Energy Conversion and Management 51 (2010) 2117–2126

Contents lists available at ScienceDirect

Energy Conversion and Management

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

A numerical study of a turbulent axisymmetric jet emerging in a co-flowing stream

Houda Mahmoud [a,*], Wassim Kriaa [a], Hatem Mhiri [a], Georges Le Palec [b], Philippe Bournot [b]

[a] Unité de thermique et thermodynamique des procédés industriels, Ecole Nationale d'Ingénieurs de Monastir, route de Ouardanine, 5020 Monastir, Tunisia
[b] IUSTI, UMR CNRS 6595, 5 Rue Enrico Fermi, Technopôle de Château-Gombert, 13013 Marseille, France

A R T I C L E I N F O

Article history:
Received 30 March 2009
Received in revised form 18 November 2009
Accepted 14 March 2010
Available online 2 April 2010

Keywords:
Axisymmetric jet
Turbulence
Co-flow
K–ε model
RSM model
Recirculation bubbles

A B S T R A C T

In this work, we propose a numerical study of an axisymmetric turbulent jet discharging into co-flowing stream with different velocities ratios ranging between 0 and ∞. The standard k–ε model and the RSM model were applied in this study. The numerical resolution of the governing equations was carried out using two computed codes: the first is a personal code and the second is a commercial CFD code FLUENT 6.2. These two codes are based on a finite volume method.

The present predictions are compared with the experimental data. The results show that the two turbulence models are valid to predict the average and turbulent flow sizes.

Also, the effect of the velocities ratios on the flow structure was examined. For $R_u > 1$, it is noted the appearance of the fall velocity zone due to the presence of a trough low pressure. This fall velocity becomes increasingly intense according to R_u and changes into a recirculation zone for $R_u \geqslant 4.5$. This zone is larger and approaches more the nozzle injection when R_u increases.

1. Introduction

The jet type flows occur in a variety of applications, especially in the industrial sector. For over a century, the theory of turbulent jets and their practical applications have attracted the specialists attention in many research fields [1,2]. The turbulent co-flowing jets represent one of the most frequent cases of engineering applications which involve the interaction between two parallel flows, such as in the propelling systems, where the role of the aerodynamics on the engine function (efficiency, power, pollution, etc.) is critical. In fact, the mixture between the fuel and the oxidizing is conditioned by the structure of the flow (presence of eddies). In this context, several experimental and numerical works have been devoted. They have considered different velocities ratios $R_u = u_\infty/u_0$ (u_0 is the jet velocity and u_∞ is the co-flow velocity) in order to understand and model the fundamental mechanisms governing the mixture between the jet and it is ambient.

In practice, the jets discharged into co-flowing ambient stream are turbulent. They have been studied in experiments with low velocities ratios (R_u ranging between 0.01 and 0.17) [3–6]. The objective of studying such flow is to solve the problems of seeding found at the jet edge and to avoid the development of oscillations in the primary flow. So, we can provide good conditions for measurements and reduce errors in region of high turbulence levels. In this context, Mesnier [7] investigated three density ratios: 7.2, 1 and 0.66 which correspond, respectively, to the mixing of helium, air or CO_2 jet merging, with 40 m s^{-1} velocity, into a uniform co-flowing air stream with 0.36 m s^{-1} velocity. He was concerned to study the density and the injector geometry effect on the jet development. Antoine et al. [8] performed measurements in turbulent water jet flowing with an injection velocity equal to 10 m s^{-1} and discharging into a low velocity co-flowing water stream equal to 0.5 m s^{-1}. These authors have studied experimentally and numerically the mass and momentum transport properties of this flow. The effects of the co-flow on the turbulent mixing process were highlighted. The major visible effect of the co-flow was the jet spreading rate reduction with approximately 30% from the free jet.

Numerically, the study of turbulent jets emerging in a co-flow with low velocities ratios has been the subject of several works in order to stabilize the calculations and to ensure the convergence of codes. Morgans et al. [9] are interested in the study of a propane jet emerging into moving co-flowing air stream with a velocity ratio $R_u = 0.165$. They have considered the parabolic approach (i.e. the boundary layer assumptions) and the different turbulence models: standard k–ε, modified k–ε and k–ω. They showed that the k–ω model gives a prediction of the round jet spreading rate similar to that of the modified k–ε model and better than that of the standard k–ε Gazzah et al. [10] have, also, used the parabolic approach and the second order model RSM for studying numerically the experimental configuration of Shefer et al. [11]. They have focused their work on the influence of co-flow on the characteristic parameters of the jet.

All these studies have considered low velocities ratio. However, a low velocity co-flow is not desirable in many industrial applications and its acceleration is required to respond to several implementation needs.

* Corresponding author.
E-mail address: mahhouda2003@yahoo.fr (H. Mahmoud).

0196-8904/$ - see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.enconman.2010.03.004

Nomenclature

d	nozzle diameter, m
k	turbulent kinetic energy, $m^2 s^2$
x, r	longitudinal and transverse coordinates, m
u, v	mean velocity components along x and y directions, $m s^{-1}$
U, V	dimensionless mean velocity components
U_{ex}	excess velocity ($U_{ex} = U - U_\infty$)
P	static pressure (Pa)
R_u	velocities ratio ($R_u = u_\infty/u_0$)
Re	Reynolds number, $Re = ud/v$
Re_t	turbulent Reynolds number, $Re_t = ud/v_t$
I	turbulent intensity
L_B	recirculation bubble width
H	recirculation bubble length
P	recirculation bubble position

Greek symbols

ε	dissipation rate of the turbulent kinetic energy, $m^2 s^{-3}$
v	molecular kinematic viscosity ($m^2 s^{-1}$)
v_t	turbulent kinematic viscosity ($m^2 s^{-1}$)
ρ	density ($kg\ m^{-3}$)
δ_{ij}	Kronecker delta

Subscripts

∞	ambient middle
0	nozzle exit
C	jet axis

Superscript

$\bar{}$	Reynolds average
$'$	fluctuation

To our knowledge, the effect of the high velocity co-flow on a jet type flow was not examined. However, the studies conducted on coaxial jets showed the important effect of the high velocities ratio. In fact, Kriaa et al. [12] proved that high velocities ratio generates a more significant jet acceleration accompanied with a higher training velocity. Rehab et al. [13] showed the appearance of a recirculation bubble for a velocity ratio R_u higher than a critical value R_{uc} which have a strong effect on the flow dynamic and concentration fields.

In this work, we propose a numerical study of a turbulent co-flowing jet for different velocities ratios R_u ranging between 0 and ∞, elaborated by two computer codes using the elliptic approach (complete equations governing the flow without the boundary layer approximations): the first is a personal code using the standard k–ε model and the second is elaborated using the commercial CFD code FLUENT 6.2 adopting two different turbulence models (k–ε and RSM models). The discussion about the found results will be related to the validity of the two turbulence models for the prediction of the mean and turbulent flow parameters. We will be, also, interested in determining the critical velocity of the appearance of the recirculation bubbles and its effects on the flow structure.

2. Assumptions

The experimental configuration of Antoine et al. [8] is used to study a flow issuing from cylindrical nozzle of diameter equal to *1mm* which ejects a fluid into a same fluid co-flowing (Fig. 1).

The equations are written in a reference frame of which the origin is located at the nozzle centre and the flow is supposed to be two-dimensional. The following assumptions will be, also, considered:

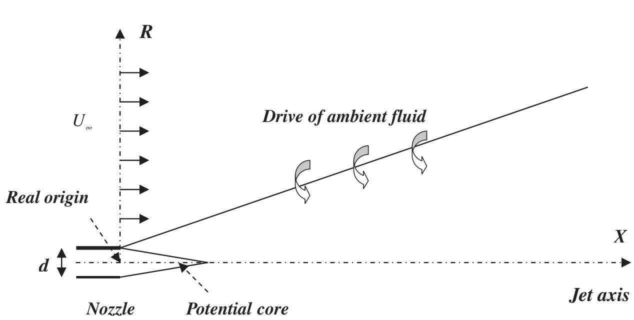

Fig. 1. Geometrical configuration of a co-flowing jet.

- The jet emitted horizontally according to X is with axial symmetry.
- The flow is average stationary.
- The flow is incompressible.
- The flow is supposed to be turbulent and fully developed (great Reynolds numbers).

3. Equations and numerical modelling

3.1. Governing equations

By using the above assumptions, the principle of the mass conservation and the Newton's second law [14], the continuity and momentum conservation equations are written in the following way:

$$\frac{\partial(ru)}{\partial x} + \frac{\partial(rv)}{\partial r} = 0 \quad (1)$$

$$\frac{\partial(uu)}{\partial x} + \frac{1}{r}\frac{\partial(rvu)}{\partial r} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(-u'u') + \frac{1}{r}\frac{\partial}{\partial r}(-ru'v') \quad (2)$$

$$\frac{\partial(uv)}{\partial x} + \frac{1}{r}\frac{\partial(rvv)}{\partial r} = -\frac{1}{\rho}\frac{\partial p}{\partial r} + \frac{\partial}{\partial x}(-u'v') + \frac{1}{r}\frac{\partial}{\partial r}(-rv'v') \quad (3)$$

For the closure of these equations, we considered two turbulence models: the standard k–ε model and the Reynolds stress model (RSM).

3.1.1. The standard k–ε model

The turbulent kinetic energy and dissipation rate equations can be written in the following way:

$$\frac{\partial(u\phi)}{\partial x} + \frac{1}{r}\frac{\partial}{\partial r}(rv\phi) = \frac{\partial}{\partial x}\left(\Gamma_\phi\frac{\partial\phi}{\partial x}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\Gamma_\phi\frac{\partial\phi}{\partial r}\right) + S_\phi \quad (4)$$

ϕ, S_ϕ and Γ_ϕ are respectively the considered variables, the sources terms and the turbulent diffusion coefficients given in Table 1.

P_k is the production term of the turbulent kinetic energy given by the following relation:

$$P_K = v_t\left(2\left(\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial r}\right)^2 + \left(\frac{v}{r}\right)^2\right) + \left(\left(\frac{\partial u}{\partial x}\right) + \left(\frac{\partial v}{\partial r}\right)\right)^2\right) \quad (5)$$

We note that the Reynolds constraints tensor terms are modelled by the following approximations [15]:

$$-\overline{u_i'u_j'} = v_t\left(\frac{\partial\overline{u_i}}{\partial x_j} + \frac{\partial\overline{u_j}}{\partial x_i}\right) - \frac{2}{3}\left(k + v_t\frac{\partial\overline{u_i}}{\partial x_i}\right)\delta_{ij} \quad (6)$$