Hindawi Publishing Corporation
Advances in Mechanical Engineering
Article ID 645351

Editorial

Numerical Simulation of Fluid Flow and Heat Transfer Processes 2014

Bo Yu,[1] Shuyu Sun,[2] Jinjia Wei,[3] Dongliang Sun,[4] Yi Wang,[1] and Weihua Cai[5]

[1] *National Engineering Laboratory for Pipeline Safety, Beijing Key Laboratory of Urban Oil and Gas Distribution Technology, China University of Petroleum, Beijing 102249, China*
[2] *Computational Transport Phenomena Laboratory, Division of Physical Science and Engineering, King Abdullah University of Science and Technology, Thuwal 23955-6900, Saudi Arabia*
[3] *State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China*
[4] *Department of Renewable Energy, North China Electric Power University, No. 2 Bei Nong Lu, Zhu Xin Zhuang, Changping District, Beijing 102206, China*
[5] *School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China*

Correspondence should be addressed to Bo Yu; yubobox@vip.163.com

Received 8 September 2014; Accepted 8 September 2014

Numerical simulation of fluid flow and heat transfer processes has been an important vehicle for scientific researches and applications. This annual special issue focuses on the up-to-date progresses in this field. We received 88 active submissions from China (including Taiwan), United States of America, Canada, Brazil, Russia, Romania, Turkey, Saudi Arabia, Iraq, India, Japan, Republic of Korea, Malaysia, and Egypt and finally accepted 46 research articles to publish them in the special issue after the strict peer review processing. The topics cover both the fundamental researches (30 articles) and applicable researches (16 articles), which will be introduced in detail.

The fundamental research articles are mainly distributed in the following 4 topics: turbulent flow, heat/mass transfer, porous medium flow, and numerical methods.

In the field of heat/mass transfer, 9 articles have been published. The article "*Investigation of regularities of heat and mass transfer and phase transitions during water droplets motion through high-temperature gases*" by R. S. Volkov et al. is a mechanism study of water evaporation at the temperature above 1000 K. The authors establish water droplet rate, single out temperature values of gas-vapor mixture in small neighborhood, and study the phase transition regularity in heat and mass transfer of water evaporation processes. In the article "*Numerical study on deformation and interior flow of a droplet suspended in viscous liquid under steady electric fields*" by Z. Wang et al., the rate of deformation and internal flow of the single droplet are simulated through coupling with hydrodynamics and electrostatics. When fluid electric properties change, the charge distribution at the interface is various, which leads to the droplet different deformation shapes. The article "*Numerical study on the thermal performance of a shell and tube phase change heat storage unit during melting process*" by W. Li and C. Kong develops a mathematical model involving heat transfer fluids and phase change materials to analyze the thermal performance of the heat storage unit. The results indicate that the air inlet velocity has great effect on the air outlet temperature and heat transfer rate, and the water inlet velocity has little effect on the water outlet temperature. According to these results, the authors declare that their work can provide a reference for designing phase change heat storage system. The article "*Experimental and numerical investigations of heat transfer characteristics for impinging swirl flow*" by S. D. Salman et al. reports modelling studies to investigate the effect of the swirl intensity on the heat transfer characteristics of conventional and swirl impingement air jets at a constant nozzle-to-plate distance. They find that the radial uniformity of Nusselt number (Nu) of swirl impingement air jets (SIJ) depends on the swirl intensity and the air Reynolds number and proper fitted SIJ can provide much more uniform local heat transfer distribution on the surface. In the article "*Using*

of multiwall carbon nanotube based nanofluid in the heat pipe to get better thermal performance" by Y. Bakhshan et al., thermal performance of a cylindrical heat pipe is investigated numerically. The MWCNT based nanofluid has lower thermal resistance, higher heat transfer coefficient, and lower temperature difference between evaporator and condenser sections, so it has good thermal specifications as a working fluid for using in heat pipes. In the article "*Heat transfer and thermal deformation characteristics of liquid-cooled laser mirror*" by P. Hu et al., a coupled finite volume-element method is developed to simulate the transient thermal deformation of water-cooled mirror. The authors find that fluid flow not only affects the magnitude of temperature rise and thermal deformation but also affects the distribution of temperature and thermal deformation which increase significantly in the first seconds and gradually become steady state in the subsequent time. In the article "*Numerical study on the effect of tube rows on the heat transfer characteristic of dimpled fin*" by X. Wu et al., the authors investigate the effects of tube-row number on the air side heat transfer characteristics of dimpled fin for velocity ranging from 1 to 3 m/s and find that the dimpled arrangement can change the mainstream direction, increase the disturbance, and enhance the heat transfer. They conclude that the performance of two-row tube is the best. In the article "*Prediction of mass transfer time relaxation parameter for boiling simulation on the shell-side of LNG spiral wound heat exchanger*" by Z.-Y. Wu et al., the authors propose an approach to predict mass transfer time relaxation parameter for boiling simulation on the shell-side of LNG spiral wound heat exchanger. Their method may be well applicable to various phase change simulations. The article "*Effect of one-target focus type on hydrodynamic characteristics of tower solar cavity receiver*" by Y. Hao et al. presents the study on heat flux of the vapor-liquid two-phase or single-phase superheated steam in the parallel heated panel bundles of the solar cavity receiver. The results show that for the evaporation panels the flow distribution can synchronize with that of low heat flux, while for the superheater sections the flow distribution decreases with the increasing heat flux. This desynchrony may give rise to stagnation or backflow of the working fluid and lead to the panels burst or erosion due to the local overheating in some extreme situation.

In the field of turbulent flow, 8 articles have been published. The article "*CFD numerical simulation of the complex turbulent flow field in an axial-flow water pump*" by W.-Y. Li et al. presents a numerical simulation procedure based on computational fluid dynamics (CFD) to obtain a thorough recognition of the characteristics of the complex turbulent flow field in the pump which is extremely difficult to be measured using the up-to-date experimental techniques. The established model can be of reference value in further optimal design of the axial-flow pump. The article "*Motion of passive scalar by elasticity-induced instability in curved microchannel*" by X.-B. Li et al. presents a direct numerical simulation (DNS) study on the elasticity-induced irregular flow, passive mixing, and scalar evolution in the curvilinear microchannel. As the elastic effect exceeds the critical value, the flow tends to a chaotic state, while the evolution of scalar gets strong and fast. Main direction of molecular stretching is perpendicular to the main direction of flow field deformation. In the article "*Direct numerical simulation and visualization of biswirling jets*" by J. Yan et al., the visualization of vortex-vortex interaction between the biswirling jets is accomplished by using direct numerical simulation. The evolution of vortex structures of the biswirling jets is found rather complicated. The article "*Numerical simulation on forced convective condensation of steam upward flow in a vertical pipe*" by G.-D. Qiu et al. presents a transient three-dimensional volume of fluid (VOF) simulation on condensation of upward flow of wet steam inside a 12 mm inner-diameter vertical pipe considering gravity and surface tension. Obvious local eddies are observed in the slug flow and the churn flow but not observed in other flow patterns that the streamlines are almost parallel to the flow direction. In the article "*Direct numerical simulation of a free particle laden round jet with point-particles: turbulence modulation*" by D. Li et al., two-way coupling between the particle and the fluid combining the point force method is proposed. Some findings will further refine the understanding of intricate mechanism of small-scale turbulence modulation for a spatially evolving free round jet. In the article "*Study on the drag reducing channel fluids by experiments and DNS using Giesekus model*" by W. Gu et al., both experimental and numerical studies are simultaneously performed for fully developed water and surfactant solution channel flow. Although large turbulent drag reduction appears in DNS, Giesekus model is found to have some limitations in describing the fluid characteristics and viscoelasticity of the surfactant solution. The article "*Rotating turbulent flow simulation with LES and Vreman subgrid-scale models in complex geometries*" by T. Guo et al. and the article "*Characteristics of large-scale structures in supersonic planar mixing layer with finite thickness*" by H. Zhang et al. are both studies using the large eddy simulation (LES) method. The former one shows the flowing character in Francis turbine passage under the small opening condition and the tangential velocity component of fluid has absolute superiority under small opening condition. The latter one investigates the fine structure of the supersonic planar mixing layer with finite thickness. The authors find the mixing layer is thicker with higher pressure of the high speed side and the development of the vortices can be affected through increasing the temperature to change the compression.

In the field of porous medium flow, 8 articles have been published. The article "*Coupling two-phase fluid flow with two-phase Darcy flow in anisotropic porous media*" by J. Chen et al. reports a novel numerical method of coupling two-phase flows in a free fluid region and a porous medium region. The flow in the free fluid region is modeled by coupled Cahn-Hilliard and Navier-Stokes equations while the flow in porous medium region is modeled by the two-phase Darcy's law. The two regions are coupled by the Robin-Robin domain decomposition method with the generalized Beavers-Joseph-Saffman condition on the interface. Obtained results show the anisotropic properties effect on the velocity and pressure of the two-phase flow. In the article "*Theoretical simulation study on controlling factors in horizontal well CO_2 stimulation of heavy oil*" by T. Liu et al., the components and geology model according to typical heavy oil reservoirs are established firstly

to improve the research on the controlling factors and a quick filter criterion to choose heavy oil reservoirs for CO_2 stimulation is proposed. In the article "*Compositional simulation on the flow of polymeric solution alternating CO_2 through heavy oil reservoir*" by M. S. Jeong et al., polymer alternating gas (PAG) process involving a combination of polymer flooding and CO_2 injection is suggested to examine the applicability of carbon dioxide to recover viscous oil from highly heterogeneous reservoirs. It is found that PAG method would increase oil recovery over 45% compared with water alternating gas (WAG) process. In the article "*An estimation method of water drive displacement based on formation permeability distribution and inverse problem modeling*" by R. Huang et al., an inverse problem modeling of formation permeability distribution is taken to study reservoir vertical heterogeneity and oil production performance. Through the established objective function, an optimal solution for water production rate is obtained. The article "*Boundary-layer flow and heat transfer of nanofluids over a permeable moving surface in the presence of a coflowing fluid*" by A. Noor et al. demonstrates the theoretical investigation on the steady boundary-layer flow of a nanofluid past a permeable moving flat plate in the presence of a coflowing fluid. The governing equations are first transformed into ordinary differential equations before they are solved numerically using a finite difference scheme along with a shooting method. Dual solutions exist when the plate and the free stream move in the opposite directions. In the article "*Effect of thermophysical properties on coupled heat and mass transfer in porous material during forced convective drying*" by W. Cai et al., the convective drying kinetics of porous medium was investigated numerically. The authors find that thermal diffusion coefficient has significant positive impact on temperature distribution and mass diffusion coefficient might directly affect the moisture content distribution. Soret effect has a significant impact on heat flux and temperature distribution in the presence of large temperature gradient. In the article "*Magnetohydrodynamic stagnation flow and heat transfer toward a stretching permeable cylinder*" by A. Mastroberardino and J. I. Siddique, the authors analyze the magnetohydrodynamic flow and heat transfer on a stretching and permeable cylinder, prove existence of solutions for all values of the relevant parameters, and provide unique results in the case of a monotonic solution with nonlinear boundary value problems. In the article "*Separation process by porous membranes: a numerical investigation*" by A. L. Cunha et al., the authors predict the behavior of the concentration polarization boundary layer along the length of a permeable tubular membrane and evaluate the effects of axial Reynolds and Schmidt numbers on the concentration polarization boundary layer thickness during the cross-flow filtration process.

There are 5 articles for numerical methods in fluid flow and heat transfer. In the article "*Fast algorithm of numerical solutions for strong nonlinear partial differential equations*" by T. Liu et al., a new algorithm for "one-step calculation for pressure, one-step calculation for velocity, and multistep calculation for fractional flow and saturation" is proposed. The algorithm can avoid the nonconvergence caused by the saturation directly calculated by pressure, which has an important reference value in the numerical simulations of chemical and gas flooding for oil reservoirs. In the article "*Performance analyses of IDEAL algorithm on highly skewed grid system*" by D. Sun et al., the feasibility of the IDEAL algorithm on highly skewed grid system is analyzed to be more robust and more efficient than the traditional SIMPLER algorithm. The convergence rate of the IDEAL algorithm is 6.3 times faster than that of the SIMPLER algorithm, and the IDEAL algorithm can converge almost at any size of time step. In the article "*Numerical investigation for the modeling of the magnetic buoyancy force during the natural convection of air in a square enclosure*" by K. Song et al., two kinds of the expressions for the magnetizing force are considered and compared in the numerical computations. The numerical results reveal that the natural convection inside the enclosure does not depend on the types of the expressions for magnetizing force. In the article "*Simulation-based optimization of a vector showerhead system for the control of flow field profile in a vertical reactor chamber*" by H. Xia et al., a profile error feedback (PEF) optimization with a cyclic iterative approximation idea is proposed to optimize a vector showerhead gas delivery system for the control of mass transport to parameterize the geometry configuration of the holes in high resolution. In the article "*Multiobjective optimization of a counterrotating type pump-turbine unit operated at turbine mode*" by J.-H. Kim et al., an optimization method for improving the turbine output and efficiency of a counterrotating type pump-turbine unit operated at turbine mode is carried out. The blade geometry of both runners is optimized using a hybrid multiobjective evolutionary algorithm coupled with a surrogate model, Reynolds-averaged Navier-Stokes equations and the Latin hypercube sampling method. The turbine outputs and efficiencies of optimized pump-turbine units are simultaneously improved.

The applicable research articles cover colorful topics directly solving real problems or strongly related to them, including solar energy, fuel, U-type tube, human environment, pump design and operations, noise detection, manufacturing, ware storage, bridge engineering, and bionics. They will be introduced subsequently as follows.

The article "*Thermal effect on a CIGS thin-film solar cell P2 layer by using a UV laser*" by D.-C. Chen et al. deals with solar energy problems. It uses ANSYS software to analyze an ultraviolet (UV) (355 nm) laser processing system and confirms that the laser apparatus is effective when applied to a stainless steel CIGS solar cell P2 layer.

The article "*Modeling of the transport phenomena in passive direct methanol fuel cells using a two-phase anisotropic model*" by Z. Miao et al. and the article "*Compressed biogas-diesel dual-fuel engine optimization study for ultralow emission*" by H. Koten et al. solve the fuel related problems. The former one shows the simulation of transport phenomena in a passive direct methanol fuel cell by the proposed model considering anisotropy and deformation of the gas diffusion layer (GDL). Methanol concentration between 3 M and 4 M was recommended for passive liquid feed DMFC in order to achieve a balance between the cell performance and the methanol crossover. The latter one is to find out the optimum operating conditions in a diesel engine fueled with

compressed biogas and pilot diesel dual-fuel. The results showed that significantly lower NO_x emissions are emitted under dual-fuel operation for all cases compared to single-fuel mode at all engine load conditions.

The article "*Numerical thermodynamic analysis of two-phase solid-liquid abrasive flow polishing in U-type tube*" by J. Li et al. and the article "*Numerical analysis and optimization on flow distribution and heat transfer of a U-type parallel channel heat sink*" by X. Hao et al. are researches on U-type tube which are widely used in military and civilian fields. In the former one, the authors carry out numerical simulations of the surface processing characteristics of a two-phase solid-liquid abrasive flow polishing a U-type tube. A theoretical basis is developed for controlling the quality of abrasive flow polishing. In the latter one, a multiobjective optimization design of a U-type parallel channel heat sink is performed. The method offers sufficiently accurate predictions for practical designs, being quite straightforward to use.

The article "*The influence of the bed with a semiopen hood on bacteria removal in a negative-pressure isolation room*" by J.-M. Huang and H.-T. Cheng and the article "*Numerical simulation of particles deposition in a human upper airway*" by D. Li et al. study the human environment related topics. The authors in the former article numerically investigate the influence of a sickbed with a semiopened hood on bacteria removal in a negative-pressure isolation room. They find that the hood has excellent effect on the removal of contaminated air, which is much better than the case without hood. The flow field of patient's face up cough is very different from face side cough, and the contaminated air cannot be removed properly through one air outlet. The increase of hood height has a negative impact when the patient coughs lying on back. The authors in the latter article rebuild a realistic geometric model from nasal cavity to upper six-generation bronchia and conclude that the particle diameter plays a weaker role on nanoparticle depositions than micron particles.

The topic of pump design and operations is distributed in the following 3 articles. In the article "*Research on blade thickness influencing pump as turbine*" by S.-S. Yang et al., numerical research on three different specific speeds of pump as turbine (PAT) with different blade thickness is carried out. The results show that the blade thickness of PAT should be as thin as possible if its strength could be met. In the article "*Numerical study on the characteristics of pressure fluctuations in an axial-flow water pump*" by Z.-J. Shuai et al., three-dimensional unsteady numerical simulations are conducted on the complex turbulent flow field in an axial-flow water pump to investigate the flow induced vibration problem. The authors find that the first blade passing frequency dominates the frequency spectrum for all monitoring locations inside the pump. In the article "*Startup characteristics of a centrifugal pump delivering gas-liquid two-phase flow*" by Y.-L. Zhang et al., the transient startup flow characteristics inside the high specific-speed prototype centrifugal pump delivering the gas-liquid two-phase flow are numerically simulated during the startup period using the dynamic slip region method. The main reason for evolution of the internal flow field during the startup period is attributed to the continuous variation of the attack angle at the leading edges of blades.

In the article "*Simulation of broadband noise sources of an axial fan under rotating stall conditions*" by L. Zhang et al., the authors analyze the aerodynamic noise before and after the phenomenon of rotating stall by solving Navier-Stokes equations. They find that the aerodynamic noise source of the fan is mainly the rotation noise. The vortex noise accounts for the major part of fan noise after the occurrence of stall, and the maximum acoustic power level of the fan appears in the rotor domains. In the article "*Numerical simulation of temperature field and residual stress distribution for laser cladding remanufacturing*" by L. Hua et al., the cladding process of Ni–Cr–B–Si coatings on 16 MnR steel under different parameters of laser power, scanning speed, and spot diameter is simulated. A new criterion based on the ratio of the maximum tensile residual stress and fracture strength of the substrate is proposed for optimization of the remanufacturing parameters. In the article "*The numerical investigation of temperature and velocity distribution in the high-bay depot*" by X. Wu et al., the authors conclude that the radius and spacing of the supply-air inlet have great influence on temperature distribution. The results can provide a theoretical reference for the high-bay depot design and economic operation. In the article "*Prediction of backwater profiles due to bridges in a compound channel using CFD*" by S. Kocaman, the prediction performance of backwater surface profiles for three different types of bridges with or without piers in a compound channel is assessed. The CFD model provides reasonably good description of backwater surface profiles upstream of the bridges. In the article "*Modeling and simulation of fish-like swimming in a straight-line swimming state using immersed boundary method*" by W. Wang et al., a self-propelled swimming fish model is established, reflecting the interaction between internal force generated by muscle contraction and the external force provided by fluid. The results reveal that the fish gain energy from flow field by the conversion of "C" type and "S" type of fish body.

Bo Yu
Shuyu Sun
Jinjia Wei
Dongliang Sun
Yi Wang
Weihua Cai