International Journal of Heat and Mass Transfer 126 (2018) 521–535

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Constructal Design of triangular arrangements of square bluff bodies under forced convective turbulent flows

F.B. Teixeira [a], G. Lorenzini [b,*], M.R. Errera [c], L.A.O. Rocha [d], L.A. Isoldi [a], E.D. dos Santos [a]

[a] Graduate Program of Ocean Engineering, School of Engineering, Universidade Federal do Rio Grande, Itália Avenue, km 8, Rio Grande 96201-900, Brazil
[b] Università degli Studi di Parma, Dipartimento di Ingegneria e Architettura, Parco Area delle Scienze 181/A, 43124 Parma, Italy
[c] Environmental Engineering Department, Universidade Federal do Paraná (UFPR), Cel. Francisco H. dos Santos Street, 210, Curitiba, Paraná 81531-980, Brazil
[d] Mechanical Engineering Graduate Program, Universidade do Vale do Rio dos Sinos – UNISINOS, São Leopoldo, RS, Brazil

ARTICLE INFO

Article history:
Received 5 March 2018
Received in revised form 24 April 2018
Accepted 26 April 2018

Keywords:
Constructal Design
Turbulent flow
Forced convection
Bluff bodies
Numerical study

ABSTRACT

Present numerical work consists on geometric evaluation of turbulent, transient, two-dimensional, incompressible and forced convective flows over triangular arrangements of square bluff bodies of size D employing Constructal Design. The main purpose is to evaluate the influence of the geometry of the arrangement over drag coefficient (C_D) and Nusselt number (Nu_D), i.e., multiobjective problem. It is also investigated the influence of array configuration over fluid dynamic and thermal behavior of the flow. The problem has two constraints: cross-sectional area of bodies ($3D^2$) and occupation area of the array ($6D \times 6D$) and two degrees of freedom: S_l/D and S_T/D (ratios between longitudinal and transversal pitchs and size of the bodies). For all simulations, Reynolds and Prandtl numbers are constant (Re_D = 22,000 and Pr = 0.71). Time-averaged conservation equations of mass, momentum and energy are solved with Finite Volumes Method. Closure of turbulence is solved with RANS *SST* – κ-ω modeling. Results showed a strong influence of design over fluid dynamic and thermal performance of the problem, as well as, multiplicity of scales and patterns of turbulent flows. Moreover, the optimal multi-objective configuration was the same reached for fluid dynamic purpose, showing the dominance of this purpose in the present investigation.

1. Introduction

Forced convective flows are encountered in several engineering applications. Consequently, many experimental and numerical works have been carried out to improve the comprehension about this phenomenon [1–4]. Among the studied problems, the evaluation of convective flows over arrangement of tubes, bluff bodies or even finned channels has been highly prominent since this kind of domain represents ideally several thermal systems as heat exchangers, condensers, evaporators, exhaust fans, automotive radiators, cooling of electronic components and industrial equipment [5–8]. According to Wilcox [6] many of above mentioned applications are characterized by turbulent flows. In spite of this fact, geometrical evaluations related with cylinders or bluff bodies are dominantly performed for laminar flows due to high physical complexity and expensive numerical and experimental costs related with approach of turbulent flows.

The study of convective flows over only one bluff body has been still investigated in literature for a better comprehension of the physical problem and estimative of mean parameters. Into experimental framework, Igarashi [1] studied a square bluff body subjected to forced convective turbulent flows for a series of Reynolds numbers ($1.11 \times 10^4 < Re_D < 5.19 \times 10^4$). The objective was to determine the heat transfer coefficients (h) of the bluff body at various angles of incidence ($0° \leq \alpha \leq 45°$) between the flow and the obstacle. Durao et. al. [9] performed experiments with a Laser-Doppler measurement device to analyze the velocity characteristics of a turbulent flow at Re = 14,000 around a square bluff body. The main purpose was to separate and quantify the periodic turbulent and non turbulent characteristics. Similarly, Lyn et al. [2] also performed Laser-Doppler measurements of a mixed convection turbulent external flow at Re = 21,400 over a square bluff body with the objective to improve the comprehension about fluid dynamic and thermal fields.

Into the numerical realm, several aspects as high computational cost caused by complex physical phenomena and generation of multiple spatial and temporal scales for the flow has justified the evaluation of only one bluff body or circular cylinder, as seen in Refs. [10–12]. For instance, Wiesche [12] used LES (Large Eddy Simulation) approach associated with the sub-grid scale model of Smagorinsky [13] to study by means of Computational Fluid

* Corresponding author.
E-mail address: giulio.lorenzini@unipr.it (G. Lorenzini).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.04.134
0017-9310/

Nomenclature

A_0	occupation area of the bluff bodies (m^2)
A_t	cross-sectional area of the three bluff bodies (m^2)
C_D	drag coefficient ($C_D = 2F_D/(\rho A V_\infty^2)$)
c_p	specific heat at constant pressure (J/(kg·K))
D	bluff body side (m)
F_1	blend function 1 of *SST* κ–ω turbulence model
F_2	blend function 2 of *SST* κ–ω turbulence model
g	gravitational acceleration (m/s^2)
h	heat transfer coefficient ($W/(m^2{\cdot}K)$)
H	height of computational domain (m)
H_0	height of bluff bodies occupation area (m)
H_1	distance between upstream bluff body center and lateral surfaces of domain (m)
k	thermal conductivity of the fluid (W/(m·K))
L	length of computational domain (m)
L_0	length of bluff bodies occupation area (m)
L_1	distance between inlet surface and the center of upstream bluff body (m)
Nu_D	Nusselt number of bluff body ($Nu_D = h{\cdot}D/k$)
P	pressure (N/m^2)
Pr	Prandtl number ($Pr = \mu c_p/k$)
q''	heat flux (W/m^2)
Re_D	Reynolds number as a function of side of bluff body ($Re_D = \rho V_\infty D/\mu$)
S_L	longitudinal pitch (m)
S_T	transversal pitch (m)
T_∞	temperature of the fluid at the free stream (K)
u	x-axis fluid velocity (m/s)
u_*	friction velocity ($(\tau/\rho)^{1/2}$) (m/s)
v	y-axis fluid velocity (m/s)
V_∞	velocity at the free stream (m/s)
x	spatial coordinate in free stream direction (m)
y	spatial coordinate in normal direction to free stream

Greek symbols

α	thermal diffusivity (m^2/s)
β	ad hoc constant of *SST* κ–ω turbulence model
Δi	ideal point distance ($\Delta i = (C_D(\text{norm})^2 + Nu_D(\text{norm})^2)^{1/2}$)
κ	turbulent kinetic energy (m^2/s^2)
μ	dynamic viscosity (kg/(m·s))
υ	kinematic viscosity (m^2/s)
ρ	density (kg/m^3)
σ	ad hoc constant of *SST* κ–ω turbulence model
μ_T	turbulent viscosity (m^2/s)
ω	specific dissipation rate 1/s

Subscripts

m	once maximized or minimized
mm	twice maximized or minimized
o	once optimized
oo	twice optimized
f	fluid dynamic objective
t	thermal objective

Dynamics (CFD) a turbulent flow of air ($Re = 22{,}000$ and $Pr = 0.71$) over a heated square cylinder. Results for thermal fields are compared with those obtained in Refs. [2,14–16]. Perng and Wu [17] studied mixed convective turbulent flows for a heated bluff body subjected to a vertical air channel employing LES modeling. For all cases, it is considered a flow with $Re = 5000$ and $Pr = 0.71$. The dimensions of the obstacle were varied to increase the blockage ratio of the flow. It was also studied the effects of aided and opposed buoyancy over the heat exchange for various Richardson numbers varying in the range ($-1.0 \le Ri \le 1.0$). Turbulence closure models and numerical procedures have also been investigated for this kind of flow. Ranjan and Dewan [18] carried out comparative studies between two different mesh generation strategies, one structured with wall function and one unstructured mesh with structured and refined mesh in near wall region. A numerical model based on PANS (Partially-Averaged Navier-Stokes) SST $\kappa{-}\omega$ was evaluated to solve the same problem of external turbulent flow with forced convection and compared with previous experimental and numerical works of Refs. [1,2,9,12]. More recently, Chen and Xia [19] studied a forced convective turbulent flow at $Re = 22{,}050$ over a bluff body of square cross-section with an identical computational domain to that used in [18]. The authors studied fluid dynamics and thermal behavior comparing results obtained with RANS (Reynolds-Averaged Navier Stokes) models of Spalart-Allmaras (SA) and SST κ–ω and hybrid LES-RANS model with unstructured meshes. Into the evaluation of size of one bluff bodie, Ranjan and Dewan [20] studied the effects of bluff body sides ratio over the phenomena of separation, detachment and reattachment of boundary layers, as well as, vorticity and stagnation of fluid flow.

Concerning the geometrical evaluation of cylinders or bluff bodies arrangements, Lam et al. [21] performed two and three-dimensional simulations of isothermal flows through an array of four circular cylinders aligned in square arrrangement. Laminar flows were studied for two Reynolds numbers, Re_D = 100 and 200, being investigated the effect of the ratio between the pitch and diameters of cylinders over the velocity fields, as well as, two and three dimensional structures generated in the analyzed flows. In sequence, Lam and Zou [22] continued the studies of [21], evaluating numerically and experimentally turbulent flows with Reynolds numbers in the range $11{,}000 \le Re_D \le 20{,}000$. The numerical study was carried out with LES approach, while for the experimental work the distribution of mean and fluctuating velocity fields were measured with the aid of Laser Doppler anemometer in a closed-loop water channel. A Digital Particle Image Velocimetry (DPIV) system was also used to characterize the distribution of velocity field, vorticity and other turbulence properties. Other important work was made by Sumner [23], which conducted an extensive review about flows on pairs of cylinders in aligned and staggered arrangements. The study presented an analysis of a series of works, e.g. [24–29]. The work also showed the effect of the distance between the pair of cylinders, ratio between cylinder pitch and diameters and Reynolds number on mean flow parameters such as Strouhal number (St_D), drag coefficient (C_D), lift (C_L) and Nusselt number (Nu_D). Recently, Salcedo et al. [30] carried out an experimental study on confined cylinders with constant high temperature subjected to laminar mixed convection flow. The cylinders were aligned longitudinally to the direction of the flow in a water channel and the parameters acquired through a particle image velocimetry (*PIV*) equipment. Some characteristics of the flow behavior was analyzed such as the fluid velocity fluctuation along the flow, the Strouhal number and the vortex shedding according to the Richardson number, that was varied in the range $-1.0 \le Ri \le 3.0$. Despite of several important contributions done in literature about geometrical evaluation in arrangements of cylinders and bluff bodies, few have been noticed about application of Constructal Design for investigation of external convective turbulent flows.