Analog Electronics

an integrated PSpice approach

T.E. Price

Analog Electronics With Pspice

Thomas L. Floyd

Analog Electronics With Pspice:

Analog Electronics T. E. Price,1997 *Analog Electronics* T. E. Price,1997 Analog Electronics An integrated PSpice approach concentrates on the introductory material associated with analog electronics The book starts with a description of the basic active of diodes transistors both bipolar and FET and integrated circuits There are chapters on frequency response oscillators filters power supplies and the interface between analog and digital circuitry There are many worked examples and extensive use of PSpice provides realistic demonstrations of circuit principles *Electronics* Nassir H. Sabah,2017-12-19 Electronics Basic Analog and Digital with PSpice does more than just make unsubstantiated assertions about electronics Compared to most current textbooks on the subject it pays significantly more attention to essential basic electronics and the underlying theory of semiconductors In discussing electrical conduction in semiconductors the author addresses the important but often ignored fundamental and unifying concept of electrochemical potential of current carriers which is also an instructive link between semiconductor and ionic systems at a time when electrical engineering students are increasingly being exposed to biological systems The text presents the background and tools necessary for at least a qualitative understanding of new and projected advances in microelectronics The author provides helpful PSpice simulations and associated procedures based on schematic capture and using OrCAD 16 0 Demo software which are available for download These simulations are explained in considerable detail and integrated throughout the book The book also includes practical real world examples problems and other supplementary material which helps to demystify concepts and relations that many books usually state as facts without offering at least some plausible explanation With its focus on fundamental physical concepts and thorough exploration of the behavior of semiconductors this book enables readers to better understand how electronic devices function and how they are used The book s foreword briefly reviews the history of electronics and its impact in today s world Classroom Presentations are provided on the CRC Press website Their inclusion eliminates the need for instructors to prepare lecture notes The files can be modified as may be desired projected in the classroom or lecture hall and used as a basis for discussing the course material **Laboratory Experiments and PSPICE Simulations in Analog Electronics** L. K. Maheshwari,2006 *PSPICE and MATLAB for Electronics* John Okyere Attia,2002-05-15 PSPICE has circuit simulation features unmatched by any other scientific software MATLAB s capabilities for matrix computations plotting data processing and analysis are well established throughout the world Together these two software packages form a powerful full function toolbox for electronic circuit analysis PSPICE and MATLAB for Electronics offers the first integrated presentation of both of these software packages It provides a PSPICE primer a MATLAB primer and an in depth treatment of their combined power for solving electronics problems particularly those associated with diodes op amps and transistor circuits The author takes a practical approach provides a multitude of examples and encourages readers to put what they ve learned into practice through the many exercises provided in each chapter All of the PSPICE netlists and MATLAB m files used in the examples are available on the Internet at www crcpress com Anyone working or aspiring to work in electronics needs a familiarity with these products and learning to use them together offers more than the sum of their advantages Use PSPICE for circuit analysis use MATLAB for calculating device parameters curve fitting numerical functions and plots and use PSPICE and MATLAB for Electronics to learn how they can work in tandem to effectively and efficiently explore device characteristics and analyze circuits and systems **Analog Design and Simulation Using OrCAD Capture and PSpice** Dennis Fitzpatrick,2017-12-11 New to this edition Updated to using OrCAD Release 17 2 and its new features Coverage of PSPICE extra features PSpice Designer PSpice Designer Plus Modelling Application PSpice Part Search Symbol Viewer PSpice Report Associate PSpice model New delay functions for Behavioural Simulation Models New Models Support for negative values in hysteresis voltage and threshold voltage A new chapter on PSpice Advanced Analysis Analog Design and Simulation Using OrCAD Capture and PSpice Second Edition provides step by step instructions on how to use the Cadence OrCAD family of Electronic Design Automation software for analog design and simulation The book explains how to enter schematics in Capture set up project types project libraries and prepare circuits for PSpice simulation There are chapters on the different analysis types for DC Bias point DC sweep AC frequency sweep Parametric analysis Temperature analysis Performance Analysis Noise analysis Sensitivity and Monte Carlo simulation Subsequent chapters explain how the Stimulus Editor is used to define custom analog and digital signals how the Model Editor is used to view and create new PSpice models and Capture parts and how the Magnetic Parts Editor is used to design transformers and inductors Other chapters include Analog Behaviorial models Test Benches as well as how to create hierarchical designs The book includes the latest features in the OrCAD 17 2 release and there are exercises with step by step instructions at the end of each chapter that enables the reader to progress based upon their experience and knowledge gained from previous chapters The author worked for Cadence for over eight years and supported and delivered OrCAD PSpice training courses all over Europe This book has been endorsed by Cadence In addition there are new chapters on the PSpice Advanced Analysis suite of tools Sensitivity Analysis Optimizer Monte Carlo and Smoke Analysis The chapters show how circuit performance can effectively be maximised and optimised for variations in component tolerances temperature effects manufacturing yields and component stress Provides both a comprehensive user guide and a detailed overview of simulation using OrCAD Capture and PSpice Includes worked and ready to try sample designs and a wide range of to do exercises Covers Capture and PSpice together

Analog Circuits and its Simulation in PSPICE Dr A Chrispin Jiji,2021-06-23 This book is intended to support the students of undergraduate engineering in the related fields of Electronics and Communication Engineering as well as Telecommunication Engineering courses for practicing laboratory experiments It gives relevant information on the basic understanding of circuit configurations and connectivity of BJT and FET Amplifiers and Study of frequency response It presents the design and test of Analog circuits using OPAMPs understand the feedback configurations of transistor and

OPAMP circuits and the use of circuit simulation for the analysis of electronic circuits using PSPICE It also provides various methods and techniques for conducting the experiment Clear circuit diagrams and proper calculations have been provided for all the experiments and simple language has been used throughout the book for better understanding of the concepts for the students *PSpice for Circuit Theory and Electronic Devices* Paul Tobin,2022-05-31 PSpice for Circuit Theory and Electronic Devices is one of a series of five PSpice books and introduces the latest Cadence Orcad PSpice version 10 5 by simulating a range of DC and AC exercises It is aimed primarily at those wishing to get up to speed with this version but will be of use to high school students undergraduate students and of course lecturers Circuit theorems are applied to a range of circuits and the calculations by hand after analysis are then compared to the simulated results The Laplace transform and the s plane are used to analyze CR and LR circuits where transient signals are involved Here the Probe output graphs demonstrate what a great learning tool PSpice is by providing the reader with a visual verification of any theoretical calculations Series and parallel tuned resonant circuits are investigated where the difficult concepts of dynamic impedance and selectivity are best understood by sweeping different circuit parameters through a range of values Obtaining semiconductor device characteristics as a laboratory exercise has fallen out of favour of late but nevertheless is still a useful exercise for understanding or modelling semiconductor devices Inverting and non inverting operational amplifiers characteristics such as gain bandwidth are investigated and we will see the dependency of bandwidth on the gain using the performance analysis facility Power amplifiers are examined where PSpice Probe demonstrates very nicely the problems of cross over distortion and other problems associated with power transistors We examine power supplies and the problems of regulation ground bounce and power factor correction Lastly we look at MOSFET device characteristics and show how these devices are used to form basic CMOS logic gates such as NAND and NOR gates SPICE and LTspice for Power Electronics and Electric Power Muhammad H. Rashid,2024-11-13 Power electronics can be a difficult course for students to understand and for professional professors to teach simplifying the process for both LTspice for power electronics and electrical power edition illustrates methods of integrating industry standard LTspice software for design verification and as a theoretical laboratory bench Helpful LTspice software and Program Files Available for Download Based on the author Muhammad H Rashid s considerable experience merging design content and SPICE into a power electronics course this vastly improved and updated edition focuses on helping readers integrate the LTspice simulator with a minimum amount of time and effort Giving users a better understanding of the operation of a power electronic circuit the author explores the transient behavior of current and voltage waveforms for every circuit element at every stage The book also includes examples of common types of power converters as well as circuits with linear and nonlinear inductors New in this edition Changes to run on OrCAD SPICE or LTspice IV or higher Students learning outcomes SLOs listed at the start of each chapter Abstracts of chapters List the input side and output side performance parameters of the converters The characteristics of power semiconductors diodes

BJTs MOSFETs and IGBTs Generating PWM and sinusoidal PWM gating signals Evaluating the power efficiency of converters Monte Carlo analysis of converters Worst case analysis of converters Nonlinear transformer model Evaluate user defined electrical quantities MEASURE This book demonstrates techniques for executing power conversion and ensuring the quality of output waveform rather than the accurate modeling of power semiconductor devices This approach benefits students enabling them to compare classroom results obtained with simple switch models of devices **SPICE for Power Electronics and Electric Power** Muhammad H. Rashid,2017-12-19 Power electronics can be a difficult course for students to understand and for professors to teach Simplifying the process for both SPICE for Power Electronics and Electric Power Third Edition illustrates methods of integrating industry standard SPICE software for design verification and as a theoretical laboratory bench Helpful PSpice Software and Program Files Available for Download Based on the author Muhammad H Rashid s considerable experience merging design content and SPICE into a power electronics course this vastly improved and updated edition focuses on helping readers integrate the SPICE simulator with a minimum amount of time and effort Giving users a better understanding of the operation of a power electronics circuit the author explores the transient behavior of current and voltage waveforms for each and every circuit element at every stage The book also includes examples of all types of power converters as well as circuits with linear and nonlinear inductors New in this edition Student learning outcomes SLOs listed at the start of each chapter Changes to run on OrCAD version 9 2 Added VPRINT1 and IPRINT1 commands and examples Notes that identify important concepts Examples illustrating EVALUE GVALUE ETABLE GTABLE ELAPLACE GLAPLACE EFREQ and GFREQ Mathematical relations for expected outcomes where appropriate The Fourier series of the output voltages for rectifiers and inverters PSpice simulations of DC link inverters and AC voltage controllers with PWM control This book demonstrates techniques of executing power conversions and ensuring the quality of the output waveforms rather than the accurate modeling of power semiconductor devices This approach benefits students enabling them to compare classroom results obtained with simple switch models of devices In addition a new chapter covers multi level converters Assuming no prior knowledge of SPICE or PSpice simulation the text provides detailed step by step instructions on how to draw a schematic of a circuit execute simulations and view or plot the output results It also includes suggestions for laboratory experiments and design problems that can be used for student homework assignments

Electronic Experiences in a Virtual Lab Roberto Gastaldi,Giovanni Campardo,2020-05-11 This book presents a collection of lessons on various topics commonly encountered in electronic circuit design including some basic circuits and some complex electronic circuits which it uses as vehicles to explain the basic circuits they are composed of The circuits considered include a linear amplifier oscillators counters a digital clock power supplies a heartbeat detector a sound equalizer an audio power amplifier and a radio The theoretical analysis has been deliberately kept to a minimum in order to dedicate more time to a learning by doing approach which after a brief review of the theory readers are encouraged to use directly with a simulator tool to examine the operation of circuits in a virtual laboratory Though the book is not a theory textbook readers should be familiar with the basic principles of electronic design and with spice like simulation tools To help with the latter aspect one chapter is dedicated to the basic functions and commands of the OrCad P spice simulator used for the experiments described in the book The Art of Simulation Using PSPICEAnalog and Digital Bashir Al-Hashimi,1995-02-15 This comprehensive volume covers both elementary and advanced analog and digital circuit simulation using PSpice The text includes many worked examples circuit diagrams tables and code listings It also compares practical results with those obtained from simulation PSpice for Analog Communications Engineering Paul Tobin,2007 In PSpice for Analog Communications Engineering we simulate the difficult principles of analog modulation using the superb free simulation software Cadence Orcad PSpice V10 5 While use is made of analog behavioral model parts ABM we use actual circuitry in most of the simulation circuits For example we use the 4 quadrant multiplier IC AD633 as a modulator and import real speech as the modulating source and look at the trapezoidal method for measuring the modulation index Modulation is the process of relocating signals to different parts of the radio frequency spectrum by modifying certain parameters of the carrier in accordance with the modulating information signals In amplitude modulation the modulating source changes the carrier amplitude but in frequency modulation it causes the carrier frequency to change and in phase modulation it s the carrier phase The digital equivalent of these modulation techniques are examined in PSpice for Digital communications Engineering where we examine QAM FSK PSK and variants We examine a range of oscillators and plot Nyquist diagrams showing themarginal stability of these systems The superhetrodyne principle the backbone of modern receivers is simulated using discrete components followed by simulating complete AM and FM receivers In this exercise we examine the problems ofmatching individual stages and the use of double tuned RF circuits to accommodate the large FM signal bandwidth **An Analog Electronics Companion** Scott Hamilton,2003-04-24 Engineers and scientists frequently find themselves having to get involved in electronic circuit design even though this may not be their specialty This book is specifically designed for these situations and has two major advantages for the inexperienced designer it assumes little prior knowledge of electronics and it takes a modular approach so you can find just what you need without working through a whole chapter The first three parts of the book start by refreshing the basic mathematics and physics needed to understand circuit design Part four discusses individual components resistors capacitors etc while the final and largest section describes commonly encountered circuit elements such as differentiators oscillators filters and couplers A major bonus and learning aid is the inclusion of a CD ROM with the student edition of the PSpice simulation software together with models of most of the circuits described in the book Computer-Aided Analysis and Design of Switch-Mode Power Supplies Lee,2017-10-19 This comprehensive reference text explains the development and principles of operation modelling and analysis of switch mode power supplies SMPS highlighting conversion efficiency size and steady state transient regulation characteristics

Covering the practical design techniques of SMPS this book reveals how to develop specific models of circuits and components for simulation and design purposes explains both the computer simulation of the switching behaviours of dc to dc converters and the modelling of linear and nonlinear circuit components deals with the modelling and simulation of the low frequency behaviours of converters including current controlled converters and converters with multiple outputs and regulators describes computer aided design CAD techniques as applied to converters and regulators introduces the principles and design of quasi resonant and resonant converters provides details on SPICE a circuit simulator package used to calculate electrical circuit behaviour Containing over 1000 helpful drawings equations and tables this is a valuable reference for circuit design electrical and electronics engineers and serves as an excellent text for upper level undergraduate and graduate students in these disciplines Principles of Electric Circuits Thomas L. Floyd,1997 A text CD ROM introducing basic electrical concepts and circuits featuring chapter section reviews worked examples summaries glossaries key formulas self tests problems and selected answers This fifth edition contains new PSpice sections in all chapters a full color format and related exe *American Journal of Physics* ,2002 PSPICE and MATLAB for Electronics John Okyere Attia,2002 PSPICE has circuit simulation features unmatched by any other scientific software MATLAB s capabilities for matrix computations plotting data processing and analysis are well established throughout the world Together these two software packages form a powerful full function toolbox for electronic circuit analysis PSPICE and MATLAB for Electronics offers the first integrated presentation of both of these software packages It provides a PSPICE primer a MATLAB primer and an in depth treatment of their combined power for solving electronics problems particularly those associated with diodes op amps and transistor circuits The author takes a practical approach provides a multitude of examples and encourages readers to put what they ve learned into practice through the many exercises provided in each chapter All of the PSPICE netlists and MATLAB m files used in the examples are available on the Internet at www crcpress com Anyone working or aspiring to work in electronics needs a familiarity with these products and learning to use them together offers more than the sum of their advantages Use PSPICE for circuit analysis use MATLAB for calculating device parameters curve fitting numerical functions and plots and use PSPICE and MATLAB for Electronics to learn how they can work in tandem to effectively and efficiently explore device characteristics and analyze circuits and systems Provided by publisher Electronic Design ,1994

Analog Integrated Circuits with PSPICE Dr A Chrispin Jiji,2021-12-21 This book is intended to support the students of undergraduate engineering in the related fields of Electronics and Communication Engineering as well as Telecommunication Engineering courses for practicing laboratory experiments It gives relevant information on the basic understanding of circuit configurations and connectivity of BJT and FET Amplifiers and Study of frequency response It presents the design and test of Analog Integrated circuits using OPAMPs understand the feedback configurations of transistor and OPAMP circuits and the use of circuit simulation for the analysis of electronic circuits using PSPICE It also provides various methods and techniques for conducting the experiment Clear circuit diagrams and proper calculations have been provided for all the experiments and simple language has been used throughout the book for better understanding of the concepts for the students

Right here, we have countless book **Analog Electronics With Pspice** and collections to check out. We additionally allow variant types and furthermore type of the books to browse. The up to standard book, fiction, history, novel, scientific research, as skillfully as various supplementary sorts of books are readily welcoming here.

As this Analog Electronics With Pspice, it ends stirring physical one of the favored ebook Analog Electronics With Pspice collections that we have. This is why you remain in the best website to look the unbelievable book to have.

https://gandalf.roeckerfam.com/files/browse/default.aspx/Constant%20Adolphe.pdf

Table of Contents Analog Electronics With Pspice

14. Embracing eBook Trends
    - Integration of Multimedia Elements
    - Interactive and Gamified eBooks

Analog Electronics With Pspice Introduction

In the digital age, access to information has become easier than ever before. The ability to download Analog Electronics With Pspice has revolutionized the way we consume written content. Whether you are a student looking for course material, an avid reader searching for your next favorite book, or a professional seeking research papers, the option to download Analog Electronics With Pspice has opened up a world of possibilities. Downloading Analog Electronics With Pspice provides numerous advantages over physical copies of books and documents. Firstly, it is incredibly convenient. Gone are the days of carrying around heavy textbooks or bulky folders filled with papers. With the click of a button, you can gain immediate access to valuable resources on any device. This convenience allows for efficient studying, researching, and reading on the go. Moreover, the cost-effective nature of downloading Analog Electronics With Pspice has democratized knowledge. Traditional books and academic journals can be expensive, making it difficult for individuals with limited financial resources to access information. By offering free PDF downloads, publishers and authors are enabling a wider audience to benefit from their work. This inclusivity promotes equal opportunities for learning and personal growth. There are numerous websites and platforms where individuals can download Analog Electronics With Pspice. These websites range from academic databases offering research papers and journals to online libraries with an expansive collection of books from various genres. Many authors and publishers also upload their work to specific websites, granting readers access to their content without any charge. These platforms not only provide access to existing literature but also serve as an excellent platform for undiscovered authors to share their work with the world. However, it is essential to be cautious while downloading Analog Electronics With Pspice. Some websites may offer pirated or illegally obtained copies of copyrighted material. Engaging in such activities not only violates copyright laws but also undermines the efforts of authors, publishers, and researchers. To ensure ethical downloading, it is advisable to utilize reputable websites that prioritize the legal distribution of content. When downloading Analog Electronics With Pspice, users should also consider the potential security risks associated with online platforms. Malicious actors may exploit vulnerabilities in unprotected websites to distribute malware or steal personal information. To protect themselves, individuals should ensure their devices have reliable antivirus software installed and validate the legitimacy of the websites they are downloading from. In conclusion, the ability to download Analog Electronics With Pspice has transformed the way we access information. With the convenience, cost-effectiveness, and accessibility it offers, free PDF downloads have become a popular choice for students, researchers, and book lovers worldwide. However, it is crucial to engage in ethical downloading practices and prioritize personal security when utilizing online platforms. By doing so, individuals can make the most of the vast array of free PDF resources available and embark on a journey of continuous learning and intellectual growth.

FAQs About Analog Electronics With Pspice Books

How do I know which eBook platform is the best for me? Finding the best eBook platform depends on your reading preferences and device compatibility. Research different platforms, read user reviews, and explore their features before making a choice. Are free eBooks of good quality? Yes, many reputable platforms offer high-quality free eBooks, including classics and public domain works. However, make sure to verify the source to ensure the eBook credibility. Can I read eBooks without an eReader? Absolutely! Most eBook platforms offer web-based readers or mobile apps that allow you to read eBooks on your computer, tablet, or smartphone. How do I avoid digital eye strain while reading eBooks? To prevent digital eye strain, take regular breaks, adjust the font size and background color, and ensure proper lighting while reading eBooks. What the advantage of interactive eBooks? Interactive eBooks incorporate multimedia elements, quizzes, and activities, enhancing the reader engagement and providing a more immersive learning experience. Analog Electronics With Pspice is one of the best book in our library for free trial. We provide copy of Analog Electronics With Pspice in digital format, so the resources that you find are reliable. There are also many Ebooks of related with Analog Electronics With Pspice. Where to download Analog Electronics With Pspice online for free? Are you looking for Analog Electronics With Pspice PDF? This is definitely going to save you time and cash in something you should think about.

Find Analog Electronics With Pspice :

construction accounting and financial management
construction of religious boundaries
constant exposure
construction claims analysis presentation and defense
contemporary business mathematics with canadian applications
conspiracy the plot to stop the kennedys
contemporary american philosophy volume i personal statements

Analog Electronics With Pspice :

Sylvia S. Mader Looking for books by Sylvia S. Mader? See all books authored by Sylvia S. Mader, including Human Biology, and Essentials of Biology, ... Human Biology by Mader, Sylvia Instructors consistently ask for a Human Biology textbook that helps students understand the main themes of biology through the lens of the human body. Human Biology 16th edition - VitalSource Human Biology 16th Edition is written by Sylvia Mader; Michael Windelspecht and published by McGraw-Hill Higher Education (International). Human Biology Sylvia S. Mader has authored several nationally recognized biology texts published by McGraw-Hill. Educated at Bryn Mawr College, Harvard University, Tufts ... Human Biology 17th edition 9781260710823 Jul 15, 2020 — Human Biology 17th Edition is written by Sylvia Mader, Michael Windelspecht and published by McGraw-Hill Higher Education. Human Biology by Sylvia S. Mader (2002 ... - eBay Human Biology by Sylvia S. Mader (2002, Paperback) Seventh Edition. Some check marks little writing. 20 Best Human Biology Books of All Time The 20 best human biology books, such as Human Diversity, Human Anatomy for Kids, The Complete Human Body and Cell Biology for Babies. Human Biology by Michael Windelspecht and ... Human Biology by Michael Windelspecht and Sylvia S. Mader (2015, Trade Paperback). Human Biology by Sylvia Mader 16th EDITION Hi guys, if any one of you have the 16th edition of Human Biology by Sylvia Mader and Michael Windelapecht can y'all send me pictures of the ... Human Biology, 14th Edition Sylvia Mader - Jarir.com KSA Shop for Human Biology, 14th Edition by Sylvia Mader McGraw Hill Biology Medical Books English Books jarir bookstore Kuwait. Payroll Practice Test Newly hired employees must be reported to governmental officials within 20 days of starting work for an employer. A) True. B) False. Page 4. Payroll Practice ... Payroll Accounting Quiz and Test Payroll Accounting (Practice Quiz). Print PDF. For multiple-choice and true/false questions, simply press or click on what you think is the correct answer. The Payroll Source CPP Practice Exam THE PAYROLL SOURCE. CPP PRACTICE EXAM. 1. Which of the following features is LEAST likely to be considered when looking at the security of a new payroll system? Payroll Accounting - Practice Test Questions & Chapter Exam Test and improve your knowledge of Payroll Accounting with fun multiple choice exams you can take online with Study.com. Test Your Payroll Knowledge - BASIC Sep 1, 2010 — The correct answers are listed at the bottom of this quiz. Quiz Questions: 1) What form is used to obtain a Social Security number? A) Form SS- ... study guide payroll specialist Payroll Specialist. Test #2820.r0319. Sample Questions. The following sample questions should give you some idea of the form the test will take. 1. Which SAP ... Free Fundamental Payroll Certification Practice Test (2023) Nov 2, 2023 — Fundamental Payroll Certification Exam Outline. The FPC exam contains 150 multiple-choice questions, 25 of which are unscored, and you will be ... Certified Payroll Professional Practice Test Oct 31, 2023 — The Certified Payroll Professional exam contains 190 multiple-choice questions, 25 of which are unscored, and you are given a four-hour time ... COMP XM Flashcards Study with Quizlet and memorize flashcards containing terms like Segment/Perf/Size, Prices between each round, Price for each product and more. COMP XM Exam : r/Capsim The questions are a bit hard and change a lot from exam to exam so do not trust too much the keys you find online, most of them are about ... Board Query 1 Questions and Answers for FINAL COMP ... Aug 4, 2023 — Board Query 1 Questions and Answers for FINAL COMP XM EXAM. CompXM Capsim Examination Notes - BOD QUIZ Q1) ... Q1) Rank the following companies from high to low cumulative profit, (in descending order, 1=highest,. 4=lowest). Answer 1) From Selected Financial Statistic ... Board Query 1 Questions for FINAL COMP XM EXAM.pdf The rise in the labour cost increase the price of the Jacket and the quality of the supply remain unchanged. Is this a violation of the law of supply? Explain. COMPXM answers 2024 This article provides COMPXM answers 2024 template. It offers answers for round 1 and guide make decisions for remaining comp XM rounds. This comp-xm guide ... 7 Comp-XM The Comp-XM Competency Exam is built around a simulation similar to Capstone and Foundation. ... This makes the questions comparable but the answers unique.