PROCEEDINGS OF

ACADEMICS WORLD
119th INTERNATIONAL CONFERENCE

DUBLIN, IRELAND

Organized by

Date of Event:

1st-2nd February, 2019

Event Co-Sponsored by

Corporate Address
IRAJ Research Forum
Institute of Research and Journals
Plot No- 30, Dharma Vihar, Khandagiri, Bhubaneswar, Odisha, India
Mail: info@iraj.in, www.iraj.in

TABLE OF CONTENTS

Sl No.	TITLES AND AUTHORS	Page No.
01.	**Numerical Simulation Of Pollution Dispersion From Thermal Power Plantsin The Atmosphere** ➢ *Aiymzhan Baitureyeva, Alibek Issakhov, Tong Yang*	**1-5**
02.	**Green Synthesis Of Silver NPS As A Highly Sensitive Biosensor To Detect The Mercury Inside The Soil Sample Of SORAN Area** ➢ *Karmand S. Mohammed, S. Mohammad Sajadi*	**6**
03.	**Determination Of Traffic Safety With Methods Alternative To Traditional Methods** ➢ *Coruhemine, Tortum Ahmet*	**7-12**
04.	**Concrete Mixture With Plastic As Fine Aggregate Replacement** ➢ *Chien-Chung Chen, Nathan Jaffe, Matt Koppitz, Wesley Weimer, Albert Polocoser*	**13-17**
05.	**Influence Of The Flc's Parameters Of The Upqc In The Distributed Generation** ➢ *C. Benachaiba, B. Mazari, M. Habab, C. Benoudjafer, N. M. Tandjaoui*	**18-23**
06.	**An Analysis Of Mobile Banking Customers For A Bank Strategy And Policy Planning** ➢ *Behrooz Noori*	**24-30**
07.	**Advantage Of Make-To-Stock Strategy Based On Linear Mixed-Effect Model** ➢ *Yu-Pin Liao, Shin-Kuan Chiu*	**31-42**
08.	**Faculty Researchers And Non-Researchers In The Context Of Teaching Performance And Personal Profile** ➢ *Jake M. Laguador, Joseph Cezar L. Deligero, Cecilia C. Pring*	**43-48**
09.	**The Headway of Ecopreneurship as a Paramount Research Field** ➢ *Deepa Guleria, Gurvinder Kaur*	**49-56**

★★★

NUMERICAL SIMULATION OF POLLUTION DISPERSION FROM THERMAL POWER PLANTSIN THE ATMOSPHERE

[1]AIYMZHAN BAITUREYEVA, [2]ALIBEK ISSAKHOV, [3]TONG YANG

[1,2]Department of Mathematical and Computer Modelling, Kazakh National University named after al-Farabi, Almaty, Kazakhstan
[3]Department of Design Engineering and Mathematics, Middlesex University, London, United Kingdom
E-mail: [1]baitureyeva.aiymzhan@gmail.com, [2]alibek.issakhov@gmail.com, [3]T.Yang@mdx.ac.uk

Abstract - This paper presents CFD simulation of pollution dispersion from a thermal power plant. Carbon dioxide was chosen as the scattering gas, as it constitutes the main share of emissions from the energy industry. The model was tested using experimental results performed using wind tunneldata available in the literature. A comparative analysis of the results of this article with experimental and numerical data was performed. It showed that the results of this article were closer to the experimental results than the calculations of previous authors. The minimum relative error with the experiment was less by 4.11% at the pipe exit and by 2.52% at a distance x/D = 3 from the source, than other results.Based on this verification, the spread of pollution from thermal power plant (TPP) in real physical dimensions was modeled. The k-epsilon turbulence model was used taking into account buoyancy. The calculations were performed using the ANSYS Fluent 18.1software package. As a result, the distance from the source was calculated, at which pollution will reach the ground surface (~ 2 km).Obtained distance is quite big since this TPP is located in an area which is far from residential settlements and there are no natural or architectural obstacles around.

Keywords - Navier-Stokes equations, mass transfer, numerical modeling, air pollution, concentration, thermal power plant.

I. INTRODUCTION

Air pollution every year becomes an increasingly serious large-scale problem. Plants and various energy facilities (such as thermal power plants, nuclear power plants, etc.) produce a large amount of pollutants that dispersed in the atmosphere, damage the flora, fauna, buildings and harm human health. The European Environment Agency (2018) gives the following definition of air pollution: “the presence of contaminant or pollutant substances in the air that do not disperse properly and that interfere with human health or welfare, or produce other harmful environmental effects.” [1]. According to the final emissions report for 2017, published in March 2018, global energy-related CO_2 emissions have increased and reached a historic maximum. At the same time, special attention should be paid to the energy sector, since the share of energy is more than two thirds of total greenhouse gas emissions and more than 80% of CO_2 emissions[2]. Therefore, in this paper, CO_2 was selected as the main test substance of pollution. The background annual CO_2 concentration on the Earth is equal to 400.88 ppm = 0.0004 (mass fraction) [3]. To determine the extent of air pollution impact on the environment and people, it is important to take into account the physical principles affecting the movement and dispersion of pollutants [4]. Due to the rapid growth of computer capabilities, in particular, large-scale parallel computing, it becomes advisable to use computer simulation to calculate scientific and technical engineering problems. Nowadays technologies are rapidly developing, as a result of which their productivity has increased exponentially over time. According to the data, over the past 60 years, computing power has increased in productivity by 1 trillion times. [5] Therefore, people use computational fluid dynamics to conduct large-scale computer modeling of important scientific and engineering topics [6-7]. The study of jet behavior in crossflow is important for various applications, especially for chimneys, because the interaction between the jet and crossflow fluids affects the pollution dispersion into the atmosphere [8]. A review and description of the works devoted to the study of the nature of jet motion in a crossflow is given in [9-14]. Early studies of jets in crossflow were devoted to the derivation of empirical equations for the flight path and the principles of scaling [15-18].To this end, the authors conducted numerous experimental studies. Recent research in this area is described in [19, 20]. Further, there have been many studies of vortex structures (vortex pairs rotating in opposite directions, horseshoe vortices); stability and destruction of the jet [21-22]. The purpose of this work was to assess the impact of emissions on the environment based on a numerical model of the spread of pollutants from sources. One of the pipes of Ekibastuz Thermal Power Plant-1 (Kazakhstan) was chosen as a real physical object of research [23]. Its height is 330 [m], the pipe diameter is 10 [m].

II. MATHEMATICAL MODEL

Computational fluid dynamics has proven to be an effective tool for modeling the behavior of jets in crossflow.Modeling of such problems is based on the resolution of the Navier-Stokes equations (the equation of continuity and the equation of motion) [24-25].It was found that Reynolds-averaged Navier-Stokes equations (RANS) modeling, can qualitatively predict the behavior of the total flow and concentration [26].Past studies have revealed that non-stationary large eddy simulation (LES) models provide good agreement with experimental results in pollutant dispersion problems [27]. However, the computational cost of this model is about 100 times higher than the cost required for the RANS model [28]. Important observations regarding RANS k-epsilon were noted in [29]. In present work, RANS k-epsilon model was used.

Also, a comparative analysis of the obtained results with the experimental [30] and numerical [31] data was carried out.The SIMPLE method was chosen for the calculation. This method has been applied in multiple numerical studies and, when compared with experimental data, has shown good agreement. [29].

III. TEST PROBLEM

A detailed description of the test problem and experiment is given in [30, 31]. The test problem domain is a three-dimensional channel with a pipe inside it. The pipe diameter (jet width) was D = 12.7 [mm], which was used as a characteristic unit of length. The dimensions of the geometry are shown in Fig. 1.

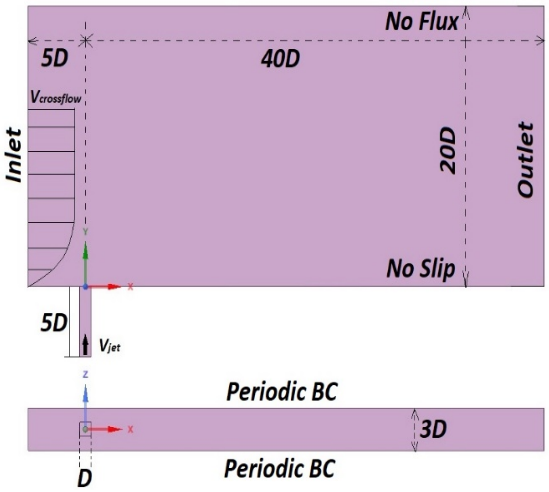

Fig.1.Configuration of the computational domain ofthe test problem

An unstructured grid was constructed, the total number of nodes was 533 697 (See Table 1). The ratio of the jet velocity to the velocity of the crossflow is denoted as $R = V_{jet} / V_{crossflow}$ .In the present work, R = 0.5 was considered: the jet velocity was 5.5 [ms^{-1}], the crossflow velocity was 11 [ms^{-1}].

Table 1: Number of grid points

NI	230
NJ	100
NK	21
Total body sizing	0.0025 [m]
Total nodes	533 697

Air was chosen as the main fluid material for the crossflow and the jet. The Reynolds number has been defined as:

$$Re_{jet} = \rho V_{jet} D/\mu = 4700. \quad (1)$$

Five types of boundary conditions were used: inlet, outlet, periodic, no slip, no flux (see Fig. 1). According to experimental data, the thickness of the boundary layer is equal to 2D. The wind velocity profile was defined by a power law with exponent 1/7 within the boundary layer and was set as constant 11 [ms^{-1}] above it.Since a smooth surface was used in the experiment, the roughness height was zero.

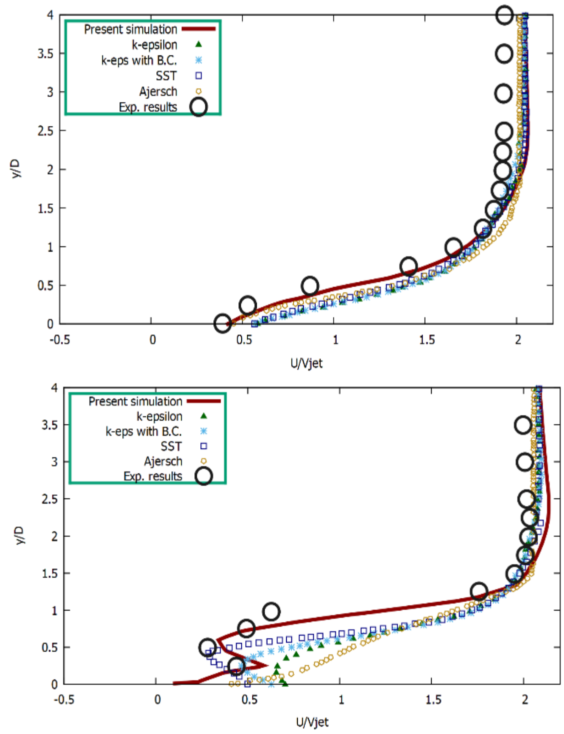

Fig.2. (a & b) Comparison of the obtained results with experiment data and calculations of other authors. (a) x/D=0.0, (b) x/D=3.0

Fig. 2 (a&b) shows a comparison of the numerical results of this study with experimental data and numerical solutions of other authors at various distances from the jet (R = 0.5; x/D = 0.0 and x/D = 3.0). At the Fig 2, b, the values of the red line (u/Vjet) at y/D = 0 approach zero, while the plots of other authors in this interval show the values of u/Vjet ~ 0.5-0.7. A zero value is more reliable from a physical point of view, as this is a near-wall field.Also, in this region (y/D ~ 0 – 1) the relative errors of present simulation are smaller, than others (see Table 2). Based on this data, the solutions obtained in this work turned out to be more accurate than the calculations obtained by other authors [30, 31]. The reason is the quality of the grid: in this work, an unstructured grid was used (the number of nodes was 533 697), while in [31] a structured grid was used (the number of nodes was 265 000).

Table 2: Relative errors of numerical simulations for R=0.5

	Min. relative error		Max. relative error	
x/D	0.0	3.0	0.0	3.0
k-epsilon	6.78%	7.54%	87.74%	207.23%
k-eps with BC	6.57%	6.2%	85.16%	169.66%
SST	6.57%	7.9%	72.9%	170.1%
Ajersch	9.65%	8.32%	51.37%	279.52%
Present paper	2.46%	3.68%	22.58%	89.13%

IV. EKIBASTUZ THERMAL POWER PLANT–1

For the real thermal power plant simulation,the computational domain was considered as three-dimensional box with a pipe inside it (see Fig. 3). The boundary conditions were set similarly to the test problem. The geometry length was 2500 [m], height - 1500 [m], width - 200 [m]. The stack was located at 250 [m] from the boundary of the wind inflow. An unstructured mesh was constructed and it was refined in the area of the trajectory of pollution and around the stack (see Fig. 4). The computational grid consists of 822 056 nodes and 4 788 517 three-dimensional elements. In order to take into account the influence of the boundary layer, the wind velocity profile was described as follows [12]:

$$v_x = v_{wind} \cdot (0.2371 \cdot \ln(Y + 0.00327) + 1.3571) \quad (2)$$

where the wind velocity (v_{wind}) was 1.5 [ms^{-1}], the pollution rate was 5 [ms^{-1}].This speed was chosen according to the wind rose of the Ekibastuz city, where the speed above 1 m/s is fixed for about 250 hours per year in each direction (See Fig. 5). $CO_2$was chosen as a pollution material. In this paper, the chemical reaction of substances was neglected. The concentration distribution of pollution is shown in Fig. 6, which is visualized using the Volume Rendering option in ANSYS. Visually, with increasing distance from the source, a diffusion effect is observed. The concentration was measured in mass fractions. The mass fraction of a substance is the mass ratio of this substance per unit mass of the mixture (for example, kg of substance per 1 kg of the mixture), i.e. it is a dimensionless quantity [32]. To convert it to ppm, the following formula is necessary: PPM = mass fraction * 10^6

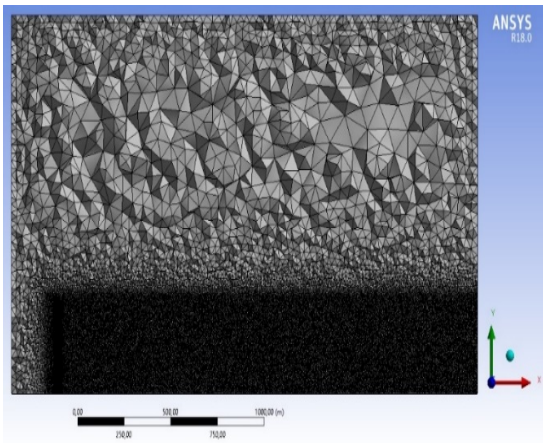

Fig.4. Computational mesh

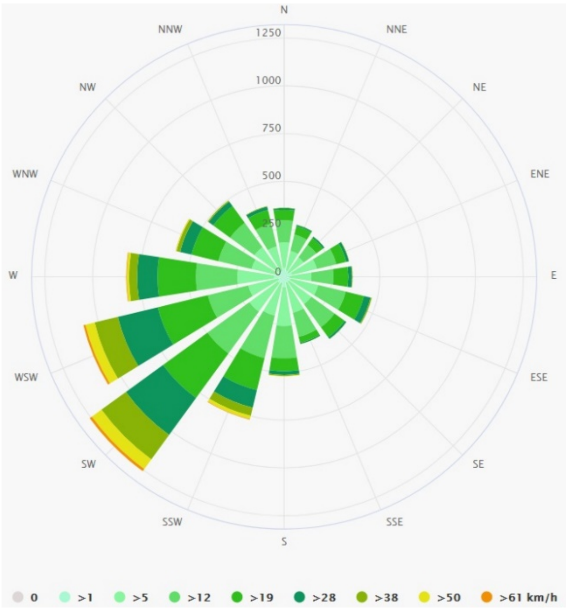

Fig.5. Annual wind rose of the Ekibastuz city.

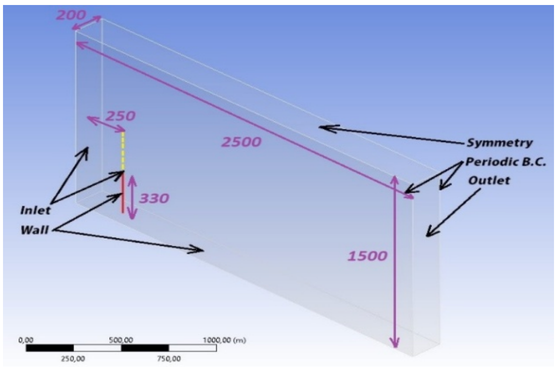

Fig.3. Configuration of computationaldomain for thermal power station

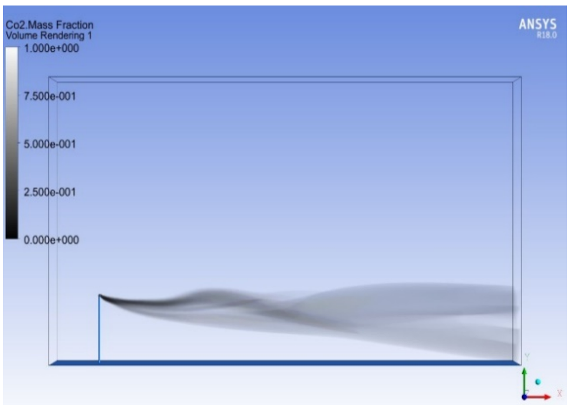

Fig.6.CO_2 distribution

CO_2distribution profiles at different distances from the source (500, 1000, 1500, 2000 and 2200 [m]) illustrated on the Fig. 7. According to the results, at a distance of 2200 [m], the concentration of emissions spreads at the height of 420 [m] from the ground surface. Two-dimensional contour of the pollution distribution is illustrated on the Fig 8. According to the plot, the pollution settles on the ground surface at a distance of about 2,220 [m] from the origin of coordinates (i.e., 1970 [m] from the source).

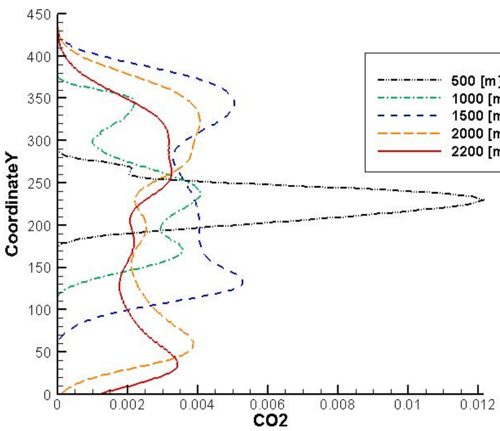

Fig.7. CO_2distribution profiles at different distances from the source: 500, 1000, 1500, 2000, 2200 [m]

CONCLUSIONS

The purpose of the study was to study the dynamics of the pollution dispersion. The mathematical model and numerical algorithm were tested using an experimental test problem. The results were closer to the experimental, compared with the data of other authors. Using the example of a real thermal power plant, CO_2dispersion was modeled. The k-epsilon model was used without any additional dispersion models. As a result, it was determined at what distance from the source the pollution accumulates on the ground surface. According to the obtained data, with increasing distance from the source, the concentration of pollution spreads more widely under the influence of diffusion. The farther the distance from the pipe, the lower the concentration of the substance.

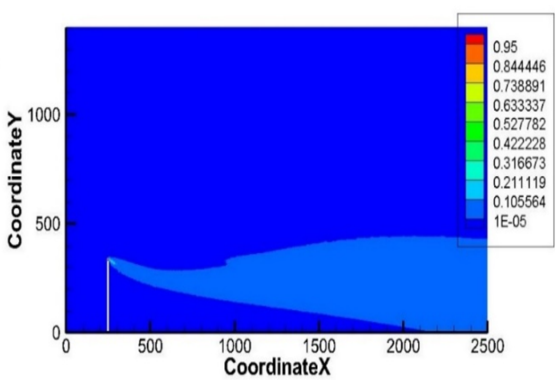

Fig.8. Two-dimensional contour of pollution distribution.

Thus, the obtained numerical data may allow to predict the optimal distance from residential areas for the construction of thermal power plants, at which the concentration of emissions will remain at a safe level in the future.

REFERENCES

[1] ECSC. (2018). Glossary: Primary & Secondarypollutant. [online] Available at: http://ec.europa.eu/health/scientific_committees/opinions_layman/en/indoor-air-pollution/glossary/pqrs/primary-pollutant-secondary-pollutant.htm (Accessed 5 Jul. 2018).

[2] International Energy Agency (IEA). CO2 Emissions from Fuel Combustion; 2018

[3] CO2 Earth. Annual CO2 Data, [online] Available at: https://www.co2.earth/annual-co2 Accessed: (15 January 2019)

[4] Zavila O. Physical Modeling of Gas Pollutant Motion in the Atmosphere, Advances in Modeling of Fluid Dynamics, Dr. Chaoqun Liu (Ed.), InTech, 2012, DOI: 10.5772/48405.

[5] Visual Capitalist. (2017). Visualizing the Trillion-Fold Increase in Computing Power. [online] Available at: https://www.visualcapitalist.com/visualizing-trillion-fold-increase-computing-power (Accessed: 15January 2019)

[6] Jiangfei Zheng "The new direction of computational fluid dynamics and its application in industry", J. Phys.: Conf. Ser. 1064 012060, 2018

[7] "Computational Fluid Dynamics Past, Present and Future". [online] Available at: http://aero-comlab.stanford.edu/Papers/NASA_Presentation_20121030.pdf (Accessed: 15 January 2019)

[8] Z. Rek et al. "Numerical simulation of gas jet in liquid crossflow with high mean jet tocrossflow velocity ratio ". Chemical Engineering Science 172 (2017) 667–676

[9] Margason, R. J. 1993 Fifty years of jet in crossflow research. In AGARD Symp. on a Jet in Cross Flow, Winchester, UK. AGARD CP 534.

[10] Broadwell, J. E. ,Breidenthal, R. E. 1984 Structure and mixing of a transverse jet in incompressible flow. J. Fluid Mech. 148, 405–412.

[11] Karagozian, A. R. 1986 An analytical model for the vorticity associated with a transverse jet. AIAA J. 24, 429–436.

[12] Hasselbrink, E. F. ,Mungal, M. G. 2001 Transverse jets and jet flames. Part 1. Scaling laws for strong transverse jets. J. Fluid Mech. 443, 1–25.

[13] Muppidi, S. , Mahesh, K. 2005 Study of trajectories of jets in crossflow using direct numerical simulations. J. Fluid. Mech. 530, 81–100.

[14] Muppidi, S. , Mahesh, K. 2008 Direct numerical simulation of passive scalar transport in transverse jets. J. Fluid Mech., 598, pp. 335–360.

[15] Callaghan, E.E., Ruggeri, R.S., 1948. Investigation of the penetration on an air jet directed perpendicularly to an air stream.

[16] Margason, R.J., 1968. The Path of a Jet Directed at Large Angles to a Subsonic Free Stream. Langley Research Center.

[17] Golbitz, W.C., 1980. Time Dependent Navier-Stokes Solution of a Turbulent Gas Jet Ejected from a Rectangular Orifice into a High-Subsonic Crossflow. Faculty of the School of Engineering of the Air Force Institute of Technology.

[18] Cutler, P.R.E., 2002. On the Structure and Mixing of a Jet in Crossflow. The Universe of Adelaide.

[19] E. Erdem, K. Kontis, S. Saravanan Penetration characteristics of air, carbon dioxide and helium transverse sonic jets in mach 5 cross flow Sensors (Switzerland), 14 (2014), pp. 23462-23489, 10.3390/s141223462

[20] T. Cambonie, N. Gautier, J.L. Aider Experimental study of counter-rotating vortex pair trajectories induced by a round jet in cross-flow at low velocity ratios Exp. Fluids, 54 (2013), pp. 1-13, 10.1007/s00348-013-1475-9

[21] Karagozian, A.R., 2014. The jet in crossflow. Phys. Fluids

26.http://dx.doi.org/10.1063/1.4895900
[22] T.F. Fric, A. RoshkoVortical structure in the wake of a transverse jet J. Fluid Mech., 279 (1994), pp. 1-47, 10.1017/S0022112094003800
[23] Wikipedia, "Ekibastuz_GRES-1". [online] Available at: https://en.wikipedia.org/wiki/Ekibastuz_GRES-1 (Accessed: 15 January 2019)
[24] Ferziger J. H., Peric M. Computational Methods for Fluid Dynamics. Springer; 3rd edition, 2013, -p. 426
[25] Chung T. J. Computational Fluid Dynamics. Cambridge University Press, 2002 - p. 1012.
[26] Tominaga, Y., Stathopoulos, T., 2009. Numerical simulation of dispersion around an isolated cubic building: comparison of various types of k–e models. Atmospheric Environment 43, 3200–3210.
[27] M. Chavez et al., "Near-field pollutant dispersion in the built environment by CFD and wind tunnel simulations". J. Wind Eng. Ind. Aerodyn. 99 (2011) 330–339
[28] Cheng, Y., Lien, F.S., Yee, E., Sinclair, R., 2003. A comparison of large eddy simulations with a standard k?e Reynolds-averaged Navier Stokes model for the prediction of a fully developed turbulent flow over a matrix of cubes. Journal of Wind Engineering and Industrial Aerodynamics 91, 1301–1328.
[29] B. Blocken et al., "Numerical evaluation of pollutant dispersion in the built environment: Comparisons between models and experiments". J. Wind Eng. Ind. Aerodyn. 96 (2008) 1817–1831
[30] Ajersch, P., Zhou, J. M., Ketler, S., Salcudean, M., and Gartshore, I. S., "Multiple Jets in a Crossflow: Detailed Measurements and Numerical Simulations," International Gas Turbine and Aeroengine Congress and Exposition, ASME Paper 95-GT-9, Houston, TX, June 1995, pp. 1–16.
[31] Keimasi M.R., Taeibi-Rahni M., "Numerical Simulation of Jets in a Crossflow Using Different Turbulence Models", AIAA journal, Vol. 39, No. 12, December 2001
[32] ANSYS Fluent Theory Guide 15, ANSYS Ltd., 2013.

★★★