International Journal of Heat and Fluid Flow 29 (2008) 1309–1318

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Numerical investigation of convective heat transfer in unsteady flow past two cylinders in tandem arrangements

Necati Mahir̆, Zekeriya Altaç *

Eskişehir Osmangazi University, School of Engineering and Architecture, Mechanical Engineering Department, 26480 Batı Meşelik, Eskişehir, Turkey

ARTICLE INFO

Article history:
Received 10 August 2007
Received in revised form 19 December 2007
Accepted 6 May 2008
Available online 20 June 2008

Keywords:
Heat transfer
Numerical simulation
Tandem arrangement
Flow past cylinder
Fluent

ABSTRACT

Unsteady laminar convective heat transfer from two isothermal cylinders of tandem arrangement is numerically investigated. The numerical simulations were carried out by the commercial CFD software—FLUENT®. The working fluid is air. The analysis is carried out for the Reynolds numbers of 100 and 200 and for center-to-center distance ratio, L/D, of 2, 3, 4, 5, 7 and 10. The flow parameters such as the lift and drag coefficients and Strouhal number are also obtained and compared with those of available in the literature. The vorticity and isotherms were generated to interpret the flow and heat transport visualization. The mean and local Nusselt numbers for the upstream and downstream cylinders were obtained. It is found that the mean Nusselt number of the upstream cylinder approaches to that of an single isothermal cylinder for $L/D > 4$ and the mean Nusselt number of the downstream cylinder is about 80% of the upstream cylinder.

1. Introduction

Forced convection heat transfer around circular cylinders has numerous applications in engineering practice such as heat exchangers, space heating, flow around arrays of nuclear fuel rods, heat losses from high-rise buildings, chimneys, power generators, and other thermal applications.

The studies involving flow over two cylinders have been the subject of numerous experimental and numerical work in the last two decades. Both the flow field and force coefficients depend highly on the configuration and the spacing of the cylinder pair due to both wake and proximity-induced interference effects. Some of the earliest experimental studies on cylinder pairs in a tandem orientation that were carried out by Kostic and Oka (1972), Tanida et al. (1973) and King and Johns (1976) demonstrated the presence of two major flow regimes with a complex transition region between them. For closely spaced cylinders, the flow separates behind the first cylinder and reattaches to the second one while, for larger spacing, vortex shedding occurs behind both cylinders.

Extensive reviews of results in tandem, side-by-side and/or staggered arrangements can be found in the published works of Zdravkovich (1977, 1985), Chen (1987), Blevins (1990) and Sumner et al. (2000). Critical spacing, which is defined as the minimum gap between the walls of the cylinders, separating different flow regimes was established for tandem arrangements as well. In tandem arrangement, the numerical studies also confirm the experimental findings; that is, if the gap is greater than the critical spacing ($\sim 3.5D - 3.8D$), the upstream cylinder sheds vortices onto the downstream cylinder. Alternatively, vortex shedding does not occur from the upstream cylinder if the gap is less than the critical spacing.

Mittal et al. (1997) numerically studied incompressible flows past a pair of cylinders at Reynolds numbers 100 and 1000 in tandem and staggered arrangements using a stabilized finite element formulation. In tandem configuration, the center-to-center distance ratio (L/D) was kept at 2.5 and 5.5. The Strouhal number, based on the dominant frequency in the time history of the lift coefficient, for both cylinders was the same. Farrant et al. (2001) studied two-dimensional unsteady incompressible flows around circular cylinders at Reynolds number 100 and 200. A hybrid boundary element method was used to discretize the spatial domain together with a second order implicit finite difference approximation in time. Meneghini et al. (2001) also numerically studied the two-dimensional shedding of vortices and flow interference between two circular cylinders in tandem and side-by-side arrangements. The simulations were performed for Reynolds numbers from 100 to 200. Kondo and Matsukuma (2005) numerically studied two- and three-dimensional flow around two circular cylinders in tandem arrangement for $Re = 1000$. The center-to-center distance between the two cylinders was $2D$ to $5D$. They observed the critical distance to be between $3.5D$ to $4D$. The numerical results were qualitatively compared with experimental data. Carmo and Meneghini (2006) investigated two- and three-dimensional simulations of the incompressible flow around a pair of circular cylinders in tandem arrangements using the spectral element

* Corresponding author. Tel.: +90 222 2393750x3365; fax: +90 222 2393613.
E-mail addresses: nmahir@ogu.edu.tr (N. Mahir̆), zaltac@ogu.edu.tr (Z. Altaç).

0142-727X/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.ijheatfluidflow.2008.05.001

method. The centre-to-centre distance of the investigated configurations was varied from 1.5D to 8D. The simulations were in the Reynolds number range from 160 to 320. Ding et al. (2007) numerically studied the flow field around two circular cylinders arranged in side-by-side and tandem configuration (L/D = 2.5 and 5.5) using the mesh-free least square-based finite difference method. The flow simulations were carried out for Re = 100 and 200.

As to the heat transfer from the cylinders, Jue et al. (2001) investigated the convection heat transfer of flow across three heated cylinders arranged in an isosceles right-angled triangle between two parallel plates. The variations of the drag coefficient and time-averaged Nusselt number around the surface of the three cylinders were investigated. In their study, L/D ratio of 0.5–1.25, Reynolds number of 100–300 and Grashoff number of 80,000–200,000 were considered. Buyruk (2002) numerically studied the heat transfer from two isothermal tandem cylinders in cross flow of air at Re = 400. In the study, three staggered isothermal cylinder and four inline isothermal cylinder configurations were also investigated for Re = 80, 120 and 200. Rahnama and Hadi-Moghaddam (2005) numerically investigated 2D unsteady laminar flow past a heated square cylinder mounted inside a plane channel. The blockage ratio was chosen as 1/8 and the Reynolds number was less than 200. The time-averaged Nusselt number, drag coefficient, recirculation length, and Strouhal number were obtained and compared with the literature. Zhou and Yiu (2006) experimentally investigated the flow structure, momentum and heat transport in the wake of two tandem circular cylinders involving the heated upstream cylinder. Their measurements were, however, conducted for Re = 7000 and for L/D = 1.3, 2.5, 4, 6, 10, 20 and 30. They showed that the cross-stream distributions of the Reynolds stresses and heat fluxes at a given x/D vary from one to another. The momentum and heat transport characteristics were then summarized for each flow structure.

It is known that vortex shedding occurs in the wake of a circular cylinder if the Reynolds number is sufficiently large. The vortex shedding affects the heat transfer characteristics of a cylinder. However, the combined wake behind multiple cylinders is different from that of a single circular cylinder; thereby, the influence of the flow interaction between multiple cylinders on heat transfer needs to be further investigated. Thus, the purpose of this study is to numerically investigate the convection heat transfer characteristics of two cylinders of equal diameter in tandem arrangements subject to cross flow of air. The local Nusselt numbers and mean Nusselt numbers for the upstream and downstream cylinders are computed and analyzed with respect to the flow patterns.

2. Governing equations

The governing equations, for unsteady incompressible viscous flow, are

for continuity

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0, \tag{1}$$

for the momentum

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right), \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right), \tag{3}$$

and for the energy

$$\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right), \tag{4}$$

where u and v are the velocity components, p is the pressure, ν is the kinematic viscosity, ρ is the density, T is the temperature of the fluid, and α is the thermal diffusivity defined as $k/\rho c$ where k is the thermal conductivity and c is the specific heat of the fluid.

The computational domain and the configuration of the cylinders are illustrated in Fig. 1. The boundary conditions can be stated as:

For the inlet, $u = U_\infty$, $T = T_\infty$,
For the outlet, $\frac{\partial u}{\partial x}=0,\ \frac{\partial v}{\partial x}=0,\ \frac{\partial T}{\partial x}=0$,
For the top and bottom, $u = U_\infty$, $v = 0$, $T = T_\infty$,
For the cylinders walls, $u = 0$, $v = 0$, $T = T_w$,
where U_∞ and T_∞ are the free-stream velocity and temperature and T_w is the cylinder wall temperature which is constant.

The lift and drag coefficients are computed from

$$C_L=\frac{2F_y}{\rho D U_\infty^2},\qquad C_D=\frac{2F_x}{\rho D U_\infty^2}, \tag{5}$$

where D is the cylinder diameter, F_x and F_y are the force components resolved in the directions x and y.

The Reynolds number is defined as $Re = U_\infty D/\nu$. On the other hand, the local heat transfer coefficient and the local Nusselt number are computed from

$$-k\frac{\partial T}{\partial n}\bigg|_w=h_\theta(T_w-T_\infty),\qquad Nu_\theta=\frac{h_\theta D}{k}, \tag{6}$$

where n is the direction normal to the cylinder surface; h_θ and Nu_θ are the local heat transfer coefficient and local Nusselt number, respectively; and θ is the polar angle measured in usual counterclockwise direction with respect to cylinder center.

The surface-averaged (mean) Nusselt number is then evaluated as follows:

$$Nu=\frac{1}{2\pi}\int_{\theta=0}^{2\pi}Nu_\theta\,d\theta, \tag{7}$$

3. Numerical verification

The governing flow and energy equations subjected to the aforementioned boundary conditions are solved using a commercial CFD package—FLUENT®. FLUENT® is capable of handling unsteady Navies-Stokes and energy equations using a finite volume method in two-and three-dimensional geometries (Fluent, 2003).

The first case for the verification of FLUENT simulations is the heat transfer problem of a single isothermal cylinder subject to cross flow of air (Pr = 0.7). The number of grid points and their distribution is an important matter in such unsteady laminar flow over cylinders because of the complex phenomena existing in this type of flow, such as separation and vortex shedding. The compu-

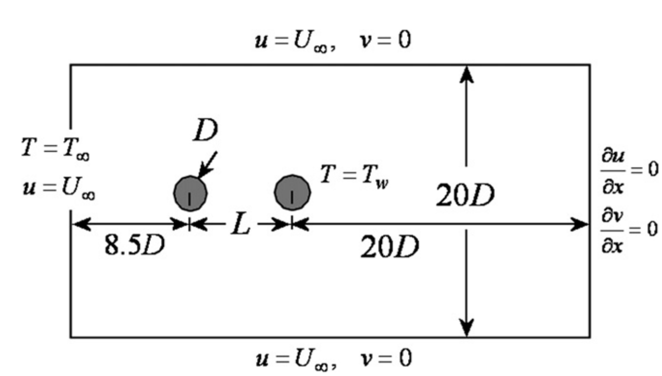

Fig. 1. Computational domain and two cylinder configuration.