energies

Editorial

Advances in Numerical Heat Transfer and Fluid Flow

Artur S. Bartosik

Department of Production Engineering, Faculty of Management and Computer Modelling, Kielce University of Technology, Al. Tysiaclecia P.P. 7, 25-314 Kielce, Poland; artur.bartosik@tu.kielce.pl

Citation: Bartosik, A.S. Advances in Numerical Heat Transfer and Fluid Flow. *Energies* **2024**, *17*, 2108. https://doi.org/10.3390/en17092108

Received: 21 April 2024
Accepted: 25 April 2024
Published: 28 April 2024

1. Introduction

Scientists continuously are looking for new methods that allow them to better understand the flow and heat transfer phenomena. They try to optimise the parameters of fluid flow and heat exchange in order to increase the efficiency of specific technological processes and enhance environmental sustainability. The aim of this article is to provide an overview of the latest results in experiments and modelling of heat exchange between laminar or turbulent flow and surroundings, which were submitted to the Special Issue "Numerical Heat Transfer and Fluid Flow 2023".

The starting points in the prediction of heat exchange between the moving fluid and the surroundings are conservation equations, like the continuity, Navier–Stokes, and energy equations. This set of equations can be solved numerically using direct or indirect methods (DNS). The direct method requires high computing power. Thanks to the theory of turbulence, the time-averaged Navier–Stokes equations, called Reynolds-averaged Navier–Stokes equations (RANS), can be solved by applying additional equations, which allow the components of turbulent stress tensors to be designated. Components of the turbulent stress tensor can be calculated using a direct or indirect method [1–9]. Again, the direct method requires high computer power. In the literature, an indirect approach is commonly used in engineering applications because it requires less computing power. In such a case, turbulence models are applied to calculate the components of the turbulent stress tensor. One can find that hybrid models are continuously developed; they use different approaches to the near-wall region, where a small scale of turbulence exists, and to the outer region, where a large scale of turbulence dominates. For instance, the turbulence model can be applied to the region close to the wall, while large eddy simulation (LES) can be applied to the outer region. If non-Newtonian flow is considered, additional equations are needed in order to include a proper relation between shear stress and shear rate.

Computational methods that allow us to decrease the computation time are still developing. As a result, we already have considerable resources of software packages which allow scientists and engineers to simulate the heat transfer between fluid and the surroundings in specific applications. Some of the software packages were applied in this Special Issue.

The articles gathered in the Special Issue are useful for researchers, engineers, and students who deal with fluid flow and heat transfer.

2. Review of New Advances

Measurements of fluid thermal dynamics are still required in order to increase the knowledge and understanding of flow phenomena and to build an experimental base for the validation of mathematical models. Holešová et al. [10] focused on the challenges associated with visualising air flow over a heating source in an open laboratory environment. The authors performed an experimental visualisation and numerical simulation of air flow and heat transfer between the heating source and the environment using natural convection. They applied particle image velocimetry to visualise air flow. The data from the measurements were used as input for numerical simulations performed using Ansys Fluent software 18. These results demonstrated the effectiveness of combining measurements and numerical simulation to develop an accurate 3D model of heat transfer in laminar air flow [10]. The authors demonstrated that their 3D model had well-identified vorticities and had the potential to be used to simulate thermal comfort in the laboratory environment. It can be further extended by using and adjusting parameters such as the output of the heater and the heating source temperature [10].

Gas turbines are commonly used in power generation. Better turbine cooling can lead to the design of a turbine with better efficiency. Ahn [11] reviewed 50 articles published over the past 20 years on the use of large eddy simulation (LES) for the internal cooling passage of gas turbines. The author focused mainly on the mid-chord ribbed channels, collected validated LES models, and presented the simulation conditions of the models, then formulated findings that are difficult to obtain experimentally and discussed the effects of rotation, buoyancy, and heat transfer on the rib. The author discussed LES studies related to the shape of the ribbed channels and the LES contribution in the prediction of internal cooling of gas turbines and those with ribbed channels [11].

Kim et al. [12] applied large eddy simulation to predict forced convection around wavy cylinders with different axes for Re = 3000. The authors considered four types of undulated cylinders: streamwise undulation, transverse undulation, in-phase undulation, and antiphase undulation. The authors found that the friction and Nusselt number for cylinders with transverse undulation and in-phase undulation were significantly influenced by the wavelength and wave amplitude, whereas cylinders with streamwise undulation and antiphase undulation showed a very weak influence. The authors showed that proper modification of the geometry can improve both heat transfer and aerodynamic performance [12].

For a fully developed turbulent channel flow in the Reynolds number range of friction of 550–5200, Usov et al. [13] recalibrated the differential Reynolds stress model of Jakirlić and Maduta [14] using direct numerical simulation data in order to use the model in the hybrid RANS/LES framework. In their hybrid method, the authors used the RANS turbulence model only for a thin, near-wall layer in which small-scale turbulence dominated, while for the outer part of the boundary layer, in which large-scale turbulence dominated, they used the LES model. The authors performed comprehensive simulations of mean stream-wise and transverse velocity profiles, distribution of friction coefficient, and Reynolds stress profiles. The authors concluded that the recalibrated hybrid model ensured more accurate profiles of mean velocity, inner Reynolds stress peak, and characteristic turbulence frequency compared to the base model of Jakirlić and Maduta [14], and the model error was reduced by 58% [13].

The efficiency of heat exchange inside a shell and a tube heat exchanger is one of the main challenges of CFD. Estupinan-Campos et al. [15] numerically analysed the influence of geometric variations in baffle angles and the tube profile on heat transfer. They considered the turbulent flow of hot water in the inner tube and cold water in the outer tube. The flow of hot and cold liquids was countercurrent. The 3D simulations for the flow of water inside a shell and a tube heat exchanger were performed using time-averaged mass, momentum, and energy equations, while the problem of closing the set of equations was solved by introducing the standard k-ε model. The results of numerical predictions using Ansys software were compared with the measurements by Bicer et al. [16], achieving an average error equal to 3.15%. The authors recommended parameters such as the height of the baffle walls and the inclination of the deflector, which are given the highest efficiency of heat exchange inside a shell and a tube heat exchanger [15].

Pillow plate heat exchangers provide an alternative to conventional shell and tube heat exchangers. The proper waviness of the channels promotes fluid mixing in the boundary layers, resulting in the intensity of the heat transfer. Heat transfer efficiency can be increased by applying secondary surface structuring. Afsahnoudeh et al. [17] performed numerical studies on the influence of various surface structures in a heat exchanger on the efficiency of heat transfer. The mathematical model consisted of the conservation form of the continuity, momentum, and energy equations, while the closure problem was solved by applying the standard k-ε turbulence model. The authors considered a symmetric flow. Using software OpenFOAM No. 8, numerical simulations were performed for turbulent single-phase flow. As a result of the simulations, the authors determined the pressure drop, the heat transfer coefficient, and the overall thermohydraulic efficiency. The numerical predictions were validated only for the pressure drop, and the relative error was below 5%. The authors confirmed that the application of secondary structure has an impact on the thermohydraulic characteristics in both the outer and inner channels and results in an increase in the efficiency of the pillow plate heat exchanger [17].

On the basis of experiments and simulations, Bartosik [18] analysed the efficiency of heat exchange between the vertical pipe and turbulent slurry flow. The author presented experimental data of four slurries, with a narrow range of solid particle diameters, i.e., two with glass spheres with diameters of 0.125 mm and 0.240 mm, respectively, and two with sand particles with diameters of 0.470 mm and 0.780 mm, respectively, in the range of volume concentration of solid particles from 10% to 40%. The author showed that the experimental data demonstrated a substantial influence of solid particle diameter on the suppression of turbulence, which was the highest for the slurry with particle diameters equal to 0.47 mm. The author developed a mathematical model for heat exchange between pipe and slurry, taking into account the RANS approach with the k-ε turbulence model, and a specially designed turbulence damping function (wall function) depending on the diameter and concentration of solid particles. Numerical simulations of heat exchange between the slurry and the vertical pipe demonstrated a semisinusoidal dependence of the Nusselt number on the average particle diameter. The maximum Nusselt number appeared for a particle diameter of 0.125 mm, while the minimum was 0.470 mm [18].

The main challenge of the modern economy is the reduction in greenhouse gas emissions and the transition toward a zero-carbon economy. The greenhouse gas emissions of air conditioning are one of the main contributors to environmental pollution. A reduction in gas emission can be achieved by applying the caloric effect. One can recognise the caloric effect depending on the applied external field. If the external field is generated due to mechanical stress, such as compression [19], tension [20], or torsion [21], it is called the elastocaloric effect [22]. This effect occurs in specific materials known as shape memory alloys [23,24]. However, if the applied field is the result of hydrostatic pressure, it is referred to as the barocaloric effect [25,26]. Cirillo et al. [27] conducted a study on the application of barocaloric solid-state technology to the cold food supply chain. The authors examined the energy efficiency of a barocaloric cooling system designed to function as a refrigeration machine for cold products. The authors used a heat transfer liquid consisting of 50% ethylene glycol and 50% water mixture, while the solid-state refrigerant was acetoxy silicone rubber. As a reference, they used a vapour compression refrigerator commonly used in industrial settings. This analysis was carried out for incompressible laminar flow using the Navier–Stokes equations and the energy equation in 2D form, and the set of equations was solved using the finite element method. The authors concluded that, for a liquid velocity greater than 0.06 m/s, the barocaloric system outperforms the vapour compression system in terms of cooling power [27].

The gain of energy from deep geothermal resources requires the determination of thermal conductivity and heat capacity; prediction of heat exchange between geothermal wells and transported liquid; sustained power output; and, in some situations, chemical reactions. Such a process is extremely complex due to the dynamics of spatiotemporal evolution, which involves changes in the temperature field, seepage field, and mechanical field. Yang et al. [28] performed simulations of the heat production capacity of an underground reservoir, taking into account time-dependent mass conservation, heat conduction, and fluid heat transfer equations. The authors analysed the sensitivity of parameters related to the thermal performance and mining life of the deep thermal reservoir to thermal output. The results of the simulations showed that the injection temperature and the spacing of the well have little effect on the thermal efficiency and useful life of the mine, while the permeability, number, and length of the fracture have a significant influence. The authors emphasised the theoretical and practical importance of such studies [28].

Photovoltaic modules (PVs) often use cooling mediums, mainly liquid or air. The cooling medium flows over the backside surface of the cells and decreases the temperature of the cells to increase the efficiency of energy production. The cooling temperature is not high, so the use of such heat is limited. In the literature, some researchers have conducted studies on spectral beam splitting technology for sunlight. This technology uses different wavelengths of light from the Sun separately. Part of wavelength of the light is allocated to photoelectric conversion, and the other to photothermal conversion [29,30]. Lu et al. [31] performed simulations of the flow characteristics using a spectral beam-splitting module. The authors applied Ansys software using time-averaged continuity, momentum, energy, and k-ε equations. The authors noted that, for certain parameters of the nanofluid flow, it was possible to obtain a uniform temperature distribution on the surface of the photovoltaic cell while obtaining considerable heat and improving the integrated use of the Sun's energy. The authors stated that their study could provide a reference to optimise nanofluid flow within a spectral beam-splitting module [31].

3. Conclusions

The articles contributing to the Special Issue "Numerical Heat Transfer and Fluid Flow 2023" demonstrate the progress in using CFD for the prediction of heat exchange between laminar, turbulent, single-phase, or two-phase flow and the surroundings. The results of the measurements and several CFD models are presented. The authors formulated a set of conservation equations for a variety of applications and solved them numerically, taking into account the convergence criteria and ensuring a mesh-independent solution. All simulations were performed using commercial software, and most of the mathematical models have been validated.

The reader of the Special Issue "Numerical Heat Transfer and Fluid Flow 2023" can find a description of various phenomena of heat and fluid flow, experimental data, and various approaches to solving engineering problems. The collected articles will allow one to contribute to a better understanding of some phenomena and the interpretation of computed and measured quantities.

Funding: This research received no external funding.

Acknowledgments: The author thanks the contributors of the Special Issue Numerical Heat Transfer and Fluid Flow 2023 for the valuable articles, and thanks for the invitation to act as a guest editor.

Conflicts of Interest: The author declares no conflicts of interest.

References

1. Reynolds, O. On the dynamical theory on incompressible viscous fluids and the determination of the criterion. *Philos. Trans. R. Soc. Lond.* **1895**, *186*, 123–164. [CrossRef]
2. Prandtl, L. Bemerkung uber den warmeubergang in rohr. *Phys. Z.* **1928**, *29*, 487–489.
3. Karman, T. Mechanische ähnlichkeit und turbulenz. *Nachr. Gesselsch. Wiss. Göttingen Math. Phys.* **1930**, *322*, 58–76.
4. Karman, T. Some aspects of the theory of turbulent motion. In Proceedings of the Fourth International Congress for Applied Mechanics, Cambridge, UK, 3–9 July 1934.
5. Kolmogorov, A.N. Equations of turbulent motion of an incompressible fluid. *Izviestia AN SSSR Ser. Fiz.* **1942**, *VI*, 56–58.
6. Van Driest, E.R. On turbulent flow near a wall. *J. Aeronaut. Sci.* **1956**, *23*, 1007–1011. [CrossRef]
7. Patankar, S.V.; Spalding, D.B. *Heat and Mass Transfer in Boundary Layers*; Morgan-Grampian: London, UK, 1967.
8. Spalding, D.B. *Turbulence Models for Heat Transfer*; Report HTS/78/2; Department of Mechanical Engineering, Imperial College London: London, UK, 1978.
9. Spalding, D.B. *Turbulence Models—A Lecture Course*; Report HTS/82/4; Department of Mechanical Engineering, Imperial College London: London, UK, 1983.
10. Holešová, N.; Lenhard, R.; Kaduchová, K.; Holubčík, M. Application of Particle Image Velocimetry and Computational Fluid Dynamics Methods for analysis of natural convection over a horizontal heating source. *Energies* **2023**, *16*, 4066. [CrossRef]
11. Ahn, J. Large Eddy Simulation of flow and heat transfer in a ribbed channel for the internal cooling passage of a gas turbine blade: A Review. *Energies* **2023**, *16*, 3656. [CrossRef]

12. Kim, M.-K.; Chang, C.-H.; Nam, S.-H.; Yoon, H.-S. Large Eddy Simulation of forced convection around wavy cylinders with different axes. *Energies* **2024**, *17*, 894. [CrossRef]
13. Usov, L.; Troshin, A.; Anisimov, K.; Sabelnikov, V. Calibration of a near-wall differential Reynolds Stress Model using the updated Direct Numerical Simulation data and its assessment. *Energies* **2023**, *16*, 6826. [CrossRef]
14. Jakirlić, S.; Maduta, R. Extending the bounds of 'steady' RANS closures: Toward an instability-sensitive Reynolds stress model. *Int. J. Heat Fluid Flow* **2015**, *51*, 175–194. [CrossRef]
15. Estupinan-Campos, J.; Quitiaquez, W.; Nieto-Londono, C.; Quitiaquez, P. Numerical simulation of the heat transfer inside a shell and tube heat exchanger considering different variations in the geometric parameters of the design. *Energies* **2024**, *17*, 691. [CrossRef]
16. Biçer, N.; Engin, T.; Yaşar, H.; Büyükkaya, E.; Aydın, A.; Topuz, A. Design optimization of a shell-and-tube heat exchanger with novel three-zonal baffle by using CFD and Taguchi method. *Int. J. Therm. Sci.* **2020**, *155*, 106417. [CrossRef]
17. Afsahnoudeh, R.; Wortmeier, A.; Holzmüller, M.; Gong, Y.; Homberg, W.; Kenig, E.Y. Thermo-hydraulic performance of pillowplate heat exchangers with secondary structuring: A numerical analysis. *Energies* **2023**, *16*, 7284. [CrossRef]
18. Bartosik, A.S. Effect of the solid particle diameter on frictional loss and heat exchange in a turbulent slurry flow: Experiments and predictions in a vertical pipe. *Energies* **2023**, *16*, 6451. [CrossRef]
19. Chen, J.; Zhang, K.; Kan, Q.; Yin, H.; Sun, Q. Ul-tra-high fatigue life of NiTi cylinders for compression-based elastocaloric cooling. *Appl. Phys. Lett.* **2019**, *115*, 093902. [CrossRef]
20. Tušek, J.; Engelbrecht, K.; Mikkelsen, L.P.; Pryds, N. Elastocaloric effect of Ni-Ti wire for application in a cooling device. *J. Appl. Phys.* **2015**, *117*, 124901. [CrossRef]
21. Wang, R.; Fang, S.; Xiao, Y.; Gao, E.; Jiang, N.; Li, Y.; Mou, L.; Shen, Y.; Zhao, W.; Li, S.; et al. Torsional refrigeration by twisted, coiled, and supercoiled fibers. *Science* **2019**, *366*, 216–221. [CrossRef]
22. Greibich, F.; Schwödiauer, R.; Mao, G.; Wirthl, D.; Drack, M.; Baumgartner, R.; Kogler, A.; Stadlbauer, J.; Bauer, S.; Arnold, N.; et al. Elastocaloric heat pump with specific cooling power of 20.9 Wg–1 exploiting snap-through instability and strain-induced crystallization. *Nat. Energy* **2021**, *6*, 260–267. [CrossRef]
23. Ahcin, Ž.; Kabirifar, P.; Porenta, L.; Brojan, M.; Tušek, J. Numerical modeling of shell-and-tube-like elastocaloric regenerator. *Energies* **2022**, *15*, 9253. [CrossRef]
24. Zhu, Y.; Tsuruta, R.; Gupta, R.; Nam, T. Feasibility investigation of attitude control with shape memory alloy actuator on a tethered wing. *Energies* **2023**, *16*, 5691. [CrossRef]
25. Cirillo, L.; Greco, A.; Masselli, C. Cooling through barocaloric effect: A review of the state of the art up to 2022. *Therm. Sci. Eng. Prog.* **2022**, *33*, 101380. [CrossRef]
26. Aprea, C.; Greco, A.; Maiorino, A.; Masselli, C. The use of barocaloric effect for energy saving in a domestic refrigerator with ethylene-glycol based nanofluids: A numerical analysis and a comparison with a vapor com-pression cooler. *Energy* **2020**, *190*, 116404. [CrossRef]
27. Cirillo, L.; Greco, A.; Masselli, C. The Application of Barocaloric Solid-State Cooling in the Cold Food Chain for Carbon Footprint Reduction. *Energies* **2023**, *16*, 6436. [CrossRef]
28. Yang, Y.; Fu, G.; Zhao, J.; Gu, L. Heat production capacity simulation and parameter sensitivity analysis in the process of thermal reservoir development. *Energies* **2023**, *16*, 7258. [CrossRef]
29. Hu, P.; Zhang, Q.; Liu, Y.; Sheng, C.; Cheng, X.; Chen, Z. Optical analysis of a hybrid solar concentrating Photovoltaic/Thermal (CPV/T) system with beam splitting technique. *Sci. China Technol. Sci.* **2013**, *56*, 1387–1394. [CrossRef]
30. Liu, M.; Du, M.; Long, G.; Wang, H.; Qin, W.; Zhang, D.; Ye, S.; Liu, S.; Shi, J.; Liang, Z.; et al. Iron/Quinone-based all-in-one solar rechargeable flow cell for highly efficient solar energy conversion and storage. *Nano Energy* **2020**, *76*, 104907. [CrossRef]
31. Lu, L.; Tian, R.; Han, X. Optimization of nanofluid flow and temperature uniformity in the spectral beam splitting module of PV/T system. *Energies* **2023**, *16*, 4666. [CrossRef]