in fig. 6.7. The layout is drawn in L-Edit (Tanner Tool). While designing layout special analog layout methods like fingering, centroid geometry etc have been used to take care of matching issues and other related problems. Interdigitization and common centroid techniques are used for matched transistors. There is a little difference in the results as compared to the schematic results. This is because of the capacitive effects consideration. The frequency response of the layout is shown in fig. 6.8.

The results are as follows:

DC Gain = 13.94 dB

Passband frequency,  $\omega_p = 15$  MHz

Roll-off rate =  $-131$  db/dec

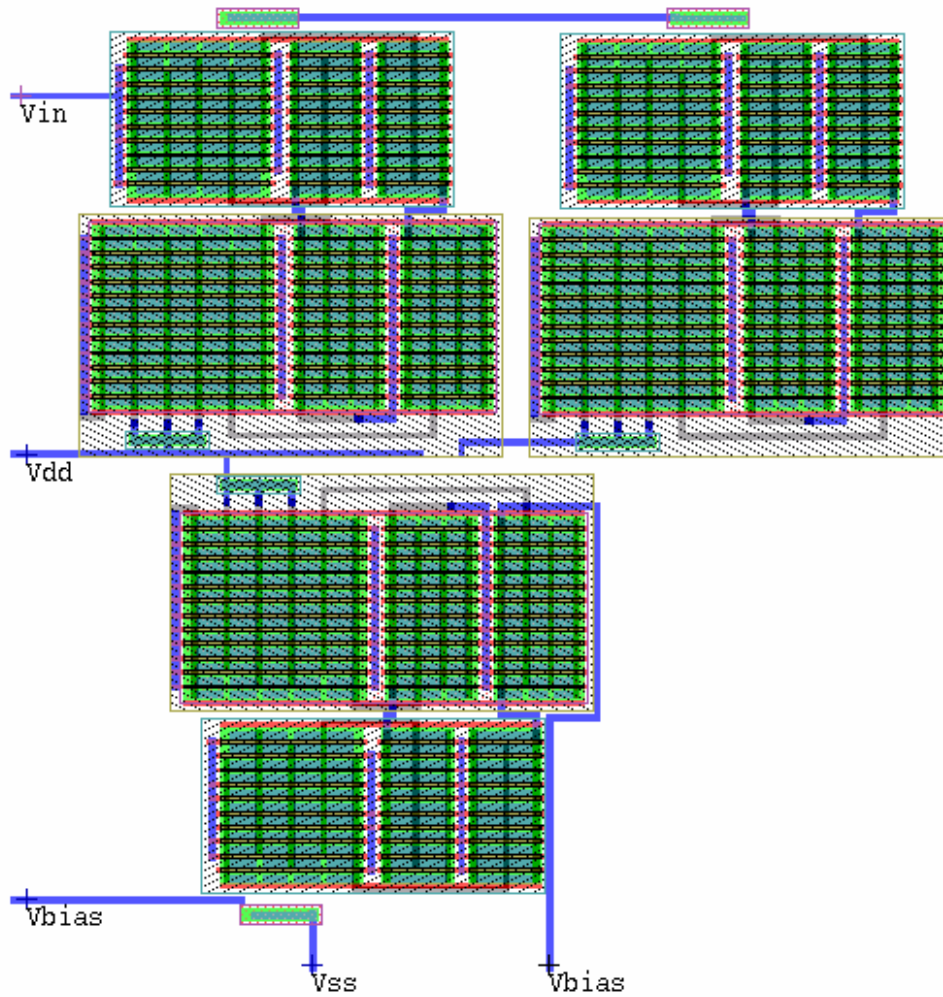


Figure 6.7. Layout of the 6<sup>th</sup> order Active-RC Butterworth filter

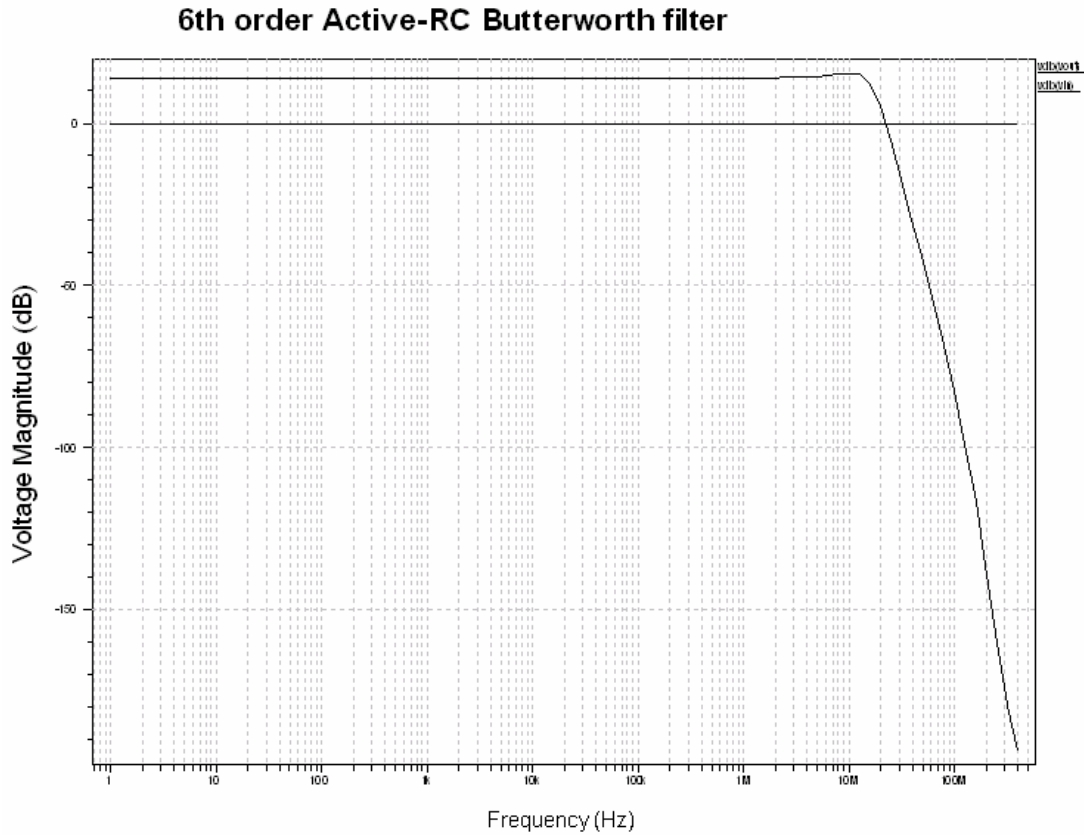


Figure 6.8. Frequency response of the 6<sup>th</sup> order Active-RC Butterworth Filter

## 6.6 Analysis of different orders of Active-RC Butterworth Filter

With the increase in filter order various parameters of the filter changes. As filter order of the Active-RC filter increases, the roll-off rate of the filter response approaches to the brick-wall response i.e. it approaches to the response of ideal filter

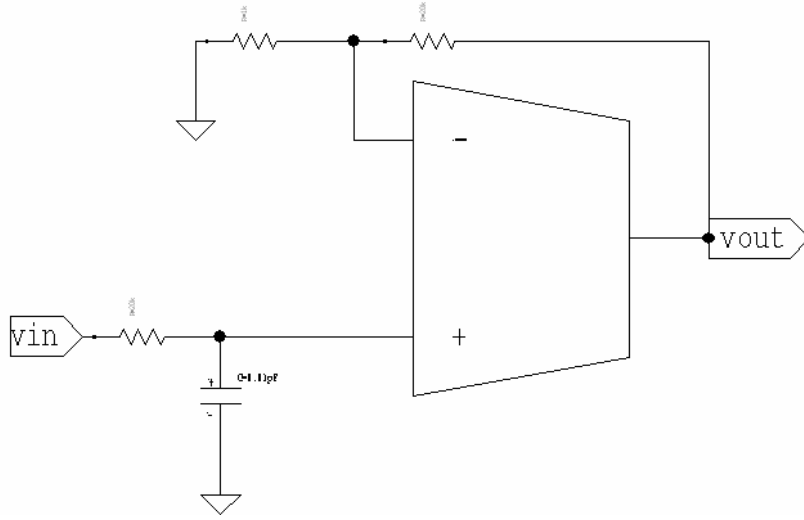
[2]. The number of components also increases with the increase in the order of the Active-RC filter [19]. As active device used in the filter decreases the filter passband frequency, the increase in active components, with increase in filter order decreases the passband frequency [2].

To analyze the above said changes in the filter parameters with increase in filter order an analysis is done starting from the 1<sup>st</sup> order Active-RC Butterworth filter to 6<sup>th</sup> order Active-RC Butterworth filter. This also shows that with the help of the design procedure laid down in previous sections, designing for other filter orders can be done easily and quickly.

Different filter specifications have been taken to analyze different order of Active-RC Butterworth filter. The OTA used is of same specification as for the 6<sup>th</sup> order Active-RC Butterworth filter. The schematic and corresponding frequency response curves of the 1<sup>st</sup> order to 5<sup>th</sup> order filters are shown in fig. 6.9 to fig. 6.13. The ratios of  $R_F/R_1$  for different orders are tabulated in Table 6.2. The response of the 6<sup>th</sup> order filter is shown in fig. 6.6. The simulation results have been compiled and shown as below in Table 6.2.

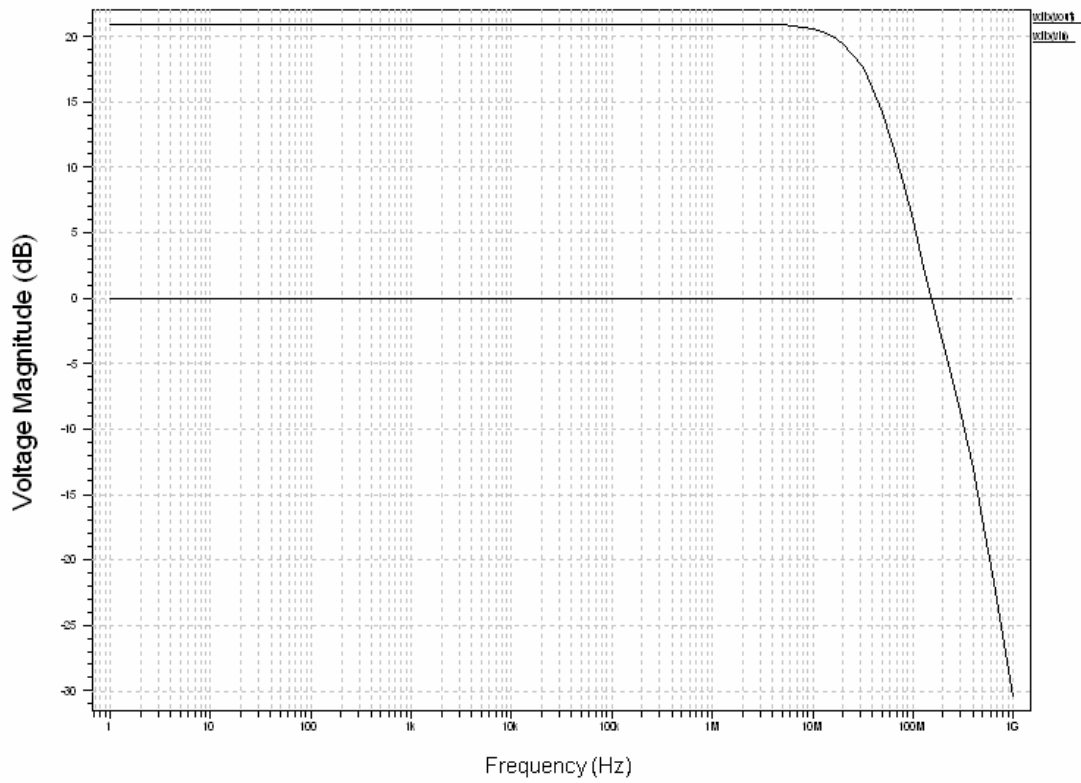
Table 6.1 Ratios of  $R_F/R_1$  for different stages in different orders of filters

Filter Order	$R_{F1}/R_{11}$	$R_{F2}/R_{12}$	$R_{F3}/R_{13}$
1	1		
2	0.586		
3	0.152	1.235	
4	0.152	1.25	
5	1	0.152	1.25
6	1.152	1.235	2



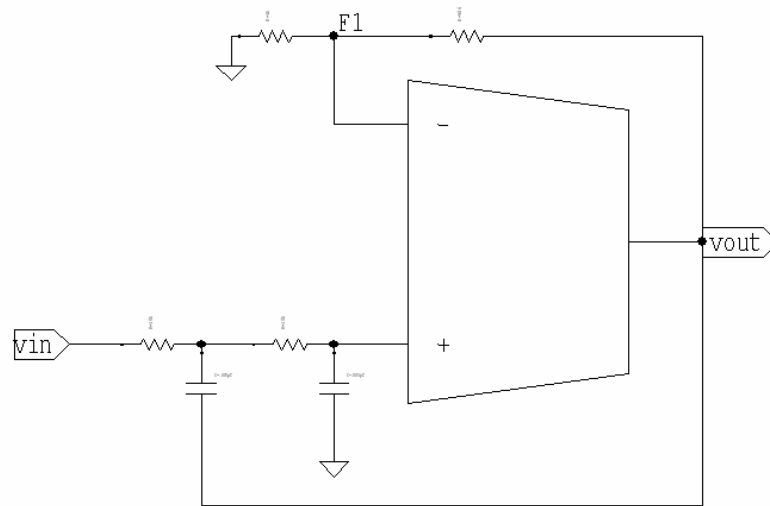
(a)

**1st order**



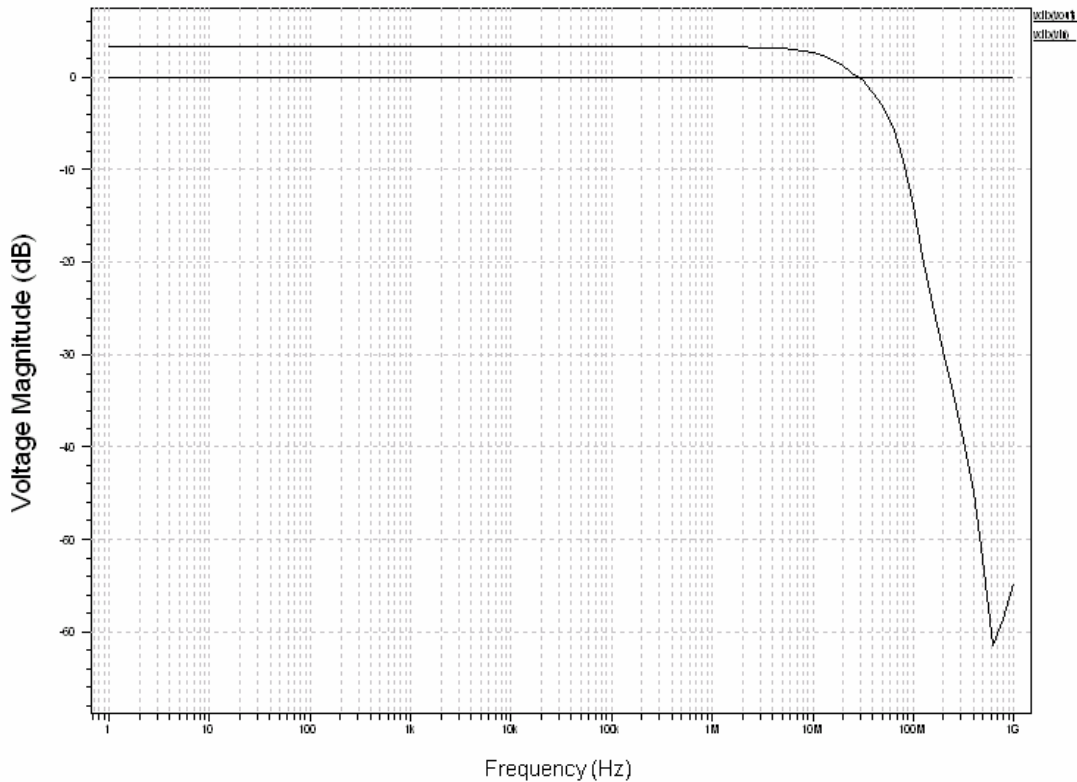
(b)

Figure 6.9 (a) Schematic of the 1<sup>st</sup> order Active-RC Butterworth filter. (b) Frequency response of the 1<sup>st</sup> order Active-RC Butterworth Filter



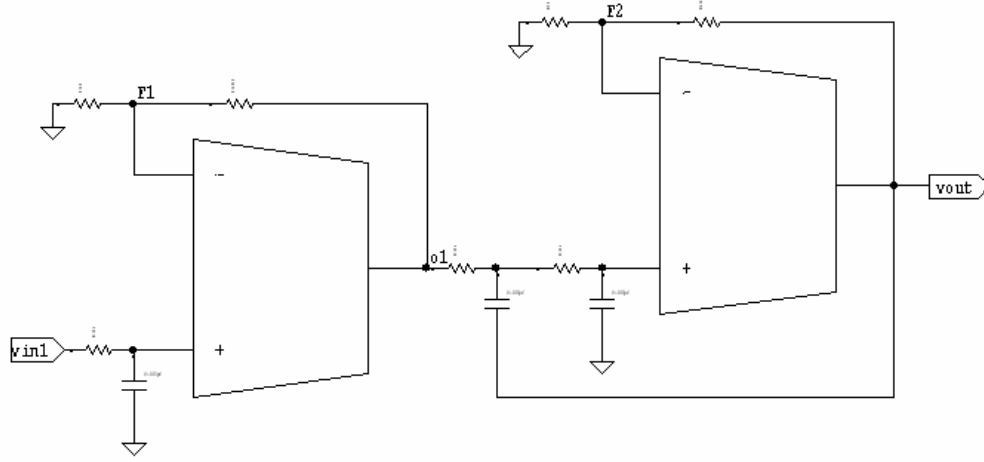
(a)

**2nd order**

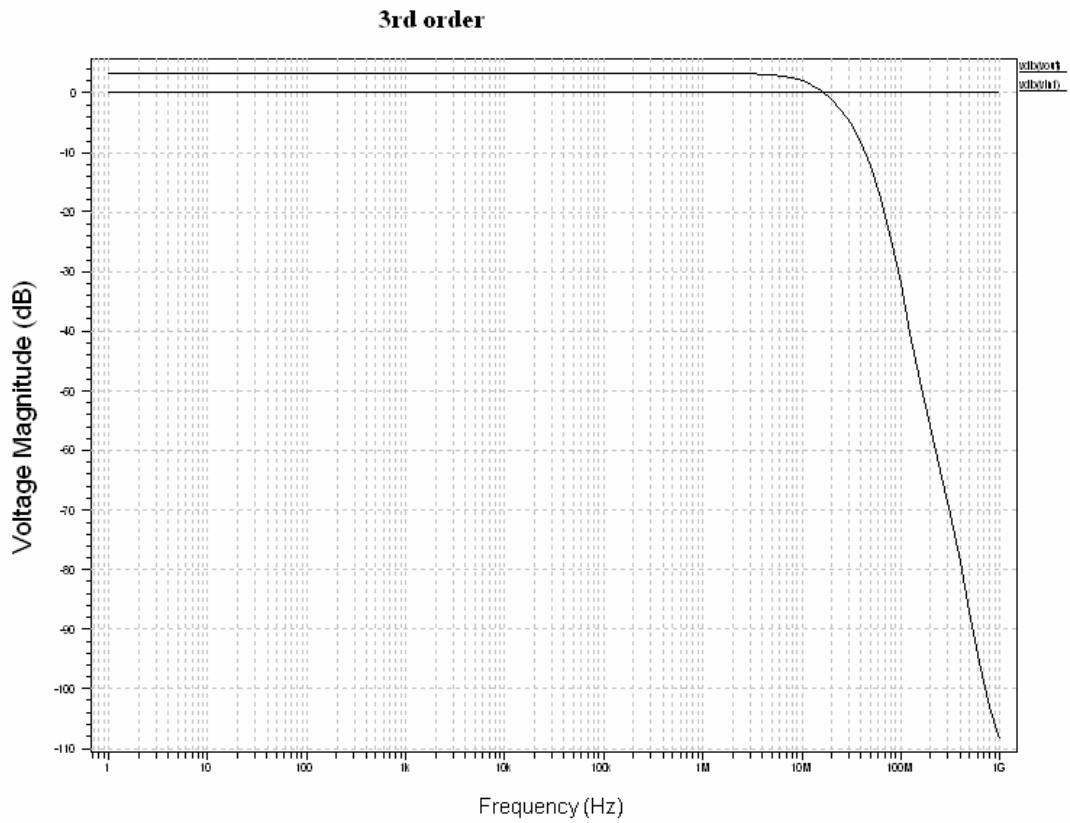


(b)

) Figure 6.10 (a) Schematic of 2<sup>nd</sup> order Active-RC Butterworth filter. (b) Frequency response of the 2<sup>nd</sup> order Active-RC Butterworth Filter

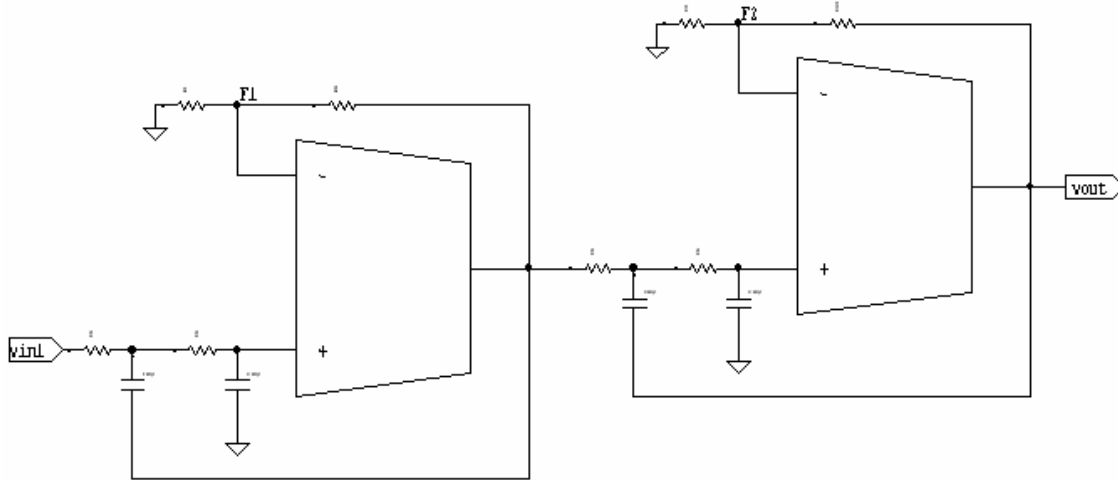


(a)



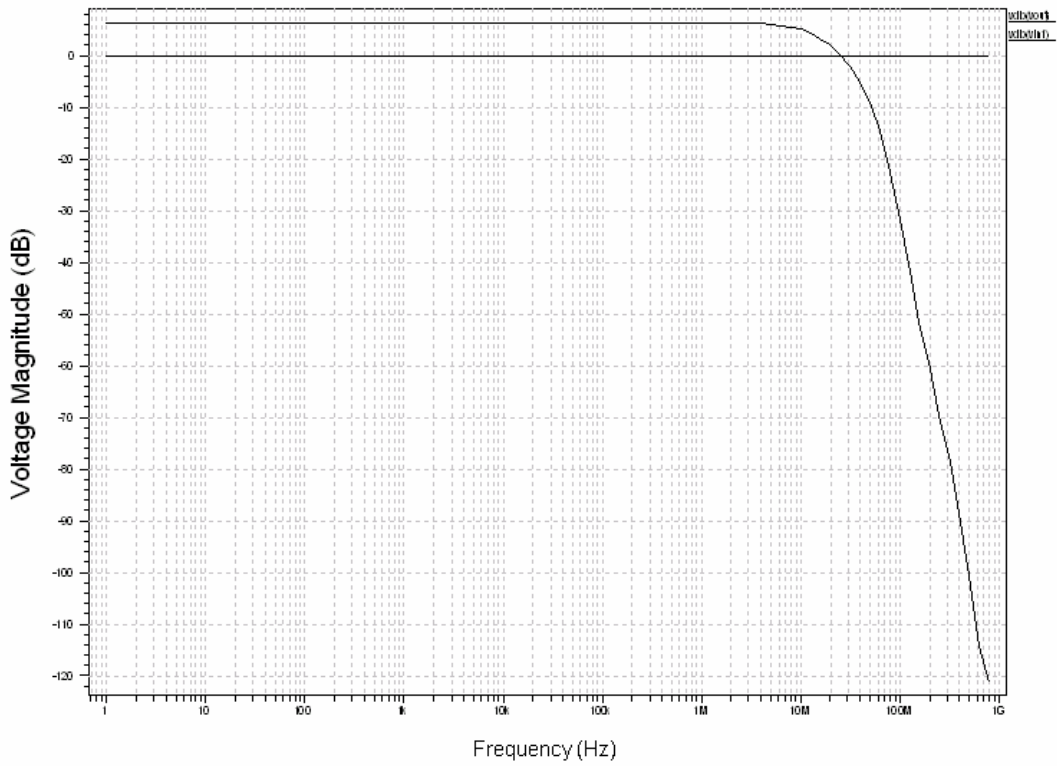
(b)

Figure 6.11 (a) Schematic of 3<sup>rd</sup> order Active-RC Butterworth filter. (b) Frequency response of the 3<sup>rd</sup> order Active-RC Butterworth Filter.



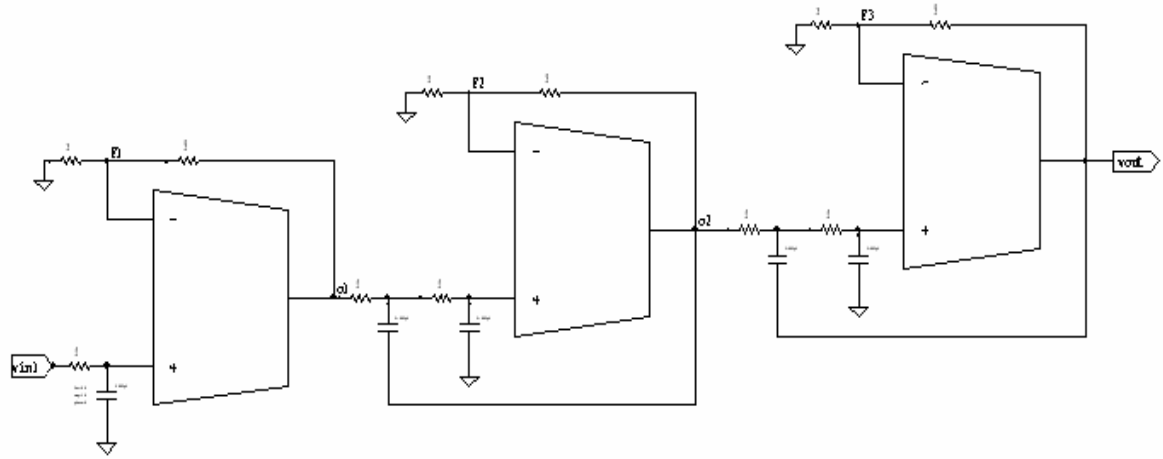
(a)

**4th order**



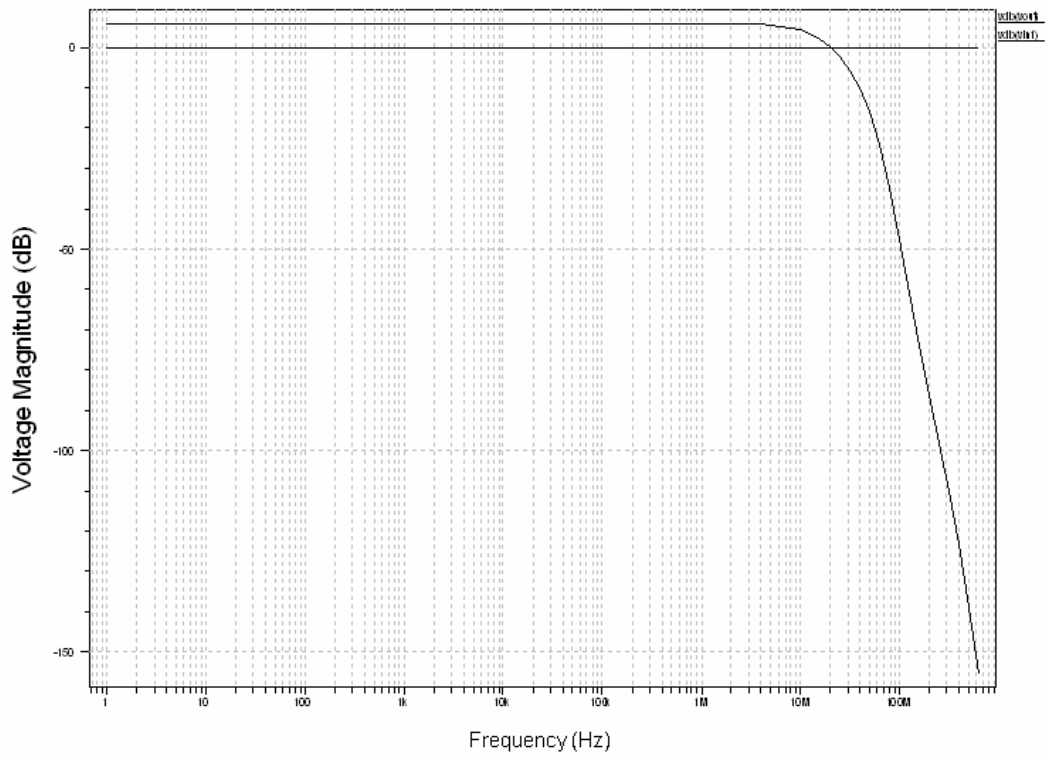
(b)

Figure 6.12 (a) Schematic of 4<sup>th</sup> order Active-RC Butterworth filter. (b) Frequency response of the 4<sup>th</sup> order Active-RC Butterworth Filter



(a)

**5th order**



(b)

Figure 6.13 (a) Schematic of 5<sup>th</sup> order Active-RC Butterworth filter. (b) Frequency response of the 5<sup>th</sup> order Active-RC Butterworth Filter

Table 6.2 Change in filter parameters with increase in filter order

Filter Order	Roll-off rate (db/dec)	$\omega_p$ (MHz)	$\omega_s$ (MHz)	$A_{min}$ simulated (dB)	$A_{min}$ calculated (dB)	Power Dissipation (mW)	Passband gain (dB)
1	-25.08	29.26	1000	50	29	2.50076	20.94
2	-48.94	26.28	536.09	55	49.5	2.5013	3.27
3	-75	17.29	709.58	100	93.85	5.00155	3.34
4	-92.33	15.06	600	110	125.09	5.00165	6.28
5	-130	13.34	593.37	150	161	7.50132	6.00
6	-140	12.63	533.22	180	192	7.50152	13.94

The response curves of the filter frequency response have been analyzed and few parameters have been computed like  $A_{min}$ , to justify the design of the filters of different orders. The error in the calculated and simulated values of  $A_{min}$  is shown with the help of a graph (fig. 6.17). The fashion in which roll-off rate, passband frequency and power dissipation changes, is shown with the help of the graph by taking the simulated values shown in Table 6.2.

In fig. 6.14 the comparison of the theoretical values of roll-off rates with the simulated values is shown in Table 6.2. As filter order is increasing, roll-off rate is also increasing with a minor deviation from the theoretical values.

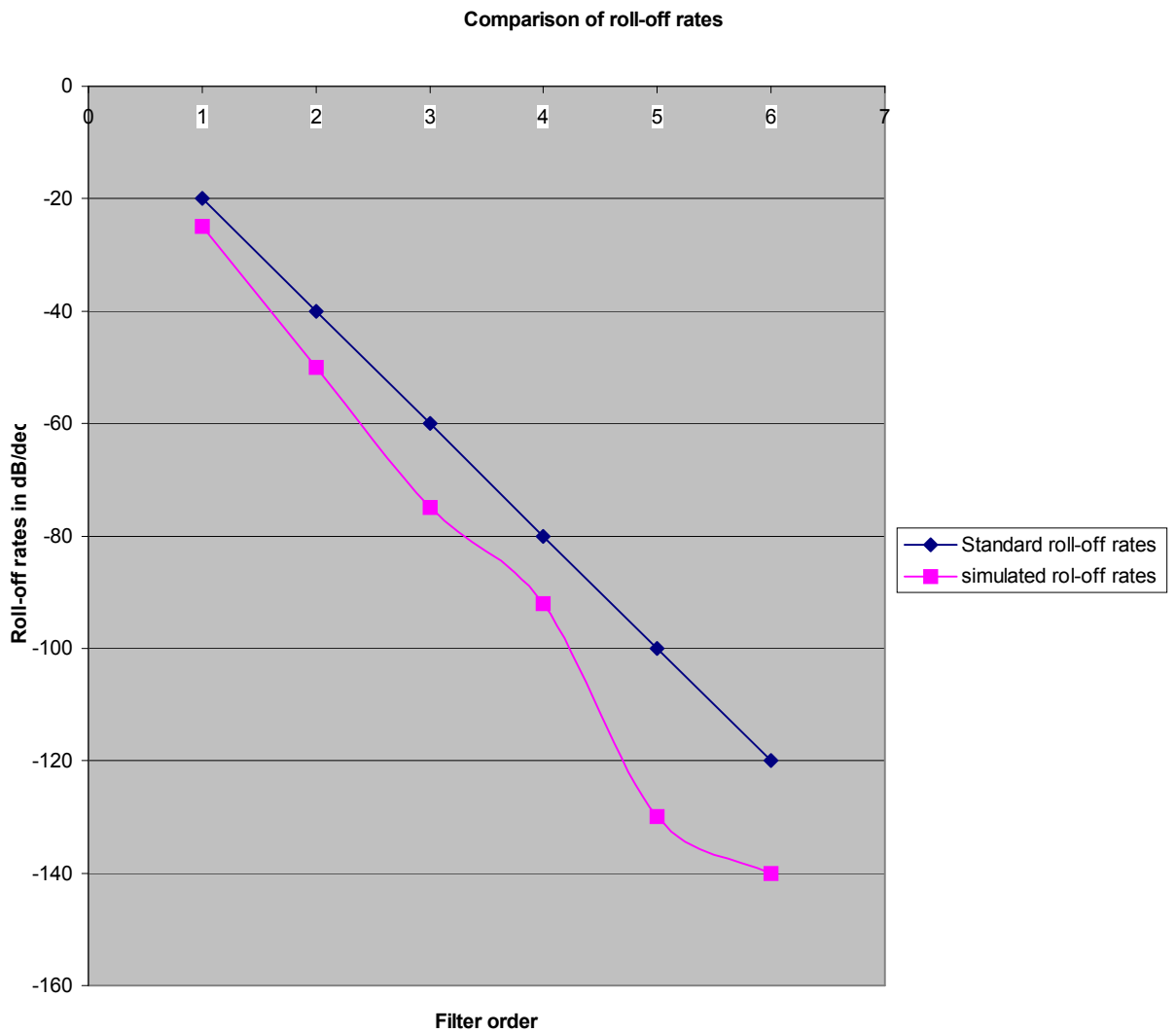


Figure 6.14 Comparison of roll-off rates of simulated and theoretical values

With the increase in the filter order the passband frequency reduces as shown in the fig. 6.15. The graph is drawn for the values shown in Table 6.2.

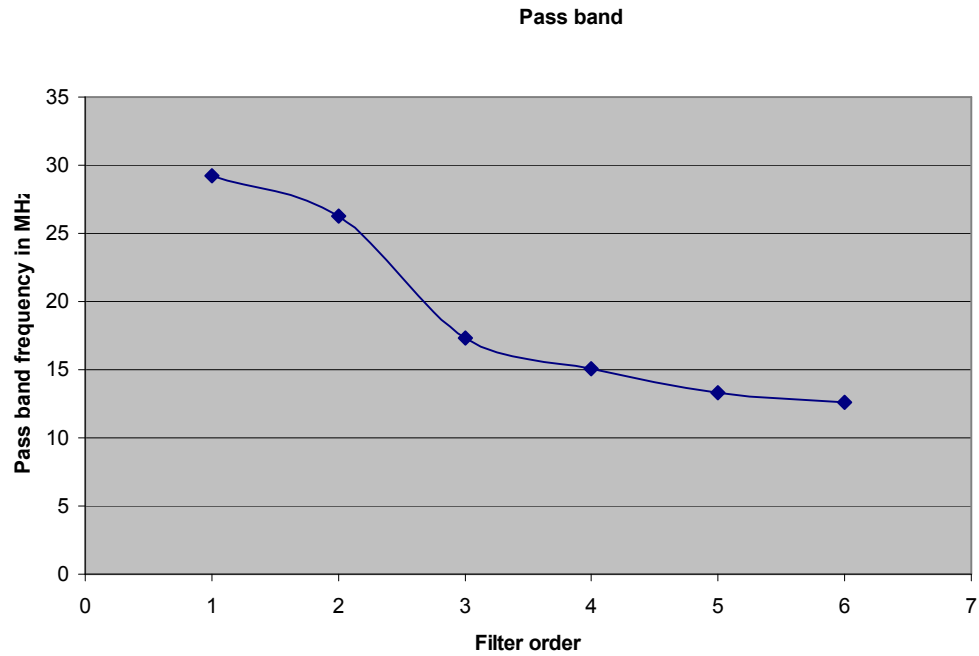


Figure 6.15 Comparison of roll-off rates of simulated and theoretical values

The number of components increases with increase of the filter order. The values have been taken from Table 6.2. The increase in power dissipation with the increase in filter order is shown in fig. 6.16.

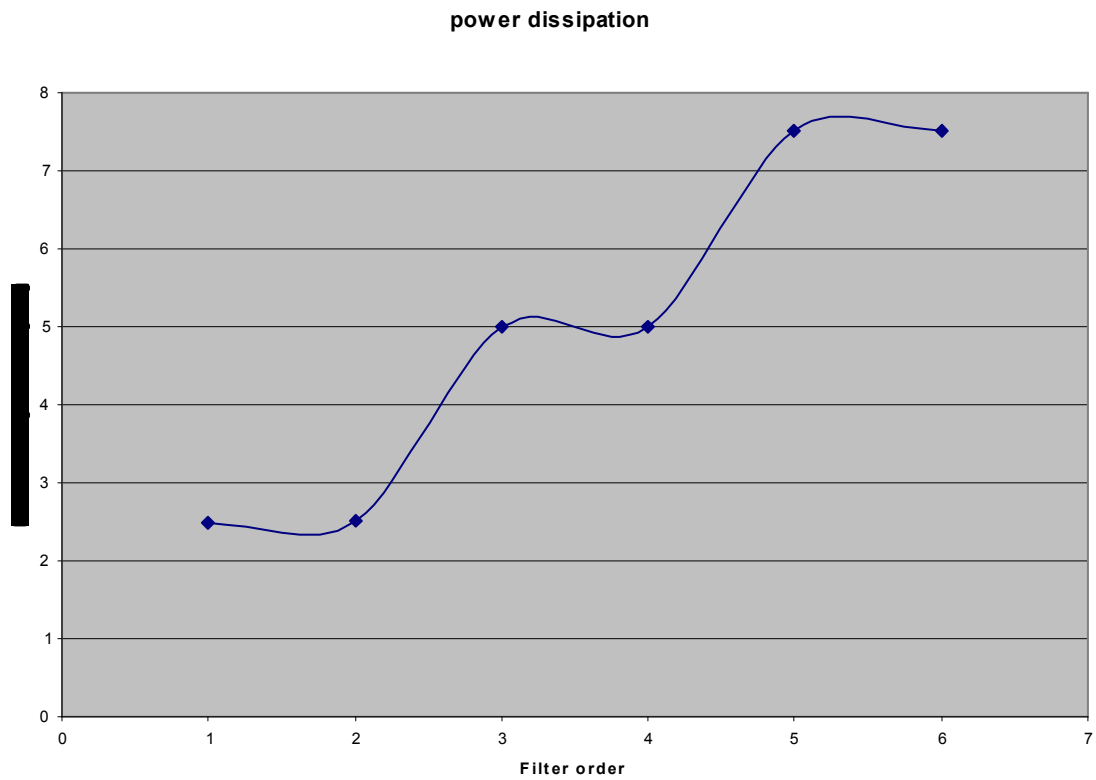


Figure. 6.16 Increase in power dissipation with increase in filter order.

In Table 6.2, calculated and simulated results of  $A_{\min}$  have been listed down. There is a little difference observed in simulated and calculated values. To view the difference between the two values a graph has been plot as shown in fig. 6.17.

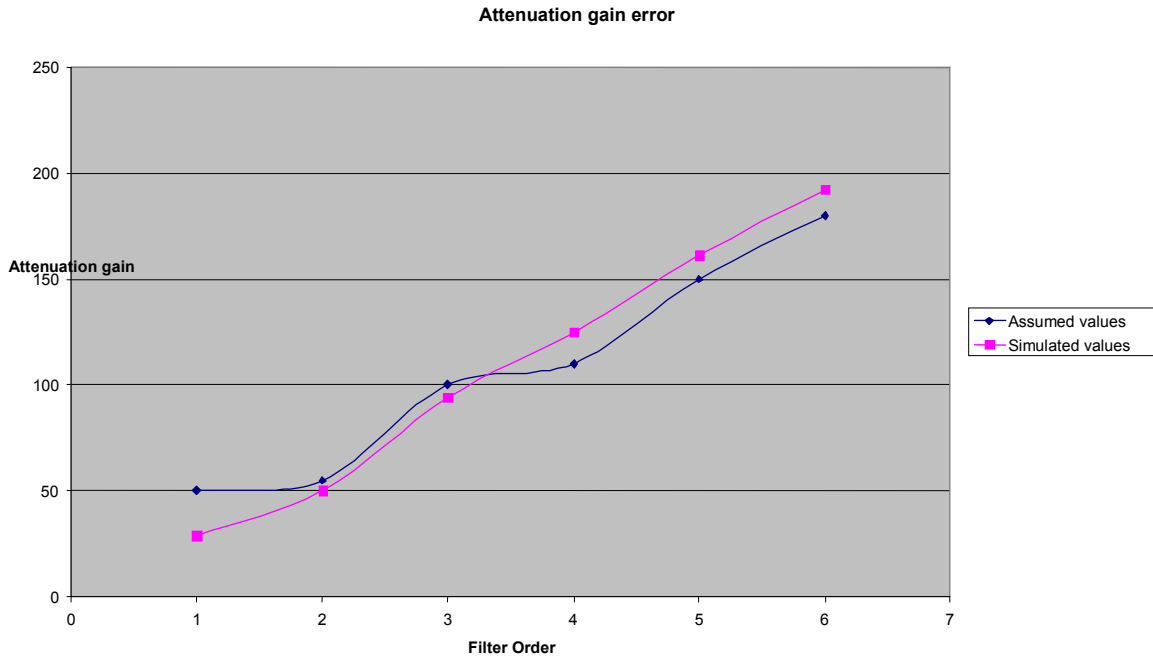


Figure 6.17. Variation in simulated values from calculated values.

From the observations integrated in Table 6.1, following conclusions are drawn that are in agreement with the literature.

1. With increase in filter order the roll-off rate is increasing.
2. The passband frequency decreases as filter order increases because of the increase in number of active components.
3. Power dissipation is increasing as the filter order increases. This is because of increase in number of components. But power dissipation is significantly due to the active component.

## Chapter 7

# CONCLUSION AND FUTURE SCOPE

### 7.1 Conclusion

Filter is a small component used in communication systems. There are various methods to design a filter depending upon the specifications and the application in which the filter has to be employed. Butterworth filter, known for its good transient response and maximally flat pass band is used widely in communication systems. The design procedure is developed for Active-RC Butterworth filter. For this first a selection is made for the active device used. Telescopic OTA is selected and a design procedure to design the Telescopic OTA is defined. The frequency response is simulated using the T-Spice environment of Tanner EDA tool. The simulated results of the Telescopic OTA are in compliance with the theoretical values.

The development of a design procedure provides a quick, well-integrated and effective mechanism for estimating the filter parameters. The steps highlighted make it easy to redesign the circuit for different sets of specifications. The design procedure is developed keeping in view few approximations that are made on the basis of basic know-how of the filter responses and the effect of the active device on the frequency response estimated by RC network. The results of SPICE simulation for the 6<sup>th</sup> order Active-RC Butterworth filter are shown in agreement with the results calculated by use of design equations and methodology.

6<sup>th</sup> order Active-RC Butterworth filter designed, has passband frequency of 12.63MHz, DC Gain of 13.94dB and roll-off rate of -140 dB/dec. So, this filter specifications are quite favourable with that of the video-frequency CMOS IC-filters reported to date.

The physical mask layout of any circuit to be manufactured using a particular process must confirm to a set of geometric constraints or rules, which are generally called layout design rules. Using the analog layout techniques of matching transistors, fingered structures and centroid geometry, the layouts are drawn. The results of the simulation of layout are nearly same as that shown by the schematic of the filter.

Further, Active-RC filters of different orders are designed. The analysis is done to show the change in filter parameters like roll-off rates, passband frequency, power dissipation etc. with increase in order. The analysis shows that with increase in filter order, the

parameters change in accordance with the changes stated in literature. This analysis validates the results of the filter design.

## **7.2 Future Scope**

- This is one method to design an Active-RC Butterworth filter using OTA. Other active devices like DDA can also be used.
- The design can be developed keeping in view low power and low voltages constraints as the filters are now being used in many portable applications that have requirement of low power and low voltages.
- Methods can be developed to reduce the steep increase in power dissipation and component count with increase in the filter order.
- Active devices can be designed in such a way that they may not restrict the passband frequency from expected passband frequency.

## APPENDIX - A

This **T-Spice** is for the Telescopic OTA with constant current source at its tail.

\* SPICE netlist written by S-Edit Win32 8.10

\* Written on May 24, 2006 at 08:56:41

\* Waveform probing commands

```
.probe
.options probefilename="om_one_end_2_input_sig.dat"
+ probesdbfile="C:\Documents and Settings\Jyoti\My Documents\nimisha\om_4th_order
butterworth_filter\telescopic_final_freeze\om_one_end_2_input_sig.sdb"
+ probetopmodule="Module0"
```

\* Main circuit: Module0

```
M1 N1 vin1 N3 N8 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M2 N2 vin2 N3 N8 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M4 vout N4 N2 N8 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M5 P3 P3 P1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M7 P1 P1 v1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M3 P3 N4 N1 N8 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M8 P2 P1 v1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M6 vout P3 P2 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
C9 vout Gnd 0.5pF
i10 N3 N8 500uA
v11 vin1 N6 0.0 AC 1.0 0.0
v12 vin2 Gnd 0.0 AC 1.0 180
v13 N4 Gnd 0.8
v14 N6 Gnd 40u
v15 Gnd N8 2.5
v16 v1 Gnd 2.5
.ac dec 10 1 1g
.print vdb(vout) vp(vout)
.op
.include "C:\Tanner\TSpice91\models\ml2_125.md"
• End of main circuit: Module0
```

## APPENDIX – B

This **T-Spice** is for the Telescopic OTA with bias circuit that provides bias at the gate of the tail transistor.

\* SPICE netlist written by S-Edit Win32 8.10

\* Written on May 18, 2006 at 10:45:39

\* Waveform probing commands

```
.probe
.options probefilename="OMNAMAHSIVAY_2end_simple tail.dat"
+ probesdbfile="C:\Documents and Settings\Jyoti\My Documents\nimisha\om namah
shivay\with simple tail\OMNAMAHSIVAY_2end_simple tail.sdb"
+ probetopmodule="Module0"
```

\* Main circuit: Module0

```
M3 P3 vb N1 N5 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M1 N1 vin1 N3 N5 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M4 vout vb N2 N5 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M2 N2 vin2 N3 N5 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u
M9 N3 O N5 N5 NMOS L=2u W=15u AD=66p PD=24u AS=66p PS=24u
M10 N10 N10 O N4 NMOS L=2u W=12.8u AD=66p PD=24u AS=66p PS=24u
M11 O O N4 N4 NMOS L=2u W=15u AD=66p PD=24u AS=66p PS=24u
M7 P1 P1 V1 V1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M5 P3 P3 P1 V1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M8 P2 P1 V1 V1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
M6 vout P3 P2 V1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u
C12 vout Gnd 0.5pF
v13 vin1 N7 0.0 AC 1.0 0.0
v14 vin2 Gnd 0.0 AC 1.0 180
v15 N7 Gnd 100u
v16 N10 Gnd 2.5
v17 V1 Gnd 2.5
v18 vb Gnd 0.8
v19 Gnd N4 2.5
v20 Gnd N5 2.5
.ac dec 10 1 1g
.print vdb(vout) vp(vout)
.op
.include "C:\Tanner\TSpice91\models\ml2_125.md"
• End of main circuit: Module0
```

## APPENDIX – C

This **T-Spice** is for 6<sup>th</sup> order Active-RC Butterworth filter.

\* SPICE netlist written by S-Edit Win32 8.10

\* Written on May 23, 2006 at 12:23:13

\* Waveform probing commands

.probe

.options probefilename="6th order.dat"

+ probesdbfile="C:\Documents and Settings\Jyoti\My Documents\nimisha\6th order\6th order.sdb"

+ probetopmodule="Module0"

.SUBCKT om vin1 vin2 vout Gnd

M1 N1 vin1 N3 N13 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u

M2 N2 vin2 N3 N13 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u

M4 vout N11 N2 N13 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u

M5 P3 P3 P1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u

M7 P1 P1 v1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u

M3 P3 N11 N1 N13 NMOS L=2u W=395u AD=66p PD=24u AS=66p PS=24u

M8 P2 P1 v1 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u

M6 vout P3 P2 v1 PMOS L=2u W=606u AD=66p PD=24u AS=66p PS=24u

i10 N3 N13 500uA

C11 vout Gnd 0.5pF

v11 N11 Gnd 0.8

v12 Gnd N13 2.5

v13 v1 Gnd 2.5

.ENDS

\* Main circuit: Module0

C1 N7 o1 .005pf

C2 N8 Gnd .005pf

C3 N5 Gnd .005pf

C4 N2 o2 .005pf

C5 N10 vout .005pf

C6 N11 Gnd .005pf

Xom\_1 N8 F1 o1 Gnd om

Xom\_2 N5 F2 o2 Gnd om

Xom\_3 N11 F3 vout Gnd om

R7 Gnd F1 2K TC=0.0, 0.0

R8 F1 o1 304 TC=0.0, 0.0

R9 vin N7 10k TC=0.0, 0.0

R10 N7 N8 10k TC=0.0, 0.0

R11 N2 N5 10k TC=0.0, 0.0

R12 o1 N2 10k TC=0.0, 0.0

R13 F2 o2 4940 TC=0.0, 0.0

```
R14 Gnd F2 4K TC=0.0, 0.0
R15 Gnd F3 3k TC=0.0, 0.0
R16 F3 vout 6k TC=0.0, 0.0
R17 o2 N10 10k TC=0.0, 0.0
R18 N10 N11 10k TC=0.0, 0.0
v19 vin N6 0.0 AC 1.0 0
v20 N6 Gnd 40u
.ac dec 10 1 150meg
.print vdb(vin) vdb(vout)
.include "C:\Tanner\TSpice91\models\ml2_125.md"
.op
* End of main circuit: Module0

# This T-Spice is for the simulation of
```

## **APPENDIX – D**

**Papers Published:**

During this work the following two review papers were accepted in the National Conference on Emerging Trends in Electronics and Communication (sponsored by IEEE) to be held on 14<sup>th</sup> - 15<sup>th</sup> September 2006 at Pardre Conceicao College of Engineering, Verna, Goa.

1. Nimisha Saini, Jyoti Jain and Alpana Agarwal, "Different Methods to Design Analog CMOS Filters: A Review" *National Conference on Emerging Trends in Electronics and Communication*, September 2006.
2. Jyoti Jain, Nimisha Saini and Alpana Agarwal, "Methods of Designing Elliptic Filter: A Review" *National Conference on Emerging Trends in Electronics and Communication*, September 2006.

## REFERENCES

- [1] Achim Gratz, “Operational Transconductance Amplifiers”, October 26, 2005.
- [2] Adel S. Sedra and Kenneth C. Smith, “ Microelectronics Circuits”, Oxford University Press, Fifth Edition, 2004
- [3] Behzad Razavi, “Design of Analog CMOS Integrated circuits”, Tata McGraw-Hill Publishing Company Limited, Edition 2002.
- [4] C. F. Wheatley and H. A. Wittlinger, “OTA Obsoletes Op-Amp”, *P. Nat. Econ. Conf.*, pp. 152-157, Dec., 1969.
- [5] Chunlei Shi, Yue Wu, Hassan O Elwan, and Mohammed Ismail, “A low-power high – linearity CMOS baseband filter for wideband CDMA applications”, *ISCAS 2000–IEEE International Symposium on Circuits and Systems*, Geneva, Switzerland, May 28- 31, 2000.
- [6] Darwin Cheung, Klaas Bult and Aaron Buchwaldy, “10-MHz 60-dB Dynamic-Range 4th-Order Butterworth Lowpass Filter”, *IEEE press*.
- [7] Deyasini Majumdar and Brent J. Maundy, “Low voltage  $G_m$ -C filters and OTAs for HDSL2”, *IEEE Canadian Conference on Electrical and Computer Engineering*, Montreal, Mayima, 2003.
- [8] Edward Sackinger and Walter Guggenbuhl, “A Versatile Building Block: The CMOS Differential Difference Amplifier”, *IEEE J. Solid-state Circuits*, Vol.sc-22, pp.287- 294, Apr., 1987.
- [9] Franco Maloberti, “Analog Design for CMOS VLSI Systems” Kluwer academic publisher, Boston/Dordrecht/London, 2001.
- [10] <http://en.wikipedia.org/filters>
- [11] J.Hughes, N.Bird and R.Soin, “A Novel Digitally Self-Tuned Continuous-Time Filter Technique”, *Digest of 1986 IEEE International Symposium on Circuits and Systems*, pp.1177- 1180, 1986.
- [12] John G.Proakis, Dimtris G.Manolakis, “Digital Signal Processing: Principles Algorithms and Applications, Third Edition, 2001.
- [13] K. Gulati, H-S. Lee, “ A  $\pm 2.45V$ -Swing CMOS Telescopic Operational Amplifier”, *Integrated Solid-State Circuits Conference*, 1998.

- [14] L.P.Huelsman and P.E.Allen, "Introduction to the Theory and Design of Active Filters", McGraw-Hill, New York, 1980.
- [15] Nicollini, G., et al., "High Frequency Fully Differential Filter Using Operational Amplifiers Without Common-mode Feedback," *IEEE Journal of Solid-State Circuits*, Vol. 24, No. 3, Jun., 1989.
- [16] Pan Wu and Rolf Schaumann, "Design considerations for CMOS and GaAs OTAs: Frequency response, linearity, tuning, and common-mode feedback", *Analog Integrated Circuits and Signal Processing*, 1(3): 247-268, Nov.,1991.
- [17] Paolo Cusinato, "Configurable direct-conversion / superheterodyne Baseband down-link channel for W-CDMA applications", *IEEE International Symposium on Circuits and Systems*, 2004.
- [18] Philip E. Allen, Douglas R. Holberg., "CMOS analog circuit design", Oxford University Press London, Second Edition, 2003.
- [19] Ramakant A. Gayakwad, "Op-Amps and linear Integrated Circuits", Third Edition., Prentice Hall, 1999.
- [20] R. L. Geiger and E. Sánchez-Sinencio, "Active Filter Design Using Operational Transconductance Amplifiers: A Tutorial", *IEEE Circuits and Devices Magazine*, Vol. 1, pp.20-32, Mar.1985.
- [21] Sergio Solís-Bustos, José Silva-Martínez, Franco Maloberti and Edgar Sánchez-Sinencio, "A 60-dB Dynamic-Range CMOS Sixth-Order 2.4-Hz Low-Pass Filter for Medical Applications", *IEEE transactions on Circuits And Systems – II: Analog And Digital Signal Processing*, Vol. 7, No. 12, Dec., 2000.
- [22] Tanner EDA User Guide
- [23] William L. Brown and Andrew Y.J. Szeto, "Reconciling Spice results and Hand Calculations: Unexpected problems", *IEEE Transaction on Education*, Vol.43, No.1, Feb., 2000.
- [24] [www.ee.nuigalway.ie/subjects/ee208/asd/notes/lecture08.pdf](http://www.ee.nuigalway.ie/subjects/ee208/asd/notes/lecture08.pdf)
- [25] [www.microchip.com/downloads/en/AppNotes/00699b.pdf](http://www.microchip.com/downloads/en/AppNotes/00699b.pdf).
- [26] [www.sensorsmag.com/articles/0701/24/main.shtml](http://www.sensorsmag.com/articles/0701/24/main.shtml)
- [27] [www.swarthmore.edu/NatSci/echeeve1/Ref/DataSheet/IntroToFilters.Pdf](http://www.swarthmore.edu/NatSci/echeeve1/Ref/DataSheet/IntroToFilters.Pdf)
- [28] [http://www.web-ee.com/primers/files/Elliptical\\_Filters.pdf](http://www.web-ee.com/primers/files/Elliptical_Filters.pdf)

[29] Y.P. Tsividis and J.O. Voorman, "Integrated Continuous-Time Filters", *IEEE press*, New York, 1992.