International Journal of Thermal Sciences 114 (2017) 155–171

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Effect of included angle on turbulent flow and heat transfer in rhombic serpentine heat exchangers

Tong-Miin Liou*, Chun-Sheng Wang, Shu-Po Chan

Department of Power Mechanical Engineering, National Tsing Hua University, Hsinchu 30013, Taiwan, ROC

A R T I C L E I N F O

Article history:
Received 16 February 2016
Received in revised form
10 December 2016
Accepted 10 December 2016

Keywords:
Serpentine heat exchanger
Rhombus channel
Heat transfer
Included angle
Rotation

A B S T R A C T

Numerical studies of included angle effect on turbulent flow and heat transfer in rhombic serpentine heat exchangers are presented for the first time. Five included angles (γ) are examined, i.e. $\gamma = 45$-deg, 60-deg, 90-deg, 120-deg, and 135-deg, with rotation numbers (Ro) and coolant-to-wall density ratios ($\Delta\rho/\rho$) ranging from 0 to 0.2 and 0.1 to 0.2, respectively, at a constant Reynolds number (Re) of 10,000. The Reynolds-averaged Navier-Stokes (RANS) equations with low-Re realizable k-ε model and enhanced wall treatment are solved with the tradeoff between accuracy and computational cost. Code validations are performed through the comparisons with previous measured local Nusselt number ratio (Nu/Nu_∞) and thermal performance factor (TPF) for both stationary and rotating channels. The results show that the value of γ significantly affects the flow fields and heat transfer behaviors in both stationary and rotating cases. In the stationary situation, the Nu/Nu_∞ distributions on leading and trailing walls of the channels with slant walls ($\gamma \neq 90$-deg) are asymmetric. As the channel rotates, there exists a critical rotating number, Ro_c, in between $Ro = 0.1$ and 0.2. Beyond Ro_c, f/f_0 decreases with increasing γ. The friction reduction of 135-deg channel to 90-deg channel for $Ro = 0.2$ and $\Delta\rho/\rho = 0.1$ is about 43%. Furthermore, above Ro_c and a critical $\Delta\rho/\rho$ around 0.2, the TPF increases with increasing γ, resulting in a 6% TPF augmentation when comparing the 135-deg channel with the 90-deg one.

1. Introduction

With increasing capacity in modern highly efficient power and propulsion systems, aggressive energy exchanging techniques such as impingement and convection heat transfer are widely employed to meet the specific heat load requirements. For convection heat transfer, the serpentine heat exchanger is most frequently used. In the construction heating, ventilation, and air conditioning (HVAC) systems, it serves as an evaporator or a condenser that controls temperature by transferring energy between the refrigerant and the ambient air [1,2]. In proton exchange membrane fuel cell (PEMFC) and phosphoric acid fuel cell (PAFC), it acts as a cooling circuit that uniformly distributes the temperature by removing waste heat from the cathode [3,4]. In turbomachinery, it functions as an internal cooling passage that maintains allowable blade temperature of advanced gas turbines (GT) by convective heat transfer [5,6]. Hence a thorough understanding of the fluid flow and heat transfer inside the serpentine channel under both stationary (HVAC and PEMFC or PAFC) and rotating (GT) conditions will help the heat exchanger designer to improve efficiency.

It is well established that the heat transfer and fluid flow in serpentine channels are affected by various parameters, such as Reynolds number (Re), rotation number (Ro), buoyancy number (Bu), density ratio ($\Delta\rho/\rho$), turn geometry, rib configuration, channel cross-sectional shape, channel orientation (β), inlet condition, channel configuration, and working fluid. These parameters have been investigated by many researchers with both experimental and numerical methods. Wagner et al. [5] conducted experimental studies on the effects of buoyancy and Coriolis forces in a smooth four-pass square serpentine channel. The results showed that compared with stationary case both Coriolis and buoyancy effects need to be considered. In addition, heat transfer coefficients varied markedly with the flow direction, i.e. radially outward or inward. Johnson et al. [6] examined the heat transfer in rotating four-pass square ducts with different β. They found that different β values resulted in different heat transfer distributions. For example, the heat transfer on the leading surfaces of radially outward smooth duct for $\beta = 45$-deg is twofold that for $\beta = 0$-deg. Schabacker et al. [7] measured the velocity distributions in a stationary smooth two-

---

* Corresponding author.
E-mail address: tmliou@pme.nthu.edu.tw (T.-M. Liou).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.12.007
1290-0729/

pass square duct. They observed that a strong bend-induced secondary flow, i.e. two counter-rotating vortices, impinges on the outer wall and influences the flow in the downstream leg. Liou and Chen [8] studied the effects of 180-deg sharp turn in a two-pass square channel on both heat transfer and fluid flow. They concluded that the secondary flow induced by the curvature and turbulence enhancement associated with unsteady separation bubble downstream of the turn were responsible for the significant heat transfer augmentation in the front half of the second pass. Iacovides et al. [9] and Liou et al. [10] investigated the effect of rotation on local heat transfer coefficients in a two-pass square channel. The results showed informative spanwise variations of heat transfer enhancement or abatement resulting from rotation. Son et al. [11], Azad et al. [12], Cho et al. [13], and Agarwal and Acharya [14] explored the effects of rib on heat transfer and fluid flow in a two-pass square and rectangular ducts. It was found that ribs not only disturbed the main flow by producing recirculation and secondary flows but also enhanced the heat/mass transfer two to three times more. Fu et al. [15,16] performed heat transfer measurements in two-pass rectangular channels with various aspect ratios. They determined that the 1:4 channel had superior thermal performance because it incurred the least pressure penalty. Wright et al. [17] investigated the influence of inlet condition on heat transfer in one-pass rectangular cooling channels with three entrance conditions: fully developed, sudden contraction, and partial sudden contraction. The results showed that the heat transfer at the entrance of smooth channel was significantly enhanced with the sudden contraction and partial sudden contraction entrances. Liou et al. [18] explored the heat transfer augmentation and pressure loss in a twin-pass parallelogram channel. Their rotating results indicated that relative to the square channel, the critical *Ro* for heat transfer recovery over the leading wall (LW) of the first pass is lower for parallelogram channel. Khoshvaght-Aliabadi et al. [38,39] measured the heat transfer and pressure drops of serpentine tubes with the nanofluid flow and agitation. They found that the addition of nanoparticles into working fluid and agitation improve the thermal performance.

In addition to the aforementioned experimental studies, there are several numerical investigations on fluid flow and heat transfer in serpentine channels. Iacovides and Launder [19] presented a parametric study of fully developed turbulent flow and heat transfer in rotating one-pass rectangular channels with Reynolds-averaged Navier-Stokes (RANS) high-*Re* k-ε model in the channel core region combined with one equation low-*Re* turbulence model in the near-wall region. The predicted trends of *Nu* levels were in good qualitative agreement with experimental results. Nonetheless, the centrifugal buoyancy effects were not incorporated. Dutta et al. [20] and Sathyamurthy et al. [21] respectively presented numerical predictions for rotating one-pass and two-pass square smooth channels. Their calculations showed that the high-*Re* k-ε model with standard wall function in the near-wall region could not provide accurate flow and heat transfer results. Therefore, more refined turbulence models such as low-*Re* k-ε model and second-moment closure model should be used. Iacovides et al. [22] tested an algebraic second moment (ASM) turbulence model in a square U-bend. The flow predictions suggested that turbulence anisotropy within the channel core region and wall sublayers caused by the strong curvature had a strong influence on the flow development. However, they did not consider the energy transport in the duct. Furthermore, Iacovides et al. [23] numerically studied developing turbulent flow and heat transfer in stationary and rotating smooth U-ducts. Four turbulence models were used, including a high-*Re* k-ε model interfaced with the low-*Re* one-equation model in the near-wall regions, a high-*Re* ASM closure with the low-*Re* one-equation model in the near-wall regions, and two versions of a low-*Re* ASM model. Among the four tested turbulence models, the low-*Re* ASM produced noticeably better results of the flow development. Wang and Chyu [24] examined the influence of three turning configurations, i.e. (1) straight-corner turn, (2) rounded-corner turn, and (3) circular turn, on secondary flow patterns and heat transfer distribution in a two-pass square duct using an extended version of the k-ε model. Their numerical results showed a reasonable agreement with experimental data. It is found that at the turn, the straight-corner case has the strongest turn-induced heat transfer enhancement while the circular turn has the weakest. Bonhoff et al. [25] presented the heat transfer predictions for a rotating U-bend coolant channel using differential Reynolds stress model (RSM) with wall function in FLUENT code. The calculated averaged *Nu* were in a satisfactory agreement with the experimental results of Wagner et al. [1]. Chen et al. [26] computed the fluid flow and heat transfer in a rotating two-pass smooth square channel with two turbulence models: a two-layer k-ε isotropic eddy viscosity model and a near-wall second-moment closure model. The predicted results provided good agreements with the data of Wagner et al. [1]. The corresponding velocity profiles, however, were not validated. Jang et al. [27] used the same second-moment closure model for fluid flow and heat transfer predictions in two-pass square channels with 60-deg ribs. Furthermore, Al-Qahtani et al. [28] and Su et al. [29] applied this model to calculate turbulent thermal flow features in rotating two-pass rectangular channels. All computed results were in reasonable agreements with experimental results. Murata et al. [30] performed large eddy simulation (LES) of turbulent flow and heat transfer in a rotating two-pass smooth square channel with an 180-deg sharp turn. The simulated results had a more favorable agreement with the experiment data, but the computational costs were also high. Sleiti et al. [31,32] studied various turbulence models with FLUENT. They tested the effects of $\Delta\rho/\rho$, *Ro*, and 90-deg rib on fluid flow and heat transfer in a two-pass square channel and concluded that the RSM with enhanced wall treatment predicted the most accurate results among the tested RANS models. Luo and Razinsky [33], and Chu et al. [34] numerically studied the effect of guide vane on thermal fluid flow features in the turn region with v^2-f model and second order closure model, respectively. The results showed that the addition of a vane in the turn did not cause much impact to the flow before the turn but it greatly affected flow behaviors and heat transfer characteristics in the turn region and the second pass. Saha and Acharya et al. [35] used the Realizable k-ε model with enhanced wall treatment in FLUENT to examine the entrance geometry effect in a narrow one-pass rectangular channel. Their validated heat transfer results showed satisfactory agreements with experimental data of Liou et al. [10]. It was concluded that entrance geometry significantly affected the inlet velocity profiles and the rotation generated secondary flows. Khoshvaght-Aliabadi et al. [37] calculated the thermal performance of helical, spiral, and serpentine tubes. They concluded that the thermal performance of helical tube is superior to that of the others. Meanwhile, different working fluids are also tested and one with higher Prandtl numbers is found to have greater values of Nusselt number.

In summary, most previous literature focus on examining the effects of turn geometry, rib configuration, channel cross-sectional shape, channel orientations, and inlet condition. However, to authors' knowledge, none of these papers studied the effects of included angle (γ) on fluid flow and heat transfer in rhombic serpentine channels. The rhombic serpentine channel features asymmetric heat transfer on its bottom and top walls [18], which is more capable of working in single side heat load or unequal double-sided heat load conditions, such as fuel cell [3,4] and gas turbine, compared to traditional square, rectangular, or circular channels. Fig. 1 shows such an example of a rhombic channel with application