energies

Editorial

Numerical Heat Transfer and Fluid Flow: New Advances

Artur S. Bartosik

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

1. Introduction

This Special Issue, titled 'Numerical Heat Transfer and Fluid Flow 2022', presents articles addressed to *Energies* and is a continuation of the 2021 edition [1]. The authors of this Special Issue describe solutions of scientific and industrial relevance in a specific field of heat transfer and fluid flow, including technical devices, nanofluids, industrial processes, dedicated perforations or mechanically deformed pipes, the transport of solid particles, etc. These articles also serve as catalysts for future directions and priorities in numerical heat transfer and fluid flow.

Numerical fluid mechanics is based on the governing equations of continuity, momentum, and energy and has been systematically developed since the second half of the twentieth century. Great interest in solving specific engineering problems requires mathematical and numerical modelling, simulations, and experiments on heat exchange and fluid flow for a variety of single- and multiphase flows and boundary conditions. The increase in the efficiency of computational methods and the availability of commercial packages makes it possible to solve complex engineering problems more accurately and faster. These also allow us to analyse the complex phenomena of the dynamic and thermal boundary layers [2].

Formulating an engineering problem of fluid flow and heat transfer requires the prior development of a physical model that includes important features of the phenomenon, such as thermophysical properties, flow geometry, boundary, and initial conditions. The next step is to develop a mathematical model. The mathematical model should start with the general form of the governing equations of continuity, momentum, and energy. As a result of the assumptions made in the physical model, a specific form of the mathematical model is formulated. The set of equations can be solved analytically, which is usually difficult and impractical, or numerically. If numerical methods are considered, direct numerical simulation (DNS), Reynolds-averaged Navier–Stokes equations (RANS), or large eddy simulation (LES) can be taken into account. The DNS method is time-consuming and expensive. The DNS method regards instantaneous parameters, which has a limited application for a large number of engineering problems as engineers are interested mainly in time-averaged parameters. Turbulence simulation using RANS or LES methods has been proven for a variety of engineering applications. Such methods are less time-consuming compared to the DNS method and, therefore, cost less; however, the set of equations requires closure. The problem of closure requires an additional equation or an additional set of equations, such as those proposed for turbulence models. The main requirements for turbulence models include universality, economy, extensionality, and reality [3,4]. The ability to simulate fluid flow and convective–diffusive heat transfer includes velocity-, pressure-, and temperature-dependent variables and remains one of the main challenges in CFD. Researchers are still looking for the best ways to influence the level of turbulence to reduce friction or increase the efficiency of mixing flowing components and heat transfer. We still try to follow nature and implement the best available engineering solutions, which is a great challenge in the development of CFD [5,6].

Considering the intensification of heat exchange, we recognise passive or active methods. Passive methods, such as increasing the area of heat transfer or inserting solid particles

Citation: Bartosik, A.S. Numerical Heat Transfer and Fluid Flow: New Advances. *Energies* **2023**, *16*, 5528. https://doi.org/10.3390/en16145528

Received: 10 July 2023
Revised: 14 July 2023
Accepted: 16 July 2023
Published: 21 July 2023

into a fluid, or increasing the intensity of turbulence by shaping an insert with a dedicated perforation or mechanically deformed pipe, have been studied for several years, and some of them have become commercial solutions [7–10]. Active methods, such as air injection, bubble or vortex generation, or proper pulsation, can lead to an increase in the heat transfer coefficient and, finally, can produce an increase in heat exchange [11–13]. Both methods contribute to changes in the dynamic and thermal boundary layers by increasing the intensity of turbulence, allowing for the better mixing of flowing components.

The articles collated in this Special Issue are useful for researchers, engineers, and students who are focused on fluid flow and heat transfer.

2. Review of New Advances

Efficient numerical methods and fast computers mean that even complex phenomena, including the 3D domain, complex boundary conditions, fluid incompressibility, and changeable thermophysical properties, have the possibility of being solved.

However, in some simplified phenomena, analytical solutions are still valuable and possible. Based on the Navier–Stokes equations, Urbanowicz et al. [14] reviewed analytical models to accelerate the laminar flow of an incompressible Newtonian fluid in a pipe. The authors reviewed the solutions, which included two main approaches, such as an imposed pressure gradient or an imposed flow rate. An extension of this theory to turbulent accelerated pipe flows was also discussed [14]. The authors concluded that the discussed solutions extend the theory of analytical solutions of simplified two-dimensional Navier–Stokes equations and can be used not only to study the behaviour of liquids during accelerated pipe flow, but also to test the accuracy of commercial CFD packages [14].

Nanoparticles suspended in a fluid, called nanofluids, exist broadly in engineering. Nanomaterials possess outstanding optical, electrical, and thermophysical properties and therefore can be found in electronics, medicine, biotechnology, and energy transport systems, of which superconductors are one of those that exploit electrical and thermophysical properties. Nanofluids have been known for many decades and are particularly useful for intensifying heat exchange in engines, heat exchangers, or cooling mechanical devices [15]. Cieslinski [16] reviewed the recent achievements in the numerical modelling of the forced convection of nanofluids in round and smooth tubes and divided them into single- and two-phase approaches. The author synthesised three main challenges for engineers who deal with nanofluids, which are as follows.

- Provide reliable thermophysical properties of nanoparticles, liquids, and nanofluids;
- Provide accurate and reliable methods to calculate the heat transfer coefficient and friction factor;
- Stabilise nanofluids as they tend to aggregate to a larger size, which can result in sedimentation and/or separation.

The author noted that the majority of mathematical models assume a steady heat flow density acting on a chosen pipe length, as well as steady wall temperature. The author recognised 30 validated mathematical models for single-phase flows and 23 for two-phase flows and collected empirical correlations for the Nusselt number for laminar and turbulent flows. Cieslinski found that some researchers, such as Lotfi et al. [17] and Mokmeli and Saffar-Avval [18], observed that the precision of a single-phase approach is similar to that of a two-phase approach, while other researchers, like Maïga [19] for instance, indicated that two-phase models more realistically reproduce the results of measurements. Some results of the simulations emphasised the enhancement of heat transfer caused by the selected solid particles, while others noted the marginal influence. This clearly indicates that the process of nanofluid flow is complex, depends on the boundary layer and solid particles, and is not yet well understood. Cieslinski noted that, in the case of turbulent flow, researchers solve the closure problem primarily by using the standard k-ε turbulence model [16].

Foam is a dispersion of particles in a continuous medium in which the particles are gas bubbles and the medium is a liquid. Bubbles consist of one or more coexisting gas components, mainly air or vapour. The lamellar interfacial length can reach orders of nanometres. Foam is present mainly in chemical and industrial processes [20]. Understanding the foam behaviour and simulating its movement is a difficult task because of the highly dispersed time and length scales where multiphase and multicomponent systems occur. Mobarak et al. [21] proposed an approach to simulate some of the chosen phenomena using the Boltzmann lattice method and discussed the limitations. The authors presented simulations of bubble rise in a partially filled flask-like container that included convective–diffusive heat transfer. The authors stated that they made progress in the direction of the modelling and parameterisation of full-scale industrial rectification columns, where foaming is a critical and often occurring problem [21].

Recently, great interest has been shown in the application of metal foams to high-power batteries, heat and cooling devices, and compact electronic heat sinks [22]. Metal foams can reduce the size and mass of devices because of their low density. Shan et al. [23] studied heat transfer in an electronic radiator filled with metal foam. The authors analysed the effect of several factors, such as flow rate, pores per linear inch, and number of fins, on the heat transfer coefficient. The authors proved that the metal foam reflects a much stronger ability of heat transfer compared to the electronic radiator without the metal foam. They found the optimal number of fins and observed that the higher the number of pores per linear inch, the greater the heat transfer coefficient [23].

The enhancement of heat transfer using passive methods demonstrates an advantage over active methods as they do not require external power. However, passive methods cause greater friction, which requires extra power to the pump, compressor, blower, or ventilator. Considering geometrically complex applications, such as ribbed pipes or cylinders with a wavy leading edge, several factors that affect turbulence and the drag coefficient can be recognised [24]. These are examples of the pitch and shape of the rib, the angle of attack of the rib, the coefficient of channel blockage, etc. [7–10]. Recently, Mousavi et al. [25] described a variety of passive methods used to intensify the heat transfer process. Kugele et al. [26] used the LES approach for a single-started helically ribbed pipe and validated the velocity and heat transfer between the ribs using experimental data obtained by Virgilio et al. [27,28]. The authors considered an acrylic glass with two different helical turbulators inserted into the pipe. The starting points for their mathematical model were continuity, Navier–Stokes equations, and energy equation in 3D form. They simulated 3D water flow in a pipe at a constant Re = 21,000 and Pr = 7.0, assuming constant thermophysical properties. Kugele et al. [26] demonstrated that the LES simulations gave fairly accurate results for the flow field, the level of turbulence, and the local heat transfer coefficient. Measurements and simulations of the Nusselt number in the pipe with and without helical turbulators confirmed a significant increase in the Nusselt number. Simulations of dynamic and thermal boundary layers performed by Kugele et al. [26] shed more light on complex turbulence processes and their influence on heat transfer in helically ribbed pipes.

The components of the gas turbine must be cooled to maintain its mechanical strength for a reliable operation. Cooling efficiency is essential in the region along the edge of the turbine blades and vanes. Lee et al. [29] applied large-eddy simulations to study turbulent flow in a channel of a certain height with a staggered array of pin fins with an appropriate diameter as a function of the heating loads that are relevant for the cooling of turbine blades and vanes. The results of the simulations were validated on the basis of data obtained from direct numerical simulation and experiments. The authors concluded that there are significant changes in the turbulent flow structure caused by heating loads, creating wall jets next to all heated surfaces [29].

Heating the steam pipeline that connects the boiler to the turbine is essential in the turbine and boiler start-up process. Rapid changes in steam temperature can cause high stresses on pipeline and turbine components. Kaczmarski [30] proposed a solution to the inverse heat transfer problem (IHP) in a steam pipeline, taking into account that the turbine manufacturer sets temporary changes in the steam temperature at the turbine input. The author proposed a method to predict the transient steam temperature at the entrance of the pipe based on the measured (known) steam temperature at the exit of the pipe. Taking into account the heat balance equations, the author formulated a set of ordinary differential equations. The author divided the pipeline domain into control volumes in the radial and longitudinal directions. Next, taking into account the boundary and initial conditions, he solved the set of equations using the Runge–Kutta method. The author presented the results of the simulations, which include time-dependent pressure, mass flow rate, wall and steam temperature, and the relative difference between inlet and outlet steam temperatures. The author concluded that his mathematical model can be used to control boiler operation [30].

The incredible increase in the use of renewable energy sources, which are unfortunately sensitive to weather conditions, has led to the development of thermal and electrical storages (accumulators). Thermal accumulators can use fluid, solid, or phase-change materials [31]. Taler et al. [32] built a test rig for a hybrid (electric–water system), for a building in which a ceramic heat accumulator was installed. The ceramic thermal accumulator contains an outer cylindrical shell inside which the ceramic cylinders and electrical resistance heaters were packed. The air flow generated by the fan was transported through the accumulator, which gave heat to the water flowing in the finned heat exchanger, which in turn was the heat source for the central heating system of the building. The calculated exit air temperature T = f(x,t) was compared with the measured data. The results of the simulations indicated good accuracy; however, for the initial air heating period, the differences were significant. The authors reasoned that their mathematical model could help to choose the best accumulator size for specific needs [32].

The global fibre optics market is rapidly developing, mainly in the electronics and telecommunication industries. Fibre strands can be made of glass or plastic and are known as microstructured optical fibres or photonic crystal fibres [33]. Mathematical models that can describe the fibre drawing process are highly desirable. Luzi et al. [34] developed a mathematical model of a fibre drawing process that includes the general equations of continuity, momentum, and energy. Assuming that the flow is incompressible and axially symmetric without a circumferential velocity component and taking into account the evolution equation for the inner and outer surfaces, developed by Fitt et al. [35], they built an asymptotic mathematical model of the unsteady 2D flow of the capillary drawing process. Luzi et al. [34] performed a successful validation of the mathematical model for annular capillaries. The authors performed simulations of the temperature and velocity distributions during the fibre production process. Their study can lead to a better understanding of the fibre drawing process and demonstrate the influence of physical quantities on the production process [34].

The high consumption of oil and gas results in an increase in explorations in deep-depth onshore and offshore territories. It is known that a high formation temperature causes failures in current operating designs [36]. In the drilling process, the drilling mud provokes a cooling effect in the formation. An accurate prediction of the temperature of the mud that flows into the drilling tube and annulus is very desirable [37]. Pioneering work was carried out by Bullard [38] who analytically predicted the temperature distribution using the diffusion equation. Jang et al. [39] proposed a mathematical model of transient heat transfer to compute the radial temperature in the drilling hole, annulus, and formation. The authors used the concept of quantifying thermal disturbance, called a thermally disturbed radius. On the basis of this concept, the authors can predict how long thermal disturbance occurs radially in the formation. Their physical model assumes that the drilling operation system consists of five subsections, such as the drilling hole, drilling pipe, annulus, casing, and reservoir formation. Taking into account some assumptions, the authors developed a mathematical model of unsteady transient heat transfer in a two-dimensional domain to derive the thermally disturbed radius. The authors performed a validation of the mathematical model based on the measurements and obtained satisfactory agreement. As a result of simulations, the dependence of temperature on the depth in the hole, temperature profiles, and thermal disturbance radius based on time and temperature were presented.

The authors reasoned that their mathematical model allows one to predict the temperature of the formulated well [39].

Great interest is expressed in increasing the efficiency of wind, water turbines, and photovoltaic modules to increase their contribution to the total amount of energy produced [40,41]. Hybrid platforms, which use more than one source of renewable energy, are becoming more popular and cost-effective. Torres et al. [42] conducted a study of a hybrid platform, which includes offshore wind and water turbines. The water turbine, which uses tidal power, is installed in a structured tower (together with a wind turbine). The main objective was to calculate the power generated by wind and water turbines [42]. Using the continuity equation and the RANS equations, the set of 3D equations was closed, taking into account the standard k-ε turbulence model. Simulations were performed assuming that the inlet velocity to the domain was the same as the velocity measured on a real platform and at the right height/depth. The results of the simulations were presented as graphs of velocity fields. On the basis of the simulations, the authors were able to calculate the energy production for the hybrid platform over a year [42].

The transport of the solid phase, using water as a carrier liquid, is commonly applied in industry. Solid–liquid transport usually requires more energy compared to a single-phase flow with the same flow rate. It is well known that a small amount of deflocculant results in a decrease in viscosity and, as a consequence, a decrease in wall shear stress. Jaworska-Jozwiak and Dziubinski [43] performed rheological measurements of fine dispersive limestone slurry with and without deflocculant in a wide range of solid concentrations. In the rheological experiments, the authors observed a significant decrease in viscosity and wall shear stress. Next, using the Bernoulli equation for the real liquid, the authors formulated an algebraic mathematical model for the hydrotransport of limestone from a reservoir to a settling tank for the chosen manufacturing enterprise. The authors reasoned that by adding a proper amount of the chosen deflocculant to the limestone slurry, it is possible to reduce the power consumption of the centrifugal pump motor by more than 50% [43].

The high level of water pollution around the world caused by the presence of plastics requires research, which helps to understand the spread of pollution. Plastic particles can settle and spread as sediment or flow freely with the carrier liquid or remain floating [44]. Kevorkijan et al. [45] conducted research on the settling process of plastic particles with diameters between 1 and 10 mm. The authors performed predictions and experiments on 3D turbulent flow in an open channel containing water and plastic particles. Their mathematical model includes CFD computations for liquid flow and the Lagrangian tracking method for plastic particles. The mathematical model dedicated to liquid flow constitutes time-averaged continuity and momentum equations, while the closure problem was solved using the standard k-ε turbulence model. The mathematical model dedicated to plastic particles constitutes the equations for the conservation of mass and Newton's second law. The authors performed simulations using Ansys software for vertical and horizontal velocity profiles, average settling time, and average downstream settling distance of plastic particles, as well as the formation of particle clouds. The authors reasoned that the proposed mathematical model of the settlement of particles in an open channel, which has been validated for turbulent flow, can be used on a larger scale to predict the propagation of mesoplastics in rivers [45].

3. Conclusions

Analysing the articles contributed to this Special Issue, titled *Numerical Heat Transfer and Fluid Flow 2022*, one can say that all articles are applied to specific environmental or engineering problems. The articles are encountered in fluid dynamics and/or heat transfer in machines, production and exploration process, environment, and other related areas of mechanical engineering.

Some articles dealt with simulations, while others presented their own experimental data. The articles, which dealt with simulations, contained a physical model with major assumptions such as the physical properties of the flowing medium, geometry of the flow domain, boundary, and initial conditions, and then a mathematical model. Mathematical models were formulated using conservation laws, such as equations of continuity, N-S, and energy. The authors formulated the set of 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 were properly validated.

Through the variety of approaches presented in this Special Issue, titled *Numerical Heat Transfer and Fluid Flow 2022*, the reader can find a description of the physical phenomenon, physical and mathematical models, and experimental data, and can gain a better understanding of the phenomena in a specific engineering application, including the interpretation of computed and measured quantities.

Funding: This research did not receive external funding.

Data Availability Statement: Data are contained within this article.

Acknowledgments: The author thanks the contributors of this Special Issue, titled *Numerical Heat Transfer and Fluid Flow 2022*, for valuable articles and MDPI for the invitation to act as a Guest Editor. Special thanks are extended to the editorial staff of *Energies* for efficient daily cooperation and great participation.

Conflicts of Interest: The author declares that there is no conflict of interest.

References

1. Bartosik, A. Numerical heat transfer and fluid flow: A review of contributions to the special issue. *Energies* **2022**, *15*, 2922. [CrossRef]
2. Patankar, S.V. (Ed.) *Numerical Heat Transfer and Fluid Flow*; Taylor and Frances Inc.: Abingdon, UK, 1980; p. 2014.
3. Spalding, D.B. *Turbulence Models for Heat Transfer*; Report HTS/78/2; Department of Mechanical Engineering, Imperial College of London: London, UK, 1978.
4. Spalding, D.B. *Turbulence Models—A Lecture Course*; Report HTS/82/4; Department of Mechanical Engineering, Imperial College of London: London, UK, 1983.
5. Castelvecchi, D. Mysteries of turbulence unravelled. *Nature* **2017**, *548*, 382–383. [CrossRef] [PubMed]
6. Klebert, P.; Su, B. Turbulence and flow field alterations inside a fish sea cage and its wake. *Appl. Ocean Res.* **2020**, *98*, 102–111. [CrossRef]
7. Zong, Y.; Bai, D.; Zhou, M.; Zhao, L. Numerical studies on heat transfer enhancement by hollow-cross disk for cracking coils. *Chem. Eng. Process. Process Intensif.* **2019**, *135*, 82–92. [CrossRef]
8. Nakhchi, M.E.; Esfahani, J.A. Numerical investigation of heat transfer enhancement inside heat exchanger tubes fitted with perforated hollow cylinders. *Int. J. Therm. Sci.* **2020**, *147*, 106153. [CrossRef]
9. Ahn, K.; Song, J.C.; Lee, J.S. Dependence of conjugate heat transfer in ribbed channel on thermal conductivity of channel wall: An LES Study. *Energies* **2021**, *14*, 5698. [CrossRef]
10. Pandey, L.; Singh, S. Numerical analysis for heat transfer augmentation in a circular tube heat exchanger using a triangular perforated Y-shaped insert. *Fluids* **2021**, *6*, 247. [CrossRef]
11. Hagiwara, Y. Effects of bubbles, droplets or particles on heat transfer in turbulent channel flows. *Flow Turbul. Combust.* **2011**, *86*, 343–367. [CrossRef]
12. Hamed, H.; Mohhamed, A.; Khalefa, R.; Habeeb, O. The effect of using compound techniques (passive and active) on the double pipe heat exchanger performance. *Egypt. J. Chem.* **2021**, *64*, 2797–2802. [CrossRef]
13. Zhao, J.; Zhang, B.; Fu, X.; Yan, S. Numerical study on the influence of vortex generator arrangement on heat transfer enhancement of oil-cooled motor. *Energies* **2021**, *14*, 6870. [CrossRef]
14. Urbanowicz, K.; Bergant, A.; Stosiak, M.; Deptuła, A.; Karpenko, M. Navier-Stokes solutions for accelerating pipe flow—A review of analytical models. *Energies* **2023**, *16*, 1407. [CrossRef]
15. Bhanvase, B.; Barai, D. *Nano Fluid for Heat and Mass Transfer—Fundamentals, Sustainable Manufacturing and Applications*; Academic Press: London, UK, 2021; p. 448. Available online: https://www.elsevier.com/books-and-journals (accessed on 29 April 2021).
16. Cieśliński, J. Numerical modelling of forced convection of nanofluids in smooth, round tubes: A review. *Energies* **2022**, *15*, 7586. [CrossRef]
17. Lotfi, R.; Saboohi, Y.; Rashidi, A.M. Numerical study of forced convective heat transfer of nanofluids: Comparison of different approaches. *Int. Commun. Heat Mass Transf.* **2010**, *37*, 74–78. [CrossRef]
18. Mokmeli, A.; Saffar-Avval, M. Prediction of nanofluid convective heat transfer using the dispersion model. *Int. J. Therm. Sci.* **2010**, *49*, 471–478. [CrossRef]

19. Maïga, S.; Nguyen, C.T.; Galanis, N.; Roy, G. Heat transfer behaviours of nanofluids in a uniformly heated tube. *Superlattices Microstruct.* **2004**, *35*, 543–557. [CrossRef]
20. Cantat, I.; Cohen-Addad, S.; Elias, F.; Graner, F.; Höhler, R.; Pitois, O.; Rouyer, F.; Saint-Jalmes, A. *Foams: Structure and Dynamics*; OUP: Oxford, UK, 2013.
21. Mobarak, M.; Gatternig, B.; Delgado, A. On the simulations of thermal liquid foams using Lattice Boltzmann method. *Energies* **2023**, *16*, 195. [CrossRef]
22. Shen, L.; Xu, S.; Bai, Z.; Wang, Y.; Xie, J. Experimental study on thermal and flow characteristics of metal foam heat pipe radiator. *Int. J. Therm. Sci.* **2021**, *159*, 106572. [CrossRef]
23. Shan, X.; Liu, B.; Zhu, Z.; Bennacer, R.; Wang, R.; Theodorakis, P.E. Analysis of the heat transfer in electronic radiator filled with metal foam. *Energies* **2023**, *16*, 4224. [CrossRef]
24. Ferreira, P.H.; Araújo, T.B.; Carvalho, E.O.; Fernandes, L.D.; Moura, R.C. Numerical investigation of flow past bio-inspired wavy leading-edge cylinders. *Energies* **2022**, *15*, 8993. [CrossRef]
25. Mousavi Ajarostaghi, S.S.; Zaboli, M.; Javadi, H.; Badenes, B.; Urchueguia, J.F. A Review of recent passive heat transfer enhancement methods. *Energies* **2022**, *15*, 986. [CrossRef]
26. Kügele, S.; Mathlouthi, G.O.; Renze, P.; Grützner, T. Numerical simulation of flow and heat transfer of a discontinuous single started helically ribbed pipe. *Energies* **2022**, *15*, 7096. [CrossRef]
27. Virgilio, M.; Mayo, I.; Dedeyne, J.; Geem, K.V.; Marin, G.; Arts, T. Effects of 2-D and 3-D helical inserts on the turbulent flow in pipes. *Exp. Therm. Fluid Sci.* **2020**, *110*, 109923. [CrossRef]
28. Virgilio, M.; Mayo, I.; Dedeyne, J.; Geem, K.V.; Marin, G.; Arts, T. Influence of obstacles on the wall heat transfer for 2D and 3D helically ribbed pipes. *Int. J. Heat Mass Transf.* **2020**, *148*, 119087. [CrossRef]
29. Lee, C.S.; Shih, T.I.P.; Bryden, K.M.; Dalton, R.P.; Dennis, R.A. Strongly heated turbulent flow in a channel with pin fins. *Energies* **2023**, *16*, 1215. [CrossRef]
30. Kaczmarski, K. Identification of transient steam temperature at the inlet of the pipeline based on the measured steam temperature at the pipeline outlet. *Energies* **2022**, *15*, 5804. [CrossRef]
31. Li, P.-W.; Chan, C.L. *Thermal Energy Storage Analyses and Designs*; Academic Press: London, UK, 2017.
32. Taler, D.; Taler, J.; Sobota, T.; Tokarczyk, J. Cooling modelling of an electrically heated ceramic heat accumulator. *Energies* **2022**, *15*, 6085. [CrossRef]
33. Russell, P.S. Photonic-crystal fibers. *J. Light. Technol.* **2006**, *24*, 4729–4749. Available online: https://opg.optica.org/abstract.cfm?URI=OFC-2009-OTuC1 (accessed on 13 July 2023). [CrossRef]
34. Luzi, G.; Lee, S.; Gatternig, B.; Delgado, A. An asymptotic energy equation for modelling thermo fluid dynamics in the optical fibre drawing process. *Energies* **2022**, *15*, 7922. [CrossRef]
35. Fitt, A.D.; Furusawa, K.; Monro, T.M.; Please, C.P.; Richardson, D.J. The mathematical modelling of capillary drawing for holey fibre manufacture. *J. Eng. Math.* **2002**, *43*, 201–227. [CrossRef]
36. Wu, B.; Zhang, X.; Jeffrey, R.G. A model for downhole fluid and rock temperature prediction during circulation. *Geothermics* **2014**, *50*, 202–212. [CrossRef]
37. Dirksen, R. Upgrading formation-evaluation electronics for high-temperature drilling environments. *J. Pet. Technol.* **2011**, *63*, 24–26. [CrossRef]
38. Bullard, E.C. The time necessary for a bore hole to attain temperature equilibrium. *Geophys. J. Int.* **1947**, *5*, 127–130. [CrossRef]
39. Jang, M.; Chun, T.S.; An, J. The transient thermal disturbance in surrounding formation during drilling circulation. *Energies* **2022**, *15*, 8052. [CrossRef]
40. Micallef, D.; van Bussel, G. A Review of urban wind energy research: Aerodynamics and other challenges. *Energies* **2018**, *11*, 2204. [CrossRef]
41. Santamaría, L.; Vega, M.G.; Pandal, A.; Pérez, J.G.; Velarde-Suárez, S.; Oro, J.M.F. Aerodynamic performance of VAWT airfoils: Comparison between wind tunnel testing using a new three-component strain gauge balance and CFD modelling. *Energies* **2022**, *15*, 9351. [CrossRef]
42. Torres, J.P.N.; De Jesus, A.S.; Lameirinhas, R.A.M. How to improve an offshore wind station. *Energies* **2022**, *15*, 4873. [CrossRef]
43. Jaworska-Jozwiak, B.; Dziubinski, M. Effect of deflocculant addition on energy savings in hydrotransport in the lime production process. *Energies* **2022**, *15*, 3869. [CrossRef]
44. Klein, S.; Dimzon, I.K.; Eubeler, J.; Knepper, T.P. Analysis, occurrence, and degradation of microplastics in the aqueous environment. In *Handook of Environmental Chemistry*; Springer: Cham, Switzerland, 2018; Volume 58, pp. 51–67.
45. Kevorkijan, L.; Žic, E.; Lešnik, L.; Biluš, I. Settling of mesoplastics in an open-channel flow. *Energies* **2022**, *15*, 8786. [CrossRef]