

Editorial

Numerical Heat Transfer and Fluid Flow: A Review of Contributions to the Special Issue

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

1. Introduction

The paper contains a summary of successful invited papers addressed to the Special Issue on ‘Numerical Heat Transfer and Fluid Flow’, which were published in 2021 in the scientific journal ‘Energies’. Invitations were addressed to specialists from all over the world who deal with mathematical modeling, simulations, and experiments on heat and/or fluid flow. The submitted papers regarded the solution of problems of scientific and industrial relevance in a specific field of heat transfer and fluid transportation, including natural resources, technical devices, industrial processes, etc. Papers addressed to the Special Issue not only solved specific engineering problems, but served as a catalyst on future directions and priorities in numerical heat transfer and fluid flow. Most papers dealt with heat transfer in single-phase flow of air, in particular technical devices, while part of them regarded liquid and solid–liquid flows. Reliable predictions require reliable measurements; therefore, the majority of the papers presented experimental data and validation of mathematical models.

The importance of heat and fluid flow is still growing in all aspects of our lives, starting from nature and ending with industrial processes. In the era of digital transformation, which includes converting any process into a quantified format suitable for analysis, there is an increasing demand for modeling, simulations, and experiments on heat exchange in fluid flow for a variety of single and multiphase flows, and also boundary conditions [1]. Thanks to computational fluid dynamics and its commercial packages, especially Ansys, we can design and optimize various industrial processes. The increasing understanding of heat and mass transfer phenomena has contributed significantly to the development of new methods and techniques for solving and effectively managing many engineering processes.

Formulating any problem of prediction of heat transfer and/or fluid flow requires developing a physical model first. The next step is developing a mathematical model which fulfills the assumptions stated in the physical model and defines boundary and initial conditions. The mathematical model should be based on general governing equations, like continuity, Navier-Stokes, and energy equations. The equation set can be solved analytically—which is complicated and impractical—or numerically. If numerical methods are considered, we can use approaches like direct numerical simulation (DNS), for instance. Such a method is time-consuming, expensive, and not practical for many engineering applications. Other methods like, for instance, modelling of turbulence, which uses random averaged Navier-Stokes equations (RANS), or large eddy simulation (LES) were proved for a variety of engineering applications and are less time consuming and less expensive; however, the set of equations require closure. The problem of closure requires additional equation or equations, like those proposed by turbulence models. Requirements for turbulence models, formulated by the honorable founder of computational fluid dynamics (CFD), that is, Dudley Brian Spalding, are the following: universality, economy, extensionality, and reality [2,3]. The ability to simulate heat transfer and/or fluid flow, which includes velocity, pressure, and temperature distributions, for engineering purposes, remains one of the main challenges in CFD.



Citation: Bartosik, A.S. Numerical Heat Transfer and Fluid Flow: A Review of Contributions to the Special Issue. *Energies* **2022**, *15*, 2922. <https://doi.org/10.3390/en15082922>

Received: 11 April 2022

Accepted: 14 April 2022

Published: 15 April 2022

Publisher’s Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



Copyright: © 2022 by the author. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

Considering the heat exchange between a transported fluid and the surrounding, we recognize methods and techniques focused on the enhancement of heat transfer, named passive or active. Passive methods, such as increasing heat transfer area and/or temperature difference, shaping an insert with dedicated perforation, or mechanically deformed pipes, have been studied for several years and have become commercial solutions [4–9]. Active methods, such as air injection, bubble or vortex generation, or proper pulsation, can lead to increased heat transfer coefficient, and, finally, can produce increased heat transfer process [10–13]. Some of such methods have been demonstrated by contributors to the Special Issue.

2. Review of Contribution to the Special Issue

The improvement of heat transfer by the use of ribs has been studied for several decades. It is known that the pitch and shape of the rib, the angle of attack of the rib, and the coefficient of channel blockage have a significant effect on heat transfer [14]. Joon Ahn et al. [15] made the LES simulation of the dependence of conjugate heat transfer in a ribbed channel on the thermal conductivity of the channel wall. The authors noted that for such a case, RANS underpredicts heat transfer and does not accurately predict local peaks, while the same predictions using LES give more accurate results. The authors considered a channel wall with a thickness three times the height of the rib. Starting points for their mathematical model were continuity, Navier-Stokes, and energy equations in 3D form, which were transformed to dimensionless governing equations. The authors assumed that the flow is fully developed with periodic boundary conditions in the stream direction. All simulations were performed for 10,000 time steps to reach a steady state. The simulations were positively validated with the available data available in the literature. On the basis of the simulations, the authors analyzed the conjugate heat transfer characteristics in a ribbed channel. The authors made simulations in a wide range of conductivity ratios between the gas turbine blade material and air. Taking into account the results of simulations of time-averaged temperature and heat transfer distribution, they analyzed whether the mechanism responsible for promoting heat transfer under pure convection conditions is valid for a variety of conductivity ratios. The authors concluded that if the conductivity ratio between the solid wall and the fluid exceeded 100, the heat transfer characteristics were similar to those under isothermal conditions, and the vortices at the corners of the ribs strongly influenced the convective heat transfer. For a conductivity ratio below 100, vortices located in the corners played an important role in heat transfer [15]. The authors concluded that the ‘thermal resistance of the solid wall of the channel to convective heat transfer was observed in the turbulent flow regime’ [15].

Heat pipes have been extensively developed for some decades. The main advantages are no moving parts, high reliability, and fair efficiency. Working fluid transport exists naturally, does not require energy input, and can transfer heat for significant distances. Recently, heat pipes heat exchangers have gained popularity in heat recovery applications such as air conditioning, dehumidifiers in air conditioning systems, technological processes, etc. [16]. Górecki et al. [17] conducted experimental and numerical studies on heat pipe heat exchangers with individually finned heat pipes. The authors conducted a study on modeling, design, and experiments on a heat pipe heat exchanger comprised of individually finned heat pipes utilized as a recuperator in small air conditioning systems with airflow between 300 m³/h and 500 m³/h. Using the available algebraic correlations, the authors developed a thermal heat pipe heat exchanger model. Based on their previous research, they used R404A refrigerant as a working fluid. The mathematical model consists of a set of algebraic equations and semi-empirical functions. On the basis of parametric studies, the authors concluded that 20 rows of finned heat pipes in the staggered arrangement guarantee stable heat exchanger efficiency equal about 60%. The authors emphasized that the designed heat pipe heat exchanger made of an individually finned heat pipe bundle is a competitive choice to the continuous plate-fin alternative. On the basis of the parametric studies, the authors designed and then constructed the heat exchanger.

Measurements made on the experimental rig were referred to the predictions made by the mathematical model, and they noted that the relative difference was about 10%. In accordance with the authors statement, the method can improve a heat pipe heat exchanger efficiency significantly. Future research should focus on intensifying heat transfer by using turbulators in an individually finned tube bundle located between heat pipes [17].

Recent interest in the liquefaction of natural gas, which occupies only 1/600 of its volume, is due to the fact that it is easier and more economical to transport and store [18–20]. Some researchers stressed that ‘the development of small-scale energy in the coming years is expected to be associated with the widespread use of liquefied natural gas, which is recognized as one of the most promising types of energy carriers’ [21]. By taking into account the real thermophysical properties of the gas, Avramenko et al. [21] analysed the method for solving steady-state natural convection of van Der Waals gas near a vertical heated plate. The authors proposed a novel simplified form of equations in analytical form for real gases, enabling the estimation of the effects of the dimensionless van der Waals parameters on the normalized heat transfer coefficient and Nusselt number. Using the integral method for the momentum and energy equations, they obtained an approximate analytical solution. The authors demonstrated changes in heat transfer intensity, which are due to effects that are considered by the van der Waals equation of state, but not by the ideal gas equation [21].

Some research is developing a fluidic oscillator. The fluid oscillator usually has two feedback channels and does not possess moving parts and can improve aerodynamics, such as lift forces, mixing processes, and heat transfer, for instance [22]. Kim and Kim [23] studied numerically a fluid oscillator with a bent outlet nozzle. The authors analyzed the influence of various mounting conditions, such as the arrangement and installation angles of the fluidic oscillator in the range from 0° to 40°, on the characteristics of the oscillator with and without external flow. The authors performed analyses for air which has been treated as an ideal gas. They considered 3D unsteady RANS equations with a shear stress transport turbulence model. The authors used an Ansys commercial CFD software. They concluded that the pitch angle of fluidic oscillators had the most sensitive effect on the flow control [23].

The construction of tube heat exchangers is complex. Cross-thin metal sheets, tubes, and fin pitches, with different sizes and numbers of rows, cause the complexity of fluid flow [24]. Usually, each row operates in a different way, which has an effect on the level of turbulence. For these reasons, the characteristics of tube heat exchangers are determined mainly experimentally. Marcinkowski et al. [25] presented a method to determine individual correlations for the air side Nusselt numbers on each row of tubes for a four-row finned heat exchanger with continuous flat fins and round tubes in a staggered tube layout. The authors’ method was formulated using CFD modelling; however, the description of the model is limited. The authors concluded that their approach enables the selection of the optimum number of tube rows for a given heat output of the heat exchanger. The authors stated that the approach allows the reduction of investment costs of constructing heat exchangers by decreasing the tube row number. In addition, operating costs can be reduced as a result of reducing air pressure losses [25].

The increase in efficiency of flow machines dedicated to the transport of gases is mainly associated with blade impeller and labyrinth seals. This is especially important if high-power generating machines are considered. Liang et al. [26] proved that a 1% increase in seal-tooth clearance height causes a significant decrease in multistage axial compressor performance and efficiency. Some researchers are using CFDs to design seals of higher leak tightness. Joachimiak [27] proposed a method of aerodynamic sealing to reduce leakage by matching the seal geometry to the flow. The authors considered a staggered labyrinth seal for steady flow conditions. The method contains CFD predictions assuming that the air is an ideal gas. The author used the RANS method for calculations with the k- ω SST turbulence model for 2D axisymmetric geometry. His calculation domain consisted of 354,000 elements. The author considered two approaches, that is, changed and unchanged

seal height. The author concluded that the newly designed geometry reveals an almost stable relative reduction in the leakage rate, irrespective of the pressure ratio upstream and downstream of the seal [27]. His conclusions require validation. However, such measurements are complex and difficult to perform.

High battery costs, which contribute more than 40% of the total price of electrical vehicles, and high charging and discharging rates, variable frequency of charging, and finally battery packing, a proper cooling system and safety, make research in this field very desirable [28]. Widyantara et al. [29] made an approach to optimize the design of the lithium-ion battery pack for electric vehicles to meet the optimal operating temperature using an air-cooling system by modifying the number of cooling fans and the inlet air temperature. A 3D numerical model of the packing of the 74 V and 2.31 kWh batteries and CFD simulations based on the lattice Boltzmann method have been applied. Furthermore, the authors developed a battery thermal management system based on consideration of temperature distribution and power consumption. The authors concluded that three cooling fans with 25 °C inlet air temperature gave the best performance, with low power required. The authors' findings could be helpful in developing a standardized battery packing module and in designing low-cost battery packing for electric vehicles. The authors emphasized that in future work, an investigation is recommended on the effect of employing variable speed for cooling fans and a study on structural strength and water protection for air conditioning systems [29].

In recent years, we have observed interest in developing new oil-cooled motors with high power density, low vibration and noise, strong overload capacity, and high efficiency. Such oil-cooled motors require modern heat exchangers with an enhanced heat transfer process. The most typical liquid used in oil-cooled motors is oil, which is relatively cheap and emphasizes long service life. Some researchers are using vortex generators in radiators to improve overall heat transfer performance [30]. Junjie Zhao et al. [31] performed a numerical study on the influence of the vortex generator arrangement on the enhancement of heat transfer in oil-cooled motors. The authors carefully formulated a physical model, and then developed a mathematical model that constituted continuity, momentum, and energy equations. The closure problem was solved using the standard $k-\epsilon$ turbulence model. The set of equations for the 2D case was solved using the commercial software Fluent. They performed grid-independent tests concluding that the grid number equal to 6,850,000 is sufficient for simulations. Validation of the mathematical model, which includes frictional coefficient and Nusselt number, is limited, which is due to the fact that the numerical results were compared with other numerical predictions available in the literature. To increase the heat transfer coefficient, the authors studied the influence of an attack angle of the vortex generator. The authors concluded that for a single pair of rectangular vortex generators, the attack angle equal to 45° gives the best results in enhancing heat transfer [31].

The water hammer phenomenon can lead to pipeline system failures; however, such a phenomenon depends on the material of the pipeline [32]. Steel pipes are better recognized in the literature, while viscoelastic pipelines are less. Kubrak et al. [33] made experiments and predictions on hydraulic transients in a viscoelastic pipeline system with sudden cross-sectional changes. The authors focused on high-density polyethylene pipelines. The authors investigated the influence of sudden cross-sectional changes in a high-density polyethylene pipeline system on pressure oscillations during the water hammer phenomenon. The authors recorded pressure changes downstream of the pipeline system during a valve-induced water hammer. They formulated a mathematical model which constitutes the continuity and momentum equations for non-steady and one-dimensional flow. A set of transient flow equations for a polymeric pipe was numerically solved using the MacCormack explicit method. The authors concluded that for high mass flow rates, the jet frequency increases with the bending angle; however, at a bending angle equal to 40°, the oscillation of the jet disappears. The authors also observed that the effect of external flow on the frequency increases with the increase of the bending angle. The predicted numerical pressure fluctuations showed satisfactory agreement with their measurements

in terms of both phase and amplitude. The authors noted that more work needs to be done to study the use of the water hammer in variable property pipeline systems. Future studies should also focus on simulating the water hammer in serially connected pipes with various inner diameters and made of different materials [33].

Saving energy is a main goal of engineers. This includes, for instance, fluid transportation and control of hydraulic systems. Controlling and adjustment of hydraulic elements, such as valves, pumps, receivers, etc., is one of the possible ways which effect the efficiency of such systems [34]. In hydrostatic systems, efficiency can be improved in a number of ways, depending on whether the system features fixed or variable displacement pumps. For example, in the system with fixed displacement pumps controlled by the throttle method, the main approach is to limit the operation of the safety valve. Bury et al. [35] performed simulations and experiments with control signals of various shapes and feedback from the hydraulic system. The authors analyzed the pressure at the pump inlet and outlet, the flow rate, and the rotational speed of the hydraulic motor as functions of time. The authors built a simple mathematical model taking into account the Hagen-Poiseuille equation for laminar flow and the Bernoulli equation for turbulent flow. The model constitutes a continuity equation at particular points in the hydraulic circuit and the equilibrium equation of torque on the shaft of the hydrostatic motor. In accordance with the authors statement, their method of modelling the opening characteristics of the proportional spool valve allows us to optimize the start-up process in terms of the execution of the objective function for the parameters like, for instance, start-up time, reaction time, or energy efficiency of the system. On the basis of the analyses, the authors concluded that the maximum pressure value at the inlet port of the hydraulic motor during its start-up can be modified by adjusting the shape of the proportional spool valve (symmetrical, asymmetrical). This solution also reduces the noise caused by the transmission during start-up and reduces the load on elements of the transmission, which extends its operational life [35].

Thermal conductivity is essential when rapid changes in temperature occur in some parts of mechanical machines. This phenomenon exists in mechanical face seals, especially during startup [36]. Błasiak [37] performed an analysis of heat transfer for non-contact mechanical face seals using the variable order derivative approach. The author proposed a physical and next mathematical model for noncontact mechanical face seals for the conduction of heat from the liquid film in the gap to the rotor and the stator, followed by convection to the water surrounding them. The author used a variable-order derivative time fractional model to describe heat transfer. His equation of heat transfer depends on time and was solved analytically; however, the characteristic features of the equation were determined through numerical simulations. The objective of the study was “to compare the results of the classical equation of heat transfer with the results of the equations involving the use of the fractional order derivative”. Author concluded that his method allows one to predict the heat transfer phenomena occurring in non-contacting face seals, especially during the startup. The author emphasized that his mathematical approach, which is based on the fractional differential equation, is suitable to develop more detailed mathematical models for similar phenomena [37].

It is well known that cavitation is a dangerous process that affects pipelines and flow machines [38]. Urbanowicz et al. [39] conducted research on modeling transient pipe flow in plastic pipes with a modified discrete bubble cavitation model. In the physical model of unsteady pipe flow, they assumed the importance of three phenomena: unsteady wall shear stress, vaporous cavitation, and pipe wall retarded strain. The authors considered plastic pipelines characterized by the retarded strain (RS) that occurs on the wall of these pipes. Using the convolution integral of the local derivative of pressure and the creep function that describes the viscoelastic behavior of the pipe wall material, they calculated the RS. The authors formulated the equations of a discrete bubble cavity model by using the continuity equations for the gas and liquid phases separately, and the momentum equation for two-phase vaporous cavitation. They transformed the set of partial differential equations into a set of ordinary differential equations. The authors concluded that the

modified unsteady discrete bubble cavity model allows one to simulate the pressure and velocity waveforms in which vapor areas appear as a result of the cavitation phenomenon in plastic pipes. Such a model seems to be suitable for complex networks, such as water supply, oil hydraulics, heating, etc. The authors noted that the comparison of computed results with measurements showed that the novel discrete bubble cavity model predicts pressure and velocity waveforms, including cavitation and retarded strain effects, with sufficient precision. The authors noticed that the influence of unsteady friction on the damping of pressure waves was much smaller than the influence of retarded strain [39].

Slurry pipeline transport is widely used in many industrial applications. In a situation where long distances of transportation are considered, predictions of heat exchange are usually focused on friction losses, heat transfer coefficient, velocity, and temperature distributions. Solving such a problem requires a proper physical model which includes physical properties of solid and carrier liquid phases, solid concentration, regime of flow, like homogeneous or heterogeneous, laminar or turbulent, rheological model, and boundary conditions. If the solid phase is very fine, the slurry demonstrates a yield shear stress, which increases with the increase in solid concentration. In such a case, it is quite common that the damping of turbulence appears [40]. Bartosik [41] proposed a mathematical model of heat transfer in turbulent flow of fine-dispersive slurry, including a specially designed wall damping function. The mathematical model constitutes the continuity, momentum, and energy equations. The closure problem was solved by taking into account the k - ϵ turbulence model and the rheological model. The study was focused on developing a new correlation of the Nusselt number for turbulent flow of fine dispersive slurry that exhibits yield shear stress and damping of turbulence. The new correlation of the Nusselt number includes Reynolds and Prandtl numbers, solid volume concentration, and dimensionless yield shear stress. The mathematical model was solved numerically and validated for heat transfer in carrier liquid flow only, as there are no data available for heat exchange in such slurries. The study demonstrates substantial differences between the slurry and velocity distributions at the pipe wall. The author concluded that for the same bulk velocity of the slurry, the Nusselt number decreases with an increase in solid volume concentration, and the highest rate of decrease in the Nusselt number is for a solid concentration below 10% by volume. The author noticed that more work needs to be done on examining the new Nusselt number for solid volume concentrations below 10% and greater than 30%, for different heat fluxes and pipe diameters [41].

3. Conclusions

Analyzing the papers contributed to the Special Issue, 'Numerical Heat Transfer and Fluid Flow', one can say that all papers are applied to specific engineering problems. All papers are encountered in fluid dynamics in machines, electronic packaging, chemical processes, and other related areas of mechanical engineering.

The majority of the papers dealt with simulations. For this reason, the papers presented physical models, which include major assumptions, physical properties of the flowing medium, boundary, and initial conditions, and also mathematical models. Mathematical models were formulated using conservation laws, such as the continuity, N-S, and energy equations. Using the boundary and initial conditions, the authors formulated the set of equations and solved them numerically, taking into account the convergence criteria and ensuring a grid-independent solution. Part of the models have been validated in full, while part have been validated in the limited scale. Some papers presented their own experimental data.

Through these approaches, the readers can find a variety of physical and mathematical models and can gain a better understanding of phenomena in the specific engineering applications of heat transfer and/or fluid flow, including interpretation of computed and measured results.

Funding: This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

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

References

1. Patankar, S.V. *Numerical Heat Transfer and Fluid Flow*; Minkowycz, W.J., Sparrow, E.M., Eds.; Taylor and Frances Inc.: Washington, DC, USA, 1980; p. 2014.
2. Spalding, D.B. *Turbulence Models for Heat Transfer*; Report No. HTS/78/2; Imperial College of London, Department of Mechanical Engineering: London, UK, 1978.
3. Spalding, D.B. *Turbulence Models—A Lecture Course*; Report No. HTS/82/4; Imperial College of London, Department of Mechanical Engineering: London, UK, 1983.
4. Fathinia, F.; Parsazadeh, M.; Heshmati, A. Turbulent forced convection flow in a channel over periodic grooves using nanofluids. *Int. J. Mech. Mechatron. Eng.* **2012**, *6*, 12–2782. [[CrossRef](#)]
5. Faruk, O.C.; Celik, N. Numerical investigation of the effect of flow and heat transfer of a semi-cylindrical obstacle located in a channel. *Int. J. Mech. Aerosp. Ind. Mechatron. Manuf. Eng.* **2013**, *7*, 891–896.
6. Sen, D.; Ghosh, R.A. Computational study of very high turbulent flow and heat transfer characteristics in circular duct with hemispherical inline baffles. *Int. J. Mech. Aerosp. Ind. Mechatron. Manuf. Eng.* **2015**, *9*, 1046–1051.
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. 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](#)]
10. Bartosik, A. Numerical Modelling of Fully Developed Pulsating Flow with Heat Transfer. Ph.D. Thesis, Kielce University of Technology, Kielce, Poland, 1989.
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. Bayat, H.; Majidi, M.; Bolhasan, M.; Karbalaie Alilou, A.; Mirabdollah, A.; Lavasani, A. Unsteady flow and heat transfer of nanofluid from circular tube in cross-flow. *Int. Sch. Sci. Res. Innov.* **2015**, *9*, 2078–2083.
13. 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](#)]
14. Yusefi, A.; Nejat, A.; Sabour, H. Ribbed Channel Heat Transfer Enhancement of an Internally Cooled Turbine Vane Using Cooling Conjugate Heat Transfer Simulation. *Therm. Sci. Eng. Prog.* **2020**, *19*, 100641. [[CrossRef](#)]
15. 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](#)]
16. Tian, E.; He, Y.L.; Tao, W.Q. Research on a new type waste heat recovery gravity heat pipe exchanger. *Appl. Energy* **2017**, *188*, 586–594. [[CrossRef](#)]
17. Gorecki, G.; Łecki, M.; Gutkowski, A.N.; Andrzejewski, D.; Warwas, B.; Kowalczyk, M.; Romaniak, A. Experimental and Numerical Study of Heat Pipe Heat Exchanger with Individually Finned Heat Pipes. *Energies* **2021**, *14*, 5317. [[CrossRef](#)]
18. Banaszkiwicz, T.; Chorowski, M.; Gizicki, W.; Jedrusyna, A.; Kielar, J.; Malecha, Z.; Piotrowska, A.; Polinski, J.; Rogala, Z.; Sierpowski, K. Liquefied Natural Gas in Mobile Applications—Opportunities. *Energies* **2020**, *13*, 5673. [[CrossRef](#)]
19. Osorio-Tejada, J.; Llera-Sastresa, E.; Scarpellini, S. Liquefied natural gas: Could it be a reliable option for road freight transport in the EU? *Renew. Sustain. Energy Rev.* **2017**, *71*, 785–795. [[CrossRef](#)]
20. Staffell, I.; Scamman, D.; Abad, A.V.; Balcombe, P.; Dodds, P.E.; Ekins, P.; Shah, N.; Ward, K.R. The role of hydrogen and fuel cells in the global energy system. *Energy Environ. Sci.* **2019**, *12*, 463–491. [[CrossRef](#)]
21. Avramenko, A.A.; Shevchuk, I.V.; Kovetskaya, Y.Y.; Dmitrenko, N.P. An Integral Method for Natural Convection of Van Der Waals Gases over a Vertical Plate. *Energies* **2021**, *14*, 4537. [[CrossRef](#)]
22. Gregory, J.; Tomac, M.A. Review of fluidic oscillator development and application for flow control. *AIAA Pap.* **2013**, 2013–2474. [[CrossRef](#)]
23. Kim, N.H.; Kim, K.Y. Effects of Bent Outlet on Characteristics of a Fluidic Oscillator with and without External Flow. *Energies* **2021**, *14*, 4342. [[CrossRef](#)]
24. Thulukkanam, K. *Heat Exchanger Design Handbook*, 2nd ed.; CRC Press: Boca Raton, FL, USA, 2013.
25. Marcinkowski, M.; Taler, D.; Taler, J.; Weglarz, K. Thermal Calculations of Four-Row Plate-Fin and Tube Heat Exchanger Taking into Account Different Air-Side Correlations on Individual Rows of Tubes for Low Reynold Numbers. *Energies* **2021**, *14*, 6978. [[CrossRef](#)]
26. Liang, D.; Jin, D.; Gui, X. Investigation of seal cavity leakage flow effect on multistage axial compressor aerodynamic performance with a circumferentially averaged method. *Appl. Sci.* **2021**, *11*, 3937. [[CrossRef](#)]
27. Joachimiak, D. Novel Method of the Seal Aerodynamic Design to Reduce Leakage by Matching the Seal Geometry to Flow Conditions. *Energies* **2021**, *14*, 7880. [[CrossRef](#)]
28. Huda, M.; Koji, T.; Aziz, M. Techno Economic Analysis of Vehicle to Grid (V2G) Integration as Distributed Energy Resources in Indonesia Power System. *Energies* **2020**, *13*, 1162. [[CrossRef](#)]

29. Widiantara, R.D.; Naufal, M.A.; Sambegoro, P.L.; Nurprasetio, I.P.; Triawan, F.; Djamari, D.W.; Nandiyanto, A.B.D.; Budiman, B.A.; Aziz, M. Low-Cost Air-Cooling System Optimization on Battery Pack of Electric Vehicle. *Energies* **2021**, *14*, 7954. [[CrossRef](#)]
30. He, Y.L.; Zhang, Y.W. Advances and Outlooks of Heat Transfer Enhancement by Longitudinal Vortex Generators. *Adv. Heat Transf.* **2012**, *44*, 119–185. [[CrossRef](#)]
31. 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](#)]
32. Soares, A.K.; Covas, D.I.; Reis, L.F. Analysis of PVC Pipe-Wall Viscoelasticity during Water Hammer. *J. Hydraul. Eng.* **2008**, *134*, 1389–1394. [[CrossRef](#)]
33. Kubrak, M.; Malesinska, A.; Kodura, A.; Urbanowicz, K.; Stosiak, M. Hydraulic Transients in Viscoelastic Pipeline System with Sudden Cross-Section Changes. *Energies* **2021**, *14*, 4071. [[CrossRef](#)]
34. Xiong, S.; Wilfong, G.; Lumkes, J.J. Components Sizing and Performance Analysis of Hydro-Mechanical Power Split Transmission Applied to a Wheel Loader. *Energies* **2019**, *12*, 1613. [[CrossRef](#)]
35. Bury, P.; Stosiak, M.; Urbanowicz, K.; Kodura, A.; Kubrak, M.; Malesinska, A. A Case Study of Open- and Closed-Loop Control of Hydrostatic Transmission with Proportional Valve Start-Up Process. *Energies* **2021**, *14*, 1860. [[CrossRef](#)]
36. Warbhe, S.D.; Tripathi, J.J.; Deshmukh, K.C.; Verma, J. Fractional Heat Conduction in a Thin Circular Plate with Constant Temperature Distribution and Associated Thermal Stresses. *J. Heat Transf.* **2017**, *139*, 44502. [[CrossRef](#)]
37. Blasiak, S. Heat Transfer Analysis for Non-Contacting Mechanical Face Seals Using the Variable-Order Derivative Approach. *Energies* **2021**, *14*, 5512. [[CrossRef](#)]
38. Wylie, E.B.; Streeter, V.L.; Suo, L. *Fluid Transients in Systems*; Prentice Hall: Englewood Cliffs, NJ, USA, 1993.
39. Urbanowicz, K.; Bergant, A.; Kodura, A.; Kubrak, M.; Malesinska, A.; Bury, P.; Stosiak, M. Modeling Transient Pipe Flow in Plastic Pipes with Modified Discrete Bubble Cavitation Model. *Energies* **2021**, *14*, 6756. [[CrossRef](#)]
40. Wilson, K.; Thomas, A. A new analysis of the turbulent flow of non-Newtonian fluids. *Can. J. Chem. Eng.* **1985**, *63*, 539–546. [[CrossRef](#)]
41. Bartosik, A. Numerical Modelling of Heat Transfer in Fine Dispersive Slurry Flow. *Energies* **2021**, *14*, 4909. [[CrossRef](#)]