ISSN 1392 - 1207. MECHANIKA. 2011. 17(2): 168-171

Numerical study of a turbulent jet flow issued from lobed diffuser

M. Boulenouar*, A. Meslem, B. Imine***, I. Nastase******
**University of Science and Technology, U.S.TO, Oran, Algeria, E-mail: boulenouar_m@yahoo.fr*
***LEPTIAB, Université de La Rochelle, Pôle Sciences et Technologie, 17000 La Rochelle, France, E-mail: amina.meslem@univ-lr.fr*
****University of Science and Technology, U.S.TO, Oran, Algeria, E-mail: imine_b@yahoo.fr*
*****Technical University of Civil Engineering, Building Services Department, Bucharest, Romania, E-mail: ilinca.nastase@gmail.com*

crossref http://dx.doi.org/10.5755/j01.mech.17.2.333

1. Introduction

The physics of turbulent jet is of considerable interest from both fundamental and practical aspects. For fundamental aspect, it has been widely suggested that the mixing process is intimately connected with turbulence transient. Furthermore, the nozzle geometry and flow initial perturbations have profound influence on its generation and transition [1]. For practical aspect, the mixing process governs jet noise level of airplane and vehicles, spread of pollutant at industrial sites, mixing rate in combustion chambers and heating, ventilation and air-conditioning (HVAC) systems in buildings. The turbulent jet development is very sensitive the diffuser geometry [2-6], which is the main object of the present work. This technique is named passive control. It is considered one of the most promising methods for improving HVAC systems in terms of energy conceptions and thermal comfort of the occupants. This technique is simple and costless mechanical modification of the boundary geometry of classical existing diffusers. Lobed diffusers widely used in aeronautic or aerospace applications represent an attractive method for HVAC in buildings. However, the moderate exit conditions and low confinement flow specific to this field are different from the currently documented applications of passive control. As the mixing performance depends heavily on the lobed geometry parameters and exit flow conditions, designing an optimal air diffuser by experimental means alone is quite expensive due to the wide range of parameters involved. Computational fluid dynamics (CFD) methods represent a better alternative to experimental methods in the case of the optimization studies. Many different cases could be simulated at fraction of the time required for the experimental method. The commercial software available in today's market can quickly provide accurate aerodynamic predictions for a wide variety of geometries and flow conditions. This way, the predictability of the turbulence models to reproduce low Reynolds lobed jet vortex structures and its mixing performance in experimentally well known reference case, must be provided. Simulations of the turbulent jets employing a standard k-ε turbulence model have been performed by [7-9]. Their studies do not appear to be sufficient for capturing the mixing in these flows. Model predictions reveal that a very small degree of anisotropy is predicted throughout the flow field in comparison with experimental results. For six circular lobed nozzle with a Reynolds number $Re_0 = 54780$ (based on nozzle inner equivalent diameter), author in [10] found that numerical simulations using four widely employed turbulence models (k-ε, realizable k-ε, k-ω and shear stress transport (SST) k-ω) agree reasonably well with the PIV measurements in terms of streamwise vorticity and spanwise vorticity. However, the lobed jet mixing is not quantitatively analyzed and turbulent kinetic energy is over predicted with about 50 to 130%. Among the four models investigated by [10], the Realizable k-ε turbulence model provides the most accurate prediction for a lobed nozzle. It should be noted, that all previous studies were performed at relatively higher Reynolds number as compared to the operational values in HVAC application. Consequently, acceptable numerical prediction of turbulent lobed jet mixing performance at low Reynolds remains questionable.

In this study, the authors analyze the predictability of SST k-ω in the low Reynolds number inlet condition of lobed diffuser jet. Numerical simulation results are compared with PIV measurements of [1].

2. Computational details

The lobed diffuser is built up from a circular section tube of $D_e = 40$ mm and 76.5 mm length. This straight tube is connected to a shorter lobed geometry having the same characteristics as indicated in the study of [1]. The thickness of the wall at the exit plane is 2.5 mm. Lobed diffuser has the inner and outer lobe penetration angles (22°, 14°). Jet exit plane has six lobes with parallel sides and six troughs of sinusoidal shape (Fig. 1). The lobed diffuser and the jet development domain are modeled and meshed using the commercial software Gambit 2.3. The fluid medium is air. The incompressible jet mixing flow is injected with an initial volumetric flow rate $Q_0 = 4.7\ 10^{-3}$ m^3/s. At the inlet, the turbulence intensities are estimated to be 1.1%. Because of symmetry, only the sixth of the physical field is considered as computational domain (Fig. 2).

This domain extends 10 equivalent diameters downstream of the exit-plane and 4 equivalent diameters in

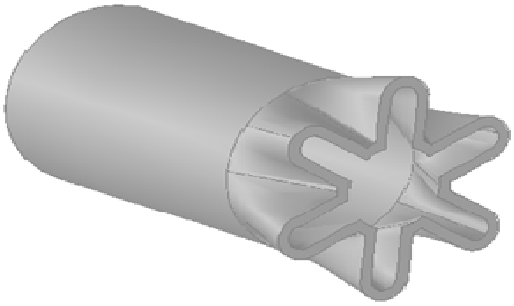

Fig. 1 Lobed diffuser

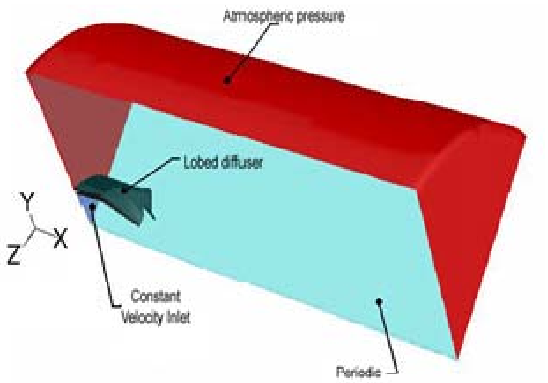

Fig. 2 Computational domain and boundary conditions

transverse plane. Constant static pressure is considered across the outer domain extends with rotationally periodic boundary conditions imposed on each side. The numerical analysis is performed using a finite volume based solver FLUENT 6.3. The SIMPLE algorithm is used for pressure-velocity coupling. The grid extends gradually in all directions in order to take into account of the jet development in the domain. The mesh was refined until no signification changes between cases with different mesh density were observed.

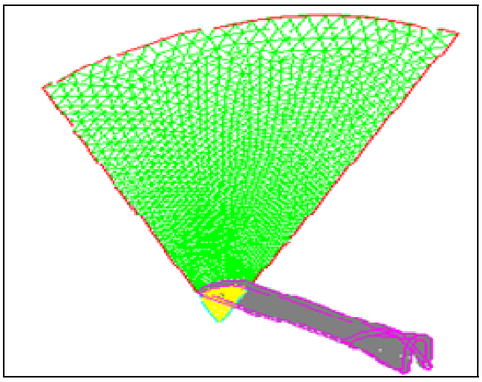

Fig. 3 Tetrahedral surface meshing on the lobed diffuser wall

The final mesh was 1.7 million of cells in the present study (Fig. 3). The values of $y+$ for the first gridline off the surfaces of the diffuser wall are less than 1. The convergence criterion required for the computed residuals is less than 10^{-6} for all equations.

3. Results and discussion

It should be noted that the entire simulated flow field is created by duplicating and periodically rotating the computational domain (periodic on 60° intervals) through a full 360°. The evolution of the centreline axial velocity normalized by the centreline velocity at jet exit U_0 is shown in Fig. 4. It is found that the turbulent model predicts 10% velocity increase at the jet exit, while the experimental results show only 2% velocity increase. The length of the potential core of the simulated flow appears to agree reasonably well with experimental data ($3D_e$ *to compare to* $2D_e$ in reality). After this region, a satisfactory agreement is obtained on the velocity levels since the difference between the simulation and the reference is less then 5%.

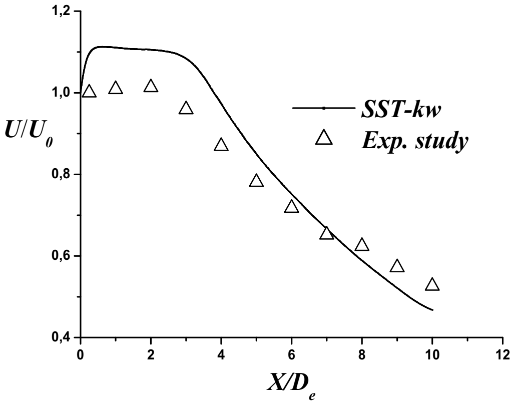

Fig. 4 Downstream evolution of longitudinal velocity

Fig. 5 shows the streamwise evolution of the normalized longitudinal volumetric flow rate, which was obtained by integrating streamwise velocity component U at each streamwise position X. Only the values larger than 0.2 m/s were taken into account.

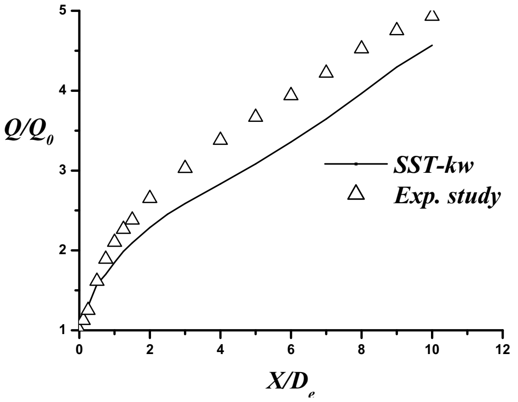

Fig. 5 Downstream evolution of volumetric flow rate

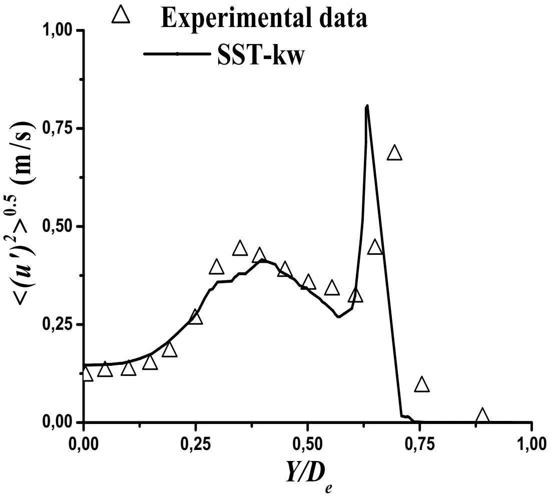

Fig. 6 Radial evolution of fluctuating longitudinal velocity at $X = 0.25D_e$

The numerical flow rate evolution law is close to the experimental one with a relative maximum deviation of 7% at $X = 10D_e$. From the position $X = 2D_e$, the curves are linear. On the exit radial distribution of the mean fluctuating streamwise velocity is shown in Fig. 6, the two peaks of the curve at $X \approx 0.4D_e$ and $X \approx 0.6D_e$ respectively, are well predicted by the model. These peaks are related to the areas of high velocity gradients due to connection between the jet core flow and the lobe flow and between the periphery of the jet flow and the zero speed in the ambient air respectively. Figs. 7 and 8 give at the streamwise position $X = 0.25D_e$, the normalized streamwise vorticity distribution of the present numerical simulation and of the experimental data respectively. The simulated transverse field agrees reasonably well with the experimental one. In fact, the six pairs of counter rotating large scale streamwise structures generated by the geometry of the lobed diffuser are predicted by the model. Each structure corresponds to the shear generated by two inverse transverse flows: outward flow due to the outer lobe penetration angle and inward flow due to the inner lobe through penetration angle. It is interesting to note that the maximal level of ϖ_x is equal the experiment maximal value.

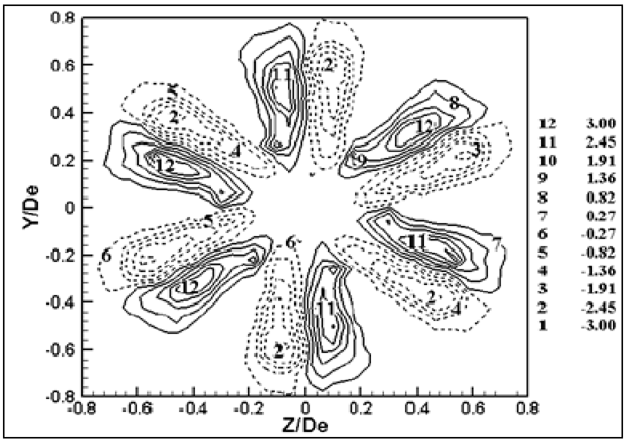

Fig. 7 Numerical distributions of the streamwise vorticity at $X = 0.25D_e$. $\varpi_x = \frac{D_e}{U_0}\left(\frac{\partial W}{\partial Y} - \frac{\partial V}{\partial Z}\right)$

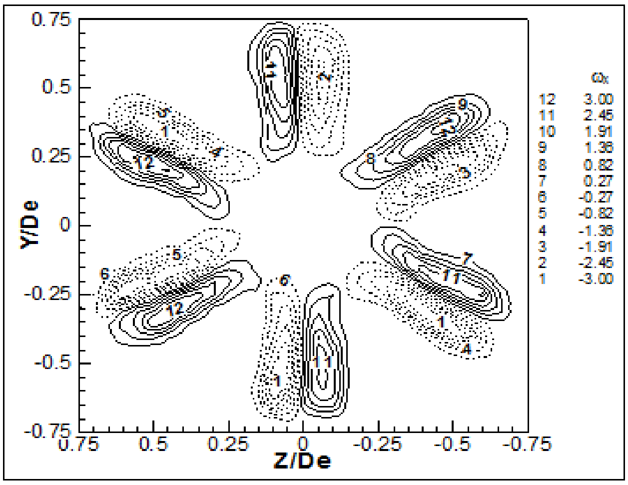

Fig. 8 Experimental distribution of the streamwise vorticity at $X = 0.25D_e$

4. Conclusion

Numerical simulation of a low Reynolds turbulent lobed jet using the SST *k-ω* model has been carried out. Results of centreline longitudinal velocity, volumetric flow rate, mean streamwise velocity fluctuating and streamwise vorticity agree reasonably well with the PIV measurements. The SST *k-ω* model can be considered for future works as an efficient tool for quickly optimizing lobed diffuser design and analyzing exit Reynolds number effect on the mixing performance of the lobed diffuser. The shown numerical method will allow many geometric parameter studies to improve mixing process for HVAC application.

References

1. **Nastase I.** 2007. Analyse des jets lobés en vue de leur intégration dans les unités Terminales de Diffusion d'air. PhD. thesis, Université de La Rochelle, France. 219p.
2. **Quinn, W.R.** 1989. On mixture in an elliptic turbulent free jet, Physics Fluids 1(10): 1716-1722.
3. **Zaman, K.B.M.Q.** 1996. Spreading characteristics and thrust of jets from asymmetric nozzles, AIAA paper No 96-200: 1-17.
4. **Mesnier I.** 2001. Etudes sur le développement de jets turbulents à masse volumique variable à géométries axisymétrique et asymétrique. Thèse de doctorat, Université d'Orléans. 327p.
5. **Hu, H.; Kobayashi, T.; Saga, T.; Taniguchi, N.** 2001. A study on a lobed jet mixing flow by using stereoscopic PIV, Physics of fluids 13(11): 3425-3441.
6. **Nastase, I.; Meslem, A.** 2010. Vortex dynamics and mass entrainment in turbulent lobed jets with and without lobe deflection angles, Experiments in Fluids 48(4): 693-714.
7. **McGuirk, J.J.; Rodi, W.** 1979. The calculation of three-dimensional turbulent free jets, First International Symposium: 874-882.
8. **Koutmos, P.; McGuirk, J.J.** 1989. Turbofan forced mixer nozzle temperature and flow modeling, Int. journal of Heat and Mass Transfer 32(6): 1141-1153.
9. **Salman, H.**; **Gary, J.P.; McGuirck, J.J.** 2003. Prediction of lobed mixer vortical structures with a *k-ε* turbulence model, AIAA Journal 41(5): 878-886.
10. **Nathan, J.C.; Parviz, M.; Hu, H.** 2005. Numerical simulation of the vortical structures in a lobed jet mixing flow, 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada: 1-13.
11. **Yahiaoui, T.; Adjlout, H.; Imine, O.** 2010. Experimental investigation of in-line tube bundles, Mechanika 5(85): 37-43.
12. **Khorsi, A.; Azzi, A.** 2010. Computation film cooling from three different holes geometries, Mechanika 6(86): 32-37.

M. Boulenouar, A. Meslem, B. Imine, I. Nastase

TURBULENTINIO ČIURKŠLĖS TEKĖJIMO IŠ MENTINIO DIFUZORIAUS SKAITINIS TYRIMAS

Reziumė

Atliktas turbulentinės laisvo tekėjimo čiurkšlės iš šešių menčių difuzoriaus skaitinis imitavimas, naudojant šlyties įtempių perdavimo k-ω modelį. Erdvinis sprendimas gautas naudojant tetraedrinių elementų tinklelį su 1.7 milijono elementų. Skaitinio prognozavimo rezultatai palyginti su žymėtųjų dalelių greičio matavimais. Šis palyginimas parodė, kad rezultatai gerai sutampa, ypač išilginio greičio čiurkšlės ašyje, tūrinio tekėjimo greičio, išilginio vidutinio fliuktuacijos greičio ir srovės vertekso. Gauti rezultatai rodo, kad šlyties įtempių perdavimo k-ω modelis gali būti naudojamas ateities tyrimams kaip efektyvi priemonė mentinių difuzorių projektavimui pagreitinti, optimizuoti ir analizuoti naudojant Reinoldso skaičiaus efektą maišymo proceso erdviniam tekėjimui pritaikyti kaitinimo, ventiliavimo ir oro kondicionavimo įrenginiuose.

M. Boulenouar, A. Meslem, B. Imine, I. Nastase

NUMERICAL STUDY OF A TURBULENT JET FLOW ISSUED FROM LOBED DIFFUSER

Summary

Numerical simulations of turbulent free jet issuing from a six-lobed diffuser, using the shear stress transport k-ω model have been carried out. The three-dimensional calculations have been done on a tetrahedral mesh with 1.7 million cells. The numerical predictions have been compared with PIV measurements. This comparison shows reasonable agreement, principally, in terms of centreline longitudinal velocity, volumetric flow rate, longitudinal mean fluctuating velocity and streamwise vorticity. With these results, the shear stress transport k-ω model can be considered for future studies as an efficient mean for quickly optimizing lobed diffuser design and analyzing exit Reynolds number effect on the mixing performance of this three-dimensional flow for HVAC application.

M. Боуленоуар, A. Меслем, Б. Имине, И. Настасе

ЧИСЛЕННОЕ ИССЛЕДОВАНИЕ ТЕЧЕНИЯ ТУРБУЛЕНТНОЙ СТРУИ ИЗ ЛОПАСТНОГО ДИФФУЗОРА

Резюме

Проведено численное имитирование турбулентной струи свободного течения шестилопостного диффузора используя k-ω модель передачи напряжения сдвига. Трехмерное решение, получено используя сетку с 1.7 миллионом тетраэдрических элементов. Результаты численного прогнозирования сопоставлены с результатами измерения скорости меченых частиц. Это сопоставление показало хорошое совпадение результатов, особенно: продольной скорости на оси струи, скорости объемного течения, продольной средней скорости флуктуаций и вертекса потока. Полученные результаты показывают, что k-ω модель передачи напряжения сдвига может быть полезна для будущих исследований, как эффективное средство для ускорения оптимизации и анализа лопастных диффузоров используя эффект числа Рейнольдса в процессе применения смешивания трехмерного течения в устройствах нагрева, вентиляции и кондиционирования воздуха.

Received November 12, 2010
Accepted April 15, 2011