RANS AND LES MODELS COMPARISON FOR A CROSS-SHAPED JET FLOW WITH APPLICATION IN PERSONALIZED VENTILATION

Florin Bode[1], Ilinca Nastase[*2], Amina Meslem[3], and Cristiana Croitoru[4]

1 Technical University of Cluj-Napoca,
Mechanical Engineering Department,
B-dul Muncii nr.103-105, D03, Cluj-Napoca, Romania
CAMBI Research Center, Technical University of Civil Engineering in Bucharest, Building Services Faculty,
Avenue Pache Protopescu 66, Bucharest, Romania

2 CAMBI Research Center, Technical University of Civil Engineering in Bucharest,
Avenue Pache Protopescu 66, Bucharest, Romania
**Corresponding author: Ilinca.nastase@cambi.ro*

3 LaSIE, University of La Rochelle, Pôle Sciences et Technologie,
Avenue Michel Crépeau, 17042 La Rochelle, France

4 CAMBI Research Center, Technical University of Civil Engineering in Bucharest,
Avenue Pache Protopescu 66, Bucharest, Romania

ABSTRACT

The need for thermal comfort and clean air for occupants in buildings or vehicles is vital since we spend more than 90% of our time inside these enclosed environments. Worldwide, current directions of the leading powers are oriented towards the reduction of the energy consumptions and HVAC systems make no exception. Personalized Ventilation (PV) applied to buildings may represent a solution to this problem. The main idea of PV is to provide clean air close to the face of each occupant and to improve thermal comfort in his microenvironment.

This study is a part of a larger research work related to the optimization of PV air diffuser for HVAC systems in terms of passive control of jets flows in order to control jet development before it impinges on the occupant's face.

The flow field of an elementary lobed jet from a cross-shaped orifice with straight edges is investigated numerically using Large Eddy Simulation (LES) model and the results are compared with those from seven RANS (Reynolds Averaged Navier-Stokes) turbulence models and with experimental results. The RANS models used are the RNG k-ε turbulence model, k-ε standard, k-ε realizable, the Shear Stress Transport (SST) k-ω, the standard k-ω, the Spalart-Allmaras turbulence models and the Reynolds Stress turbulence Model (RSM). The initial Reynolds number based on the jet centreline exit streamwise velocity and on the equivalent diameter is around 4000. Numerical results are analysed based on PIV measurements performed in the same flow.

The objective is to assess the capability and limitations of the studied viscous models to predict the significant features of the cross-shaped air jet by numerical simulation.

The study revealed that none of the turbulence models was able to predict well all jet characteristics in the same time. For instance, the Reynolds Stress turbulence Model (RSM) predicted better the local jet flow expansion in the longitudinal minor and major planes, whereas global flow expansion and ambient air induction are better predicted by the Shear Stress Transport (SST) k-ω turbulence model. Furthermore the RANS models are not able to capture the flow's temporal dynamics.

Also, the LES model allows obtaining global mean quantities as RANS models but delivers supplementary information about the temporal vortex dynamics.

KEYWORDS

Cross shaped jet, CFD, RANS turbulence models, LES

INTRODUCTION

The need for thermal comfort and clean air for occupants in buildings or vehicles is vital since we spend more than 90% of our time inside these enclosed environments. Worldwide, current directions of the leading countries are oriented towards the reduction of the energy consumptions and HVAC systems make no exception. Personalized Ventilation (PV) for buildings and HVAC systems can be one of the solutions to this problem. The main advantage of PV in comparison to mixing ventilation is to provide clean air close to the face of each occupant and to improve thermal comfort in his microenvironment. In this way we are using energy only to change for each occupant his microenvironment parameters and not to cool the entire building.

A major challenge for personalized ventilation is to be able to control the jet dynamics without causing an increase in energy consumption and occupant discomfort. An innovative concept for optimized air diffusion in buildings using passive control of air jet through lobed diffusers was proposed [1, 2]. Despite the consequent gain in ambient air induction for the lobed perforated panel flow, the streamwise maximum velocities display comparable values in the far field, which signifies comparable throws for the two flows.

Mixing and air entrainment of a lobed jet can be increased or minimized as a function of a set of geometric parameters such as: number of lobes, their heigth and width, the shape of these lobes. Designing an optimal geometry for these lobes only by means of experimental methods the cost both in financial terms and in terms of time required will be huge. Computational Fluid Dynamics (CFD) based methods are an alternative to experimental methods. Large Eddy Simulation (LES) and Reynolds-Averaged Navier–Stokes (RANS) equation solvers are used nowadays in a wide range of engineering and indoor environment modeling. By these method we can extract detailed information on the 3D flow field (wich is not possible nowaday by experiment means), and by changing the boundary conditions and nozzle geometry we can obtain a large parametric study. However the choise of the numerical model to predict well our complex flow at low or moderate Reynolds numbers is still an open question.

The general conclusion of a previous study was that none of the evaluated turbulence models correctly predicted all flow characteristics (the standard k-ε model, the Shear Stress Transport (SST) k-ω model and the Reynolds Stress Model (RSM) were investigated). However, of the seven RANS turbulence models, the SST k-ω appeared capable of reproducing reasonably well flow expansion and ambient air induction when the flow was numerically resolved through a lobed diffuser.

The RNG k–ε turbulence model is generally preferred among all RANS models in indoor airflows modelling [3-6]. Comparative studies [4] showed that the RNG k-ε model is stable and has the best overall performance among all RANS models for enclosed environments.

The RSM is the most elaborate turbulence model among the RANS based models. This turbulence model should be considered whenever non-isotropic effects are important; for example, in flows with strong curvature, swirling flows, and flows with strong acceleration/retardation. Shamami and Birouk [7] showed that the Reynolds Stress Model (RSM) was able to produce much better predictions of the features of the jet in comparison with the eddy viscosity models (the standard k-ε, Renormalization Group (RNG) k-ε, the realizable k-ε, Shear Stress Transport (SST) k-ω were evaluated by the authors). In all RANS turbulence models evaluated, RSM appeared to be the only one capable of reproducing the experimental data reasonably well.

In this study the flow field of an elementary lobed jet from a cross-shaped orifice with straight edges is investigated numerically using Large Eddy Simulation (LES) model and the results are compared with those from seven RANS (Reynolds Averaged Navier-Stokes) turbulence models. Due to the very time step necessary for LES, the corresponding results presented here are only partial. The initial Reynolds number based on the jet centerline exit streamwise velocity and on the equivalent diameter is around 4000. Numerical results are analysed based on PIV measurements performed in the same flow.

JET NUMERICAL SIMULATION

Computational details

The jet air studied in this paper is generated using a cross-shaped orifice with straight edges (Figure 1). The equivalent diameter of the cross-shaped orifice is D_e=10mm.

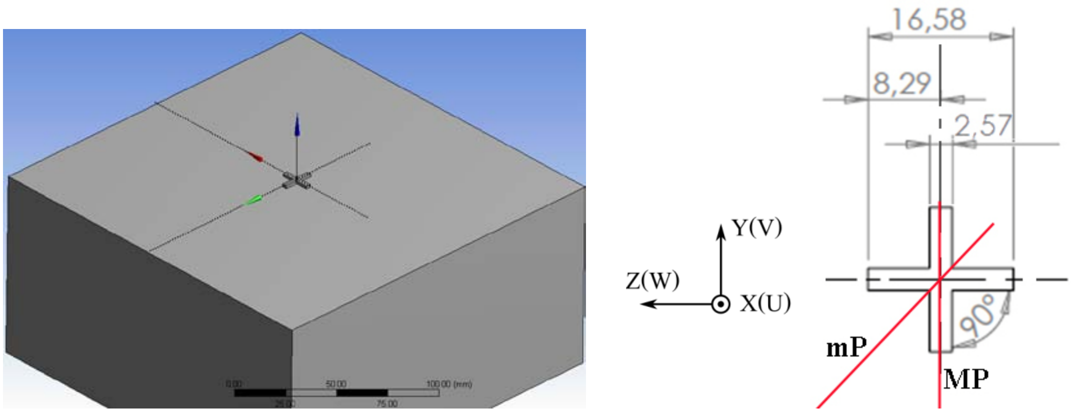

Figure 1. Investigated cross-shaped orifice

The computational domain (Figure 2) is composed of two parts separated by an orifice plate of 1.5 mm (0.15 D_e) thick. The upstream part and the downstream part of the domain has XYZ dimensions of 10×20×20 D_e and 29.85×20×20 D_e, respectively. Owing to the symmetry of the problem, just one fourth of the domain was modelled (so the dimension becomes 10 D_e in the Y and Z directions).

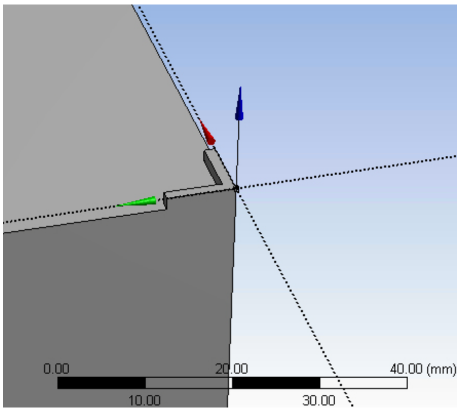

Figure 2. Geometry used for numerical simulation

The numerical analysis was performed using a finite volume based solver Fluent 13.0. Due to the low velocities, the incompressible solver was used. Fluent includes many viscous models, from the simplest laminar model to Large Eddy Simulation (LES), passing by RANS models and hybrid LES-RANS models. A direct full simulation (Direct Numerical Simulation - DNS) of the involved problem is still beyond our computational capabilities at the present time.

The inlet boundary conditions for the numerical simulation were given at the inlet plane of the upstream part of the domain. A uniform velocity normal to boundary of 0.00728 m/s, corresponding to a flow rate of 3.30×10^{-4} m^3/s and turbulence intensity (u' = 2 %) were fixed on this plane.
Given both the interest domain and the cross orifice jet symmetry, the symmetry boundary condition was used in the symmetry planes (*XY*) and (*XZ*) and the wall boundary condition for the orifice plate. There are other references in the literature with non conventional axi-symmetric jets (i.e. lobed jets) for which symmetry planes are used in order to gain computational resources [8-11]. The other boundary conditions applied to the planes which were far enough from the orifice (and are thus not affected by the flow) are atmospheric pressure boundary conditions.
The SIMPLE algorithm was used for pressure-velocity coupling. A second order upwind scheme was used to calculate the convective terms in the equations, integrated with the finite volume method.
Regarding the accuracy of results, the imposed convergence criterion was 10^{-6} for the variables residuals.
Computations were performed on a system Intel Xeon six core dual processor 2.66GHz with 96GBRAM.
In the LES case the numerical simulation was initialized with a time step, matching with a CFL number of 0.06. After solution stabilization the CFL number was raised to 0.2, which corresponds with a time step of 10^{-6} seconds.

Mesh dependency test

From our previous experience [9, 12, 13] we tried to achieve a final computational grid that would try to meet all necessary requirements for a good mesh, such as: the minimum number of cells needed in the critical section (30 cells), the smallest cell size (0.01mm), the largest cell size (2mm), $y+$ (less than 4 and its mean value was 1.3), the rate of cell growth (1.05), skweness of the cells. The resulted grid had a size of 4 million Cartesian non-uniform cells and all the simulation were performed using the same grid. This type of grid has been successfully used in our previous researches and managed to solve the flow field of different type of cross-shaped jets. A Cartesian grid is the best choice for flows with a strong velocity component in one direction such as jet flows.
For low Reynolds models the near-wall region is resolved down to the viscous sub-layer. A fine grid is nedeed in the near wall region which imposes at least one node in the viscous sublayer [14]. Values of y+ close to 1 are most desirable for the near-wall modeling. Wilcox has shown in [15] that the numerical discretization of such a function causes serious numerical errors in the viscous sub layer. He suggested enforcing the analytical solution in all points in the computational grid for which $y+ < 2.5$. The results also show that, in general, the error is the largest when thefirst cell center is located in the region $5 < y+ < 11$, despite the use of correct boundary conditions.
In our case, $y+$ was considered to be satisfactory for values less than 4, knowing that other literature recommends values of $y+$ less than 5 [16-18].
Results were compared with the experimental data of the turbulent cross-shaped jet [19] at moderate Reynolds number and the results of the comparison was satisfactory convincing us that we have a good quality mesh.
However, a mesh dependency study was conducted for five different grids (Figure 3): 0.7, 1.3, 3.3, 4 and 9 million hexaedral cells. In all cases, the initial flow rate was considered to be $Q_0=3.30\times10^{-4}$ m^3/s. From our previous experience [9, 12, 13], we used for the mesh dependency study SST k-ω turbulence model. The results confirmed that the initial grid was indeed a good choice.

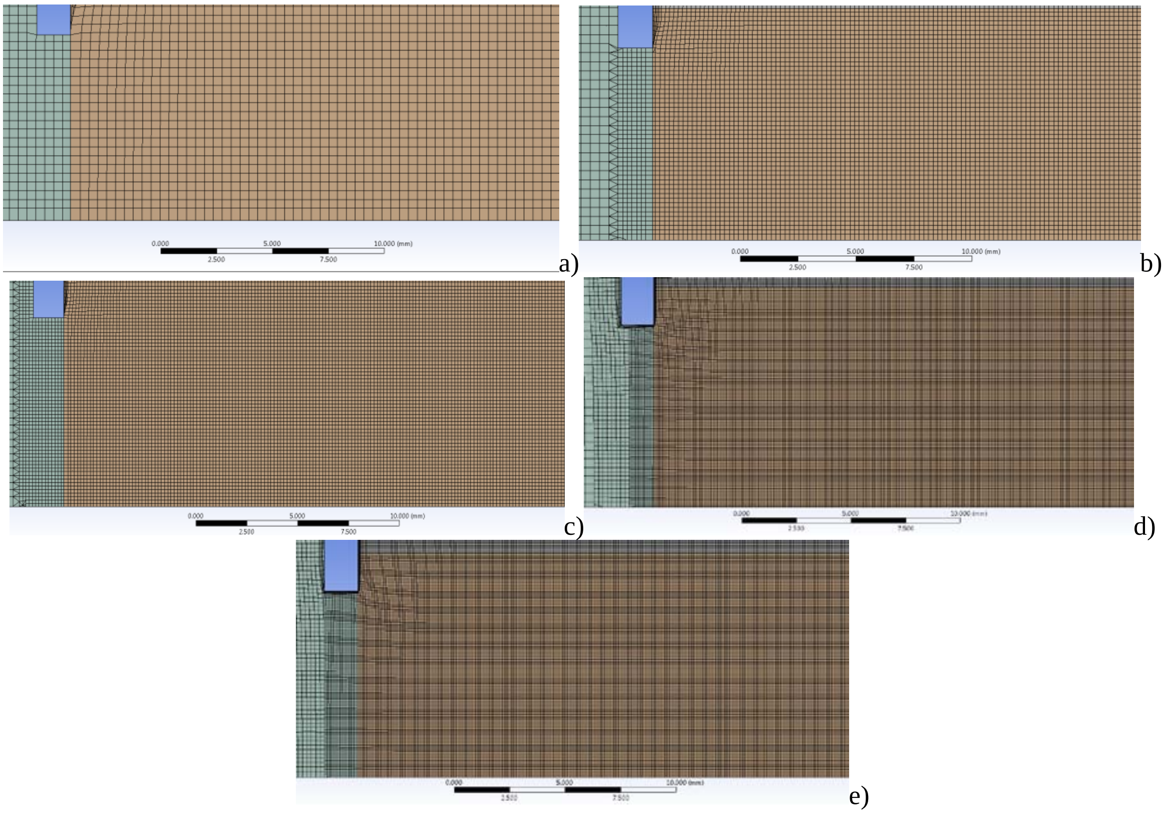

Figure 3. Details of several tested mesh grids: a) 0.7 millions of cells; b) 1.3 millions of cells; c) 3.3 millions of cells; d) 4 millions of cells; e) 9 millions of cells

Results obtained by the use of 4 and 9 million elements meshes are similar (Figure 4) and very close to the experimental results (Particle Image Velocimetry – PIV and Hot Wire Anemometry –HWA). The results obtained with grids of 0.7, 1.3 and 3.3 million elements are far away from experimental results, so grid chosen for numerical simulations was the 4 million elements.

Air entrainment is similar in all studied cases and verify experimental results obtained using PIV (Figure 5). Air entrainment is obtained by integrating streamwise velocity in the cross-planes with a minimum velocity of 0.15 m/s.

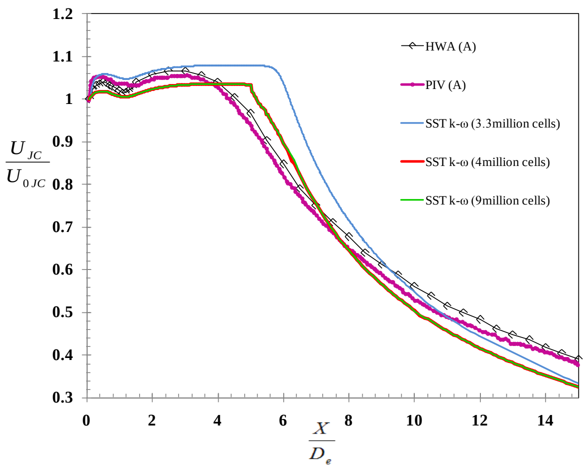

Figure 4. Normalized mean streamwise velocity on the jet centreline for 3.3, 4 and 9 million hexahedral cells compared with experimental results (PIV and HWA)

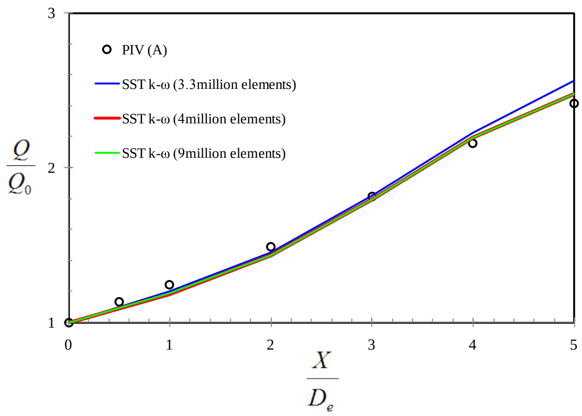

Figure 5. Air entrainment for 3.3, 4 and 9 million hexahedral cells cases compared with PIV experimental results

RESULTS AND DISCUSSION

The Reynolds number based on the Jet Centreline exit velocity (U_{0JC}) and on the equivalent diameter (D_e) is around 4000 for the experimental and numerical cases.

The velocity ratio U_{0JC}/U_{0mean} varies throughout the numerical cases, being closer to the experimental data in the case of SST k-ω and k–ε realizable turbulence models. On the other hand RSM model overestimates and RNG k–ε model underestimates the decay of the jet (Figure 6). k–ε standard, Spalart-Allmaras and k-ω standard are far away from experimental data and further we will not refer to these three turbulence models anymore. SST k-ω and k–ε realizable are the RANS models that predicts fairly well jet decay. Because of very small time step needed for LES model, the solution has not yet stabilized and results presented below for LES are only partial for now.

Ambient air induction (Figure 7) is obtained by integrating streamwise velocity in the cross-planes at 0, 1, 2, 3, 4, and 5D_e with a minimum velocity of 0.15 m/s. It is found from inspecting different velocity fields that this value is a good compromise to define the edge of the jet in the present study. As our particular application is directly interested to quantify the mixing between jets generated by HVAC terminal units and their ambience, we considered the 0.15 m/s criterion defining the extinction of the flow from the point of view of the thermal and draft comfort of the occupants [20].

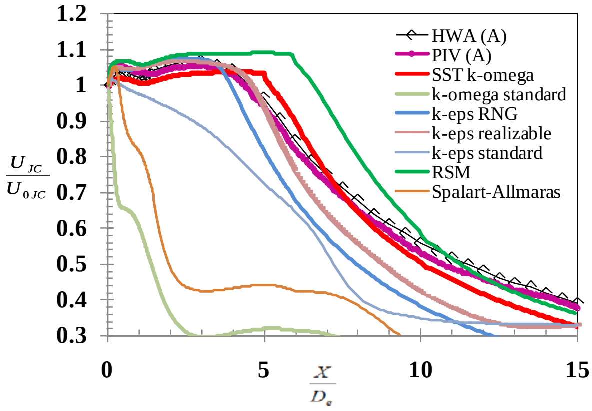

Figure 6. Normalized mean streamwise velocity on the jet centreline for studied numerical cases

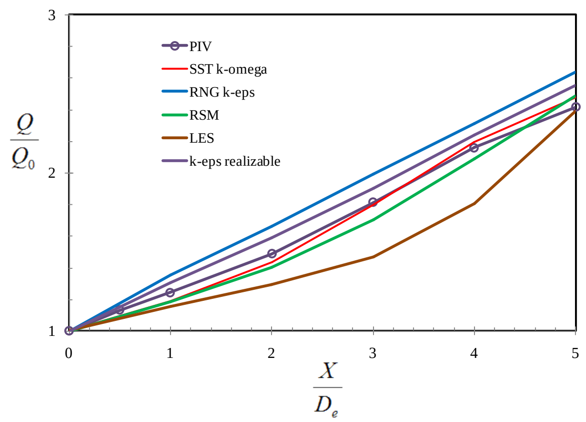

Figure 7. Streamwise evolution of the normalized volumetric flow rates for studied numerical cases

Ambient air induction is closer to the experimental data for SST k-ω and RSM turbulence models.

The difference in the predictions of the axial velocity profiles at 1 D_e in Major Plane (MP) and minor Plane (mP), as is depicted in Figure 1, is clearly visible in Figure 8.

k-ε turbulence models, predict unrealistic inflection points on MP, in the inner jet core region around Y = 0.3 D_e, whereas SST k-ω and RSM turbulence model predict realistic inflection points in the outer jet region, around Z = 0.7 D_e, in RSM case being more pronounced. The inflections are less pronounced than in the experimental situation for both numerical cases.

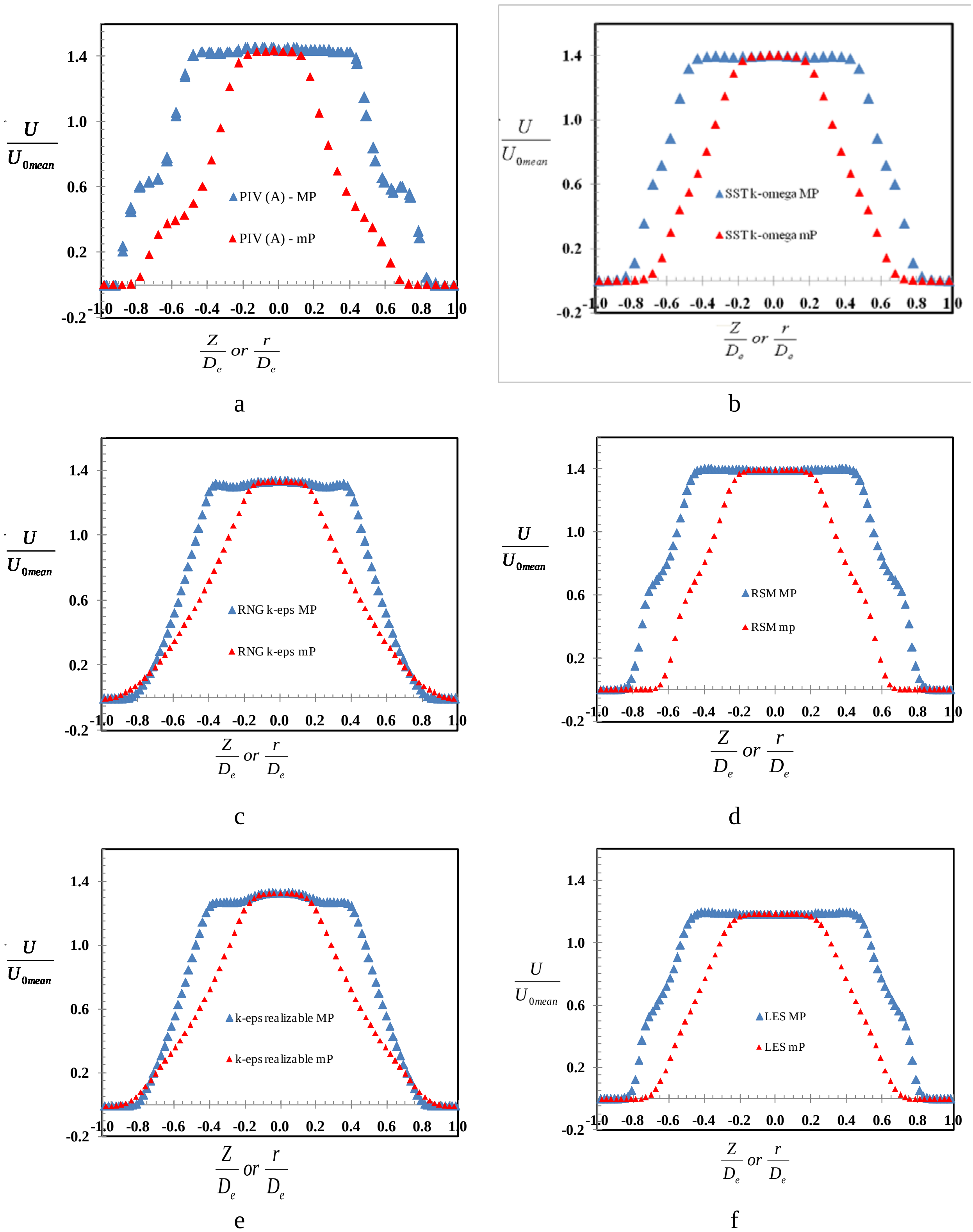

Figure 8. Streamwise velocity profiles in the MP and mP at X = 1 D_e for a. PIV, b. SST k-ω, c. RNG k–ε, d. RSM, e. k–ε realizable f. LES

The jets completly decays on Z direction at 0.7 D_e on minor Plane and at 0.8 D_e on Major Plane. Among all numerical cases SST k-ω and LES predict realistic decays of the jet both on Major Plane and minor Plane unlike RNG k–ε and k–ε realizable turbulence model.

Although there are only partial results for LES, can be seen in the Figure 9 the strong unsteady nature of the flow. Evolution of the LES results to date and comparison with experimental visualization results show close agreement between LES and experimental method.

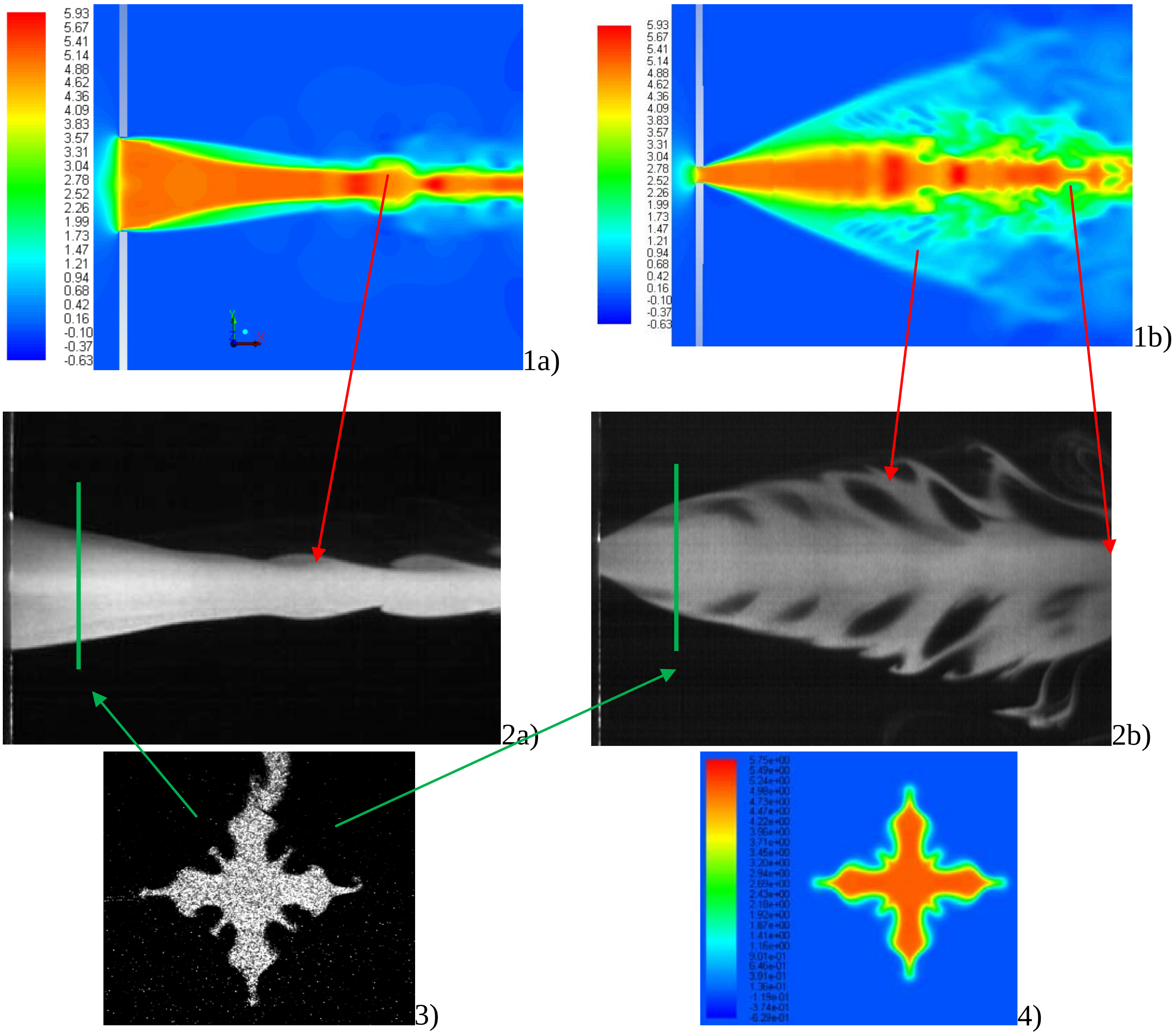

Figure 9. 1) LES isocontours of streamwise velocity, 2) High speed visualizations in the streamwise planes: a) Major Plane, b) minor Plane
3) High speed visualizations in the transverse plane at X=0.5D_e,
4) LES isocontours of streamwise velocity in the transverse plane at X=0.5D_e

CONCLUSION

In this study a three-dimensional flow at moderate Reynolds number through a cross-shaped orifice has been numerically investigated using seven turbulences models: RNG k-ε turbulence model, k-ε standard, k-ε realizable, the Shear Stress Transport (SST) k-ω, the standard k-ω, the Spalart-Allmaras turbulence models and the Reynolds Stress turbulence Model (RSM). The results are then compared with LES and experimental hot-wire anemometry and particle image velocimetry results.

The most accelerated flow at the jet inlet is given by the SST k-ω and RSM turbulence models and the value of the velocity ratio (maximum velocity on jet bulk-velocity) at the jet inlet is close to the experimental data.

From all models the k-ϵ models were not able to predict correct location of the inflection point in centreline streamwise velocity, shown by the experiments in the jet core region at 0.7D_e. Closest results to the experimental data were obtained by RSM and LES.

The study reveals that none of the turbulence models is able to predict well all jet characteristics. The Reynolds Stress turbulence Model (RSM) leads to better agreement between the numerical results and experimental data for the local jet flow expansion in the longitudinal minor and major planes, whereas global flow expansion and ambient air induction are better predicted by the Shear Stress Transport (SST) k-ω turbulence model.

Ambient air induction is closer to the experimental data for SST k-ω and RSM turbulence models.

As stated in [21] we confirm the superiority of the SST k-ω in comparison to all analyzed k–ε turbulence models, k-ω standard, Spalart-Allmaras and RSM, to correctly handle the transition of the flow through the diffuser.

The LES model still needs to be run. Because of very small time step needed for this model, the results presented in this paper are only partial. Evolution of the LES results to date and comparison with experimental visualizations shows very close agreement. Based on these enrouraging first results, LES is expected to well predict the nean flow of the lobed jet and its votex dynamics at moderate Reynolds number.

ACKNOWLEDGEMENTS

„This work was supported by the grants of the Romanian National Authority for Scientific Research, CNCS – UEFISCDI, project numbers: PN-II-PD-PCE-2011- 3-0099, PN-II-ID-PCE-2011-3-0835 and PN-II-PT-PCCA-2011-3.2-1212".

REFERENCES

1. Meslem, A., I. Nastase, and F. Allard, *Passive mixing control for innovative air diffusion terminal devices for buildings.* Building and Environment, 2010. **45**: p. 2679-2688.
2. Nastase, I., et al., *Lobed grilles for high mixing ventilation - An experimental analysis in a full scale model room.* Building and Environment, 2011. **46**(3): p. 547-555.
3. Chen, Q., et al., *Ventilation performance prediction for buildings: Model assessment.* Building and Environment, 2010. **45**: p. 295–303.
4. Zhang, Z., et al., *Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part 2- comparison with experimental data from literature.* HVAC&R Research, 2007. **13.**(6): p. 871-886.
5. Zhai, Z.J., W. Zhang, and Q. Chen, *Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part 1 - summary of prevalent turbulence models.* HVAC&R Research, 2007. **13**(6): p. 853 - 870.
6. Haidong Wang, Z.J.Z., *Analyzing grid independency and numerical viscosity of computational fluid dynamics for indoor environment applications.* Building and environment, 2012. **52**: p. 107-118.
7. Shamami, K.K. and M. Birouk, *Three-dimensional numerical simulation of an equilateral triangular turbulent free jet.* International Journal of Numerical Methods for Heat & Fluid Flow, 2009. **19**(7): p. 894 - 912.
8. McGuirk, J.J. and W. Rodi. *The calculation of three-dimensional turbulent free jets*. in *Symposium on Turbulent Shear Flows*. 1977. University Park, Pa ; United States.
9. Meslem, A., et al., *A comparison of three turbulence models for the prediction of parallel lobed jets in perforated panel optimization.* Building and Environment, 2011. **46**(11): p. 2203-2219.
10. Nathan, J.C., M. Parviz, and H. Hu. *Numerical Simulation of the Vortical Structures in a Lobed Jet Mixing Flow*. in *43rd AIAA Aerospace Sciences Meeting and Exhibit*. 2005. Reno, Nevada.

11. Yuan, Y., *Jet Fluid Mixing Control Through Manipulation of Inviscid Flow Structures*. 2000, Virginia Polytechnic Institute and State University: Ph.D. Thesis.
12. Bode, F., I. Nastase, and C. Croitoru, *Mesh dependency study using large eddy simulation of a very low reynolds cross-shaped jet.* Mathematical modelling 2011. **4**: p. 16-23.
13. Bode, F., I. Nastase, and A. Meslem, *Numerical investigation of very low Reynolds cross-shaped orifice jet.* Submitted to International Journal of Heat and Mass Transfer, 2012.
14. Georgi Kalitzin, G.M., Gianluca Iaccarino, Paul Durbin, *Near-wall behavior of RANS turbulence models and implications for wall functions.* Journal of Computational Physics, 2005. **204**: p. 265-291.
15. Wilcox, D.C., *Turbulence modelling for CFD.* La Canada- California: DCW Industries Inc. 536., 1993.
16. N. Martinho, A.L., M. Gameiro da Silva, *Evaluation of errors on the CFD computation of air flow and heat transfer around the human body.* Building and Environment, 2012. **58**: p. 58-69.
17. Tengfei (Tim) Zhang, L.T., Chao-Hsin Lin, Shugang Wang, *Insulation of commercial aircraft with an air stream barrier along fuselage* Building and Environment, 2012. **57**: p. 97-109.
18. Vieser W, E.T., Menter F., *Heat transfer predictions using advanced two-equation turbulence models.* ANSYS CFX Validation Report, 2004(1-62).
19. El-Hassan, M., A. Meslem, and K. Abed-Meraïm, *Experimental investigation of the flow in the near-field of a cross-shaped orifice jet.* Phys. Fluids, 2011. **23**(045101): p. 16 p.
20. Fanger, P.O., *Air turbulence and sensation of draught.* Energy and Buildings, 1988. **12**(1): p. 21-39.
21. El-Behery, S.M. and M.H. Hamed, *A comparative study of turbulence models performance for separating flow in a planar asymmetric diffuser.* Computers & Fluids, 2011. **44**: p. 248–257.