CFD Techniques and Energy Applications

Zied Driss · Brahim Necib
Hao-Chun Zhang
Editors

CFD Techniques and Energy Applications

Springer

Editors
Zied Driss
Department of Mechanical Engineering
National School of Engineers of Sfax
Sfax
Tunisia

Hao-Chun Zhang
School of Energy Science and Engineering
Harbin Institute of Technology
Harbin
China

Brahim Necib
Faculty of Sciences and Technology
University of Constantine 1
Constantine
Algeria

ISBN 978-3-319-70949-9 ISBN 978-3-319-70950-5 (eBook)
https://doi.org/10.1007/978-3-319-70950-5

Library of Congress Control Number: 2017963004

Printed on acid-free paper

This Springer imprint is published by Springer Nature
The registered company is Springer International Publishing AG
The registered company address is: Gewerbestrasse 11, 6330 Cham, Switzerland

Preface

This book focuses on CFD (Computational Fluid Dynamics) techniques and the recent developments and research works in energy applications. It is also devoted to the publication of basic and applied studies broadly related to this area. The chapters present the development of numerical methods, computational techniques, and case studies in the energy applications. Also, they offer the fundamental knowledge for using CFD in energy applications through new technical approaches. Besides it describes the CFD process steps and provides benefits and issues for using CFD analysis in understanding the flow complicated phenomena and its use in the design process. The best practices for reducing errors and uncertainties in the CFD analysis are further described. The Book is expected not only to reveal the recent advances and future research trends of CFD Techniques but also to provide the reader with valuable information about energy applications.

The chapters present the development of numerical methods, computational techniques, and energy application case studies. Such studies and development approaches aim to provide the readers, engineers and Ph.D. students with the fundamentals of CFD prior to embarking on any real simulation project. Additionally, engineers supporting or being supported by CFD analysts can take advantage from the information of the Book's different chapters.

In the first chapter, a numerical analysis on the performance of a solar chimney power plant using steady state Navier-Stokes and energy equations in cylindrical coordinate system was presented. The fluid flow inside the chimney is assumed to be turbulent and simulated with the k–ε turbulent model, using FLUENT software package. The computed results were in good agreement with the experimental measurements of the Spanish Manzanares power plant. The numerical model was then coupled with a mathematical one for a geothermal heat exchanger to investigate the option of coupling solar and geothermal sources for a continuous day and night operation. The results show the benefits of the hybrid solar-geothermal plant compared to the single solar chimney plant for day and night periods.

The second chapter presents the behavior of the air flow characteristics inside the Solar Chimney Power Plant (SCPP). A two-dimensional (2D) steady model was carried out using the commercial computational fluid dynamics (CFD) code

Ansys-Fluent 17.0. In this chapter, five turbulence models were tested to present the air flow characteristics distribution such as magnitude velocity, temperature, pressure and turbulence. The above work showed that the turbulence model types have a direct effect on the numerical results. Its computational results were compared to the experimental data found by Kasaeian et al. (2014) to choose the adequate model.

In the third chapter, the behavior of a negatively buoyant jet in laminar conditions that results from the injection of lighter fluids downwards into a large container of homogeneous fluid of denser density was studied numerically using Open Foam with the finite volume method. The fluid characteristics effect on the evolution of the pure water injected in a tank full of salt water was investigated particularly the molecular diffusivity that affects the mixing layer, the density relative difference between two liquids and the salt water kinematic viscosity that has an important effect on the transient phase as well as the subsequent steady state in terms of stationary penetration depth and jet profile.

The fourth chapter aimed to perform a numerical simulation of the liquid sloshing using the "Fluent" software. The simulation of the two phase application was achieved using the Volume Of Fluid (VOF). The container was subjected to a sinusoidal excitation. To impose the external excitation, a user defined function was developed and interpreted in "Fluent". Four numerical simulations were developed with different turbulence models; standard k–ε, RNG k–ε, Realizable k–ε and standard k–ω. The fluid flow characteristics for the different simulation cases were presented and discussed. The numerical results were compared with the experimental data. The comparison results show a good agreement with the turbulence model standard k–ε.

This fifth chapter presents a numerical model in order to capture the flow fields within a vanned volute under steady conditions. Numerical simulations were conducted using the CFX 17.0 package to solve Navier–Stokes equations by means of a finite volume discretization method. The good agreement between the experimental and numerical results of the turbine performance confirms the validation of the numerical model. Then, many computed flow discharge parameters, such as the averaged volute exit flow angle, were plotted to understand the behavior of the volute under different turbine expansion ratios. Furthermore, several loss coefficients distribution and entropy contours were plotted to characterize the occurring losses. In addition, pressure distributions, velocity and turbulence parameters as well as streamlines were numerically obtained to analyze the flow behavior within the turbine volute.

The sixth chapter presents some CFD simulation results of the hydrodynamic structure around a modified anchor system. Using the CFD code Ansys-FLUENT 17.0, the finite volume method was used to solve the Navier-Stokes equations. This study was carried out using the standard k-ε turbulence model. The comparison between the numerical and the experimental results found in the literature shows a good agreement.

The seventh chapter focuses on the numerical study of forced convection, for a thermo dependent Newtonian fluid in an eccentric horizontal annular duct. The inner and outer cylinders were heated with a constant heat flux. The governing equations were solved numerically by a finite difference method with implicit scheme. The dynamic profile was assumed to be fully developed while the temperature profile was assumed uniform at the entrance. The aim of this work was to present the eccentricity effects on the dynamic and thermal fields along the duct. The thermo dependency effect of the fluid was also examined, and some interesting results regarding the reduction of the dynamic blocking phenomenon of the flow in the narrow part of the duct for large eccentricities were presented. These results reduce the strictness of precautions for neglecting the axial diffusion when making computations in such geometries.

The eighth chapter investigates the effect of the incidence angle on the aerodynamic characteristics of the flow around a Savonius wind rotor. Six configurations with different incidence angles $\theta = 0°$, $\theta = 30°$, $\theta = 60°$, $\theta = 90°$, $\theta = 120°$ and $\theta = 150°$ were studied. To this end, a numerical simulation was developed using the Computational Fluid Dynamic (CFD) code "Fluent". The considered numerical model is based on the resolution of the Navier-Stokes equations together with the k-ε turbulence model. These equations were solved by a finite volume discretization method. The results confirm that the variation of the incidence angle has an effect on the local characteristics. The numerical results were compared to those obtained by previous findings showing a good agreement and confirming the numerical method efficiency.

The ninth chapter predicts numerically the flow effects of two coaxial jets with different swirl numbers on the characteristics of the turbulent diffusion flame. The study focused on the rotation influence of the secondary flow; that is to say two configurations were processed and compared: co-swirl and counter swirl. Obviously, the latter showed higher shear than the first. The calculation results were validated by actual measurement of the same configuration for two cases: co and counter swirl for reactant with combustion. The calculation results focus on the characteristics of the average flow and its turbulence for the two cases cited above. The obtained results confirm the swirl effects to stabilize the flame.

Sfax, Tunisia — Zied Driss
Constantine, Algeria — Brahim Necib
Harbin, China — Hao-Chun Zhang

Acknowledgements

First and foremost, I would like to thank Dr. Nabil Khélifi, Springer Editor who invited me to edit this new book after awarding the conference on CFD techniques and Thermo-Mechanics Applications, which was held at the National School of Engineers of Sfax (University of Sfax, Tunisia) in April 2016. All the ideas have developed further with my co-editors and many reviewers; especially in the second edition of the International Conferences on Mechanics and Energy (ICME'2016) which was held in Hammamet (Tunisia) in December 2016 and the third edition ICME'2017, held in Sousse (Tunisia) in December 2017.

I would like to thank all the authors who submitted chapters at our requests. Especially, I wish to express my gratitude to all the reviewers who participated to this book, provided support, talked things over, read, wrote, offered comments and allowed us to quote their remarks.

Many colleagues have generously provided comments and material from their past and current research. Particularly, I thank my co-editors Prof. Brahim Necib from the University of Mentouri Constantine (Algeria) and Prof. Hao-Chun Zhang from the Harbin Institute of Technology (China). Without them, this book would never find its way to so many reserchers, engineers and Ph.D. students.

I would like to express my gratitude to all those who provided support and assisted in the editing and proofreading. Particularly, I thank Prof. Abdelmajid Dammak for the Linguistic improvements of all chapters in the book. In addition, I would like to thank Reyhaneh Majidi, Shahid Mohammed, Kavitha Palanisamy and Suganya Manoharan from Springer for helping me in the process of selection, editing and design.

Last and not least: I beg forgiveness of all those who have been with me over the course of the years and whose names I have failed to mention.

Sfax, Tunisia
January 2018

Prof. Dr. Zied Driss

Contents