JOURNAL OF MATERIALS AND ENGINEERING STRUCTURES 6 (2019) 375–382

Journal of Materials and Engineering Structures

Research Paper

Structure of the out- flows behind buildings and Influence of the geometry of the streets on the out-flows

Medjahed Bendida [a,*]***, Amel Djellouli*** [a]***, Djouhar Hamidat*** [a] ***, Mohamed Bouzit*** [b]

[a] *Laboratory of Numerical and Experimental Modeling of Mechanical phenomena, University of Abdelhamid Ibn Badis, Faculty of Science and Technology , Mostaganem, Algeria.*

[b] *Laboratory of Applied Mechanics; University Of Science and Technology of Oran, Faculty of Mechanical Engineering, BP. 1505 El-Mnaouar, Oran, Algeria.*

ARTICLE INFO

Article history:

Received : 3 October 2018

Revised : 23 April 2019

Accepted : 27 April 2019

Keywords:

Numerical simulation

Wind speed

Building

Street canyon

ABSTRACT

This paper intends to study the latest results from the research methods available with special intentions given to the architectural effects of the street valley and wind speed. Extensive research has been carried out through several research approaches to understand the effect of wind flow formation in the streets and the current wind condition on the structure of the wind current. The main goal of this paper is to study the structure of out-flows of buildings and the effect of street engineering on external flows. The numerical modeling is to simulate the effect of the wind flow and the layer limit on different building structures using the ANSYS Fluent package. The program was based on the K-ε model to incorporate the potential of differential equations forming the mathematical model. Three cases were considered; the first case is the height-to-width ratio of the valley (h/w), the second is the width of the dome (b3/b) and the third case the ratio of the height of the valley (h3/h) to see the effect of street valley engineering and wind speed effect.

1 Introduction

In recent years, the problem of air quality has become the major problem of large industrialized countries with high population density worldwide. Air quality in many densely populated urban areas is threatened by based pollutants produced by factories have a significant impact on the urban population. The reason for the lack of ventilation is due to the lack of respect for the structure of external flows behind the buildings, which leads to its impact on the engineering of the streets and even on external flows, leading to deterioration of air quality.

---

* *Corresponding author. Tel.: +213655203399.*
E-mail address: medjahedb@gmail.com

e-ISSN: 2170-127X,

RESEARCH REVIEW of Sciences and Technologies

To understand this problem, many laboratories worldwide have recruited many researchers to conduct scientific research to clarify the effects and proportion of building aspects in the structure of external flows behind buildings and the impact of street engineering on external flows. Flow visualisation studies by Perry et al. (1969) showed that the flow within these cavities between the elements is largely self contained, being separated from the outer flow by a shear layer across the top of the cavity [1]. In 1988; Oke has the paper sought to show some answers about understanding the effects and proportion of building aspects in the structure of external flows behind buildings and the impact of street engineering on external flows [2]. It has been and many researchers are still doing research to find a clear solution to this phenomenon of dispersion within the valley in the streets, and has been conducting extensive research in this area, namely in urban streets in recent years because of its effects on air quality in addition to the international attention to resolve this problem which affects the health Population. This research was conducted through the structure of the out- flows behind buildings and Influence of the geometry of the streets on the out- flows e.g., Huang and al., 2000 [3]; Kim and al., 2001 [4]; Pirouzpanah, 2001 [5]; Kastner-Klein and al., 2004 [6]; Bert Blocken and al., 2004 [7]; Xie Xiaomin and al., 2006 [8]; Ali-Toudert a and al., 2007 [9]; Nastaran Shishegar, 2013[10]; Jonas Allegrini and al., 2018 [11]. Many the numerical study by CFD software have contributed to our understanding and computational fluid dynamics (CFD) model simulations. These include flow regime e.g, Sotiris Vardoulakis and al., 2003[12]; Tabejamaat, 2003 [13] ;Kim and al., 2004 [14], Xian-Xiang and al., 2006[15]; Zhang and al., 2006 [16]; Najafpour and al, 2010 [17]; Mauro Scungio and al., 2013 [18].

In April 2001, Chan Andy T. and al.; did two studies; the first for the strategic guidelines for street canyon geometry to achieve sustainable street air quality. In this simulation work, we consider a canyon model with l/h and h/w, where l is the canopy length, h is the building height in consideration and w is the width. The second in March 2003; for the Strategic guidelines for street canyon geometry to achieve sustain able street air quality (part II: multiple canopies). In this work, the canyon model has been modified in order to investigate the multi-block effects on canyon flow (Individual canyon height-to-width ratio, canopy breadth ratio and canyon height ratio) [19, 20]. In 2014, I posted a paper titled: job a comparison work between numerical models and CHENSI with experimental data (MUST) in the case of wind direction 0 ° [21]. I published another paper in 2016 under the title: an article on the numerical study of the dispersion of the pollutant in an Urban Canopy [22]. In July 2016, we did the job by PhD thesis at Mostaganem University in Algeria whose theme is the numerical study of the dispersion of a passive scalar in urban simplifies. The study presented two main objectives, including a study of the flow and dispersion of pollutants in cities, by modelling both average concentrations and their fluctuations using a CFD (Computational Fluid Dynamics) model that uses the CHENSI code [23].

2 Problematic

The present work is study of structure of the out-flows behind buildings and Influence of the geometry of the streets on the out-flows, the canyon model has been modified in order to investigate the multi-block effects on canyon flow. The system with canyon domain in dimension H x W (60 x m 350 m) and canyon building height h; breadth b and the canyon width w is shown in Figure. 1. Canyon ratio h/w is selected in reference with the regimes classified by Oke [2].

The standard height h; breadth b and canyon width w are thus 20, 10 and 60 m, giving h/w=0.33: This implies that there is a lateral spacing of 20 m between the canopy blocks with the adjacent domain and this effectively constitutes the two-dimensionality of the problem. Numerical results presented in this study for the mean flow field for three cases (03 cases): 1st case is individual canyon height-to-width ratio (h/w), 2nd case is canopy breadth ratio (b3/b) and 3rd case is canyon height ratio (h3/h) to see the effect of street canyon geometry and effect of wind speed.

In 1st case (individual canyon height-to-width ratio (h/w)): in this study for the mean flow field for three types (03 types) by the flow pattern classifications: a) is isolated roughness flow (see figure 2.a and table 02), b) wake interference flow (see figure 2.b and table 02) and c) skimming flow (see figure 2.c and table 02); between the second and third buildings, to see the effect of street canyon geometry and effect of wind speed.

In 2nd case (canopy breadth ratio (b3/b)): in this study for the mean flow field for three types (03 types); the first type b3/b = 0.5, the second type b3/b = 2 and the third type b3/b = 4; on the third building (The report made reference to the one of the Canyon #3).

3rd case (canyon height ratio (h3/h)): in this study for the mean flow field for three types (03 types); the first type h3/h = 0.25, the second type h3/h = 0.5 and the third type h3/h = 2; on the third building (The report made reference to the one of the Canyon #3).

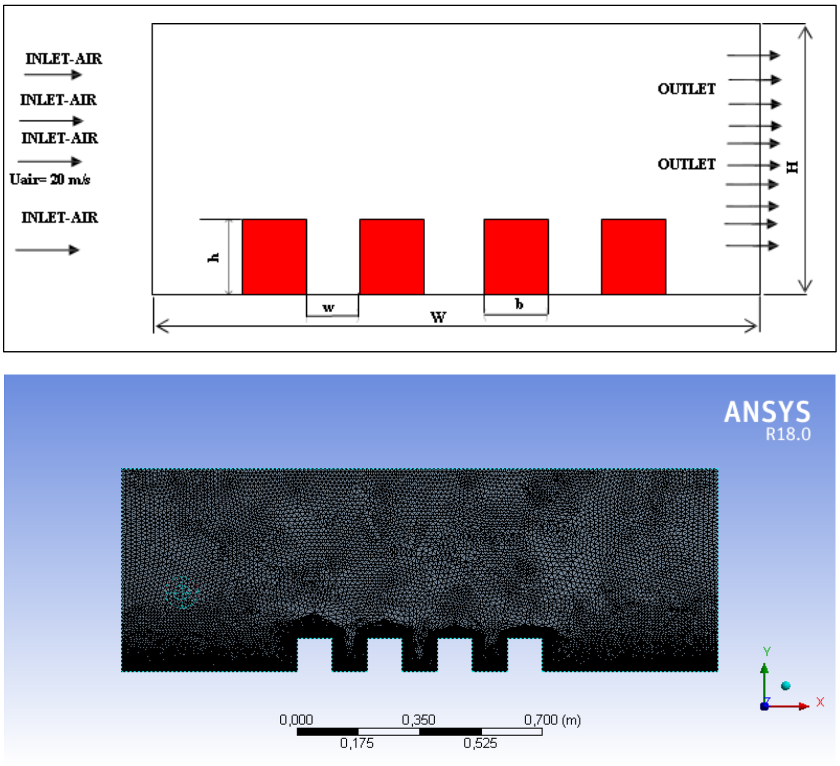


Fig. 1 –Domain of simulation and Mesh used for numerical simulations (boundary conditions on each surface are given in Table 1).

Table 1- Boundary conditions.

Nature	Input value
Inlet	Normal speed of Air at 20 m/s
Outlet	Standard temperature and pressure
Overall exposed building walls	No slip condition

The process of generating the 2D CFD model was done inside the ANSYS FLUENT 18 (Workbench 18).

Particularly, the numerical parameters the workflow of the model was as follow:

- Generation of a simplified 2D and computational domain;
- High quality meshing of the domain to meet the specifics of the turbulence models and reach grid independent solutions;
- Simulations for steady state based on Reynolds-Averaged Navier-Stokes equations (RANS) with constant fluid properties;
- Standard k-e turbulence model with standard constants and optimization of the Transition Turbulence model;
- Setting the Fluent Solver and calibrating the model;
- Double Precision (fine);
- Governing equations solved by means of a collocated grid system using finite volume method;
- Numerical approximations;
- Iterative convergence criteria with10-6 for all variables never reached for continuity and turbulence.

3 Flow pattern classifications and interaction of flows around obstacles

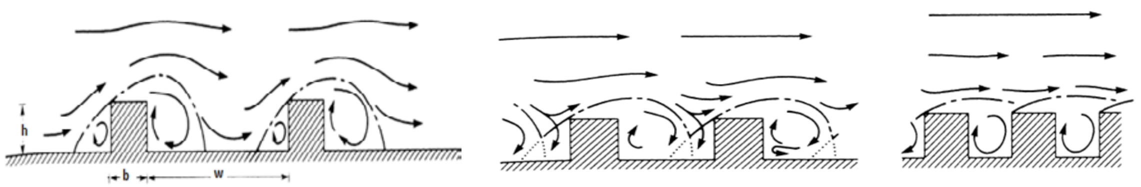
The street-level dispersion study involves the interaction of kinematic fields specific to each obstacle. We consider the simplified case of an incident flow perpendicular to the street, itself delimited by two obstacles of height equal to h. Depending on the distance between these two obstacles, noted s, one can observe three flow regimes (see figure 2 and table 2 [2].

We thus distinguish:

a) Isolated roughness flow (w / h> 3, 5 for a two-dimensional obstacle and w / h> 2.2 for a cube) the recirculation zones downstream of the first obstacle and upstream of the second obstacle do not interact. The flows observed are those described in the previous paragraph (see figure 2.a and table 2).

b) Wake interference flow (1, 6 <w/h<3, 5 for a two-dimensional obstacle and 1.3 <w/h<2.2 for a cube) this is a complex inteminational regime that sees puffs Turbulent reach the wall between the two obstacles. The two internal recirculation zones on the street interact strongly (see figure 2.b and table 2).

c) Skimming flow (w / h <1, 6 for a two-dimensional obstacle and w / h <1.3 for a cube) In this case the distance between the obstacles is insufficient to allow the two recirculation zones to coexist. There is only one current that passes both roofs obstacles without penetrating directly inside the street (see figure 2.c and table 2).



a) Isolated roughness flow w/h> 3.33 b) Wake interference flow 1.53 < w/h < 3.33 c) Skimming flow w/h < 1.53

Fig. 2 –Flow pattern classifications (Oke [2]).

Table 2- Boundary between flow regimes (see figure 2).

Isolated roughness flow	Wake interference flow	Skimming flow
w/h = 6 [25]	4 > w/h > 2 [25]	1 > w/h > 0.5 [25]
w/h = 4 [24]	w/h = 3.713 [1]	w/h = 0.9 [1]
w/h > 3.33 [2]	3.33 > w/h > 1.53 [2]	w/h < 1.53 [2]

4 Results and discussion

In this article, this work is structure of the out-flows behind buildings and Influence of the geometry of the streets on the out-flows, ratio of the roughness array (h/w) it is of great importance of gap for the separation and reattachment of the stream lines in the canyon. Numerical results presented in this study for the mean flow field for three cases (03 cases): 1st case is canyon height-to-width ratio (h/w), 2nd is canopy breadth ratio (b3/b) and 3rd case is canyon height ratio (h3/h) to see the effect of street canyon geometry and effect of wind speed.

In figure 3, the 1st case (canyon height-to-width ratio (h/w)): in this study for the mean flow field for three types (03 types) by the flow pattern classifications: a) is isolated roughness flow (see figure 2.a and table 02), b) wake interference flow (see figure 2.b and table 02) and c) skimming flow (see figure 2.c and table 02); between the second and third buildings. To see the effect of street canyon geometry and effect of wind speed. Are presented the 1st case (canyon height-to-width ratio (h/w), in figure 3.a (isolated roughness flow), for wide canyons, the roughness elements are well spaced and act essentially as isolated roughness elements. In figure 3.b (wake interference flow), as roughness become more closely spaced, the disturbed flow has insufficient distance to readjust and the type of skimming flow, the bulk of the flow skims over the canyon (figure 3.c). In this system of four canopies, only the canyon width between Buildings 2 and 3 will vary in this section while the other canyon widths remain constant. The variation will show the h/w that cover the three regimes reported by Oke

[2]. The results obtained in the simulations show the similar phenomenon and the current regime of the isolated roughness is developed in wider canyon, with h / w below 0.5 and wakes regime of the intervention current which elongates between h / w ratio 0.5-0.8 before transition in skim the current (Figure. 3). It is noted that the value of the retention shows the respective current regimes distinctly (see figure 4).

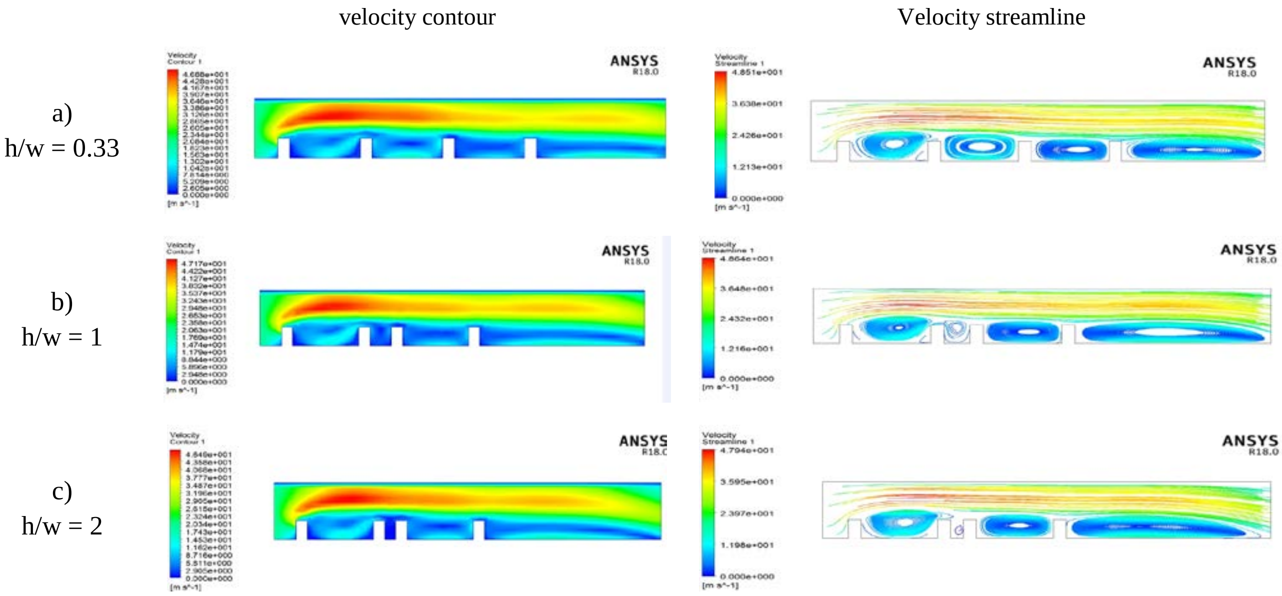


Fig. 3 – Velocity contour and velocity streamline in 1st case (individual canyon height-to-width ratio (h/w)), on flow pattern to: a) h/w = 0.33, b) h/w = 1 and c) h/w = 2; between the second and third buildings

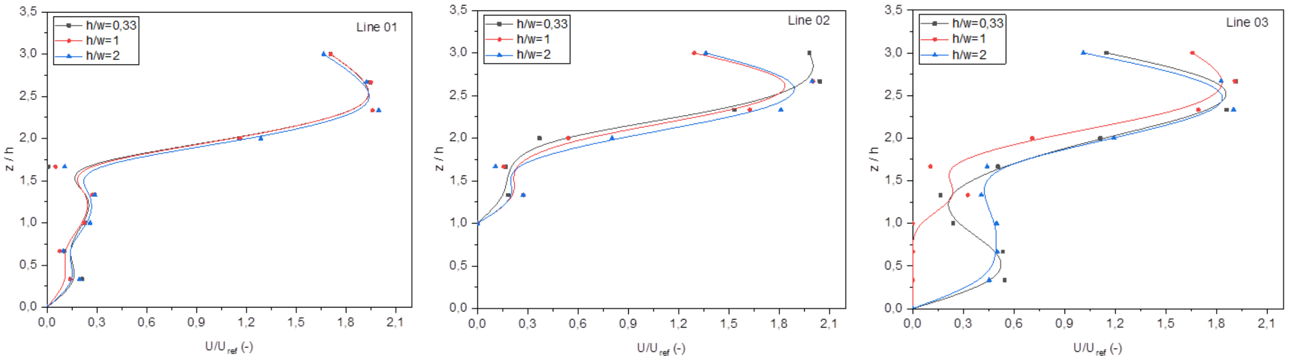


Fig. 4 – Vertical profiles of speed; line 01: x_1/h=3, line 02: x_2/h=4.75 and line 03: x_3/h=6.5 (see figure 3)

In figure 5, the 2nd case (canopy breadth ratio (b3/b)): in this study for the mean flow field for three types (03 types); the first type b3/b = 0.5, the second type b3/b = 2 and the third type b3/b = 4; on the third building. Increasing the width of the canopy increases the intensity of turbulence and the depth of friction influence. The width of the canopy to build 3 more than 2 in this case serves to examine the effect of the roughness imposed by building it downstream of the canyon as shown in figure. 5. as the building width increases to a value of 2.0, a critical relative roughness and thus a maximum intensity of turbulence is reached (see figure 6).

In figure 7, the 3rd case (canyon height ratio (h3/h)): in this study for the mean flow field for three types (03 types); the first type h3/h = 0.25, the second type h3/h = 0.5 and the third type h3/h = 2; on the third building (The report made reference to the one of the Canyon #3). A building on the lower wind side (h3/h <1) do not encourage canyon air acceleration over the roof, the weaker whirlpool inside the canyon. This defect can be compensated by the re-entrainment of fresh air into the canyon cavity downstream of building 3. The value of maximum retention occurs at h3/h = 1.2 because of stagnation upstream of Building 3. This ratio of height obtained is close to the previous study for a system of two canopies. The zone of the current of stagnation extends until h3/h= 2; Where to build 3 almost block the entire quill and the scatter box occurs only at the lateral sides of the canopies (see figure 8).

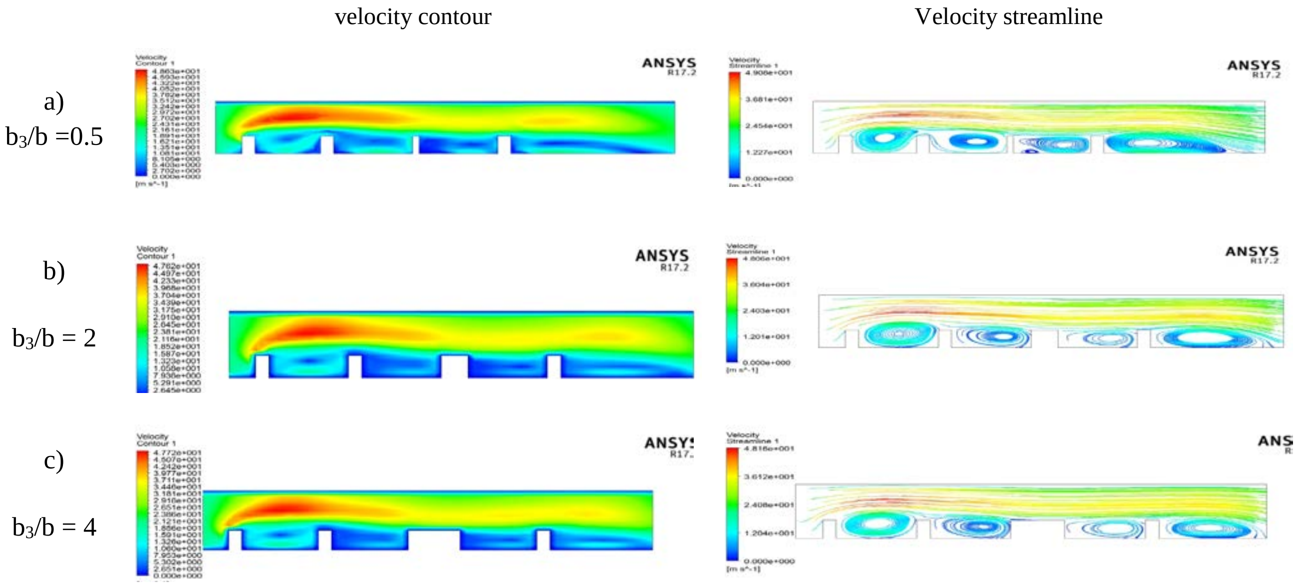


Fig. 5 – Velocity contour and velocity streamline in 2nd case (canopy breadth ratio (b_3/b)): in this study for the mean flow field for three types (03 types); the first type $b_3/b = 0.5$, the second type $b_3/b = 2$ and the third type $b_3/b = 4$; on the third building

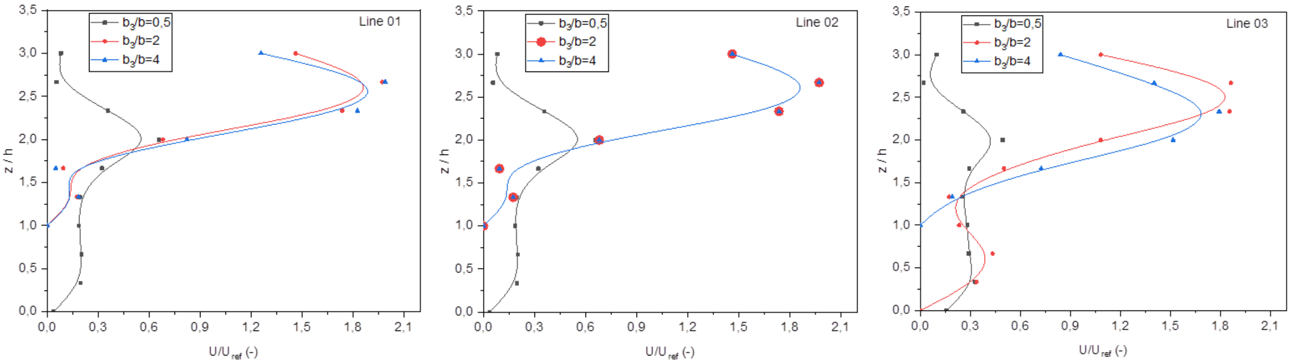


Fig. 6 – Vertical profiles of speed; line 01: $x_1/h=3$, line 02: $x_2/h=4.75$ and line 03: $x_3/h=6.5$ (see figure 5)

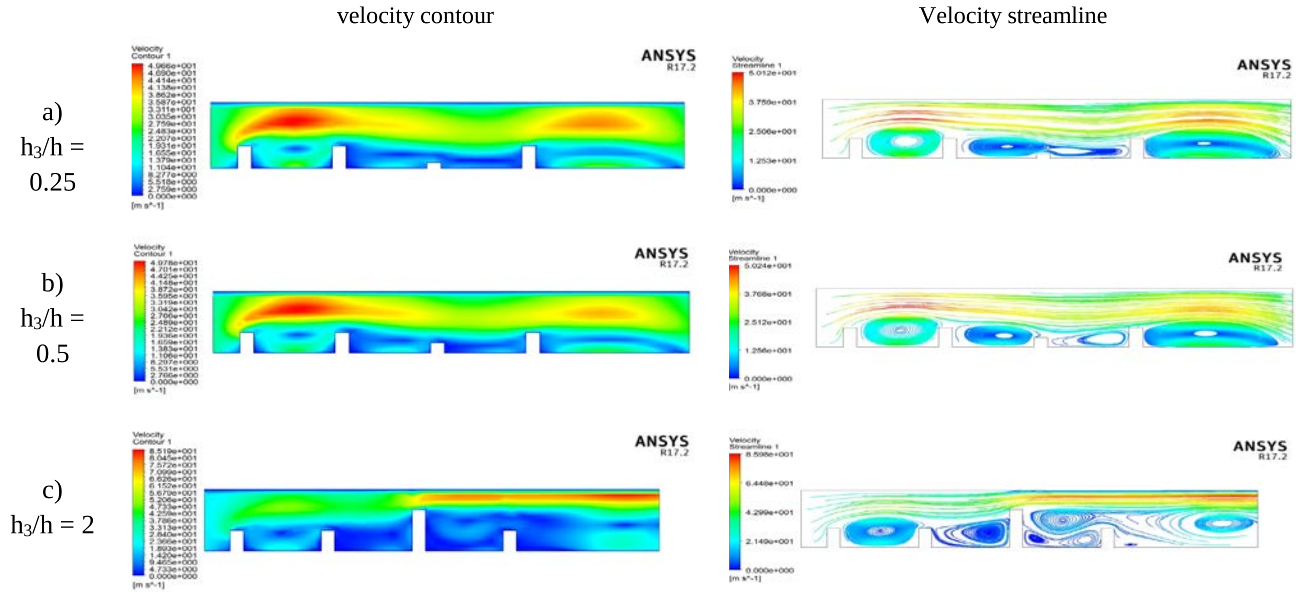


Fig. 7 – Velocity contour and velocity streamline on flow pattern in 3rd case (canyon height ratio (h_3/h)): in this study for the mean flow field for three types (03 types); the first type $h_3/h = 0.25$, the second type $h_3/h = 0.5$ and the third type $h_3/h = 2$; on the third building (The report made reference to the one of the Canyon #3)

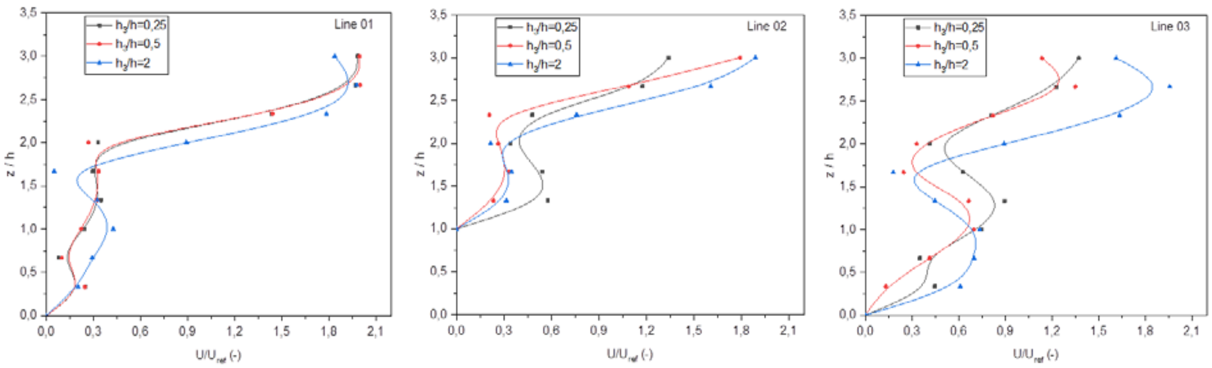


Fig. 8 – Vertical profiles of speed; line 01: x1/h=3, line 02: x2/h=4.75 and line 03: x3/h=6.5 (see figure 7)

5 Conclusion

Numerical results presented in this study for the mean flow field for three cases (03 cases): 1st case is individual canyon height-to-width ratio (h/w), 2nd case is canopy breadth ratio (b3/b) and 3rd case is canyon height ratio (h3/h) to see the effect of street canyon geometry and effect of wind speed. Researchers in this area have discussed many previous research that urban climate science has become a more perfect and predictable science so that its results can be of direct value to the planning and design of urban buildings and structures. The purpose of this paper is to show some solutions and answers on how to understand the structure of external flows behind buildings and the impact of street engineering on external flows.

In this paper, several questions have been addressed to understand the reasons for the impact on quality and quality of air within population areas and the fundamental question we ask: "Are we adhering to valuable advice and guidance to engineers on street engineering by scientific research on urban climate and air quality in urban areas? ". To think about answering such a question, there are many valuable ideas and unlimited possibilities in our imagination. For example, different climatic contexts, urban geometry, climate variables and engineering design.

The choices are among the alternatives available to us. In the case of street climate design, objectives may be inconsistent. An example is, while open engineering leads to dispersion of air pollution and access to solar energy, the arrangement of the densest clusters is favourable. This problem is investigated by re-reading the results of studies and recent field research carried out in the Urban Canyon and on the scale and mathematical modelling by several researchers. By focusing on measurable relationships, it is possible to find a range of valley geometry that conforms to conflicting design goals in urban cities and multitudes.

REFERENCES

[1]- A.E. Perry, P.N. Joubert, Rough-wall boundary layers in adverse pressure gradients. J. Fluid Mech. 17 (2) (1963) 193-211. doi:10.1017/S0022112063001245

[2]- T.R. Oke, Street Design and Urban Canopy Layer Climate. J. Energ. Build. 11(1-3) (1988) 103-113. doi:10.1016/0378-7788(88)90026-6

[3]- H. Huang, Y. Akutsu, M. Arai, M. Tamura, A two dimensional air quality model in an urban street canyon: evaluation and sensitivity analysis. Atmos. Environ. 34(5) (2000) 689–698. doi:10.1016/S1352-2310(99)00333-7

[4]- J.J. Kim, J.J. Baik, Urban street canyon flows with bottom heating. Atmos. Environ. 35(20) (2001) 3395–3404. doi:10.1016/S1352-2310(01)00135-2

[5]- V. Pirouzpanah, S.M. Sami, Y. Ajabshirchy, The theoretical and experimental investigation fot eh effect of variation of thermodynamical parameters of egr fluid on NOx emissions in S-I engines. Int. J. Eng. 14 (3) (2001) 263-272.

[6]- P. Kastner-Klein, R. Berkowicz, R. Britter, The influence of street architecture on flow and dispersion in street canyons. Meteo. Atmos. Phys. 87(1-3) (2004) 121–131. doi:10.1007/s00703-003-0065-4

[7]- B. Blocken, J. Carmeliet, Pedestrian Wind Environment around Buildings: Literature Review and Practical Examples. J. Build. Physics. 28(2) (2004) 107-159. doi:10.1177/1097196304044396

[8]- X. Xiaomin, H. Zhen, W. Jiasong, The impact of urban street layout on local atmospheric environment. J. Build Environ. 41(10) (2006) 1352–1363. doi:10.1016/j.buildenv.2005.05.028

[9]- F. Ali-Toudert, H. Mayer, Numerical study on the effects of aspect ratio and orientation of an urban street canyon

on outdoor thermal comfort in hot and dry climate. J. Build. Environ. 41(2) (2007) 94-108. doi: 10.1016/j.buildenv.2005.01.013

[10]- N. Shishegar, Street Design and Urban Microclimate: Analyzing the Effects of Street Geometry and Orientation on Airflow and Solar Access in Urban Canyons. J. Clean. Ener. Technol. 1(1) (2013) 52-56. doi: 10.7763/jocet.2013.v1.13

[11]- J. Allegrini, A wind tunnel study on three-dimensional buoyant flows in street canyons with different roof shapes and building lengths. J. Build and Environ. 143 (2018) 71–88. doi:10.1016/j.buildenv.2018.06.056

[12]- S. Tabejamaat, Numerical Study of Reduction of NOx Emmission by High Temperature Air Combustion Technology, Int. J. Eng. 16(3) (2003) 301-310.

[13]- S. Vardoulakis, B.E.A. Fisher, K. Pericleous, N.G. Gonzalez-Flescac, Modelling air qualityin street canyons: a review. Atmos Environ. 37(2) (2003) 155–182. doi:10.1016/S1352-2310(02)00857-9

[14]- J.J. Kim, J.J. Baik, A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k-ε turbulence model. Atmos Environ. 38(19) (2004) 3039–3048. doi:10.1016/j.atmosenv.2004.02.047

[15]- X.X. Li, C-H. Liu, D.Y.C. Leung, K.M. Lam, Recent progress in CFD modelling of wind field and pollutant transport in street canyons. Atmos Environ. 40(29) (2006) 5640 – 5658. doi:10.1016/j.atmosenv.2006.04.055

[16]- N. Zhang, W. Jiang, S. Miao, A large eddy simulation on the effect of buildings on urban flows. Win. Struc. 9(1) (2006) 23–35. doi: 10.12989/was.2006.9.1.023

[17]- G. Najafpour, S. Mehdizadeh, M. Asadi, Environmental impact of fossil fuel utilization in neka thermal power plant, Int. J. Eng. 23(2) (2010) 115-120.

[18]- M. Scungio, F. Arpino, L. Stabile, G. Buonanno, Numerical Simulation of Ultrafine Particle Dispersion in Urban Street Canyons with the Spalart-Allmaras Turbulence Model. Aeros. Air Qual. Res. 13(5) (2013) 1423–1437. doi: 10.4209/aaqr.2012.11.0306

[19]- A.T. Chan, E.S.P. So, S.C. Samad, Strategic guidelines for street canyon geometry to achieve sustainable street air quality. J. Atmos. Environm. 35 (24) (2001) 4089-4098. doi: 10.1016/S1352-2310(01)00212-6

[20]- A.T. Chan, W.T.W. Au, E.S.P. So, Strategic guidelines for street canyon geometry to achieve sustainable street air quality-part II: multiple canopies and canyons. Atmos. Environm. 37(20) (2003) 2761–2772. doi: 10.1016/S1352-2310(03)00252-8

[21]- B. Medjahed, M. Abidat, J.F. Sini, Comparison between numerical models and CHENSI with experimental data (MUST) within the case of the 0° approach flow. MATEC Web of Conf. 11 (2014) 02-05. doi:10.1051/matecconf/20141102005

[22]- B. Medjaheda, M. Bouzit, Numerical Study of the Dispersion of the pollutant (CH4) in a canopy Urban. J Mater Environ. Sci. 7 (12) (2016) 4767-4777.

[23]- B. Medjahed, Numerical Study of the Dispersion of a Passive Scalar in Simplified Urban, Ph.D. Thesis, University of Mostaganem, 2016.

[24]- R.J. Stuart, Three-dimensional characteristics of coherent flow structure in a turbulent boundary layer over a rough surface, Ph.D. Thesis, University College of London, 1984.

[25]- C.H. Chang, N.R. Meroney, Numerical and physical modelling of bluff body flow and dispersion in urban street canyons. J. Wind. Eng. Indus. Aerodyn. 89(14-15) (2001) 1325-1334. doi:10.1016/S0167-6105(01)00129-5