processes

Editorial

Special Issue on "CFD Based Researches and Applications for Fluid Machinery and Fluid Device"

Jin-Hyuk Kim [1,2,*], Sung-Min Kim [3], Minsuk Choi [4], Lei Tan [5], Bin Huang [6] and Ji Pei [7]

1 Clean Energy R&D Department, Korea Institute of Industrial Technology, Cheonan 31056, Korea
2 Industrial Technology (Green Process and Energy System Engineering), Korea University of Science and Technology, Daejeon 34113, Korea
3 School of Mechanical Engineering, Sungkyunkwan University, Suwon 16419, Korea; smkim@skku.edu
4 Department of Mechanical Engineering, Myongji University, Yongin 17058, Korea; mchoi@mju.ac.kr
5 Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China; tanlei@mail.tsinghua.edu.cn
6 Institute of Ocean Engineering and Technology, Ocean College, Zhejiang University, Hangzhou 310058, China; binhuang@zju.edu.cn
7 National Research Center of Pumps, Jiangsu University, Zhenjiang 212013, China; jpei@ujs.edu.cn
* Correspondence: jinhyuk@kitech.re.kr

The demand for computational fluid dynamics (CFD)-based numerical techniques is increasing rapidly with the development of the computing power system. These advanced CFD techniques are applicable to various issues in the industrial engineering fields and especially contributing considerably to the design of fluid machinery and fluid devices, which have very complicated unsteady flow phenomena and physics. In other words, with the rapid development of CFD techniques, the performances of fluid machinery and fluid devices with complicated unsteady flows have been enhanced significantly. In addition, many persistently troublesome problems of fluid machinery and fluid devices such as flow instability, rotor–stator interaction, surging, cavitation, vibration, and noise are solved clearly using advanced CFD techniques.

The Special Issue on "CFD-Based Researches and Applications for Fluid Machinery and Fluid Device" in Processes deals with topics related to CFD techniques and applications in various fluid machines and devices. Specifically, the 31 papers published in this Special Issue focus on the advancement in the detailed internal flow analyses and methodologies for designing various fluid machines and devices, as the following summaries indicate. The influence of a low-pressure environment on the aerodynamic and noise characteristics of a centrifugal fan was studied numerically and experimentally by Zhang et al. [1]. Luo et al. [2] analyzed systematically the dynamic characteristics of mechanical seals under different fault conditions. Yu et al. [3] numerically studied the influence of step casing on unsteady cavitating flows and instabilities in inducers with equal and varying pitches. A methodology to improve the aerodynamic design with low cost and high accuracy for a 1–1/2 axial compressor was presented by Xie et al. [4]. Wang et al. [5] conducted the multi-condition optimization to enhance the cavitation performance of a double-suction centrifugal pump based on an artificial neural network (ANN) and nondominated sorting genetic algorithm II (NSGA-II). Zhang et al. [6] proposed an improved aerodynamic optimization method for designing effectively a low Reynolds number cascade. The design optimization of a two-vane pump for wastewater treatment using machine-learning-based surrogate modeling was carried out by Ma et al. [7]. Wang et al. [8] analyzed numerically the axial vortex characteristics in a centrifugal pump as a turbine with an S-blade impeller. Li et al. [9] studied numerically and experimentally the transient characteristics of a centrifugal pump during the startup period with assisted valve. The thermal performance with the geometric parametrization of T-shaped obstacles in a solar air heater was performed by Ahn and Kim [10].

Citation: Kim, J.-H.; Kim, S.-M.; Choi, M.; Tan, L.; Huang, B.; Pei, J. Special Issue on "CFD Based Researches and Applications for Fluid Machinery and Fluid Device". *Processes* **2021**, *9*, 1137. https://doi.org/10.3390/pr9071137

Received: 29 June 2021
Accepted: 29 June 2021
Published: 30 June 2021

On the other hand, the effect of rotor spacing and duct diffusion angle on the aerodynamic performances of a counter-rotating ducted fan in a hover mode was analyzed numerically by Kim et al. [11]. Lei et al. [12] applied the CFD method to analyze the aerodynamic performance of an octorotor small unmanned aerial vehicle with different rotor spacing in hover. Shrestha and Choi [13] proposed a CFD-based shape design optimization process to improve the flow uniformity in the fixed flow passages of a Francis hydro turbine model. The influence analyses of the blade outlet angle on the flow and pressure pulsation characteristics in a centrifugal fan were carried out by Ding et al. [14]. Rui et al. [15] performed experimental and numerical studies to investigate the effect of the radius of a volute tongue on the aerodynamic and aeroacoustic characteristics of a Sirocco fan. The effect and mechanism of the triple hole on the film-cooling performance based on large eddy simulation (LES) were identified by Baek and Ahn [16]. Song et al. [17] verified the influence of tip clearance on the flow characteristics of an axial compressor through the CFD technique. Hur et al. [18] numerically investigated the effect of clearance and cavity geometries on the leakage performance of a stepped labyrinth seal. The effect of root clearance on the mechanical energy dissipation of an axial flow pump based on entropy production was analyzed by Li et al. [19]. Gao et al. [20] explored the hydraulic performance with different blade wrap angles of an impeller in an open-design vortex pump.

Moreover, the numerical and experimental studies on the waviness mechanical seal of a reactor coolant pump were conducted by Feng et al. [21]. Benišek et al. [22] suggested a new design of the reversible axial jet fan impeller with symmetrical and adjustable blades. Stelmach et al. [23] confirmed the influence of hydrodynamic changes in a system with a pitched blade turbine on mixing power using a particle image velocimetry (PIV) method. Wang et al. [24] optimized the shapes of the impeller and diffuser of a mixed-flow pump using the inverse design method and CFD analysis. Rakibuzzaman et al. [25] designed numerically a new prototype propeller-type tubular turbine utilizing discharge water from a fish farm, and its performance was verified experimentally. The characteristic implicit method based on the upwind differencing and implicit finite difference scheme to solve the mixed free-surface-pressurized flow in a hydropower station was suggested by Wang et al. [26]. Park et al. [27] carried out the multi-objective numerical optimization to simultaneously enhance the heat transfer efficiency and reduce the pressure loss of a wavy microchannel heat sink. Chen et al. [28] investigated numerically the dynamic stresses of the runner during start-up in the turbine mode of a pump turbine. Portal-Porras et al. [29] tested the accuracy of the cell-set model applied on vane-type sub-boundary layer vortex generators by using CFD techniques. Shamsuddeen et al. [30] suggested a new inducer-type guide vane to reduce the hydraulic losses at the inter-stage flow passage of a multistage centrifugal pump. The effect of micro-tab on the lift enhancement of airfoil S-809 with trailing edge flap in wind turbine blades was analyzed numerically by Ye et al. [31].

Finally, as the guest editors of this Special Issue, we would like to especially thank the Section Managing Editor, Ms. Shirley Wang, for organizing and helping the Special Issue of *Processes*. We are also thankful to all the reviewers for their valuable comments for improving the quality of papers published in this Special Issue. In addition, this valuable Special Issue is available at https://www.mdpi.com/journal/processes/special_issues/CFD_Fluid_Device.

Author Contributions: Writing—original draft preparation, J.-H.K.; writing—review and editing, J.-H.K.; S.-M.K.; M.C.; L.T.; B.H., and J.P. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Zhang, X.; Zhang, Y.; Lu, C. Flow and noise characteristics of centrifugal fan in low pressure environment. *Processes* **2020**, *8*, 985. [CrossRef]

2. Luo, Y.; Fan, Y.; Han, Y.; Zhang, W.; Acheaw, E. Research on the dynamic characteristics of mechanical seal under different extrusion fault degrees. *Processes* **2020**, *8*, 1057. [CrossRef]
3. Yu, L.; Zhang, H.; Chen, H.; Zuo, Z.; Liu, S. Numerical study on the influence of step casing on cavitating flows and instabilities in inducers with equal and varying pitches. *Processes* **2020**, *8*, 1103. [CrossRef]
4. Xie, H.; Song, M.; Yang, B. Research on the application of partial similarity for a 1-1/2 axial compressor. *Processes* **2020**, *8*, 1121. [CrossRef]
5. Wang, W.; Li, Y.; Osman, M.K.; Yuan, S.; Zhang, B.; Liu, J. Multi-condition optimization of cavitation performance on a double-suction centrifugal pump based on ANN and NSGA-II. *Processes* **2020**, *8*, 1124. [CrossRef]
6. Zhang, S.; Yang, B.; Xie, H.; Song, M. Applications of an improved aerodynamic optimization method on a low Reynolds number cascade. *Processes* **2020**, *8*, 1150. [CrossRef]
7. Ma, S.-B.; Kim, S.; Kim, J.-H. Optimization design of a two-vane pump for wastewater treatment using machine-learning-based surrogate modeling. *Processes* **2020**, *8*, 1170. [CrossRef]
8. Wang, X.; Kuang, K.; Wu, Z.; Yang, J. Numerical simulation of axial vortex in a centrifugal pump as turbine with S-blade impeller. *Processes* **2020**, *8*, 1192. [CrossRef]
9. Li, Q.; Ma, X.; Wu, P.; Yang, S.; Huang, B.; Wu, D. Study on the transient characteristics of the centrifugal pump during the startup period with assisted valve. *Processes* **2020**, *8*, 1241. [CrossRef]
10. Ahn, S.-Y.; Kim, K.-Y. Thermal performance of T-shaped obstacles in a solar air heater. *Processes* **2020**, *8*, 1305. [CrossRef]
11. Kim, W.-Y.; Senguttuvan, S.; Kim, S.-M. Effect of rotor spacing and duct diffusion angle on the aerodynamic performances of a counter-rotating ducted fan in hover mode. *Processes* **2020**, *8*, 1338. [CrossRef]
12. Lei, Y.; Huang, Y.; Wang, H. Aerodynamic performance of an octorotor SUAV with different rotor spacing in hover. *Processes* **2020**, *8*, 1364. [CrossRef]
13. Shrestha, U.; Choi, Y.-D. A CFD-based shape design optimization process of fixed flow passages in a Francis hydro turbine. *Processes* **2020**, *8*, 1392. [CrossRef]
14. Ding, H.; Chang, T.; Lin, F. The influence of the blade outlet angle on the flow field and pressure pulsation in a centrifugal fan. *Processes* **2020**, *8*, 1422. [CrossRef]
15. Rui, X.; Lin, L.; Wang, J.; Ye, X.; He, H.; Zhang, W.; Zhu, Z. Experimental and comparative RANS/URANS investigations on the effect of radius of volute tongue on the aerodynamics and aeroacoustics of a Sirocco fan. *Processes* **2020**, *8*, 1442. [CrossRef]
16. Baek, S.I.; Ahn, J. Large eddy simulation of film cooling with triple holes: Injectant behavior and adiabatic film-cooling effectiveness. *Processes* **2020**, *8*, 1443. [CrossRef]
17. Song, M.; Xie, H.; Yang, B.; Zhang, S. Influence of tip clearance on flow characteristics of axial compressor. *Processes* **2020**, *8*, 1445. [CrossRef]
18. Hur, M.S.; Lee, S.I.; Moon, S.W.; Kim, T.S.; Kwak, J.S.; Kim, D.H.; Jung, I.Y. Effect of clearance and cavity geometries on leakage performance of a stepped labyrinth seal. *Processes* **2020**, *8*, 1496. [CrossRef]
19. Li, Y.; Zheng, Y.; Meng, F.; Osman, M.K. The effect of root clearance on mechanical energy dissipation for axial flow pump device based on entropy production. *Processes* **2020**, *8*, 1506. [CrossRef]
20. Gao, X.; Zhao, T.; Shi, W.; Zhang, D.; Shi, Y.; Zhou, L.; Chang, H. Numerical investigation of an open-design vortex pump with different blade wrap angles of impeller. *Processes* **2020**, *8*, 1601. [CrossRef]
21. Feng, X.; Su, W.; Ma, Y.; Wang, L.; Tan, H. Numerical and experimental study on waviness mechanical seal of reactor coolant pump. *Processes* **2020**, *8*, 1611. [CrossRef]
22. Benišek, M.H.; Čantrak, Đ.S.; Ilić, D.B.; Janković, N.Z. New design of the reversible jet fan. *Processes* **2020**, *8*, 1671. [CrossRef]
23. Stelmach, J.; Kuncewicz, C.; Szufa, S.; Jirout, T.; Rieger, F. The influence of hydrodynamic changes in a system with a pitched blade turbine on mixing power. *Processes* **2021**, *9*, 68. [CrossRef]
24. Wang, M.; Li, Y.; Yuan, J.; Osman, F.K. Matching optimization of a mixed flow pump impeller and diffuser based on the inverse design method. *Processes* **2021**, *9*, 260. [CrossRef]
25. Rakibuzzaman, M.; Suh, S.-H.; Kim, H.-H.; Ryu, Y.; Kim, K.Y. Development of a hydropower turbine using seawater from a fish farm. *Processes* **2021**, *9*, 266. [CrossRef]
26. Wang, X.; Fan, H.; Liu, B. Optimization control on the mixed free-surface-pressurized flow in a hydropower station. *Processes* **2021**, *9*, 320. [CrossRef]
27. Park, M.-C.; Ma, S.-B.; Kim, K.-Y. Optimization of a wavy microchannel heat sink with grooves. *Processes* **2021**, *9*, 373. [CrossRef]
28. Chen, F.; Bi, H.; Ahn, S.-H.; Mao, Z.; Luo, Y.; Wang, Z. Investigation on dynamic stresses of pump-turbine runner during start up in turbine mode. *Processes* **2021**, *9*, 499. [CrossRef]
29. Portal-Porras, K.; Fernandez-Gamiz, U.; Aramendia, I.; Teso-Fz-Betoño, D.; Zulueta, E. Testing the accuracy of the cell-set model applied on vane-type sub-boundary layer vortex generators. *Processes* **2021**, *9*, 503. [CrossRef]
30. Shamsuddeen, M.M.; Ma, S.-B.; Kim, S.; Yoon, J.-H.; Lee, K.-H.; Jung, C.; Kim, J.-H. Effect of an inducer-type guide vane on hydraulic losses at the inter-stage flow passage of a multistage centrifugal pump. *Processes* **2021**, *9*, 526. [CrossRef]
31. Ye, J.; Salem, S.; Wang, J.; Wang, Y.; Du, Z.; Wang, W. Effects of micro-tab on the lift enhancement of airfoil S-809 with trailing-edge flap. *Processes* **2021**, *9*, 547. [CrossRef]