Available online at www.CivileJournal.org

Civil Engineering Journal

Vol. 2, No. 3, March, 2016

Numerical Study of Flow Pattern in Buildings with Different Heights

Ali Hooshmand Aini [a*]

Department of Civil Engineering, Ayandegan Institute of Higher Education, Tonekabon, Iran

[a] Received 17 March 2016; Accepted 8 April 2016

Abstract

Understanding the flow pattern around the building, results in an accurate analysis of structure performance. Furthermore, having a proper configuration of the buildings next to each other we can provide a situation in which the buildings use the wind to make the air movement and natural ventilation. In this paper we use the FLUENT software to verify numerical flow pattern in buildings with different heights, and the results are provided in the form of distribution of velocities, velocity in Y direction, flow patterns and counters of turbulent.

Keywords: Numerical Method; Configuration of Buildings; Flow Pattern; Computational Fluid Dynamics.

1. Introduction

Comfort has been one of the most important needs of man in all his life. This comfort has been made or forbidden from various ways for man. But having comfort in structures is the most important issue in this relation. One of the major factors considered for this, is an atmosphere of comfort that is available by creating a suitable environment in terms of temperature and ventilation. Creating air flow in the living area will have an important role in providing comfort. On the other hand wind flow can be named as one of the main factors on the dispersal of pollutants in urban environments and among the buildings. However it is an obvious issue that pollutants have some effect on the wind flow. In fact, urban areas have row buildings with different heights which streets have separated them from each other and cars on the street are the main generators of pollutants in these areas. Various parameters like buildings and local users influence on the amount of their distribution and wind flow [1, 2].

For the first time Lipman, published an article in 1952 about the turbulent flow of wind, he mentioned near the earth’s surface. These models Later in 1961, by Vlazy and Cohen was diagnosed with a complete and correct method of Davenport that the fluctuations pressure on the surface of the wind in some way depends on the structure function the way back to the wind (Figure 1).

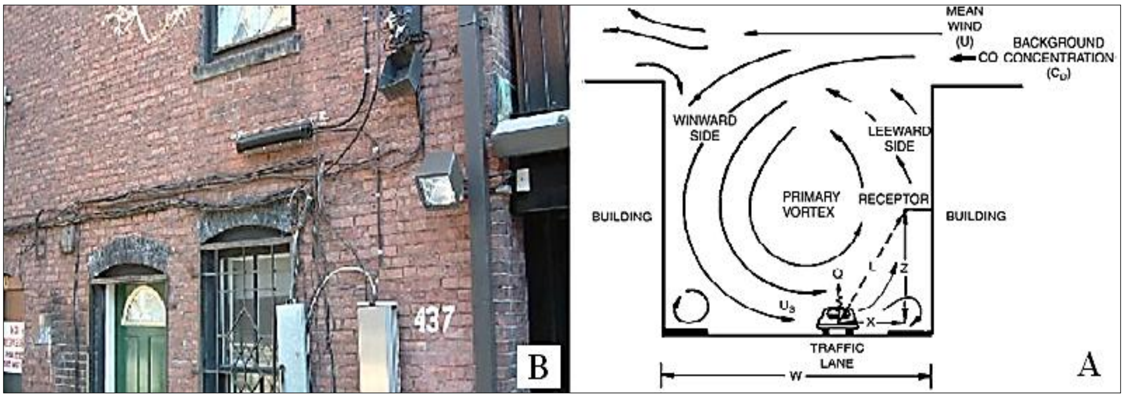

Figure 1. A) Pollution incidence in street regular canyon B) Smoke and pollution accumulation of wall of houses

* Corresponding author: Ali_hooshmand1983@yahoo.com

In Figures 2 to 4 that are achieved by the examinations of researchers, it shows that a structure with high volume and height against the wind flow can have an undeniable role in blocking flow, slope, conduction, veer, or reducing its speed. Especially when the structure is exactly against the wind it leads to creating a phenomenon called stagnation in back of the structure which pollutants generated from sources inside the space is focused for a while in this area that It can also lead to dangerous conditions. In such a situation the use of elements like pilot, given the openness of the input and output it can somewhat works as square ventilation. Shape and details of the structure like indentation, extrusion, and elements related to the façade and examples like that is involved in wind blowing [3].

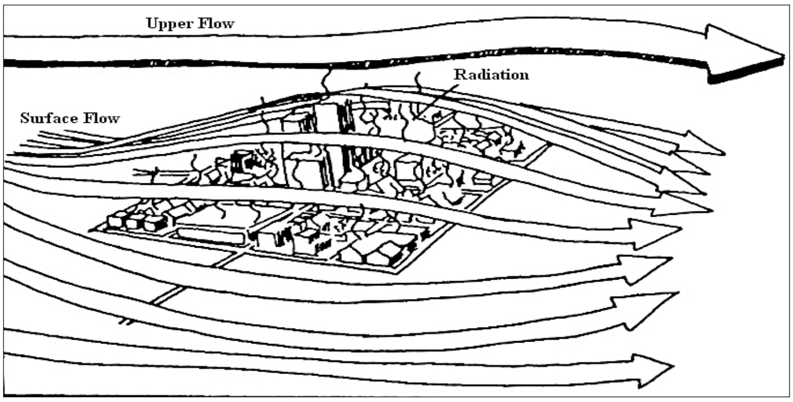

Figure 2. Surface wind flow and the City form

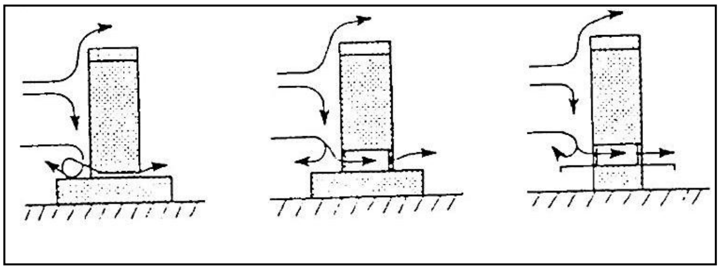

Figure 3. The impact of the pilot in the air flow

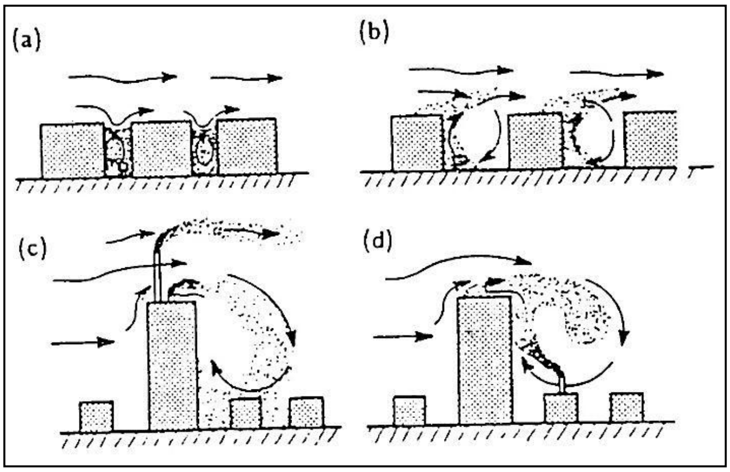

Figure 4. The impact of the buildings combination on air flow

The issues that impact on the amount of the proposed effects can be noted the height of the buildings. In fact there are tall buildings in front of the other buildings causing the accumulation of pollutants in the surrounding atmosphere and act like an impermeable spirit against the wind flow. Wind contact with the structure the important phenomenon is formation of turbulent flows. These flows even in the ideal case that the structure is concurrent don't appear to be concurrent. Turbulent flows are randomly generated. Study of turbulent flows by relations theory would be very difficult [4].

Tall buildings will divert strong winds and will redirect them to the ground that as result of this situation, unpleasant and dangerous conditions cause for pedestrians and the pedestrian must use more energy for walking. Increase in wind speed of 15 meters per second cause creating unstable situation conditions for the pedestrian. Furthermore when the wind speed is high it is possible to observe the accumulation of fumes made from vehicles and other pollutants in the pedestrian part. Due to this cause during the process of designing tall buildings, considering the wind flow and flow pattern around the building is important.

2. Flow Pattern around the Building

The air flow pattern in the concurrent direction of the wind around the building is shown in Figure 4. Two-thirds of the height of the floor is called stagnation point which from that point to the top the air flow goes upwards and passes the roof of the building. Below this point the air flow is driven downwards which leads to the formation of a vortex in front of the building and finally passes around the building. In the direction of behind the wind we observe the large flow separation and re-rounded (Figure 5). The air flow passes through the corners of the building, both horizontal and vertical angular momentum accelerate. Configuration of the buildings beside each other and the appearance of structure would also create turbulent currents and complex aerodynamic problems. These factors have a major effect on making the structural analysis be more complex (Figure 6) [5].

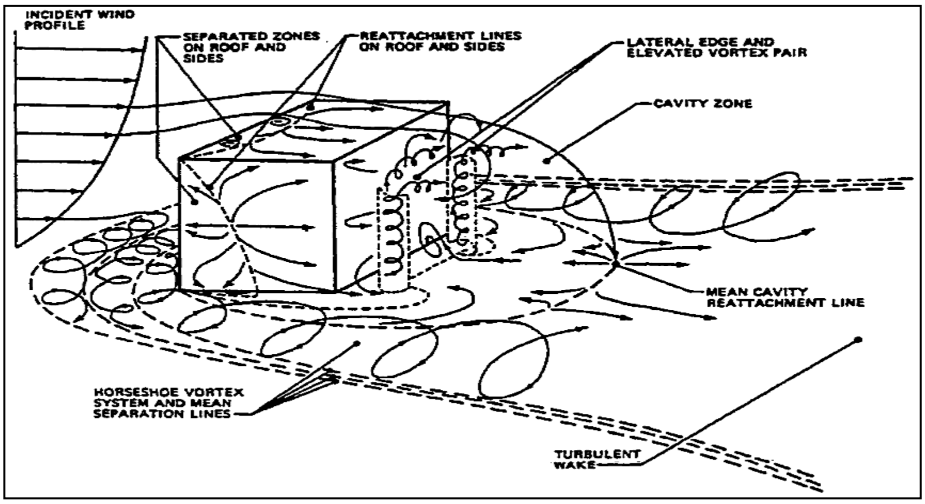

Figure 5. Flow pattern obtained around the building with the sharp edge (Hosker 1979)

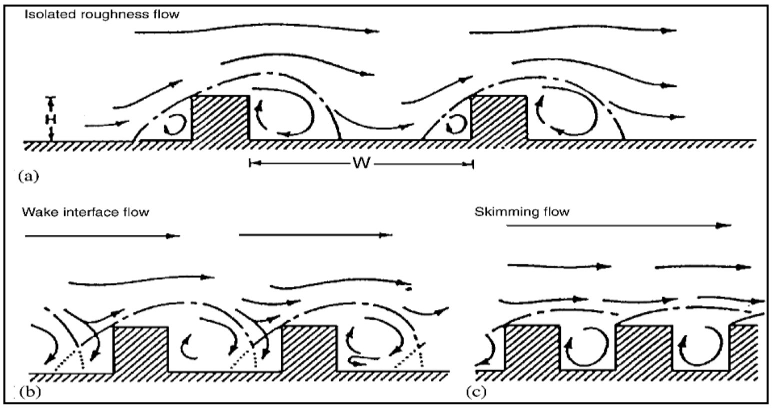

Figure 6. Typical flow pattern among the buildings of various sizes (Oke, 1988) [6]

3. Numerical Modeling

Today, in addition to wind tunnel that is typically used to simulate structures, CFD (Computational Fluid Dynamic) has also become a very powerful tool to predict the flow patterns around the structures. All flows that are considered in practical engineering including simple cases such as the two-dimensional jets, wakes, boundary layer flow tubes or very complex three - dimensional cases are the type of upward unstable certain Reynolds number. At low Reynolds numbers below the flow is slow. High Reynolds numbers can be observed that the flow will go turbulent. In simple cases, the continuity equation (Equation 1) and Navier – Stokes can be solved through numerical method with CFD methods which is similar to finite volume method [7].

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \qquad (1)$$

In this paper, we have used FLUENT software that is based on finite volume method. We have also chosen K-ε as the turbulent model. The K-ε model is fairly complete and general but very expensive that is used to describe the turbulence and is also useful to express transport properties of turbulence by the mean flow and turbulence penetration and for production and depreciation turbulence. In this model, the two transport equation (partial differential equation PDE), one is resolved for the turbulent kinetic energy k and the other one for the turbulent kinetic energy dissipation rate ε. The standard k-ε model uses the transport equations that are practiced ε and K in FLUENT software:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[(\alpha+\frac{\alpha_t}{\sigma_k})\frac{\partial}{\partial x_j})\right]+G_k+G_b \tag{2}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_i}\left[(\alpha+\frac{\alpha_t}{\sigma_k})\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2s}\rho\frac{\varepsilon^2}{k} \tag{3}$$

$$k=\frac{1}{2}(\overline{u^2}+\overline{v^2}+\overline{w^2}) \tag{4}$$

K= Kinetic energy (per unit mass) related to the turbulence
α_t =Turbulent viscosity

Equations contain five adjustable constants $C_{2\epsilon}, C_{1\epsilon}, \sigma_{\epsilon}, \sigma_k, C_\mu$ whose values are:

$C_{2\epsilon}=1.92, C_{1\epsilon}=1.44, \sigma_{\epsilon}=1.30, \sigma_k=1.0, C_\mu=0.09$

G_k=Production term of turbulent kinetic energy due to mean velocity gradient
G_b=Production term of turbulent kinetic energy due to the Buoyancy force [8, 9].

To evaluate the effect of building height difference on the pattern flow, two states are assumed. In the first case a building which has a height-to-width ratio of 1.5 to 1 in front of the wind flow with the speed of 5 meters per second is considered. Height-to-width ratio is 1 to 1 for all other buildings. In the latter case the building with the higher height is placed at the end of the model. As we observe in figures 7 and 8 due to the flow conditions the boundary conditions is used in terms of input speed conditions and output pressure conditions. By solving the flow the results are shown in Figures 9 to 14.

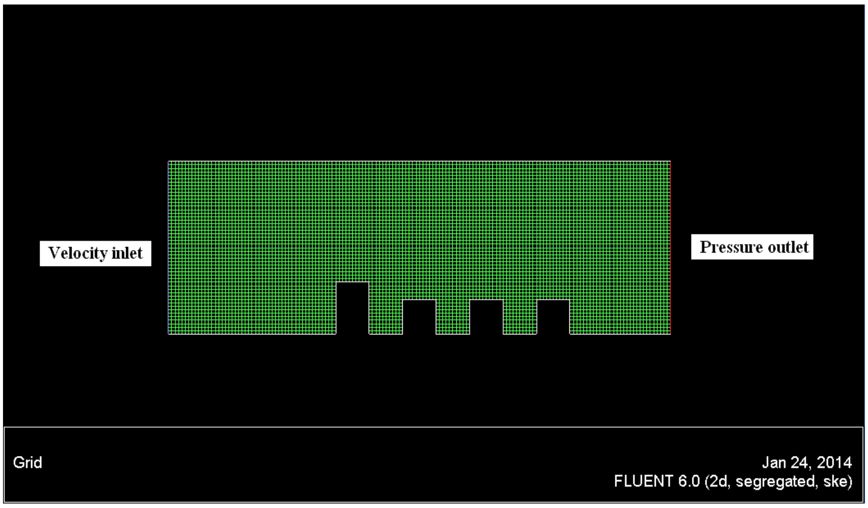

Figure 7. Geometric model, meshed and boundary conditions done were imposed in the case 1.

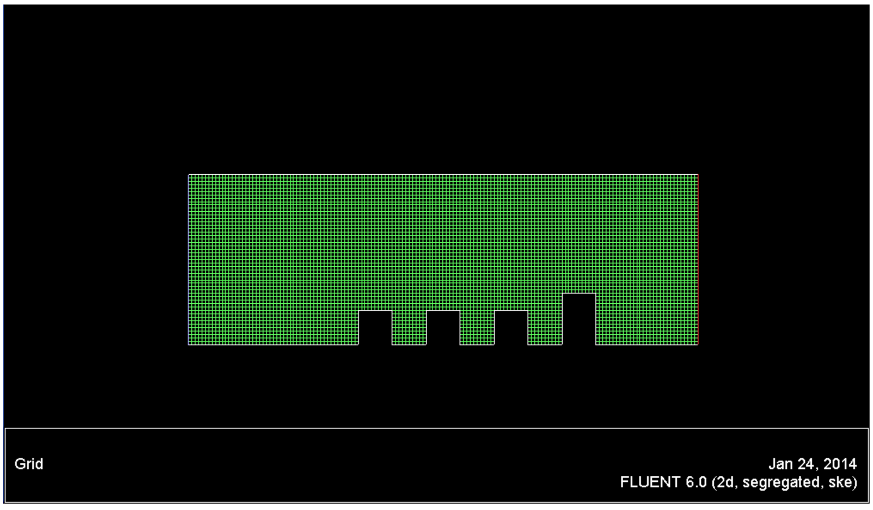

Figure 8. Geometric model, meshed and boundary conditions done were imposed in the case 2.

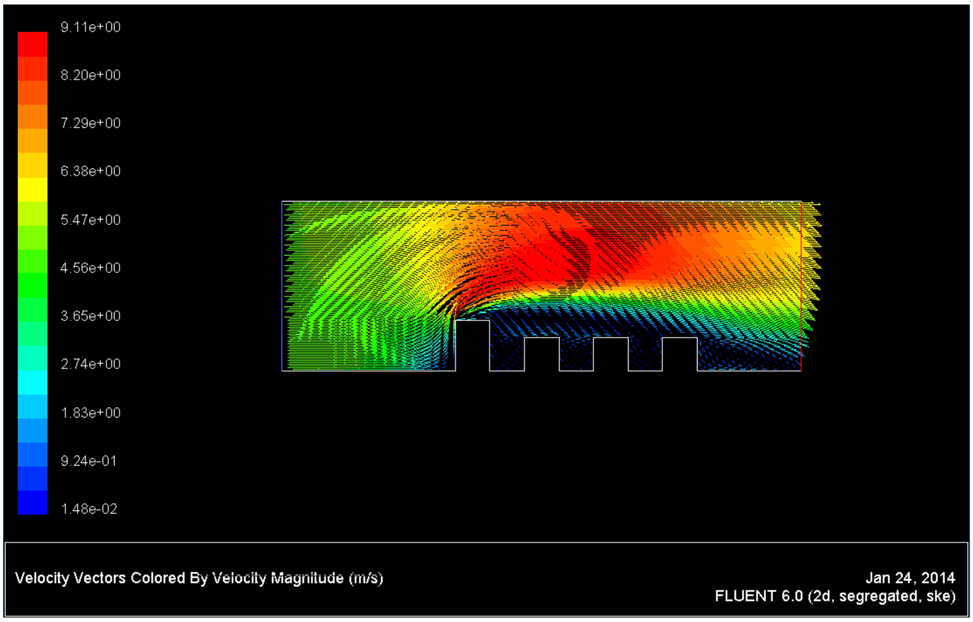

Figure 9. Display velocity vectors in Case 1.

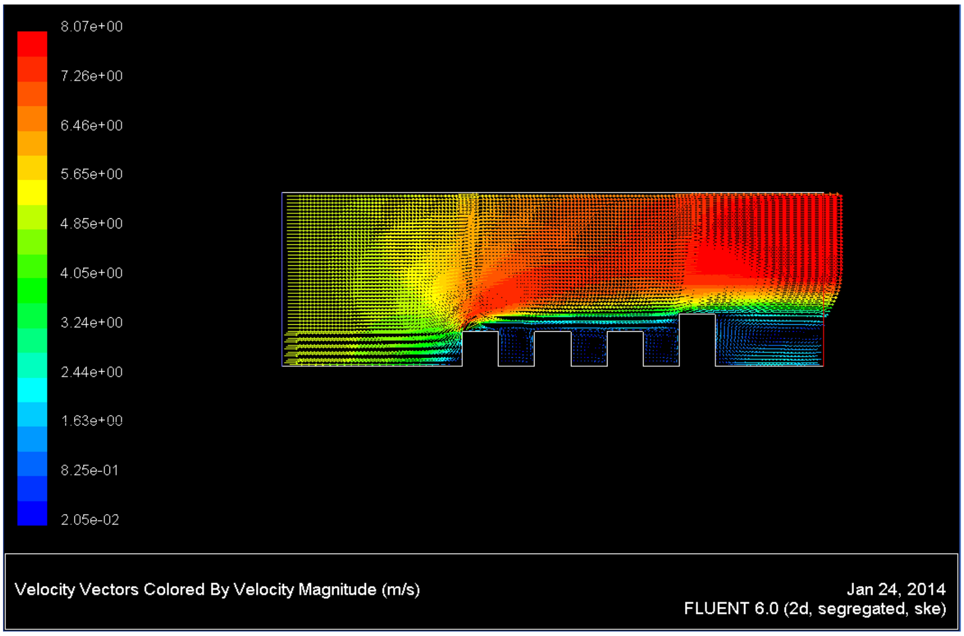

Figure 10. Display velocity vectors in Case 2.

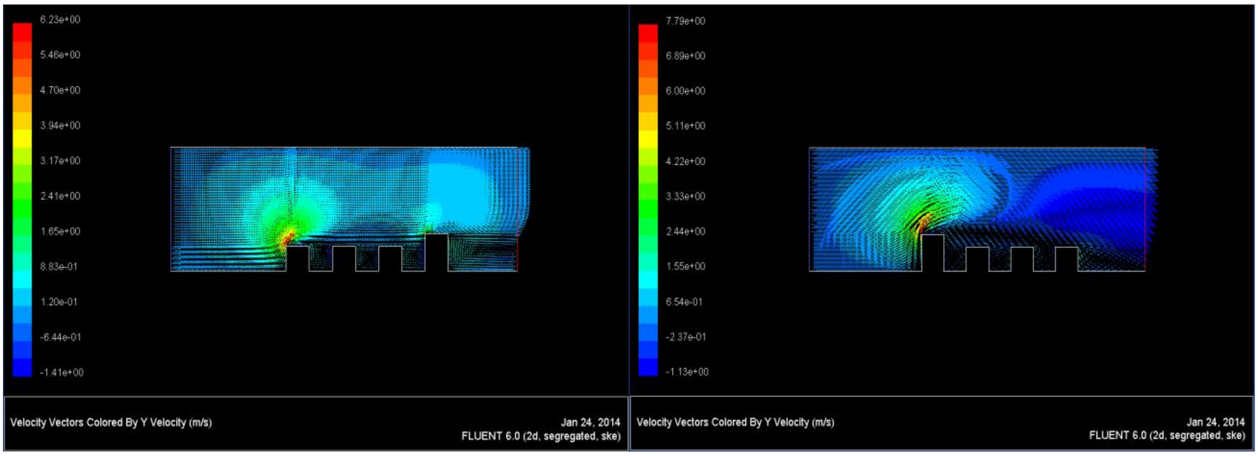

Figure 11. Display velocity vectors in the Y direction in Case 1 and 2.

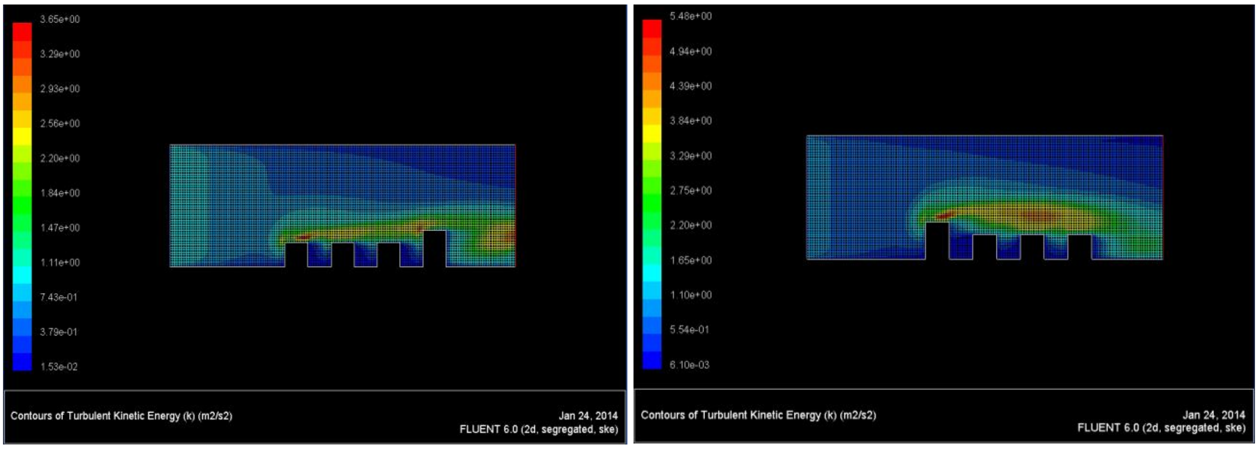

Figure 12. Display Counters of Turbulent in Case 1 and 2.

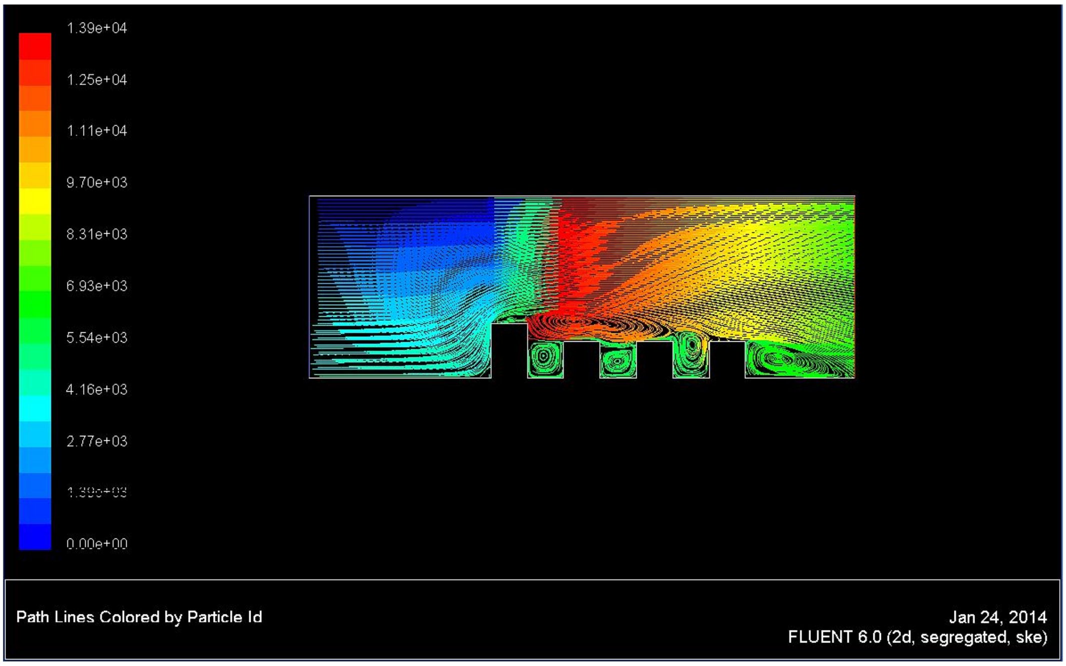

Figure 13. Display the flow pattern in Case 1.

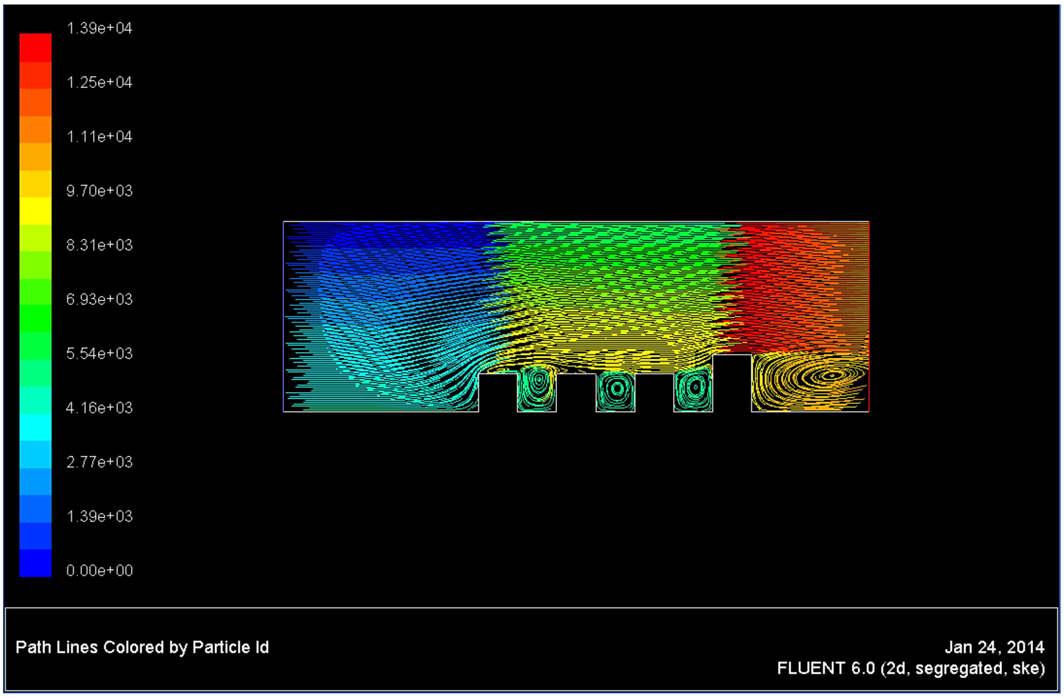

Figure 14. Display the flow pattern in Case 2.

4. Conclusion

According to the figures and results obtained in this study it can be concluded that maximum velocity vectors occurred at the top of the building (Figures 9 and 10). The maximum speed is created in the mode 1 (Figure 9). The maximum speed in the Y direction (Where the flow has suddenly changed its direction) is also occurred at the top of the first building in front of the wind (Figure 12).

Furthermore according to the flow pattern obtained (Figures 13 and 14) in case 2 tornados formed between the buildings are all in the clockwise direction but in case 1 between the first and second building that are in front of the wind the formed tornado is in the opposite of the clockwise direction. This is due to the impact of the upper flow on the Vortex flow between the buildings. Moreover in case 1 because of existing a taller building in front of the flow in addition to vortex created in case 2, we observe a tornado on the back of the building. Naturally, the dimensions of this tornado is directly related to building height and wind speed and as we see in wind blowing case at a speed of 5 meters per second, this tornado has gone through till the end of the roof of the third building. So we can conclude that the tornados direction created between the buildings is entirely dependent on the height of buildings. According to the obtained values and shapes, we can conclude that the FLUENT software has a strong ability in flow modelling around the buildings and by using this software the parameters related to the flow can be obtained desirably.

5. References

[1] Mahmoodi, Mahnaz and Pourmosa, Mahbobe,‘’Potential valuating of wind energy and its foundamental role in suitable ventilation and moisture removal, Case Study: Rasht City (Golsar Area), ‘’ Armanshahr, (2010): 147-156.

[2] Assimakopoulos, V. D., H. M. ApSimon, and N. Moussiopoulos. "A numerical study of atmospheric pollutant dispersion in different two-dimensional street canyon configurations." Atmospheric Environment 37, no. 29 (2003): 4037-4049.

[3] Bahraini, Hosain and Arefi, Mahyar, Barakpour, Naser, Khoshpoor, Hasan, ‘’ Meteorological studies of air pollution in urban design: a special case of Tehran, "Ecology , No. 18, (1997).

[4] The impact of the wind on structures,(1378), first printing, Mahamood Yahyaee, Nasir al-Din Tusi University Press, Tehran.

[5] Ye Li, “Numerical Evaluation of Wind-Induced Dispersion of Pollutants around Building,” A Thesis in the Department of Building, Civil and Environmental Engineering, Concordia University, Montreal, Quebec, Canada, October, (1998).

[6] Xie, Xiaomin, Zhen Huang, and Jia-song Wang. "Impact of building configuration on air quality in street canyon." Atmospheric Environment 39, no. 25 (2005): 4519-4530.

[7] Wolfgang, R. "Turbulence Models and their Application in Hydraulic: a state-of-the art review." International Association for Hydraulic Research, Delft (1984): 22-24.

[8] Soltani, Majid and Rahimi, Rohollah, Computational Fluid Dynamics by using FLUENT software, Publication Designer, (2006): 446 pages.

[9] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.