Wave Simulation Fluent Tutorial

BIM Handbook
Water Wave Mechanics For Engineers And Scientists
Multiphysics Modeling with Application to Biomedical Engineering
Ocean Wave Energy Systems
Mechanical Components and Control Engineering III
Advances in Fluid Dynamics
Numerical Study of the Hydrodynamic Performance of a Point-Absorbing Wave Energy Converter
Multiphysics Simulation by Design for Electrical Machines, Power Electronics and Drives
CFD Simulations of Marine Hydrodynamics
Numerical Study on Scaling Effects and Decoupled Network-based Simulation of Gaseous Explosion
Monthly Catalog of United States Government Publications
Computational Methods for Fluid Dynamics
Modeling and Simulation Fundamentals
Scientific and Technical Aerospace Reports

Ocean Wave Energy
Aeroacoustics of Low Mach Number Flows
Advances in Hybrid RANS-LES Modelling
An Introduction to Hydrodynamics and Water Waves
Research on Ship Design and Optimization Based on Simulation-Based Design (SBD) Technique
Hydrodynamics Around Cylindrical Structures
The Software Encyclopedia
Luces en el camino: filosofía y ciencias sociales en tiempos de desconcierto
Investigation of Wings in Ground Effect using Computational Fluid Dynamics
Handbook of Aseptic Processing and Packaging
Maritime Technology and Engineering 5 Volume 2
Transfer Phenomena in Fluid and Heat Flows X
Monthly Catalogue, United States Public Documents
Wave Ship Interaction in Transforming Seas
Design, Application, Performance and Emissions of Modern Internal Combustion Engine Systems and Components
Proceedings of the 5th International Conference on Industrial Engineering (ICIE 2019)
Multiphase Flow Dynamics 2
Handbook of Ocean Wave Energy

Renewable and Sustainable Energy II
Numerical Modelling of Wave Energy Converters
Evaluation of Liquid Fuel Spray Models for Hybrid RANS/LES and DLES Prediction of Turbulent Reactive Flows
Evaluation of Various Turbulence Models for Shock-wave Boundary Layer Interaction Flows
Scramjet Propulsion
The Multi-Agent Transport Simulation MATSim
An Introduction to Computational Fluid Dynamics The Finite Volume Method, 2/e
An Introduction to ANSYS Fluent 2020

SHEPARD VILLEGAS

BIM Handbook Trans Tech Publications Ltd
This dissertation, "Numerical Study of the Hydrodynamic Performance of a Point-absorbing Wave Energy Converter" by Qingjie, Du, □□□, was obtained from The University of Hong Kong (Pokfulam, Hong Kong) and is being sold pursuant to Creative Commons: Attribution 3.0 Hong Kong License. The content of this dissertation has not been altered in any way. We have altered the formatting in order to facilitate the ease of printing and reading of the dissertation. All rights

not granted by the above license are retained by the author. Abstract: As fossil energy is depleting and global warming effect is worsening rapidly, developing renewable energies becomes the top priority in most countries. In recent years, wave energy has attracted more and more attention due to its high energy density and enormous global capacity. The goal of this study is to carry out a numerical study of the hydrodynamic performance of a point-absorbing wave energy converter. In this study, an accurate and efficient numerical wave fume was established first. Commercial software code FLUENT?, which is a state-of-the-art computer program package for modeling fluid flow and heat transfer, was used for the numerical simulation. Based on the Navier-Stokes equations for viscous, incompressible fluid and Volume of fluid (VOF) method, a numerical wave tank was developed. Dynamic meshing method was used to simulate the wavemaker, and Geo-Reconstruct scheme was used to capture and reconstruct the free surface. A wave-absorbing method employing porous medium model was proposed to act as the wave absorbing beach, which can absorb the wave energy efficiently. A series of regular waves were simulated using the proposed numerical method. Validation has been made by physical experiments. After developing the wave flume model, a cylinder, which represents the

point-absorbing wave energy converter (WEC), was added into the wave flume. The hydrodynamic behavior of the WEC was studied. The numerical results were also compared with physical experiments. Based on the numerical simulation results, suggestions on optimizing the point-absorber are provided. In this study, eight wave cases, with different wave period and wave length were simulated. The results show that the numerical simulation can match well with the physical wave tank result. Both the wave height and wave period in different cases can match well between the numerical simulation and physical wave tank results. In the wave-cylinder simulation, the results also show a good match in the numerical study and physical study. This numerical model is very significant in ocean structure design. The cylinder tested in this study can be easily changed to a ship or an offshore-platform. Compared with the physical experiment, numerical simulation is more flexible. The simulation can be carried on a large time span and spatial scale. The geometry can be changed easily. Also the cost of numerical simulation is relatively cheap compared with the physical experiment. DOI: 10.5353/th_b4715284 Subjects: Hydroelectric generators Ocean wave power

Water Wave Mechanics For Engineers And Scientists GRIN Verlag

The MATSim (Multi-Agent

Transport Simulation) software project was started around 2006 with the goal of generating traffic and congestion patterns by following individual synthetic travelers through their daily or weekly activity programme. It has since then evolved from a collection of stand-alone C++ programs to an integrated Java-based framework which is publicly hosted, open-source available, automatically regression tested. It is currently used by about 40 groups throughout the world. This book takes stock of the current status. The first part of the book gives an introduction to the most important concepts, with the intention of enabling a potential user to set up and run basic simulations. The second part of the book describes how the basic functionality can be extended, for example by adding schedule-based public transit, electric or autonomous cars, paratransit, or within-day replanning. For each extension, the text provides pointers to the additional documentation and to the code base. It is also discussed how people with appropriate Java programming skills can write their own extensions, and plug them into the MATSim core. The project has started from the basic idea that traffic is a consequence of human behavior, and thus humans and their behavior should be the starting point of all modelling, and with the intuition that when simulations with 100 million particles are possible in computational

physics, then behavior-oriented simulations with 10 million travelers should be possible in travel behavior research. The initial implementations thus combined concepts from computational physics and complex adaptive systems with concepts from travel behavior research. The third part of the book looks at theoretical concepts that are able to describe important aspects of the simulation system; for example, under certain conditions the code becomes a Monte Carlo engine sampling from a discrete choice model. Another important aspect is the interpretation of the MATSim score as utility in the microeconomic sense, opening up a connection to benefit cost analysis. Finally, the book collects use cases as they have been undertaken with MATSim. All current users of MATSim were invited to submit their work, and many followed with sometimes crisp and short and sometimes longer contributions, always with pointers to additional references. We hope that the book will become an invitation to explore, to build and to extend agent-based modeling of travel behavior from the stable and well tested core of MATSim documented here.

Multiphysics Modeling with Application to Biomedical Engineering

Springer Nature

Discover BIM: A better way to build better buildings Building Information Modeling (BIM) offers a novel approach to design, construction, and facility

management in which a digital representation of the building product and process is used to facilitate the exchange and interoperability of information in digital format. BIM is beginning to change the way buildings look, the way they function, and the ways in which they are designed and built. The BIM Handbook, Third Edition provides an in-depth understanding of BIM technologies, the business and organizational issues associated with its implementation, and the profound advantages that effective use of BIM can provide to all members of a project team. Updates to this edition include: Information on the ways in which professionals should use BIM to gain maximum value New topics such as collaborative working, national and major construction clients, BIM standards and guides A discussion on how various professional roles have expanded through the widespread use and the new avenues of BIM practices and services A wealth of new case studies that clearly illustrate exactly how BIM is applied in a wide variety of conditions Painting a colorful and thorough picture of the state of the art in building information modeling, the BIM Handbook, Third Edition guides readers to successful implementations, helping them to avoid needless frustration and costs and take full advantage of this paradigm-shifting approach to construct better buildings that

consume fewer materials and require less time, labor, and capital resources.

Ocean Wave Energy Systems Springer Science & Business Media This special issue Transfer Phenomena in Fluid and Heat Flows X in the journal "Defect and Diffusion Forum" presents a collection of peer-reviewed works associated with diffusion phenomena, the motion of the fluid flow and heat transfer in the technical and natural systems.

Mechanical Components and Control Engineering III CRC Press La sociedad contemporánea se enfrenta, parca en ropajes, al vendaval provocado por diversas crisis simultáneas: crisis sanitaria, crisis económica, crisis política, crisis educativa, crisis climática, etc. En esta situación, corresponde al mundo de la academia, al ámbito erudito e intelectual el tratar de hacerse valer y demostrar la importancia que tiene para afrontar el desafío del futuro. En la biografía de cualquier persona suele haber alguien que ejerce una influencia benéfica especial.Esos individuos actúan como luces en el camino que orientan en las, a menudo, procelosas situaciones vitales que nos toca vivir. Precisamente eso es lo que nos proponemos en este volumen que, a falta de un término más adecuado, podemos calificar como monumental. Pretendemos ser una luz en el camino del lector

que se aproxime a los diversos estudios que se acogen en estas páginas. Aquí se encuentran los resultados de investigaciones muy variopintas que tratan, en la medida de sus posibilidades, de dar respuestas a interrogantes diversos que atañen al quehacer cotidiano, a la forma de entender la vida y la realidad, en definitiva, a la construcción del mundo.

Advances in Fluid Dynamics John Wiley & Sons

As an engineer, you may need to test how a design interacts with fluids. For example, you may need to simulate how air flows over an aircraft wing, how water flows through a filter, or how water seeps under a dam. Carrying out simulations is often a critical step in verifying that a design will be successful. In this hands-on book, you'll learn in detail how to run Computational Fluid Dynamics (CFD) simulations using ANSYS Fluent. ANSYS Fluent is known for its power, simplicity and speed, which has helped make it a world leader in CFD software, both in academia and industry. Unlike any other ANSYS Fluent textbook currently on the market, this book uses applied problems to walk you step-by-step through completing CFD simulations for many common flow cases, including internal and external flows, laminar and turbulent flows, steady and unsteady flows, and single-phase and multiphase flows. You will also learn how to

visualize the computed flows in the post-processing phase using different types of plots. To better understand the mathematical models being applied, we'll validate the results from ANSYS Fluent with numerical solutions calculated using Mathematica. Throughout this book we'll learn how to create geometry using ANSYS Workbench and ANSYS DesignModeler, how to create mesh using ANSYS Meshing, how to use physical models and how to perform calculations using ANSYS Fluent. The twenty chapters in this book can be used in any order and are suitable for beginners with little or no previous experience using ANSYS. Intermediate users, already familiar with the basics of ANSYS Fluent, will still find new areas to explore and learn. An Introduction to ANSYS Fluent 2020 is designed to be used as a supplement to undergraduate courses in Aerodynamics, Finite Element Methods and Fluid Mechanics and is suitable for graduate level courses such as Viscous Fluid Flows and Hydrodynamic Stability. The use of CFD simulation software is rapidly growing in all industries. Companies are now expecting graduating engineers to have knowledge of how to perform simulations. Even if you don't eventually complete simulations yourself, understanding the process used to complete these simulations is necessary to be an effective team member. People with experience using ANSYS

Fluent are highly sought after in the industry, so learning this software will not only give you an advantage in your classes, but also when applying for jobs and in the workplace. This book is a valuable tool that will help you master ANSYS Fluent and better understand the underlying theory.

Numerical Study of the Hydrodynamic Performance of a Point-Absorbing Wave Energy Converter Mdpi AG

Turbulence modelling has long been, and will remain, one of the most important t- ics in turbulence research, challenging scientists and engineers in the academic world and in the industrial society. Over the past decade, Detached Eddy Simulation (DES) and other hybrid RANS-LES methods have received increasing attention from the turbulence-research community, as well as from industrial CFD engineers. Indeed, as an engineering modelling approach, hybrid RANS-LES methods have acquired a remarkable profile in modelling turbulent flows of industrial interest in relation to, for example, transportation, energy production and the environment. The advantage exploited with hybrid RANS-LES modelling approaches, being - tentially more computationally efficient than LES and more accurate than (unsteady) RANS, has motivated numerous research and development activities. These activities, together with industrial

applications, have been further facilitated over the recent years by the rapid development of modern computing resources. As a European initiative, the EU project DESider (Detached Eddy Simulation for Industrial Aerodynamics, 2004-2007), has been one of the earliest and most systematic international R&D effort with its focus on development, improvement and applications of a variety of existing and new hybrid RANS-LES modelling approaches, as well as on related numerical issues. In association with the DESider project, two subsequent international symposia on hybrid RANS-LES methods have been arranged in Stockholm (Sweden, 2005) and in Corfu (Greece, 2007), respectively. The present book is a result of the Second Symposium on Hybrid RANS-LES Methods, held in Corfu, Greece, 17-18 June 2007. *Multiphysics Simulation by Design for Electrical Machines, Power Electronics and Drives* AIAA

"This research seeks to improve the prediction efficiency of gaseous explosions realized by numerical simulations in a full-scale underground network using a decoupled method. To provide quick predictions of overpressure distribution of methane explosions in underground airway networks, a two-section theory is employed. The explosion space is divided into a driver section and a blast-wave section. Governing equations including conservation of mass,

momentum, and energy, together with chemical reaction and turbulence models are solved for the driver and the blast-wave sections using computational fluid dynamics (CFD) solver ANSYS Fluent (3D-based) and Flowmaster (1D-based) respectively. The three dimensional (3D) and one dimensional (1D) numerical analyses are preceded separately (decoupled). In the driver section, the numerical calculation results with three variables (FLSF, HDSF, and concentration) considering the size of explosion space and methane concentration level for the driver section are stored in a database tool Microsoft SQL Server Express aims to generate a methane explosion source database. To validate the selected combustion and turbulent models, a series of lab-scale methane explosion experiments were conducted. In the blast-wave section, the influences of geometric changes are quantified by using 2D Euler equations, whereas the simulation results are used to adjust the 1D network-based modeling. The decoupled method is applied in two case studies and proved capable to predict the pressure distribution of methane explosions that occurs in a complex airway network."--Abstract, page iii.

CFD Simulations of Marine Hydrodynamics

Ubiquity Press

Multi-phase flows are part of our natural environment such as tornadoes, typhoons, air and water pollution and volcanic activities as well

as part of industrial technology such as power plants, combustion engines, propulsion systems, or chemical and biological industry. The industrial use of multi-phase systems requires analytical and numerical strategies for predicting their behavior. In its third extended edition this book contains theory, methods and practical experience for describing complex transient multi-phase processes in arbitrary geometrical configurations. This book provides a systematic presentation of the theory and practice of numerical multi-phase fluid dynamics. In the present second volume the mechanical and thermal interactions in multiphase dynamics are provided. This third edition includes various updates, extensions, improvements and corrections.

Numerical Study on Scaling Effects and Decoupled Network-based Simulation of Gaseous Explosion John Wiley & Sons

An insightful presentation of the key concepts, paradigms, and applications of modeling and simulation Modeling and simulation has become an integral part of research and development across many fields of study, having evolved from a tool to a discipline in less than two decades. Modeling and Simulation Fundamentals offers a comprehensive and authoritative treatment of the topic and includes definitions, paradigms, and applications to equip readers with the skills needed to work

successfully as developers and users of modeling and simulation. Featuring contributions written by leading experts in the field, the book's fluid presentation builds from topic to topic and provides the foundation and theoretical underpinnings of modeling and simulation. First, an introduction to the topic is presented, including related terminology, examples of model development, and various domains of modeling and simulation. Subsequent chapters develop the necessary mathematical background needed to understand modeling and simulation topics, model types, and the importance of visualization. In addition, Monte Carlo simulation, continuous simulation, and discrete event simulation are thoroughly discussed, all of which are significant to a complete understanding of modeling and simulation. The book also features chapters that outline sophisticated methodologies, verification and validation, and the importance of interoperability. A related FTP site features color representations of the book's numerous figures. Modeling and Simulation Fundamentals encompasses a comprehensive study of the discipline and is an excellent book for modeling and simulation courses at the upper-undergraduate and graduate levels. It is also a valuable reference for researchers and practitioners in the fields of computational statistics, engineering,

and computer science who use statistical modeling techniques. Monthly Catalog of United States Government Publications Dykinson In near-shore transforming seas, as waves approach the shoreline, wave shoaling and sometimes wave breaking take place due to the decreasing water depth. When a ship advances through the transforming seas, the ship body and waves interact with each other substantially and can lead to unknown motions of the ship hull. The physical process of how the wave transforms in the surf zone and how the vehicle actually behaves when it passes through the transforming seas is a complicated issue that triggers considerable research interest. The goal of my research is to characterize the dynamics of a high-speed surface ship model in transforming seas through a parametric numerical study of the ship wave interactions. In this study, the vehicle of interest is a surface effect ship (SES) and we aim to contribute to developing a methodology for simulating the transforming wave environment, including wave breaking, and its interactions with the SES. The thesis work uses a commercial software package ANSYS Fluent to generate numerical waves and model the interface between water and air using the volume of fluid (VoF) method. A ship motion solver and the dynamic mesh are used to enable the modeled ship to perform three degree-

of-freedom (DoF) motion and the near-region of the ship hull to deform as well as re-mesh. Non-conformal meshes with hybrid compositions of different cell types and various grid sizes are used in the simulations for different purposes. Five user-defined functions (UDFs) are dynamically linked with the flow solver to incorporates ship/grid motions, wave damping and output of the numerical results. A series of steps were taken sequentially: 1) validation for ship motions including simulation of a static Wigley hull under steady flows to compare against previous experimental results by other researchers, and the comparison between the static SES model under steady flows and the moving SES model advancing in the calm water; 2) study of the ship with 3 DoF advancing in calm water of both constant depth and varying depth; 3) validation for numerical waves, including predictions of numerically progressive waves in both a regular tank and a tank with a sloped fringing reef to compare with theoretical and experimental results, respectively; 4) investigation of the transforming characteristics of the wave traveling over the sloped fringing reef, which mimics the near-shore wave environment and a study of the dynamics of the SES through transforming waves. We find that the flow solver used in this study reliably models the wave profiles along the ship hull. The

comparison between a static SES in a current and a moving SES in calm water at the same Froude number shows that although the velocity fields around the vehicle are significantly different, the wave profiles inside and outside the rigid cushion of the vehicle are similar and the resistance force experienced by the vehicle in the two scenarios agree well over time. We conducted five numerical simulations of the vehicle traveling from shallow water to deep water across the transition zone for different Froude numbers. From the results, we find that as the Froude number increases, the wave resistance force on the vehicle becomes larger in both shallow water and deep water. In addition, the overall mean resistance force experienced by the vehicle over the whole trip increases with the Froude number. Statistical analysis of the wave motions suggests that the energy flux decreases dramatically in the onshore direction as the waves break. The more severe the wave-breaking process, the greater the decrease in energy flux. Both the increase of Froude number and the wave steepness apparently increase the resistance force on the vehicle in the shallow water. This thesis work captures the impact of the transforming characteristics of the waves and closely replicates the behavior of how waves interact with a ship in transforming seas through numerical modeling and simulation.

Computational Methods for Fluid Dynamics
Academic Press
Despite the modeling capabilities of current computational fluid dynamics (CFD), there still exist problems and inconsistencies in simulating fluid flow in certain flow regimes. Most difficult are the high-speed transonic, supersonic and hypersonic wall-bounded turbulent flows with small or massive regions of separation. To address the problem of the lack of computational accuracy in turbulence modeling, NASA has established the Turbulence Modeling Resource (TMR) website and has issued the NASA 40% Challenge. The aim of this challenge is to identify and improve/develop turbulence and transition models as well as numerical techniques to achieve a 40% reduction in the predictive error in computation of benchmark test cases for turbulent flows. One of the phenomena of considerable interest in the 40% Challenge is the shock-wave boundary layer interaction (SWBLI) that occurs on aircraft surfaces at transonic and supersonic speeds and on space vehicles at hypersonic speeds. The correct modeling of shock-waves is complex enough, but the occurrence of SWBLI adds to the complexity by promoting flow separation, heat transfer, and pressure gradients on the surface. SWBLI may occur in both the external and internal flow path of air and space vehicles; therefore, it is important

to accurately predict this phenomenon to improve the design of aircraft and space vehicles. The majority of CFD codes utilize the Reynolds Averaged Navier-Stokes (RANS) equations and employ various turbulence models. The most common among these turbulent models are the one-equation Spalart-Allmaras (SA) model and the two-equation Shear Stress Transport (SST) k-[omega] model. In recent years the CFD community has, in greater number, also started to adopt Large-Eddy Simulation (LES), Direct Numerical Simulation (DNS), and hybrid RANS-LES approaches for improving the accuracy of simulations. However currently, solving the RANS equations with eddy-viscosity turbulence models remains the most commonly used simulation technique in industrial applications. In this research, the one-equation Wray-Agarwal (WA), SA, and SST k-[omega] turbulence models are used to simulate supersonic flows in a 2D compression corner at angles of 8° and 16°, a partial axisymmetric flare of 20°, a full-body conical axisymmetric flare of 20°, and an impinging shock over a flat plate at 6°, 10°, and 14°. The ANSYS Fluent and OpenFOAM flow solvers are employed. Inflow boundary conditions and mesh sensitivity are examined to ensure the grid independence of computed solutions. For each of the three turbulence models, heat

transfer, surface pressure, skin friction, and velocity profiles are compared with the available experimental data. It is found that the results from the WA model are in similar or better agreement with the experimental data compared to the SA and SST k-[omega] models for the majority of cases considered.

Modeling and Simulation Fundamentals CRC Press

This set of two volumes comprises the collection of the papers presented at the 5th International Conference on Maritime Technology and Engineering (MARTECH 2020) that was held in Lisbon, Portugal, from 16 to 19 November 2020. The Conference has evolved from the series of biennial national conferences in Portugal, which have become an international event, and which reflect the internationalization of the maritime sector and its activities. MARTECH 2020 is the fifth of this new series of biennial conferences. The set comprises 180 contributions that were reviewed by an International Scientific Committee. Volume 2 is dedicated to ship performance and hydrodynamics, including CFD, maneuvering, seakeeping, moorings and resistance. In addition, it includes sections on ship machinery, renewable energy, fishing and aquaculture, coastal structures, and waves and currents.

Scientific and Technical Aerospace Reports Academic Press

Numerical Modelling of

Wave Energy Converters: State-of-the Art Techniques for Single WEC and Converter Arrays presents all the information and techniques required for the numerical modelling of a wave energy converter together with a comparative review of the different available techniques. The authors provide clear details on the subject and guidance on its use for WEC design, covering topics such as boundary element methods, frequency domain models, spectral domain models, time domain models, non linear potential flow models, CFD models, semi analytical models, phase resolving wave propagation models, phase averaging wave propagation models, parametric design and control optimization, mean annual energy yield, hydrodynamic loads assessment, and environmental impact assessment. Each chapter starts by defining the fundamental principles underlying the numerical modelling technique and finishes with a discussion of the technique’s limitations and a summary of the main points in the chapter. The contents of the chapters are not limited to a description of the mathematics, but also include details and discussion of the current available tools, examples available in the literature, and verification, validation, and computational requirements. In this way, the key points of each modelling technique can be identified without

having to get deeply involved in the mathematical representation that is at the core of each chapter. The book is separated into four parts. The first two parts deal with modelling single wave energy converters; the third part considers the modelling of arrays; and the final part looks at the application of the different modelling techniques to the four most common uses of numerical models. It is ideal for graduate engineers and scientists interested in numerical modelling of wave energy converters, and decision-makers who must review different modelling techniques and assess their suitability and output. Consolidates in one volume information and techniques for the numerical modelling of wave energy converters and converter arrays, which has, up until now, been spread around multiple academic journals and conference proceedings making it difficult to access Presents a comparative review of the different numerical modelling techniques applied to wave energy converters, discussing their limitations, current available tools, examples, and verification, validation, and computational requirements Includes practical examples and simulations available for download at the book’s companion website Identifies key points of each modelling technique without getting deeply involved in the mathematical representation

Ocean Wave Energy
World Scientific
Collection of selected, peer reviewed papers from the 3rd Asian Pacific Conference on Mechanical Components and Control Engineering (MCCE 2014), September 20-21, 2014, Tianjin, China. Volume is indexed by Thomson Reuters CPCI-S (WoS). The 367 papers are grouped as follows: Chapter 1: Materials Science and Processing Technologies, Chapter 2: General Mechanical Engineering, Applied Mechanics and Dynamics, Chapter 3: Mechatronics and Robotics, Chapter 4: Control Technologies, Automation, Design and Simulation of Manufacturing, Chapter 5: Electrical Engineering and Electric Machines, Chapter 6: Power System and Energy Engineering, its Applications, Chapter 7: Electronics and Integrated Circuits, Embedded Technology and Applications, Chapter 8: Measurements, Testing, Monitoring, Analysis and Methodology, Chapter 9: Signal and Image Processing, Data Mining and Computational Mathematics, Chapter 10: Communication, Networks and Information Technologies, Chapter 11: Construction Technologies, Urban Planning and Urban Traffic, Chapter 12: Earth Science and Environmental Engineering, Chapter 13: Biomedical Engineering, Chapter 14: Product Design, Planning, Projects Management and Industrial Engineering

Aeroacoustics of Low Mach Number Flows
Springer Science &

Business Media
CFD is an emerging area and is gaining popularity due to the availability of ever-increasing computational power. If used accurately, CFD methods may overcome the limitations of experimental and other numerical methods, in some respects. This Special Issue focuses on Computational Fluid Dynamics (CFD) Simulations of Marine Hydrodynamics with a specific focus on the applications of naval architecture and ocean engineering, and it comprises 24 original articles that advance state-of-the-art CFD applications in marine hydrodynamics and/or review the progress and future directions of research in this field. The published articles cover a wide range of subjects relevant to naval architecture and ocean engineering, including but not limited to; ship resistance and propulsion, seakeeping and maneuverability, hydrodynamics of marine renewable energy devices, validation and verification of computational fluid dynamics (CFD), EFD/CFD combined methods, fouling/coating hydrodynamics.

Advances in Hybrid RANS-LES Modelling

World Scientific Publishing Company
This book discusses the subject of wave/current flow around a cylinder, the forces induced on the cylinder by the flow, and the vibration pattern of slender structures in a marine environment. The primary aim of the book is

to describe the flow pattern and the resulting load which develops when waves or current meet a cylinder. Attention is paid to the special case of a circular cylinder. The development in the forces is related to the various flow patterns and is discussed in detail. Regular as well as irregular waves are considered, and special cases like wall proximities (pipelines) are also investigated. The book is intended for MSc students with some experience in basic fluid mechanics and for PhD students. Contents:Flow Around a Cylinder in Steady CurrentForces on a Cylinder in Steady CurrentFlow Around a Cylinder in Oscillatory FlowsForces on a Cylinder in Regular WavesMathematical and Numerical Treatment of Flow Around a CylinderDiffraction Effect. Forces on Large BodiesForces on a Cylinder in Irregular WavesFlow-Induced Vibrations of a Free Cylinder in Steady CurrentsFlow-Induced Vibrations of a Free Cylinder in WavesVibrations of Marine PipelinesMathematical Modelling of Flow-Induced Vibrations. Readership: Civil and ocean engineers. keywords:Pipelines;Offshore Structures;Hydroelastic Vibrations;Flow-induced Vibrations;Forces on Offshore Structures;Flow Around Offshore Structures;Wave Loading;Vibrations;Waves; Steady Currents;Pipeline Stability;Diffraction;Irregular Waves;Oscillatory Flow;Mathematical

Modelling;Coastal Structures;Marine Structure;Flow Loading;Vibration of Marine Pipelines “The figures are very good. Many of them are photographs and sketches of aspects of flow that are sometimes difficult to explain in words. The references are extensive, quoting many recent papers. The treatment of the subjects is up-to-date and particularly the chapters on numerical simulation and vibrations contain excellent synopses of new research, much of it by the authors themselves. The style is lucid and the text is well-organized. This book can be highly recommended to anyone who deals with cylindrical structures.” Professor J W Kamphuis Coastal Engineering

An Introduction to Hydrodynamics and Water Waves Pearson Education India

Selected, peer reviewed papers from the 2012 International Conference on Energy and Environmental Protection (ICEEP 2012), June 23-24, 2012, Hohhot, China

Research on Ship Design and Optimization Based on Simulation-Based Design (SBD) Technique SDC Publications

This book is open access under a CC BY-NC 2.5 license. This book offers a concise, practice-oriented reference-guide to the field of ocean wave energy. The ten chapters highlight the key rules of thumb, address all the main technical engineering aspects and describe in detail all the key aspects to be

considered in the techno-economic assessment of wave energy converters. Written in an easy-to-understand style, the book answers questions relevant to readers of different backgrounds, from developers, private and public investors, to students and researchers. It is thereby a valuable resource for both newcomers and experienced practitioners in the wave energy sector.

Hydrodynamics Around Cylindrical Structures

Springer

Ship optimization design is critical to the preliminary design of a ship. With the rapid development of computer technology, the simulation-based design (SBD) technique has been introduced into the field of ship design. Typical SBD consists of three parts: geometric reconstruction; CFD numerical simulation; and optimization. In the context of ship design, these are used to alter the shape of the ship, evaluate the objective function and to assess the hull form space respectively. As such, the SBD technique opens up new opportunities and paves the way for a new method for optimal ship design. This book discusses the problem of optimizing ship's hulls, highlighting the key technologies of ship optimization design and presenting a series of hull-form optimization platforms. It includes several improved approaches and novel ideas with significant potential in this field