

## Research Paper

# Air flow simulation through the skylight in the historical house of Mosavar-al-Molki

E. Nabizadeh Shahrehabak<sup>1</sup>, S. Golafshan<sup>2</sup>, M.R. Chamani<sup>3,\*</sup>

<sup>1</sup>Research Assistant, Department of Mechanical Engineering, University of Nevada-Las Vegas

<sup>2</sup>Lecturer, Department of Civil Engineering, Isfahan University of Technology

<sup>3</sup>Associate Professor, Department of Civil Engineering, Isfahan University of Technology

Received: 29 June 2015, Revised: 18 April 2016, Accepted: 1 May 2016, Available online: 29 June 2016

---

### Abstract

One of the main architectural characteristics in the central area of Iran is the relation between life and climatic conditions; e.g., the four-seasonal house which is a typical kind of the central-yard house. As the climate of Isfahan is more pleasant than other desert regions in the central part of Iran, wind towers are replaced with skylights to ventilate the pool area of hoze-khane. So far, several researches have been carried out to study the stack effect characteristics and to review wind tower performance. In this paper, a case study is carried out to investigate the effect of the skylight of Mosavar-al-Molki house on the flow pattern and natural ventilation. The performance of skylight on the internal air flow patterns as an effective tool in improving the natural ventilation is investigated by using the Ansys Fluent software. Other elements which have been discussed in this paper are the vault and the columns. These elements have a distinguished effect on distribution of inlet streams in different parts of the hoze-khane. Analysis of the simulation results were obtained for wind velocities of 2.5 m/s and 8 m/s. The numerical results show that wind speed of 2.5 m/s has better performance than wind speed of 8 m/s in terms of ventilation.

**Keywords:** Skylight, Natural ventilation, Flow pattern, Simulation, Ansys fluent software.

---

## 1. INTRODUCTION

Thermal comfort is usually defined as a range of temperature and humidity in which mechanisms of body temperature regulation have their least activity. Creating thermal comfort in open spaces is not possible unless a thorough understanding of the factors affecting the thermal condition of these spaces is known. These factors change their surrounding by intensifying or weakening the climatic factors at micro-scale [1]. Recently, a rising trend has been witnessed in the use of ventilation systems in buildings to enhance the efficiency of heating and cooling systems. Ventilation systems are classified into the three types of active mechanical systems, passive natural systems, and hybrid ones, the latter being a combination of the first two [2]. Natural ventilation systems in buildings can be classified into the two categories of stack-ventilation and cross-ventilation systems. Natural ventilation systems are based on the temperature buoyancy which is called the 'stack effect', and the pressure difference which creates a wind-driven force that ventilates a building. The adequacy of a natural ventilation system for the peak ventilation of a specific building must be determined through an investigation

of the air flow patterns. Such an investigation also helps to evaluate the thermal comfort provided in each space within the building. It must be noted that the air flow is the most effective parameter involved in creating thermal comfort for the dwellers and it is the difference between indoor and outdoor temperatures that causes an even distribution of heat in the different spaces inside the building. For negligible outdoor/indoor differences in temperature, it is only the air flow that gives rise to better perspiration of the residents, and thereby, their feelings of thermal comfort [3]. The present study aims to investigate the airflow patterns through a historical building and its effects on ventilation through different places inside the building.

## 2. RESEARCH METHODOLOGY

A major concern for modern architects since the emergence of the non-renewable energy crisis in the 1970s is the design of environment-friendly buildings. The growing concern has heightened in recent years to the extent that sustainable architecture is now based on fluid dynamic diagrams as a major requirement for justifying every construction project. These diagrams are used in predicting the consumption of non-renewable energies in buildings.

To derive environment-friendly design principles for

---

\* Corresponding author: mchamani@cc.iut.ac.ir  
Tel: 031-33913831 ; Fax: 031-33912700

contemporary buildings, the present study takes a climatologic approach to the quantitative and qualitative analysis of the performance of skylights in historic buildings. From among the many instances, the skylight at Mossavor-ul-Molki's house is selected for the purposes of this study as one of the best examples. Using numerical methods and computer simulations in the Ansys Fluent [4] environment, the functions of both the skylight and structural elements such as columns, vaults, and the partitioning walls in providing natural ventilation are studied. In fluid dynamic simulation, partial differential equations of fluid flow and heat transfer systems are approximated by finite difference methods of which, finite volume difference is one example. In these methods, partial equations are solved by transforming the problem environment from a continuous to a discrete state.

### **3. RESEARCH BACKGROUND**

Ohba et al. [5] used experiments and numerical simulations to investigate the factors involved in air flow patterns in cross-ventilation systems. They found that, despite the reduced discharge from sashes, the inflow into the building would increase with the impact angle between the wind and the building inlet from 40 to 60 degrees. They also showed that the inflow air tends to flow toward the floor while large clockwise winds blow toward the entrance sash throughout the upper space in the room.

Van Moesike et al. [6] used numerical analysis to study the airflow patterns around buildings. They showed that changing the distance between two adjacent buildings from 3 to 1.5 times their height has a significant effect on the airflow around the buildings. They also compared pressure distribution in the three open, suburban, and urban spaces as a result of changing the wind impact angle. They concluded that wind impact angle has little effect on ventilation as measured by ACH criterion, but this effect was significant in open or in suburban areas.

Evola and Popov [7] studied airflow streamlines in the three models of single-sided windward ventilation, single-sided leeward ventilation, and two-sided ventilation, all of which are subtypes of the cross-ventilation system. They compared simulation results and experimental measurements to find out that, when the windward sash was open in the single-sided natural ventilation system, a perceptible circulation zone was created that blew throughout the whole room space. They also observed that a similar circulation zone was created in the single-sided natural ventilation system when the windward sash was open, except that air velocity at the bottom was increased considerably. In this situation, the center of the circulation current moved from the upper to the lower part of the room space. In a cross-ventilation system in which the inlet sash is as wide as the outlet sash, the outlet air velocity in the upper half of the outlet sash significantly reduced when compared to that in the lower half. They finally realized that the  $k-\varepsilon$  RNG model yielded satisfactory results for natural ventilation systems.

Cheung and Liu [8] carried out both experiments and numerical simulations to investigate the effect of building spacing on improving ventilation rate and the indoor and outdoor airflow lines during cross ventilation. For residential buildings with square plans, they found that the pressure difference on the two sides of the building would increase with increasing inter-building distances beyond 5 times their floor plan dimensions. Clearly, increasing pressure difference has a great impact on creating natural ventilation. Experimental results revealed that air tends to flow toward the floor surface of the building after it enters the building to exit through the outlet sash. Simulation results, however, showed that the airflow would pave a direct, non-diverging path between the two sashes.

Bu and Kato [9] employed a wind tunnel and used numerical simulation to study the indoor and outdoor airflow patterns in a building that had one basement. The basement had a single-sided ventilation system through an annexed front yard. Results showed that the current through the inlet sash to the yard was a function of the angle of the wind impacting the building. The authors used the two turbulent models of Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS). Comparison of the results revealed that LES results fitted better than those of the RANS model due to the large inflow current.

Ngyuen and Reiter [10] simulated six common roof configurations to investigate the effect of roof configuration on natural ventilation. They verified the simulation results with experimental measurements obtained from wind tunnel studies. Comparison of the values obtained for air flow rate, air velocity, and wind distribution in the buildings indicated that roof configuration has an effect of only 4% on natural ventilation parameters while roof height has a great effect on ventilation. They concluded that air flow rate increases with reducing roof height.

### **4. NATURAL VENTILATION AND DOMESTIC ARCHITECTURE IN CENTRAL IRAN**

Four-season residences based on an inward physical pattern are typical constructions in central Iran. The yard in these houses are shaded by their lateral walls to create a desirable and pleasant environment with obvious effects on thermal comfort inside the building. The northern front in these houses is more important in winters due to the collection of solar energy from the yard; this front is called the Sun-oriented, or southerly side and is used as a winter residence. In the summer, the southern front which receives the northern light and is shaded most of the times provides more desirable conditions. This side of the building is called the northerly or shaded side and is used for summertime dwelling. One environmentally creative aspect of residences in central Iran is the skylight and wind tower designs on top of the building. These elements not only strengthen natural ventilation within the interior spaces but play a major role in creating an authentic urban view in these regions as well. These architectural elements

are often located in one corner of the central yard on top of the pool courtyard. They enhance the ventilation effect of the evaporative cool current over the pond on the thermal comfort within the residential spaces.

In what follows, we attempt a close examination of the airflow patterns in the historic house of Mossavor-ul-Molki located in the Mahalle Nou District in Isfahan. Fig. 1 presents a 3D schematic plan of the house which

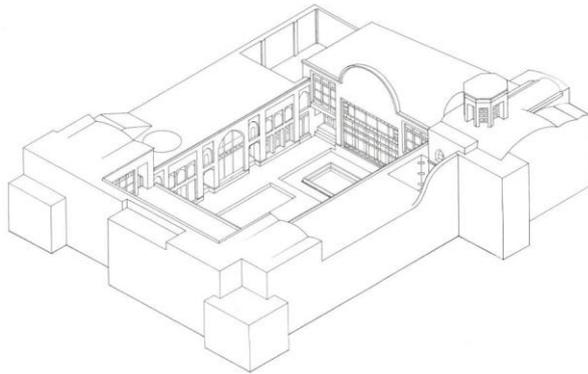


Fig. 1 A 3D plan of Mossavor-ul-Molki's House [11]

includes a general view of the house along with the locations of the skylight and other structural elements. Wind towers and skylights may be considered to be essentially the same with almost similar functions. However, they have major differences in their geometries and structures (Figs. 2 and 3), which lead to differences in the wind velocity within the building and, consequently, in the airflow patterns.



Fig. 2 Wind tower in Aghazadeh Abarghu House



Fig. 3 The Skylight in Mossavor-ul-Molki's House

## 5. STUDY BUILDING LOCATION AND SPECIFICATIONS

In this study, the air flow pattern in the pool courtyard of Mossavor-ul-Molki's house is investigated. The house is architecturally an inward one in which the yard is located in the center of the building plan (part 1 in Fig. 4). In the northern porch of the house, there is a master guest hall (part 2.1 in Fig. 4) with a five-span uplifting window (called *Orsi*). On both sides of this room which have heights equal to two stories are located two earring-like rooms (part 3 in Fig. 4) each of which has an uplifting (*Orsi*) window

overlooking the yard. On each of the eastern and western fronts of the building, there is one hall (part 2.2 in Fig. 4) in the middle and two three-door rooms on its sides (part 4 in Fig. 4).

The hall height is equal to the main guest hall height located on the northern front and the hall also have beautifully decorated uplifting windows overlooking the yard. There also exists a hall on the southerly front (part 2.3 in Fig. 4) which is used as the sitting room during hot summer days. On top of this hall, there is an open balcony, called *Mahtabi* (receiving the moonlight), with a narrow porch at the far end of the space.

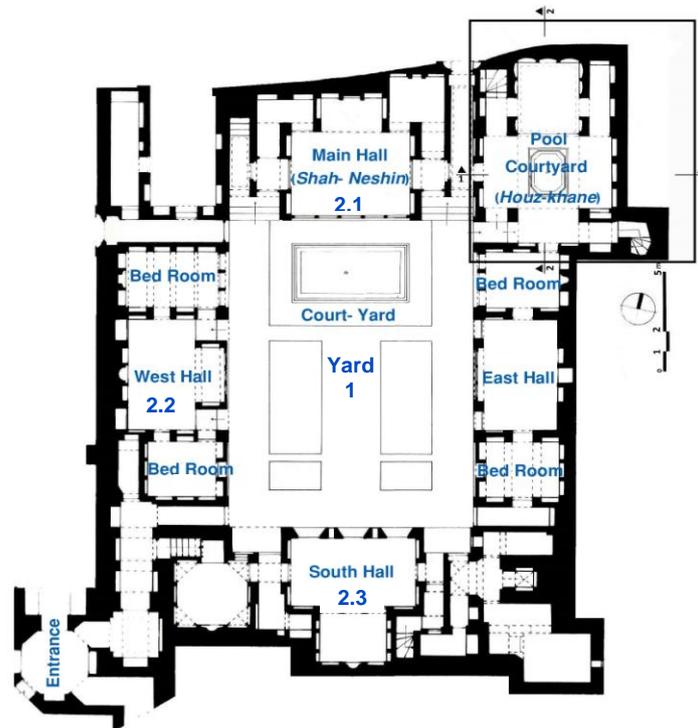


Fig. 4. First floor plan, adapted from [11]

On the northeastern corner of the house, a pool courtyard is located with a height equal to that of two stories (part 4 in Fig. 4) in which there is a water pool and a skylight on top with relatively large sashes. Both the pool

and the skylight are octahedrons. The ceiling of the pool courtyard is embellished with embossed plasterwork. The interior space is almost cubic in shape (7.5 m × 6.5 m × 9.5 m) with an approximate volume of 460 m<sup>3</sup>, Fig. 5.

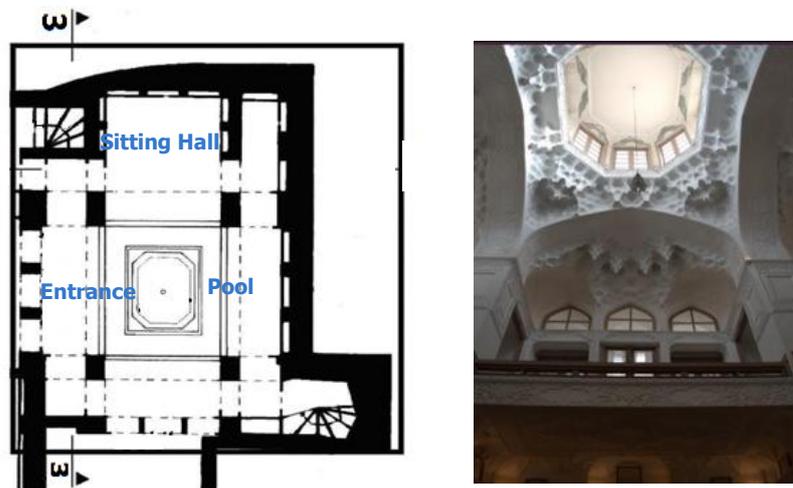


Fig. 5 The pool courtyard; left) Floor plan, right) The interior view, adapted from [11]

## 6. DESCRIPTION OF THE SOFTWARE AND MESH SPECIFICATION

The Ansys Fluent is a software package used in computational fluid dynamics which is capable of modeling systems such as turbulent and non-turbulent flows, compressible and incompressible fluids, Newtonian and non-Newtonian fluids, open surface flows, steady and unsteady

flows, and multi-phase flows. In this software, the finite volume numerical method is used for discretizing equations. In this method, partial differential equations are integrated in a physical space and then solved numerically. The method is especially applicable to fluid mechanic and heat transfer problems or problems with complex geometries.

In the present work, the Gambit software is used for generating the mesh. The mesh is of the quadrilateral type with different number of nodes on different sides. After a

series of sensitivity analyses and comparison of the results, the mesh sizing was found to yield acceptable results. The interval count instruction was used for meshing; it divides each side of the building into predetermined equal intervals. The number of existing cells is 159167, the smallest of which has a volume equal to  $8.191768e^{-7} \text{ m}^3$  and the volume of the largest is  $2.567409e^{-2} \text{ m}^3$ .

## 7. GOVERNING EQUATIONS

The main purpose of this paper is to study the air flow patterns. Although thermal properties may affect the flow properties (like density), the velocities are high enough to neglect the thermal effect on the flow patterns. The equations governing the fluid consist of Reynolds (Momentum) equation and Continuity equation. The latter equation for an incompressible fluid in a Cartesian coordinate is as follows:

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

where  $u_i$  [m/s] is the time-averaged velocity components. The Reynolds Equation in the Cartesian system is as follows:

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_i}(u_i u_j) = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} (-\overline{u'_i u'_j}) \quad (2)$$

where  $p$  [N/m<sup>2</sup>] is the piezometric pressure,  $\nu$  [m<sup>2</sup>/s] is the kinematic viscosity of the fluid,  $\rho$  [kg/m<sup>3</sup>] is the density of fluid, and  $u'_i$  and  $u'_j$  [m/s] are, respectively, the turbulence components of velocity in  $i$  and  $j$  directions. For modeling the incompressible turbulent flow, the *RNG*  $k-\varepsilon$  turbulent model is used as

$$\frac{\partial (k)}{\partial t} + \frac{\partial (k u_i)}{\partial x_i} = \frac{1}{\rho} \frac{\partial}{\partial x_i} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_i} \right) + \frac{1}{\rho} (G_k + G_b) - \varepsilon \quad (3)$$

$$\frac{\partial (\varepsilon)}{\partial t} + \frac{\partial (\varepsilon u_i)}{\partial x_i} = \frac{1}{\rho} \frac{\partial}{\partial x_i} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right) + \frac{1}{\rho} C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \frac{\varepsilon^2}{k} - R_\varepsilon \quad (4)$$

where  $k$  [m<sup>2</sup>/s<sup>2</sup>] is the turbulent kinetic energy,  $\varepsilon$  [m<sup>2</sup>/s<sup>3</sup>] is the depreciation rate,  $\mu_{eff}$  [m<sup>2</sup>/s] is the effective viscosity,  $\alpha_\varepsilon$  and  $\alpha_k$  are, respectively, the inverse effective Prandtl numbers for  $\varepsilon$  and  $k$  computed using the following formula derived analytically by the *RNG* theory:

$$\left| \frac{\alpha - 1.3929}{\alpha_0 - 1.3929} \right|^{0.6321} \left| \frac{\alpha - 2.3929}{\alpha_0 - 2.3929} \right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}} \quad (5)$$

where  $\alpha_0 = 1.0$ . For the high-Reynolds-number limit ( $\mu_{mol}/\mu_{eff} \ll 1$ ),  $\alpha_\varepsilon = \alpha_k \approx 1.393$ ,  $C_{1\varepsilon} = 1.42$  and  $C_{2\varepsilon} = 1.68$  are the standard coefficients in the  $k-\varepsilon$  equation,  $G_b$  [J] is the generation of turbulent kinetic energy due to buoyancy,  $G_k$  [J] is the generation of turbulent kinetic energy due to

changes in average velocity as

$$G_b = \beta \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \rho C_\mu \frac{k^2}{\varepsilon} ; G_k = -\overline{\rho u'_i u'_j} \frac{\partial u_j}{\partial x_i} \quad (6)$$

where  $Pr_t$  is the turbulent Prandtl number for energy. In the case of the *RNG*  $k-\varepsilon$  model,  $Pr_t = 1/\alpha$ , where  $\alpha$  is given by Equation (5), but with  $\alpha_0 = 1/Pr = k/\mu c_p$ . The coefficient of thermal expansion,  $\beta$  [1/k], is defined as

$$\beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p \quad (7)$$

and  $R_\varepsilon$  is a special parameter in the *RNG*  $k-\varepsilon$  model which is defined as

$$R_\varepsilon = \frac{C_\mu \eta (1 - \eta/\eta_0) \varepsilon^2}{1 + \beta \eta^3} \frac{1}{k} \quad (8)$$

where  $\eta_0 = 4.38$ ,  $\beta = 0.012$  and  $\eta \equiv Sk/\varepsilon$  in which  $S$  [1/s] represents the average deformation rate of the fluid element defined as

$$S = \sqrt{2S_{ij}S_{ij}} \quad (9)$$

## 8. BOUNDARY CONDITIONS

Given the steady state of the flow, a velocity boundary condition has been employed for the inlet of the pool courtyard (the blue part in Fig. 6). Two steady inflow velocities of 2.5 and 8 m/s are considered at the inlet of the skylight. Battle McCarthy Consulting Engineers [12] recommended the comfort wind speed of 2.5 m/s. According to the Report by Isfahan Synoptic Station, the predominant and the maximum wind speed in the study region are, respectively, 2.5 m/s and 8 m/s. In the present model, the flow escapes through a gallery connected to the yard. To study the flow in the pool courtyard, the outflow pressure was assumed to be equal to atmospheric pressure.

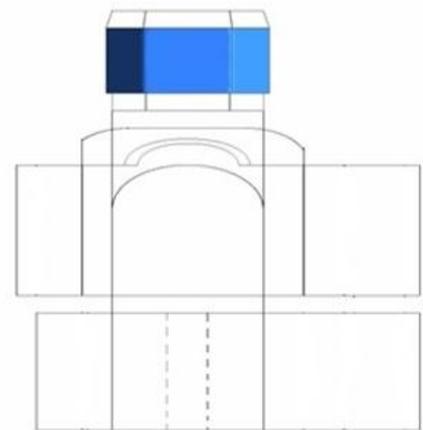


Fig. 6 Vertical section of the pool courtyard and the skylight

### 9. ANALYSIS OF NUMERICAL RESULTS

Assuming a wind speed of 8 m/s, the results of the velocity contour (the velocity distribution) and velocity vector were initially analyzed. The problem conditions were assumed to be steady and the analyses were performed at a convergence level of 3-10 in the software environment with 1300 replications accomplished to reach the desirable convergence. Air circulation within the building is affected by such factors as wall arrangements, velocity of inflow air, air pressure at different places within the building, the stack effect (i.e., the distance between the inlet and outlet), window shapes and their sashes, and the shape effect of the building on creating low pressures. Fig. 7 shows that the maximum velocity occurs at the exit section (i.e., the entrance door of pool courtyard in Fig. 5). The velocity changes from the door edges up to its center where it reaches its maximum. That maximum velocity occurs due to the low ratio of the cross section to the skylight's inlet.

The results of the numerical model (Figs. 8 and 9) obtained for velocities of 2.5 and 8 m/s at the sections in the center of the room (Section 1-1 in Fig. 4) indicate that the acceleration of the main current covering the path between the inlet window and the exit section increases with increasing inflow wind speed. Comparison of the results reveals that a wind velocity of 2.5 m/s provides better ventilation than one of 8 m/s. Based on the conservation of mass and due to the fact that the inflow and outflow discharges are equal, the air entering each space stays for a longer time when the wind velocity is lower. This long time creates adequate flow lines to affect the neighboring spaces. In contrast, a main current velocity of

8 m/s will have less effect. For airflow of 2.5 m/s (Fig. 9), more exchanges occur between the lateral and the main currents so that the lateral ones are drawn toward the main current, which leads to the constant change of air in the space.

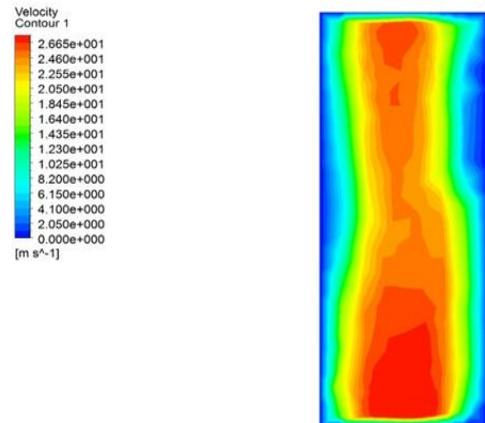


Fig. 7 Velocity contours at the pool courtyard entrance door (exit section)

Figures 10 and 11 show the flow pattern at Section 2-2 (Fig. 4) in the center of the structure for the two above velocities. As shown in Fig. 10 (an airflow velocity of 2.5 m/s), the current circulates within the spaces inside the building. These whirling currents flow regularly. With a wind velocity of 2.5 m/s, the maximum air speed near the ceiling on the second floor and at the outlet of the skylight reaches 4 m/s and reaches 11 m/s for the wind speed of 8 m/s. Figure 11 (for a wind speed of 8 m/s) shows that the flow pattern at this section is similar to that for a wind speed of 2.5 m/s.

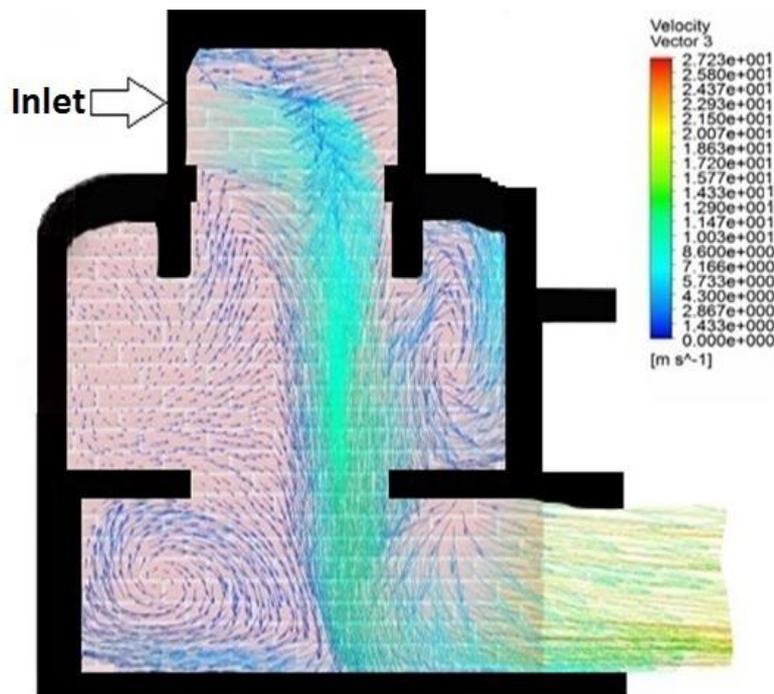


Fig. 8 Airflow pattern at Section 1-1 (Fig. 4) for an inflow velocity of 8 m/s

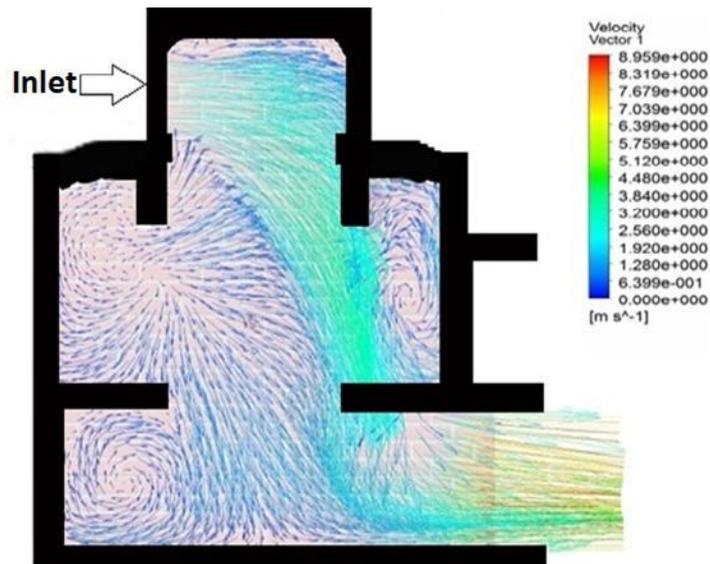


Fig. 9 Airflow pattern at Section 1-1 (Fig. 4) for an inflow velocity of 2.5 m/s

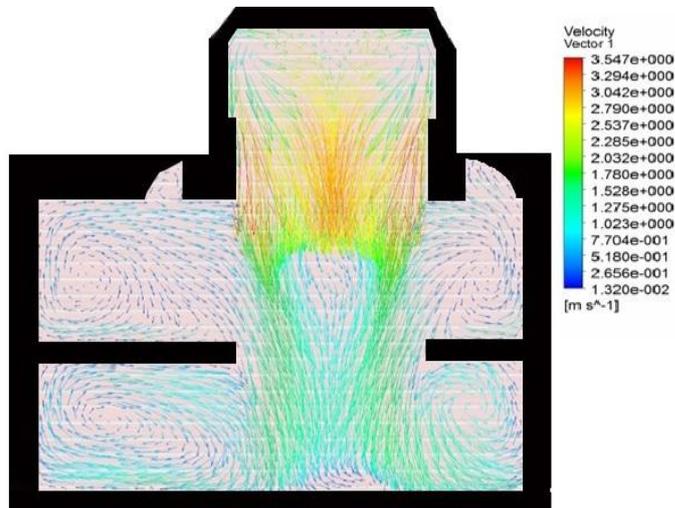


Fig. 10 Airflow pattern at Section 2-2 (Fig. 4) for an inflow velocity of 2.5 m/s

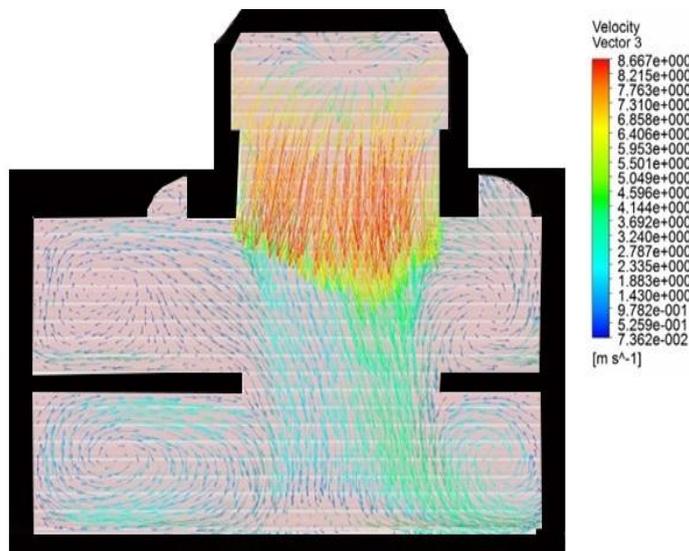


Fig. 11 Airflow pattern at Section 2-2 (Fig. 4) for an inflow velocity of 8 m/s

Generally speaking, ventilation at higher wind speeds is not as good as that at lower speeds. The lower wind speeds reduce the eddies and the turbulence to provide more comfort for the dwellers. The regularity observed in the flow pattern with a wind speed of 2.5 m/s makes it possible to achieve the desired effect of the inflow current on all the inside spaces.

Columns and vault in the building are the other factors that create circulations, because they partitioned space and guiding wind inside the building. The impact of columns and vault on the air flow pattern are shown in Figs. 12 and 13. Figure 12 is a horizontal section near the ceiling of the first floor and Fig. 13 is a vertical section at the north part of the pool courtyard (Section 3-3 in Fig. 5).

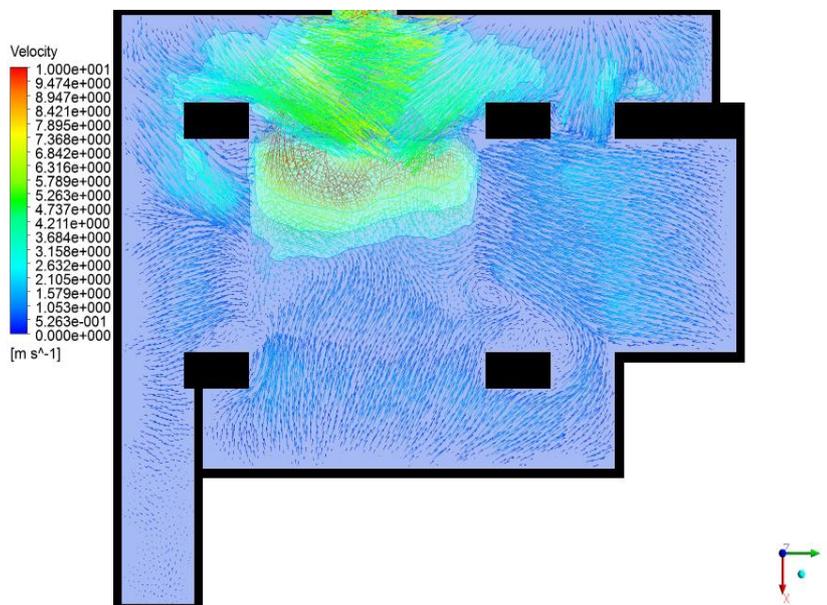


Fig. 12 Airflow pattern near the ceiling of the first floor for an inflow velocity of 8 m/s

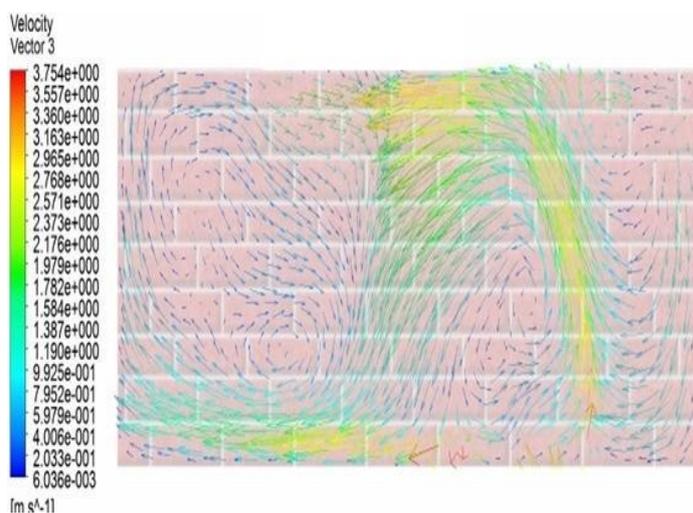


Fig. 13 Airflow pattern at Section 3-3 (Fig. 5) for an inflow velocity of 8 m/s

The presence of columns and the impact of the whirling flow lines against them in the center due to the vaults cause the flow lines to converge in this region of the building so that higher wind speeds may not provide the desired comfort. The centers of the circulations in Fig. 12 are located at the columns which may be considered as important factors in determining paths of air conduction. Due to the narrow outlet section, a high-speed zone along with irregular flow lines is observed between two columns near the section. Unlike this, the two columns located at a farther distance from the outlet section have more regular flow lines where

the wake separation zone is also better delineated.

Another prominent characteristic of the building is its duplex design which creates spaces shared between the first and the second floors. Using the flow patterns in Fig. 14 (Section 3-3 in Fig. 5), we may infer that when the two downward currents collide against the vault separating the two stories, the air is returned upward to start circulating within the sitting room on the second floor. The currents move at a velocity of 4 to 5 m/s to provide uniform ventilation on the second floor.

Figure 15 presents a view of the building's vault. The

curvature in it located around the columns causes part of the upward airflow to be conducted downward and, thus, a circulation (eddy) is created in this region. Clearly, the air

velocity near the ceiling toward which the current tends to move is higher than those formed in other parts of the ceiling.

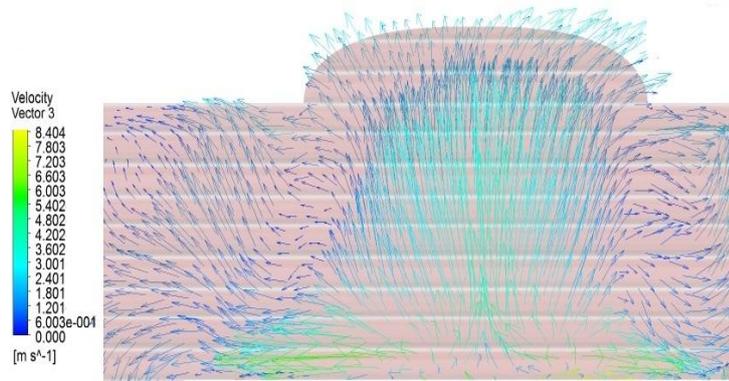


Fig. 14 Airflow pattern at Section 3-3 (Fig. 5) on the second floor for an inflow velocity of 8 m/s

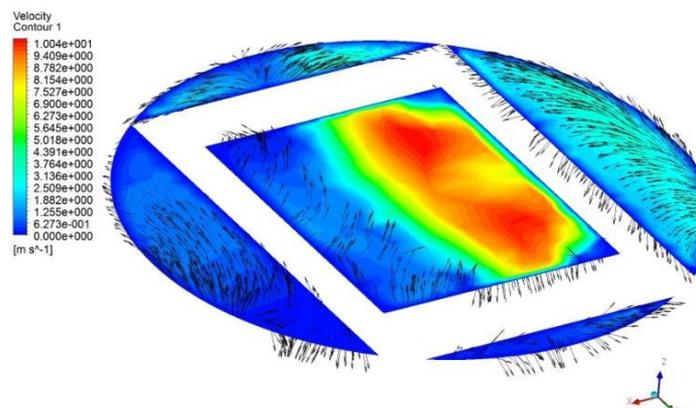


Fig. 15 Airflow pattern at the ceiling for an inflow velocity of 8 m/s

As seen in Fig. 16 (2<sup>nd</sup> floor ceiling), a major portion of the current flows among the four columns and does not diffuse in the inside spaces. The velocity of this current near the two columns facing at the upstream reaches the maximum air velocity inside the building.

The reason why the maximum velocity occurs at this region is that the air velocity reaches zero as a result of

colliding with the columns (stagnation point). Moreover, since this region is exposed to direct wind blows and due to the high velocity of the air current, a slight deviation occurs in the flow path and, therefore, the major portion of the current passes through this region, which inevitably leads to increased velocity in order to satisfy the continuity equation.

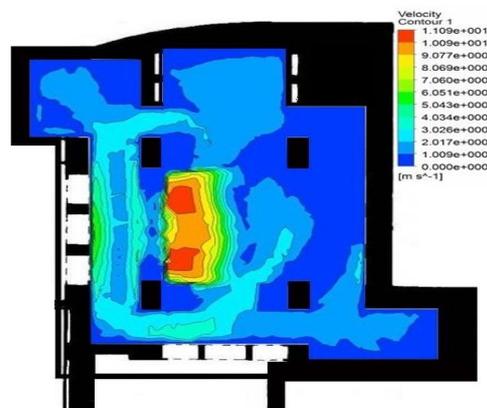


Fig. 16 Velocity contours at the upper space on the second floor for an inflow velocity of 8 m/s

## 10. CONCLUSION

In this study, the flow patterns within the space over the pool courtyard of Mosavar-al-Molki's house was investigated using simulations of the inflow air current for two velocities of 2.5 and 8 m/s. Based on the flow patterns and velocity contours obtained, the analyses revealed that the skylight has different effects on the flow patterns created with different air velocities. The vault and the columns within the pool courtyard were also found to be effective factors involved in the diffusion and conduction of the flow within the building spaces and improving the ventilation. The formation of circulations on the second floor due to the presence of columns, their collision, and the presence of the vaults were found to cause the flow lines to converge in this zone. It was found that the spaces on the second floor may not provide a desirable thermal comfort at high air velocities. Further, it was found that the currents formed in the lateral spaces are basically drawn toward the main airstream, which gives rise to the constant and regular circulation of air. This even circulation of air blowing at an appropriate velocity enhances the thermal comfort and creates a more desirable environment due to the sustained influence of the outdoor conditions. Comparison of the results revealed that an air velocity of 2.5 m/s (i.e., the predominant wind speed during the hot seasons) provides better ventilation in the building than the higher velocity of 8 m/s. Investigation of the flow patterns formed due to temperature differences in the indoor spaces and the evaporative cooling effect of the pool courtyard is recommended for future research. Also, comparison of wind tower performance and skylight performance would be a good topic for future studies.

**Acknowledgement:** The authors would like to extend their most sincere thanks to Mahmood Rezaei Haftadari for his assistance in implementing this research. Mr. Mossavor-ul-Molki, Isfahan Cultural Heritage Organization, and Faculty of Art University of Isfahan also deserve our gratitude for their support during this project.

## REFERENCES

- [1] Mahmoudi Zarandi M, Kolbadi Nezhad M, Pourmousa M. Climatic guidelines for designing open spaces in residential complexes of yazd, International Journal of Architectural Engineering and Urban Planning, 2014, No. 1, Vol. 24, pp. 24-36.
- [2] Allard F, Santamouris M. Natural Ventilation in Buildings: A Design Handbook, London, Routledge Publishing, 1998.
- [3] Khan N, Su Y, Riffat S. A review on wind driven ventilation techniques, Energy and Buildings, 2008, Vol. 40, pp. 1586-604.
- [4] Ansys 14 for Microsoft Windows. Software Version 14, Ansys Inc. Pennsylvania, 2011.
- [5] Ohba M, Iriea K, Kurabuchi T. Study on airflow characteristics inside and outside across-ventilation model and ventilation flow rates using wind tunnel experiments, Journal of Wind Engineering and Industrial Aerodynamics, 2001, Vol. 89, pp. 1513-24.
- [6] Van Moeseke G, Gratia E, Reiter S, De Herde A. Wind pressure distribution influence on natural ventilation for different incidences and environment densities, Energy and Buildings, 2005, Vol. 37, pp. 878-89.
- [7] Evola G, Popov V. Computational analysis of wind driven natural ventilation in buildings, Energy and Buildings, 2006, Vol. 38, pp. 491-501.
- [8] Cheung JOP, Liu CH. Cfd Simulations of natural ventilation behavior in high-rise buildings in regular and staggered arrangements at various spacing, Energy and Buildings, 2011, Vol. 43, pp. 1149-58.
- [9] Bu Z, Kato S. Wind-induced ventilation performances and airflow characteristics in an areaway-attached basement with a single-sided opening, Building and Environment, 2011, Vol. 46, pp. 911-21.
- [10] Nguyen AT, Reiter S. The effect of ceiling configurations on indoor air motion and ventilation flow rates, Building and Environment, 2011, Vol. 46, pp. 1211-22.
- [11] Haji Qassem K. Mansions of Esfahan Ganjnameh. Tehran, Faculty of Architecture and Urban Planning, Documentation and Research Center Publishing, 2004, Vol. 4.
- [12] Battle McCarthy Consulting Engineers. Wind Towers: Detail in Building, London, Academic Press, 1999.
- [13] Razjooyan M. In the Comfort of Wind. Tehran (In Persian), Shahid Beheshti University Publications, 2007.

### AUTHOR (S) BIOSKETCHES

**Nabizadeh Shahrehabak, E.,** Research Assistant, Department of Mechanical Engineering, University of Nevada-Las Vegas.

Email: *Eb.nabizadeh@gmail.com*

**Golafshan, S.,** Lecturer, Department of Civil Engineering, Isfahan University of Technology.

Email: *s.golafshan@cc.iut.ac.ir*

**Chaman, M.R.,** Associate Professor, Department of Civil Engineering, Isfahan University of Technology.

Email: *mchamani@cc.iut.ac.ir*

### COPYRIGHTS

Copyright for this article is retained by the author(s), with publication rights granted to the journal.

This is an open-access article distributed under the terms and conditions of the Creative Commons Attribution License (<http://creativecommons.org/licenses/by/4.0/>).

**HOW TO CITE THIS ARTICLE**

*Nabizadeh Shahrebabak, E., Golafshan, S., Chamani, M.R. (2016). Air flow simulation through the skylight in the historical house of Mosavar-al-Molki. Int. J. Architect. Eng. Urban Plan, 26(1): 41-51, June 2016.*

URL: <http://ijaup.iust.ac.ir/article-1-256-en.html>

