International Journal of Heat and Mass Transfer 128 (2019) 1184–1200

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Impact of an oscillating guide vane on the thermo-hydraulic fields in a square cavity with single inlet and outlet ports

Yang-Cheng Shih [a], J.M. Khodadadi [b,*], Shih-Wei Nien [a], Yi Zeng [b], Xuan-Long Huang [a]

[a] Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, 1, Sec. 3, Chung-Hsiao, E. Road, Taipei 10608, Taiwan, ROC
[b] Department of Mechanical Engineering, Auburn University, 1480 Wiggins Hall, Auburn, AL 36849-5341, USA

ARTICLE INFO

Article history:
Received 10 June 2018
Received in revised form 12 September 2018
Accepted 13 September 2018
Available online xxxx

Keywords:
Computational fluid dynamics
Enclosures
Enhanced heat transfer
Flow oscillation
Flow separation
Forced convection
Guide vane
Rotating vortices

ABSTRACT

This study reports on a numerical investigation of the impact of an oscillating rigid plate guide vane on the transient behavior of thermo-hydraulic fields within a square cavity with single inlet and outlet ports. The computational fluid dynamics software ANSYS® FLUENT was employed to perform the numerical simulations. The numerical analysis focused on the effect of the frequency and amplitude of an oscillating rigid guide vane on modification of the transverse throughflow, internal flow re-distribution, re-arrangement of multiple rotating vortices and the ensuing heat transfer performance. According to the numerical results, it is shown that for a combination of the Strouhal number and amplitude of inclination angle of the guide vane, the instantaneous Nusselt number that is a measure of the thermal performance of the system exhibits cyclical changes once the respective periodic state was reached. Moreover, the cycle-averaged and peak-to-peak Nusselt number increases with the increase of the amplitude of oscillations as the frequency of oscillation is fixed. For a fixed amplitude of oscillation, the cycle-averaged and peak-to-peak Nusselt number decreases as the frequency increases. Instantaneous variations of the dimensionless pressure drop of the cavity also exhibited periodic oscillations. Enhancement of heat transfer due to oscillations of the guide vane was accompanied with the expense of increasing the pressure drop.

1. Introduction

Fluid flow and heat transfer problems in enclosures and cavities widely occur in engineering applications including cooling of electronics, ventilation systems of buildings, solar energy collectors, thermal storage systems, mixing chambers, heat exchangers, lubricated journal bearings, microfluidic devices, etc. Given the diversity of these applications, control of fluid flow within cavity systems, improving heat transfer performance and reducing energy consumption have received great attention by researchers.

Practical multi-dimensional problems associated with fluid flow and heat transfer in cavities (e.g. piezoelectrically-actuated microvalve for liquid flow control [1]) can be investigated in a simple enclosure model. Using active and passive approaches to control and regulate convective flow in cavities is very important to the design of these systems. To achieve this, applications of active or passive techniques in a simple rectangular cavity have been addressed for years. These include the problem of the lid-driven cavity flow subject to oscillating lid and protruding fixed thin fin on stationary side wall [2–4], buoyancy-driven convection within a differentially-heated cavity affected by presence of a stationary thin fin [5], and forced convection within a square cavity with inlet and outlet ports under steady [6] and oscillating inlet flow [7] conditions. Velazquez et al. [8] extended the work of [7] by investigating the effects of the aspect ratio of the enclosure, duct extensions at inlet/outlet ports and variable properties of the fluid, whereas Sourtiji et al. [9,10] studied the influence of buoyancy-driven convection and a water-alumina nanofluid, respectively, on modifying thermal performance of the model problem of [7].

One of the active approaches for regulating heat transfer in a convection-dominated cavity system is to control the movement of a wetted boundary, e.g. [2,4]. Khayat [11] reported on flow mixing in a cavity due to a block rotating at a constant angular velocity using the boundary element method through simulation of three-dimensional transient creeping flow of Newtonian or Jeffreys-type linear viscoelastic fluids. Monitored values of traction associated with the block tip and cavity face exhibited non-linear periodic dynamical behavior with time. Three-dimensional Stokes flow in a periodic domain driven by movement of a rotating screw that corresponds to the flow within mixers with a stationary outer barrel was studied by Kim et al. [12]. General expansion of the flow

* Corresponding author.
E-mail address: khodajm@auburn.edu (J.M. Khodadadi).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.09.060
0017-9310/

field and the solution of the coefficients were handled using a hybrid spectral/finite-difference approach. Elongation and shear zones were examined and two mechanisms responsible for mixing were identified. To further assess the viability of this approach for heat transfer applications, a computational study of flow and thermal fields due to isothermal or insulated rotating cylinders of various cross-sectional shapes (circle, square, and equilateral triangle) in a square cavity was studied by Shih et al. [13]. In their study, similarity of flow and thermal fields for rotating objects of various shapes with the same hydraulic diameter pointed to the proper choice of equivalency based on the hydraulic diameter. The periodic states of fluid flow and heat transfer within the cavity system were elucidated. However, the effects of natural convection, radiation, and viscous dissipation were neglected. Shih and Cheng [14] studied a problem similar to [13] that involved rotation of a circular cylinder within stationary outer cylinders with their cross-sections being circles, squares, and equilateral triangles. Viscous dissipation affecting the flow of the fluid contained between two wetted surfaces was considered. The triangular enclosure was found to be very effective in dissipating the internal energy, whereas the performance of the circular enclosure was the poorest. Karimi Talkhoncheh et al. [15] have reported on a variation of [6,13] through time-dependent simulation of forced convection affected by two fixed heated horizontal cylinders positioned within a 2-dimensional square enclosure that included inlet and outlet ports. Shifting of steady flow and heat transfer to periodic oscillation and then to chaotic oscillation for cylinders with dimensionless distance of 0.1 between them were clearly observed, whereas for greater cylinder spacing, the sequence of these regimes differed markedly. Particle image velocimetry measurements of swirl generation in vertical and horizontal experimental flow facilities using a flow-driven rotating vane modeling similar passive devices located on the top of the structural grids in the fuel assemblies of nuclear power plants were presented by Seo et al. [16]. Three types of vanes, i.e. spacer grid, fixed split and moving rotational vane were used. Pressure drops in the flow facilities for three types of vanes were compared to computational predictions. Moving rotational vane led to a higher value of swirl ratio compared to the fixed split vane.

Another active approach for regulating heat transfer in a convection-dominated cavity system is to introduce an object following prescribed or fluid-modified motion. For example, Shi and Khodadadi [17,18] employed a computational fluid dynamics (CFD) method within the framework of an in-house computer code utilizing the finite volume method to investigate the effect of addition of an oscillating thin fin on the transient and periodic states of fluid flow and heat transfer in a lid-driven cavity system. For enhancing heat transfer of an electronic device, Fu and Yang [19,20] proposed an innovative approach by swinging extremely thin fins in a moving fluid, an approach that belongs to the topic of the moving boundary problems. The influence of the velocities of the flowing fluid, the swinging speed and amplitude of the fins on the dynamic behavior of thermal and flow fields was investigated numerically. Dey and Chakrborty [21] conducted a numerical study to investigate how an oscillating fin disturbed thermal boundary layers within a duct and resulted in enhancement of heat transfer. The effects of the Reynolds number, amplitude and frequency of the oscillating fin on the Nusselt number were also addressed in their study. Moreover, they also discussed the impact of a pair of parallel oscillating fins on heat transfer enhancement. Extensions of systems utilizing rigid structures [17–21] to the case of flexible structures inspired by biological systems have also been reported. Joshi et al. [22] employed the sharp-interface immersed boundary approach and the finite element method to study the fluid-structure interaction of a laminar cross flow passing over twin thin flexible fins in a heated two-dimensional channel. Their numerical results revealed that the vortex ring generated by the flow-induced deformation of the fins could augment the heat transfer rate on the channel walls. The effects of some parameters, including the Young's modulus, pulsation frequency and the Prandtl number on the thermal augmentation were also discussed. Recently, Ghalambaz et al. [23] used the Arbitrary Lagrangian-Eulerian method to consider the fluid-structure interaction of a flexible oscillating fin attached on the vertical hot wall of a two-dimensional differentially-heated square cavity. The parameters including the Rayleigh number, fin length, oscillating amplitude and period, thermal conductivity ratio and the Young's modulus affecting promotion of the Nusselt number were discussed.

Another approach for modifying flow and heat transfer deals with employing guide vanes (GV) that could be thin planar or curved surfaces rotating about an axis by the flow, programmed to rotate or fixed. Yilmaz et al. [24] reported experimental findings of heat transfer augmentation (maximum of 148%) within a straight pipe flow caused by re-directing of flow at the inlet by different fixed radial guide vanes. Experimental and computational findings in relation to average pressure distributions on a circular cylinder placed in the branch of a tee-branch with/without vanes were addressed by Wu et al. [25]. Lin [26] gave an in-depth review of low-profile GV for managing flow separation in boundary layers over airfoils, whereas co- and counter-rotating GV vortex generators in various configurations were employed by Bur et al. [27] in a transonic channel to control the interaction of a separated turbulent boundary layer and a shock wave. Zhen et al. [28] compared experimental and computational results of their studies involving effects of placement of fixed thin planar GV on wings of unmanned aerial vehicles in relation to the lift coefficient. Surge and rotating stall compression system instabilities being natural limits to the performance of all compressors and particularly disturbing in operation of gas turbine engines can be avoided by employing fixed variable-angle GV and placing bleed valves along the compressor as reviewed by Paduano et al. [29]. Improving low-end torque and transient response of a turbocharged engine by placement of fixed yet variable inlet GV upstream of a turbocharged compressor was investigated by Uchida et al. [30]. The dynamic mesh method and the shear stress transition (SST) $k-\omega$ turbulence model were used by Li et al. [31] to simulate the three-dimensional closing airfoil-shaped GV in a pump-turbine in the pump mode. Elastic vibration behavior of an airfoil-shaped GV of a Francis turbine passage due to flow excitation was studied computationally by Liew et al. [32] utilizing low-order velocity–pressure finite elements, whereas the arbitrary Lagrangian–Eulerian formulation was adopted for solving the flow field. Through combining a large-eddy simulation flow solver and a structure solver, Wang et al. [33] revisited the same problem. Sparrow et al. [34] have reviewed how advances in computational techniques have led to integration of heat exchangers and associated fan/blower components that involve blades/vanes.

Despite the broad utilization of guide vanes to control aerodynamic instabilities, turbulence and stall, no report of its use to modify heat transfer was found. In contributing to the existing knowledge base of management of heat transfer in cavity flow systems, the objective of the present study is to use a dynamic CFD method to investigate the effect of a periodically-swinging guide vane on the thermo-hydraulic fields in a square cavity with single inlet and outlet ports. Prior studies focusing on the influence of such moving structures on mixing and improved transport are limited to cases where fins were attached on the wetted solid boundaries [21–23]. Specifically, this study is focused on elucidating the effects of the amplitude of a hinged guide vane's inclination angle and frequency of oscillation on the flow and thermal fields, that in turn affect the thermal performance of these systems.