Experimental And Cfd Analysis Of A Perforated Inner Pipe
7f4d5b0b406334cef1559e493f29d938

This book contains selected and peer-reviewed papers presented at the International Conference on Efficient Solar Power Generation and Energy Harvesting (ESPGEH 2019). The primary focus of the book is on latest advances and scientific developments in the field of solar energy. The book covers various topics such as solar photovoltaics, solar energy harvesting, smart materials for energy applications, hybrid renewable energy plant, and on-grid and off-grid power plant. The book also discusses current techniques to produce energy-efficient solar cells, emerging materials and processes to develop cost-effective solar cells, and different issues in energy management. Given the scope of the contents, this book will be of interest for researchers, professionals as well as policy makers.

In the past Computational Fluid Dynamics (CFD) was confined to large organisations capable of developing and supporting their own codes. But recently there has been a rapid increase in the availability of reasonably priced commercial codes, and many more industrial organisations are now able to routinely use CFD. Advances of CFD in Fluid Machinery Design provide the perfect opportunity to find out what industry is doing and this book addresses how CFD is now being increasingly used in the design process, rather than as a post-design analysis tool. COMPLETE CONTENTS Trends in industrial use of CFD Challenges and methodologies in the design of axial flow fans for high-bypass-ratio, gas turbine engines using steady and unsteady CFD A three-dimensional inverse method based on pressure loading for the design of turbomachinery blades Application of CFD to the design and analysis of axial and centrifugal fans and compressors The design and performance of a transonic flow deswirling system – an application of current CFD design techniques tested against model and full-scale experiments Recent developments in unsteady flow modelling for turbomachinery aeroelasticity Computational investigation of flow in casing treatments for stall delay in axial flow fans Use of CFD for the three-dimensional hydrodynamic design of vertical diffuser pumps Recommendations to designers for CFD pump impeller and diffuser simulations Three dimensional CFD – a possibility to analyse piston pump flow dynamics CFD analysis of screw compressor performance Prediction of aerothermal phenomena in high-speed discstator systems Use of CFD in the design of a shaft seal for high-performance turbomachinery Users and potential users, of CFD for the design of fluid machinery, managers, designers, and researchers working in the field of ‘industrial flows’, will all find Advances of CFD in Fluid Machinery Design a valuable volume discussing state-of-the-art developments in CFD.

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.

This book comprises selected peer-reviewed proceedings of the International Conference on Applications of Fluid Dynamics (ICAFD 2018) organized by the School of Advanced Sciences, Vellore Institute of Technology, India, in association with the University of Botswana and the Society for Industrial and Applied Mathematics (SIAM), USA. With an aim to identify the existing challenges in the area of applied mathematics and mechanics, the book emphasizes the importance of establishing new methods and algorithms to address these challenges. The topics covered include diverse applications of fluid dynamics in aerospace dynamics and propulsion, atmospheric sciences, compressible flow, environmental fluid dynamics, control structures, viscoelasticity and mechanics of composites. Given the contents, the book is a useful resource for students, researchers as well as practitioners.

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 CFX 4.2. 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.
A comprehensive reference for engineers and researchers, Gas Turbine Heat Transfer and Cooling Technology, Second Edition has been completely revised and updated to reflect advances in the field made during the past ten years. The second edition retains the format that made the first edition so popular and adds new information mainly based on select

In this chapter, an analysis of convection heat transfer in an experimental heat exchanger using experimental data and numerical simulation data (by means computational fluid dynamics (CFD)) is presented. Work was realized in four stages. In the first stage, experimental data were obtained from a heat exchanger installed in Thermohydraulic Laboratory from CIICAp. Analytic calculus with experimental data was realized in the second stage to establish proper values in boundary and operation conditions for numerical simulation. The third stage includes numerical simulation using CFD of the heat exchanger domain with both working fluids (air-water). At the fourth stage, an analysis of the results was performed.

This book collects invited lectures and selected contributions presented at the Enzo Levi and XVIII Annual Meeting of the Fluid Dynamic Division of the Mexican Physical Society in 2012. It is intended for fourth-year undergraduate and graduate students, and for scientists in the fields of physics, engineering and chemistry with an interest in Fluid Dynamics from experimental, theoretical and computational points of view. The invited lectures are introductory in nature and avoid the use of complicated mathematics. The other selected contributions are also suitable for fourth-year undergraduate and graduate students. The Fluid Dynamics applications include oceanography, multiphase flows, convection, diffusion, heat transfer, rheology, granular materials, viscous flows, porous media flows and astrophysics. The material presented in the book includes recent advances in experimental and computational fluid dynamics and is well-suited to both teaching and research.

This paper compares results from a computational fluid dynamics (CFD) simulation of airflow and pollutant dispersion under mixed-convection conditions with experimental data obtained in our 7m x 9m x 11m high experimental facility. A tracer gas was continuously released from a 1 m² horizontal source 0.5 m above the floor. Path-integrated concentrations were measured along multiple short and long sampling paths in three horizontal planes. A steady state CFD analysis was used to model these experiments. The Reynolds Averaged Navier-Stokes (RANS) equations were solved for the flow and temperature field using the commercial CFD software, StarCD. CFD results were compared with the measured path-integrated
concentrations. Accuracy of CFD predictions was found to improve with inclusion of thermal effects, and further by using a low-Re turbulence model.

Turbulence is one of the key issues in tackling engineering flow problems. As powerful computers and accurate numerical methods are now available for solving the flow equations, and since engineering applications nearly always involve turbulence effects, the reliability of CFD analysis depends increasingly on the performance of the turbulence models. This series of symposia provides a forum for presenting and discussing new developments in the area of turbulence modelling and measurements, with particular emphasis on engineering-related problems. The papers in this set of proceedings were presented at the 5th International Symposium on Engineering Turbulence Modelling and Measurements in September 2002. They look at a variety of areas, including: Turbulence modelling; Direct and large-eddy simulations; Applications of turbulence models; Experimental studies; Transition; Turbulence control; Aerodynamic flow; Aero-acoustics; Turbomachinery flows; Heat transfer; Combustion systems; Two-phase flows. These papers are preceded by a section containing 6 invited papers covering various aspects of turbulence modelling and simulation as well as their practical application, combustion modelling and particle-image velocimetry.

This book offers timely insights into research on numerical and experimental fluid mechanics and aerodynamics, mainly for (but not limited to) aerospace applications. It reports on findings by members of the STAB (German Aerospace Aerodynamics Association) and DGLR (German Society for Aeronautics and Astronautics) and covers both nationally and EC-funded projects. Continuing on the tradition of the previous volumes, the book highlights innovative solutions, promoting translation from fundamental research to industrial applications. It addresses academics and professionals in the field of aeronautics, astronautics, ground transportation, and energy alike.

This book comprises select proceedings of the 46th National Conference on Fluid Mechanics and Fluid Power (FMFP 2019). The contents of this book focus on aerodynamics and flow control, computational fluid dynamics, fluid structure interaction, noise and aero-acoustics, unsteady and pulsating flows, vortex dynamics, nuclear thermal hydraulics, heat transfer in nanofluids, etc. This book serves as a useful reference beneficial to researchers, academicians and students interested in the broad field of mechanics.
Three examples of recent investigations combining CFD analysis and wind tunnel tests at the von Karman Institute for Fluid Dynamics (VKI) are presented to illustrate the integration process between experimental tests and numerical results. For each project, the requirements and the methodology were different but the final goal was to obtain an optimized aerodynamic design. The first project presented is dealing with a fluid-structure interaction problem. The second project concerns the aerodynamic design of a new Belgian polar base station for Antarctica and the last project concerns the design of an ultra-streamlined land vehicle for the next Solar World Challenge.

The Nuclear Materials Storage Facility (NMSF) at the Los Alamos National Laboratory is being renovated for long-term storage of canisters designed to hold heat-generating nuclear materials, such as powders, ingots, and other components. The continual heat generation within the canisters necessitates a reliable cooling scheme of sufficient magnitude which maintains the stored material temperatures within acceptable limits. The primary goal of this study was to develop both an experimental facility and a computational fluid dynamics (CFD) model of a subsection of the NMSF which could be used to observe general performance trends of a proposed passive cooling scheme and serve as a design tool for canister holding fixtures. Comparisons of numerical temperature and velocity predictions with empirical data indicate that the CFD model provides an accurate representation of the NMSF experimental facility. Minor modifications in the model geometry and boundary conditions are needed to enhance its accuracy, however, the various fluid and thermal models correctly capture the basic physics.

This document presents guidelines for assessing the credibility of modeling and simulation in computational fluid dynamics. The two main principles that are necessary for assessing credibility are verification and validation. Verification is the process of determining if a computational simulation accurately represents the conceptual model, but no claim is made of the relationship of the simulation to the real world. Validation is the process of determining if a computational simulation represents the real world. This document defines a number of key terms, discusses fundamental concepts, and specifies general procedures for conducting verification and validation of computational fluid dynamics simulations. The document's goal is to provide a foundation for the major issues and concepts in verification and validation. However, this document does not recommend standards in these areas because a number of important issues are not yet resolved. It is hoped that the guidelines will aid in the research, development, and use of computational fluid dynamics simulations by establishing common...
Read Free Experimental And Cfd Analysis Of A Perforated Inner Pipe

terminology and methodology for verification and validation. The terminology and methodology should also be useful in other engineering and science disciplines.

A high efficiency Stirling Radioisotope Generator (SRG) is being developed for possible use in long-duration space science missions. NASA’s advanced technology goals for next generation Stirling convertors include increasing the Carnot efficiency and percent of Carnot efficiency. To help achieve these goals, a multi-dimensional Computational Fluid Dynamics (CFD) code is being developed to numerically model unsteady fluid flow and heat transfer phenomena of the oscillating working gas inside Stirling convertors. In the absence of transient pressure drop data for the zero mean oscillating multi-dimensional flows present in the Technology Demonstration Convertors on test at NASA Glenn Research Center, unidirectional flow pressure drop test data is used to compare against 2D and 3D computational solutions. This study focuses on tracking pressure drop and mass flow rate data for unidirectional flow though a Stirling heater head using a commercial CFD code (CFD-ACE). The commercial CFD code uses a porous-media model which is dependent on permeability and the inertial coefficient present in the linear and nonlinear terms of the Darcy-Forchheimer equation. Permeability and inertial coefficient were calculated from unidirectional flow test data. CFD simulations of the unidirectional flow test were validated using the porous-media model input parameters which increased simulation accuracy by 14 percent on average.

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 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.

Industrial tray air-dryers are increasingly used for the drying of agricultural products. The main drawback of these dryers is the non-uniform velocity distribution in the drying zone resulting in a non-uniform drying of the product. Computational Fluid Dynamics (CFD) software was implemented to predict and decrease the non-uniform velocity distribution of various dryer configurations. Tunnel dryers in commercial use were used to obtain experimental data. The CFD results were correlated with the test data. Trolley and tray tunnel dryers provide a relatively simple, low capital intensive and versatile method for drying a wide range of products. Artificial drying has the advantage of controlled drying conditions compared to traditional sun drying. The main focus of every tunnel design should be the improvement of the quality of the product in terms of colour, texture and aroma. Increasing the evaporation rate without increasing the energy required to do so, should always be done in-line with this main objective. Many studies focus on the mango structure and food dehydration principles that influence the uniform drying product with the assumption that the airflow over the produce is uniform. Few have been conducted on the air movement inside industrial dryers. CFD analysis predicts the airflow without influencing the airflow pattern compared to the measuring equipment inside test dryers. The experimental data were obtained from an empty dryer without a flow diverter. This was compared to dryer with the flow diverter included and compared to a dryer with the trolleys, trays and mango slices included. The test results showed that turbulence created by this configuration, still played a major role in the non-uniform velocity distribution along the drying zone of the tunnel. The inclusion of a flow diverter did however dampen the swirl effect of the main fan. Measuring the velocity distribution was practically difficult with the handheld devices used, which influenced the accuracy of the measurements taken. This justified the CFD analysis in order to better visualise and predict the airflow pattern inside the dryer. The total average speed CFD results of the sections in the drying zone (without mangoes and trolleys) of the dryer without a flow diverter was 11.2% higher compared to the test results. It was 14% higher for the dryer with the flow diverter included. The dryer with the mangoes, trays, trolleys and flow diverter showed a large difference where the total average speed of the CFD analysis was 49% higher compared to the test results. The main reason for the difference of the CFD analysis compared to the measured results are the factors that influenced the uncertainty of the experimental set up. The CFD analysis showed that the coefficient of variance (CV) of the dryer with the flow diverter (mangoes and trolleys included) was 3% lower...
various dryer configurations were analysed using the CFD software to investigate what the best combination of flow diverter, vanes and blanking-off plates would be. A dryer configuration where flow diverters (up- and downstream of the main fan) above the false ceiling and inside the drying zone was analysed. A 16% decrease in terms of the CV value was obtained compared to the dryer with just the flow diverter downstream of main fan above the false ceiling. There was however a large region of swirl upstream of the main above the false ceiling resulting in a larger loss of heated air through the outlet fan before it reached the drying zone. The cost of manufacturing a simple vane and flow diverter for an existing dryer is 4% of the initial building costs (excluding the initial cost of the trolleys). The overall drying uniformity of this dryer is improved according to the CFD analysis by 7%. A cost analysis (taking into account the 15 year life cycle of a dryer) in terms of the energy requirement to evaporate water from the drying zone, showed that the dryer with the flow diverter was 6% less expensive to run on a yearly basis. Labour costs will be lower due to man-hours saved in terms of sorting out the wet slices from the dried product. Resources (dryers and trolleys) that would have been used for re-drying the wet produce, could now be implemented to increase the production rate of the plant. Copyright.

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.
The objective of this project is to study the fundamental physical phenomena in the reactor cavity cooling system (RCCS) of very high-temperature reactors (VHTRs). One of the primary design objectives is to assure that RCCS acts as an ultimate heat sink capable of maintaining thermal integrity of the fuel, vessel, and equipment within the reactor cavity for the entire spectrum of postulated accident scenarios. Since construction of full-scale experimental test facilities to study these phenomena is impractical, it is logical to expect that computational fluid dynamics (CFD) simulations will play a key role in the RCCS design process. An important question then arises: To what extent are conventional CFD codes able to accurately capture the most important flow phenomena, and how can they be modified to improve their quantitative predictions? Researchers are working to tackle this problem in two ways. First, in the experimental phase, the research team plans to design and construct an innovative platform that will provide a standard test setting for validating CFD codes proposed for the RCCS design. This capability will significantly advance the state of knowledge in both liquid-cooled and gas-cooled (e.g., sodium fast reactor) reactor technology. This work will also extend flow measurements to micro-scale levels not obtainable in large-scale test facilities, thereby revealing previously undetectable phenomena that will complement the existing infrastructure. Second, in the computational phase of this work, numerical simulation of the flow and temperature profiles will be performed using advanced turbulence models to simulate the complex conditions of flows in critical zones of the cavity. These models will be validated and verified so that they can be implemented into commercially available CFD codes. Ultimately, the results of these validation studies can then be used to enable a more accurate design and safety evaluation of systems in actual nuclear power applications (both during normal operation and accident scenarios).

The Prismatic Modular Reactor (PMR) is one of the major Very High Temperature Reactor (VHTR) concepts, which consists of hexagonal prismatic fuel blocks and reflector blocks made of nuclear grade graphite. However, the shape of the graphite blocks could be easily changed by neutron damage during the reactor operation and the shape change can create gaps between the blocks inducing the bypass flow. In the VHTR core, two types of gaps, a vertical gap and a horizontal gap which are called bypass gap and cross gap, respectively, can be formed. The cross gap complicates the flow field in the reactor core by connecting the coolant channel to the bypass gap and it could lead to a loss of effective coolant flow in the fuel.
blocks. Thus, a cross flow experimental facility was constructed to investigate the cross flow phenomena in the core of the VHTR and a series of experiments were carried out under varying flow rates and gap sizes. The results of the experiments were compared with CFD (Computational Fluid Dynamics) analysis results in order to verify its prediction capability for the cross flow phenomena. Fairly good agreement was seen between experimental results and CFD predictions and the local characteristics of the cross flow was discussed in detail. Based on the calculation results, pressure loss coefficient across the cross gap was evaluated, which is necessary for the thermo-fluid analysis of the VHTR core using a lumped parameter code.

Aerodynamic analysis using computational fluid dynamics (CFD) is most fruitful when it is combined with a thorough program of wind tunnel testing. The understanding of aerodynamic phenomena is enhanced by the synergistic use of both analysis methods. A technique is described for an integrated approach to determining the forces and moments acting on a wind tunnel model by using a combination of experimentally measured pressures and CFD predictions. The CFD code used was FLO57 (an Euler solver) and the wind tunnel model was a heavily instrumented delta wing with 62.5 deg of leading-edge sweep. A thorough comparison of the CFD results and the experimental data is presented for surface pressure distributions and longitudinal forces and moments. The experimental pressures were also integrated over the surface of the model and the resulting forces and moments are compared to the CFD and wind tunnel results. The accurate determination of various drag increments via the combined use of the CFD and experimental pressures is presented in detail. Melton, John E. and Robertson, David D. and Moyer, Seth A. Ames Research Center

Inhaltsangabe:Abstract: Wind energy is an increasingly important source of renewable, clean energy. In spite of this, only the methods and the materials of construction have improved over time, while the basic working principle of the wind turbine is still the same as it was centuries ago. In this thesis we have increased the power of a wind turbine by a factor of 4 in a fluid dynamic simulation, using a very simple external shroud system. We have also extended the theory of wind turbines (limit of Betz), to include this new kind of device and show why past attempts to augment the power of a wind turbine by means of shroud systems have failed. A detailed analysis of the device and its functioning principle is presented in this thesis - optimization studies need to be done in the future. Inhaltsverzeichnis:Table of Contents: Abstract I Index II List of Figures IV List of Symbols VI Introduction 1.1 Theory of Wind Turbines 5 1.1.1 Introduction 5 1.2 The Betz Law 6 1.3 Aerodynamics of the rotor 13 1.4 Rotor Power Characteristics 18 1.5 Number of Rotor Blades 20 1.6 Horizontal Axis Wind Turbines (HAWT) 22 1.7 Shrouded / Ducted Wind
Computational fluid dynamics (CFD) is recognized as a powerful engineering tool. That is, CFD has advanced over the years to the point where it can now give us deep insight into the analysis of very complex processes. There is a danger, though, that an engineer can place too much confidence in a simulation. If a user is not careful, it is easy to believe that if you plug in the numbers, the answer comes out, and you are done. This assumption can lead to significant errors. As we discovered in the course of a study on behalf of the Department of Energy's Savannah River Site in South Carolina, CFD models fail to capture some of the large variations inherent in complex processes. These variations, or scatter, in experimental data emerge from physical tests and are inadequately captured or expressed by calculated mean values for a process. This anomaly between experiment and theory can lead to serious errors in engineering analysis and design unless a correction factor, or safety factor, is experimentally validated. For this study, blending times for the mixing of salt solutions in large storage tanks were the process of concern under investigation. This study focused on the blending processes needed to mix salt solutions to ensure homogeneity within waste tanks, where homogeneity is required to control radioactivity levels during subsequent processing. Two of the requirements for this task were to determine the minimum number of submerged, centrifugal pumps required to blend the salt mixtures in a full-scale tank in half a day or less, and to recommend reasonable blending times to achieve nearly homogeneous salt mixtures. A full-scale, low-flow pump with a total discharge flow rate of 500 to 800 gpm was recommended with two opposing 2.27-inch diameter nozzles. To make this recommendation, both experimental and CFD modeling were performed. Lab researchers found that, although CFD provided good estimates of an average blending time, experimental blending times varied significantly from the average.
This study aims to investigate the heave and pitch responses of systematic series of high-speed displacement hull forms attached with Hull Vane®. AMECRC systematic series hull forms are developed based on the HSDHF series originally developed at the MARIN, The Netherlands. These hull forms are obtained by systematically varying the length to breadth ratio, breadth to draft ratio and block coefficient. Hull Vane® used for this analysis has a profile similar to NACA 4412 and principal dimensions such as chord length, span and flap angle have been kept constant. For the purpose of this study, five models from the 14 models of the AMECRC systematic series were chosen. Motion responses are calculated for head seas in an irregular sea way, transiting at Froude numbers ranging from 0.25 to 1.0 with an increment of 0.25. In addition to the above analysis, an attempt has been made to predict the bare hull motion response of models 4, 8, 13 using MAXSURF motions and Star CCM+. Heave and Pitch RAOS were obtained and these results are compared with the experimental results obtained by Sahoo and Doctors (2004) through towing tank experiments. The CFD package Star CCM+ has been validated with these results and it is then used to predict the motion response of the hull forms with Hull Vane®. Heaving and pitching RAOS of both bare hull form and hull form with Hull Vane® are presented as graphs and then the results have been compared. A discussion of the results obtained and the effect of Hull Vane® on the motion characteristics of the high-speed displacement hull forms were presented at the end. The results obtained from this analysis shows significant reduction in the motion response of hull-forms with Hull Vane®. It is expected that this study would help provide a platform for obtaining in-depth knowledge into the study of Hull Vane® as a stern appendage. It is hoped that the conclusion would be beneficial to anyone whose interest lies in advancing Hull Vane® research.

These proceedings contain the papers presented at the 4th International Symposium on Engineering Turbulence Modelling and Measurements held at Ajaccio, Corsica, France from 24-26 May 1999. It follows three previous conferences on the topic of engineering turbulence modelling and measurements. The purpose of this series of symposia is to provide a forum for presenting and discussing new developments in the area of turbulence modelling and measurements, with particular emphasis on engineering-related problems. Turbulence is still one of the key issues in tackling engineering flow problems. As powerful computers and accurate numerical methods are now available for solving the flow equations, and since engineering applications nearly always involve turbulence effects, the reliability of CFD analysis depends more and more on the performance of the turbulence models. 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 CFD software validation.

The analysis of the turbulent flows in nuclear fuel bundles is a very interesting task to optimize the efficiency of modern nuclear power plants. The proposed study utilizes Computational Fluid Dynamics (CFD) to characterize the flow pattern generated in a fuel bundle with Spacer Grids (SG) and Mixing Vanes (MV). CFD calculations were performed using different turbulence models for steady state simulations. Large Eddy Simulations (LES) scheme was applied to time dependent cases. The simulations were compared with the experimental data measured at Texas A & M University fuel bundle experimental facility. Also, another objective is to develop some new coarse mesh approaches for modeling MV to include these structures in the prospective of quarter of core simulations; MV and SG are usually modeled with porous media, since the computational power required to solve the full geometry is still unacceptable. The new contribution of the study is the definition and implementation of a Momentum Sources Forcing approach that allows a detailed definition of MV and SG for coarse mesh calculations. The proposed method was investigated using different turbulence models and different numerical schemes. Also, LES calculations allowed the study of Fluid Structure Interaction (FSI), that generates vibration problems and failure of nuclear fuel pins. A spectral analysis of the forces acting on the fuel pins walls was developed. In conclusion, a comprehensive study of fuel bundle problem was proposed with benchmark of the computational techniques to the experimental data.

This paper documents testing methods, statistical data analysis, and a comparison of experimental results to CFD models for blending of fluids, which were blended using a single pump designed with dual opposing nozzles in an eight foot diameter tank. Overall, this research presents new findings in the field of mixing research. Specifically, blending processes were clearly shown to have random, chaotic effects, where possible causal factors such as turbulence, pump fluctuations, and eddies required future evaluation. CFD models were shown to provide reasonable estimates for the average blending times, but large variations -- or scatter -- occurred for blending times during similar tests. Using this experimental blending time data, the chaotic nature of blending was demonstrated and the variability of blending times with respect to average blending times were shown to increase with system complexity. Prior to this research, the variation in blending times caused discrepancies between CFD models and experiments. This research addressed this discrepancy, and determined statistical correction factors that can be applied to CFD models, and thereby quantified techniques to permit the application of CFD models
to complex systems, such as blending. These blending time correction factors for CFD models are comparable to safety factors used in structural design, and compensate variability that cannot be theoretically calculated. To determine these correction factors, research was performed to investigate blending, using a pump with dual opposing jets which re-circulate fluids in the tank to promote blending when fluids are added to the tank. In all, eighty-five tests were performed both in a tank without internal obstructions and a tank with vertical obstructions similar to a tube bank in a heat exchanger. These obstructions provided scale models of vertical cooling coils below the liquid surface for a full scale, liquid radioactive waste storage tank. Also, different jet diameters and different horizontal orientations of the jets were investigated with respect to blending. Two types of blending tests were performed. The first set of eighty-one tests blended small quantities of tracer fluids into solution. Data from these tests were statistically evaluated to determine blending times for the addition of tracer solution to tanks, and blending times were successfully compared to Computational Fluid Dynamics (CFD) models. The second set of four tests blended bulk quantities of solutions of different density and viscosity. For example, in one test a quarter tank of water was added to a three quarters of a tank of a more viscous salt solution. In this case, the blending process was noted to significantly change due to stratification of fluids, and blending times increased substantially. However, CFD models for stratification and the variability of blending times for different density fluids was not pursued, and further research is recommended in the area of blending bulk quantities of fluids. All in all, testing showed that CFD models can be effectively applied if statistically validated through experimental testing, but in the absence of experimental validation CFD model can be extremely misleading as a basis for design and operation decisions.

Current Trends and Future Developments on (Bio-) Membranes: Techniques of Computational Fluid Dynamic (CFD) for Development of Membrane Technology provides updates on new progress in membrane processes due to various challenges and how many industrial companies and academic centers are carrying out these processes. Chapters help readers understand techniques of computational fluid dynamic (CFD) for the development of membrane technology, including an introduction to the technologies, their applications, and the advantages/disadvantages of CFD modeling of various membrane processes. In addition, the book compares these modeling methods with other traditional separation systems and covers fouling and concentration polarization problems. The book is a key reference for R&D managers interested in the development of membrane technologies as well as academic researchers and postgraduate students working in the wider areas of strategic treatments, separation and purification processes. Includes developments of membrane technologies in different applications by using CFD tools Describes CFD methods for evaluation