International Journal of Heat and Mass Transfer 66 (2013) 164–176

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

The effect of corrugation on heat transfer and pressure drop in channel flow with different Prandtl numbers

Maryam Mirzaei [a], Lars Davidson [b,*], Ahmad Sohankar [c], Fredrik Innings [d]

[a] Department of Mechanical Engineering, Yazd University, Yazd, Iran
[b] Chalmers University of Technology, Gothenberg, Sweden
[c] Department of Mechanical Engineering, Isfahan University of Technology, Isfahan, Iran
[d] TetraPak, Lund, Sweden

ARTICLE INFO

Article history:
Received 4 November 2012
Received in revised form 18 June 2013
Accepted 19 June 2013

Keywords:
Wavy channel
Plane channel
Turbulent heat transfer
Boundary layer
LES

ABSTRACT

Large Eddy Simulation and Direct Numerical Simulation are applied to study the turbulent flow field in a wavy channel at two Prandtl numbers, $Pr = 0.71$ and $Pr = 3.5$, and Reynolds number $Re_b = 10{,}000$. The characteristics of the separated shear layer and the near wall recirculating zone are discussed in relation to the turbulent heat transfer. Special attention is paid to the behavior of the flow and thermal boundary layers and various turbulent characteristics and their effects on the distribution of the Nusselt number and friction coefficient in the separation and reattachment regions. The results indicate that the thickness of the thermal boundary layer rather than the turbulent fluctuations has a significant effect on the local variation of the averaged Nusselt number. The results are compared with Direct Numerical Simulation results of a plane channel at the same Reynolds number.

1. Introduction

In various heat transfer devices, optimization of wall heat transfer is a key requirement to make the devices more compact or effective. The use of the surface protuberances is a passive heat transfer augmentation method and is based on developing boundary layers or streamwise fluctuations, creating swirl or vortices and flow destabilization or turbulence intensification. One typical technique is to use corrugations (waves) on the wall. Providing considerable heat transfer enhancement with a relatively low pressure drop as well as the simplicity of the manufacture method make these geometries more attractive than other passive enhancement methods such as ribs or vortex generators. The periodic changes of pressure gradient and the streamline curvature effects produce turbulence structures. The corrugated wall enhances heat transfer by destroying or decreasing the thermal boundary layer thickness and increasing the bulk flow mixing. The recirculating bubble in the wavy part of the channel and the separated shear layer formed above the separation point raise the turbulence intensity near the wall. The corrugated channels can be considered to be fully corrugated (fully-wavy or sinusoidal) or half corrugated channels. In a fully corrugated channel, the corrugation is applied to the entire channel length whereas in a half corrugated channel, part of the channel is corrugated and part of it remains smooth (see Fig. 1). The half corrugated channel is chosen as the geometry in the present study (referred to as the wavy channel).

Various studies have been conducted on turbulent flow over the fully corrugated walls [1–5]. Hudson et al. [1], Cherukat et al. [2] and Choi and Suzuki [3] studied the turbulent flow in a fully wavy channel with experimental, Direct Numerical Simulation (*DNS*) and Large Eddy Simulation (*LES*) methods, respectively. They provided an extensive study of different turbulent flow characteristics such as Reynolds shear stress, turbulent intensities and turbulence production over a wave with a wave amplitude to wave length ratio of 0.05. Calhoun and Street [4], Dellil et al. [5] and Yoon et al. [6] carried out numerical investigations of the effect of the wave amplitude in a fully wavy channel. They used different wave amplitude to wave length ratios and compared the turbulent flow patterns, the location of the separation and reattachment points and the mean distributions of heat transfer and pressure drop along the channel. As it has been mentioned in the literature review, most of the previous studies have been concentrated on the flow in a fully wavy channels, whereas in the present study a half corrugated channel is considered as the computational domain (see Fig. 1). Comparing the results of both geometries, some similarities and differences can be identified. For the wavy channels with identical wave amplitudes (fully and half corrugated), a small

* Corresponding author. Tel.: +46 31 7221404.
E-mail addresses: m.mirzaei@stu.yazduni.ac.ir (M. Mirzaei), lada@chalmers.se (L. Davidson), asohankar@cc.iut.ac.ir (A. Sohankar), fredrik.Innings@tetrapak.com (F. Innings).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.06.047

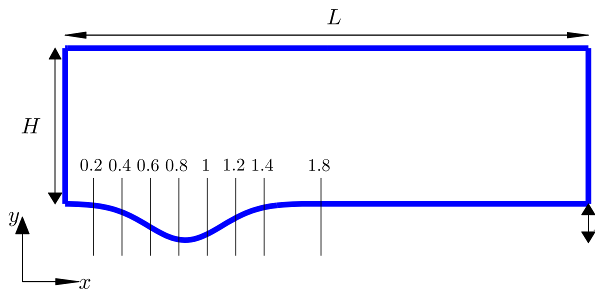

Fig. 1. Schematic view of the computational domain indicating different cross sections in the streamwise direction.

bubble appears at the bottom of the wave, where its location is nearly the same for both cases. In both cases, the peak values of the averaged Nusselt number and friction coefficient distributions occur around the separation and reattachment points. However, in half corrugated channel (the present study), due to the presence of the smooth part of the channel after the wave, the boundary layer formed after the reattachment point continues downstream of the wave, while for the fully wavy channel the boundary layer continues only until the next wave. The available studies concentrated mainly on the investigation of turbulent flow field whereas the present study pays detailed attention to both turbulent flow and temperature fields. We aim here to provide insight into the characteristics of the flow and heat transfer in a wavy channel by using Large eddy simulation (*LES*) and Direct Numerical Simulation (*DNS*). This study particularly discusses the variation of the temperature and velocity boundary layers for two different Prandtl numbers and their effects on the heat transfer. The results are also compared with the plane channel flow.

The paper is organized as follows. A presentation of the numerical method, boundary conditions, the turbulence modeling, the computational domain and the grid is given in Section 2. Section 3 is devoted to a description of the results including an analysis of the global, time-averaged and instantaneous results for both plane and wavy channels. Some conclusions are drawn in the final section.

2. The modeling methodology

2.1. Governing equations

The Navier–Stokes and energy equations for an incompressible viscous flow read:

$$\frac{\partial \bar{u}_i}{\partial t}+\frac{\partial(\bar{u}_i\bar{u}_j)}{\partial x_j}=\beta\delta_{1i}-\frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[(\nu+\nu_t)\frac{\partial \bar{u}_i}{\partial x_j}\right] \tag{1}$$

$$\frac{\partial \bar{T}}{\partial t}+\frac{(\partial \bar{u}_j\bar{T})}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\frac{\nu}{Pr}+\frac{\nu_t}{Pr_t}\right)\frac{\partial \bar{T}}{\partial x_j}\right]-\gamma\bar{u}_1 \tag{2}$$

x_1, x_2, $x_3(x,y,z)$ denote the Cartesian coordinate system, where the x_1 axis is aligned with the flow direction, the x_2 axis is the wall-normal direction and the x_3 axis is the spanwise direction. δ_{1i} is the streamwise component of Kronecker delta.

The periodic boundary condition is employed in the streamwise and spanwise directions for both the plane and the wavy channel. Using the periodic boundary condition for the velocity and temperature requires special modifications in the Navier–Stokes equation in the x direction and in the energy equation. The first term on the right-hand side of Eq. (2) represents the streamwise driving pressure gradient. For the plane channel, the Reynolds number based on the wall friction velocity (Re_τ) is prescribed, so β is equal to 1. For the wavy channel (see Fig. 1), to achieve the desired Reynolds number based on the bulk velocity (Re_b), β is defined on the basis of the flow rate and is modified in each time step. The last term on the right-hand side of the energy equation provides the temperature gradient in the streamwise direction. For the constant heat flux (q_w) condition on the wall, γ is obtained based on the inlet and outlet energy balance as $\gamma=\frac{q_w}{\rho u_{ref}Hc_p}$ where u_{ref} is the wall friction velocity (u_τ) for the plane channel and the bulk velocity (u_b) for the wavy channel.

For the wavy channel, the symmetry boundary condition is used for the upper wall and the no-slip condition and uniform heat flux are employed for the lower wall. For the plane channel, the no-slip condition and uniform heat flux are applied at both the upper and lower walls.

2.2. Model description

The present study uses the WALE turbulence model [8] as the subgrid scale model which reads:

$$\nu_{sgs}=(C_s\Delta)^2\frac{(s^d_{ij}s^d_{ij})^{3/2}}{(\bar{S}_{ij}\bar{S}_{ij})^{5/2}+(s^d_{ij}s^d_{ij})^{5/4}} \tag{3}$$

$$s^d_{ij}=\frac{1}{2}(\bar{g}_{ij}{}^2+\bar{g}^2_{ji})-\frac{1}{3}\delta_{ij}\bar{g}^2_{kk},\quad \bar{S}_{ij}=\frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j}+\frac{\partial \bar{u}_j}{\partial x_i}\right) \tag{4}$$

where C_s=0.325, $\bar{g}_{ij}$ is the velocity gradient tensor and s^d_{ij} is the traceless symmetric part of the square of velocity gradient tensor.

2.3. Numerical method

An incompressible finite volume code based on an implicit, fractional step technique with a multigrid pressure Poisson solver and a non-staggered grid arrangement is employed [9]. The second order Crank–Nicolson scheme is used for time discretization. The spatial discretization is based on the second-order central differencing scheme.

2.4. Computational domain and grid influence

The original domain is introduced as a periodic corrugated channel. To reduce computational costs, only one pitch of the domain is regarded as the computational domain. A two-dimensional schematic view (in the x–y plane) of the three-dimensional computational domain and the corresponding geometrical parameters are displayed in Fig. 1 and Table 1, respectively. The shape of the lower wall in the wavy part is given by $y=-a\exp(-((7x-x_0)^2)/(2\delta^2))$, where $a=0.14$, $x_0=5.907$ and $\delta=1.8219$. The Reynolds number based on the bulk flow velocity and channel height is $Re_b=u_bH/\nu=10{,}000$. All the results presented below are normalized with bulk flow velocity, channel height, kinematic viscosity and bulk temperature ($T_b=q_w/(u_b\rho c_p)$). Simulations are performed for two Prandtl numbers, $Pr=0.71$ and 3.5. The time step is chosen as $\Delta t=0.005$. It should be mentioned that three different time steps, $\Delta t=0.005$, $\Delta t=0.01$ and $\Delta t=0.015$, have been tested in the present study and it was observed that the accuracy of the results becomes weaker, when a larger value of Δt is employed. Therefore, $\Delta t=0.005$ is finally chosen as the proper time step. Based on the

Table 1
Geometrical parameters of the computational domain.

H	Height of channel	1
L	Channel pitch	3.68H
H_1	Corrugation height	0.2H
W	Width of channel	0.5H