International Journal of Heat and Mass Transfer 78 (2014) 50–57

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Slot jet impingement heat transfer in the presence of jet-axis switching

J.M. Gorman [a], E.M. Sparrow [a,*], J.P. Abraham [b]

[a] Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55455, USA
[b] School of Engineering, University of St. Thomas, St. Paul, MN 55105, USA

ARTICLE INFO

Article history:
Received 4 May 2014
Received in revised form 9 June 2014
Accepted 12 June 2014
Available online 14 July 2014

Keywords:
Jet impingement
Rectangular jet
Jet-axis switching
Large aspect-ratio jet
Laminar flow
Turbulent flow

ABSTRACT

Jet-axis switching is an established phenomenon whereby a non-circular, three-dimensional free jet undergoes a major change in cross-sectional shape with increasing downstream distance from the jet origin. This phenomenon has been demonstrated here to also occur for impinging jets. The focus of the present work is to investigate the heat transfer and fluid flow characteristics of rectangular slot jets which experience jet-axis switching. The jets in question have initial cross-sectional aspect ratios of 5:1 and 10:1. The jet cross sections, although highly skewed at first, evolve through near circularity and subsequently become skewed in the direction perpendicular to that of the initial skewness. In addition to the two initial aspect ratios, parametric variations were made of the Reynolds number, the distance of the impingement plate from the jet origin, and the contraction experienced by the flow passing through the aperture of the jet-forming orifice (i.e., the blockage ratio). The investigation was implemented by means of numerical simulation from which local and average Nusselt numbers were determined as functions of the foregoing parameters. Higher Reynolds numbers, greater downstream distances of the impingement plate, and greater blockages served to enhance the Nusselt number values. It remains to be seen whether there is an initial jet aspect ratio that is large enough to preclude axis switching and thereby allow two-dimensional modeling.

1. Introduction

It is widely accepted that jet impingement provides the highest local heat transfer coefficients among all forced convection fluid flows. This realization has motivated a steady stream of relevant research which has been well summarized and correlated [1–7]. A different stream of jet-related research has focused on the fluid mechanics of free jets. That research stream has discovered a flow transformative phenomenon termed *axis switching* [8–17]. This phenomenon is encountered for free jets of non-circular cross section. In particular, it is found that a rectangular jet whose initial configuration is defined by a horizontal long dimension and a vertical short dimension undergoes a metamorphosis so that its downstream configuration is characterized by a horizontal short dimension and a vertical long dimension. This fluid flow phenomenon has been extensively investigated to reveal complex and esoteric structures, but it appears that no consideration has been given to the possible practical implications of axis switching.

The focus of the present investigation is to definitively determine the response of jet-impingement heat transfer to the axis-switching phenomenon. A necessary first step in the research is to demonstrate that axis switching continues to exist in the presence of jet impingement, the phenomenon having been previously observed only for free jets. Subsequent to that demonstration, the focus of the work is directed to the identification of jet axis switching as a heat transfer controlling mechanism for impinging jets.

The physical situation to be considered encompasses the creation of rectangular jets of different aspect ratio as the result of flow passing through and subsequently emerging from an orifice-capped rectangular duct into an expansion space. Axis-switching transitions are experienced by the jet as it passes through the expansion space. These transitions may be affected by the presence of impingement surfaces. The differences between the nature of jet switching for a free jet and an impinging jet must be recognized as a factor in the analysis of jet impingement heat transfer.

The analysis tool to be employed in the investigation is numerical simulation. Whereas the flow in the jet-creating duct is modeled as laminar, the emerging jet may be either laminar or turbulent. A metric is defined to quantify the regime of the flow.

2. Physical situation

The description of the physical model is facilitated by reference to Fig. 1. That figure is a perspective view of a rectangular duct of constant cross section through which a fluid passes in laminar

* Corresponding author. Tel.: +1 612 625 9881.
E-mail address: esparrow@umn.edu (E.M. Sparrow).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.06.041
0017-9310/

Nomenclature

A	model constant
a	width of the slot opening
b	height of the slot opening
c_p	specific heat at constant pressure
F_1, F_2	blending functions in the SST model
H	height of the duct
h	local heat transfer coefficient
$\bar{h}$	average heat transfer coefficient
k	thermal conductivity
k_{turb}	turbulent thermal conductivity
L	length of the duct, $100b$
P_k	production term for the turbulent kinetic energy
Pr	Prandtl number
Pr_{turb}	turbulent Prandtl number
Q	rate of heat transfer
R	distance from duct centerline to solution domain outer boundary, $y = z = 50b$
Re	Reynolds number, Ub/ν
S	absolute value of the shear strain rate
s	solution domain extension upstream of duct exit, $10b$
T	temperature
T_w	temperature of the impingement plate
T_∞	inlet fluid temperature
U	mean velocity in the duct
u	velocity component in the x-direction
u_i	velocity component in the i-direction
W	width of the duct
x_i	tensor coordinate direction
α	SST model constant
β_1, β_2	SST model constants
ε	turbulence dissipation
κ	turbulent kinetic energy
μ	molecular or dynamic viscosity
μ_{turb}	turbulent viscosity
ν	kinematic viscosity
ρ	fluid density
σ	diffusion coefficient
ω	specific rate of turbulence dissipation

flow. The exit cross section of the duct may be partially blocked by an orifice plate or may otherwise be unobstructed. In either situation, the exiting fluid expands into a large open space. The fluid entering the space may flow either freely without blockage or may impinge on a flat surface where heat transfer occurs. Details of the exit cross section of the rectangular duct are conveyed in Fig. 2, where dimensional nomenclature is also displayed. The aspect ratio of the duct proper is W/H while the aspect ratio of the rectangular slot is a/b. The flow direction is x, while the cross-sectional coordinates are y and z.

3. Governing equations

The solution of the physical problem defined in the preceding section was achieved by means of numerical simulation. The simulation model was developed to take account of the possible change in the flow regime that may occur in the duct proper and in the downstream expansion space. To accommodate such an event, the governing equations are written for turbulent flow, and a turbulence model that reduces to the laminar regime, when appropriate, is selected. The relevant physical principles that govern the flow and heat transfer are: momentum conservation (Reynolds Averaged Navier–Stokes equations), mass conservation, and the First Law of Thermodynamics. These equations are written for incompressible, constant property flow, but no restrictions are placed on the type of fluid.

The RANS equations, as written in Cartesian tensor form, are

$$\rho\left(u_i\frac{\partial u_j}{\partial x_i}\right) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_i}\left((\mu+\mu_{turb})\frac{\partial u_j}{\partial x_i}\right), \quad i=1,2,3,\ j=1,2,3 \tag{1}$$

The mass conservation equation is

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2}$$

The quaintly μ_{turb} is designated as the turbulent viscosity. For the RANS model, it is defined as

$$\mu_{turb} = -\rho u_i'' u_j'' \tag{3}$$

where u_i'' represents the fluctuating component of u_i.

For the flow-regime characterization, it is necessary to select a turbulence model for which the ratio μ_{turb}/μ reduces to a value $\leqslant 0.01$ in the laminar flow regime. It has been demonstrated in [18] that the Shear Stress Transport turbulence model (SST) displays this characteristic. It represents a blending of two previously developed models, the κ–ε and κ–ω models. The quantity κ is the turbulence kinetic energy, and ω is the specific rate of turbulence dissipation. The κ–ε model has been demonstrated to give satisfactory descriptions of the velocity field at locations that are distant from bounding walls, and the κ–ω model provides a valid characterization of velocities in the near neighborhood of the wall.

The equations of the SST model are

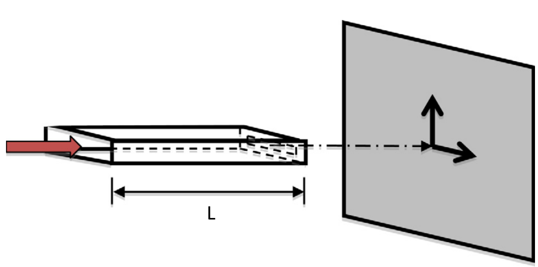

Fig. 1. Pictorial view of the physical situation.

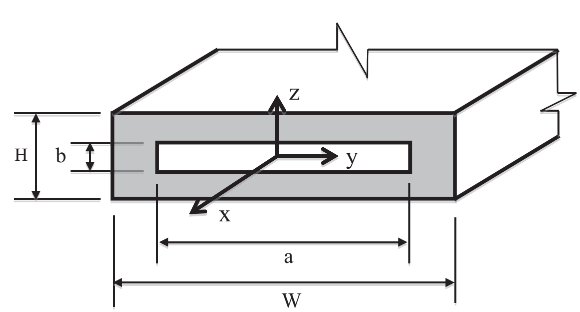

Fig. 2. Details of the exit plane of the duct.