Handbook of Computational Fluid Mechanics

Most natural and industrial flows are turbulent. The atmosphere and oceans, automobile and aircraft engines, all provide examples of this ubiquitous phenomenon. In recent years, turbulence has become a very lively area of scientific research and application, and this work offers a grounding in the subject of turbulence, developing both the physical insight and the mathematical framework needed to express the theory. Providing a solid foundation in the key topics in turbulence, this valuable reference resource enables the reader to become a knowledgeable developer of predictive tools. This central and broad ranging topic would be of interest to graduate students in a broad range of subjects, including aeronautical and mechanical engineering, applied mathematics and the physical sciences. The accompanying solutions manual to the text also makes this a valuable teaching tool for lecturers and for practising engineers and scientists in computational and experimental and experimental fluid dynamics.

Analysis of Turbulent Flow with Computer Programs

Proceedings of the world renowned ERCOFTAC (International Symposium on Engineering Turbulence Modelling and Measurements). The proceedings include papers dealing with the following areas of turbulence: Eddy-viscosity and second-order RANS models, Direct and large-eddy simulations and deductions for conventional modelling, Measurement and visualization, experimental control, Transition and effects of curvature, rotation and buoyancy on turbulence, Aero-acoustics, Heat and mass transfer and chemically reacting flows, Compressible flows, Shock phenomena, Two-phase flows, Applications in aerospace engineering, turbomachinery and reciprocating engines, Industrial aerodynamics and wind engineering, and selected chemical engineering problems. Turbulence remains one of the key issues in tackling engineering flow problems. These problems are solved now and more by CFD analysis, the reliability of which depends strongly on the performance of the turbulence models employed. Successful simulation of turbulence requires the understanding of the complex physical phenomena involved and suitable models for describing the turbulent momentum, heat and mass transfer. For the understanding of turbulence phenomena, experiments are indispensable, but they are equally important for providing data for the development and testing of turbulence models and hence for software validation. As in other fields of Science, in the rapidly developing discipline of turbulence, swift progress can be achieved only by keeping up to date with recent advances all over the world and by exchanging ideas with colleagues active in related fields.

Analysis of Turbulent Flow with Computer Programs

This book is intended to serve as a reference text for advanced scientists and research engineers to solve a variety of fluid flow problems using computational fluid dynamics (CFD). Each chapter arises from a collection of research papers and discussions contributed by the practiced experts in the field of fluid mechanics. This material has encompassed a wide range of CFD applications concerning computational scheme, turbulence modeling and its simulation, multiphase flow modeling, unsteady-flow computation, and industrial applications of CFD.

Handbook of Computational Fluid Mechanics

Noise pollution around airports, trains, and industries increasingly attracts environmental concern and regulation. Designers and researchers have intensified the use of large-eddy simulation (LES) for noise reduced industrial design and acoustical research. This 2007 book, written by 30 experts, presents the theoretical background of acoustics and of LES, followed by details about numerical methods, e.g. discretization schemes, boundary conditions, coupling aspects. Industrially relevant, hybrid RANS/LES techniques for acoustic source predictions are presented in detail. Many applications are featured ranging from simple geometries for mixing layers and jet flows to complex wing and car geometries. Selected applications include scientific investigations at industrial and university research institutions.
Professor of aerodynamics at SUPAERO and director of DMAE at ONERA, the author calls on both his academic and industrial experience when presenting this work. The field of CFD is strongly represented by the following corporate companies: Boeing; Airbus; Thales; United Technologies and General Electric, government bodies and academic institutions also have a strong interest in this exciting field. Each chapter has also been specifically constructed to constitute as an advanced textbook for PhD candidates working in the field of CFD, making this book essential reading for researchers, practitioners in industry and MSc and MEng students. * A broad overview of the development and application of Computational Fluid Dynamics (CFD), with real applications to industry * A Free CD-Rom which contains computer program’s suitable for solving non-linear equations which arise in modelling turbulent flows * Professor Cebeci has published over 200 technical papers and 14 books, a world authority in the field of CFD

An Introduction to Computational Fluid Dynamics

This textbook presents numerical solution techniques for incompressible turbulent flows that occur in a variety of scientific and engineering settings including aerodynamics of ground-based vehicles and low-speed aircraft, fluid flows in energy systems, atmospheric flows, and biological flows. This book encompasses fluid mechanics, partial differential equations, numerical methods, and turbulence models, and emphasizes the foundation on how the governing partial differential equations for incompressible flow can be solved numerically for an accurate and efficient manner. Extensive discussions on incompressible flow solvers and turbulence modeling are also offered. This text is an ideal instructional resource and reference for students, research scientists, and professional engineers interested in analyzing fluid flows using numerical simulations for fundamental research and industrial applications.

Analysis of Turbulent Flows with Computer Programs

In various branches of fluid mechanics, our understanding is inhibited by the presence of turbulence. Although many experimental and theoretical studies have significantly helped to increase our physical understanding, a comprehensive and predictive theory of turbulent flows has not yet been established. Therefore, the prediction of turbulent flow relies heavily on simulation strategies. The development of reliable methods for turbulent flow computation will have a significant impact on a variety of technological advancements. These range from aircraft and car design, to turbomachinery, combustors, and process engineering. Moreover, simulation approaches are important in materials such as prediction of biologically relevant flows, and also significantly contribute to the understanding of environmental processes including weather and climate forecasting. The material thing is compiled in this book presents a coherent account of contemporary computational approaches for turbulent flows. It aims to provide the reader with information about the current state of the art as well as to stimulate directions for future research and development. The book puts particular emphasis on computational methods for incompressible and compressible turbulent flows as well as on methods for analyzing and quantifying numerical errors in turbulent flow computations. In addition, it presents turbulence modelling approaches in the context of large eddy simulation, and unfolds the challenges in the field of simulations for multiphase flows and computational fluid dynamics (CFD) of engineering flows in complex geometries. Apart from reviewing main research developments, new material is also included in many of the chapters.

Statistical Theory and Modeling for Turbulent Flows

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.

Turbulent Flow Computation

Publisher Description

Development and Application of Rotation and Curvature Correction to Wray-Agarwal Turbulence Model

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

Computational Fluid Dynamics Analysis of the Heat Transfer in Turbulent Flow Over a 2-dimensional Double Cavity

This book covers a wide variety of topics related to advancements in different stages of mass transfer modelling processes. Its purpose is to create a platform for the exchange of recent observations, experiences, and achievements. It is recommended for those in the chemical, biotechnological, pharmaceutical, and nanotechnology industries as well as for students of natural sciences, technical, environmental and employees in companies which manufacture machines for the above-mentioned industries. This work can also be a useful source for researchers and engineers dealing with mass transfer and related issues.

Applied Computational Fluid Dynamics and Turbulence Modeling

This book presents the fundamentals of computational fluid dynamics for the novice. It provides a thorough yet user-friendly introduction to the governing equations and boundary conditions of viscous fluid flows and its modelling.

Computational Fluid Dynamics Analysis of Turbulent Flows

Analysis of Turbulent Boundary Layers focuses on turbulent flows meeting the requirements for the boundary-layer or thin-shear-layer approximations. Its approach is devising relatively fundamental, and often subtle, empirical engineering correlations, which are then introduced into various forms of describing equations for final solution. After introducing the topic on turbulence, the book examines the conservation equations for compressible turbulent flows, boundary-layer equations, and general behavior of turbulent boundary layers. The latter chapters describe the CS method for calculating two-dimensional and axisymmetric laminar and turbulent boundary layers. This book will be useful to readers who have advanced knowledge in fluid mechanics, especially to engineers who study the important problems of design.
Turbulent Flows

This title provides the fundamental bases for developing turbulence models on rational grounds. The main different methods of approach are considered, ranging from statistical modelling to numerical simulations of turbulence. Each of these various methods has its own specific performances and limitations, which appear to be complementary rather than competitive. After a discussion of the basic concepts, mathematical tools and methods for closure, the book considers second order closure models. Emphasis is placed upon this approach because it embodies potential for clarifying numerous problems in turbulent shear flows. Simpler, generally older models are then presented as simplified versions of the more general second order models. The influence of extra physical parameters is also considered. Finally, the book concludes by examining large Edy numerical simulations methods. Given the book’s comprehensive coverage, those involved in the theoretical or practical study of turbulence problems in fluids will find this a useful and informative read.

Statistical Theory and Modeling for Turbulent Flows

It is a truism that turbulence is an unsolved problem, whether in scientific, engineering or geophysical terms. It is strange that this remains largely the case even though we now know how to solve directly, with the help of sufficiently large and powerful computers, equations that govern turbulent fluid flows, not with our numerical approximations but with the size of the computational task and the complexity of the solutions we can earn, which match the complexity of real turbulence precisely in so far as the computations mimic the real flows. The fact that we can now solve some turbulence in this limited sense is nevertheless an enormous step towards the goal of full understanding. Direct and large-eddy simulations are these numerical solutions of turbulence. They reproduce with remarkable fidelity the statistical, structural and dynamical properties of physical turbulent and transitional flows, though since the simulations are necessarily time-dependent and three-dimensional they demand the most advanced computer resources at our disposal. The numerical techniques vary from accurate spectral methods and high-order finite differences to simple finite-volume algorithms derived on the principle of embedding (‘first-principles’). Indeed, CFD analysis has reached a fairly mature level of development, including the commercial level. However, CFD experts are aware that even though commercial codes are capable of simulating local fluid and thermal physics, great care must be taken in their application to avoid errors caused by such things as inappropriate grid meshing, low-order discretisation schemes, lack of iterative convergence and inaccurate time-stepping. Just as important is the choice of a turbulence model for turbulent flow simulation. Turbulence models model the effects of turbulent transport of mass, momentum and energy, but are not necessarily applicable for wide ranges of flow types. Therefore, there is a well-recognised need to establish practices and procedures for the proper application of CFD to simulate flow physics accurately and establish the level of uncertainty of such computations. The present document represents contributions of CFD experts on what the basic practices, procedures and guidelines should be to aid CFD analysts to obtain accurate estimates of the flow and energy transport as applied to nuclear reactor safety. However, it is expected that these practices and procedures will require updating from time to time as research and development affect them or replace them with better procedures. The practices and procedures are categorised into five groups. These are: 1. Code Verification 2. Code and Calculation Documentation 3. Reduction of Numerical Errors 4. Quantification of Numerical Uncertainty (Calculation Verification) 5. Calculation Validation. These five categories have been identified from procedures currently required of CFD simulations such as those required for publication of a paper in the ASME Journal of Fluids Engineering and from the literature such as Roache [1998]. Code verification refers to the demonstration that the equations of fluid and energy transport have been correctly written in the CFD code, that the implementation simply casts these equations in terms of their discretizations, etc., and boundary and initial conditions used to pose the fluid flow problem are fully described in available documentation. Reduction of numerical error refers to practices and procedures to lower numerical errors to negligible or very low levels as is reasonably possible (such as avoiding use of first-order discretisations). The quantification of numerical uncertainty is also known as calculation uncertainty verification. This means that estimates are made of numerical error to allow the characterization of the numerical.

CFD Analysis of Laminar Oscillating Flows

Provides unique coverage of the prediction and experimentation necessary for making predictions. * Covers computational fluid dynamics and its relationship to direct numerical simulation used throughout the industry. * Covers vortex methods developed to calculate and evaluate turbulent flows. * Includes chapters on the state-of-the-art applications of research such as control of turbulence.

Computational Fluid Dynamics Analysis of the Heat Transfer in Turbulent Flow Over a 3-dimensional Cavity

Computational Fluid Dynamics (CFD) is increasingly playing a significant role in the analysis and design of aircrafts, turbomachines, automobiles, and in many other industrial applications. In majority of the applications, the fluid flow is generally turbulent. The accurate prediction of turbulent flows to date remains a challenging problem in CFD. In almost all industrial applications, Reynolds-Averaged Navier-Stokes (RANS) equations in conjunction with a turbulence model are employed for simulation and prediction of turbulent flows. Currently the one-equation (namely the Spalart-Allmaras (SA) and Wray-Agarwal (WA) and two-equation (namely the k-[epsilon] and Shear Stress Transport k-[omega]) turbulence models remain the most widely used models in industry. However, improvements and new developments are needed to improve the accuracy of the turbulence models for wall bounded flows with separation in the presence of adverse pressure gradients, and for flows with rotation and curvature (RC) such as those encountered in turbomachinery, centrifugal pumps and the rotating machinery in other industrial devices. The goal of this research is to enable the
eddy–viscosity type turbulence models to accurately account for the rotation and curvature effects. To date, there have been two approaches for inclusion of RC effects in turbulence models, which can be categorized as the “Modified Coefficients Approach” which parameterizes the model coefficients such that the growth rate of turbulent kinetic energy is either suppressed or enhanced depending upon the effect of system rotation and streamline curvature on the pressure gradient in the flow and the “Bifurcation Approach” which parameterizes the eddy–viscosity coefficient such that the equilibrium solution bifurcates from the main branch to decaying solution branches. In this research, the uncertainty quantification (UQ) analysis of the sensitivity of RC correction coefficients and the coefficients are modified based on the UQ analysis to improve the model’s behavior. Both these approaches are applied to the widely used turbulence models (SA, SST k-ω and WA) and they show some improvement in predictions of turbulent flow in all benchmark test cases considered, namely the flow in a 2D curved duct, flow in a 2D U-turn duct, fully developed turbulent flow in a 2D rotating channel, fully developed turbulent flow in a 2D rotating backward-facing step, flow in a rotating cavity, flow in a stationary and rotating serpentine channel, flow in a rotor-stator cavity and in a hydrocyclone as well as two wall-unbounded turbulent flow cases. All the simulations are conducted using the commercial software ANSYS Fluent and the open source CFD software OpenFOAM. The success of this research should enhance the ability of the RANS modeling for more accurate prediction of complex turbulent flows with rotation and curvature effects. In addition to the RANS modeling of RC effects, a new DES model incorporating the WA2017m-RC turbulence model (referred to as the WA2017m-RC-DES model) is developed and validated against experimental and DNS data. Further improvements are obtained with the DES model in some test cases.

Numerical Simulation

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 to focus exclusively on broad coverage of CFD applications 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 overview of the application areas where CFD can be used. 20% new content

Turbulence

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.

Direct and Large–Eddy Simulation I

Most natural and industrial flows are turbulent. The atmosphere and oceans, automobile and aircraft engines, all provide examples of this ubiquitous phenomenon. In recent years, turbulence has become a very lively area of scientific research and application, and this work offers a grounding in the subject of turbulence, developing both the physical insight and the mathematical framework needed to express the theory. Providing a solid foundation in the key topics in turbulence, this valuable reference resource enables the reader to become a knowledgeable developer of predictive tools. This central and broad ranging topic would be of interest to graduate students in a broad range of subjects, including aeronautical and mechanical engineering, applied mathematics and the physical sciences. The accompanying solutions manual to the text also makes this a valuable teaching tool for lecturers and for practising engineers and scientists in computational and experimental and experimental fluid dynamics.

CFD Modeling of Turbulent Flows Around the Main Injector Assembly Using Porosity

This is the first book specifically designed to offer the student a smooth transitionary course between elementary fluid dynamics (which gives only last-minute attention to turbulence) and the professional literature on turbulent flow, where an advanced viewpoint is assumed. The subject of turbulence, the most forbidding in fluid dynamics, has usually proved treacherous to the beginner, caught in the whirls and eddies of its nonlinearities and statistical imponderables. This is the first book specifically designed to offer the student a smooth transitionary course between elementary fluid dynamics (which gives only last-minute attention to turbulence) and the professional literature on turbulent flow, where an advanced viewpoint is assumed. Moreover, the text has been developed for students, engineers, and scientists with different technical backgrounds and interests. Almost all flows, natural and man-made, are turbulent. Thus the subject is the concern of geophysical and environmental scientists (in dealing with atmospheric jet streams, ocean currents, and the flow of rivers, for example), of astrophysicists (in studying the photospheres of the sun and stars or mapping gaseous nebulae), and of engineers (in calculating pipe flows, jets, or wakes). Many such examples are discussed in the book. The student is warned against the mathematical development on the one hand and the morass of experimental detail and empirical data on the other. As a result of following its midstream course, the text gives the student a physical understanding of the subject and deepens his intuitive insight into those problems that cannot now be rigorously solved. In particular, dimensional analysis is used extensively in dealing with those problems whose exact solution is mathematically elusive. Dimensional reasoning, scale arguments, and similarity rules are introduced at the beginning and are applied throughout. A discussion of Reynolds stress and the kinetic theory of gases provides the contrast needed to put mixing-length theory into proper perspective: the authors present a thorough comparison between the mixing-length models and dimensional analysis of shear flows. This is followed by an extensive treatment of vorticity dynamics, including vortex stretching and vorticity budgets. Two chapters are devoted to boundary-free shear flows and well-bounded turbulent shear flows. The examples presented include wakes, jets, shear layers, thermal plumes, atmospheric boundary layers, pipe and channel flow, and boundary layers in pressure gradients. The spatial structure of turbulent flow has been the subject of analysis in the book up to this point, at which a compact but thorough introduction to statistical methods is given. This provides the reader to understand the stochastic and spectral
structure of turbulence. The remainder of the book consists of applications of the statistical approach to the study of turbulent transport (including diffusion and mixing) and turbulent spectra.

Mass Transfer

This is an advanced textbook on the subject of turbulence, and is suitable for engineers, physical scientists and applied mathematicians. The aim of the book is to bridge the gap between the elementary accounts of turbulence found in undergraduate texts, and the more rigorous monographs on the subject. Throughout, the book combines the maximum of physical insight with the minimum of mathematical detail. Chapters 1 to 5 may be appropriate as background material for an advanced undergraduate or introductory postgraduate course on turbulence, while chapters 6 to 10 may be suitable as background material for an advanced postgraduate course on turbulence, or as a reference source for professional researchers. This second edition covers a decade of advancement in the field, streamlining the original content while updating the sections where the subject has moved on. The expanded content includes large-scale dynamics, stratified and rotating turbulence, the increased power of direct numerical simulation, two-dimensional turbulence, Magnetohydrodynamics, and turbulence in the core of the Earth.

Computational Fluid Dynamics: Principles and Applications

Large-Eddy Simulation for Acoustics

This paper describes a numerical simulations of oscillating flow in a constricted duct and compares the results with experimental and theoretical data. The numerical simulations were performed using the computational fluid dynamics (CFD) code CFX4.2. The numerical model simulates an experimental oscillating flow facility that was designed to test the properties and characteristics of oscillating flow in tapered ducts, also known as jet pumps. Jet pumps are useful devices in thermoacoustic machinery because they produce a secondary pressure that can counteract an unwanted effect called streaming, and significantly enhance engine efficiency. The simulations revealed that CFX could accurately model velocity, shear stress and pressure variations in laminar oscillating flow. The numerical results were compared to experimental data and theoretical predictions with varying success. The least accurate numerical results were obtained when laminar flow approached transition to turbulent flow.

Unsteady Turbulent Flow Modelling and Applications

Turbulent Flow

Analysis of Turbulent Boundary Layers

Most natural and industrial flows are turbulent. The atmosphere and oceans, automobile and aircraft engines, all provide examples of this ubiquitous phenomenon. In recent years, turbulence has become a very lively area of scientific research and application, and this work offers a grounding in the subject of turbulence, developing both the physical insight and the mathematical framework needed to express the theory. Providing a solid foundation in the key topics in turbulence, this valuable reference resource enables the reader to become a knowledgeable developer of predictive tools. This central and broad ranging topic would be of interest to graduate students in a broad range of subjects, including aeronautical and mechanical engineering, applied mathematics and the physical sciences. The accompanying solutions manual to the text also makes this a valuable teaching tool for lecturers and for practising engineers and scientists in computational and experimental fluid dynamics.

Engineering Applications of Computational Fluid Dynamics

Processes and Procedures for Application of CFD to Nuclear Reactor Safety Analysis

This paper is concerned with the implementation of a computational model of turbulent flow in a section of the lower plenum of Very High Temperature Reactor (VHTR). The proposed model has been encoded in a state-of-the-art CFD code, NPHASE. The results of NPHASE predictions have been compared against the experimental data collected using a scaled model of a sub-region in the lower plenum of a modular prismatic gas-cooled reactor. It has been shown that the NPHASE-based model is capable of predicting a three-dimensional velocity field in a complex geometrical configuration of VHTR lower plenum. The current and future validations of computational predictions are necessary for design and analysis of new reactor concepts, as well as for safety analysis and licensing calculations.

Computational Fluid Dynamics

The master thesis of David Roos Launchbury deals with the implementation and validation of a numerical solver for incompressible large eddy simulation (LES) with heat transfer in OpenFOAM. Academic and industrial cases, ranging from flow between parallel plates to film cooling, are investigated utilising existing and newly-implemented turbulence models. Simulations using no turbulence models, i.e. under-resolved DNS (UDNS) simulations, are performed for comparison. Very good results are obtained in all cases with variations among the individual models, with the UDNS simulations performing surprisingly well. The study shows that the developed software is able to simulate complex cases reliably and accurately.

CFD Analysis of Power-law Fluid in a Partially Blocked Eccentric Annulus Under Turbulent Flow Conditions

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.

Analysis of Turbulent Flows with Computer Programs
This handbook covers computational fluid dynamics from fundamentals to applications. This text provides a well documented critical survey of numerical methods for fluid 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.

Computational Fluid Dynamics

Computational Methods for Heat and Mass Transfer

Hot gas turbulent flow distribution around the main injector assembly of the Space Shuttle Main Engine (SSME) and LOX flow distribution through the LOX posts have a great effect on the combustion phenomena inside the main combustion chamber. In order to design a CFD model to be an effective engineering analysis tool with good computational turn-around time (especially for 3-D flow problems) and still maintain good accuracy in describing the flow features, the concept of porosity was employed to describe the effects of blockage and drag force due to the presence of the LOX posts in the turbulent flow field around the main injector assembly of the SSME. Two-dimensional numerical studies were conducted to identify the drag coefficients of the flows, both through tube banks and round the shielded posts, over a wide range of Reynolds numbers. Empirical, analytical expressions of the drag coefficients as a function of local flow Reynolds number were then deduced. The porosity model was applied to the turbulent flow around the main injector assembly of the SSME, and analyses were performed. The 3-D CFD analysis was divided into three parts: LOX dome, hot gas injector assembly, and hydrogen cavity. The numerical results indicate that the mixture ratio at the downstream of injector face was close to stoichiometric around baffle elements. Cheng, Gary C. and Chen, Y. S. and Farmer, Richard C. Unspecified Center

CFD Analysis of Turbulent Flow Phenomena in the Lower Plenum of a Prismatic Gas-Cooled Reactor

Modelling and Computation of Turbulent Flows has been written by one of the most prolific authors in the field of CFD. Professor of aerodynamics at SUPAERO and director of DMAE at ONERA, the author calls on both his academic and industrial experience when presenting this work. The field of CFD is strongly represented by the following corporate companies; Boeing; Airbus; Thales; United Technologies and General Electric, government bodies and academic institutions also have a strong interest in this exciting field. Each chapter has also been specifically constructed to constitute as an advanced textbook for PhD candidates working in the field of CFD, making this book essential reading for researchers, practitioners in industry and MSc and MEng students. * A broad overview of the development and application of Computational Fluid Dynamics (CFD), with real applications to industry * A Free CD-Rom which contains computer program’s suitable for solving non-linear equations which arise in modeling turbulent flows * Professor Cebeci has published over 200 technical papers and 14 books, a world authority in the field of CFD

Engineering Turbulence Modelling and Experiments

A First Course in Turbulence

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

Copyright code: c769c1cb1548bd1f66fdef4d36e52d51b