International Journal of Heat and Mass Transfer 95 (2016) 636–649

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical investigation on the performance of low-Reynolds number $k-\epsilon$ model for a buoyancy-opposed wall jet flow

Sushil Kumar Rathore [1,2], Manab Kumar Das *

Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, West Bengal 721302, India

ARTICLE INFO

Article history:
Received 9 January 2015
Received in revised form 11 September 2015
Accepted 31 October 2015
Available online 4 January 2016

Keywords:
Low-Reynolds number model
Yang and Shih model
Negatively buoyant flow
Wall jet
Buoyancy opposed flow

ABSTRACT

The negatively buoyant wall jet flow in a rectangular channel subjected to a slowly moving counter-current stream has been declared as an application challenge. Numerical simulation has remained elusive for last one decade since the two moderately successful papers of Addad et al. (2004) and Craft et al. (2004). The present investigation reports the successful reproduction of the experimental results of He et al. (2002) for the non-buoyant as well as the buoyant cases. The computations have been carried out using low-Reynolds number turbulence model proposed by Yang and Shih (YS). The performance assessment of YS model has been done by comparing the computational results with the experimental results of He et al. Based on the comparison, it has been observed that YS model shows a very good agreement with the experimental results for this complex flow situation.

1. Introduction

1.1. Background and relevant literature

The study of buoyancy opposed jet is important as it is encountered in many industrial applications and also in nature. There are many situations in which buoyancy acts in a direction opposite to that of flow. In case of fire in an enclosure, the fire plume flows downward after hitting the ceiling but buoyancy acts in upward direction. The discharge of heated water or industrial effluent in the ambient and flow in nuclear reactor core belong to the similar flow situation. The important parameter in downward flowing buoyancy opposed jet flow is the depth of penetration. Depth of penetration of jet is defined as the distance from the point of injection to the point at which the non-dimensional wall temperature falls to about a value of 0.02 (He et al. [10]). The opposing buoyancy force causes externally induced flow to retard. As a result, flow reaches a stagnation point and then flows in the opposite direction.

The first detailed experimental study of buoyancy opposed jet flow is reported by Goldman and Jaluria [9]. They have carried out an experimental study of a two-dimensional buoyancy opposed wall jet discharged adjacent to a vertical surface and buoyancy opposed free jet to determine basic flow and thermal characteristics of such a flow. They have utilized hot-wire anemometry and thermocouples for measurements of mean velocity and temperature, respectively. They have performed flow visualization using smoke prior to the experimental study for investigation of basic nature of the flow. They have reported that the depth of penetration is mostly dependent on Richardson number $\left(Gr/Re^2\right)$ and reduces with increase in the Richardson number. They have also found that the experimental mass flow rate increases with increase in Richardson number due to a stronger reverse flow. Kapoor and Jaluria [12] further investigated the work done by Goldman and Jaluria [9] to obtain the heat transfer characteristics. They have experimentally studied the heat transfer characteristics of a two-dimensional negatively buoyant wall jet flow over an adiabatic and an isothermal vertical surfaces. They have obtained the Nusselt number variation and the heat transfer to the vertical surfaces. They have reported that the rate of heat transfer and the depth of penetration both decrease with increase in Richardson number or mixed convection parameter $\left(Gr/Re^2\right)$.

He et al. [10] have experimentally studied the flow and thermal characteristics of a negatively buoyant wall jet which is produced by injecting hot water down one wall of a vertical passage of rectangular cross-section into a counter-stream of cold water. The measurements of local mean velocity and temperature have been carried out using the laser Doppler anemometry and thermocouples, respectively. They have carried out the

* Corresponding author. Tel.: +91 3222 282924; fax: +91 3222 282728.
E-mail addresses: isushilrathore@gmail.com (S.K. Rathore), manab@mech.iitkgp.ernet.in (M.K. Das).

[1] Former Research Scholar.

[2] Present address: Mechanical Engineering Department, National Institute of Technology Patna, Ashok Rajpath, Patna, Bihar 800005, India.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.10.067
0017-9310/

Nomenclature

C_{fx}	skin friction coefficient, $\tau_w/\frac{1}{2}\rho V_0^2$
G	production by shear
g	acceleration due to gravity, m s^{-2}
Gr	Grashof number, $g\beta\triangle Th^3/\nu^2$
h	width of the jet, m
k	turbulent kinetic energy, m^2 s^{-2}
$\overline{p_0}$	ambient pressure, Pa
$\overline{p}$	static pressure, Pa
P	non-dimensional static pressure
Pr	Prandtl number, ν/α
R	ratio of velocity at inlet of counter-flow stream to inlet jet velocity, V_c/V_0
Re	Reynolds number, V_0h/ν
Re_t	turbulence Reynolds number, $k^2/\nu\epsilon$
Re_y	non-dimensional distance, $\sqrt{k}y/\upsilon$
Ri	Richardson number, Gr/Re^2
T	dimensional temperature, K
T_c	temperature at inlet of counter-flow stream, K
T_{in}	jet inlet temperature, K
$\overline{u}, \overline{v}$	dimensional mean velocities in x, y -directions respectively, m s^{-1}
U, V	non-dimensional velocities in X, Y -directions, respectively
u_τ	friction velocity, $\sqrt{\tau_w/\rho}$, m s^{-1}
V_0	average inlet jet velocity in downward direction, m s^{-1}
V_c	non-dimensional velocity at inlet of counter-flow stream
X, Y	non-dimensional coordinates
x, y	dimensional coordinates, m
x^+	dimensionless distance, xu_τ/ν
y^+	dimensionless distance, yu_τ/ν
Greek	
α, α_t	laminar and turbulent thermal diffusivities, respectively, m^2 s^{-1}
β	coefficient of thermal expansion, K^{-1}
$\triangle T$	temperature difference, $(T_{in} - T_c)$, K
ϵ	rate of dissipation of turbulent kinetic energy, m^2 s^{-3}
ν, ν_t	laminar and turbulent kinematic viscosity, respectively, m^2 s^{-1}
ρ	density, kg m^{-3}
τ_w	wall shear stress, Pa
θ	non-dimensional temperature
Subscript	
n	non-dimensional quantity

investigation for mixed convection parameter $\left(Gr/Re^2\right)$ in the range of $0.0 - 0.052$. They have observed that the depth of penetration and lateral spread of jet reduce with increase in the Richardson number. They have also reported a concentrated mixing layer at the interface of two stream in case of stronger buoyancy influence. Craft et al. [3] have numerically investigated the performance of different turbulence models for buoyancy opposed wall jet flow similar to that studied by He et al. [10]. They have applied the low-Reynolds number model of Launder and Sharma [14], the high-Reynolds number $k - \epsilon$ model and the two second-moment closures (Gibson and Launder [8], Craft and Launder [5]) with standard wall function and analytical wall function (Craft et al. [4]). They have presented vertical velocity contours, vector plots and temperature contours for isothermal flow and one of the buoyant test cases of He et al. [10]. They have reported that second-moment closure with analytical wall functions leads to a good agreement with the available results. They have also mentioned that numerical problem have presented them to obtain converged results for one of the two buoyant test case with counter-flow to jet velocity ratio 0.077 using second-moment closures.

The experimental configuration studied by He et al. [10] is computationally investigated by Addad et al. [1] using large eddy simulation. To quote Addad et al. [1], “Based on the experiment of He et al. (2002), this flow was suggested as an “application challenge” by the power generation industrial sector to the Qnet-CFDEU network.” Addad et al. [1] argue that numerical predictions vary significantly with the types of RANS (Reynolds-averaged Navier–Stokes) models used. They claim that the most advanced models used by Craft et al. [3] could yield a reasonable agreement with the experimental data of He et al. [10]. However, Addad et al. [1] have failed to mention that the real challenge is to simulate the effect of buoyancy and Craft et al. [3] have actually presented the validation of isothermal case and not the buoyancy opposed thermal problem. As mentioned, Addad et al. [1] have attempted with a hope to confirm the experimental data. They have used an LES (large eddy simulation) with half-a-million nodes due to the limitation of resources. They have provided results for non-buoyant and buoyant cases. They have considered the Reynolds number 4000 and compared their computational results with He et al. [10]. However, the experimental data utilized for velocity comparisons are absent in He et al. [10]. The manuscript does not mention the Richardson number for which the computations have been carried out. So their results cannot be taken as a good comparison of He et al.’s [10] results. They have applied two codes (Star-CD and *Code_Saturne*) to study isothermal and buoyant test cases, and have reported that both codes return satisfactory results for isothermal flow and moderately buoyant test cases.

In the companion paper, Craft et al. [3] have classified the negatively buoyant turbulent wall jet to be a **more complex flow** than other relatively numerically amenable flows like buoyancy-modified up-and down-flow through pipes and annuli. They argue that the collision of a heated downward wall jet flow with the low-velocity upward moving cold uniform stream results in a stagnation point of the wall jet and turning of the jet upwards; this further leads to buoyant as well as dynamic influence on the stagnation point position. They further argue that a numerical prediction of the flow would require a Reynolds stress transport model (RSM) rather than an isotropic eddy viscosity turbulence model. Craft et al. [3] have used the new analytical wall function (AWF) developed by Craft et al. [4]. Craft et al. [3] have shown the comparison of their numerical results (using two-component-limit (TCL) model with AWF and standard $k - \epsilon$ model with AWF) with the experimental results of He et al. [10] and LES results of Addad et al. [1] at downstream location 0.4 m. However, the results shown in Fig. 4(b) of Craft et al. [3] correspond to a downstream location of 0.6 m; in the archival literature of He et al. [10], they have provided the results up to 0.5 m. Also to be noted that, Craft et al. [3] have carried out computations for $\mathrm{Re} = 4000$ whereas He et al. [10] have carried out experiments for $\mathrm{Re} = 4754$. He et al. [10] have carried out their experiments where V_c/V_0 was maintained very close to 0.077 for all the cases. Craft et al. [3] have shown some general results for buoyant cases