International Journal of Thermal Sciences 120 (2017) 321–336

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

A PANS study of fluid flow and heat transfer from a square cylinder approaching a plane wall

Pritanshu Ranjan, Anupam Dewan*

Department of Applied Mechanics, Indian Institute of Technology Delhi, Hauz Khas, New Delhi 110016, India

ARTICLE INFO

Article history:
Received 4 July 2016
Received in revised form
6 June 2017
Accepted 16 June 2017

Keywords:
PANS
Turbulent heat transfer
Separation
Vortex shedding
Wall resolve
Cylinder near wall

ABSTRACT

Simulations were carried out to study the forced convection from a heated square cylinder placed near a plane wall boundary in a channel. The parameters that were studied are Reynolds number (Re) and the distance between the cylinder and the plane wall (gap ratio). At higher Re, turbulence modeling was performed using a dynamic PANS approach along with wall resolving approach. Simulations were carried out using a finite volume based open source software, OpenFOAM. The cylinder was kept at a constant temperature such that the difference in temperature between the incoming velocity and cylinder was small enough to treat temperature as a scalar quantity. For $Re = 22000$ the critical value of the gap ratio was found to be 0.3 at which the vortex shedding was completely suppressed. The face averaged value of the Nusselt number for the back face dropped below the front face value for the gap ratio lower than 0.5. The effect of Re was also studied for the gap ratio of 0.4, and it was observed that up to $Re = 1000$, increase in total Nusselt number from the cylinder was linear but as the wake became unsteady and turbulent the Nusselt number increased exponentially with Re. Three different flow regimes were observed in terms of Re and are investigated in detail for the heat transfer associated with it.

1. Introduction

Fluid flow and heat transfer from a bluff body has always remained a topic of intense research. It presents a very challenging problem for academicians because of various complex fluid mechanical characteristics involved. Bluff bodies with rectangular shapes often find a practical application in various areas, such as, heat exchangers, electronic cooling, civil and mechanical structures, etc. These type of structures involve large-scale vortex formation due to flow separation from sharp corners of rectangular faces. The heat transfer mechanism highly depends on these unsteady large-scale vortices. In the past decade many researchers have studied, experimentally or computationally, unconfined free standing rectangular/square bluff bodies. The flow behavior around these bodies changes significantly in the presence of external disturbances, such as, a solid surface or wall. This behavior is due to interaction between the separated shear layer from the rectangular body and shear layer formed on the external solid surface. Due to this interaction, vortices shed from the bluff body are asymmetric and of unequal strength and this asymmetry increases as the cylinder approaches the wall. The present paper deals with the study of flow over infinite heated square cylinder in the presence of a plane solid wall.

In the past, many researchers have studied the influence of wall on the flow dynamics of square cylinders. Durao et al. [1] were the first to study the effect of gap between the square cylinder and the wall on Strouhal number (St). They used Laser Doppler Velocimetry to conduct the experiments at Reynolds number (Re) = 13600 and observed that the vortex shedding was completely suppressed for a gap to cylinder height ratio (G/D) = 0.35. Further [2,3] performed experiments and simulations, respectively, for a square cylinder placed near a wall at $Re = 22000$. They both observed that the critical gap width (G/D) was in the range of 0.35–0.5. Liou et al. [4] first presented 3D large eddy simulation (LES) of turbulent flow over a square cylinder located near a solid wall at $Re = 2.2 \times 10^4$. They showed that, due to the presence of solid wall, the celerity of vortex shed from the side facing the wall is smaller than that of the other side of the cylinder. Further experimental studies were presented by Refs. [5,6] to consider the influence of the wall proximity on the vortex shedding from a square cylinder at $Re = 2.2 \times 10^4$. Bailey et al. [5] concluded that at $G/D = 0.4$ the vortex shedding is suppressed due to the inability of the shear layer between cylinder and wall to interact with the instabilities present in the upper shear

* Corresponding author.
E-mail address: adewan@am.iitd.ac.in (A. Dewan).

http://dx.doi.org/10.1016/j.ijthermalsci.2017.06.015
1290-0729/

layer. Martinuzzi et al. [6] reported that the four gap height (G/D) dependent flow regimes exist and below $G/D = 0.3$ the periodicity of the vortex shedding is completely absent. Many other researchers have also performed numerical studies for e.g. Refs. [7–9] used the ε-SST model and [10] used the second moment closure model for a 2D simulation of square cylinder placed near a wall at $Re = 2.2 \times 10^4$ and reported similar results as those by experimental studies.

All the studies mentioned above are in the range of Re, based on the cylinder diameter, equal to 10^4. In the low Re regime, Hwang and Yao [11] numerically studied the effect of ground on shedding from a 2D square cylinder in Re ranging from 500 to 1500 and found that the vortex shedding is suppressed below $G/D = 0.5$ and similarly Bhattacharya and Maiti [12] also performed 2D numerical simulations for $Re = 40$. Kumaran and Vengadesan [13] and Mahir [14] performed 3D numerical simulations for a square cylinder placed near a wall in the same Re regime (250–450) and found that no periodicity can be observed for $G/D \leq 0.3$. Recently Shi et al. [15] and Liu et al. [16] performed PIV measurements over a square cylinder for gap ratios in the range of 0.1–0.8 at $Re = 2250$ and 13200, respectively. They used Proper Orthogonal Decomposition (POD) technique to study the energy associated with the wake structures. Samani and Bergstrom [17] also used POD in the numerical investigation of wake dynamics affected by a wall for an infinite square cylinder. They performed 3D LES simulations at $Re = 500$ and showed that as the wall is approached, the wake flow pattern becomes highly asymmetric and the values of lift and drag coefficients increase and decrease, respectively.

However all the above mentioned studies deal with the fluid flow only and did not investigate the heat transfer in the near wake region of a square cylinder. Rosales et al. [18] investigated the convective heat transfer in laminar flow ($Re = 500$) past a single and tandem pair of square cylinders confined in a channel. Similarly Sharma and Eswaran [19] studied the combined effect of buoyancy and channel confinement on 2D heat transfer from a square cylinder for $Re = 100$. Bhattacharyya et al. [20] numerically studied the heat transfer from a square cylinder placed near a wall ($G/D = 0.5$) over Re range of 100–250 and Grashof number (Gr) range of 0–10^5. However, all these heat transfer studies are in laminar flow regions and cannot be applied to 3D turbulent flow regimes. Recently He et al. [21] performed experiments for a square cylinder located near a heated plane wall. They used PIV and TSP techniques for flow and heat transfer measurements, respectively. The gap ratio was varied from 0 to 0.75 and Re from 3800 to 11400. The results of He et al. [21] are also used in the present study for the validation purpose. Still, there is a lack of studies dealing with the combined effects of turbulence and wall on heat transfer characteristics from a square cylinder.

In the present study, we carry out 3D numerical simulation of heat transfer from a square cylinder placed asymmetrically in a channel near a solid boundary. A variable resolution model, Partially-Averaged Navier-Stokes Method (PANS) given by Girimaji [22], was used for solving the turbulent flow. Its SST $k-\omega$ variant which was used in our previous studied [23,24] is employed in the present study.

2. Modeling formulation

An incompressible flow field is assumed to study the heat transfer from a square cylinder placed near a wall. The temperature difference between the hot cylinder and the free stream flow is small enough to neglect the effect of buoyant forces on the flow. To model turbulence, a variable resolution modeling approach, namely, Partially-Averaged Navier-Stokes (PANS) approach, first given by Girimaji [22] is used. The details of the modeling approach are presented in the subsequent sections.

2.1. Partially-Averaged Navier-Stokes (PANS) approach

Considering the flow to be incompressible the governing equations for turbulent flow can be written as

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + U_j\frac{\partial U_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu\frac{\partial^2 U_i}{\partial x_j \partial x_j} - \frac{\partial \tau(V_i, V_j)}{\partial x_j} \tag{2}$$

where, a general arbitrary filter is used to decompose the instantaneous velocity field, V_i, which can commute with spatial and temporal differentiations. The decomposed velocity is given by $V_i = U_i + u_i$, where U_i denotes the filtered/resolved velocity and u_i its fluctuating part which requires modeling. The term $\tau(V_i, V_j)$ is the sub filtered stress (SFS) and is given by $\tau(V_i, V_j) = \langle V_i V_j\rangle - \langle V_i\rangle\langle V_j\rangle$. In the PANS approach the SFS is modeled using the Boussinesq assumption which states that

$$\tau(V_i, V_j) = -\nu_u\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \frac{2}{3}k_u\delta_{ij} \tag{3}$$

$$\nu_u = C_\mu \frac{k_u^2}{\varepsilon_u} \tag{4}$$

where, ν_u, k_u and ε_u are the PANS eddy viscosity, unresolved turbulent kinetic energy and unresolved turbulent dissipation, respectively. To close the system of equations (1)–(4) model equations for k_u and ε_u are needed. Initially [22,25] used the standard k-ε model as the base model to derive the transport equations for k_u and ε_u. Many other researchers have also developed other variants to close the system of equations (1)–(4). Basara et al. [26] derived the PANS model using the k-ε-ζ-f model. The model given by Ref. [26] was further used by other researchers [27–29] to study the flow field over bluff body. In our previous studies [23,24] we used the SST k-ω model given by Menter [30] to derive the PANS model and the same modeling strategy is employed in the present study. The corresponding governing equations are

$$\frac{\partial(\rho k_u)}{\partial t} + \frac{\partial(\rho U_j k_u)}{\partial x_j} = \overline{P_{ku}} - \beta^*\rho\omega_u k_u + \frac{\partial}{\partial x_j}\left(\Gamma_{ku}\frac{\partial k_u}{\partial x_j}\right) \tag{5}$$

$$\frac{\partial(\rho\omega_u)}{\partial t} + \frac{\partial(\rho U_j\omega_u)}{\partial x_j} = \frac{\gamma}{\nu_u}\overline{P_{ku}} - \left(\frac{1}{f_\omega} - 1\right)\frac{\gamma\beta^*}{\nu_u}\omega_u k_u - \frac{\beta\rho\omega_u^2}{f_\omega} + \frac{\partial}{\partial x_j}\left(\Gamma_{\omega u}\frac{\partial\omega_u}{\partial x_j}\right) + (1 - F_{1u})2\rho\sigma_{\omega 2}\frac{f_\omega}{f_k}\frac{1}{\omega_u}\frac{\partial k_u}{\partial x_j}\frac{\partial k_u}{\partial x_j} \tag{6}$$

The model coefficients are

$$\Gamma_{ku} = \mu + \frac{\mu_u}{\sigma_k}\frac{f_\omega}{f_k},\quad \Gamma_{\omega u} = \mu + \frac{\mu_u}{\sigma_\omega}\frac{f_\omega}{f_k}, P_{ku} = \tau_{xy}\frac{\partial U_i}{\partial x_j},\quad \overline{P_{ku}} = \min(P_{ku}; c_l\varepsilon_u) \tag{7}$$

$$F_{1u} = \tanh\left(\arg_1^4\right), \arg_1 = \min\left(\max\left(\frac{\sqrt{k_u}}{\beta^*\omega_u y}; \frac{500\nu}{y^2\omega_u}\right); \frac{4\rho\sigma_{\omega 2}k_u}{CD_{k\omega u}y^2}\right) \tag{8}$$