International Journal of Heat and Mass Transfer 125 (2018) 710–721

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Constructal design of forced convective flows in channels with two alternated rectangular heated bodies

B.C. Feijó [a], G. Lorenzini [b,*], L.A. Isoldi [a], L.A.O. Rocha [c], J.N.V. Goulart [d], E.D. dos Santos [a]

[a] Graduate Program of Ocean Engineering, School of Engineering, Universidade Federal do Rio Grande – FURG, Italia 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] Mechanical Engineering Graduate Program, University of Vale do Rio dos Sinos – UNISINOS, São Leopoldo, RS, Brazil
[d] Group of Experimental and Computational of Mechanics, University of Brasilia, PO box 8114, Gama, DF 72.405-610, Brazil

ARTICLE INFO

Article history:
Received 20 February 2018
Received in revised form 10 April 2018
Accepted 17 April 2018

Keywords:
Channel flows
Heated bodies
Forced convection
Numerical simulation
Constructal design

ABSTRACT

Present numerical study performs a geometrical optimization by means of Constructal Design and Exhaustive Search of two alternated rectangular heated bodies mounted in channel surfaces subjected to steady, two-dimensional, incompressible, laminar and forced convective air cooled flows. The problem has two purposes, maximize the heat transfer rate between the bodies and surrounding flow (q) and minimize pressure drop (ΔP) in the channel, i.e., a multi-objective problem. The system is subjected to five constraints, but only two are evaluated here: area fractions of first and second bodies (ϕ_1 and ϕ_2). The problem has two degrees of freedom: ratio between the height and length of upward and downward bodies (H_1/L_1 and H_2/L_2) placed in lower and upper surfaces of the channel, respectively. The influence of fraction areas on the system performance is also investigated. All simulations are performed with constant Reynolds and Prandtl numbers, $Re_H = 100$ and $Pr = 0.71$. As expected, highest intrusion and areas of the bodies were benefical for heat exchange, while the opposite was noticed for pressure drop. For multi-objective optimization, intermediate optimal shapes with assimetric sizes were achieved. The best multiobjective performance is reached for the upward body higher than the downstream one ($H_1/L_1 > H_2/L_2$).

1. Introduction

Many experimental and numerical studies have been performed to improve the comprehension about convection heat transfer in channels with mounted heated bodies. This kind of problem can represent ideally several real engineering problems as those found in electronic packaging, heat exchangers for heating or cooling, solar air heater (SAH) and Heating, Ventilation and Air Conditioning (HVAC) devices [1–3]. According to Sewall et al. [4] ribbed internal cooling ducts are also commonly used in gas turbine vanes and blades to enhance heat transfer coefficient. Other example is related with the cooling of electronic devices, which is highly important in strategically areas like aerospace, defense and biomedical engineering, where shortcomings cannot be afforded [5,6]. In this sense, the development of cooling strategies to achieve high performance of heat removal systems is an important subject [7].

Several works have been performed to improve the comprehension about fluid dynamic and thermal behavior of forced convective laminar flows in channels with one body or array of mounted bodies. For instance, Young and Vafai [8] investigated numerically forced convective flows with one mounted block in a channel surface considering heat conduction in the obstacle. The influence of geometrical parameters of the bodies as its height and width, as well as, the thermal conductivity of solid and fluid over flow and heat transfer characteristics was evaluated. Results showed that shape and material of the obstacle significantly influenced the behavior of fluid flow and heat transfer. Korichi and Oufer [9] made a similar study considering an array with three blocks mounted alternately in upper and lower surfaces of the channel. The influence of Reynolds number, the obstacle dimensions and their conductivities over heat transfer behavior was investigated. Korichi and Oufer [10] extended the study of Ref. [9] for oscillatory flows in the channel over an arrangement of alternating square blocks mounted on the channel walls. In general, results showed that the insertion of upper obstacle caused the generation of vortexes that increased the heat transfer rate between the blocks and surrounding flow. Moreover, it was verified that higher obstacles also conducted to an augmentation of heat exchange. Luviano-Ortiz et al. [11] evaluated numerically the insertion of curvilinear deflectors above the heated blocks on

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

https://doi.org/10.1016/j.ijheatmasstransfer.2018.04.086
0017-9310/

Nomenclature

A_{aux}	Auxiliary area [m^2]
A_c	Area [m^2]
A_s	Surface area [m^2]
C_p	Specific heat at constant pressure [J/(kg K)]
h	Heat transfer coefficient [W/(m^2 K)]
H	Height of the channel [m]
H_1	Height of the body 1 [m]
H_2	Height of the body 2 [m]
k	Thermal conductivity of the fluid [W/(m K)]
L	Length of the channel [m]
L_1	Length of the body 1 [m]
L_2	Length of the body 2 [m]
L_3	Length of the auxiliary area 1 [m]
L_4	Length of the auxiliary area 2 [m]
P	Pressure [Pa]
P_{in}	Spatial averaged pressure at the inlet of the channel [Pa]
P_{out}	Spatial averaged pressure at the exit of the channel [Pa]
Pr	Prandtl number ($\mu c_p/k$)
q	Heat transfer rate between the heated bodies and free stream [W]
Re_D	Reynolds number ($\rho u_\infty H/\mu$)
T_∞	Fluid temperature at the free stream [K]
T_w	Temperature of the body surface [K]
T_{out}	Spatial averaged temperature at the exit of the channel [K]
u_∞	Velocity at the free stream [m/s]
u	Velocity in x-direction [m/s]
v	Velocity in y-direction [m/s]
x	Spatial coordinate in x-direction [m]
y	Spatial coordinate in y-direction [m]
W	Depth of the channel (z-direction) [m]

Greek symbols

ΔP	Pressure drop [Pa]
μ	Dynamic viscosity [kg/m s]
υ	Kinematic viscosity [m^2/s]
ρ	Density [kg/m^3]
ϕ	Fraction area of

Subscripts

max	Maximized
min	Minimized
c	channel
o	Once optimized
F	Fluid dynamic objective
T	Thermal objective
1	body 1
2	body 2
$2o$	Twice optimized
2,max	Twice maximized
2,min	Twice minimized
$3o$	Three times optimized
3,max	Three times maximized
3,min	Three times minimized
$4o$	Four times optimized
4,max	Four times maximized
4,min	Four times minimized

the heat transfer rate between forced convective flow and obstacles. To achieve this purpose, a comparison was performed between cases of convective flows with and without deflectors. According to the authors, the employment of deflectors was more recommended for flows with high Reynolds numbers. Afterwards, Yemenici et al. [12] studied experimentally forced convective laminar and turbulent flows. It was investigated the combined effects of free stream velocities and different sizes of rectangular blocks on the flow and heat transfer characteristics. The increase of surface areas of the blocks led to an augmentation of flow separation and heat transfer rate. Recently, Durgam et al. [13] studied experimentally and numerically a steady state forced convective air flow in a vertical cooling channel with heat sources mounted in a substrate board. The main purpose was to find out the optimal distribution of seven heat sources which minimizes the substrate temperature. It was obtained from the results a correlation for substrate temperature as a function of Reynolds number and ratio between the thermal conductivities of substrate board and air.

The study of mixed and natural convective flows in channels with mounted blocks has also been performed, as can be seen in Refs. [1,14,15]. These works led to interesting recommendations about the influence of dimensionless parameters of the convective flow (Reynolds and Grashof numbers) and geometric dimensions of the blocks over the heat transfer rate, fluid dynamic and thermal patterns. Important studies have also been dedicated to numerical and experimental evaluation of forced, mixed and natural convective turbulent flows in channels with mounted blocks [2,16–18]. With exception of the work of Ref. [2], the above mentioned studies about turbulent flows have not been dedicated to geometric optimization, mainly due to high level of complexity of this kind of fluid flow, high computational effort for numerical predictions and expensive costs for development of several experiments. Then, most of investigations have been devoted to improve the comprehension about fluid dynamics and thermal patterns with few different geometrical configurations. Other complex shapes have also been investigated, for instance, Handoyo et al. [3] performed a numerical study of a turbulent air flow in a V-shaped corrugated channel with internal delta-shaped obstacles that mimics a Solar Air Heater (SAH). The influence of spacing between obstacles on the heat transfer and pressure drop of channel flow was analised. Beyond the previous studies, important contributions have been done in the investigation of convective nanofluid flows, imposition of porous blocks mounted in the channels, annular flows and new methods for optimization of the problem [19–22]. In spite of important contributions performed in the literature, Constructal Design has not been employed for geometrical evaluation of convective flows with mounted blocks on the channel surfaces.

Constructal Design is a method based in constraints and performance indicators (objectives) for evaluation of design of any flow system (animated or inanimated) with finite dimensions. Constructal Design has been used to show that the design of flow systems and its evolution along the time are predictable by a physical principle named Constructal Law. Constructal Law states that a finite flow system with freedom to morph along the time will evolve in such way to improve the access to the internal currents that flow through it [23–26]. For study of engineering problems, Constructal Design has been successfully employed to improve the performance of several problems. Examples have been noticed in solids with heat generation cooled by cavities, finned systems, high conductive pathways, heat exchangers, refrigeration, solid mechanics and renewable energy [27–38]. Constructal Design is used for definition of search space of geometrical configurations to be evaluated. For optimization, in the present study it is employed Exhaustive Search, which consists in the simulation of