fluids

Editorial

Recent Advances in Fluid Mechanics: Feature Papers, 2022

Mehrdad Massoudi

U. S. Department of Energy, National Energy Technology Laboratory (NETL), Pittsburgh, PA 15236, USA; mehrdad.massoudi@netl.doe.gov

This Special Issue is a collection of papers from some of the leading researchers discussing new findings or cutting-edge developments relating to all aspects of fluid mechanics.

Amongst the various forces acting on particles in a fluid, the Basset force, related to the fluid inertial effects, is one of the most difficult to study. Procopio and Giona [1] developed a modal expansion of the force acting on a micrometric particle. They show that the viscoelastic effects of the fluids studied induce the regularization of inertial memory, stemming from the finite propagation velocity. They derive an analytical expression for the fluid inertial kernel for a Maxwell fluid, and they also propose a general method to provide accurate approximations of this expression for complex fluids.

Hydropower plants have a high storage capacity and are capable of quick responses; as a result, they are increasingly being used to facilitate and integrate the intermittent energy from other renewable sources of energy, for example, wind and solar energy. At times, the operation of hydro turbines is limited by the formation of a Rotating Vortex Rope (RVR) in the draft tube. Arabnejad et al. [2] studied this phenomenon by using scale-resolving methods, namely, SST-SAS, wall-modeled LES (WMLES), and zonal WMLES. Their numerical simulations consider the effects of different scale-resolving methods on capturing flow, and the results indicate that for a small amount of vapor, cavitation induces broadband high-frequency fluctuations, and as the amount of cavitation increases, these fluctuations tend to have a dominant frequency different from that of the RVR.

There has been a tremendous increase in the applications of fractional calculus as a new and efficient mathematical tool for analyzing the properties of non-linear materials and relating the parameters in the models to experimental results. Lenzi et al. [3] studied the solutions of a generalized diffusion-like equation using a spatial and time-fractional derivative; in their equations, the presence of the non-local terms, related to reaction or adsorption–desorption processes, are also accounted for. They used the Green function approach to obtain solutions. Their study can help us to understand the different scenarios that can occur in connection with diffusion and anomalous diffusion processes.

The study of oscillating airfoils at moderate Reynolds numbers is a suitable candidate for testing the transition and modification needed in the standard Reynolds-Averaged Navier–Stokes (RANS) equations. Alberti et al. [4] used a high-order discontinuous Galerkin solver to study two-dimensional flapping foils at moderate Reynolds numbers when subject to different prescribed harmonic motions. Their simulations show an increase in the effectiveness in predicting loads, which is the case at low Strouhal numbers. Furthermore, their transition model seems to accurately predict wake topology, which is directly related to thrust/drag generation.

Methane pyrolysis appears to be among the new benign technologies for producing hydrogen with zero greenhouse gas emissions, and it is especially suitable for solar energy applications with high-temperature process heat. Msheik et al. [5] examined the possibility of using solar methane pyrolysis as a decarbonization process, producing both hydrogen gas and solid carbon with zero CO_2 emissions. They designed a novel hybrid solar/electric reactor at the PROMES-CNRS laboratory to handle the difficulties associated with direct normal irradiance (DNI). They also used Computational Fluid Dynamics (CFD)

check for updates

Citation: Massoudi, M. Recent Advances in Fluid Mechanics: Feature Papers, 2022. *Fluids* **2023**, *8*, 262. https://doi.org/10.3390/fluids8100262

Received: 19 September 2023
Accepted: 22 September 2023
Published: 26 September 2023

simulations and investigated the performance of this reactor under different operating conditions. The results of their numerical simulations agree with their experimental results, indicating the applicability of the proposed solar hybrid reactor design for efficient methane decomposition performance.

Vortex stretching and bursting are among the main causes of turbulence and the interactions at different scales of energy transfer; these are generally related to some of the terms in equations of motion, inertia, compression, diffusion, and dissipation. The vortex filament is also noticed in the Taylor–Green vortex, which is different from the simulation results based on the Navier–Stokes equations. Caltagirone [6] studied this problem using spectral analysis and by allowing the decay of kinetic energy to be a function of the wave number.

Waris and Lappa [7] studied the mixed buoyancy–Marangoni convection of a fluid over an inclined layer heated from below and unbounded from above. They used a thermographic visualization technique and took multiple temperature measurements at different points. Using a computer-based reconstruction of the spatial distribution of wavelengths, they show that this flow arrangement can develop interesting patterns, such as spatially localized cells, longitudinal wavy rolls, and finger-like structures.

Flow over rough surfaces occurs in many engineering applications and in nature (for example, flow inside pipes, around turbine blades, atmospheric boundary layers, etc.). The effects of roughness and its impact on flow have been studied extensively. Salomone et al. [8] studied flow over strips placed regularly along the mean stream. They used wall-modeled large-eddy simulations (WMLES) and improved delayed detached-eddy simulations (IDDES) (a hybrid method solving the Reynolds-averaged Navier–Stokes (RANS)) equations near the wall, while the large-eddy simulations (LES) were used in the core of the flow. They noticed that the modifications due to roughness can produce certain non-equilibrium effects, and memory of the upstream conditions also seem to be an important factor in the computational modeling of this flow.

Peristaltic flow occurs in many biological processes, such as digestion, which is an important component of any in silico model of the stomach. Obtaining an analytical solution that can be used for model verification is highly desirable. Liu et al. [9] used a smooth particle hydrodynamics (SPH) code (from CSIRO) and developed a model for use in the stomach wherein wall motion, buoyancy, acid secretion, and food breakdown are included. They used two different numerical methods, namely, the Finite Volume Method (FVM) and the SPH, to study this problem. The simulations show that both methods provide very good agreement with the analytical model.

Studies on synthetic jet actuators (SJAs) have shown a potential to delay flow separation over surfaces, offering applications in aerodynamics, where flow control can be achieved via injection through the external excitation of an enclosed cavity volume. In general, SJAs are smaller than an aircraft's wingspan, and as a result, they are used in an array form. Arafa et al. [10] experimentally studied the effect of the excitation frequency of SJAs on the mean jet velocity issuing from an array of circular orifices. They focused on the acoustic excitation characteristics of the actuator's cavity. They noticed that a large-aspect-ratio-cavity volume with multiple peaks can correspond to the standing-wave-mode shapes of the cavity.

The rheological responses of complex materials such as suspensions, dispersion, slurries, etc., are generally different from those of Newtonian fluids; some of these fluids exhibit non-linear effects such as yield stress, and among the most-used models with yield stress are the Bingham, Herschel–Bulkley, and Casson models. Calus et al. [11] studied the two-dimensional linear stability of a regularized Casson fluid flowing down an incline. Their results, which were obtained using the long-wave approximation method, indicate that the critical Reynolds number at which instability arises depends on the material parameters, the angle of inclination, and the prescribed inlet discharge. They also show that the flow of a Casson-type material over an inclined plane becomes increasingly stable as yield stress increases. This behavior, interestingly, is the opposite of that for a Bingham fluid.

Double-diffusive convection in a porous medium is a challenging topic of study due to the non-linearity inherent in the problem, where, in addition, the Darcy law can cause large-scale damping. Liu and Knobloch [12] studied thermal convection with salinity as a passive scalar; they used direct numerical simulations (DNS), and through the single-mode solutions, they reproduced the root-mean-square and mean temperature profiles of time-dependent states at high Rayleigh numbers. Their results show the potential for this single-mode approach to be applied to other flow configurations where coherent structures are dominant due to the presence of large-scale damping.

With the further requirement and demand for the use of renewable energy to produce electricity, steam turbine power plants are being operated at a low load, where it is possible for the steam turbine rear stages to absorb power from the turbine shaft; this can lead to the so-called "ventilation phenomenon". Mambro et al. [13] correlated the state of the steam within the rotor channel to the measurements obtained downstream of the blades for different ventilation regimes. In their case, the ventilation power was related to the drag force that acts on the moving blades. Their results indicate that the drag coefficient is highly correlated with the Reynolds number based on the reverse blade height.

Computational studies related to membrane system design have shown the effectiveness of performance measures, where, for example, fouling and flow unsteadiness can be induced via different spacer configurations. Heinz et al. [14] numerically studied the local mass distributions in membrane systems and showed that the collective interaction of operation conditions (OCs) can provide further insight into understanding the related problems in the advection–diffusion equation. Using a Fourier series model (FSM), they obtained the exact solutions of an advection–diffusion equation for a wide range of OCs.

Amongst the promising approaches to turbulence modeling, one can mention the two-equation turbulence models in the framework of Reynolds-averaged Navier–Stokes (RANS) equations. As pointed out by Heinz [15], the existing hybrid RANS-LES methods suffer from some inherent problems, which can be alleviated by using a generalization of the continuous eddy simulation (CES) methods. It is also shown that the minimal error methods associated with flows of incompressible fluids can be extended to stratified and compressible flows; this can provide valuable input for the design of consistent turbulence models for cases with significant modeling uncertainties.

The measurement of mass transfer intensity in bubbly flows is an important challenge in innovative bioreactor design. To acquire a better understanding of these multiphase flows, Computational Fluid Dynamics (CFD) approaches can be used to describe flows in the bioreactor loop. Starodumov et al. [16] presented the results they obtained when using a developed thermometry method to evaluate the key performance in a bioreactor, for example, the volumetric mass transfer coefficient, which is an important parameter in the design, operation, scaling-up, and optimization of bioreactors. They designed a mass-transfer apparatus for growing different microorganisms to study a jet bioreactor with the recirculation of liquid and gas phases of a given rheology system.

Bubble dynamics, including bubble formation and dissolution, significantly impact industrial applications, ranging from the production of beverages to foam-manufacturing processes; the rate of bubble expansion or contraction is one of the most important parameters affecting these processes. Maloth et al. [17] studied the motion and expansion of an isolated bubble due to mass transfer in a pool of a supersaturated gas–liquid solution. They numerically solved the advection–diffusion equation and examined the effects of gas–liquid solution parameters, such as the inertia, viscosity, surface tension, diffusion coefficient, system pressure, and solubility of the gas, on the solution. They noticed that surface tension and inertia do not significantly influence bubble expansion, whereas viscosity, pressure, diffusion, and solubility have a noticeable impact on bubble growth.

Recent experimental results indicate that rising gas columns can produce interesting oscillations. Gergely and Néda [18] numerically studied the convective flows of heated fluid columns in a gravitational field using a simplified 2D geometry. They used the FEniCS package to solve the coupled Navier–Stokes and heat equations. In their study, they examined a hydrodynamics computer simulation where, for simplicity, heated fluid columns are used instead of ascending Helium columns; this way, they were able to reproduce the experimentally observed data.

The flow of a bubble through a confining pore can be affected by the surface roughness and the geometry of the pore. Studies have shown that pore-scale interactions, in addition to the entrance critical pressure and a strong interaction of an isolated dispersed phase and pore geometry, can lead to additional pressure at the exit of the pore geometry. Ansari and Nobes [19] investigate the motion of an isolated bubble through different pore geometries; their simulations indicate that pore shape and surface roughness have a significant effect on the passage of the isolated phase. They were also able to detect the phase-pinning pressure, which can cause a delayed response in multiphase flows in the pore structures.

Dilute flows of gas–solid particles occur in various aspects of industry, such as pneumatic transport, fluidized beds, vertical risers, cyclones, flow-mixing devices, etc. To understand the interaction between a fluid and particles, it is important to know the forces acting on the particles. In the numerical simulation of such flows, the Lagrangian Particle-Tracking method is often used, where packets of individual particles are tracked, recognizing that the main forces acting on the particles are those of gravity and drag. Dodds et al. [20] performed a CFD analysis to study the effects of particles situated both perpendicular and parallel to the flow direction; their results show that the neighboring particles perpendicular to the flow seem to increase the drag force at close separation distances, whereas when entrained particles are co-aligned with the flow, the drag force seems to be reduced for close separation distances and increases as the distance increases.

The main role of the mitral valve (MV), which has an elliptical shape and is composed of an annulus and two leaflets, is to enable and regulate the appropriate flow of blood into the left ventricle (LV). Valve asymmetry presents a special challenge for modern cardiac surgery. Collia and Pedrizzetti [21] performed a systematic numerical study using a healthy ventricle and an ideal valve with varying degrees of valve asymmetry. As they indicate, their computational model should not be confused with an FSI model, since, in their approach, they do not include the elastic properties of the tissues. Their results can provide some important pre-surgical information as to which type of valve asymmetry can be used for correct valve repair/replacement.

Fluid/fluid interfaces and interfacial rheology are important areas of research for rheologists and modelers. These types of processes can occur in flows of foam and emulsion-based applications along with certain chemical processes such as liquid–liquid extraction, froth flotation, wastewater treatment, or tertiary oil recovery. Guzmán et al. [22] provide a review of this topic, focusing on the study of the fluid/fluid interfaces with dilatational stresses. The authors examine the available experimental and theoretical models for the dilatational rheology of fluid/fluid interfaces and discuss the effect of the non-linear character of dilatational deformation on the rheological response of these interfaces.

Finally, I would like to thank all the authors who contributed to this Special Issue. Without their contributions and without the help of qualified reviewers, it would not have been possible to organize this Special Issue. I am also grateful to all the anonymous reviewers for their help. I would like to extend a personal note of appreciation and gratitude to Ms. Wing Wang and the Editorial staff of the *Fluids* Office; without their help and assistance, *Fluids* could not publish high-quality papers in a short period of time.

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

References

1. Procopio, G.; Giona, M. Modal Representation of Inertial Effects in Fluid–Particle Interactions and the Regularity of the Memory Kernels. *Fluids* **2023**, *8*, 84. [CrossRef]
2. Arabnejad, M.H.; Nilsson, H.; Bensow, R.E. Investigation of Flow-Induced Instabilities in a Francis Turbine Operating in Non-Cavitating and Cavitating Part-Load Conditions. *Fluids* **2023**, *8*, 61. [CrossRef]
3. Lenzi, E.K.; Somer, A.; Zola, R.S.; da Silva, L.R.; Lenzi, M.K. A Generalized Diffusion Equation: Solutions and Anomalous Diffusion. *Fluids* **2023**, *8*, 34. [CrossRef]

4. Alberti, L.; Carnevali, E.; Crivellini, A. Assessment of a RANS Transition Model with Flapping Foils at Moderate Reynolds Numbers. *Fluids* **2023**, *8*, 23. [CrossRef]
5. Msheik, M.; Rodat, S.; Abanades, S. CFD Simulation of a Hybrid Solar/Electric Reactor for Hydrogen and Carbon Production from Methane Cracking. *Fluids* **2023**, *8*, 18. [CrossRef]
6. Caltagirone, J.-P. The Role of Inertia in the Onset of Turbulence in a Vortex Filament. *Fluids* **2023**, *8*, 16. [CrossRef]
7. Waris, W.; Lappa, M. Patterning Behavior of Hybrid Buoyancy-Marangoni Convection in Inclined Layers Heated from Below. *Fluids* **2023**, *8*, 12. [CrossRef]
8. Salomone, T.; Piomelli, U.; De Stefano, G. Wall-Modeled and Hybrid Large-Eddy Simulations of the Flow over Roughness Strips. *Fluids* **2023**, *8*, 10. [CrossRef]
9. Liu, X.; Harrison, S.M.; Cleary, P.W.; Fletcher, D.F. Evaluation of SPH and FVM Models of Kinematically Prescribed Peristalsis-like Flow in a Tube. *Fluids* **2023**, *8*, 6. [CrossRef]
10. Arafa, N.; Sullivan, P.E.; Ekmekci, A. Jet Velocity and Acoustic Excitation Characteristics of a Synthetic Jet Actuator. *Fluids* **2022**, *7*, 387. [CrossRef]
11. Calusi, B.; Farina, A.; Fusi, L.; Palade, L.I. Stability of a Regularized Casson Flow down an Incline: Comparison with the Bingham Case. *Fluids* **2022**, *7*, 380. [CrossRef]
12. Liu, C.; Knobloch, E. Single-Mode Solutions for Convection and Double-Diffusive Convection in Porous Media. *Fluids* **2022**, *7*, 373. [CrossRef]
13. Mambro, A.; Congiu, F.; Galloni, E.; Lanni, D. Blade Drag Resistance in Windage Operating of Low Pressure Steam Turbines. *Fluids* **2022**, *7*, 372. [CrossRef]
14. Heinz, S.; Heinz, J.; Brant, J.A. Mass Transport in Membrane Systems: Flow Regime Identification by Fourier Analysis. *Fluids* **2022**, *7*, 369. [CrossRef]
15. Heinz, S. From Two-Equation Turbulence Models to Minimal Error Resolving Simulation Methods for Complex Turbulent Flows. *Fluids* **2022**, *7*, 368. [CrossRef]
16. Starodumov, I.; Nizovtseva, I.; Lezhnin, S.; Vikharev, S.; Svitich, V.; Mikushin, P.; Alexandrov, D.; Kuznetsov, N.; Chernushkin, D. Measurement of Mass Transfer Intensity in Gas–Liquid Medium of Bioreactor Circuit Using the Thermometry Method. *Fluids* **2022**, *7*, 366. [CrossRef]
17. Maloth, R.K.N.; Khayat, R.E.; DeGroot, C.T. Bubble Growth in Supersaturated Liquids. *Fluids* **2022**, *7*, 365. [CrossRef]
18. Gergely, A.; Néda, Z. Computational Fluid Dynamics Approach for Oscillating and Interacting Convective Flows. *Fluids* **2022**, *7*, 339. [CrossRef]
19. Ansari, S.; Nobes, D.S. Effect of the Pore Geometry on the Driving Pressure across a Bubble Penetrating a Single Pore. *Fluids* **2022**, *7*, 333. [CrossRef]
20. Dodds, D.; Sarhan, A.A.R.; Naser, J. CFD Investigation into the Effects of Surrounding Particle Location on the Drag Coefficient. *Fluids* **2022**, *7*, 331. [CrossRef]
21. Collia, D.; Pedrizzetti, G. The Influence of Mitral Valve Asymmetry for an Improved Choice of Valve Repair or Replacement. *Fluids* **2022**, *7*, 293. [CrossRef]
22. Guzmán, E.; Maestro, A.; Carbone, C.; Ortega, F.; Rubio, R.G. Dilational Rheology of Fluid/Fluid Interfaces: Foundations and Tools. *Fluids* **2022**, *7*, 335. [CrossRef]