Indian Journal of Fundamental and Applied Life Sciences ISSN: 2231– 6345 (Online)
An Open Access, Online International Journal Available at www.cibtech.org/sp.ed/jls/2014/03/jls.htm
2014 Vol. 4 (S3), pp. 1664-1668/Aini and Sendesi
Research Article

INVESTIGATION OF FLOW PATTERN BETWEEN GABLE ROOF BUILDINGS BY USING FINITE VOLUME METHOD

***Ali Hooshmand Aini[1] and Sayyed Amin Teimouri Sendesi[2]**
[1]Department of Civil Engineering, Roudbar Branch, Islamic Azad University, Roudbar, Iran
[2]Department of Mechanical Engineering, Roudbar Branch, Islamic Azad University, Roudbar, Iran
**Author for Correspondence*

ABSTRACT
Calculating the wind load on structures is a very important issue in designing the structures today. Discovering the flow pattern around the building, despite causing a correct analysis about the performance of the buildings, it can also play an important role with issues related to energy efficiency. In this paper, we have examined the flow pattern between buildings with gable roof which are mostly used in the northern areas of Iran, by using the FLUENT software. And, we have provided the results in the velocity distribution, flow pattern.

Keywords: *Gable Roofs, Computational Fluid Dynamics, FLUENT*

INTRODUCTION
Roof is a part of the structure that protects us against atmospheric factors such as rain, snow, heat and sunlight. The roofs vary in different parts of the world and the leading cause of these differences can be found in the existing materials as well as area's weather conditions. We can see some examples of common shapes of the roof in Figure 1.A. The roof has a long history in the world architecture, this can be confirmed by the surviving buildings from ancient times (Figure 1.B).

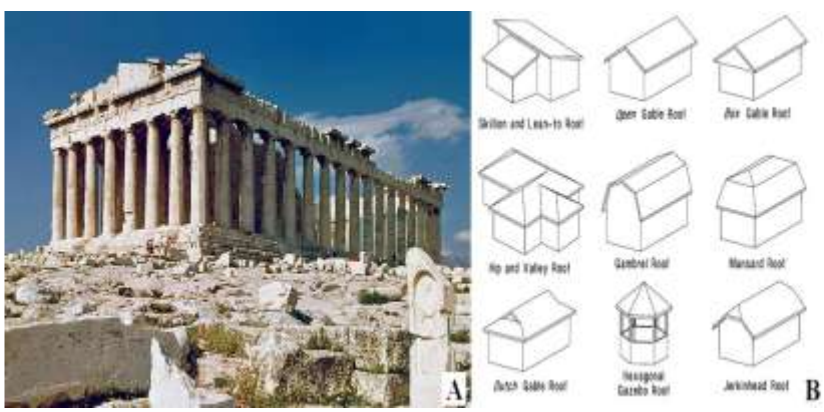

Figure 1: A) Parthenon Temple in Acropolis, Athens B) Different Types of Common Roofs

Roof or roofing is also part of the living space, in traditional Iranian architecture and despite of the existence of complex and beautiful volumes, it has been used as a yard. In some buildings in cities such as Nain, around the roof has been heightened about a meter and a half by walls to create a yard that has been used for sleep on summer nights. These walls also had a secondary role in climate by overcastting on some parts of the roof at different hours of day. Such places are also used in mosques. Above the roof of Sepahsalar mosque is a Vuzukhaneh which is named Chehelshir, and the space of this vuzukhaneh is the same.The other example is the Abbasian house in Kashan, which has the same yard on its roof where the architects have used delicate framing of reticular brick, to provide it's ventilation. The other application of the roof in Iranian architecture is building funnel on it for ventilating the house (Figure 2.A). Indeed, the funnel is an example of Iranian engineering masterpiece.(Yahyayi, 2000) Another amazing example of roof in Iranian architecture can be found in the historical city of Masuleh, in which the yard of the above building is the roof of the beneath building (Figure 2.B).

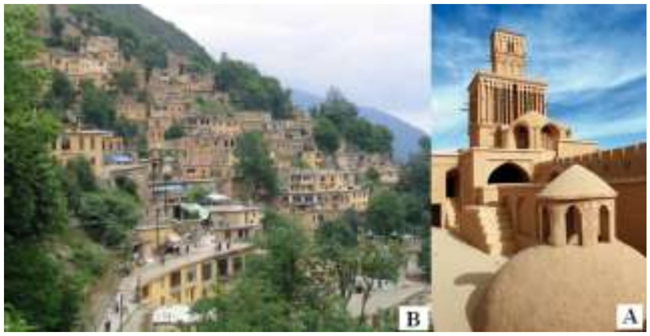

Figure 2: A) A Model Of An Iranian Funnel B) A View From Historical City Of Masuleh

Flow Pattern around the Building

The pattern of wind flow around the building as you can see on figure 3 is in favored of wind. Two-thirds of the height of the floor is called the stillness that from that point upwards, the air flow upward and passes through the roof of the building. Below this point, the air flow downwards, this is driven to cause vortexes in front of the building and at last passes around the building. On the other hand if the flow is on the opposite direction of the wind, we will see high flow of separation (Figure 3). The air flow accelerates when it passes through the horizontal or vertical angular corners of the building. Appearance of buildings and structures, as well as their exposure near each other cause turbulent flows and complex aerodynamic problems, that these situations have an important role in structure analysis difficulty.

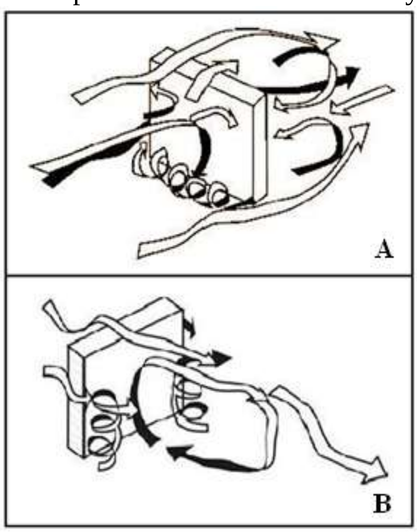

Figure 3: A) Airflow pattern in favor of wind B) Airflow pattern up wind (Baskaran and Kashef, 1996)

Over all, when a structure is under aerodynamic loads, two major forces will apply to it:

1. Tensile force that is created along the wind direction. This force is caused due to the pressure and suction on top and rear faces of wind in structure.
2. The Lift force, this force effects on the structure along the perpendicular wind flow which is arises due to the vortex phenomenon behind the structure.

The important thing about the wind blowing structure is the formation of turbulent flows. These flows are not symmetrical, even in ideal cases that the structures are symmetrical. Turbulent flows are generated randomly. The study of turbulent flows by theoretical relations is a very difficult task. Planning and evaluation of wind effects on high buildings takes place in following general contents:

1. The general effect of wind on structure
2. The study of local effects of wind

Considering the above mentioned issues, the scientists decided to study the effects of this natural phenomenon on the structure. For the first time, in 1952 Lipman published an article in which he noted the effects of the turbulent flow of air near the earth's surface. For studying and displaying these effects, he also provided models. By supplementing and amending the Davenport method later in 1961,Vlazy and Cohen diagnosed that the wind pressure fluctuations on the top surface of wind depends on the performance of the building on rear face of wind (Bungale, 2005; Paterson and Apelt,1986).

Numerical Modeling using the Fluent Software

Today, calculating the wind load on structures is a very important issue in designing them. In most developed countries there are regulations for calculating this force that the design and building engineers are required to run them. The important thing about the wind blowing structure is the formation of turbulent flows. Examining the turbulent flow by theoretical relationship is very difficult. Today, in addition to the wind tunnel which is typically used for structure simulation, CFD is also became a very powerful tool for predicting the flow pattern around the structure (Soltani and Rahimi, 2007)

All flows that are considered in practical engineering, ranging from simple instances, such as two-dimensional fountain, suites, boundary-layer flows in pipes or very complex three-dimensional instances which are higher than certain Reynolds number are unstable. At low Reynolds numbers, the flow is slow. At high Reynolds numbers, it is observed that the flow becomes turbulent. In simple cases of continuity equations (Equation-1) and the Navier-Stokes can be solved by numerical and CFD methods such as finite volume method.

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

In this paper, we have used FLUENT software which is based on finite volume method. We have also chosen k-ε as the turbulence model. k-ε is a relatively complete and universal model that is used for describing turbulence, but it is too expensive, It is also useful for expressing turbulence properties transport by mean flow, influence, production and dissipation of turbulence.

In this model, two transport equations (partial differential equation PDE) are dissolved, one for the turbulent kinetic energy k and the other for the dissipation rate of turbulent kinetic energy ε. The standard k-ε has used transport equations (2) and (3) in FLUENT software which have been used for k and ε:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\alpha + \frac{\alpha_t}{\sigma_k}\right)\frac{\partial}{\partial x_j}\right] + G_k + G_b \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_i}\left[\left(\alpha + \frac{\alpha_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \quad (2)$$

$$+ C_{1s}\frac{\varepsilon}{k}(G_k + C_{3s}G_b) - C_{2s}\rho\frac{\varepsilon^2}{k}$$

$$k = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \quad (3)$$

K=kinetic energy (per unit mass) of the turbulence

α_t=turbulent viscosity

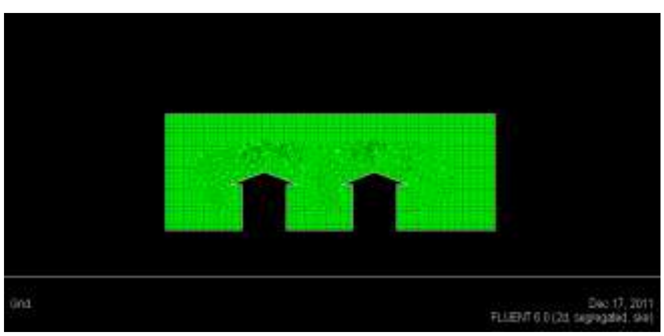

Figure 1: Geometric Model, Mesh And Boundary Conditions Applied In The Model

Indian Journal of Fundamental and Applied Life Sciences ISSN: 2231– 6345 (Online)
An Open Access, Online International Journal Available at www.cibtech.org/sp.ed/jls/2014/03/jls.htm
2014 Vol. 4 (S3), pp. 1664-1668/Aini and Sendesi
Research Article

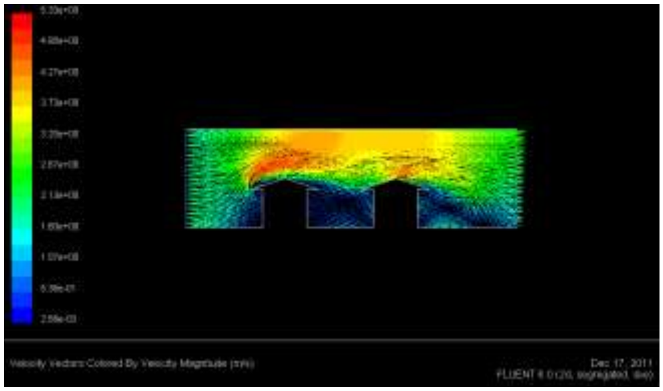

Figure 2: Presentation Of Velocity Vectors

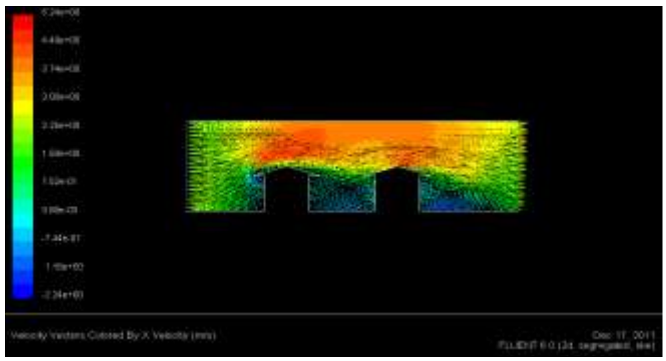

Figure 3: Presentation Of Velocity Vectors In X Direction

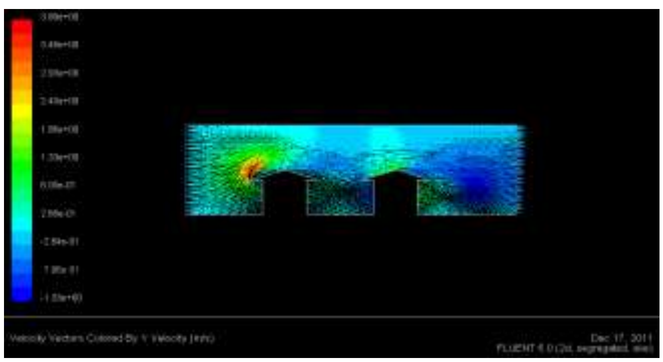

Figure 4: Presentation Of Velocity Vectors In Y Direction

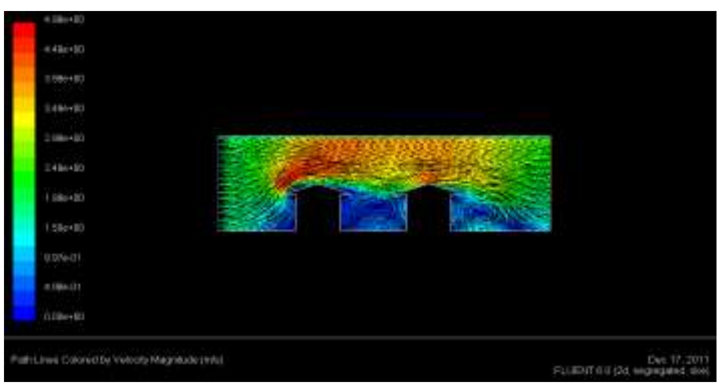

Figure 5: Presentation of Flow Pattern

Research Article

Equations contain five adjustable constants $C_{2\varepsilon}$, $C1\varepsilon$, $\sigma\varepsilon$, σk, $C\mu$ $_{\text{which}}$ values are:
σ_k=1.0 C_μ=0.09 $C_{1\varepsilon}$=1.44 $C_{2\varepsilon}$=1.92 σ_ε=1.30
G_k=Terms of turbulent kinetic energy due to the mean velocity gradient.
G_b=Terms of turbulent kinetic energy due to the buoyancy force (Shojaie *et al.,* 2007),
In order to solve this problem, a model has been built, which you can see in Figure 4. According to the terms of flowing, we have used boundary conditions as velocity inlet and pressure outlet. By solving the problem, the results are presented in Figure 5 to 8.

Conclusion

First, confirm that you have the correct in this research; finite volume method was used to solve the differential equations. According to the results that we have reached, we can say that the model is a good answer for stimulating the flow between buildings with gable roofs. According to Figure 5,when the flow blows the structure it has its maximum speed above the roof and also, due to the diversion of the flow in front of the building we can see the maximum velocity in the Y direction (Figure 3), and the greatest turbulent in this section. The flow pattern would be similar to what had been given in Figure 8. As we observe, we can see the formation of vortexes between two buildings. The results indicate the ability of the software in stimulating wind flow over structures in order to avoid undesirable and unpleasant conditions for pedestrians, preventing the incoming vibration on the structure and accumulation of pollution from vehicles and at last energy-related issues are optimized as well.

REFERENCES

Baskaran A and Kashef A (1996). Investigation of air flow around building using computational fluid dynamics techniques, *Engineering Structures* **18**(11) 861-875.
Bungale S (2005). Wind and Earthquake Resistant Buildings , Marcel Dekker, New York.
Paterson D and Apelt CJ (1986). Computer modelling of turbulent wind flows over buildings, 9thaustralasian fluid mechanics conference auckland.
Shojaie Fard M and Noorpour Hashtroudi A (2007). Introduction to Computational Fluid Dynamics, Iranian University of Science and Technology 131-139.
Soltani M and Rahimi Asl R (2007). Computational Fluid Dynamics using Fluent, Tarrah 84-92.
Yahyayi M (2000). *Effect of Wind on Structure,* First Edition (KN Toosi University Press).