International Journal of Heat and Mass Transfer 100 (2016) 250–266

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

DNS of conjugate heat transfer in presence of rough surfaces

P. Orlandi [a,*], D. Sassun [a], S. Leonardi [b]

[a] Dipartimento di Ingegneria Meccanica e Aerospaziale, Università di Roma "La Sapienza", Via Eudossiana 16, Roma, Italy
[b] Dept. Mechanical Engineering, The University of Texas at Dallas, USA

ARTICLE INFO

Article history:
Received 22 December 2015
Received in revised form 25 March 2016
Accepted 12 April 2016
Available online 9 May 2016

Keywords:
Heat transfer
Roughness
Conjugate heat transfer

ABSTRACT

Often in the numerical simulations of turbulent heat transfer in a channel, the temperature is assigned as boundary condition at the wall. This condition is rather different from that in real experiments, where solid walls are thick and a constant temperature is imposed at the exterior of the walls. In the present work, rough channel flows with a Prandtl number $Pr_F = \nu/\alpha_F = 1$ have been simulated transporting thermal fields in presence of walls with two different thermal diffusivities α_{Si}. In one case, the thermal diffusivity corresponds to that of a material with conductivity 10 times greater than copper. In the other, the thermal diffusivity is that of glass. The upper wall of the channel is smooth and $0.5h$ thick (h being the half height of the channel). The lower wall is made of a rough layer of height $k = 0.2h$ superimposed to a uniform layer $0.3h$ thick. Several simulations were performed varying the roughness elements. Above the crests plane the total stress has a linear behaviour as in a smooth channel. By normalising the total stress with the stress at the smooth wall, it has been found that the surface made by three-dimensional staggered cubes leads to a drag 2.3 times greater than that of smooth walls. On the other hand, for triangular longitudinal bars the drag reduces. The heat transfer is reduced for low conducting walls, however the influence of the shape of the roughness is similar to that for high conducting walls. This has been observed through flow visualizations and joint pdf between velocity and temperature fields.

1. Introduction

In laminar flows the convective heat transfer can be augmented or reduced only by changing the transporting fluid. In turbulent flows, a reduction or an increase of the heat transfer can be achieved by changing the vortical structures near the walls by passive or active means. In this study, passive disturbances are considered, which are obtained by a modification of the shape of the surface of the wall. Not only the shape, but also the orientation of the geometrical disturbances has large effects on the global quantities such as resistance and heat transfer. The influence of the flow within the rough surface on that above the plane of the crests can be analysed in detail by the direct numerical simulation (DNS) of the Navier–Stokes equations.

Experimental and numerical studies were devoted to flows past orthogonal two-dimensional grooves, which is the simplest geometry to generate flows past rough surfaces. Square and cylindrical bars with several pitch, w, to height, k, ratio were considered. A complete literature review on flows past transverse bars can be found in [1]. Among these, the geometrical set-up of [2] is similar to that employed in the present paper, with one rough wall, and, the other smooth. The effort to perform numerical simulations of flows past rough surfaces is increasing even if the DNS are limited to low and intermediate Reynolds numbers [3–5]. The DNS provide a wealth of information and the possibility to evaluate any kind of statistics. For instance the DNS of [3] of flows past transverse square bars, with a large spectrum of pitch to height ratio, explained why the maximum drag occurs for $w/k = 7$ (with w the cavity width).

Despite a large number of studies aimed at clarifying the effect of roughness on the flow field, much less is known on the effect of roughness on the thermal field due to the difficulties to perform measurements and to realise the apparatus to control the temperature at the boundary. Miyake et al. [6] studied the transport of a passive scalar for $w/k = 7$, Nagano et al. [7] were fixing $w/k = 3$ by changing the height H of the square ribs. Leonardi et al. [8], by assigning the temperature T_0 on the lower rough wall and $-T_0$ on the upper smooth wall, made a comparison between transverse square bars and circular ribs for different values of w/k. At large w/k the dependency of the sum of the frictional and the form drag on the shape of the elements is negligible for the reason that the recirculating region behind the bar does not largely depend on

* Corresponding author.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.04.035
0017-9310/

its shape. For $w/k \approx 1$, the dependence of the total drag on the shape of the obstacles is large [9] with a consequent influence on the total heat flux [8].

For obstacles aligned with the flow directions experimental and numerical studies were focused to understand whether and under which circumstances drag reduction is possible. The experiments cited by [10] reported that the triangular shaped geometries are the most efficient, and, only, those of approximately 15 wall units in size reduce the drag. The numerical simulations, in particular that by [11], confirmed the experimental results, and asserted that the riblets reduce the drag by restricting the location of the streamwise vortices above the wetted surface, such that only a limited area of the surface is exposed to the downwash of high-speed fluid that the vortices induce. Orlandi et al. [9] considered triangles with two w/k, circular and square shaped longitudinal elements and compared the turbulence characteristics, in the near wall layer, with those of transverse bars. They observed that, in all cases, the viscous contribution to the total friction decreases and that the "form drag" increases. The "form drag", for these geometries with longitudinal straight grooves is related to the turbulent stress at the crests plane, that is due to the recirculating motion within the rough surface. Only few geometries could generate a weak motion in the cavities and therefore a small turbulent stress leading to a drag reduction. With regards to the heat transfer, there are few DNS and among them [12] considered the drag reducing triangular grooves.

In DNS of turbulent heat transfer in channels, for the thermal field, iso-flux or iso-thermal boundary conditions can be imposed at the walls. The latter set-up differs from that in real experiments, where the solid walls are thick and the constant temperature is maintained at the exterior of the walls. Temperature and heat flux at the interface between fluid and solid depend on the design of the experiment. In these circumstances the heat transfer strongly depends on the conductivity and thickness of the solid wall. Two smooth walls were considered by [13] in a channel with a constant heat source at the interior to investigate the dependence of the heat transfer with the Prandtl number. Orlandi et al. [14] studied the conjugate heat transfer in a plane channel at the fluid Prandtl number $Pr_F = \nu/\alpha_F = 1$ (ν is the kinematic viscosity and α_F is the thermal diffusivity of the fluid) for four different values of thermal diffusivity of the walls. For sake of simplicity and analogy with the fluid, the ratio between the viscosity of the fluid ν and the thermal diffusivity of the solid has been indicated with $Pr_{Si} = \nu/\alpha_{Si}$ and referred in the text as solid Prandtl number. In the [14] simulations one velocity field was transporting four passive scalars with Pr_{Si} set equal to $0.0134, 0.134, 10.5$ and 30.8 These values correspond to different materials: the first with $Pr_{Si} = 0.0134$ is an ideal material with conductivity ten times higher than copper ($Pr_{Si} = 0.134$) and the other two correspond respectively to marble ($Pr_{Si} = 10.5$) and glass ($Pr_{Si} = 30.8$).

In the present paper, the previous study has been extended to investigate the effects of the solid thermal diffusivity on the heat transfer for flows past rough walls. The channels have rough surfaces of different shape in one side, and smooth walls in the opposite side. The shapes of the rough surfaces are such to give higher friction or drag reduction, with respect to that of smooth walls. This numerical set-up can be reproduced in a laboratory and therefore the evaluation of any kind of statistics by the DNS may help in analysing the laboratory data, where difficulties in the measurements do exist. Near smooth walls high values of the correlations coefficients between temperature and streamwise velocity fluctuations have been observed. The choice of rough surfaces slightly or strongly perturbing the overlying turbulent flow allows to investigate whether the correlations coefficients remain high. If strong changes are not found, flow visualizations can still be used to detect the modifications of the high and low speed streaks. If the DNS, at low Reynolds numbers, demonstrate that high correlations are found in flows past rough surfaces, the flow visualizations in laboratories could give insights on the variations of the shape of the streaks at high Reynolds numbers. Orlandi et al. [14] for smooth walls evaluated the probability density function of the streamwise velocity component and that of the fluctuating temperature for the different Pr_{Si}, and observed a decrease of the correlation in presence of low conducting materials.

Direct Numerical Simulations solve all the scales of the flow from the energy containing to the Kolmogorov ones, albeit at large computational cost. By increasing the Reynolds number the smallest reduce in size and the DNS is unfeasible. In presence of smooth walls even the near wall energy containing scales reduce in size and the resolution is limiting the friction Reynolds number R_τ defined as $R_\tau = u_\tau h/\nu$. u_τ is the friction velocity, for smooth wall $u_\tau = \sqrt{\nu \partial U/\partial y}|_{y=0}$, y is the distance from the wall and h is half channel height. The number of grid points to have solutions at $R_\tau = 4000$ are given by [15]. For flows past rough walls the energy containing scales are linked to the shape of the geometry, therefore the influence of the Reynolds number on the friction is reduced, as it depicted in the Moody diagram [16]. At the intermediate Reynolds number the DNS allows to evaluate any statistics, which help to make closures in Large Eddy (LES) or Reynolds Averaged Navier Stokes (RANS) simulations of flows of practical interest at high Reynolds number. In the case of heat transfer a fundamental quantity for the closures is the turbulent Prandtl number Pr_T defined as the ratio between the eddy diffusivity of momentum ν_T and the eddy diffusivity of heat α_T later on defined. The different behavior of Pr_T near smooth and rough surfaces is described in the present paper, and, in particular for applications where the material of the solid layer plays an important role.

2. Numerical procedure

The incompressible non-dimensional Navier–Stokes and continuity equations are

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \Pi \delta_{i1} + \frac{1}{Re}\frac{\partial^2 U_i}{\partial x_j^2}; \quad \frac{\partial U_j}{\partial x_j} = 0, \tag{1}$$

where $Re = (U_P h/\nu)$ is the Reynolds number, U_P is the centreline laminar Poiseuille velocity, Π is the pressure gradient required to maintain a constant flow rate, U_i are the components of the velocity vector in the i directions, P is the pressure, x_1, x_2 and x_3 are the streamwise, wall-normal and spanwise directions respectively. A finite difference scheme combined with the immersed boundary method described in [17] is used. The equations are integrated in time with an implicit Crank Nicholson for the viscous terms, and a 3rd order Runge Kutta. In our previous papers the viscous terms were treated by an implicit procedure and for the MPI coding the computational box was subdivided in parallel layers to the walls. This assumption is limiting the use of a large number of processors, which, on the other hand, is possible by subdividing the domain in rectangular pencils. In this case the CPU time to transfer data among the processors increases therefore it is preferable to use an explicit scheme with a reduction of the CFL from 1.5 to 1.

The physical box has dimensions $L_1 = 8h$, $L_2 = 3.0h$ and $L_3 = 4h$. The origin in the vertical direction is at the centreline of the channel. The roughness is placed on the lower wall, the crests plane being at $x_2 = -1$. The upper smooth wall is at $x_2 = 1$. Both solid walls have a thickness equal to $0.5h$. The computational grid has $640 \times 256 \times 320$ points in x_1, x_2 and x_3 respectively. In the wall normal direction x_2, the non-uniform grid allows to have a better resolution at the interface between solid and fluid. The