13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

TRAJECTORIES OF A PLANE JET INTO A CROSS-FLOW

X. Wang and L. Cheng

Department of Civil Engineering
The University of Western Australia, Nedlands, WA 6907, AUSTRALIA

ABSTRACT

A plane jet into a cross flow is simulated using a RNG based k-ε turbulence model. The results are compared with the experimental results and those by LES model. It is shown that the distributions of velocity and turbulent intensity simulated by the RNG k-ε model agree well with the experimental results. The effect of jet width on the flow is studied, and it is found that the mixing process of velocity and temperature fields is only dependent on a single quantity, i.e. the momentum flux ratio. Empirical formulas of jet velocity and temperature trajectories are proposed and validated in the present study.

INTRODUCTION

Study of a heated (or cooled) jet injected normally into a cooled (or heated) cross-flow is important for the design of gas engine combustors and waste water discharges. Fluid is normally injected into a cross-flow through a single hole, a row of orifices or through a line nozzle, which were investigated by Kamotani and Greber (1972), Holdeman and Walker (1977), and Chen and Hwang (1991), respectively. It was found that at the constant width ratio the velocity and temperature trajectories were mainly dependent on the momentum flux ratio, which is defined as $J=(\rho_j u_j)^2/(\rho_b u_b)^2$, where ρ_j, and u_j are density and velocity of the jet and ρ_b and u_b are the density and velocity of the bulk flow. It was also found that the density (or temperature) ratio has only weak influence on the temperature distribution, and almost no influence on the velocity distribution. Chen and Hwang (1991) investigated the mixing characteristics of one- and dual-line heated jet injected normally into a cold cross-flow in a rectangular duct. Distributions of mean velocity, temperature and turbulent intensity in several nearby cross sections downstream the jet exit were presented. Correlation formulas of jet velocity and temperature trajectories with momentum flux ratio J and downstream distance X/b were also presented. Since only a single nozzle width was studied in their experiment, the effect of the jet width on the trajectories was not considered in the formulas. However, the experience with a similar type of flow, i.e. a side discharge into an open channel flow, indicates that the jet width has significant effect on the flow characteristics. The studies of a side discharge into an open channel flow by Mikhail et al. (1977), Demuren et al. (1983) and Wang et al. (1997) showed that the geometry of the recirculation zone formed downstream the jet is uniquely dependent on the momentum flux ratio defined as $M=(bu_j)^2/(Bu_b)^2$, where b and B are the width of jet and main channel, respectively. The theoretical study of Willi et al. (1984) and experimental study of Fujita (1990) of the channel junction flow also demonstrated the dependence of the width of the recirculation zone on the momentum flux ratio M. In addition, since the jet width normally needs to be designed in practice, the study of the dependence of the mixing process on the momentum flux ratio M is of great importance.

Jones and Wille (1996) studied the case of a single plane jet, previously investigated by Chen and Hwang (1991) using Large Eddy Simulation (LES) models. Both rectangular mesh and an adapted mesh were used with the standard Smagorinsky, one-equation and dynamic models. The calculated distribution of mean axial velocity component at a few cross sections downstream the jet exit did not agree well with the experimental results. Although the reasons for the disagreement were explained in the paper, it seems more likely that the computational mesh used (83×30×30) was too coarse for the LES models to resolve the small scales of motion in the vicinity of the wall. The experience of the present authors with the simulation of a side-discharge into a channel flow (Wang and Cheng 1997) shows that this mesh is not even fine enough to produce mesh-independent results for two-equation turbulence models.

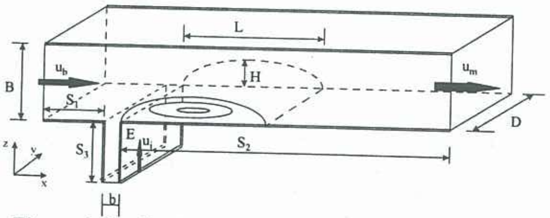


Figure 1 Configuration of a plane jet into a cross flow ($D=2B$, $b=0.0417B$, $S_1=B$, $S_2=7B$, $S_3=B$)

The purposes of the present study are to examine the effects of jet width ratio on the flow characteristics, such as jet trajectories of velocity and temperature, and to investigate the correlation between the momentum flux ratio M and the mixing process of the flow. A flow configuration of a plane jet perpendicularly injected into a cross flow is schematically shown in Figure 1.

CALCULATION DETAILS

In this study, a finite volume method embedded in Fluent 4.4®was used to solve the Reynolds-averaged Navier-Stokes equations in a curvilinear coordinate. A nonstaggered grid system was used for storage of the discrete velocities and pressure. The RNG turbulence model of Yakhot et al. (1992) was used for its improved performance in simulating separated flow over standard k-ε model. The boundary conditions used in the present study are similar to those applied in the side discharge into a open channel flow by Wang and Cheng (1997). QUICK scheme (Leonard 1979) was chosen to account for the convection terms in the momentum equations. The equations were iterated with a line-by-line matrix solver and a multi-grid acceleration to the steady state by means of SIMPLEC pressure-correction scheme. Solution was assumed to have converged when the sum of the normalized residuals for velocities, pressure, turbulent energy and dissipation rate fell below 0.001, and residual for enthalpy below 1.0×10^{-5}. Firstly, simulations were carried out for a case with nozzle open ratio b/D=0.0208 (see Figure 1). This flow geometry was experimentally investigated by Chen and Hwang (1991) and numerically studied by Jones and Wille (1996). The Reynolds number, based on duct height B and bulk velocity u_b was Re=19,000, while the velocity ratio of jet velocity and the inlet cross flow R (=u_j/u_b) is 7.34. The Reynolds number based on the nozzle width b and the jet exit velocity was Re=5815. Three other nozzle open ratios b/B=0.0167, 0.0667 and 0.1 were also investigated in combination with five momentum flux ratios M=0.0417, 0.167, 0.375, 1.0417 and 2.245, respectively. Smooth wall assumption is adopted for all wall boundary conditions in the present study, and the density of the fluid is assumed constant. The calculation results presented in this paper, such as distributions of velocity and turbulent intensity, and velocity and temperature trajectories, are extracted from the symmetrical plane of Y=0.5D.

In numerical simulations, the following two factors may strongly affect the accuracy of the simulation, if they are not considered properly: (1) the size of the computation domain; (2) the density and distribution of the grid. Wang and Cheng (1997) conducted a mesh-independence study for a side discharge into an open channel flow, in which the effects of the computation domain and computation grids were thoroughly investigated. Since the flow to be studied here is very similar to the flow studied by Wang and Cheng, a calculation domain as illustrated in Figure 1, with an $120\times45\times30$ non-uniform mesh are used in the present study, based on their results. Boussinesq approximation was used to examine the effect of buoyancy induced by temperature difference, and it was found there was no obvious difference for the calculated velocity and temperature distributions. Therefore, all results presented in this paper were obtained without considering the buoyancy effect.

RESULTS and DISCUSSION

Velocity and Temperature Distribution

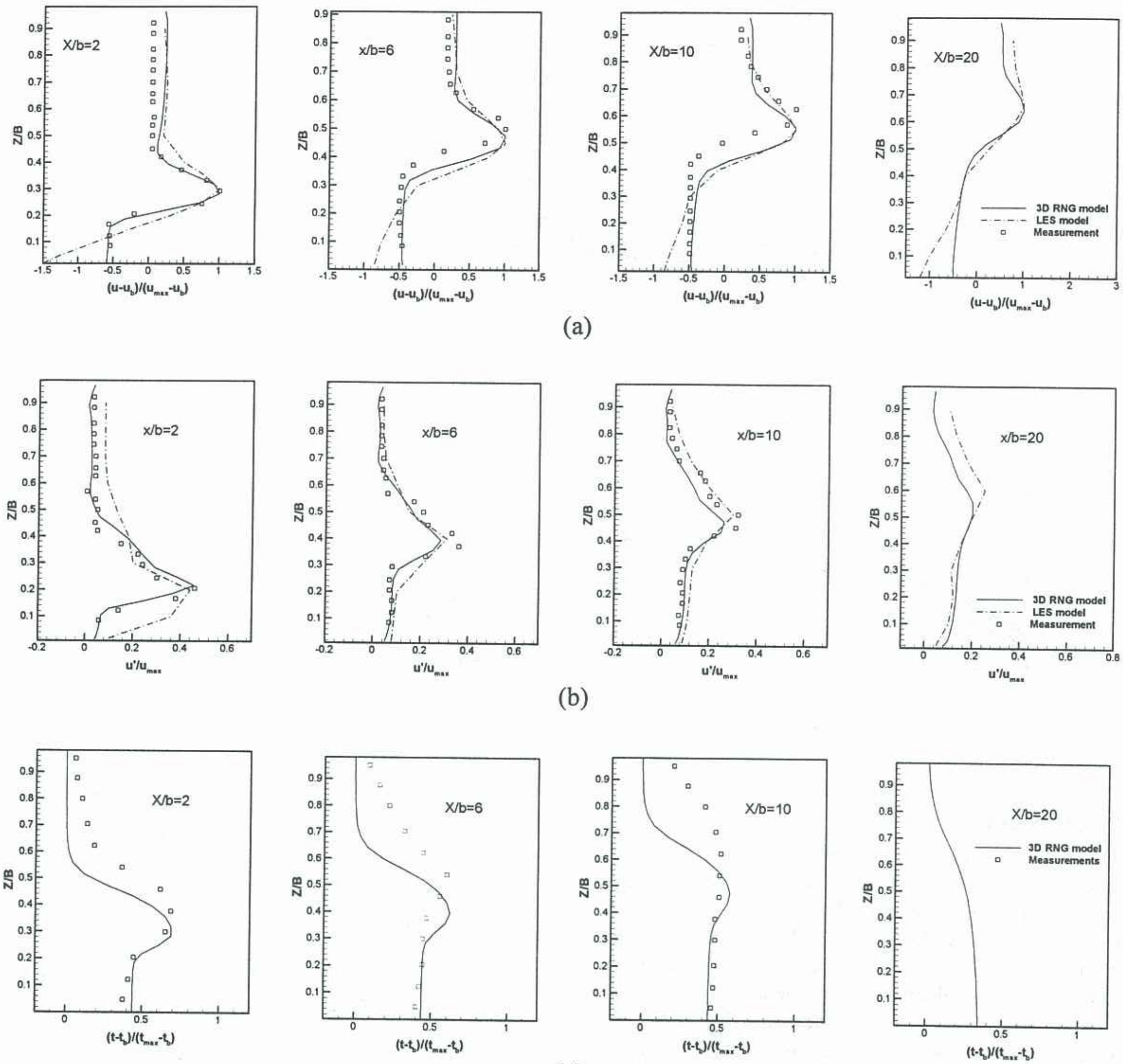
Figure 2 presents the normalized (a) mean axial velocity, (b) turbulent intensity and (c) normalized temperature profiles, at four cross sections (X/b) relative to the jet exit E. To compare with the experimental results by Chen and Hwang (1991), the calculation results are presented in the following non-dimensional form:

$$\widetilde{U}=\frac{u-u_b}{u_{\max}-u_b}\quad \widetilde{T}=\frac{t-t_b}{t_{\max}-t_b}\quad I=\frac{u'}{u_{\max}} \tag{1}$$

where, u_{max} and t_{max} are the maximum mean axial velocity and temperature in a cross section at a downstream location X/b; u_b and t_b are the velocity and temperature of the inlet cross flow. u' is the rms value of fluctuating velocity component. Results of mean velocity and turbulent intensity obtained with a dynamic LES model by Jones and Wille (1996) are also included in Figure 2. It is shown that good agreement between the simulated velocity distributions by the three-dimensional RNG k-ε model and the measured velocity distributions by Chen and Hwang (1991) exists. Since only the normalized axial velocity component was presented in the experimental results of Chen and Hwang (1991), it is impossible to identify the actual experimental maximum axial velocity component. To give an indication, the maximum axial velocity component obtained by the present three-dimensional RNG k-ε model at X/b=2.0, 6.0, 10.0, and 20.0 are $2.45u_b$, $3.07u_b$, $3.29u_b$, and $3.42u_b$ respectively. Generally speaking, the RNG k-ε model and the LES model gave almost identical velocity distributions in most of the flow regions, except inside the separation zone immediately downstream the jet. It is seen that the axial velocity distributions given by Jones and Wille (1996) using the LES model depart considerably from the experimental results, as well as from the present numerical results. The reason for that may be due to the use of a rather coarse mesh in Jones and Wille's calculations. In the LES model of Jones and Wille (1996), the wall stress was assumed to be linearly related to the instantaneous spatially averaged tangential velocity at first grid node adjacent to the wall. This requires that the first grid node be placed within the viscous sublayer. But the relatively coarse mesh ($87\times30\times30$) used in their study did not seem to meet this requirement. In addition, the coarse mesh very likely results in excessive levels of numerical diffusion in the calculation. This may explain why large velocity gradients were predicted in the near field of the wall in Jones and Wille's study.

The axial turbulent fluctuations, defined as u'/u_{max}, where u' is the rms value of the fluctuating velocity, are presented in Figure 2(b), to compare with the experimental results of Chen and Hwang (1991) and the LES results of Jones and Wille (1997). The rms value of the fluctuating axial velocity obtained using the RNG k-ε model was derived from the calculated velocity field. It is seen that both RNG k-ε model and the LES model predicted the axial turbulent fluctuations quite well, compared with the experimental results. It can be observed that the maximum u'_{max} occurs roughly at the same height as the maximum axial velocity component, although the numerical models tend to slightly under-predict this height at most of the cross sections given in Fig. 2(b). The disappearance of the experimental data at section of X/b=20 was due to the limited experiment data presented in Chen and Hwang (1991).

The temperature distributions predicted using the RNG k-ε model together with the experimental results of Chen and Hwang (1991) are presented in Figure 2 (c). The simulation and the experiment show the same tendency of the thermal mixing process that the temperature increases with the increasing height z/B to a maximum value and decrease monotonously thereafter. Because of the energy exchange of a hot jet and a cold cross flow, the position of the maximum temperature at each location moves up from the jet side with the downstream distance. However it can be observed that the simulated heights of the maximum temperature are generally lower than those by the experiment. This difference increases with the downstream distance. Although considerable differences exist for the temperature distributions, the model performance on the prediction of temperature is quite consistent with the model performance on the prediction of the mean velocity components and the velocity fluctuations, where the predicted heights of the maximum values of velocity and the fluctuating velocity are generally slightly lower than those obtained experimentally.



(a)

(b)

(c)

Figure 2 Comparison of (a) velocity, (b)turbulent intensity and (c) temperature at different locations

Velocity and Temperature Trajectories

The velocity (or temperature) trajectory is defined as the curve formed by the position of the maximum axial velocity (or temperature) at each cross section downstream the jet. Velocity and temperature trajectories are important to depict jet mixing and spreading process. Chen and Hwang (1991) experimentally investigated the velocity and temperature trajectories within X/b=10 downstream the jet. Correlation formulas of velocity and temperature trajectories with the momentum flux ratio J and downstream distance X/b were proposed using the least-squares regression method. Since only one nozzle width b=5mm was investigated, the effect of nozzle width on the trajectories was not included in the formulas. This greatly limits the use of these formulas. To investigate the influence of the nozzle width on the trajectories, calculations were carried out with different nozzle widths at a constant value of J. Figure 3 shows the velocity trajectory calculated with different nozzle widths at J=50.4. It can be seen from the figure that the calculated velocity trajectory agrees very well with that obtained experimentally (Chen and Hwang, 1991) for b/B=0.0417, and the velocity trajectory is strongly dependent on the nozzle width. This suggests that the trajectory is not uniquely dependent on J. Therefore any formula derived in terms of the momentum flux ratio J is only of limited value.

In order to study the dependence of the trajectories on the momentum flux ratio M, calculations with four nozzle widths b/B=0.0167, 0.04167, 0.0667 and 0.1 in combination with five momentum flux ratios M=0.0417, 0.167, 0.375, 1.0417 and 2.245, were carried out. Figure 4 shows the calculated velocity and temperature trajectories with different nozzle widths and momentum flux ratio M. It can be found that the trajectories depend on the momentum flux ratio M. Based on the calculation results, correlation formulas for the velocity and temperature trajectories, expressed in terms of M, are presented in Figure 4. It can be observed that excellent correlation exists between the data obtained with different nozzle widths. However, caution needs to be taken when these formulations are used. The correlation formulas are expected to be accurate only in the near field of jet. In the further downstream region, the velocity and temperature trajectories are almost parallel to the cross flow and can not be represented using the current formulas.

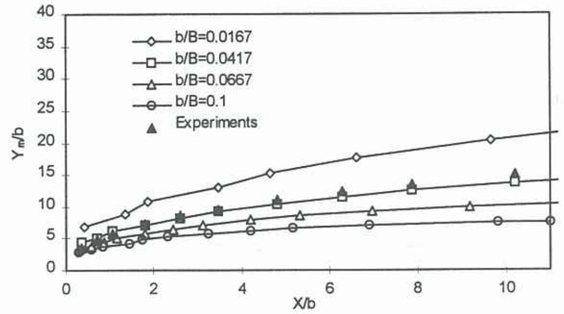
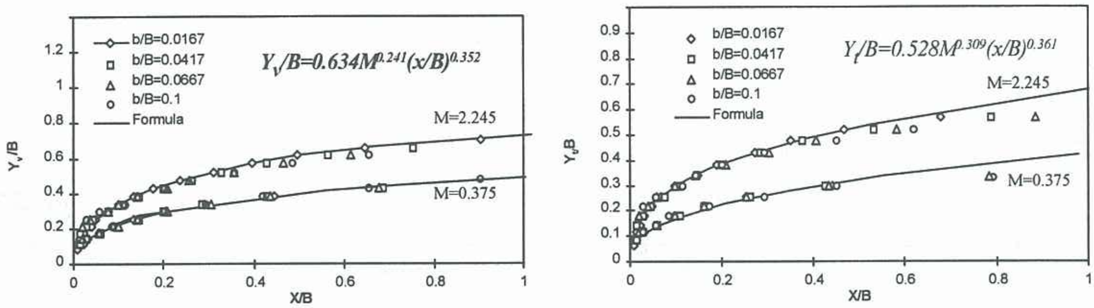


Figure 3 Influence of nozzle width on the velocity trajectory (J=50.4)



(a) (b)

Figure 4 Correlation of jet trajectories of (a) velocity and (b) temperature

CONCLUSION

A plane jet injects normally into a cross flow has been studied using a RNG based k-ε model. The calculated velocity distribution and turbulent intensity at different positions downstream from the jet, show good agreement with the experiment results. The jet width has great influence on the velocity and temperature trajectories, which mainly depend on momentum flux ratio M. Correlation formulas of jet velocity and temperature trajectories, expressed in terms of the momentum flux ratio M and downstream distance X/b are proposed.

REFERENCES

Chen, K. S. and Hwang, J. Y., "Experimental study on the mixing of one- and dual-line heated jets with a cold crossflow in a confined channel", *AIAA Journal*, Vol. **29**, No. 3, 353-360, 1991.

Demuren, A. O. and Rodi, W., "Side discharge into open channels: mathematical model." *J. of Hydr. Engrg.*, ASCE, 109(12), 1707-1722, 1983.

Fujita, I., "*Study of the flow structure at a confluence*", Ph.D. thesis (in Japanese), Gifu University, Japan, 1990.

Jones, W. P. and Wille, M., "Large-eddy simulation of a plane jet in a cross-flow", *Int. J. Heat and Fluid flow*, **17**, 296-306, 1996.

Holdeman, J. D. and Walker, R. E., "Mixing of a row of jets with a confined crossflow", *AIAA Journal*, Vol. **15**, 243-249, 1977.

Kamotani, Y. and Greber, I., "Experiments on a turbulent jet in a cross flow", *AIAA Journal*, Vol. **10**, 1425-1429, 1972.

Mikhail, R., Chu, V. H. and Savage, S. B., " The reattachment of a two-dimensional turbulent jet in a confined cross flow."*Proc. 16th IAHR Cong., Sao Paulo, Brazil,* Vol. 3, 414-419, 1975.

Willi, H. and Hager, M., " Separation zone at open-channel junctions", *J. of Hydr. Engrg.*, ASCE, 110(11), 539-543, 1984.

Wang, X. and Cheng L., "Numerical analysis of a side discharge into a open channel flow", *J. of Hydraulic Research*, IAHR. 1997. (submitted).

Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B., and Speziale, C. G., "Development of turbulence models for shear flows by a double expansion technique", *Phys. Fluids, A* 7, 1510-1520, 1992.