ISSN 1392 - 1207. MECHANIKA. 2013 Volume 19(5): 512-517

Numerical simulation of a very low Reynolds cross-shaped jet

Florin Bode*, Amina Meslem, Cristiana Croitoru*****
**Technical University of Cluj-Napoca, Mechanical Engineering Department, 101-103 Muncii Bd., Cluj-Napoca, Romania Technical University of Civil Engineering in Bucharest, Building Services Department - CAMBI, 66 Avenue Pache Protopopescu, Bucharest, Romania, E-mail: florin.bode@termo.utcluj.ro*
**LEPTIAB, Université de La Rochelle, Pôle Sciences et Technologie, 17000 La Rochelle, France, E-mail: amina.meslem@univ-lr.fr
****Technical University of Civil Engineering in Bucharest, Building Services Department - CAMBI, 66 Avenue Pache Protopopescu, Bucharest, Romania, E-mail: cristianaverona@yahoo.com*

crossref http://dx.doi.org/10.5755/j01.mech.19.5.5537

1. Introduction

On March 2007, a series of environmental new regulations were adopted by the European Council. Regulations have stipulated that the European Union states must increase energy efficiency in order to reduce by 20% the EU energy consumption by 2020. Also they have to reduce emissions of greenhouse gases by 20% by 2020 [1]. All the 27 Heads of State and Government agreed to apply this objectives by 2020.

Considering these facts, the main task for the Heating Ventilation and Air Conditioning (HVAC) design engineers is to maintain at least the same thermal comfort for occupants in vehicles or buildings with less energy consumption. Inevitably, this will lead to a lower air flow introduced in the ventilated area.

Given these new conditions, innovative types of diffusers must be designed to accomplish two tasks: supply as more as possible mostly fresh air near the face of the user, and avoid draught and other thermal discomfort disagreements.

Since we spend more than 90% from our time indoors or in vehicles the need for clean air in terms of comfort is now a necessity.

Nowadays, total volume ventilation method (TV) is applied in buildings ventilation and the principle being mixing between the supplied air and polluted air present in the room. TV proves his inefficiency because clean air is supplied from diffusers placed far away from the users and it has time to mix with the polluted air from the room before it is inhaled by them. In the same time, TV is energy inefficient because his final goal is to generate an uniform environment in the entire ventilated room. On the other hand, a new development in the field of air distribution for confined spaces like buildings is personalized ventilation (PV) which is being used for decades in passenger transportation (automotive, trains, airplanes) [2-4]. The goal of personalized ventilation is to supply fresh and clean air close to the face of each occupant and to improve thermal comfort in his microenvironment.

One way to achieve this is through geometry passive control of the lobed diffusers in order increase air diffusion in vehicles and buildings [5-8]. Because of the special geometry of the lobed orifice, a lobed jet is able to cover a larger area around the human head compared to a circular jet for the same effective exit area and exit volumetric flow rate. This lead to the fact that PV is more protective for the indoor population from airborne transmission of infection [9-12] on one hand, and PV is saving energy because it may be used only when the occupant is present at the workplace [13, 14] on the other hand. Due to vortex dynamics intensification in the lobed jet, it is possible to control the maximum velocity of the jet, which will avoid draught discomfort.

The optimization of the lobed diffuser has to be correlated to all the previous noted parameters that are influencing the efficiency of a PV system.

The flow through a lobed cross-shaped orifice is very complex as many physical phenomena and time scales are involved.

Experimental investigations can be time consuming and since even a small perturbation can affect the development of the studied jet at very low Reynolds number. Given this, a numerical parametric study and a evaluation in a non-intrusive way of the flow behaviour is better suited for this case.

Computational Fluid Dynamics (CFD) doubled by experimental validations through modern investigation techniques like Laser Doppler Velocimetry (LDV) may provide an answer to this issue [15].

The purpose of this paper is to find the optimal viscous model to be used for the CFD study of an very low Reynolds air cross-shaped jet.

The increase in computational power from the last years enable us to conduct numerical simulation in order to capture the largest scales from the flow and to view the possible instabilities with Large Eddy Simulation (LES) viscous model. At this time, an approach to this problem by Direct Numerical Simulation (DNS), would be beyond our computing capability.

2. Computational details

In this paper a lobed cross-shaped orifice is used to generate a cross shaped air jet (Fig. 1). The results of experimental measurements have been made available by CAMBI laboratory (TUCEB) and Laboratoire d'Etudes des Phénomènes de Transfert et de l'Instantanéité : Agro-industrie et Bâtiment (LEPTIAB).

The studied cross-shaped orifice has an equivalent diameter of D_e = 10 mm.

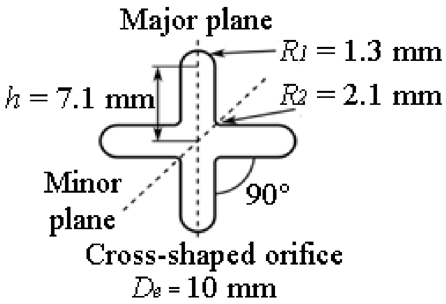


Fig. 1 Lobed cross-shaped orifice

The 3D geometrical model is realized in Ansys Design Modeler (Fig. 2).

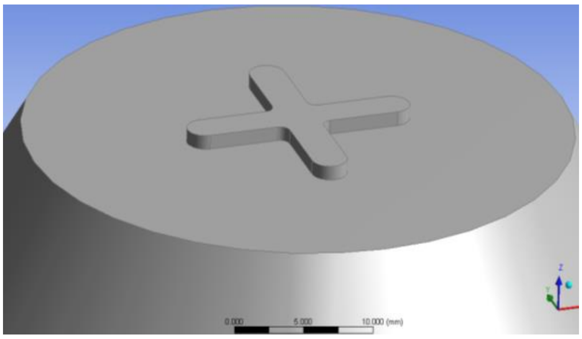


Fig. 2 Investigated cross-shaped orifice

The flow Reynolds number is 515, based on the mean velocity through the lobed cross-shaped orifice $U_0 mean$ (0.92 m/s) and on the equivalent diameter D_e.

Fluent software included in Ansys 13.0 package was used for the numerical simulation of the flow.

The mesh independence test for the flow field of the cross shaped jet was carried out in [16] on three unstructured grids of 0.4, 1.35, 2.2 million tetrahedral elements and the results were compared by Laser Doppler Velocimetry (LDV) measurement results [17].

All the grids used were highly refined in the orifice section. The conclusion was that the grid of 2.2 million tetrahedral elements is enough for the numerical simulation of the flow for the first 5 D_e (Fig. 3).

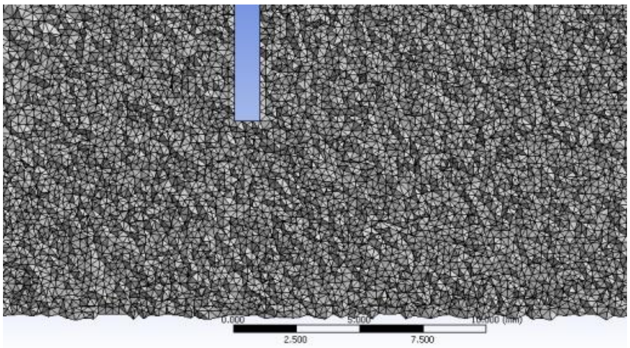


Fig. 3 Detail of the 2.2 million tetrahedral elements mesh in the orifice zone

A lobed cross shaped orifice plate having 1mm in thickness is separating the investigated domain. The upstream part of the computed domain has the dimensions $20D_e$ length and a diameter of $16D_e$ and the downstream part of the have the dimensions $40D_e$ length and a diameter of $20D_e$. Like in the experimental setup, the reduction of the turbulence level at the jet exit from the lobed cross shaped orifice is achieved by a convergent part placed at the end of the upstream part.

The flow downstream of the cross-shaped orifice is highly dependent of the upstream flow, so the inlet condition were specified at the inlet plane of the upstream part of the domain. A velocity magnitude, normal to boundary, of 0.0036 m/s was specified. Intensity and hydraulic diameter was the chosen specification method for the turbulence and a turbulence intensity of 2% was imposed on this boundary.

Only one eighth of the flow is modeled (Fig. 4) due to the domain symmetry. The other boundary conditions are represented in Fig. 4.

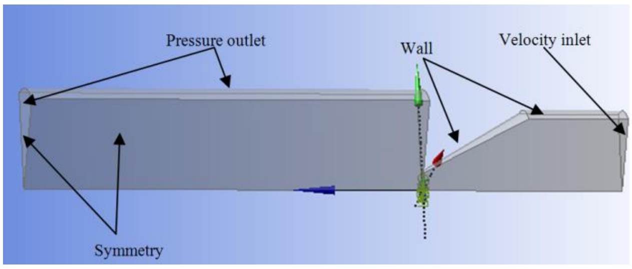


Fig. 4 Boundary conditions [16]

For the viscous model independence test we made numerical simulation with the following viscous models: the laminar model, LES model and two Reynolds Averaged Navier Stokes (RANS) models which gave good results on low Reynolds numbers: Reynolds Stress Model (RSM) - a second order closure model and RNG k-ε a linear eddy viscosity model.

The pressure-based solver was used for the numerical simulations as the flow is considered incompressible, given the very low velocities. For the LES case we used Smagorinsky-Lilly Subgrid-Scale model (SGS).

For the transient formulation we used a second order implicit scheme.

The SIMPLE algorithm is used for RANS numerical simulation for pressure-velocity coupling, [18] as for the LES, a second order, Bounded Central Differencing scheme is used for the convection discretization schemes for all transport equation [19].

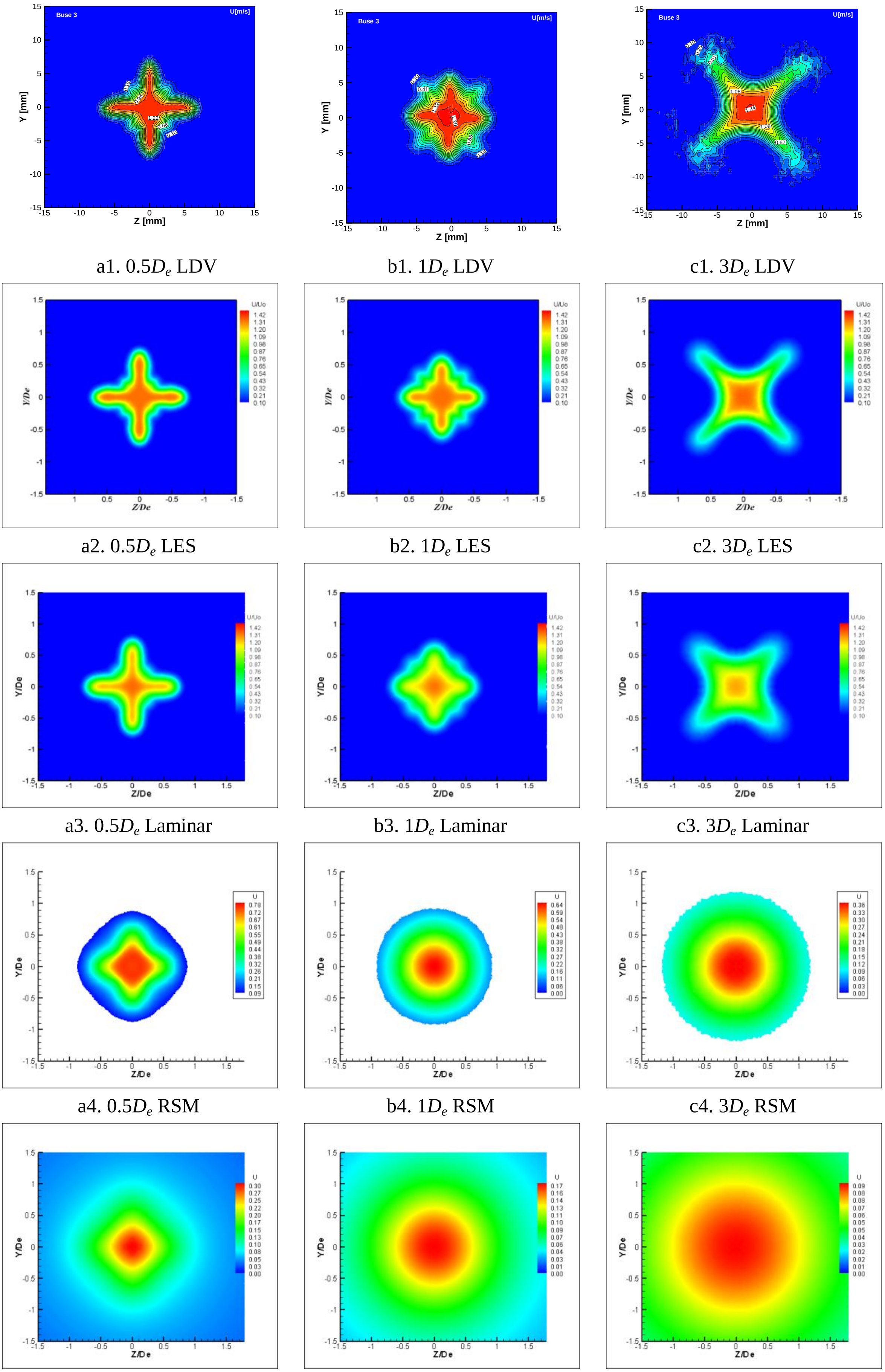
The imposed convergence criterion was 10^{-6} for all the variables residuals.

3. Results and discussion

In Fig. 5 are represented transverse velocity field at 0.5, 1 and 3 D_e for all the analyzed viscous models. We chose to represent the flow at this distances because we wanted capture an interesting phenomenon that occurs in the flow namely axis switching.

In this way we can analyze which one of the viscous models used in this numerical simulation can capture better this phenomenon.

The axis switching phenomenon occurs between the exit from the cross-shaped orifice ($0D_e$) and $3D_e$ farther downstream. Characteristic of this phenomenon is that the major axis of the jet becomes the secondary axis and secondary axis becomes major axis [20].



a1. $0.5D_e$ LDV b1. $1D_e$ LDV c1. $3D_e$ LDV

a2. $0.5D_e$ LES b2. $1D_e$ LES c2. $3D_e$ LES

a3. $0.5D_e$ Laminar b3. $1D_e$ Laminar c3. $3D_e$ Laminar

a4. $0.5D_e$ RSM b4. $1D_e$ RSM c4. $3D_e$ RSM

a5. $0.5D_e$ RNG k-ε b5. $1D_e$ RNG k-ε c5. $3D_e$ RNG k-ε

Fig. 5 Transverse velocity field at a) $0.5D_e$, b) $1D_e$ and c) $3D_e$ for 1. LDV measurements [21] 2. LES 3.Laminar 4.RSM 5. RNG k-ε

At $0.5D_e$ axis switching phenomenon has just begun, but this distance is imposed by LDV experimental measurements (Fig. 5, a1).

At $1D_e$ we can see the axis switching in the midst of change (Fig. 5, b1), and at $3D_e$ the phenomenon is clearly over (Fig. 5, c1).

The expected axis switching [22] is well reproduced by LES viscous model, but RSM and RNG k-ε failed to capture this phenomenon. In the numerical simulation using laminar model the axis switching is present but is slightly underestimated.

At $0.5D_e$ from the jet exit plane we can see that the contour plots corresponding to the LDV measurements, and respectively to the LES and laminar are similar but on the other hand RMS and RNG k-ε models have already differences.

As it could observed in Fig. 4 the LES model is reproducing better than the laminar model, the shape of the small instabilities formed in the orifice's troughs that will later develop into streamwise structures [17], while the other RANS models are far away from reproducing even the initial shape of the jet.

The laminar model displays also lower maximum velocity values than LES model, but RMS and RNG k-ε have lower velocities than the laminar model. At $0.5D_e$ RMS turbulence model is reproducing poorly the cross-shaped flow, but RNG k-ε is not able to capture even a feature of flow from all analyzed models. LES model is reproducing very well the LDV experimental velocity fields.

Farther, at $X = 1D_e$ (Fig. 5, b1 - b5), the values obtained by LES are clearly better reproducing the cross-shaped dynamics than the laminar ones. The laminar values are in good agreement with LDV ones, but RSM and RNG k-ε numerical simulation results failed to predict the flow.

At $3D_e$ both laminar and LES models successfully predicted the axis switching phenomenon but only LES model is able to accurately capture the real flow characteristics (Fig. 5, c1 - c5).

As from the aimed application, the streamwise velocity decay is very important, so in Fig. 6 we represented the comparison between the numerical models and the experimental LDV data.

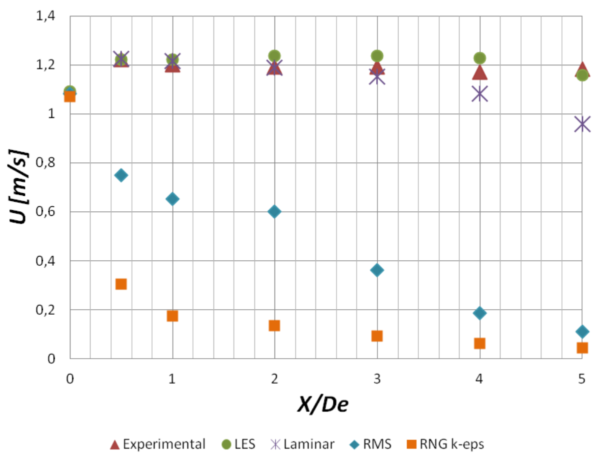


Fig. 6 Axial velocity in the axis of the jet

As it could be observed in Fig. 6 the RNG k–ε model and RSM underestimate the jet core length so at $1D_e$ RNG k–ε axial velocity is 0.2 m/s and RSM axial velocity is 0.2 m/s at $4D_e$.

The LDV experimental measurements at $4D_e$ have 1.2 m/s, almost the same as LES and laminar model. The dissipation of the flow for the laminar model is greater in comparison to the measurements results beginning at $3D_e$. On the other hand LES model tends to follow the LDV experimental measurements at least until $5D_e$.

Another important feature for our application is the ability of the air jet to mix with the ambient air. We also wanted to check the ability of the studied viscous models to reproduce the volumetric flow rate evolution of the studied jet.

Q the volumetric flow rate was calculated by:

$$Q = 2\pi \int rU\, dr\,.$$

To determine the jet boundaries, we used a fixed criterion of axial velocity, so there was no need to integrate all the measurement points at an axial location. Because in our particular application the direct interest is to quantify the air mixing between the jet generated by HVAC diffusers and the air from the room, we considered a value of 0.1 m/s for the axial velocity as a criterion, for both experimental and numerical case, which is defining in our case the boundary of the flow from the point of view of the draft and thermal comfort of the inhabitants.

In Fig. 7 is given the streamwise evolution of the entrainment rate of the jet. In the same way, for the evaluation of the volumetric flow rate we considered the axial velocity of 0.1 m/s as a criterion for defining the extinction of the flow from the point of view of the thermal and draft comfort of the inhabitants [23].

The RNG k–ε turbulence models overestimate jet expansion. Also, the RSM viscous model overestimate, as well as RNG k–ε turbulence model the entrainment rate.

In both the last figures is clearly visible that LES provides the best results.

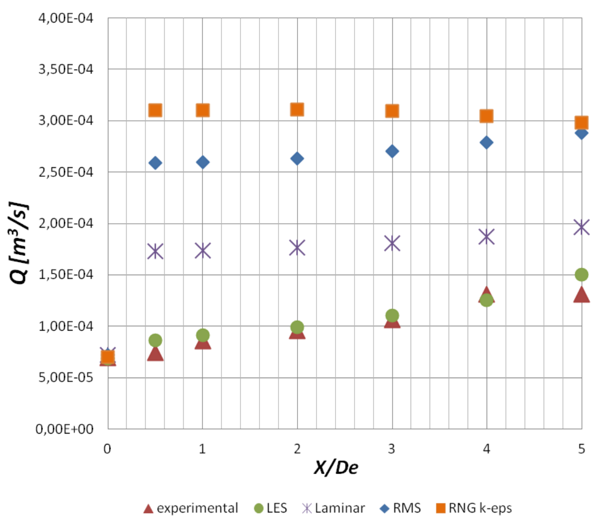


Fig. 7 Streamwise evolution of the entrainment rate

Due to the vena contracta effect of the jet velocity (this phenomenon is commonly encountered in orifice jets [24]), the maximum value of centreline axial velocity near the jet exit from the cross-shaped orifice is about 20% higher than bulk-mean exit velocity. Numerical simulation carried out in this study shows that vena contracta phenomenon is well predicted by the LES and laminar models unlike the other studied models.

4. Conclusions

In this article, the flow field of a very low Reynolds cross-shaped jet was numerically investigated using the RSM, RNG $k–\varepsilon$, laminar, and the Large Eddy Simulation viscous models. The corresponding results were compared with LDV measurements for a reference jet configuration which was numerically simulated. The objective of this study was to assess the capability and limitations of the CFD numerical simulations in predicting the significant features of the cross-shaped jet flow when the flow is numerically resolved through the lobed diffuser.

The RSM model is found to be better than that the RNG $k–\varepsilon$ viscous model, even so, the performance of these two viscous models was very poor for the analyzed case.

Jet behaviour predicted by the laminar model is very good up to $3D_e$ and by the LES model is up to $X = 5D_e$. The laminar model was found to overestimate the jet entrainment rate.

The evolution of the axial velocity and the volumetric flow rate predicted by the LES viscous model are reasonably well reproduced from the perspective of the aimed application.

Numerical simulation carried out in this study shows that vena contracta phenomenon is well predicted by the LES and laminar models unlike RSM and RNG k-ε turbulence models.

Only LES and laminar viscous models were able to capture the axis switching phenomenon, but in the laminar case is slightly underestimated.

As the good prediction of the cross section's shape of the jet and of its expansion is important for the aimed application up to about $X = 15D_e$, a supplementary refining of the mesh downstream to $X = 5D_e$ is necessary. This will be possible in the very next future through the reinforcement of our computational means.

Acknowledgments:

This work was supported by the grant of the Romanian National Authority for Scientific Research, CNCS – UEFISCDI, project number: PN-II-PD-PCE-2011-3-0099.

References

1. **Brussels European Council**, 8/9 March 2007 Presidency conclusions.
2. **Melikov, A.K**. 2004. Personalized ventilation, Indoor Air Pollution 14(7): 157-167. http://dx.doi.org/10.1111/j.1600-0668.2004.00284.x.
3. **Gao, N.; Niu**, **J.** 2004. CFD study on micro-environment around human body and personalized ventilation, Building and Environment 39: 795-805. http://dx.doi.org/10.1016/j.buildenv.2004.01.026.
4. **Sun, W.; et al**. 2007. Thermal performance of a personalized ventilation air terminal device at two different turbulence intensities, Building and Environment, 42: 3974-3983. http://dx.doi.org/10.1016/j.buildenv.2006.04.028.
5. **Meslem, A.; Nastase, I.; Allard, F.** 2010. Passive mixing control for innovative air diffusion terminal devices for buildings, Building and Environment 45(12): 2679-2688. http://dx.doi.org/10.1016/j.buildenv.2010.05.028.
6. **Meslem, A.; El-Hassan, M.; Nastase, I.** 2010. Analysis of jet entrainment mechanism in the transitional regime by time-resolved PIV, Journal of Visualization, online first: p. 1-12. http://dx.doi.org/10.1007/s12650-010-0057-7.
7. **Nastase, I.; et al**. 2011. Lobed grilles for high mixing ventilation An experimental analysis in a full scale model room, Building and Environment 46(3): 547-555. http://dx.doi.org/10.1016/j.buildenv.2010.08.008.
8. **Kristiawan, M.; et al.** 2012. Wall shear rates and mass transfer in impinging jets: Comparison of circular convergent and cross-shaped orifice nozzles, International Journal of Heat and Mass Transfer 55(1-3): 282-293. dx.doi.org/10.1016/j.ijheatmasstransfer.2011.09.014.
9. H**e, Q.; et al.** 2011. CFD study of exhaled droplet transmission between occupants under different ventilation strategies in a typical office room, Building and Environment 46(2): 397-408. http://dx.doi.org/10.1016/j.buildenv.2010.08.003.
10. **Khalifa, H.E.; Janos, M.I.; Dannenhoffer**, **J.F.** 2009. Experimental investigation of reduced-mixing personal ventilation jets, Building and Environment 44(8): 1551-1558. http://dx.doi.org/10.1016/j.buildenv.2008.11.006.
11. **Melikov, A.; Ivanova, T.; Stefanova**, **G.** 2012. Seat headrest-incorporated personalized ventilation: Thermal comfort and inhaled air quality, Building and Environment 47(0): 100-108. http://dx.doi.org/10.1016/j.buildenv.2011.07.013.
12. **Melikov, A.K.; Cermak, R.; Majer**, **M.** 2002. Personalized ventilation: evaluation of different air terminal devices, Energy and Buildings 34(8): 829-836. http://dx.doi.org/10.1016/S0378-7788(02)00102-0.
13. **Schiavon, S.; Melikov**, **A.K.** 2009. Energy-saving strategies with personalized ventilation in cold climates, Energy and Buildings 41(5): 543-550. http://dx.doi.org/10.1016/j.enbuild.2008.11.018.
14. **Schiavon, S.; Melikov, A.K.; Sekhar, C.** 2010. Energy analysis of the personalized ventilation system in hot and humid climates, Energy and Buildings 42(5): 699-707. http://dx.doi.org/10.1016/j.enbuild.2009.11.009.
15. **Meslem, A.; et al.** 2011. A comparison of three turbulence models for the prediction of parallel lobed jets in perforated panel optimization, Building and Environment 46(11): 2203-2219. http://dx.doi.org/10.1016/j.buildenv.2011.04.037.
16. **Bode, F., Nastase, I.; Croitoru, C.** 2011. Mesh dependence study using large eddy simulation of a very low Reynolds cross-shaped jet, Mathematical

Modelling in Civil Engineering – Scientific Journal, 7(4): p. 7.
17. **Nastase, I.; Meslem, A.; Gervais, P.** 2008. Primary and secondary vortical structures contribution in the entrainment of low Reynolds number jet flows, Experiments in Fluids 44(6): 1027-1033. http://dx.doi.org/10.1007/s00348-008-0488-2.
18. **Bode, F.; Benea, R.; Hodor**, **V.** 2010. Investigation on thermo-acoustical instabilities related to a confined swirling burner. Mechanika 1(81): 37-42.
19. **FLUENT**, User's Guide, Software and Documentation, in Fluent, Inc, V. 6.1.18, Editor. 2003: Centerra Park Lebanon, NH.
20. **Meslem, A.; Dia, A.; Beghein, C.; Ammar, A.; Nastase, I.; El Hassan, M.** 2012. Numerical simulation of free cross-shaped jet, Mechanika 18(4): 403-408. http://dx.doi.org/10.5755/j01.mech.18.4.2328.
21. **Nastase, I**. 2007. Analyse des jets lobés en vue de leur intégration dans les Unités Terminales de Diffusion d'air. Université de La Rochelle: Ph.D. Thesis.
22. **Nastase, I.; Meslem, A.** 2008. Vortex dynamics and entrainment mechanisms in low Reynolds orifice jets, Journal of Visualization 11(4): 309-318. http://dx.doi.org/10.1007/BF03182199.
23. **Fanger, P.O.; Christensen, N.K** 1986. Perception of draught in ventilated spaces, Ergonomics 29(2): 215-235. http://dx.doi.org/10.1080/00140138608968261.
24. **Mi, J.; Nathan, G.J.; Nobes, D.S.** 2001. Mixing characteristics of axisymmetric free jets from a contoured nozzle an orifice plate and a pipe, Journal of Fluids Engineering 123(4): 878-883. http://dx.doi.org/10.1115/1.1412460.

F. Bode, C. Croitoru

LABAI ŽEMO REINOLDSO KRYŽIAUS FORMOS SRAUTO SKAITINĖ ANALIZĖ

R e z i u m ė

Atlikti laisvo kryžiaus formos srauto, naudojamo buitiniuose ventiliatoriuose (BV), skaičiuojamosios skysčių dinamikos (SSD) tyrimai, siekiant nustatyti, kuris klampio modelis geriausiai tinka SSD srautui tiksliai modeliuoti. Didelės indukcijos difuzorių naudojimo pasyviam srautui valdyti koncepcija padarė perversmą šildymo, vėdinimo ir oro kondicionavimo (ŠVOK) procesuose. Šio skaitmeninio modeliavimo tyrimo tikslas – nustatyti optimalią kryžiaus formos čiurkšlės konfigūraciją. Todėl buvo lyginami lazerinio Doplerio greičio matuoklio (*Laser Doppler Velocimetry* – LDV) eksperimentiniai ir skaitmeninio modeliavimo, naudojantis Fluent programa, rezultatai su rezultatais, gautais naudojant keturis skirtingus klampio modelius: laminarinį, RNG k-ε, RSM (*Reynold's Stress Model* – Reinoldso įtempių modelį) ir LES (*Large Eddy Simulation* – plataus sūkurio modelį). Tuo buvo siekiama įvertinti ištirtų klampio modelių galimybes ir apribojimus ir skaitmeniniu modeliavimu nustatyti svarbias kryžiaus formos oro srauto savybes. Tyrimai parodė, kad LES yra vienintelis modelis, fiksuojantis tekėjimo charakteristikas.

F. Bode, C. Croitoru

NUMERICAL SIMULATION OF A VERY LOW REYNOLDS CROSS-SHAPED JET

S u m m a r y

A Computational Fluid Dynamics (CFD) investigation of a free cross-shaped jet used in personalized ventilation (PV) was conducted in order to determine the best viscous model to be used for an accurate CFD numerical simulation of the flow. The conception of high induction diffusers by means of passive jet control became a challenge in Heating, Ventilation and Air Conditioning (HVAC). The aim of this study is to identify the optimal configuration for the numerical simulation study of anair cross-shaped jet. For this we compared the Laser Doppler Velocimetry (LDV) experimental measurements on a cross-shaped jet with the results obtained by numerical simulation in Fluent software with four different viscous models: laminar, RNG k-ε, RSM (Reynold's Stress Model) and LES. 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 reveals that LES is the only model that is able to capture the flow characteristics.

Keywords: lobed cross shaped orifice**,** axis switching, CFD, laminar flow, personalized ventilation.

Received July 05, 2012
Accepted October 10, 2013