energies

MDPI

Editorial

On CFD-Assisted Research and Design in Engineering

Dmitry Eskin

Skolkovo Institute of Science and Technology, 121205 Moscow, Russia; d.eskin@skoltech.ru

At present, computational fluid dynamics (CFD) is an inherent component of the development procedure of a majority of technological processes involving fluid flows and/or heat and mass transfers. Practicing engineers and investigators employ different commercial CFD software, open-source codes and even develop their own computational codes (in house) for solving tasks, requiring accounting for nonstandard effects.

The most popular commercial CFD codes are ANSYS Fluent and STARCCM+. Although these products compete on a global market, the latter is so remarkably expansive and continuously expanding that sales of both software packages are constantly growing. In general, both codes possess similar capabilities and are based on, more or less, the same modelling approaches. Since each code has its own advantages and disadvantages, an experienced user can choose the optimal package necessary for the solution of a certain task. Additionally, both codes allow users to incorporate their own models describing specific physics or chemistry processes through so-called user-defined functions (UDFs). Note, a number of other commercial codes (e.g., ANSYS CFX) possess similar capabilities.

In recent years, *Energies* has published a considerable number of manuscripts dedicated to solving CFD problems, encompassing different approaches applied to solving various tasks. In the present Editorial, we briefly consider five works published within a relatively short period of time; these works are not directly related to each other, but present interesting examples of CFD-assisted engineering research.

Su et al. [1] delivered a paper considering the motion of solid particles in a centrifugal slurry transport pump for deep-ocean mining. Such a device is capable of lifting a slurry from big depths (~5000–10,000 m) to the surface. The slurry is usually composed of a mixture of valuable minerals and waste materials. Multistage centrifugal pumps are regularly employed for this purpose, with a stage of the studied pump consisting of a bowl diffuser and impeller. Due to a relatively small impeller diameter, restricted by a diameter of a pipe housing, a significant number of stages is necessary to create a head needed for lifting slurry from a significant depth to the surface. The authors conducted CFD studies of a two-stage vertical pump, designed it and experimentally investigated in their lab employing relatively low-volume fractions (<10%) of large-sized particles (>10 mm). A numerical method based on CFD–DEM coupling was employed for the analysis. ANSYS Fluent was used for the CFD simulations and a discrete element method (DEM)-based software EDEM for the DEM computations. The DEM technique allows for the accounting of interactions between particles and pump surfaces and with each other. The computed pump characteristics turned out to be in a good agreement with the measured ones. Su et al. [1] presented colorful pictures illustrating particle distributions over the impellers and diffusers, and identified areas of high and low concentrations of solids. The described work is a reasonably good example of CFD application for solving an engineering task. Conceivably, such computations could be useful for erosion analyses and the development of novel stage designs characterized by the reduced erosion of pump elements.

Another interesting paper on CFD application to pump design was recently published by Casoli et al. [2], who performed an analysis of the use of textured surfaces in hydraulic pumps. This subject is rather rarely studied and related to the area of tribology. Hydrodynamic effects, accompanying sliding metallic surfaces in hydraulic machines, contribute to

check for updates

Citation: Eskin, D. On CFD-Assisted Research and Design in Engineering. *Energies* **2022**, *15*, 9233. https://doi.org/10.3390/en15239233

Received: 22 August 2022
Accepted: 26 October 2022
Published: 6 December 2022

the support of the heavy loads experienced by these surfaces. The authors studied an effect of geometrical parameters of dimples on metallic surfaces on tribological properties of surface coupling, employing the ANSYS CFX commercial CFD code for the computations. In their simulations, the authors also accounted for cavitation, which definitely plays a negative role in coupling performance. Moreover, the formation and collapse of cavitation bubbles were attentively investigated. An important practical result was a demonstration of pressure distribution over a textured surface for different dimple configurations. The authors identified these configurations and flow regimes when the cumulative force, contributing to the disjoining of the sliding surfaces, caused a reduction in the contact load. Note, although this work looked interesting and useful for engineering applications, Casoli et al. [2] did not discuss the effect of possible textured surface configuration changes, caused by cavitation, on surface coupling. These changes may be significant because the dimples, forming the texture, can be quickly destroyed by cavitation.

Another paper (Zubrowska-Sudol et al. [3]) deserving attention is related to relatively novel energy sources, with the topic being the intensification of methane production using a hydrodynamic disintegrator used for agricultural substrate sludge pretreatment. It is worth stating that the current concept of the development of modern energetics assumes a reduction in the consumption of fossil fuels. From this standpoint, biogas produced from agricultural wastes plays a noticeable role as an energy source in Europe, a role that is expected to grow. The authors refer to studies showing that an efficient pretreatment of substrate supplied to a fermentation chamber leads to a significant increase in methane output. Zubrowska-Sudol et al. [3] used CFD modelling for the optimal designing an efficient disintegrator. The existing device was modified by adding shredding knives on the top of the disintegrator rotor, as well as ribs on the rotor's periphery set to rapidly decelerate the radial flow in order to enhance the disintegration intensity. A flow pattern in a device of the newly suggested design was studied by the ANSYS CFX commercial software conducting transient computations. Despite calculations being performed for water instead of sludge, characterized by a complex rheology, multiple cavitation zones, as well as regions of enhanced pressure, were revealed in a computational domain. The computed flow patterns allowed for the confirmation of a better disintegration performance of the new device design compared to the original. The disintegrator of the new design was manufactured and meticulously tested and it was demonstrated that the pretreatment of the maize silage provided a significant increase (up to 34%) in the methane output. This briefly described project is a convincing example of efficient CFD employment for the optimum design of industrial equipment.

Considering that bioreactors play a significant role in different modern technologies, it makes sense to pay attention to one more relevant CFD application example. Hoehne and Mamedov [4] built a CFD model for a full-scale oxidation ditch (OD). The biological wastewater treatment is the main part of a waste water treatment plant for removing biodegradable organic wastes and suspended solids. An OD is a modified process of the biological treatment of activated sludge characterized by long solid retention times for the removal of biodegradable organics. Aeration plays a major role in the OD process, since it delivers oxygen to a biodegradable material and supports its mixing. The authors employed the commercial CFD code ANSYS CFX to simulate the full OD; the corresponding setup dimensions were impressive ($75 \times 18 \times 6$ m). The sensitivity of the employed unstructured meshing was tested by using three different meshes, with the finest one consisting of 20.47 mln elements. The problem complexity was additionally enhanced by the presence of moving elements (agitators) of the chosen design. Hoehne and Mamedov [4] reported that a transient computation of 225 s of the OD operation required approximately 2 weeks of computational time. As a result of the calculations, detailed information on flow structure was obtained. A number of conclusions regarding an enhancement of the OD efficiency were formulated by the authors. This work, to a significant extent focused on computational aspects, can be considered to be an example of a deliberate approach to large-scale CFD computations aimed at a complex setup design improvement.

Let us also briefly discuss a paper dedicated to a problem completely different from the previously considered works, but representing an important example of CFD application. Kim et al. [5] simulated the discharge of a hydrogen gas from a high-pressure vessel into the atmosphere using the ANSYS Fluent CFD code. This problem is well known and important for the analysis of extremely hazardous situations if a discharge was to occur due to either vessel damage or a failure of one of the valves. The authors considered that hydrogen is stored under a high pressure (up to 70 MPa) and meticulously studied different aspects of the discharge process with several different thermodynamic models for the description of real gas behavior in the vessel being studied. Additionally, in contrast to previous studies known in the open literature, the authors accounted for the effect of turbulence on the discharge by testing several different turbulence models for this purpose. Numerous computations of the hydrogen discharge process were conducted. The calculated pressures and temperatures in the vessel vs. the discharge time were compared with the measured data, and the result analysis showed that a combination of the Redlich–Kwong equation for a real gas and the shear stress transport (SST) turbulence model provided the best computational accuracy. It is worth stating that the data, presented by Kim et al. [5], showed that none of the tried combinations of thermodynamic and turbulence models actually failed. This observation can be explained as follows: (1) hydrogen behavior under a high pressure is rather reliably described by several known equations of state; (2) a discharge of a high-velocity flow from an orifice is primarily determined by a rarefaction wave structure in an underexpanded jet and, therefore, turbulence does not play a primary role in this process. Nevertheless, the research by Kim et al. [5] represents a good example of a wise CFD technique application for solving a practical engineering problem.

Thus, through the five CFD-related paper examples, recently published in *Energies*, we showed how the authors actively employed CFD for solving various engineering tasks, demonstrating good subject knowledge and inventiveness. Considering the increasing role of CFD in complex process design, there are well-grounded expectations that a number of CFD-related submissions to *Energies* could rapidly grow along with an enhancement in their quality.

Funding: This research received no external funding.

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

References

1. Su, X.; Tang, Z.; Li, Y.; Zhu, Z.; Mianowicz, K.; Balaz, P. Research of Particle Motion in a Two-Stage Slurry Transport Pump for Deep-Ocean Mining by the CFD-DEM Method. *Energies* **2020**, *13*, 6711. [CrossRef]
2. Casoli, P.; Scolari, F.; Rundo, M.; Lettini, A.; Rigosi, M. CFD Analyses of Textured Surfaces for Tribological Improvements in Hydraulic Pumps. *Energies* **2020**, *13*, 5799. [CrossRef]
3. Zubrowska-Sudol, M.; Dzido, A.; Garlicka, A.; Krawczyk, P.; Stępień, M.; Umiejewska, K.; Walczak, J.; Wołowicz, M.; Sytek-Szmeichel, K. Innovative Hydrodynamic Disintegrator Adjusted to Agricultural Substrates Pre-treatment Aimed at Methane Production Intensification—CFD Modelling and Batch Tests. *Energies* **2020**, *13*, 4256. [CrossRef]
4. Höhne, T.; Mamedov, T. CFD Simulation of Aeration and Mixing Processes in a Full-Scale Oxidation Ditch. *Energies* **2020**, *13*, 1633. [CrossRef]
5. Kim, M.-S.; Ryu, J.-H.; Oh, S.-J.; Yang, J.-H.; Choi, S.-W. Numerical Investigation on Influence of Gas and Turbulence Model for Type III Hydrogen Tank under Discharge Condition. *Energies* **2020**, *13*, 6432. [CrossRef]