

Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

Computational Fluid Dynamics (CFD) is an important design tool in engineering and also a substantial research tool in various physical sciences as well as in biology. The objective of this book is to provide university students with a solid foundation for understanding the numerical methods employed in today's CFD and to familiarise them with modern CFD codes by hands-on experience. It is also intended for engineers and scientists starting to work in the field of CFD or for those who apply CFD codes. Due to the detailed index, the text can serve as a reference handbook too. Each chapter includes an extensive bibliography, which provides an excellent basis for further studies.

In this translation of the German edition, the authors provide insight into the numerical simulation of fluid flow. Using a simple numerical method as an expository example, the individual steps of scientific computing are presented: the derivation of the mathematical model; the discretization of the model equations; the development of algorithms; parallelization; and visualization of the computed data. In addition to the treatment of the basic equations for modeling laminar, transient flow of viscous, incompressible fluids - the Navier-Stokes equations - the authors look at the simulation of free surface flows; energy and chemical transport; and turbulence. Readers

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

are enabled to write their own flow simulation program from scratch. The variety of applications is shown in several simulation results, including 92 black-and-white and 18 color illustrations. After reading this book, readers should be able to understand more enhanced algorithms of computational fluid dynamics and apply their new knowledge to other scientific fields.

Nowadays mathematical modeling and numerical simulations play an important role in life and natural science. Numerous researchers are working in developing different methods and techniques to help understand the behavior of very complex systems, from the brain activity with real importance in medicine to the turbulent flows with important applications in physics and engineering. This book presents an overview of some models, methods, and numerical computations that are useful for the applied research scientists and mathematicians, fluid tech engineers, and postgraduate students.

Since many processes in the food industry involve fluid flow and heat and mass transfer, Computational Fluid Dynamics (CFD) provides a powerful early-stage simulation tool for gaining a qualitative and quantitative assessment of the performance of food processing, allowing engineers to test concepts all the way through the development of a process or system. Published in 2007, the first edition was the first book to address the use of CFD in food processing applications, and its aims were to present a comprehensive review of CFD applications for the food industry and pinpoint the research and development trends in the development of

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

the technology; to provide the engineer and technologist working in research, development, and operations in the food industry with critical, comprehensive, and readily accessible information on the art and science of CFD; and to serve as an essential reference source to undergraduate and postgraduate students and researchers in universities and research institutions. This will continue to be the purpose of this second edition. In the second edition, in order to reflect the most recent research and development trends in the technology, only a few original chapters are updated with the latest developments. Therefore, this new edition mostly contains new chapters covering the analysis and optimization of cold chain facilities, simulation of thermal processing and modeling of heat exchangers, and CFD applications in other food processes.

The content of this book covers several up-to-date topics in fluid dynamics, computational modeling and its applications, and it is intended to serve as a general reference for scientists, engineers, and graduate students. The book is comprised of 30 chapters divided into 5 parts, which include: winds, building and risk prevention; multiphase flow, structures and gases; heat transfer, combustion and energy; medical and biomechanical applications; and other important themes. This book also provides a comprehensive overview of computational fluid dynamics and applications, without excluding experimental and theoretical aspects.

This handbook covers computational fluid dynamics from fundamentals to applications. This text provides a well documented critical survey of numerical methods for fluid

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

mechanics, and gives a state-of-the-art description of computational fluid mechanics, considering numerical analysis, computer technology, and visualization tools. The chapters in this book are invaluable tools for reaching a deeper understanding of the problems associated with the calculation of fluid motion in various situations: inviscid and viscous, incompressible and compressible, steady and unsteady, laminar and turbulent flows, as well as simple and complex geometries. Each chapter includes a related bibliography. Covers fundamentals and applications. Provides a deeper understanding of the problems associated with the calculation of fluid motion.

An introduction to CFD fundamentals and using commercial CFD software to solve engineering problems, designed for the wide variety of engineering students new to CFD, and for practicing engineers learning CFD for the first time. Combining an appropriate level of mathematical background, worked examples, computer screen shots, and step by step processes, this book walks the reader through modeling and computing, as well as interpreting CFD results. The first book in the field aimed at CFD users rather than developers. New to this edition: A more comprehensive coverage of CFD techniques including discretisation via finite element and spectral element as well as finite difference and finite volume methods and multigrid method. Coverage of different approaches to CFD grid generation in order to closely match how CFD meshing is being used in industry. Additional coverage of high-pressure fluid dynamics and meshless approach to provide a broader

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals, Catalyst Bed

overview of the application areas where CFD can be used. 20% new content

In this Special Issue, one review paper highlights the necessity of multiscale CFD, coupling micro- and macro-scales, for exchanging information at the interface of the two scales. Four research papers investigate the hydrodynamics, heat transfer, and chemical reactions of various processes using Eulerian CFD modeling. CFD models are attractive for industrial applications.

However, substantial efforts in physical modeling and numerical implementation are still required before their widespread implementation.

Fire and combustion presents a significant engineering challenge to mechanical, civil and dedicated fire engineers, as well as specialists in the process and chemical, safety, buildings and structural fields. We are reminded of the tragic outcomes of 'untenable' fire disasters such as at King's Cross underground station or Switzerland's St Gotthard tunnel. In these and many other cases, computational fluid dynamics (CFD) is at the forefront of active research into unravelling the probable causes of fires and helping to design structures and systems to ensure that they are less likely in the future. Computational fluid dynamics (CFD) is routinely used as an analysis tool in fire and combustion engineering as it possesses the ability to handle the complex geometries and characteristics of combustion and fire. This book shows engineering students and professionals how to understand and use this powerful tool in the study of combustion processes, and in the engineering of safer or more fire resistant (or conversely,

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

more fire-efficient) structures. No other book is dedicated to computer-based fire dynamics tools and systems. It is supported by a rigorous pedagogy, including worked examples to illustrate the capabilities of different models, an introduction to the essential aspects of fire physics, examination and self-test exercises, fully worked solutions and a suite of accompanying software for use in industry standard modeling systems. · Computational Fluid Dynamics (CFD) is widely used in engineering analysis; this is the only book dedicated to CFD modeling analysis in fire and combustion engineering · Strong pedagogic features mean this book can be used as a text for graduate level mechanical, civil, structural and fire engineering courses, while its coverage of the latest techniques and industry standard software make it an important reference for researchers and professional engineers in the mechanical and structural sectors, and by fire engineers, safety consultants and regulators · Strong author team (CUHK is a recognized centre of excellence in fire eng) deliver an expert package for students and professionals, showing both theory and applications. Accompanied by CFD modeling code and ready to use simulations to run in industry-standard ANSYS-CFX and Fluent software.

Computational Fluid Dynamics enables engineers to model and predict fluid flow in powerful, visually impressive ways and is one of the core engineering design tools, essential to the study and future work of many engineers. This textbook is designed to explicitly meet the needs engineering students taking a first course in CFD or computer-aided engineering. Fully

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals, Catalyst Bed

course matched, with the most extensive and rigorous pedagogy and features of any book in the field, it is certain to be a key text. The only course text available specifically designed to give an applications-lead, commercial software oriented approach to understanding and using Computational Fluid Dynamics (CFD). Meets the needs of all engineering disciplines that use CFD. The perfect CFD teaching resource: clear, straightforward text, step-by-step explanation of mathematical foundations, detailed worked examples, end-of-chapter knowledge check exercises, and homework assignment questions

Computational Fluid Dynamics A Practical Approach Butterworth-Heinemann

This book introduces readers to the fundamentals of simulating and analyzing built and natural environments using the Computational Fluid Dynamics (CFD) method. CFD offers a powerful tool for dealing with various scientific and engineering problems and is widely used in diverse industries. This book focuses on the most important aspects of applying CFD to the study of urban, buildings, and indoor and outdoor environments. Following the logical procedure used to prepare a CFD simulation, the book covers e.g. the governing equations, boundary conditions, numerical methods, modeling of different fluid flows, and various turbulence models. Furthermore, it demonstrates how CFD can be applied to solve a range of engineering problems, providing detailed hands-on exercises on air and water flow, heat transfer, and pollution dispersion problems that typically arise in the study of buildings and environments. The

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

book also includes practical guidance on analyzing and reporting CFD results, as well as writing CFD reports/papers.

Computational Fluid Dynamics: Principles and Applications, Third Edition presents students, engineers, and scientists with all they need to gain a solid understanding of the numerical methods and principles underlying modern computation techniques in fluid dynamics. By providing complete coverage of the essential knowledge required in order to write codes or understand commercial codes, the book gives the reader an overview of fundamentals and solution strategies in the early chapters before moving on to cover the details of different solution techniques. This updated edition includes new worked programming examples, expanded coverage and recent literature regarding incompressible flows, the Discontinuous Galerkin Method, the Lattice Boltzmann Method, higher-order spatial schemes, implicit Runge-Kutta methods and parallelization. An accompanying companion website contains the sources of 1-D and 2-D Euler and Navier-Stokes flow solvers (structured and unstructured) and grid generators, along with tools for Von Neumann stability analysis of 1-D model equations and examples of various parallelization techniques. Will provide you with the knowledge required to develop and understand modern flow simulation codes Features new worked programming examples and expanded coverage of incompressible flows, implicit Runge-Kutta methods and code parallelization, among other topics Includes accompanying companion website that contains the sources of 1-D and 2-D flow solvers as

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

well as grid generators and examples of parallelization techniques

This textbook explores both the theoretical foundation of the Finite Volume Method (FVM) and its applications in Computational Fluid Dynamics (CFD). Readers will discover a thorough explanation of the FVM numerics and algorithms used for the simulation of incompressible and compressible fluid flows, along with a detailed examination of the components needed for the development of a collocated unstructured pressure-based CFD solver. Two particular CFD codes are explored. The first is uFVM, a three-dimensional unstructured pressure-based finite volume academic CFD code, implemented within Matlab. The second is OpenFOAM®, an open source framework used in the development of a range of CFD programs for the simulation of industrial scale flow problems. With over 220 figures, numerous examples and more than one hundred exercise on FVM numerics, programming, and applications, this textbook is suitable for use in an introductory course on the FVM, in an advanced course on numerics, and as a reference for CFD programmers and researchers.

Although many books have been written on computational fluid dynamics (CFD) and many written on combustion, most contain very limited coverage of the combination of CFD and industrial combustion.

Furthermore, most of these books are written at an advanced academic level, emphasize theory over practice, and provide little help to engineers who need to use CFD for combustion modeling. Computational Fluid

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

Dynamics in Industrial Combustion fills this gap in the literature. Focusing on topics of interest to the practicing engineer, it codifies the many relevant books, papers, and reports written on this combined subject into a single, coherent reference. It looks at each topic from a somewhat narrow perspective to see how that topic affects modeling in industrial combustion. The editor and his team of expert authors address these topics within three main sections: Modeling Techniques-The basics of CFD modeling in combustion Industrial Applications-Specific applications of CFD in the steel, aluminum, glass, gas turbine, and petrochemical industries Advanced Techniques-Subjects rarely addressed in other texts, including design optimization, simulation, and visualization Rapid increases in computing power and significant advances in commercial CFD codes have led to a tremendous increase in the application of CFD to industrial combustion. Thorough and clearly representing the techniques and issues confronted in industry, Computational Fluid Dynamics in Industrial Combustion will help bring you quickly up to date on current methods and gain the ability to set up and solve the various types of problems you will encounter.

Covered from the vantage point of a user of a commercial flow package, Essentials of Computational Fluid Dynamics provides the information needed to competently operate a commercial flow solver. This book provides a physical description of fluid flow, outlines the strengths and weaknesses of computational fluid dynamics (CFD), presents the basics of the discretization of the equations, focuses on the understanding of how

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals, Catalyst Bed

the flow physics interact with a typical finite-volume discretization, and highlights the approximate nature of CFD. It emphasizes how the physical concepts (mass conservation or momentum balance) are reflected in the CFD solutions while minimizing the required mathematical/numerical background. In addition, it uses cases studies in mechanical/aero and biomedical engineering, includes MATLAB and spreadsheet examples, codes and exercise questions. The book also provides practical demonstrations on core principles and key behaviors and incorporates a wide range of colorful examples of CFD simulations in various fields of engineering. In addition, this author: Introduces basic discretizations, the linear advection equation, and forward, backward and central differences Proposes a prototype discretization (first-order upwind) implemented in a spreadsheet/MATLAB example that highlights the diffusive character Looks at consistency, truncation error, and order of accuracy Analyzes the truncation error of the forward, backward, central differences using simple Taylor analysis Demonstrates how the of upwinding produces Artificial Viscosity (AV) and its importance for stability Explains how to select boundary conditions based on physical considerations Illustrates these concepts in a number of carefully discussed case studies

Essentials of Computational Fluid Dynamics provides a solid introduction to the basic principles of practical CFD and serves as a resource for students in mechanical or aerospace engineering taking a first CFD course as well as practicing professionals needing a brief, accessible introduction to CFD.

Bridging the gap in understanding between the spray drying industry and the numerical modeler on spray drying, *Computational Fluid Dynamics Simulation of Spray Dryers: An Engineer's Guide* shows how to numerically capture important physical phenomena within a spray drying process using the CFD technique. It includes numerical strategies to effectively describe these phenomena, which are collated from research work and CFD industrial consultation, in particular to the dairy industry. Along with showing how to set up models, the book helps readers identify the capabilities and uncertainties of the CFD technique for spray drying. After briefly covering the basics of CFD, the book discusses airflow modeling, atomization and particle tracking, droplet drying, quality modeling, agglomeration and wall deposition modeling, and simulation validation techniques. The book also answers questions related to common challenges in industrial applications.

Computational fluid dynamics, CFD, has become an indispensable tool for many engineers. This book gives an introduction to CFD simulations of turbulence, mixing, reaction, combustion and multiphase flows. The emphasis on understanding the physics of these flows helps the engineer to select appropriate models to obtain reliable simulations. Besides presenting the equations involved, the basics and limitations of the models are explained and discussed. The book combined with tutorials, project and power-point lecture notes (all available for download) forms a complete course. The reader is given hands-on experience of drawing, meshing and simulation. The tutorials cover flow and

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

reactions inside a porous catalyst, combustion in turbulent non-premixed flow, and multiphase simulation of evaporation spray respectively. The project deals with design of an industrial-scale selective catalytic reduction process and allows the reader to explore various design improvements and apply best practice guidelines in the CFD simulations.

Computational Fluid Dynamics: A Practical Approach, Third Edition, is an introduction to CFD fundamentals and commercial CFD software to solve engineering problems. The book is designed for a wide variety of engineering students new to CFD, and for practicing engineers learning CFD for the first time. Combining an appropriate level of mathematical background, worked examples, computer screen shots, and step-by-step processes, this book walks the reader through modeling and computing, as well as interpreting CFD results. This new edition has been updated throughout, with new content and improved figures, examples and problems. Includes a new chapter on practical guidelines for mesh generation Provides full coverage of high-pressure fluid dynamics and the meshless approach to provide a broader overview of the application areas where CFD can be used Includes online resources with a new bonus chapter featuring detailed case studies and the latest developments in CFD

Computational mechanics is a scientific discipline that marries physics, computers, and mathematics to emulate natural physical phenomena. It is a technology that allows scientists to study and predict the performance of various products--important for research and

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

development in the industrialized world. This book describes current trends and future research directions in computational mechanics in areas where gaps exist in current knowledge and where major advances are crucial to continued technological developments in the United States.

Provides a clear, concise, and self-contained introduction to Computational Fluid Dynamics (CFD) This comprehensively updated new edition covers the fundamental concepts and main methods of modern Computational Fluid Dynamics (CFD). With expert guidance and a wealth of useful techniques, the book offers a clear, concise, and accessible account of the essentials needed to perform and interpret a CFD analysis. The new edition adds a plethora of new information on such topics as the techniques of interpolation, finite volume discretization on unstructured grids, projection methods, and RANS turbulence modeling. The book has been thoroughly edited to improve clarity and to reflect the recent changes in the practice of CFD. It also features a large number of new end-of-chapter problems. All the attractive features that have contributed to the success of the first edition are retained by this version. The book remains an indispensable guide, which: Introduces CFD to students and working professionals in the areas of practical applications, such as mechanical, civil, chemical, biomedical, or environmental engineering Focuses on the needs of someone who wants to apply existing CFD software and understand how it works, rather than develop new codes Covers all the essential topics, from

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

the basics of discretization to turbulence modeling and uncertainty analysis Discusses complex issues using simple worked examples and reinforces learning with problems Is accompanied by a website hosting lecture presentations and a solution manual Essential Computational Fluid Dynamics, Second Edition is an ideal textbook for senior undergraduate and graduate students taking their first course on CFD. It is also a useful reference for engineers and scientists working with CFD applications.

Instabilities of fluid flows and the associated transitions between different possible flow states provide a fascinating set of problems that have attracted researchers for over a hundred years. This book addresses state-of-the-art developments in numerical techniques for computational modelling of fluid instabilities and related bifurcation structures, as well as providing comprehensive reviews of recently solved challenging problems in the field.

Direct Modeling for Computational Fluid Dynamics --
Introduction to Gas Kinetic Theory -- Introduction to Nonequilibrium Flow Simulations -- Gas Kinetic Scheme -- Unified Gas Kinetic Scheme -- Low Speed Microflow Studies -- High Speed Flow Studies -- Unified Gas Kinetic Scheme for Diatomic Gas -- Conclusion -- Appendix A: Non-dimensionalizing fluid dynamic variables -- Appendix B. Connection between BGK, Navier Stokes and Euler equations -- Appendix C. Moments of Maxwellian distribution

function and expansion coefficients -- Appendix D.
Flux evaluation through stationary and moving cell
interfaces

This book provides an introduction, overview, and
specific examples of computational fluid dynamics
and their applications in the water, wastewater, and
stormwater industry.

The chosen semi-discrete approach of a reduction
procedure of partial differential equations to ordinary
differential equations and finally to difference
equations gives the book its distinctiveness and
provides a sound basis for a deep understanding of
the fundamental concepts in computational fluid
dynamics.

Uniquely outlines CFD theory in a manner relevant
to environmental applications. This book addresses
the basic topics in CFD modelling in a thematic
manner to provided the necessary theoretical
background, as well as providing global cases
studies showing how CFD models can be used in
practice demonstrating how good practice can be
achieved , with reference to both established and
new applications. First book to apply CFD to the
environmental sciences Written at a level suitable for
non-mathematicians

This book contains twelve chapters detailing
significant advances and applications in fluid
dynamics modeling with focus on biomedical,
bioengineering, chemical, civil and environmental

engineering, aeronautics, astronautics, and automotive. We hope this book can be a useful resource to scientists and engineers who are interested in fundamentals and applications of fluid dynamics.

This book is the result of a careful selection of contributors in the field of CFD. It is divided into three sections according to the purpose and approaches used in the development of the contributions. The first section describes the "high-performance computing" (HPC) tools and their impact on CFD modeling. The second section is dedicated to "CFD models for local and large-scale industrial phenomena." Two types of approaches are basically contained here: one concerns the adaptation from global to local scale, - e.g., the applications of CFD to study the climate changes and the adaptations to local scale. The second approach, very challenging, is the multiscale analysis. The third section is devoted to "CFD in numerical modeling approach for experimental cases." Its chapters emphasize on the numerical approach of the mathematical models associated to few experimental (industrial) cases. Here, the impact and the importance of the mathematical modeling in CFD are focused on. It is expected that the collection of these chapters will enrich the state of the art in the CFD domain and its applications in a lot of fields. This collection proves that CFD is a highly

interdisciplinary research area, which lies at the interface of physics, engineering, applied mathematics, and computer science.

The book *Modeling in Membranes and Membrane-Based Processes* is based on the idea of developing a reference which will cover most relevant and “state-of-the-art” approaches in membrane modeling. This book explores almost every major aspect of modeling and the techniques applied in membrane separation studies and applications. This includes first principle-based models, thermodynamics models, computational fluid dynamics simulations, molecular dynamics simulations, and artificial intelligence-based modeling for membrane separation processes. These models have been discussed in light of various applications ranging from desalination to gas separation. In addition, this breakthrough new volume covers the fundamentals of polymer membrane pore formation mechanisms, covering not only a wide range of modeling techniques, but also has various facets of membrane-based applications. Thus, this book can be an excellent source for a holistic perspective on membranes in general, as well as a comprehensive and valuable reference work. Whether a veteran engineer in the field or lab or a student in chemical or process engineering, this latest volume in the “Advances in Membrane Processes” is a must-have, along with the first book in the series, *Membrane Processes*, also available from Wiley-Scrivener.

This volume presents the results of Computational Fluid Dynamics (CFD) analysis that can be used for

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

conceptual studies of product design, detail product development, process troubleshooting. It demonstrates the benefit of CFD modeling as a cost saving, timely, safe and easy to scale-up methodology.

This unique text provides engineering students and practicing professionals with a comprehensive set of practical, hands-on guidelines and dozens of step-by-step examples for performing state-of-the-art, reliable computational fluid dynamics (CFD) and turbulence modeling. Key CFD and turbulence programs are included as well. The text first reviews basic CFD theory, and then details advanced applied theories for estimating turbulence, including new algorithms created by the author. The book gives practical advice on selecting appropriate turbulence models and presents best CFD practices for modeling and generating reliable simulations. The author gathered and developed the book's hundreds of tips, tricks, and examples over three decades of research and development at three national laboratories and at the University of New Mexico—many in print for the first time in this book. The book also places a strong emphasis on recent CFD and turbulence advancements found in the literature over the past five to 10 years. Readers can apply the author's advice and insights whether using commercial or national laboratory software such as ANSYS Fluent, STAR-CCM, COMSOL, Flownex, SimScale, OpenFOAM, Fuego, KIVA, BIGHORN, or their own computational tools. Applied Computational Fluid Dynamics and Turbulence Modeling is a practical, complementary companion for academic CFD textbooks and senior project courses in mechanical,

Online Library Computational Fluid Dynamics Modeling Of Trickle Bed Reactor Hydrodynamics Reactor Internals Catalyst Bed

civil, chemical, and nuclear engineering; senior undergraduate and graduate CFD and turbulence modeling courses; and for professionals developing commercial and research applications.

This book serves as a complete and self-contained introduction to the principles of Computational Fluid Dynamic (CFD) analysis. It is deliberately short (at approximately 300 pages) and can be used as a text for the first part of the course of applied CFD followed by a software tutorial. The main objectives of this non-traditional format are: 1) To introduce and explain, using simple examples where possible, the principles and methods of CFD analysis and to demystify the 'black box' of a CFD software tool, and 2) To provide a basic understanding of how CFD problems are set and which factors affect the success and failure of the analysis. Included in the text are the mathematical and physical foundations of CFD, formulation of CFD problems, basic principles of numerical approximation (grids, consistency, convergence, stability, and order of approximation, etc), methods of discretization with focus on finite difference and finite volume techniques, methods of solution of transient and steady state problems, commonly used numerical methods for heat transfer and fluid flows, plus a brief introduction into turbulence modeling.

[Copyright: c90fb726d392b33c1c61e424679b30ee](http://c90fb726d392b33c1c61e424679b30ee)