On the flow interactions of multiple jets in cross-flow

MOHAMED MAIDI and YUFENG YAO
Faculty of Engineering
Kingston University
Roehampton Vale, Friars Avenue, London, SW15 3DW
UNITED KINGDOM
http://www.kingston.ac.uk

Abstract: - Direct numerical simulations have been performed to study the behaviour of multiple square jets issuing normally into a cross-flow. The jets are arranged side-by-side in spanwise direction in twin jet configuration and with addition third jet downstream along the centre line in tandem jets configuration. The simulations are carried out for the jet-to-jet adjacent edge distances of 1D and 2D in twin jets case and 2D for in tandem jets case, where D is the jet width. The simulation uses the jet to cross-flow velocity ratio of 2.5 and the Reynolds number based on the free-stream quantities and the jet width is 225. The predicted flow features show that the downstream merging between the two counter rotating vortex pairs (CRVP) in the twin jets configuration is strongly dependent on the jet-to-jet edge distance. Further downstream, the far-field of side-by-side jets is mostly dominated by a larger CRVP accompanied with a smaller inner vortex pair. The originally inner vortex is found no longer to be survived and it has been almost dissipated before exiting the computational domain. The obtained results are in good qualitative agreement with the experimental findings in the literature. The resulting flow structures associated with both the twin and the tandem jets in cross-flow configuration show complex flow interactions between individual jets. Indeed, the interactions between the three jets and cross-flow cases have led to a complex vortex system in which six vortex pairs can be identified. The evidence of these flow structures could provide valuable information for related industrial applications.

Key-Words: Direct numerical simulation, Multiple jets in cross-flow, Jet-to-jet spacing, Vortex structures, Cross-flow entrainment.

1 Introduction

The jet in cross-flow (JICF) dynamics has long been a subject for extensive investigation by numerous research workers due to its great practical relevance in various engineering and industrial applications. Examples include mixing and pollutant dispersion from chimney stacks, film cooling of turbine blades, V/STOL aircraft. Margason [1] provided a review paper covering fifty years of jet in cross-flow research up to 1993.

By using smoke-wire visualization techniques, Fric and Roshko [2] identified the flow features of a JICF with four main vortex structures: the horseshoe vortices, the jet shear layer vortices, the wake vortices, and the counter-rotating vortex pair. The horseshoe vortices form upstream of the jet exit and wrapping around the exiting jet orifice. The jet shear layer consists of the vortex rollers in the upstream side of the jet. The wake structures form downstream of the jet column, which persist and convectively transport to further downstream of the exit nozzle. The counter-rotating vortex pair (CRVP) which is originated as an effect of the bending of the jet itself constitutes the dominant structure of the vortex system. These observations were also confirmed by other researcher, e.g., Smith *et al.* [3], Smith and Mungal [4], Lozano *et al.* [5], Eiff *et al.* [6].

Although multiples jets are commonly used in gas turbine film cooling applications, the available literature and research on the latter is very limited. Schwendemann [7] conducted an experimental study on multiple jets in cross flow in a subsonic wind tunnel. He produced data for jet trajectories of tandem jets injected normal and inclined to a cross stream as well as data for side-by-side jets injected normal to a cross stream. Ziegler and Wooler [8] proposed a physical model for the study of the flow of a double jet system exhausting normally into a cross-flow for both a side-by-side and tandem orientation. The integral model was also extended for single jet to handle the flow of two jets with different relative distances between them. They assumed that the deflection of each jet was due to both the entrainment of the mainstream fluid and the pressure forces acting on the boundary of each jet.

Toy *et al.* [9] investigated the interaction of twin side-by-side and inline JICF. The mixing region in the far-field is reported to be similar in shape but larger for twin JICF than for single JICF. This means especially that in the far-field of side-by-side jets there is only one CRVP. The mechanism for the existence of the one CRVP is still unclear. Ibrahim and Gutmark [10] experimentally studied the penetration of such twin JICF by PIV. They demonstrated that the jet in the lee-side deflects is less while compared to the jet in the wind-side.

While recent experiments have shed much new light upon the effects of large-scale structures in the JICF configuration, many questions remain unanswered. In general, numerical simulations could provide more complete information on the complex flow structures and their interactions, but most simulations was unable to accurately reproduce the complicated flow behavior not until recently with the advancement in numerical method in conjunction with powerful parallel computing such as direct numerical simulations (DNS). DNS has become indispensable to obtain extract information, which are often difficult or sometime even impossible to obtain at the laboratory conditions.

The present paper aims to investigate the flow interactions associated with multiple jets in cross-flow by direct numerical simulation. An in-house parallel solver is used to simulate the flow field. The study has focused on the flow physics and the underneath mechanisms of the interactions.

2 Numerical Method

2.1 Governing equations

In the present numerical investigation, the compressible time-dependant three dimensional Navier-Stokes and energy equations are solved. By using reference values at free-stream, the non-dimensional form of these equations can be written as

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u_i}{\partial x_i}=0 \qquad (1)$$

$$\frac{\partial \rho u_i}{\partial t}+\frac{\partial \rho u_i u_j}{\partial x_j}=-\frac{\partial p}{\partial x_i}\delta_{ij}+\frac{1}{\mathrm{Re}}\frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

$$\frac{\partial E}{\partial t}+\frac{\partial (E+p)u_j}{\partial x_j}=-\frac{\partial q_j}{\partial x_j}+\frac{1}{\mathrm{Re}}\frac{\partial u_i \tau_{ij}}{\partial x_j} \qquad (3)$$

where ρ is the density, u_i the velocity components, p the thermodynamic pressure, E the total energy, τ_{ij} the shear stress tensor, q_i the heat flux vector, and the Reynolds number $\mathrm{Re}=U_\infty D/\nu$ based on the free-stream velocity U_∞ and the jet width D.

Assuming thermally perfect gas, the shear stress τ_{ij} and heat flux q_i can be written as

$$\tau_{ij}=\mu\left(\frac{\partial \rho u_i}{\partial x_j}+\frac{\partial \rho u_j}{\partial x_i}-\frac{2}{3}\frac{\partial \rho u_k}{\partial x_k}\delta_{ij}\right), \qquad (4)$$

$$q_j=\frac{-\mu}{(\gamma-1)\mathrm{Pr}\,\mathrm{Re}\,M^2}\frac{\partial T}{\partial x_j}, \qquad (5)$$

where the viscosity μ is specified through the power law ($\mu=T^{\Omega}$ with T the non-dimensional temperature and Ω is a constant which equals to 0.76 for air) and the Prandtl number *Pr* is set to 0.72.

2.2 Numerical technique

In the present investigation, the three-dimensional compressible Navier-Stokes and energy equations are numerically solved by using high-order finite difference scheme in space and multi-stage Range-Kutta algorithm for time advancement. An entropy splitting concept is used to improve the stability of the numerical scheme, and stable boundary treatment technique is adopted at the boundaries. The Navier-Stokes equations The code has been parallelized using the MPI library and code validations has been performed previously through various projects with numerous configurations including laminar and turbulent boundary layers and channel flows ([11, 12]).

3 Problem configuration

The problem configuration used in the simulations consists of square jets issuing perpendicularly into a main cross-flow domain. For twin jets case, the computational box has a rectangular cross-flow domain with dimensions of 24D×8D×8D (note: the spanwise length was increased to 9D for edge distance of 2D case) in longitudinal, wall-normal and transverse directions, respectively, and two jet hole domains of 1D×1D, with D the jet width The jets are located in [4D, 5D] from the cross-flow inlet plane in the streamwise direction side-by-side and the distance of 2.5D to side-planes of main domain was used, similar to that in single JICF case [13]. For tandem jets case, one additional third jet was introduced along the centre line, with edge distance of 2D to the trailing-edge of upstream twin jets. The computational grid has 241 points in the streamwise (with 11 points in the jet domain), 81 points in the wall-normal, and 81 points in the spanwise direction (with 11 points in both two jet domains, increasing to 91 points while two jets has a edge distance of 2D). The cross-flow velocity profile is initialized using a similarity solution of laminar boundary-layer at same Reynolds number and a Poiseuille-type profile is given at the inlet of jet orifice. The characteristics boundary conditions are used at both outlet plane and upper surface, and periodic conditions for the side-walls. Simulations are performed for a jet to cross-flow velocity ratio R = 2.5 and a Reynolds number Re = 225, based on the free-stream quantities of the cross-flow and the jet width (D). The low Reynolds number used in this investigation permits the clear observation of large-scale structures dynamics, which are difficult to study at high Reynolds number regime.

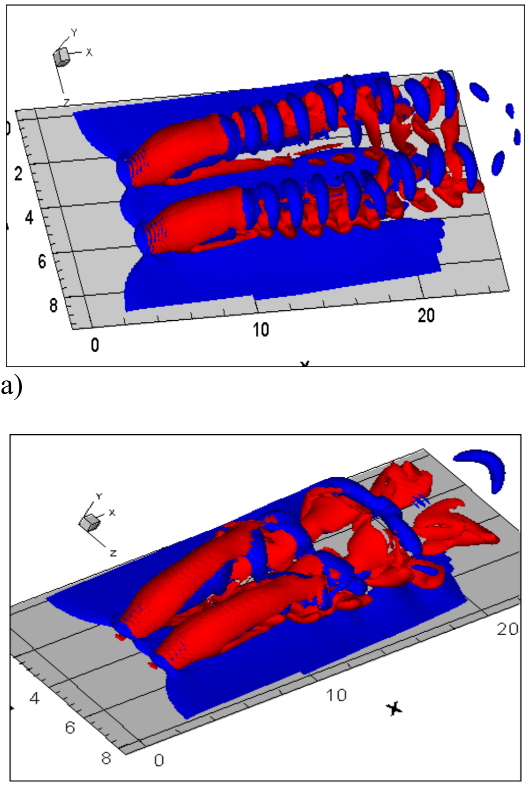
4 Results and Discussions

In the present study we present our results at a particular instant of time, t = 24, since the main objective of the present work is to obtain a deeper insight into the physical mechanism of multiple jets in cross-flow.

4.1 Twin jets in cross-flow (TJICF)

In this section we aim to investigate the vortex interaction mechanisms associated with twin jets issuing normally into a cross-flow. The two jets are placed side-by-side in spanwise direction with nozzle spacing of 2D and 1D. We focus on the merging mechanism of two adjacent jets and jets cross-flow entrainment phenomenon.

Fig. 1 (a) and (b) shows the resulting iso-surfaces of spanwise vorticity for the two configurations of twin jets in cross flow, whose nozzle spacing corresponds to 2D and 1D respectively. At the first stage the two jets produce two CRVPs similar to those observed in single JICF. Further downstream the two jets attract each other and start to merge. As shown in this figure, the merging mechanism between the two jets depends strongly on the jet-to-jet spacing. At closely separation distance (nozzle spacing of 1D), the merging process starts at about x= 10D, while by doubling the nozzle spacing the two jets seem to merge starting from x= 20D. This is also clear in Fig. 2 representing the spanwise vorticity contours in vertical median plane (x, y z= 4D) between the jets. Further downstream, the flow is shown to be dominated by one single vortex pair. It is worth noting that due to the strong intermittent interaction between the two jets, Kelvin Helmholtz rollers in the upstream side of the jets are not clear by nozzle spacing of 1D (Fig. 1(b)).

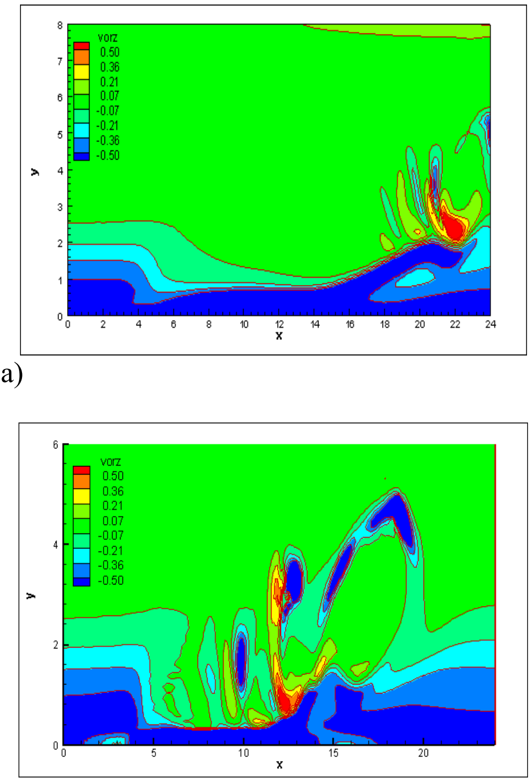


(a)

(b)

FIG 3: Simulated iso-surfaces of the spanwise vorticity ($\omega z=0.5,\ u^2/D^2$, red positive and blue negative): (a) twin jets with nozzle spacing of 2D , (b) twin jets with nozzle spacing of 1D.

For nozzle edge spacing of 2D and 1D TJICF configurations, Fig. 3 and Fig. 4 show velocity vector in the plane (z, y) at two x locations. This figure indicates that in the zone between the two jets the entrainment is disturbed and the jets bend towards the other one as each jet tries to entrain the other. Further downstream, the two jets to form one single CRVP instead of two. Fig. 4 (a) indicates the two CRVPs are quenched and leads one near wall vortex pair. The originally inner vortices seem to not survive for longer and dissipate at about x=20D. This agrees with the measurements of Toy *et al.* [9]. It is worth noting that the intensity of the inner vortex pair by wide jet spacing (nozzle spacing of 2D) is very weak compared to that found in closely jet spacing configuration case.

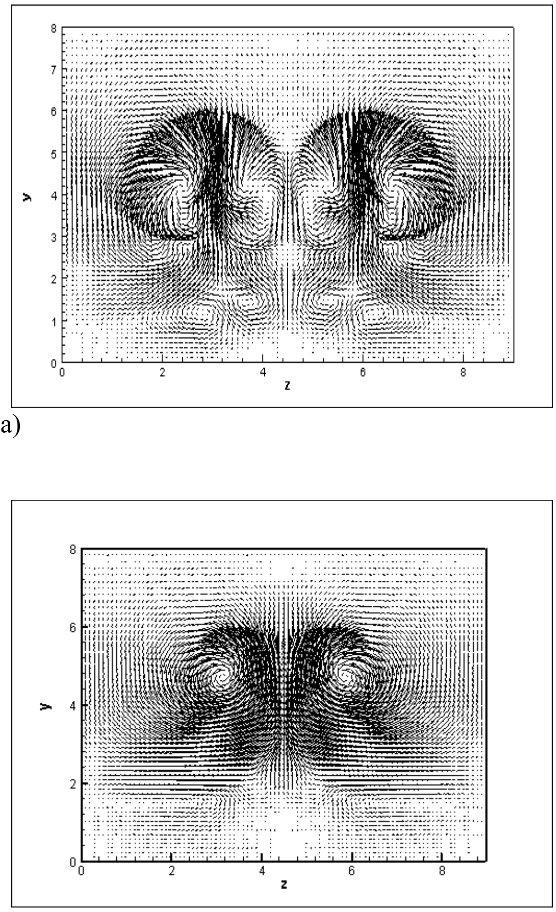


(a)

(b)

FIG 2: Contours of spanwise vortices ω_z at z =Lz/2 plane: (a) twin jets with nozzle spacing of 2D , (b) twin jets with nozzle spacing of 1D.

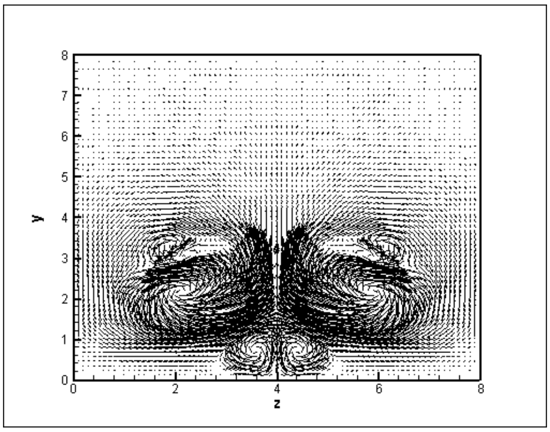
Figure 5 shows the simulated streamlines at (x, z, y=0.1D) plane for both cases of TJICF. One can see that the deflection and entrainment mechanisms of cross-flow in the wake region are strongly dependent on the jet-to-jet spacing. Indeed, by decreasing the jet-to-jet spacing, the flow separation and cross-flow penetration establishing, respectively, upstream and downstream of the jet hole are different to these predicted by wide gaps. Fig 5 shows also that the reverse flow between the two jets increases by decreasing the jet-to-jet spacing.



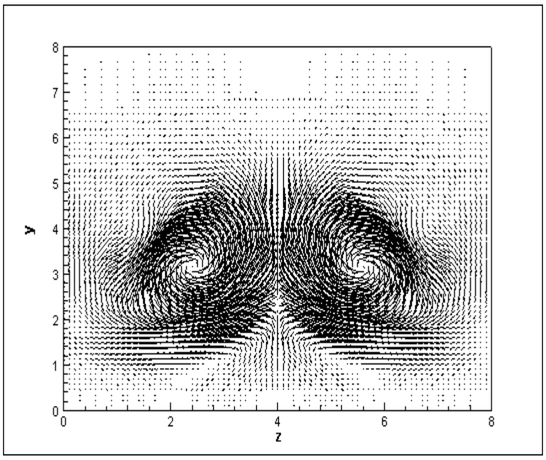
(a)

(b)

FIG 3: Vector plots associated with twin jets in cross-flow with nozzle spacing of 2D at two streamwise locations: (a) at x=20D, (b) at x=23D.

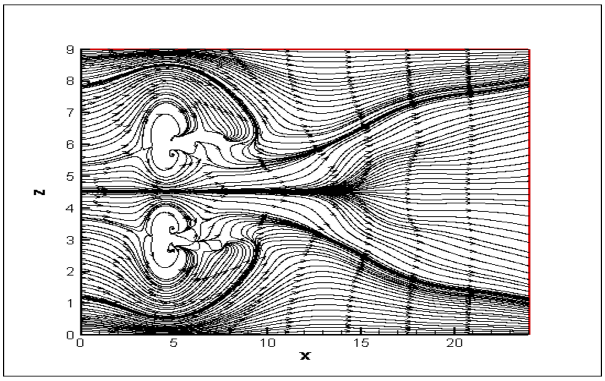


(a)

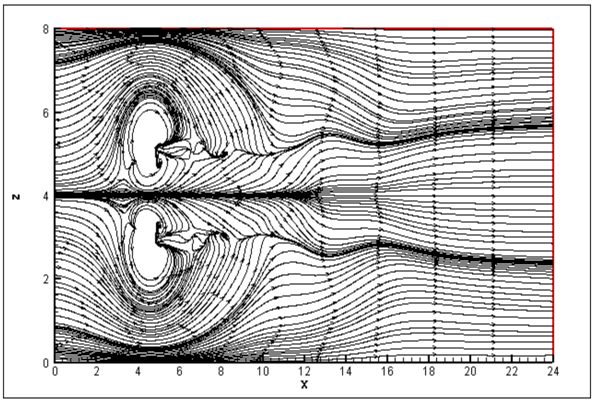


(b)

FIG 4: Vector plots associated with twin jets with nozzle spacing of 1D at two streamwise locations: (a) x=20D, (b) x=23D.



(a)



(b)

FIG 5: Streamline pattern on the horizontal plane (X, Y/D=0.1, Z).

4. 2 Multiple tandem jets in cross-flow

We now study the vertical structures interaction associated with twin front jets placed side-by side in the spanwise direction and rear jet injected normally into cross-stream.

The simulated iso-surfaces in Fig. 6 show a complex flow interaction between the three jets. The lateral spreading of the rear jet seems to be more significant than that in single jet. This is mainly due to the fact that the rear jet is protected by the twin front jets from the oncoming cross-flow.

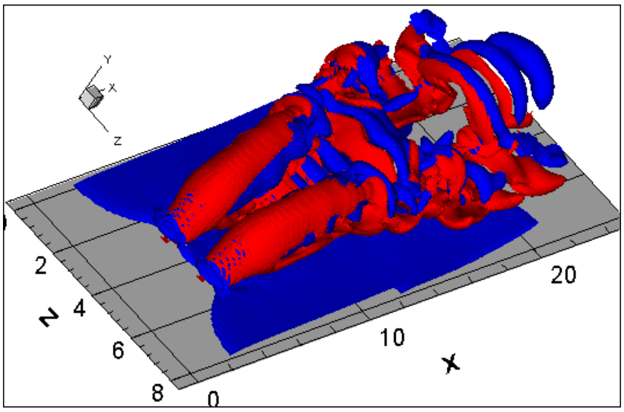


FIG 6: Simulated iso-surfaces of the spanwise vorticity ($\omega z=0.5,\ u^2/D^2,$ red positive and blue negative): (a) twin jets with nozzle spacing of 2D, (b) twin jets with nozzle spacing of 1D.

Fig. 7 shows velocity vector in the plane (z, y) at streamwise location x=18D. The interaction between the three jets and cross-flow has led to the complex vortex system shown in this figure in which six smaller vortex pairs can be identified

The simulated streamlines in the (x, y=0.1D, z) plane are presented in Fig. 11. The rear jet seems to be protected by the front jets from the oncoming cross-flow and so deflects much less with respect than a single JICF. Indeed, the presence of the twin front jets hinders the entrainment characteristics of the rear jet. Also the presence of the rear jet affects the front jets deflecting and entrainment.

Reverse or back flow region in the centre plane formed by the twin front jets are also shown in Figures 7. It may be seen that the magnitude of the reverse flow region in the centre plane upstream the side-by-side jets is decreased compared to that shown in twin jets case (Fig. 5).

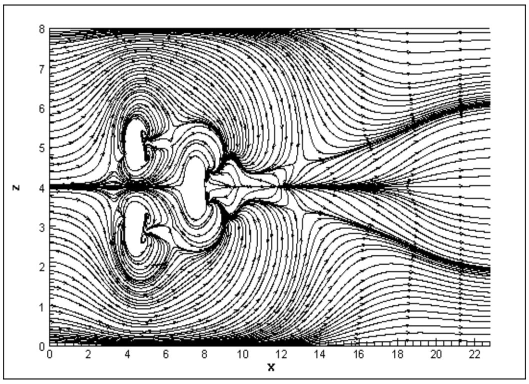


FIG 7: Streamline pattern on the horizontal plane (X, Y/D=0.1, Z).

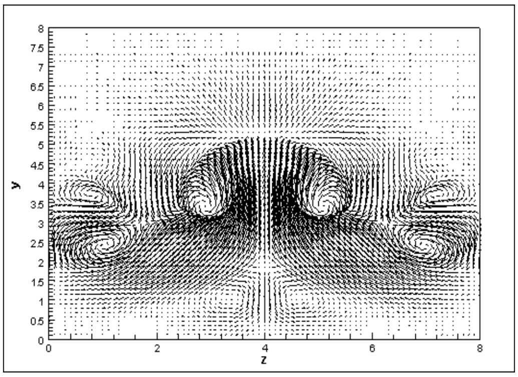


FIG 8: Vector plots associated with combined twin jets third downstream jet at streamwise locations x=18D.

CONCLUSIONS

The interaction between laminar multiple jets exiting in a cross-flow are examined using direct numerical simulations. Two cases of twin jets configuration arranged side-by-side with nozzle spacing of 1D and 2D (with D the jet width are studied). The third case of study consists of a combination of twin jets with nozzle spacing of 1D and a third downstream jet with inline nozzle spacing of 2D.

The flow features of twin jets show that the downstream merging between the two CRVPs is strongly dependent on the jet-to-jet spacing. At closely separation distance, the merging process starts earlier with respect to wide spacing jets case. Further downstream, the flow is shown to be dominated by one single vortex pair for both cases of jet-to-jet spacing.

Further downstream, the far-field of side-by-side jets is dominated by only one CRVP accompanied with inner vortex pairs. Our simulations identified also the presence of inner vortex pairs accompanying the CRVP. The originally inner vortices are shown to not survive for longer and dissipate before the computational domain exit. The obtained results are in good qualitative agreement with existing experimental findings in the literature.

The simulated flow structures associated with combined twin and inline jets in cross-flow show a complex flow interaction between the jets. Indeed, the interaction between the three jets and cross-flow has led to a complex vortex system in which six vortex pairs can be identified.

Acknowledgements

The support of the UK Engineering and Physical Science Research Council (EPSRC) through the research grant (EP/C014979/01) is gratefully acknowledged

References:

[1] Margason, R.J., Fifty year of jet in cross flow research, AGARD-CP-534, 1993, pp. 1-41.

[2] Fric, T.F. and Roshko, A.**,** Vortical structure in the wakeof a transverse jet, *J. Fluid Mech*., **279:1-47,** 1994, .

[3] Smith, S.H., Lozano, A., Mungal, M.G. and Hanson, R.K., Scalar mixing in the subsonic jet in cross-flow. AGARD-CP-534, 1993, pp. 6.1-6.13.

[4] Smith, S.H. and Mungal, M.G., Mixing structure and scaling of the jet in crossflow, *J. Fluid Mech,* **357:83-122,** 1998.

[5] Lozano, A., Smith, S.H., Mungal, M.G., Hanson, R.K., Concentration measurements in a transverse jet by planar laser-induced fluorescence of acetone, *AIAA J*., **32:** 218-221, 1994.

[6] Eiff, O.S., Kawall, J.G., Keffer, J.F, Lock-in of vortices in the wake of an elevated round turbulent jet in a crossflow. *Exps. Fluids,* **19:** 203-213, 1995.

[7] Schwendemann, M.F., A Wind Tunnel Investigation of Stratified Jets and Closely Spaced Jets Exhausting into a Cross Flow, Northrop Aircraft Division, Hawthorne, CA, Rept. NOR 73-98, May 1973.

[8] Ziegler, H. and Wooler, P.T., Multiple Jets Exhausting into a Crossflow, *Journal of aircraft*, **8, No. 6:** 414-420, 1971.

[9] Toy N., Savory E., McCusker S. and Disimile P.J., The interaction region associated with twin jets and a normal crossflow, Proc 72nd AGARD Fluid Dynamics Panel Symposium on Computational and Experimental Assessment of Jets in Crossflow, Winchester, April 1993, pp. 7.1-7.10.

[10] Ibrahim I.M. and Gutmark E.J. Dynamics of Single and Twin Circular Jets in Cross Flow, AIAA-2006-1281. In 44th AIAA Aerospace Science Meeting Proceedings. AIAA, 2006. ISBN 1-56347-807-2.

[11] Yao, Y., Lawal, A.A., Sandham, N.D., Wolton, I., Ashworth, M., Emerson, D., Massively parallel simulation of shock/boundary-layer interactions, in Proc. of Applied CFD, Beijing, China, 2000, pp. 728-735.

[12] Sandham, N.D., Li, Q., Yee, H.C., Entropy splitting for high-order numerical simulation of compressible turbulence, *J. of Comp. Phys*, **178:** 307-322, 2002.

[13] Maidi M., Yao Y., Direct numerical simulation of jet in cross-flow, UKTC workshop, New Forest, September, 2006.