Computational Investigation of Turbulent Round Jet in a Significant Co-flow

Prof A Dewan, *Member*

L Alluru, *Non-member*

S Maurya, *Non-member*

In the present paper, turbulent round jet in a significant co-flowing ambient has numerically been studied using two widely used turbulence models, namely, the standard k-ε model and Reynolds-stress transport (RST) model. The performance of the two models has been analyzed by means of a detailed comparison between the predictions and the experimental observations reported in the literature. The comparisons show that the mean flow quantities predicted by both models agree well with the experimental observations. However, some deviations are observed in the predicted turbulent quantities such as Reynolds shear stress and turbulence intensities. The deviation between the predictions by the RST model and experimental data is large compared to that by the k-ε model. Thus it can be concluded that overall the standard k-e model is superior to RST model for this flow configuration.

Keywords : RANS equations; Computational fluid dynamics; Free-shear flow; Self-similarity

NOTATION

$\overline{u_i'}$: fluctuating velocity component

u_i : mean velocity component

$\overline{u'^2}$: Reynolds axial normal stress

$\overline{v'^2}$: Reynolds radial normal stress

$\overline{u'v'}$: Reynolds shear stress

b : width of the jet

C_{ij} : convection term

$D_{T,ij}$: turbulent diffusive transport term

k : turbulent kinetic energy

P_k : rate of production of k

r : radial coordinate

$r_{1/2}$: half width of the jet

U : mean velocity of jet in the axial (streamwise) direction

U_1 : co-flow velocity

U_{co} : centreline excess velocity of jet

u_i : instantaneous velocity component

U_j : velocity of jet at the discharge

U_o : velocity of jet at the axis

V : mean velocity of jet in the radial direction

δ_{ij} : Kronecker delta

ε : turbulent dissipation rate

ε_{ij} : dissipation tensor

ϕ_{ij} : pressure-strain term

μ : molecular viscosity

μ_t : eddy viscosity

ρ : mass density

$\sigma_k, \sigma_\varepsilon$: turbulent Prandtl numbers for k and ε, respectively

υ_t : kinematic eddy viscosity

Prof A Dewan is with the Department of Applied Mechanics, while L Alluru and S Maurya are with the Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati 781 039.

This paper (modified) was received on May 22, 2008. Written discussion on this paper will be entertained till September 30, 2009.

INTRODUCTION

Most of the fluid flows encountered in engineering applications are turbulent. Turbulent flows are characterized by unsteadiness, vorticity, three-dimensionality, dissipation, wide spectrum of length and time scales and large mixing rate. With the emergence of computers and different computational techniques, the computational fluid dynamics (CFD) has become an important tool for investigating turbulent flows. In CFD, the governing equations are numerically solved over a computational domain along with suitable boundary conditions to obtain the flow field. The aim of the present work is to predict the flow field of a turbulent round jet in a significant co-flowing stream of the same fluid. This is an important test case for turbulence modelling, because it is more complex than that of a standard turbulent round jet in

a quiescent ambient. For an axisymmetric turbulent jet in a quiescent ambient, it has been well established that the mean and turbulent profiles are self-preserving. Antonia and Bilger[1] experimentally showed that the self-preservation does not hold in the case of a round jet discharged in a significant coflow and the flow far downstream depends strongly on the complete flow history, unlike that for self-similar flows. Gaskin and Wood[2] showed that in such flows the excess mean velocities are approximately self-preserving far downstream, but the profiles of turbulent shear stress and streamwise velocity fluctuations may not be self-preserving even far downstream of the jet discharge.

There have also been a number of studies on jets released both in a quiescent ambient and parallel and in the direction of a flowing fluid[3-11]. It is known that the k-ε model correctly predicts the growth rate of the plane jet but that of the round jet is over-predicted by about 40%. This is termed as the plane jet/round jet anomaly. Some modifications to the standard k-ε model have been proposed in the literature to rectify this weakness (Pope[3]; Cho and Chung[4]). Of these, the modification due to Cho and Chung[4] is found to be the most effective. Lately, direct numerical simulation (DNS) and large eddy simulation (LES) techniques have been used to study these flows, where unsteady three-dimensional Navier-Stokes equations are numerically solved (Boersma, *et al* [5] and Ribault, *et al*[6]). Chu, *et al*[10] performed LIF measurements in the transition region between the weakly and strongly advected regions of turbulent jet in a coflow and showed that spread constant in the weak jet region is half that of strong jet region. Maczynski[11] showed experimentally that in a jet in a coflowing ambient the centerline mean velocity decreases inversely as the distance from the jet source and this observation is contrary to the common assumption of turbulence being in local equilibrium.

It may be noted that to the best of the knowledge of authors no study has been reported in the literature on a detailed assessment of the performance of various turbulence models in predicting turbulent round jet in a coflowing ambient. Here results have been presented from the computational investigation of the flow field of a turbulent round jet in a co-flowing stream of the same fluid using the standard k-ε and Reynolds stress transport (RST) models. The predictions are compared with the experimental data of Antonia and Bilger[1]. The governing equations including two turbulence models used are described later. The computational procedure including the boundary conditions, mesh used, and convergence criterion are also described afterwards. The predictions of different quantities, including mean velocity and width of jet, turbulent shear and normal stresses, by the two models and their comparison with the measurements have also been presented followed by the conclusion.

MATHEMATICAL FORMULATION

The flow configuration and the associated parameters used in the present study are the same as those used by Antonia and Bilger[1] for their measurements and are shown in Figure 1(a). A turbulent jet with the discharge velocity U_j issues out from a circular pipe of the diameter, D in a co-flowing atmosphere of the same fluid which has a velocity of U_1. The centreline excess velocity of the jet is shown as U_{co} which is the difference of the centreline jet velocity (U_o) and the co-flow velocity (U_1).

Governing Equations

The flow is assumed to be incompressible, axisymmetric and steady in mean. The Reynolds-averaged Navier-Stokes (RANS) equations used can be written as follows.

Continuity equation

$$\frac{\partial U}{\partial x} + \frac{1}{r}\frac{\partial}{\partial r}(rV) = 0 \qquad (1)$$

Momentum equation in the axial direction

$$U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial r} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x}\left[\left(\overline{-u'^2}\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\overline{-u'v'}\right)\right] \qquad (2)$$

Momentum equation in the radial direction

$$U\frac{\partial V}{\partial x} + V\frac{\partial V}{\partial r} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial r} + \frac{\partial}{\partial x}\left[\left(\overline{-u'v'}\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\overline{-v'^2}\right)\right] \qquad (3)$$

Two turbulence models, namely, the standard k-ε model and Reynolds stress transport (RST) model are chosen to cover the wide spectrum of modelling options that are available for the prediction of the present flow characteristics. This is because, on one hand the k-ε model is known to handle many engineering flows with reasonable accuracy and on the other hand the RST model is one of the most elaborate turbulence models[7] and therefore expected to provide more accurate predictions compared to that by any other model for RANS equations. These two models have been widely used to compute turbulent flows of different complexities. Therefore it is important to assess the suitability of these models to new flow configurations, such as, that encountered in the present study. It may be noted that the computational cost and time involved in the use of RST model are quite high compared to that of the standard k-ε model. It can be shown in the next section that the standard k-ε model is superior to RST model in predicting the present flow configuration.

Standard k-ε Model

In the standard k-ε model, the turbulent stresses are modelled by using the Boussinesq approximation as

$$-\overline{u'v'} = \nu_t\left(\frac{\partial U}{\partial r} + \frac{\partial V}{\partial x}\right),\quad \overline{-u'^2} = 2\nu_t\frac{\partial U}{\partial x} - \frac{2}{3}k,$$

$$\overline{-v'^2} = 2\nu_t\frac{\partial V}{\partial r} - \frac{2}{3}k \qquad (4)$$

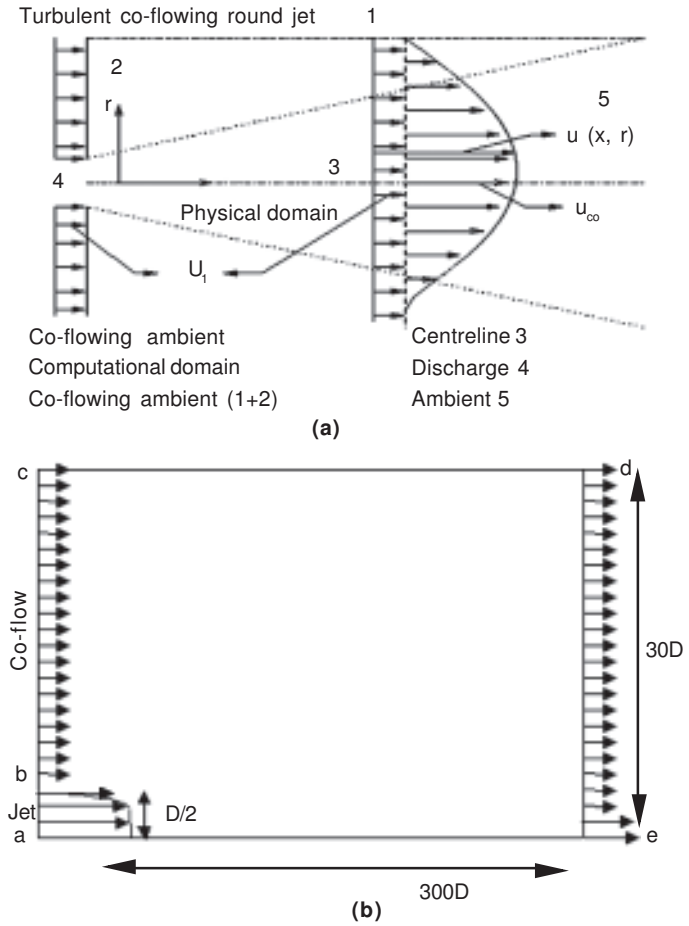


Figure 1 (a) Flow configuration of turbulent round jet in a co-flowing ambient of the same fluid and (b) computational domain used in the present investigation (not to scale)

where

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

Here, ν_t is the eddy viscosity, C_μ, a model constant, k, the turbulent kinetic energy and ε is the dissipation rate of turbulent kinetic energy. The k-ε model is based on the transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε) and these can be written as

$$U\frac{\partial k}{\partial x} + V\frac{\partial k}{\partial r} = \frac{\partial}{\partial x}\left[\left(\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x}\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial r}\right] + P_k - \varepsilon \tag{6}$$

$$U\frac{\partial \varepsilon}{\partial x} + V\frac{\partial \varepsilon}{\partial r} = \frac{\partial}{\partial x}\left[\left(\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial r}\right] + C_1\frac{\varepsilon}{k}P_k - C_2\frac{\varepsilon^2}{k} \tag{7}$$

where P_k is the rate of production of k given by

$$P_k = \left[\left(\frac{\partial U}{\partial r} + \frac{\partial V}{\partial x}\right)^2 + 2\left(\frac{\partial U}{\partial x}\right)^2 + 2\left(\frac{\partial V}{\partial r}\right)^2 + 2\left(\frac{V}{r}\right)^2\right] \tag{8}$$

The values of the model constants used in the above equations are $C_\mu = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$ as explained by Launder and Spalding[12].

REYNOLDS-STRESS TRANSPORT (RST) MODEL

The Reynolds stress transport model[13-15] involves calculation of the individual Reynolds stresses using differential transport equations. For the present flow configuration, five additional equations are need to be solved to obtain the flow field using a RST model. The general transport equation for Reynolds shear stress is

$$\frac{\partial}{\partial x_k}\left(u_k\overline{u_i'u_j'}\right) = -\frac{\partial}{\partial x_k}\left[\left(\overline{u_i'u_j'u_k'} + \overline{\frac{p'}{\rho}\left(\delta_{kj}u_i' + \delta_{ik}u_j'\right)}\right)\right] + \frac{\partial}{\partial x_k}\left[\nu\frac{\partial}{\partial x_k}\left(\overline{u_i'u_j'}\right)\right] + G_{ij} + \phi_{ij} - 2\nu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}} \tag{9}$$

Where G_{ij} represents stress production and Φ_{ij} pressure-strain given as

$$G_{ij} = -\left(\overline{u_i'u_k'}\frac{\partial u_j}{\partial x_k} + \overline{u_j'u_k'}\frac{\partial u_i}{\partial x_k}\right) \tag{10}$$

$$\phi_{ij} = \overline{\frac{p'}{\rho}\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right)} \tag{11}$$

The pressure-strain term ϕ_{ij} was modelled according to the proposals in the literatures[13-16].

COMPUTATIONAL METHODOLOGY

The jet was assumed to be axi-symmetric for the computational purposes. The Reynolds-averaged Navier-Stokes (RANS) equations for the primitive variables on a staggered variable arrangement were used. The geometry and the mesh were created using the commercial software GAMBIT and the numerical simulations were performed using the commercial flow solver FLUENT- 6.1.22 based on the finite volume method. While solving the flow with the k-ε and RST models, steady and implicit method was used for discretizing the equations. The pressure equation was discretized using the central differencing scheme, pressure-velocity coupling using the SIMPLE algorithm, momentum, Reynolds stress transport, turbulence kinetic energy and turbulence dissipation rate equations using the first order

upwind scheme. The first order accuracy was thought to suffice[17] because the computational mesh used was an extremely fine. For all the cases, the convergence criteria of $1x10^{-4}$ was applied based on the optimization of time and accuracy in arriving at a converged result. This is more stringent convergence criterion compared to that recommended by the FLUENT- 6.1.22 ($1x10^{-3}$) for these kinds of turbulent flows.

The computational domain used in the present study is shown in Figure 1(b). The boundary *ab* is the round jet inlet where velocity inlet condition is used. At the inlet, the flow was assumed to be fully developed turbulent flow. A top hat velocity profile was specified equal to 137 ms^{-1} according to the experimental conditions[1]. Turbulence was specified in terms of the turbulent intensity and hydraulic diameter and it was given a value of 4.14% based on the empirical relation involving Reynolds number and intensity[17]: $u'/u_{avg} = 0.16 Re_D^{-1/7}$. Here u' is the turbulent intensity, u_{avg}, the average jet velocity, Re_D the Reynolds number of the flow defined as $U_j D/\upsilon$ and ν is the kinematic viscosity. The boundary *bc* is the co-flow inlet as shown in Figure 1(b). The velocity inlet condition[1] was specified with a velocity equal to 30.5 m/s. According to the conditions in the experiments of Antonia and Bilger[1], the co-flow was assumed to be non-turbulent. The boundary *cd* is the free-stream boundary where the velocity component method was used with the specification of the *u*-velocity and assuming the *v*-velocity to be zero. The boundary *de* is the outflow boundary, where the normal gradient of all the parameters were taken to be zero. The streamwise length of the computational domain was taken as 300D and width in the radial direction as 30D. This size was considered after several test runs to compute the size of the domain that would suffice to describe the entire flow accurately using the above-mentioned boundary conditions. The ratio of the velocity of the jet to the velocity of co-flow was taken as 3.5 based on the experimental conditions[1] and similarly the diameter of inlet was taken to be 5.28 mm. The jet and the co-flowing fluid were assumed to be the same as air[1]. A uniform grid was used and the flow was solved for two different grid sizes of 1×10^6 and 2×10^6 elements. The k-ε model was employed and the profile of Reynolds shear stress was used for the grid independence study as it encapsulates important flow information (not shown here). It was found that results were grid independent after a size of 1×10^6 grid points and therefore, this mesh was selected for further computational purposes using both the k-ε and RST models.

RESULTS AND DISCUSSION

Mean Flow Field

The non-dimensional mean velocity φ(η) profiles against non-dimensional radial co-ordinate η [$\varphi(\eta) = (U - U_1)/(U_o - U_1)$ and $\eta = r/r_{1/2}$, where $r_{1/2}$ is the velocity half width] at various downstream locations predicted by both the k-ε and RST models (Figure 2) are consistent with the experimental data[1] at low x/D values {Figures 2(a) and 2(b)} and but deviate somewhat far downstream {Figures 2(c) and 2(d)}. Overall, the k-ε model provides better predictions than those by RST

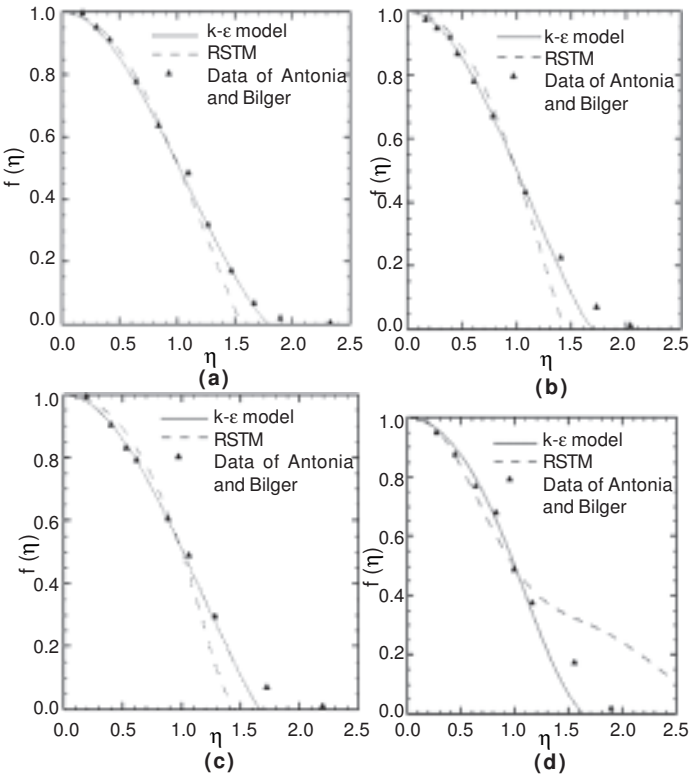
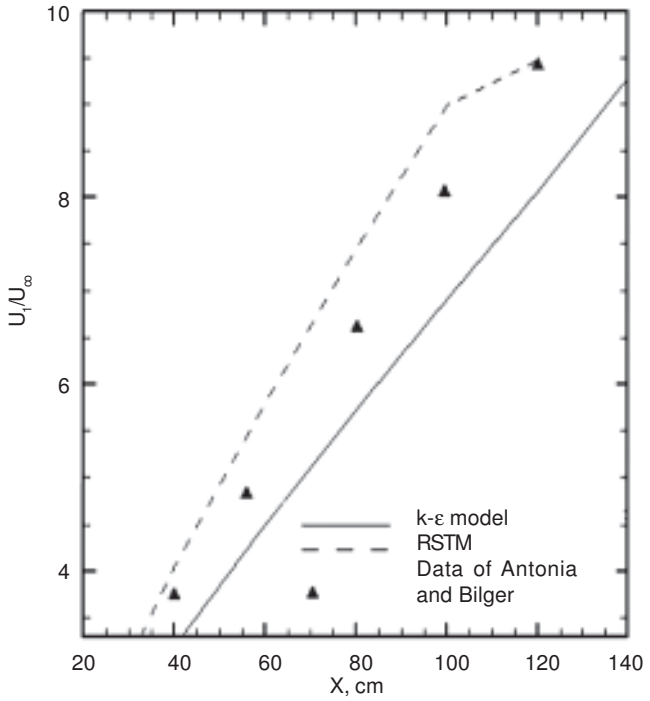


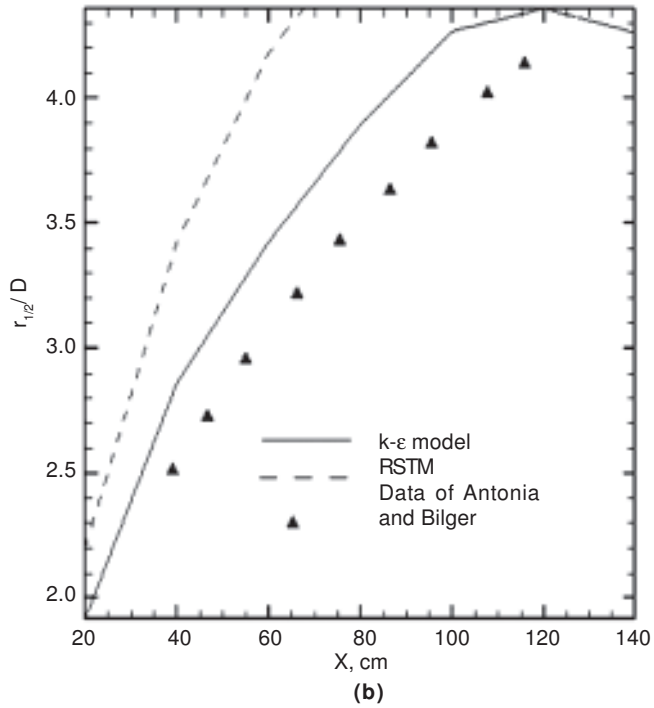
Figure 2 Non-dimensional mean velocity distributions at (a) x/D = 38, (b) x/D = 76, (c) x/D = 152 and (d) x/D = 266

model. Figure 3(a) shows that the general trend of the centreline variation of the streamwise velocity is captured well by both models. U_1/U_{co} increases along the axial direction showing that the centreline excess velocity decreases in the streamwise direction. However, the computed values are slightly offset from the experimental values[1]. The variation of the half-width predicted by both the turbulence models is non-linear as shown in Figure 3(b) compared to an almost linear variation in the experimental data[1]. Further, the predictions are not in complete accordance with the experimental data[1]. However, the k-ε model predicts half widths better than those by the RST model.

At any x/D location the centreline velocity of the present jet in coflow is larger than that of a jet in a quiescent ambient and the decay rate of the jet without a co-flow (*ie*, in a quiescent ambient) faster than that of a jet in a co-flow. The width of jet in a quiescent ambient follows a linear variation with the axial distance. On the other hand, the growth rate of a jet in a co-flow is non-linear and smaller than that in the former case. For instance, it was found that at x/D = 100, the velocity half width for the quiescent case was 20.75D and for co-flow case it was 4.5D which is 4.6 times smaller. This is because as the co-flowing fluid tries to inhibit the growth of the jet due to its velocity, the jet cannot grow freely as it encounters the co-flowing fluid and at some distance in the radial direction from axis, it attains the velocity of co-flowing fluid. However, in the case of jet in a quiescent



(a)



(b)

Figure 3 Variation of the non-dimensional (a) centreline excess velocity U_{co} and (b) velocity half-width $r_{1/2}$ along the streamwise direction

ambient, the velocity of jet will fall to zero at some point as in the radial direction is moved.

REYNOLDS SHEAR STRESS

The predicted Reynolds shear stress profiles by the k-ε model provided better agreement with the data of Antonia and Bilger[1] than those by RST model. Figure 4 presents the variation of the maximum shear stress $\overline{uv}_m$ in the streamwise direction. The experimental data show that the maximum value of the

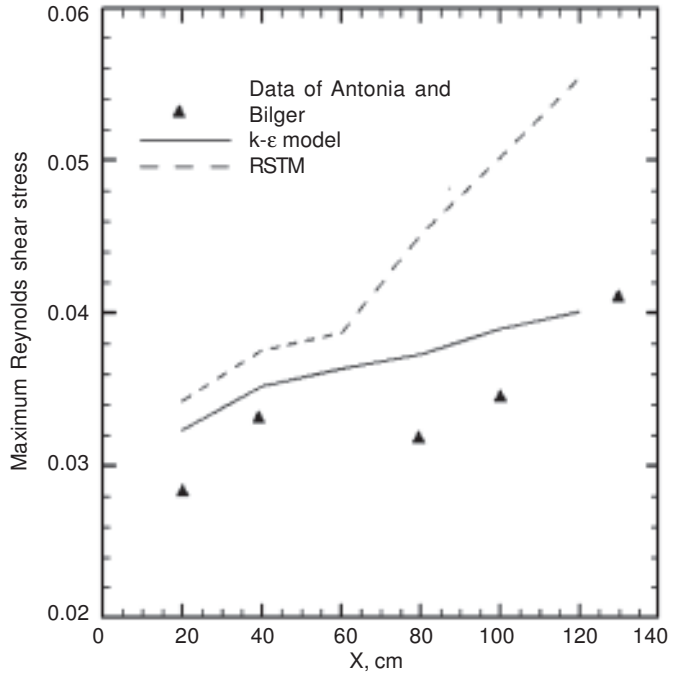


Figure 4 Variation of the maximum non-dimensional shear stress along the streamwise direction

Reynolds shear stress occurs at approximately $r/r_{1/2}$ = 0.7 at the most stream-wise locations[1]. The k-ε model predicts the maximum shear stress value at approximately the same value of $r/r_{1/2}$= 0.8 as in the experiments[1], but RST model gives a considerably different value of $r/r_{1/2}$ approximately 1.0 (not shown here). Further the RST model overpredicts the maximum shear stress value by a large amount. In general predictions by both models differ considerably from the experimental data. However, the predictions by the k-ε model are superior to those by the RST model (Figure 4). This is surprising because the RST model is the most elaborate turbulence model. In general a large deviation between the predictions by the two models and experimental data is mainly due to the inadequacy of RANS equations to model complex turbulent flows, such as, the present one. DNS or LES though computationally much more expensive compared to treatment based on RANS equations may provide more accurate results of the present flow configuration.

TURBULENCE INTENSITIES

The variation of the streamwise component of the turbulence intensity along the axis of symmetry (centreline) in the vicinity of the nozzle (including the potential core) is shown in Figure 5(a). The longitudinal turbulence intensity $\overline{u'^2}$ is normalized with the centreline excess velocity. In the computations $\overline{u'^2}$ was obtained from the expression $\overline{u'^2}$ = 2/3k with the assumption of the isotropy for both models. The trend is captured well by both models including a fall in the turbulence intensities after x = 6 cm as observed experimentally[1]. Figure 5(b) shows the u-intensity profile at x/D = 152. The predictions by the two models are not in good agreement with the experimental results[1]. However, the k-ε model performs better in this case as well. The RST model overpredicts the turbulence intensity as shown in Figure 5(b).

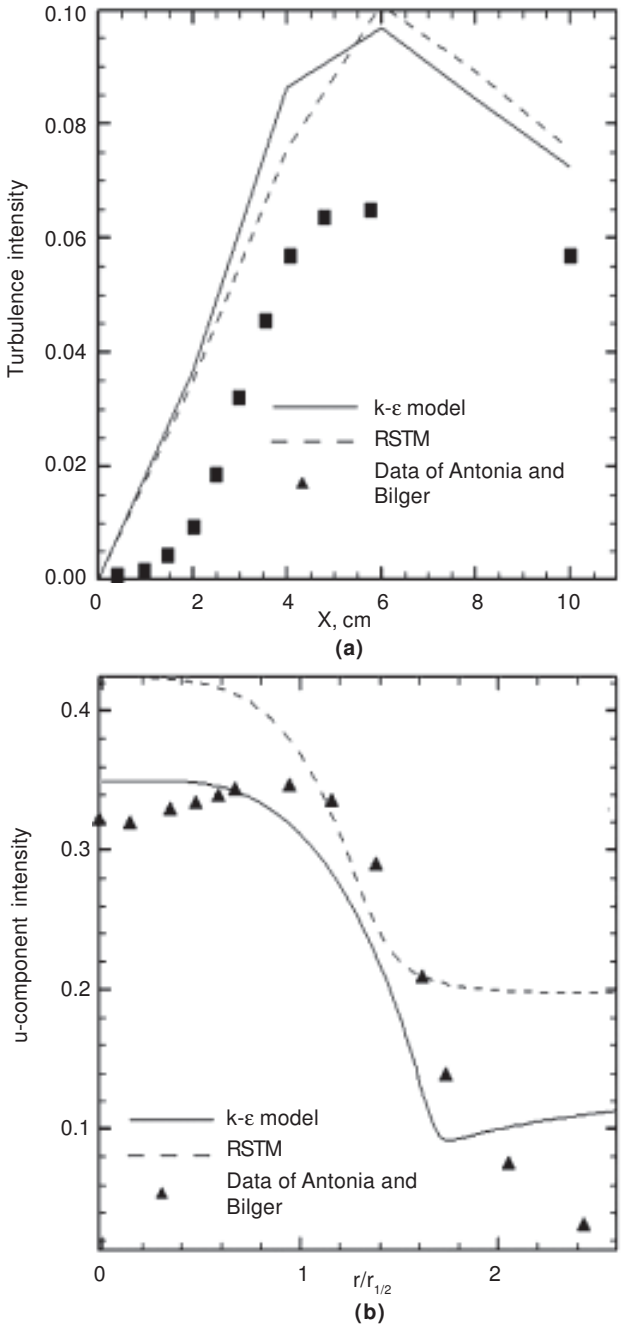


Figure 5 Non-dimensional streamwise component of the turbulence intensity (a) distribution in the vicinity of the nozzle and (b) profile across the flow at x/D = 152

CONCLUSION

The flow configuration of jet in a co-flowing ambient was solved in the present work using both the k-ε model and Reynolds stress transport (RST) model and the predictions were compared with the experimental data reported in the literature. The mean velocity profiles, turbulent flow parameters, such as, Reynolds shear stress and turbulent intensities, variation of centreline excess velocity, and velocity half-width were compared with the experimental results. In general, the agreement between the computational results and experimental ones was good. In almost all the cases the trend was captured well by both models, although in certain cases the values were not in accordance with the experimental data. Overall, the simple k-ε model predicted better than the RST model. It can therefore be concluded that it may not be always worthwhile to use the computationally intensive RST model when relatively simpler k-ε model may provide predictions of at least similar or sometimes better accuracy than those by RST model. However, it may be noted that none of two models seem to provide entirely accurate predictions of the present flow configuration. DNS or LES, which treat instantaneous, three-dimensional flows, may provide better predictions. However, these are computationally much more expensive compared to the most elaborate model for RANS equations, namely, RST model considered in the present paper.

REFERENCES

1. R A Antonia and R W Bilger. 'An Experimental Investigation of an Axisymmetric Jet in a Co-flowing Air Stream.' *Journal of Fluid Mechanics*, vol 61, 1973, p 805.

2. S Gaskin and I R Wood. 'The Axisymmetric and Plane Jet in a Co-flow.' *Journal of Hydraulics Research*, vol 39, no 5, 2001, p 451.

3. S B Pope. 'An Explanation of the Turbulent Plane Jet/Round Jet Anomaly.' *Journal of AIAA*, vol 16, 1978, p 279.

4. J R Cho and M K Chung. 'A k-ε-γ Equation Turbulence Model.' *Journal of Fluid Mechanics*, vol 237, 1992, p 301.

5. B J Boersma, G Brethouwer and F T M Nieuwstadt. 'A Numerical Investigation on the Effect of the Inflow Conditions on the Self-similar Region of a Round Jet.' *Physics of Fluids*, vol 10, no 4, 1998, p 899.

6. C L Ribault, S Sarkar and S A Stanley. 'Large Eddy Simulation of Plane Jet.' *Physics of Fluids*, vol 11, no 10, 1999, p 3069.

7. A Dewan, J H Arakeri and J Srinivasan. 'A New Turbulence Model for the Axisymmetric Plume.' *Applied Math Mod*, vol 21, 1997, p 709.

8. A Dewan, K Kalita and A K Dass. 'Computation of the Turbulent Plane Plume using k-ε-$\overline{t^2}$-γ Model.' *Applied Math Mod*, vol 24, 2001, p 815.

9. L J S Bradbury and J Riley. 'The Spread of a Turbulent Plane Jet Issuing into a Parallel Moving Air Stream.' *Journal of Fluid Mechanics*, vol 27, 1967, p 381.

10. P C K Chu, H J Lee and V H Chu. 'Spreading of Turbulent Round Jet in a Co-flow.' *ASCE Journal of Hydraulics Engineering*, vol 20, 1999, p 195.

11. J F J Maczynski. 'A Round Jet in an Ambient Co-axial Stream.' *Journal of Fluid Mechanics*, vol 13, 1961, p 597.

12. B E Launder and D B Spalding. 'The Numerical Prediction of Turbulent Flow.' *Composite Methods of Applied Mechanics and Engineering*, vol 3, 1974, p 269.

13. B E Launder and M M Gibson. 'Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer.' *Journal of Fluid Mechanics*, vol 86, 1978, p 491.

14. B E Launder. 'Second-moment Closure : Present and Future?.' *International Journal of Heat Fluid Flow*, vol 10, no 4, 1989, p 282.

15. B E Launder, G J Reece and W Rodi. 'Progress in the Development of a Reynolds-stress Turbulence Closure.' *Journal of Fluid Mechanics*, vol 68, no 3, 1975, p 537.

16. S Fu, B E Launder and M A Leschziner. 'Modelling Strongly Swirling Recirculating Jet Flow with Reynolds-stress Transport Closures.' *Proceedings of Sixth Symposium on Turbulent Shear Flows*, Toulouse, France, 1987.

17. 'FLUENT 6.1.22 User Manual.' *Fluent Inc*, Lebanon, USA, 2003.