

## Experimental And Cfd Analysis Of A Perforated Inner Pipe

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.

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.

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.

Computational Fluid Dynamics (CFD) of Chemical Processes

Analysis of Non-Newtonian Behavior of Crude Oil

Water Measurement Manual

Experimental Investigation and Modeling of Friction Stir Processing Using 3D CFD Analysis

How to Plan a CFD Analysis

**div="" style="" 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. ^**

**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.**

**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.

**Computational Fluid Dynamic (CFD) Analysis of a Six Cylinder Diesel Engine Cooling System with Experimental Correlations**

**Experimental and CFD Study of the Dynamics of Wheat Starch/pyrolysis Gases Hybrid Mixtures in the 20-L Sphere Standard Test**

**Theoretical, Computational, and Experimental Solutions to Thermo-Fluid Systems**

**Proceedings of FMFP 2019**

**Analysis of Heat Transfer in an Experimental Heat Exchanger Using Numerical Simulation**

*Experimental and CFD Analysis of the Flow in the Wake of a Vertical Axis Wind Turbine*  
*Experimental and CFD Analysis of Advanced Convective Cooling Systems*

*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.*

*This book contains selected and peer-reviewed papers presented at the International Conference on Efficient Solar Power Generation and Energy Harvesting (ESPEH 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.*

*Experimental and CFD Analysis of Advanced Convective Cooling Systems*

*Proceedings of the 7th International Conference on Advances in Energy Research*

*A Guide to Methods and Standards for the Measurement of Water Flow*

*A Theoretical, Experimental and CFD Analysis of Regenerative Flow Compressors and Pumps for Microturbine and Automotive Fuel Applications*

*CFD Analysis and Experimental Investigation of Proton Exchange Membrane Fuel Cells*

**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.**

**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...**

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

**AIAA Guide for the Verification and Validation of Computational Fluid Dynamics Simulations**

**CFD Analysis of Experimental Wing and Winglet for FalconLAUNCH 8 and the ExFIT Program**

**CFD Analysis of Nuclear Fuel Bundles and Spacer Grids for PWR Reactors**

**Selected Papers from CSNDD 2016**

**The 3-D Viscous Flow CFD Analysis of the Propeller Effect on an Advanced Ducted Propeller Subsonic Inlet**

*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 through 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.*

*Validation of Computational Fluid Dynamics (CFD) codes is an essential element of the code development process. Typically, CFD code validation is accomplished through comparison of computed results to previously published experimental data that were obtained for some other purpose, unrelated to code validation. As a result, it is a near certainty that not all of the information required by the code, particularly the boundary conditions, will be available. The common approach is therefore unsatisfactory, and a different method is required. This paper describes a methodology developed specifically for experimental validation of CFD codes. The methodology requires teamwork and cooperation between code developers and experimentalists throughout the validation process, and takes advantage of certain synergisms between CFD and experiment. The methodology employs a novel uncertainty analysis technique which helps to define the experimental plan for code validation wind tunnel experiments, and to distinguish between and quantify various types of experimental error. The methodology is demonstrated with an example of surface pressure measurements over a model of varying geometrical complexity in laminar, hypersonic, near perfect gas, 3-dimensional flow.*

*This book presents selected papers from the 7th International Conference on Advances in Energy Research (ICAER 2019), providing a comprehensive coverage encompassing all fields and aspects of energy in terms of generation, storage, and distribution. Themes such as optimization of energy systems, energy efficiency, economics, management, and policy, and the interlinkages between energy and environment are included. The contents of this book will be of use to researchers and policy makers alike. .*

*Experimental Study and Numerical Modeling Using Computational Fluid Dynamics (CFD) Technique*

*Experimental Investigation and CFD Analysis on Cross Flow in the Core of PMR200*

*Integration of CFD and Experimental Results at VKI in Low-Speed Aerodynamic Design*

*Advances of CFD in Fluid Machinery Design*

*Flow Around Wing Mirrors*

Computational fluid dynamics, CFD, has become an indispensable tool for many engineers. This book gives an introduction to CFD simulations of turbulence, mixing, reaction, combustion and multiphase flows. The emphasis on understanding the physics of these flows helps the engineer to select appropriate models to obtain reliable simulations. Besides presenting the equations involved, the basics and limitations of the models are explained and discussed. The book combined with tutorials, project and power-point lecture notes (all available for download) forms a complete course. The reader is given hands-on experience of drawing, meshing and simulation. The tutorials cover flow and reactions inside a porous catalyst, combustion in turbulent non-premixed flow, and multiphase simulation of evaporation spray respectively. The project deals with design of an industrial-scale selective catalytic reduction process and allows the reader to explore various design improvements and apply best practice guidelines in the CFD simulations.

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.

Inhaltsangabe:Abstract: Wind energy is an increasingly import 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: AbstractI IndexII List of FiguresIV List of SymbolsVI

Introduction1  
 1.Theory of Wind Turbines5  
 1.1Introduction5  
 1.2The Betz Law6  
 1.3Aerodynamics of the rotor13  
 1.4Rotor Power Characteristics18  
 1.5Number of Rotor Blades20  
 1.6Horizontal Axis Wind Turbines (HAWT)22  
 1.7Shrouded / Ducted Wind Turbines28  
 1.7.1Ducted Rotor29  
 1.7.2Turbine with a Diffuser Duct29  
 2.Methodology33  
 2.1Introduction33  
 2.2Computational Domain34  
 2.3Computational Code41  
 2.3.1Conservation Equations42  
 2.3.2K-Epsilon Turbulence Model43  
 2.3.3Discretization of the Conservation Equations45  
 2.4MFR - Multiple Frame of Reference45  
 2.5Parallel Processing46  
 2.6Simulations47  
 3.Results48  
 3.1Introduction48  
 3.2Conventional Turbine49  
 3.2.1Velocity Field49  
 3.2.2Static Pressure Field52  
 3.2.3Total Pressure Field53  
 3.2.4Power of the Conventional Turbine55  
 3.2.5Energy and Momentum Transfer57  
 3.3Shrouded Turbine59  
 3.3.1Velocity Field59  
 3.3.2Static Pressure Field62  
 3.3.3Total Pressure Field63  
 3.3.4Power of the Shrouded Turbine65  
 3.3.5Energy and Momentum Transfer66  
 3.3.6The Betz Limit68  
 3.3.7Cross Check Analysis with Traditional Shrouded Turbines69  
 Conclusions72  
 Bibliography73  
 Acknowledgments  
 Computational Fluid Dynamics Analysis of Flow Patterns in a Thermal Tray Dryer

## Heterogeneous Photocatalysis

### CFD and Experimental Analysis of an R141b Ejector Used in a Jet Refrigerator

#### Selected Proceedings of ICAFD 2018

#### Fluid Mechanics and Fluid Power

This book presents select proceedings of the International Conference on Innovations in Thermo-Fluid Engineering and Sciences (ICITFES 2020). It covers topics in theoretical and experimental fluid dynamics, numerical methods in heat transfer and fluid mechanics, different modes of heat transfer, multiphase flow, fluid machinery, fluid power, refrigeration and air conditioning, and cryogenics. The book will be helpful to the researchers, scientists, and professionals working in the field of fluid mechanics and machinery, and thermal engineering.

"This study had the objective of studying and analyzing the dynamic behavior of a wheat starch/pyrolysis gases hybrid mixture at different operating conditions of the 20-L Sphere Standard Test and different explosion regimes through an experimental approach and a Lagrangian-Eulerian computational formulation. The content of this study was divided into three stages. In the first stage, the pyrolysis step of organic particles in the sphere was studied along with the combustion of the pyrolysis gases. For these purposes, a modified version of the Godbert-Greenwald furnace and chromatography techniques were employed to determine the particular composition of the combustible gases emitted from the particles. Several one-phase explosion tests and simulations were carried out in to determine the most severe explosion that can be achieved separately by the particles and the pyrolysis gases. These results provided a comparative basis for the correct assessment of the influence of an additional phase in the explosive parameters measured in the 20-L Sphere. Furthermore, this stage provided a thorough analysis of the reacting gas system in terms of the main kinetic characteristics, the differential production/consumption of the species, the total heat released during the process, and the evolution of the flame front. The second stage evaluated the influence of two operating parameters on the dispersion and explosion steps of a wheat starch/pyrolysis gases hybrid mixture. The two parameters were the Ignition Delay Time and the nozzle geometry. In this stage, the interactions between the motion of the phases, the chemical reactions, and the turbulence levels of the gaseous phase were studied through CFD modeling to provide detailed explanations of the results obtained. In the latter stage, three different approaches were employed to analyze the behavior of wheat starch/pyrolysis gases explosions at different concentrations of the phases."--Tomado del Formato de Documento de Grado.

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).

#### A CFD Analysis and Validation Through Comparison with Experimental Results

#### CFD Analysis of Laminar Oscillating Flows

#### Advances in Solar Power Generation and Energy Harvesting

#### An Integrated CFD/experimental Analysis of Aerodynamic Forces and Moments

#### Recent Advances

This book presents contributions on the most active lines of recent advanced research in the field of nonlinear mechanics and physics selected from the 4th International Conference on Structural Nonlinear Dynamics and Diagnosis. It includes fifteen chapters by outstanding scientists, covering various aspects of applications, including road tanker dynamics and stability, simulation of abrasive wear, energy harvesting, modeling and analysis of flexoelectric nanoactuator, periodic Fermi-Pasta-Ulam problems, nonlinear stability in Hamiltonian systems, nonlinear dynamics of rotating composites, nonlinear vibrations of a shallow arch, extreme pulse dynamics in mode-locked lasers, localized structures in a photonic crystal fiber resonator, nonlinear stochastic dynamics, linearization of nonlinear resonances, treatment of a linear delay differential equation, and fractional nonlinear damping. It appeals to a wide range of experts in the field of structural nonlinear dynamics and offers researchers and engineers an introduction to the challenges posed by nonlinearities

in the development of these topics

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.

The series Topics in Current Chemistry Collections presents critical reviews from the journal Topics in Current Chemistry organized in topical volumes. The scope of coverage is all areas of chemical science including the interfaces with related disciplines such as biology, medicine and materials science. The goal of each thematic volume is to give the non-specialist reader, whether in academia or industry, a comprehensive insight into an area where new research is emerging which is of interest to a larger scientific audience. Each review within the volume critically surveys one aspect of that topic and places it within the context of the volume as a whole. The most significant developments of the last 5 to 10 years are presented using selected examples to illustrate the principles discussed. The coverage is not intended to be an exhaustive summary of the field or include large quantities of data, but should rather be conceptual, concentrating on the methodological thinking that will allow the non-specialist reader to understand the information presented. Contributions also offer an outlook on potential future developments in the field. The chapter "Mechanochemical Forces as a Synthetic Tool for Zero and One-Dimensional Titanium Oxide-Based Nano-photocatalysts" is available open access under a CC BY 4.0 License via [link.springer.com](http://link.springer.com).

Select Proceedings of ICITFES 2020

Advances in Fluid Dynamics

Select Proceedings of ESPGEH 2019

CFD Analysis of the Characteristics of a Shrouded Turbine

Tesi Di Dottorato

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 nonuniform 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 compared to the dryer without one. 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.

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.

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 through 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. Wilson, Scott D. and Dyson, Rodger W. and Tew, Roy C. and Demko, Rikako Glenn Research Center NASA/TM-2006-214246, AIAA Paper 2005-5539, E-15491

Experimental and CFD Analysis of the Flow in the Wake of a Vertical Axis Wind Turbine

CFD Analysis and Experimental Investigation Associated with the Design of the Los Alamos Nuclear Materials Storage Facility

Experimental Methodology for Computational Fluid Dynamics Code Validation

Recent Trends in Applied Nonlinear Mechanics and Physics