



Handbook on  
**Simulation of  
OPAMP based  
Circuits using LTspice**

---

Dr. Soorya Krishna K



---

# Handbook on Simulation of OPAMP Based Circuits using LTspice

---

**First Edition**

**Author**  
Dr. Soorya Krishna K



**Title of the Book:**Handbook on Simulation of OPAMP Based Circuits using LTspice

**First Edition - 2023**

**Copyright 2023 © Dr. Soorya Krishna K,**Srinivas Institute of Technology  
Mangalore.

No part of this book may be reproduced or transmitted in any form by any means, electronic or mechanical, including photocopy, recording or any information storage and retrieval system, without permission in writing from the copyright owners.

### **Disclaimer**

The author is solely responsible for the contents published in this book. The publishers don't take any responsibility for the same in any manner. Errors, if any, are purely unintentional and readers are requested to communicate such errors to the editors or publishers to avoid discrepancies in future.

**ISBN: 978-93-5747-573-0**

**MRP Rs. 170/-**

### **Publisher, Printed at & Distribution by:**

Selfypage Developers Pvt Ltd.,  
Pushpagiri Complex,  
Beside SBI Housing Board,  
K.M. Road Chikkamagaluru, Karnataka.  
Tel.: +91-8861518868  
E-mail:info@iiponline.org

**IMPRINT:IIP Iterative International Publishers**

### **For Sales Enquiries**

Contact: 91- 8861511583  
E-mail: sales@iipbooks.com



# Preface

The new National Education Policy (NEP) document places greater emphasis on practical teaching over theoretical classroom instruction. Consequently, several universities have introduced an increased number of labs, subjects coupled with labs, internships, mini-projects, main projects for extended durations, industry-related subjects, courses offered by industries, and more. The primary objective is to equip students with skills upon graduation. Furthermore, companies in the electronics field now expect students entering their ranks to be proficient in simulating circuits using software rather than relying solely on theoretical knowledge.

This signifies a drastic change aimed at enhancing teaching methodologies and the curriculum. Many engineering subjects can be effectively taught using simulations. Students will become acquainted with modern tools. From my teaching experience, I've observed that students grasp circuit operations more effectively when they witness circuit results. I've employed this approach while teaching subjects such as analog circuits, basic electronics, Verilog HDL, and linear integrated circuits. Hands-on experience tends to engage students more and facilitates better learning.

The intention behind writing this book is to advocate teaching OPAMP-based subjects in conjunction with simulations, rather than solely through theoretical instruction. The book focuses on fundamental circuits using operational amplifiers (OPAMPs), serving as the basic circuits for simulation. Design and simulations of inverting amplifiers, differentiators, integrators, comparators, filters, oscillators, multivibrators, precision rectifier circuits, etc., are covered in this book.

Faculty members are encouraged to incorporate simulations into their teaching methods, while students are encouraged to learn the subject by simulating and verifying the theory they've learned. The book emphasizes understanding circuit operations, designing circuits, observing parameters, examining output waveforms, and briefly discussing resultant waveforms. It's not aimed at absolute beginners in OPAMP circuits and does not include derivations for expressions like gain, etc.

LTspice software is used for these simulations. It's freely available for download from their website, chosen for its convenience in allowing simulations on laptops/desktops from the comfort of our homes. Additionally, this software can handle reasonably complex circuits and allows the utilization of corresponding models if specific components are unavailable.

Although this book employs LTspice software, these circuits can be simulated using other commercial software. The process of circuit design, verification of output waveforms, and circuit operations remains consistent. The book is structured to enable users to create and verify circuits provided they understand how to use the software; the simulation concepts themselves remain unchanged.

All circuits covered in this book are simulated using LTspice, and their output waveforms are cross-checked with theoretical knowledge. After completing each simulation, tasks are provided to encourage students to practice circuit design and verify outputs. I urge you to utilize this book when learning OPAMP and 555 timer-based circuits.



## **Acknowledgement**

I am deeply indebted to several individuals and to my institution Srinivas Institute of Technology (SIT) whose unwavering support and encouragement were pivotal in the creation of this book.

First and foremost, I extend my heartfelt appreciation to Dr. Shrinivasa Mayya D. for his constant encouragement. His guidance and belief in academic pursuits have been instrumental in fostering an environment conducive to writing this book.

I am profoundly grateful to the management of SIT for their continuous support and facilitation throughout the journey of crafting this book. Their commitment to academic excellence and their provision of resources have significantly contributed to the completion of this project.

The Department of Electronics and Communication Engineering (ECE) holds a special place in my heart for its relentless efforts in imparting knowledge and fostering an atmosphere of intellectual growth. My heartfelt gratitude goes to the SIT ECE department faculty members and students whose interactions have been invaluable in shaping my understanding and perspective.

To my family, whose unwavering love, patience, and unwavering support have been my pillar of strength throughout this endeavor, I express my deepest gratitude. Your understanding during the demanding periods of writing and research has been a constant source of motivation.

Additionally, I wish to acknowledge the contributions and support of Mr. Mahesh, who provided invaluable insights, feedback, and encouragement. I would also like to thank Dr. R.K. Hegde for the inputs given on the book publications. At this point, I remember my mentor, Dr. M.S. Bhat, for preparing me for lifelong learning. I thank Mrs. Flavita for the timely suggestions on some of the topics. These supports played a significant role in shaping the direction and content of this work.

I extend my heartfelt gratitude to the IIP publishing Team for their unwavering support, professionalism, and commitment to excellence throughout the publication process. Their dedication and guidance have been instrumental in

bringing this work to light, and I am deeply appreciative of their invaluable contributions.

To all those whose names may not appear here but whose support, guidance, and belief in this project were significant, I offer my heartfelt thanks. Your encouragement propelled me forward, and this book would not have been possible without your collective support.

This book is a testament to the collaborative effort and support of these remarkable individuals and institutions. I am profoundly grateful for their contributions to this endeavor.



# Contents

Introduction to LTspice, OPAMP, and 555 Timer	1-12
1. Realizing (i)Inverting Amplifier (ii)Non-Inverting Amplifier and (iii)Voltage Follower using OPAMP	13-22
2. Realizing (i)Summing and (ii)Difference Amplifier using OPAMP	23-30
3. Realizing (i)Differentiator and (ii)Integrator Circuit using OPAMP	31-49
4. Realizing Instrumentation Amplifier using OPAMPs	50-56
5. Realizing (i)Non-inverting Comparator (ii)Zero Crossing Detector and (iii)Schmitt Trigger using OPAMPs	57-72
6. Realizing (i)Halfwave and (ii)Fullwave Precision Rectifiers using OPAMPs	73-79
7. Realizing (i)Peak Detector and (ii)Sample and Hold Circuit using OPAMP	80-90
8. Realizing (i)Astable Multivibrator and (ii)Monostable Multivibrator using OPAMP	91-98
9. Realizing (i)RC Phase Shift and (ii)Wein Bridge Oscillator circuits using OPAMP	99-109
10. Generation of Triangular and Sawtooth Waveforms using OPAMP	110-114
11. Realizing (i)Clipper and (ii)Peak Clamper circuits using OPAMP	115-120
12. Realizing (i)First-Order and Second-Order Low-Pass Butterworth Filter and (ii) First-Order and Second-Order High-Pass Butterworth Filter using OPAMPs	121-131
13. Realizing (i) 4-bit Binary Weighted DAC and (ii) 4-bit R-2R DAC using OPAMP	132-138
14. Realizing (i)Astable Multivibrator and (ii)Monostable Multivibrator using 555 Timer	139-145
15. Realizing 555 Timer Applications: (i)Frequency Divider (ii)Schmitt Trigger and (iii)Pulse Width Modulation	146-152
Reference	153



## ABOUT THE BOOK

The  $\mu$ A741 OPAMP is one of the most widely used ICs in a variety of applications. Understanding the basic operations using an op-amp is highly desirable. In this book, all circuits are designed and simulated using LTspice. The simulation waveforms are cross-verified with the theoretical values obtained from equations.

Introduction to LTspice, op-amps, and the 555 timer is discussed in the opening chapter of this book. Chapters 1 through 4 cover the design of inverting amplifiers, non-inverting amplifiers, summers, difference amplifiers, differentiators, integrators, and instrumentation amplifiers. Chapters 5 through 11 delve into various op-amp applications such as comparators, multivibrators, precision rectifiers, waveform generators, and waveform shapers. Chapter 12 discusses first-order and second-order high-pass and low-pass filters. ADC and DAC circuits are explored in Chapter 13. Chapters 14 and 15 describe the 555 timer as a multivibrator and its applications.

Each chapter discusses circuit design, necessary procedures, and setup required for simulations. At the end of each chapter, tasks are provided to independently practice circuit design. Additionally, a brief discussion is included on the simulated waveforms.

## ABOUT THE AUTHOR



**Soorya Krishna K** obtained his Ph.D. from NITK Surathkal. He obtained his M. Tech in Digital Electronics and Communication from NMAMIT Nitte. Presently, he is working as a professor in the Department of Electronics and Communication Engineering at Srinivas Institute of Technology in Mangalore. His research area was VLSI and High-Speed Interconnects. Currently, he is actively involved in the research areas of VLSI, IoT, and Machine Learning.



Selfypage Developers Pvt Ltd

EISBN: 978-93-5747-573-0



MRP. 170/-