processes

MDPI

Editorial

Special Issue on "Advancement in Computational Fluid Mechanics and Optimization Methods"

Krzysztof Rogowski * and Piotr Lichota

Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology, 00-665 Warsaw, Poland; piotr.lichota@pw.edu.pl

* Correspondence: krogowski@meil.pw.edu.pl; Tel.: +48-506-193-919

Citation: Rogowski, K.; Lichota, P. Special Issue on "Advancement in Computational Fluid Mechanics and Optimization Methods". *Processes* **2022**, *10*, 1100. https://doi.org/10.3390/pr10061100

Received: 30 May 2022
Accepted: 31 May 2022
Published: 1 June 2022

In recent years, CFD methods have become a universal engineering tool for modeling many classes of flows. New turbulence models, as well as algorithms for solving governing equations, are still being developed. Various types of rotors, turbines, and pumps are classic engineering objects dedicated to CFD methods. This collection of articles included in the Special Issue (SI): "Advancement in Computational Fluid Mechanics and Optimization Methods" shows the much wider use of these methods.

CFD approaches are also used for much more advanced physical problems. Peng et al. [1] developed a partially premixed combustion model based on the flamelet generated manifold model to investigate the propagation of methane-air combustion flame. These authors simulated the transient pressure field along with flame propagation employing the unsteady RANS (URANS) approach. This paper proved that when the methane concentration accumulated in the regenerative thermal oxidizer reached at least 7.5%, the flame caused by the explosion could have caused explosions in underground mines. The unsteady RANS approach, specifically the standard k-ϵ turbulence model with standard log-law wall functions.

Another interesting paper in this SI collection deals with the fluid mechanics within a pump annular seal [2]. Fluid reaction forces can significantly affect the vibration characteristics and the stability of the pump. This work investigated transient flow parameters using the dynamic mesh approach. These authors obtained satisfactory results using the known turbulence model—the realizable k-ε.

Square-section structures are common in the field of civil and marine engineering. Flow characteristics around two square cylinders in a tandem arrangement for different spacing ratios were studied by Wang et al. [3]. In this paper, Proper Orthogonal Decomposition and Dynamic Mode Decomposition methods were employed to separate the main modes of the flow field and the corresponding time coefficient. The main purpose of these authors was to use CFD methods to investigate modes with higher energy in the flow field.

The scientific papers presented in this SI include two important works on modeling the flow around an airfoil, often used to construct wind turbine rotors with a vertical axis of rotation [4,5]. The four-digit NACA series airfoil with a relative thickness of 18% has been used in aviation for many years. However, the flow velocities occurring under such conditions are much higher than those observed during airflow around the blades of even large rotors of wind turbines. Despite the development of this branch of wind energy (vertical-axis wind turbines), since 1933, for almost 75 years, no experimental tests have been carried out on the performance of this profile for low and medium Reynolds numbers. Modern results of experimental research and new turbulence models have made it possible to revise the current research results and allowed for a better understanding of the phenomena occurring in the boundary layer. These two publications used a relatively new approach: the gamma-Re-Theta four-equation turbulence model. This model captures some phenomena in the boundary layer based on correlation—the research results provided by this model posed two new research problems. One of them is the influence of the wingspan on the characteristics of the laminar separation bubble, and the other is the need to calibrate the turbulence model. The simulation results published in these two papers have practical applications.

CFD methods can be used not only as tools that are helpful in understanding the physical phenomena occurring during the flow. One of the important applications of these methods is the ability to optimize both the geometry of an investigated object as well as the flow process. Huang et al. [6] did this kind of optimization work and used CFD methods to investigate the cryogenic expander. These authors' model took into account the generator, shaft, runner, and guide vanes. The simulations were made using commercial ANSYS CFX software and the SST k-ω turbulence model. The result of this work are recommendations for expander design and the blade lean angles.

In this SI, there is one more paper dealing with optimization. Zamora [7] studied buoyancy-driven airflow that included two isothermal inner plates established in a vented cavity. Obtained by this author, the results can be applied to the optimization of electronic equipment cooling. The selected turbulence model was the standard two-transport equations k-ω model proposed by Wilcox.

Furthermore, a less common use of CFD approaches may be to analyze the flow field of a wheel loader engine compartment. This problem was considered by Yu et al. [8] using three classical turbulence models as well as two more advanced approaches: Detached Eddy Simulation (DES) and Large Eddy Simulation (LES). This work shows that although RANS models provide a reasonable estimate of velocity fields and flow structure, LES methods have a much better ability to predict the turbulence distribution. Therefore, they seem to be the future of CFD modeling. The DES hybrid model is a good alternative to the still computationally-costly LES approach because it combines the advantages of the LES model and RANS model for simulating three-dimensional unsteady turbulent flows.

Another very interesting issue is Faraday instability. This phenomenon occurs during the relative displacement of the interface separating two immiscible fluids in a closed container. Blanco et al. [9] compared different numerical schemes to solve the Navier-Stokes equations for modeling this phenomenon. These authors used commercial ANSYS-CFX software and three different codes based on Front Tracking and Volume of Fluid schemes.

This compilation of articles in this SI shows that the classic RANS models will remain in common use for a long time. This is due to significantly lower computational costs than LES models and the acceptable results of aerodynamic loads and flow parameters around the analyzed object. However, the further development of models that capture the transition phenomena in the boundary layer in the case of external flow issues is important. It seems, however, that LES-type methods should be developed in parallel to enable a deeper analysis of turbulence. A very interesting approach that has begun to develop in recent years is the Lattice Boltzmann method. Jamalabadi [10] has shown the great possibilities of this approach in his paper. Perhaps soon, this approach will become an alternative method to CFD.

Author Contributions: Conceptualization, K.R. and P.L.; writing—original draft preparation, K.R. and P.L.; writing—review and editing, K.R. and P.L.; supervision, K.R. and P.L.; project administration, K.R. and P.L.; funding acquisition, K.R. All authors have read and agreed to the published version of the manuscript.

Funding: This research was carried out with the support of the Interdisciplinary Centre for Mathematical and Computational Modelling (ICM) University of Warsaw under computational allocation no GB83-33. Research was funded by POB Energy of Warsaw University of Technology within the Excellence Initiative: Research University (IDUB) programme. Grant No. 1820/355/Z01/POB7/2021.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Peng, Z.; Zanganeh, J.; Moghtaderi, B. CFD Modeling of Flame Jump across Air Gap between Evasé and Capture Duct for Ventilation Air Methane Abatement. *Processes* **2021**, *9*, 2278. [CrossRef]
2. Li, F.; Cui, B.; Zhai, L. Research on Rotordynamic Characteristics of Pump Annular Seals Based on a New Transient CFD Method. *Processes* **2020**, *8*, 227. [CrossRef]
3. Wang, F.; Zheng, X.; Hao, J.; Bai, H. Numerical Analysis of the Flow around Two Square Cylinders in a Tandem Arrangement with Different Spacing Ratios Based on POD and DMD Methods. *Processes* **2020**, *8*, 903. [CrossRef]
4. Rogowski, K.; Królak, G.; Bangga, G. Numerical Study on the Aerodynamic Characteristics of the NACA 0018 Airfoil at Low Reynolds Number for Darrieus Wind Turbines Using the Transition SST Model. *Processes* **2021**, *9*, 477. [CrossRef]
5. Michna, J.; Rogowski, K. Numerical Study of the Effect of the Reynolds Number and the Turbulence Intensity on the Performance of the NACA 0018 Airfoil at the Low Reynolds Number Regime. *Processes* **2022**, *10*, 1004. [CrossRef]
6. Huang, N.; Li, Z.; Zhu, B. Cavitating Flow Suppression in the Draft Tube of a Cryogenic Turbine Expander through Runner Optimization. *Processes* **2020**, *8*, 270. [CrossRef]
7. Zamora, B. Thermally Optimum Spacing between Inner Plates in Natural Convection Flows in Cavities by Numerical Investigation. *Processes* **2020**, *8*, 554. [CrossRef]
8. Yu, C.; Xue, X.; Shi, K.; Shao, M.; Liu, Y. Comparative Study on CFD Turbulence Models for the Flow Field in Air Cooled Radiator. *Processes* **2020**, *8*, 1687. [CrossRef]
9. Blanco, A.; Oliva, R.; Machado, D.; Legendre, D. Comparison of Different Numerical Interface Capturing Methods for the Simulation of Faraday Waves. *Processes* **2021**, *9*, 948. [CrossRef]
10. Abdollahzadeh Jamalabadi, M.Y. Lattice Boltzmann Simulation of Ferrofluids Film Boiling. *Processes* **2020**, *8*, 881. [CrossRef]