11th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2007, Hammamet, Tunisia, 05-09 September, 2007.

CFD INVESTIGATION OF NORMAL JET IN A CROSSFLOW BY USING REYNOLDS STRESS TURBULENCE MODEL

Seyfettin Bayraktar and Tamer Yılmaz
Yıldız Technical University
Department of Naval Architecture and Marine Engineering
34349, Beşiktaş, İstanbul, TÜRKİYE

ABSTRACT
This paper aims to apply the CFD technique to show the evolution of the vortex structures seen in jet in a crossflow (JICF) by using Reynolds Stress Turbulence Model (RSTM). The flow is considered 3-Dimensional (3-D) and fully turbulent because of its nature. The Reynolds number of flow is 45347 and 90694 for two different velocity ratios. The numerical results are compared by an experimental data found in open literature and it is seen that the velocity components and turbulent kinetic energy variations are in good agreement.
Keywords: CFD, vortex structures, transverse jets, turbulence

1. INTRODUCTION
The study of JICF has many applications, such as vertical/short take off/landing aircraft exhaust and chimney flows, gas turbine engines, etc. Up to now, many researchers have experimentally and numerically studied a jet issued into a confined rectangular crossflow. The majority of past studies have been performed to understand the flow structures such as jet trajectory, velocity and temperature fields and dominant vortex structures. Beside experimental researches [1, 2, 3] some recent numerical papers [4, 5, 6] are reported on evolution and growth of counter-rotating vortex pair (CVP). When a jet issuing into a channel, it stroll around the jet as if it encounters with an obstacle and then it is deflected both in vertically and laterally. Close to the wall and upstream of the jet exit, a reverse flow is seen because of the blockage of the main flow and a vortex structure called horseshoe vortex is seen. Downstream of the jet, another vortex namely CVP is occurred. When the crossflow skirts around laterally the jet, it shears the jet fluid along its edges and then folds the face of the jet over itself to form the CVP [7]. The most important feature in JICF phenomena is jet-to-crossflow velocity ratio, R. In present study two different velocity ratios are investigated to show how CVP and other vortices are occurred.

2. MATHEMATICAL MODEL
In this work, a single round JICF is studied. The Reynolds (Re) number based on jet velocity and pipe diameter is 45347 and 90694 for two different jet velocity values of 6.95 m/s and 13 m/s. The channel velocity value is kept as 13 m/s. Based on these assumptions there are two different jet-to-velocity ratio, R=0.5 and R=1.0. Validation of the present study is done by making a comparison with experimental data of Ref. [8] as can be seen in Fig. 1. Comparisons are made for mean U, V and W-velocity components and turbulent kinetic energy at z/D=0 for R=0.5. It is seen that both present numerical and experimental data are in good agreement. Working domain which main dimensions is given in Fig. 2a are divided finite volumes as shown in Figure 2b. Totally 262000 mesh elements are generated. Mesh elements are graded from the vicinity of the jet to the far field. Grid sensitivity study is performed with three different meshes, namely; coarse, normal and fine meshes by checking U-velocity changes and it is seen that changes in velocity not greater than 1%. According to this result all the analyses are made with normal mesh.

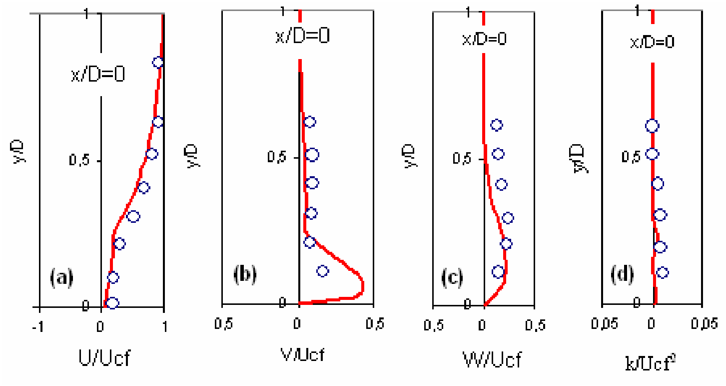


Figure 1.U-velocity (a), V-velocity (b) and W-velocity (c) comparisons of RSTM data with experimental results of Ref. [8] for z/D=0, R=0.5, Circle: exp., solid: CFD.

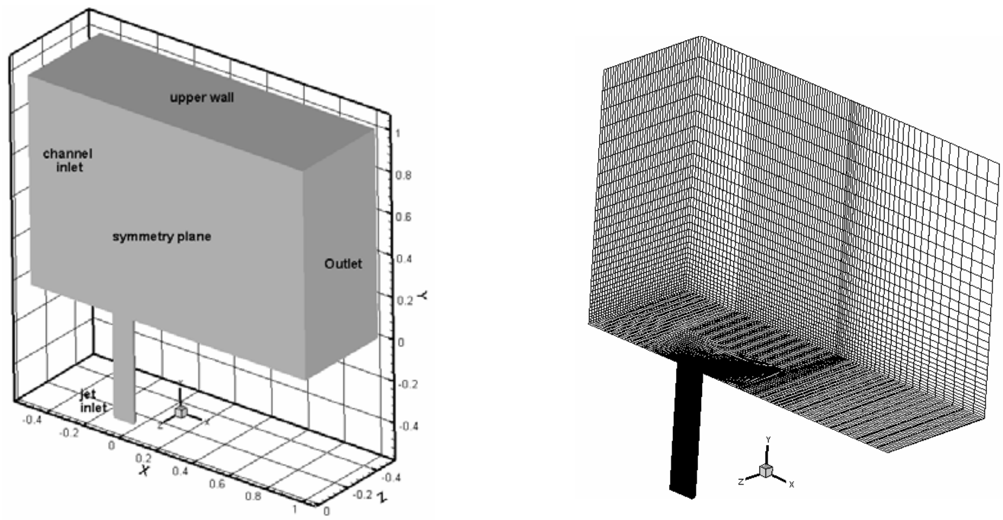
The channel and jet nozzle inlets are specified as velocity inlet while channel outlet is depicted as pressure outlet. All of the regions left are defined as wall type. At the inlets uniform velocity, temperature, turbulent kinetic energy and its dissipation values are speficied. 3-D steady-state Reynolds averaged Navier-Stokes equations and RSTM are used in the simulation. Detailed information about RSTM can be obtained from Ref. [9-10]. At the inlets, turbulence kinetic energy and its dissipation rate for jet nozzle and crossflow channel inlets were calculated by using Eq.1-Eq-4 [11];

$$k_j = 0.0020(V_j)^2 \tag{1}$$

$$k_{cf} = 0.00010(U_{cf})^2 \tag{2}$$

$$\varepsilon_j = \frac{k_j^{\,3/2}}{0.5\times D} \tag{3}$$

$$\varepsilon_{cf} = \frac{k_{cf}^{\,3/2}}{0.2\times H_{cf}} \tag{4}$$



(a) (b)

Figure 2.Working domain and coordinate system (a), grid structure (b).

3. RESULTS AND DISCUSSION

In this study, it is aimed to generate vortices seen in JICF phenomena numerically. Some of the obtained figures are seen in Fig.3-Fig.5. Longitudinal size and evolution of CVP for two different streamwise stations can be seen in Fig. 3. While jet velocity increased two times, CVP increases both in lateral and normal directions and occupy almost the whole channel at higher velocity ratio (R=1.0) especially towards the channel exit.

(a) (b)

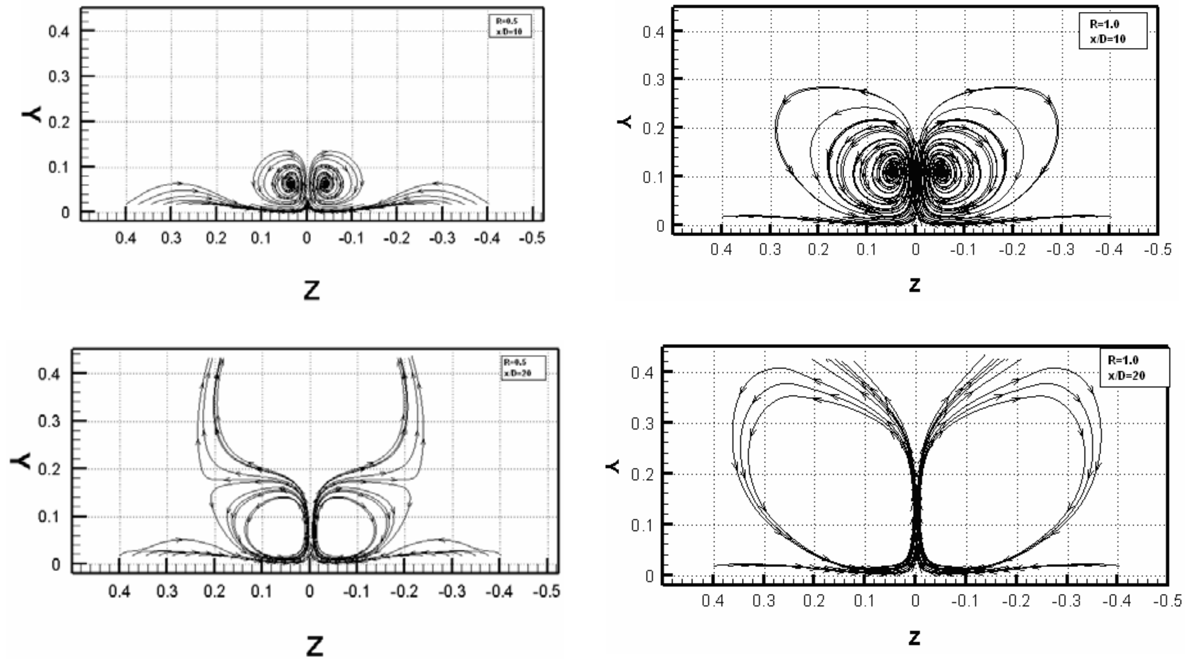


Figure 3. CVP evolution at different streamwise stations for a) R=0.5 and b) R=1.0.

a) (b)

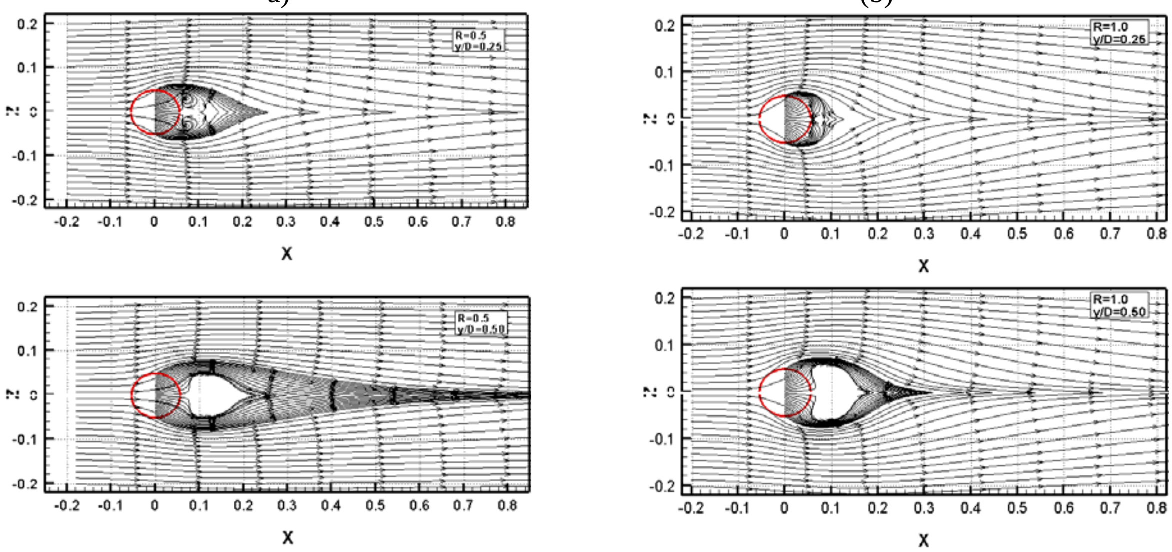


Figure 4. CVP evolution at different normal stations for a) R=0.5 and b) R=1.0.

As can be seen the striking feature of CVP is that it occurs near the jet pipe. The same result can be observed in normal stations as in Fig. 4. This figures revealed that CVP is more dominant at lower stations than that of higher ones for both two velocity ratios. Lateral sections of domain are shown in Fig. 5. In this figures it is seen that CVP getting weaker from the jet pipe centre to the side walls.

Beside the increment in the lateral and spanwise directions, the height of this vortices increases by increment in jet-to-crossflow velocity ratio. It is known that the wake vortices occur due to incoming boundary layer and origin in fluid at the back of the jet close to the wall. Wake vortices seen at downstream directions as can be seen in Fig. 5

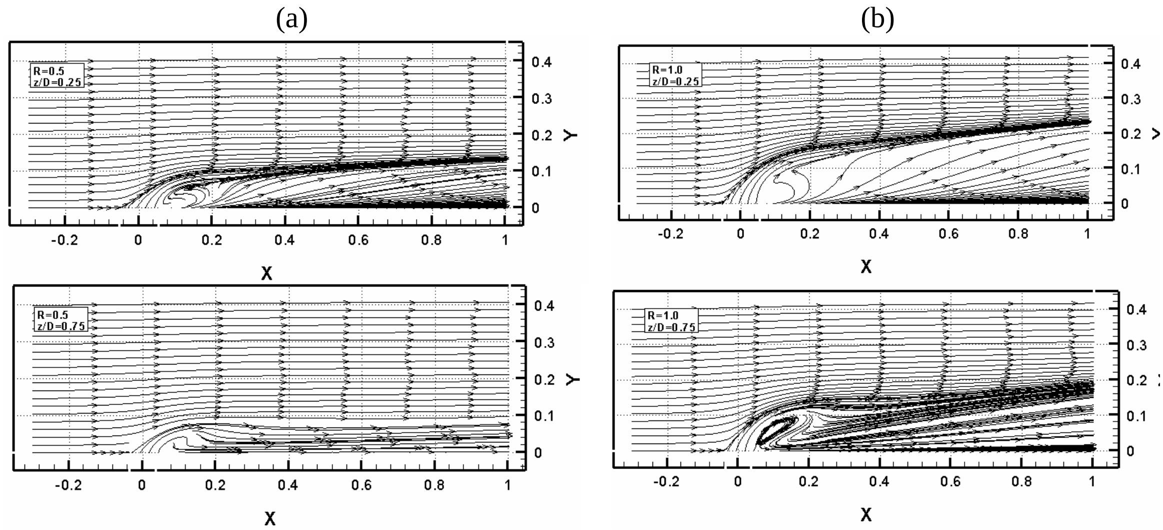


Figure 5. CVP evolution at different spanwise stations for a) R=0.5 and b) R=1.0

As a conclusion, a CFD investigation is performed to reveal one of the most dominant vortex structures, namely counter-rotating vortex pair. A validation of the study is done by making a comparison with experimental data. Results are obtained for two different jet-to-crossflow velocity ratio, R=0.5 and R=1.0 and it is seen that CVP is formed close to the jet exit and its strength is initially proportional to the jet-to-crossflow velocity ratios, R.

4. ACKNOWLEDMENT

Authors acknowledge the support provided by Türkiye Republic State Planning Organization (DPT) under the project number of 25-DPT-06-02-01.

5. REFERENCES

[1] Fric T. F., Roshko A.: Vortical Structure in the Wake of a Transverse Jet, Journal of Fluid Mechanics, 279, pp. 1-47, 1994.

[2] Perry E., Kelso R. M., Lim T.T.: Topological Structure of a jet in a Cross Flow, AGARD Meeting on "Computational and Experimental Assessement of Jet in Cross Flow, April, 1993.

[3] Cortelezzi L., Karagozian R., "On the Formation of the Counter-Rotating Vortex Pair in Transverse Jets", J. Fluid Mech., vol.446, pp.347-373, 2001.

[4] Majander P., Siikonen T., "LES of Round Jet in a Cross-flow", Int. J. of Heat and Fluid Flow 27, 402-415, 2006.

[5] Pathak M., Dewan A., Dass A.K., "DNS of Jets in Crossflow", Int. J. of CFD 20, 279-285, 2006.

[6] Sau A., Sheu T.W.H., Hwang R.R., Yang W.C., "Three-Dimensional Simulation of Square Jets in Cross-flow", Physical Review E 69, 2004.

[7] Blanchard J.N., Brunet Y., Merlen A., „"Influence of a Counter Rotating Vortex Pair on the Stability of a Jet in a Cross Flow", Experiments in Fluids 26, 63-74, 1999.

[8] Andreopoulos J., W. Rodi, "Exp. Inv. of JICF", J. of Fluid Mechanics, vol. 138, pp.361, 1984.

[9] Manabendra P., Anupam D., Anoop K.D., "Computational Prediction of a Slightly Heated Turbulent Rectangular Jet Discharged into a Narrow Channel Crossflow Using Two Different Turbulence Model", Int. J. of Heat and Mass Transfer 49, pp.3914-3928, 2006.

[10] FLUENT 6.2, Users Guide, vol. 2, 10.1-pp.10.82, 2005.

[11] Demuren, A.O., Rodi, W., "Three-Dimensional Numerical Calculations of Flow and Plumes Spreading Past Cooling Towers", Journal of Heat Transfer, 109, pp.113-119, 1987.