International Journal of Heat and Mass Transfer 128 (2019) 161–170

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Experimental and numerical study on the heat transfer downstream of a confined rectangular cylinder in the laminar regime

M. Meis [a], F. Varas [b], J.R. Arias [c], A. Velazquez [c,*], L. Carretero [c]

[a] Fluid Mechanics Group, School of Industrial Engineering, Universidad de Vigo, Campus Lagoas Marcosense, 36310 Vigo, Pontevedra, Spain
[b] Applied Mathematics Department, School of Aerospace Engineering, Universidad Politécnica de Madrid, Plaza del Cardenal Cisneros 3, 28040 Madrid, Spain
[c] Fluid Mechanics and Aerospace Propulsion Department, School of Aerospace Engineering, Universidad Politecnica de Madrid, Plaza del Cardenal Cisneros 3, 28040 Madrid, Spain

ARTICLE INFO

Article history:
Received 15 January 2018
Received in revised form 20 August 2018
Accepted 23 August 2018

Keywords:
Heat transfer
Vortex promoter
Laminar flow
Stagnation chamber

ABSTRACT

This article presents an experimental and numerical study on the heat transfer behavior in the channel flow past a confined rectangular cylinder at (relatively) low Reynolds number. The work is motivated by a previous article by the authors (Meis et al. (2010)) that illustrated through a 2D numerical model that a significant heat transfer improvement can be achieved for some particular configurations of the cylinder (suggesting an interesting guideline for the design of micro channel heat exchangers). The objective of the present study was to make a comparison between the results of the actual experimental tests (using a setup as close as possible to a practical micro heat exchanger device) and the previous numerical results of the 2D simplified model, and to understand the differences. The experimental setup consisted of a rectangular cross section channel in which a cylinder with rectangular cross section was placed. The blockage ratio was 2:1, and the channel aspect ratio in the span-wise direction was 17:1. A heated aluminum block in the stream-wise direction was inserted in the lower channel wall right downstream of the rectangular cylinder. A mechanical device was implemented to allow the rectangular cylinder to be inclined at will with respect to the incoming flow. A stagnation chamber, representative of an actual practical engineering situation, was placed upstream of the channel entrance. Three different flow Reynolds numbers (Re) based on twice the channel cross-section height (the channel hydraulic diameter) were tested: 200, 400, and 600. The Nusselt number (Nu) was measured as a function of Re and the rectangular cylinder inclination angle. Regarding the results obtained, it was found that the hysteretic heat transfer behavior that was present in some previous 2D simulations and was responsible for a remarkable heat transfer enhancement did not appear in the experimental tests. This difference can be ascribed to the fact that idealized inflow boundary conditions were used in the 2D computations while an actual stagnation chamber with finite dimensions was used in the experiments. This is corroborated by 3D numerical simulations of the experimental setup, showing that heat transfer enhancement (relative to a parallel channel flow) is associated with vorticiy generated in the stagnation chamber. In addition, 3D numerical simulations showed that further modifications in the stagnation chamber design did not change significantly these results. Finally, obvious as it may seem, it is worth highlighting that micro channel heat exchanger design guidelines based on 2D models should be considered with much care, since flow in realistic configurations can be dominated by effects associated to the overall device design.

1. Introduction

The present 3D experimental and numerical study is motivated by the results of a series of previous 2D numerical articles that deal with the effect that obstacles placed in confined flow at low Re may have on heat transfer. Some representative examples are those of Turki et al. [1], Rahnama and Moghaddam [2], Cheraghi et al. [3], and Meis et al. [4] that considered 2D laminar flows and obstacles of different cross-sections.

In reference [1], the authors considered laminar confined air flow past a heated (uniform surface temperature) square cylinder. Re, based on the side length of the square cylinder varied between 62 and 200. Two different blockage ratios (1/4 and 1/8) were considered. The channel inlet section, where numerical boundary conditions were applied, was located at a distance of 10 cylinder side lengths upstream of the cylinder itself. At this section, the inlet velocity was prescribed to be of parabolic shape; i.e. a fully

* Corresponding author.
E-mail address: angel.velazquez@upm.es (A. Velazquez).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.08.113
0017-9310/

Nomenclature

A_{wall}	Area of the heated aluminum block
C_p	Specific heat transfer
k	Water thermal conductivity
g	Gravity acceleration
G	Gain factor
Gr	Grashof number
h	Convection heat transfer coefficient
H	Channel height
L_e	Entrance length
Nu	Nusselt number
p	Pressure
P	Thermal power
Pr	Prandtl number
Q	Flow rate
Q_f	Q factor
r_h	Hydraulic radius
Re	Reynolds number
t	time
T	Temperature
T_{in}	Channel inlet flow temperature
T_{out}	Channel outlet flow temperature
T_{wall}	Temperature of the heated aluminum block
u	Velocity vector
U_{mean}	Mean flow velocity in the channel

Greek letters

λ	Expansion coefficient
μ	Water viscosity
ρ	Water density
θ	Rectangular cylinder inclination angle
ω	Vorticity

hydrodynamically developed inlet profile was considered. In practice, this means that: (a) inlet conditions implicitly assumed the presence of a very long inlet channel length where the flow could be allowed to reach fully developed conditions, and (b) inlet conditions where not perturbed by the presence of the obstacle itself. Under these assumptions, the authors found that average Nu on the surface of the square cylinder grows along with Re raised the power of 0.3, and that it is very slightly affected, only, by the blockage ratio. In practical terms, this means that doubling Re implies multiplying Nu by a factor of 1.23. In reference [2], a similar problem (2D confined flow past a square cylinder) was addressed. Re, based on the square cylinder side length, was varied between 10 and 200. The blockage ratio was 1/8. The inlet fully developed velocity profile was placed 5 cylinder side lengths upstream of the cylinder itself. The main difference with reference [1] was that, in this case, a constant heat flux was prescribed in the square cylinder surface instead of a constant temperature. After redefining Nu to account for this new boundary condition, the authors found that Nu scaled with Re raised to the power of 0.4 that is consistent with the result provided in reference [1].

The problem considered in reference [3] belonged to the same family but the rectangular cylinder section was circular, the rectangular cylinder surface was adiabatic, and walls were heated with a constant heat flux. Although the authors studied channel heat transfer behavior as a function of the rectangular cylinder distance to the channel walls, their parametric study included the case in which the rectangular cylinder was centred in the channel axis. All simulations were performed at Re 100. The blockage ratio was 1/3. Pr was varied between 0.1 and 10, including the case Pr = 1 that is closest to the cases presented in references [1,2]. Fully developed parabolic flow was prescribed at the computational domain inlet section located 4 diameters upstream of the rectangular cylinder. The authors compared their results with the case of the clean channel (no rectangular cylinder present) that was considered as their reference configuration. For the case of the centred rectangular cylinder at Pr = 1, the authors reported an average thermal gain factor (averaged Nu with rectangular cylinder divided by averaged Nu without rectangular cylinder) of 1.1 (a 10% increase). Interestingly enough, the authors, in their conclusions section, stated that the use of vortex generators is recommended for gas flows only (Pr < 1).

In reference [4], a 2D numerical study was presented on the effect that the shape of a vortex promoters has on the heat transfer and pressure drop in a confined laminar water flow. Different shapes (triangular, rectangular, square, and elliptical) and orientations of the vortex promoters were considered. The blockage ratios were varied between 1/10 to 1/2. Re, based on the rectangular cylinders side lengths, varied between 30 and 150. The inlet velocity profile was flat, not parabolic. Since the channel computational domain was fixed and the obstacles varied, the distance from the cylinders to the inlet section varied from case to case. For example, for a blockage ratio of 1/10 (close to that of references [1,2]) the distance to the inlet section was 19 diameters. For a blockage ratio of 1/4 (close to that of reference [3]) the distance to the inlet section was 7 diameters. One of the main results presented was that from the point of view of an engineering compromise between heat transfer and pressure drop, the 1/2 ratio rectangular rectangular cylinder is, possibly, the most favourable shape. Interestingly enough, the authors found an unexpectedly complex flow behavior. In particular, a multiplicity of stable flow states was identified, and, in some cases, transition between different states involved larger heat transfer with a small increase in pressure drop.

It is to be said, also, that all these studies that involve confined channel flow with obstacles (references [1–4] and many others that cannot be quoted here because of space limitations) apart from shedding light into basic Thermo-Fluid Mechanics aspects are of interest when optimizing actual engineering designs. This is important, for instance, when developing products aiming to act as heat sinks for compact electronic equipment, avionics systems, etc. For a view on this closely related subject of optimization, the reader is directed to the works of Icoz and Jaluria [5] and Jaluria [6].

Then, it is, precisely, in view of the practical engineering application of this type of studies that this article has been motivated. In particular, there are, among many other, three aspects that deserve to be analyzed in some detail. First, the question of three-dimensionality that is present in nearly all practical engineering flow devices versus the use of simplified 2D numerical models. Second, the use of simplified boundary conditions in the simulations that might not be fully representative of the actual situation. This is apparent, for example, when prescribing fully developed inlet velocity profiles that would actually require the presence of either a long inlet channel or a large stagnation chamber to homogenize the flow; circumstances difficult to achieve when dealing, for instance, with cooling of compact electronics systems. Third, the fact that 2D solutions are special in the sense that, sometimes, they are dominated by effects which are not dominant in the actual 3D flow.

In this context, it is somewhat surprising that the number of articles published in the specialized Journal literature dealing with