International Journal of Heat and Fluid Flow 35 (2012) 168–175

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Fluid Flow

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

Flow structures and heat transfer on dimples in a staggered arrangement

Johann Turnow [a,*], Nikolai Kornev [a], Valery Zhdanov [b], Egon Hassel [b]

[a] Institute of Modeling and Simulation, University of Rostock, Albert-Einstein-Str. 2, 18059 Rostock, Germany
[b] Institute of Technical Thermodynamics, University of Rostock, Albert-Einstein-Str. 2, 18059 Rostock, Germany

ARTICLE INFO

Article history:
Received 14 October 2011
Received in revised form 11 January 2012
Accepted 12 January 2012
Available online 9 February 2012

Keywords:
LES
Dimples
Vortex structures
Heat transfer

ABSTRACT

Vortex structures and heat transfer enhancement mechanism of turbulent flow over a staggered array of dimples in a narrow channel have been investigated using Large Eddy Simulation (LES), Laser Doppler Velocimetry (LDV) and pressure measurements for Reynolds numbers $Re_H = 6521$ and $Re_H = 13{,}042$.

The flow and temperature fields are calculated by LES using dynamic mixed model applied both for the velocity and temperature. Simulations have been validated with experimental data obtained for smooth and dimpled channels and empiric correlations. The flow structures determined by LES inside the dimple are chaotic and consist of small eddies with a broad range of scales where coherent structures are hardly to detect. Proper Orthogonal Decomposition (POD) method is applied on resolved LES fields of pressure and velocity to identify spatial–temporal structures hidden in the random fluctuations. For both Reynolds numbers it was found that the dimple package with a depth h to diameter D ratio of $h/D = 0.26$ provides the maximum thermo-hydraulic performance. The heat transfer rate could be enhanced up to 201% compared to a smooth channel.

1. Introduction

Several methods of heat transfer enhancement like ribs, fins or dimples have been thoroughly investigated in the last decades with the aim to improve the heat transfer at a minimum hydraulic loss. In general, dimples are known already from golf ball aerodynamics. In this classical case applications of dimples are understood as a special form of surface roughness which shift the typical drop down of the flow resistance for blunt bodies into the low Reynolds number range. Initially, the idea was to use dimples for drag reduction. Wieghardt (1953) documented the influence of dimples in terms of friction factor. Remarkably results obtained in DLR have been published by Wuest (2004). In this study the application of dimples on ICE trains reduces the forces from sidewinds up to 17%. Lienhardt et al. (2008) performed experimental and numerical studies using dimples for drag reduction in a channel. No drastic reduction of drag has been documented for the investigated geometrical configurations of dimples. However, a high increase of drag resistance was also not observed which implies that dimples could be used to enhance heat transfer without a significant increase of the hydraulic losses. Afanasyev et al. (1993) investigated the pressure drop and heat transfer on a plate with dimples in the turbulent flow regime. The maximum of heat transfer augmentation of about 40% with only a low increase of the pressure drop. Most motivating results have been performed by Chyu et al. (1997), who investigated the pressure drop and heat transfer improvement for the surface roughened by an array of hemisphere and tear-drop shaped cavities in the range of Reynolds number $Re_{Dh} = 10{,}000–50{,}000$ based on the hydraulic diameter. The heat transfer coefficients on the roughened surface for both configurations were determined to be 2.5 times higher than these on the opposite smooth wall whereas the induced flow resistance was twice as small as that of rib turbulators. The effect of the channel height and dimple depth on heat transfer in the turbulent flow regimes has been studied by Moon et al. (2003) and Burgess and Ligrani (2004). It was revealed that the heat transfer and friction factor increase for a decreasing channel height and a rising dimple depth. Today a large number of experimental investigations have been performed for different dimple geometries (see e.g. Ligrani et al., 2003 and Ekkad and Nasir, 2003).

The results published in literature point out that concave formed dimples in comparison to other conventional methods show the best thermo-hydraulic performance which is defined as the ratio between the heat exchange and flow resistance. Depending on the geometrical configuration and on the flow properties, the spatially averaged Nusselt number Nu_m referred to that of a smooth channel Nu_{m0} varies from $Nu_m/Nu_{m0} = 1.1$ to $Nu_m/Nu_{m0} = 2.5$ related to the literature. The relative flow resistance increase varies in a range of $C_p/C_{p0} = 1.05–3.5$ which is fairly low compared to other methods like rib turbulators.

High values of the thermo-hydraulic performance $(Nu_m/Nu_{m0})/(C_p/C_{p0})^{1/3}$ are attained on dimples because of an efficient way of vortex generation. The vortices on ribs and fins, which are necessary to mix the fluid, are created through the flow separation on

* Corresponding author.
E-mail address: johann.turnow@uni-rostock.de (J. Turnow).

0142-727X/$ - see front matter © 2012 Elsevier Inc. All rights reserved.
doi:10.1016/j.ijheatfluidflow.2012.01.002

protruding elements what results in a high flow resistance. The vortices on dimpled surfaces are created inside of concave cavities preventing a blockage of the channel and keeping the additional resistance at a minimum. Its formation was in the focus of many studies, but unfortunately, main attention has been paid to time averaged values whereas the flow structures within the cavities and their contribution to the heat transfer mechanism remain still unclear and are not completely understood. Especially, in the turbulent range and at large ratio of dimple depth to dimple print diameter h/D the flow is complicated. Since the form of the vortex has a strong impact on the heat transfer, the deep understanding of physics inside the dimple becomes important for a further improvement of heat exchanger efficiency.

The most detailed experimental study of unsteady flow characteristics on dimpled packages has been performed by Ligrani et al. (2003). Flow visualization by smoke patterns revealed a primary vortex pair and additionally two secondary vortices arise at the spanwise edges of each dimple which enhance the mixing process. Unfortunately, these first valuable observations of flow physics have never been quantified properly and analyzed using modern non-intrusive measurement techniques and advanced numerical technologies like LES and DNS.

The objective of this investigation is to clarify the role of the vortex formations with respect to the heat transfer on a staggered dimple package using LES and measurements of pressure and velocity. In our previous work Turnow et al. (2010) vortex structures have already been investigated for a single spherical dimple. LES simulations revealed the formation of asymmetric structures with an orientation switching between two extreme positions. The emphasis of the present work is the identification of vortex structures on dimple package with a staggered arrangement. Since the maximal augmentation of heat transfer is documented for dimples with a depth to diameter ratio of $h/D \geqslant 0.22$, the range of h/D between 0.195 and 0.326 is investigated at Reynolds numbers $\mathrm{Re}_H = 6521$ and $\mathrm{Re}_H = 13{,}042$ corresponding to the fully developed flow regime. The paper is organized as follows. The mathematical model and the experimental setup are described in Sections 2 and 3. The numerical results are validated using experimental data available for smooth and dimpled channels in Section 4.1. Unsteady flow structures are analyzed in details in Section 4.2. Their impact on flow resistance and heat transfer is discussed in Section 4.3. Further in Section 4.4, the discrete Proper Orthogonal Decomposition (POD) method is applied on resolved LES fields to identify coherent structures hidden in the random fluctuations. A section of conclusions completes the paper.

2. Computational methods

Large Eddy Simulations (LESs) have been performed based on a 3-D finite volume method. The filtered transport equations are solved on a non staggered Cartesian grid. The discretisation in space and time of the quantities at the cell faces is of second order using central differencing scheme. The LES equations are obtained by filtering the continuity equation, the Navier–Stokes equations and the transport equation for the non-dimensional temperature θ at the filter width $\widetilde{\Delta}$:

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_j \tilde{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \widetilde{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) - \tau_{ij}\right] \tag{1}$$

$$\frac{\partial \tilde{\theta}}{\partial t} + \frac{\partial \tilde{u}_j \tilde{\theta}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{\nu}{\mathrm{Pr}}\frac{\partial \tilde{\theta}}{\partial x_j} - J_j^{SGS}\right]. \tag{2}$$

The unclosed subgrid stress tensor $\tau_{ij} = \widetilde{u_i u_j} - \tilde{u}_i \tilde{u}_j$ and the subgrid contribution to the scalar dynamics $J_j^{SGS} = \widetilde{\theta u_j} - \tilde{\theta}\tilde{u}_j$ are modeled in terms of the filtered quantities $\tilde{u}_i$ and $\tilde{\theta}$, using the localized dynamic mixed model (LDMM) on the basis of the dynamic model proposed by Zang et al. (1993). In Eq. (2) the non-dimensional temperature θ is defined as $\theta = T - T_{lowerwall}/(T_{lowerwall} - T_{upperwall})$ and further treated as a passive scalar without buoyancy effects. The molecular Prandtl number Pr was set to Pr = 0.7, whereas the turbulent viscosity μ_t and the turbulent Prandtl number Pr_t are determined dynamically using LDMM. Investigations revealed that the chosen SGS model (LDMM) shows fairly good agreement with measurements in respect to heat transfer and recirculating flows in contrast to other SGS models (see Turnow, 2011).

Since the whole dimple package starting from the first row with an uniform inflow cannot be simulated using LES due to restricted computer resources, only a part of dimples located within the fully developed flow has been considered (Fig. 1). It should be noted that the domain size should be chosen carefully to capture typical flow structures. In the most of the previous works, time resolved calculations of heat exchangers with dimples or protrusions have been performed in a domain containing either a half or only one whole dimple using periodic conditions in streamwise and spanwise directions. To our opinion this domain size is not sufficient because the typical vortex structures can be larger than the dimple diameter what results in a significant degradation of accuracy of numerical simulations even for the integral values. Therefore, an extended domain (presented in Fig. 1) with a dimension $4.66H \times H \times 4.66H$, where H is the channel height, including several dimples was chosen to guarantee capturing the largest flow structures. While the diameter D of the dimple with a sharp edge was kept constant at $D = 23$ mm, three dimple depths h of $h/D = 0.196$, $h/D = 0.26$ and $h/D = 0.326$ have been studied. Periodic boundary conditions have been applied in streamwise and spanwise directions. No slip wall conditions were enforced on the lower and the upper solid walls for the velocity. The temperature was fixed at the lower (hot surface, $\theta = 1$) and the upper (cold surface, $\theta = 0$) channel walls. To drive the flow with a constant massflow rate, the overall losses are calculated within the domain and simply added as driving force to the momentum equation at every time instant.

3. Experimental setup

Experimental investigations have been performed in a closed water loop channel for isothermal conditions presented in Fig. 2.

Main parts of the test rig are the head tank with an overflow weir inside, a settling chamber, the converging nozzle (contraction) followed by the test section and a back tank with a metering system. Further a water reservoir, main pump, pipelines and four controlling valves are placed between each device to ensure stable water levels. Water is pumped from the reservoir into the head tank by the main pump which can be additionally tuned to control the mass flow rate. A weir is installed inside the main tank to

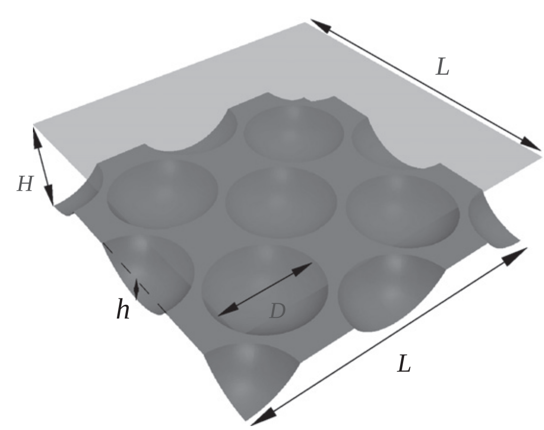

Fig. 1. Computational domain of the channel with dimples at the lower wall.