Armanshahr Architecture & Urban Development, 9(16), 39-57, Spring Summer 2016

ISSN: 2008-5079

Natural Ventilation: Analysis of Indoor Airflow in an Assumed Cubic Building with Opposite Openings by CFD Investigations

Omid Rahaei[1*]

[1]Assistant Professor of Architecture, Department of Architecture, Shahid Rajaei Teacher Training University, Tehran, Iran

Received 16 March 2015; Revised 11 October 2015; Accepted 13 October 2015

ABSTRACT: The natural ventilation is an easy way to exchange the indoor polluted warm air with outdoor fresh air. The wind power injects outdoor fresh air into the building. A good indoor air current and subsequently a proper exhaust depend on the openings' conditions and their situations. A serious architectural question is under what conditions of the openings the wind-cross ventilation can be effective, and the required indoor air current in the enclosure is established. The purpose of this article is analyzing the conditions of indoor airflow in an analytical architectural model to upgrade the natural ventilation by focusing on opposite opening's conditions. This research considers some wind driven ventilation manner with respect to openings circumstances in an assumed cubic model. The research method includes a numerical simulation using a validated computational fluid dynamics (CFD) model. It investigates and compares the performances of different models of airflow currents in a natural ventilation process and subsequently the indoor airflow paths, under the different conditions of the openings in a fixed boundary condition model; the simulations are performed in an assumed model (a 6×6×6m cubic building with just 2 opposite openings in stationary walls as boundaries of the model) by using Gambit and Fluent software. With an analytical method (using Fluent) the gathered data would be analyzed. Finally the results are presented and generalized: the results demonstrate that whatever the wind speed is, the indoor airflow condition depends on the situations of the openings. It means that the quality of wind-driven cross ventilation and its path is not depended on the wind speed. Besides for establishing proper natural ventilation, the opposite windows must not be installed in front of each other, or in the same level.

Keywords: Wind-driven Cross Ventilation, Large Openings, CFD Method, Sealed Buildings, Indoor Airflow.

INTRODUCTIONS

In an enclosed space, Air should continuously be withdrawn and replaced by fresh air from a clean external source to maintain good indoor air quality (Oakley, 2002). Natural ventilation driven by winds is widely used in hot and humid climate. Yet wind-driven natural ventilation is difficult to analyze and control because airflows around buildings are complex and generally turbulent. Nevertheless, for engineering purposes, wind pressure coefficients for sealed buildings and flow relations based on approximate applications of the Bernoulli equation are commonly used for predicting wind-driven cross ventilation in corresponding buildings with openings (Liddament, 1986). In a simplified diagram by Seifert, j. et al., (2006), it is schematically assumed that depending on the windows' positions and the velocity of the outdoor wind, the air velocities within buildings are variable (Fig. 1). The outdoor airflow and its coefficients around the buildings are depending on direction of the wind, surface orientations of buildings, surrounding shielding, and the topography in the upwind direction. In architectural purposes large openings (open windows and doors) are necessarily found in wind-ventilated buildings when this

*Corresponding author email: o.rahaei@srttu.edu

ventilation is used for cooling. For a simple building with two opposite openings (Fig. 1) the difference of static pressure between both sides is less than what is expected for the corresponding sealed building also. The question is when and under what conditions of large openings (open windows and doors) in enclosures, a controlled indoor airflow (using natural ventilation process) is established and the wind pressure coefficients can be used in calculating natural ventilation flow rates. Thus a proper approach to building indoor airflow analysis is calculation of airflow velocities and the static pressure in all inner points.

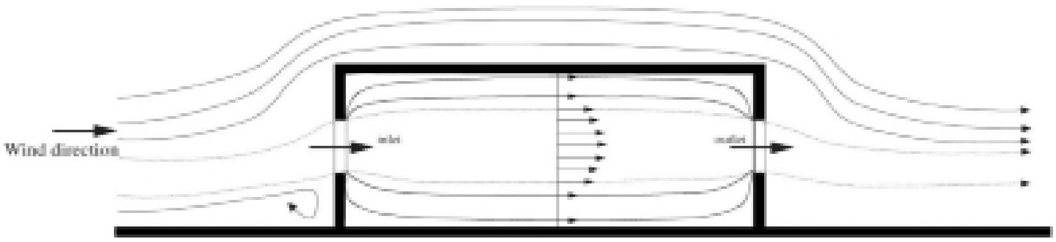


Fig. 1. Wind Flows around and through a Single-zone Building with Two Openings (Seifert et al., 2006).

LITERATURE REVIEW

Akins and Cermak (1976) have published parameter studies of the local pressure coefficients on a rectangular prism-shaped model. In 1980 an Investigation of Wind Forces on Three Dimensional Roughness Elements in a Simulated Atmospheric Boundary Layer were done (Hussein & Lee, 1980). A technique for predicting wind pressure coefficients which takes into account the effects of building form and surrounding shielding is given (Knoll et al., 1995). Wind pressure coefficients for complex buildings are not readily available although wind tunnel tests, but computational fluid dynamics (CFD) methods may be used to estimate pressure coefficients (Kato et al., 1997). As interests in low-energy cooling strategies have grown in recent years, research investigations of wind-driven natural ventilation in buildings with large openings have proliferated (Seifert et al,. 2006). The similar questions have been asked a number of times in the past (Aynsley et al., 1977) (Jensen True et al., 2003) (Sandberg, 2003), but further investigations are still needed to find a complete answer. Prediction of indoor airflow in buildings and the rate of heat and pollution transmissions can give useful information to designers for optimizing designs. Indoor airflow's information in enclosures is notable for 3 reasons: thermal comfort, indoor air quality and energy consumption. The studies show that "indoor airflow" as a distinct new science especially in last 2 decades was appeared (Amidpoor, 2010). There are 2 methods for airflow analysis: experimental methods and digital simulations. Since experimental methods needs under-controlled real buildings and are expensive. The device's errors that are not ignorable depend on the device's accuracy and the circumstances (Loomans et al., 1995). Arenz (2000) used and presented experimental thermometers, density, and wind speed in a real plant's workshop and office (Arenz, 2000). Rahaei (2014) presented an experimental method in a plant to investigate the indoor airflow tube (Rahaei, 2014). The digital simulations in comparison with the experimental methods are easier, more accurate, and under control, but need some proximate assumptions (Amidpoor, 2010). Furthermore ventilation systems were considered and analyzed by Novoselac (Novoselac et al., 2002, pp. 497–509), Rees (2001) and Ghali (Ghali et al., 2007, pp. 743–759) in combination with cold roofs using digital simulations.

However, Computational Fluid Dynamics (CFD) codes are now commercially available, and over the last ten years, many authors since the original studies performed by Van Gerwen and Van Oort (1989) and Wang and Touber (1990) have investigated the use of CFD as a tool for rationalizing design and operation. The standard $k-\varepsilon$ turbulence model was used frequently for the calculations also (Foster et al., 2002; Hoang et al., 2000; Hu & Sun, 2000; Mariotti et al., 1995; Mirade et al., 1995; Mirade & Daudin, 1998a; Mirade et al., 2002; Scott, 1994; Scott & Richardson, 1997; Xia & Sun, 2002; Xie et al., 2006). General purpose CFD codes such as Fluent, CFX, Star-CD and Phoenics have been designed for solving turbulent fluid flow problems coupled with heat and mass transfers in a given geometry by the use of a mesh where all the Navier–Stokes transport equations are solved across each mesh cell by means of an iterative procedure requiring specific algorithms. The above-mentioned studies highlight how the application of CFD will improve the understanding of dynamics and physics of the ventilating operation and thus help to optimize existing equipment and design new solutions; As Mirade and Picgirard (2001) used CFD techniques to improve air circulation around beef carcasses in a continuous-type chiller.

Following the above, it's clear that solving the airflow problems in contemporary buildings is necessary and the subject of ventilation as mentioned in introduction (especially the natural ventilation) is wildly investigated by many researchers. However the CFD analyses of indoor air currents in simple rooms were not performed and all the existed data are supposedly estimated. As an example, Kasmaei (2011) has many supposed diagrams that may works correctly, or Ghobadian (2012) has some researches in this field. But the presented methods are not accurately examined by experimental methods or digital calculations. So it is necessary to examine the indoor air currents in architectural spaces to find some architectural rules for better designs. It is notable that this research is about natural ventilation in enclosures. The research will analyze the indoor airflow conditions in a simple room with different positions of the opposite windows, as described before. The indoor airflow analyzes will be performed by digital simulations in CFD method also.

METHODOLOGY

The purpose of this article is proposing a solution for better natural ventilation in enclosed spaces by making alterations in large openings' situations (open windows and doors as architectural variables) to improve indoor airflow condition and develop some optimized-economical methods in natural (wind-cross) ventilation. Since the study involves some fields and "many architectural researches are interdisciplinary and require special combined techniques" (Groat et al., 2004), this study is somehow interdisciplinary. This paper reports an investigation of wind flow patterns in an enclosure with 2 opposite large openings: an assumed cubic building with 10% Porosity as one opening in just 2 opposite walls. Wind flow around a square-plan one-zone building (as model) was investigated by computational fluid dynamics (CFD) method. The validity of the simulation's method in CFD was proved with retest the similar projects and recalculation of the pressure coefficient as well as a comparison with experimental results. The study is based on CFD simulations (using $k-\varepsilon$ turbulence model) which are validated by experimental measurements that were carried out at the normal wind incidence case. In the unstructured finite volume method, the first-order upwind scheme is chosen for all convection terms and the second-order central differencing is used for the diffusion terms (Seifert et al., 2005). So this research, by the mentioned methods, will present some wind pattern in CFD analyzes on indoor airflow through an enclosure with 2 opposite large openings, under different conditions of the openings, at just one wind incidence angles: perpendicular to porous walls. Finally the results will be generalized.

PRESUMPTIONS

A macroscopic diagram of a simple two-opening single-zone building with unidirectional cross-ventilation is shown in Fig. 2. In this diagram three control volumes may be distinguished, i.e., the inlet control volume between reference planes 1 and 2, the zone (room) itself between 2 and 3, and the outlet control volume between 3 and 4. Under the action of the wind and the conditions of the openings, air flows in through reference plane 1 at the inlet and out through reference plane 4 at the outlet. In general, the airflow velocity profiles will vary over the area of each reference plane, here, defined in terms of coordinates y and z local to each plane. This variation is likely to be significant in larger openings. Likewise, pressures will also vary over each of the four reference planes. In simplified approaches to ventilation modeling, this variation is most often ignored (Seiferd, et al, 2006). Following Seifert's approaches, in this research all the variations in opening are ignored too.

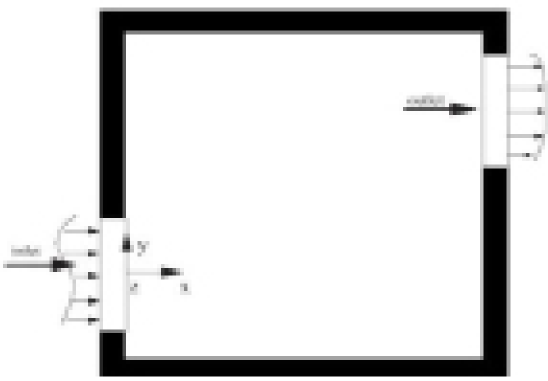


Fig. 2. Schematic Diagram for a Simple Single-zone Building with Wind Cross-ventilation.

Following the literature review and many researchers, in this research, in fluent processes (for wind cross-ventilation, illustrated in Fig. 2), the pressure and velocity fields are assumed uniform, inlet and outlet areas are equal, the room resistance is ignored, a laminar current is assumed in cubic model, and inlet and outlet resistances are ignored. No thickness for walls is assumed also.

A CFD ARPOACH AND ITS VALIDITY

In this research, the commercial CFD software, FLUENT 6.0, was applied and the standard $k-\varepsilon$ turbulence model was used for all of the CFD alculations (following the article's literature review). First of all, the validity of CFD calculations in this research must be proved: to validate the simulations, it is important to simulate the indoor airflow in a real cubic sample that was experimentally examined with experimental methods and the digital devices. Comparing the CFD results with the experimental tests would validate the simulations. However the results may be somehow theoretical, such as the approach of "large eddy simulation (LES)", reported in Kurabuchi et al. (2004), the comparative methods to validate calculations will led the researchers to realistic results (Chiu & Etheridge, 2007). Many researchers, mentioned in literature review of this article, have the same method to validate their CDF approaches.

In order to evaluate the CFD approach of this research, the calculation's method and its results is compared with the surface mean wind pressure coefficient distribution measured by Castro and Robins (1977) for wind flows around an isolated building. Castro and Robins (1977) assumed a cubic structure for their investigations with a side of 0.2 m. They measured Cp (wind pressure coefficient) values on all facades for a wind speed of 7.675m/s in both uniform and sheared approaching wind flows. Their cube is located in a large computational domain (with length x = 80 m, width y = 70 m and height z = 40 m). Pressure boundary conditions are applied at the downstream, side and top boundaries, and the non-slip boundary condition is applied at the lower boundary of the computational domain. Therefore, the selected boundary conditions are chosen to be the same as the conditions used in the wind tunnel experiment (temperature of 20°C) by Castro and Robins (1977). For generating the grid, Gambit was applied: a Tet grid manner was used (Castro & Robins, 1977).

In this research, Castro's method is applied and retested. All simulations in fluent were done and the results are considered when the residuals of all governing equations are reduced by three orders of magnitudes from their initial values: In the other word, the residuals of the equations in the software (fluent) are converged (Seiferd et al., 2006).

Fig. 3 shows the comparison of the predicted and measured Cp values. The agreement between the two results is reasonably good for all turbulence models on the windward wall (line A and line D). Castro's simulation method is applied in this research too. For validating this method in our research some experimental tests were necessary.

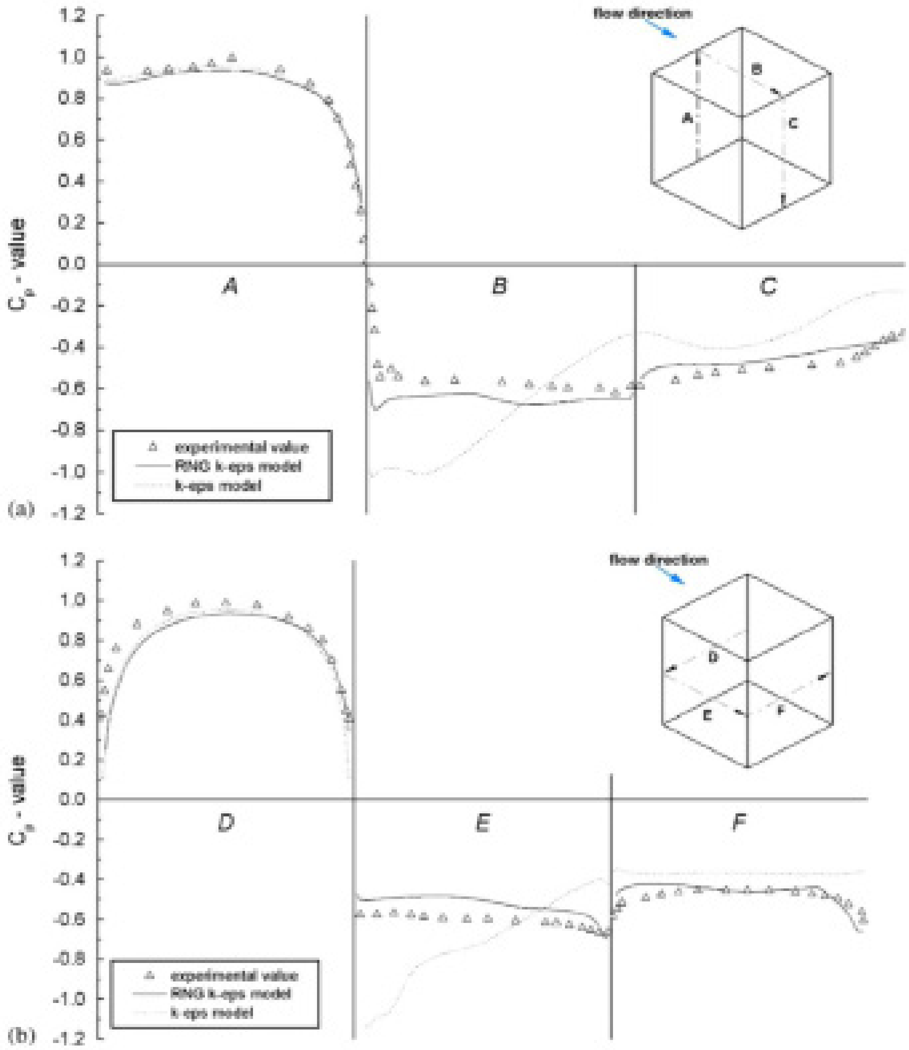


Fig. 3. Comparison of Wind Pressure Coefficient on the Cube (0.2 m x 0.2 m x 0.2 m) Surface Between Computational and Experimental Results of (Castro & Robins, 1977): (a) Along the Line A–B–C; (b) Along the Line D–E–F

For experimental testes, a real case study was selected and examined: In a flat field in Saveh (a city situated in south west of Tehran) a cubic cabin was selected (Fig. 4). It was a rectangular 2x3m cabin with the height of 2.5m. Castro's method was used to simulate and the experimental tests confirmed that. The experimental tests were done with 2 digital anemometers simultaneously: one is applied to check the outdoor wind speed, and the other one to check the indoor airflow speed. In order to measure the outdoor wind speed, the speedometer Lutron Machine, Model AM-4204, was used. This device, with its hot wire sensor, is able to connect with the computer and record the fluctuations of wind speed in chart track. When the outdoor wind's speed was 3m/s, the other device (Kimo) was applied to measure the indoor wind speed. The gathered data were analyzed by fluent software and the Cp values were measured by the software. Fig. 5 illustrates a comparison between our real experimental observations calculated by fluent, and the Castro's Cp values' calculations in wind tunnel. In this chart (Fig. 5) the average Cp in standard of $k-\varepsilon$ model was computed and compared with average Cp values by Castro. The results are comparable with Fig. 4 that the whole walls surfaces were calculated. The results of Cp values (wind pressure coefficient distribution) in Fig. 5, evaluate the research method in this study. All the meshes by Gambit and the fluent way to calculate and simulate the data are performed by this evaluated and validated method.

Fig. 4. A Real Case Study for Experimental Tests for This Study in Saveh.

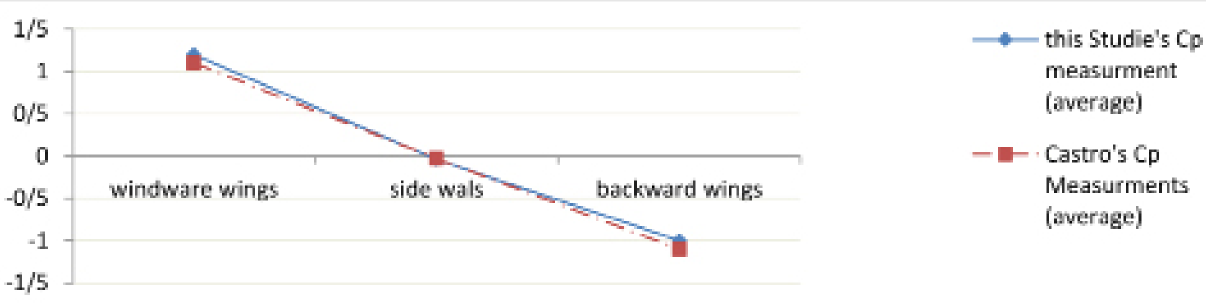


Fig. 5. A Comparison between the Average of Cp Calculations in this Research and the Average Cp Values by Castro to Validate the Research Method of this Article

INITIAL SETTINGS FOR FLUENT

Following the above, for all the remaining simulations to be reported in this paper, a simple cubic building with a length of 6m is considered. The standard thickness of the wall (walls and roof) is 0.2 m. The cube is located in a large computational domain of 80 m long, 70 m wide and 40m high. The numerical schemes and grid used are the same as described before in Fig. 3. The wind is simulated as a uniform profile with a speed of 3 m/s and the air current is assumed laminar. The turbulence intensity level at the inlet is 1%. A total of nine building configurations are considered to study. Pressure boundary conditions are applied at the downstream, side and top boundaries, and the non-slip boundary condition is applied at the lower boundary of the computational domain. Therefore, the selected boundary conditions in this research are chosen to be the same as the conditions used in the wind tunnel experiment by Castro and Robins (1977). All thermal physical properties of the air are evaluated at a temperature of 20°C.

INITIAL CONFIGURATIONS OF THE MODELS

For the building configurations here, both windward and leeward openings areas are equal. For the three building configurations shown in Fig. 6, dominant stream tubes can be expected to form when the openings are sufficiently large (2m assumed). However, the curvature of these stream tubes can be different among the three configurations. For the configuration of Case A, a dominant stream tube flow may not emerge when the two openings are offset. As the two openings become ‘‘overlapped’’, i.e., part of the two openings can ‘‘see each other’’, the wind flow is expected to be ‘‘short-circuited’’ and then a relatively straight dominant stream tube may appear. This is expected to occur when the wall porosity is sufficiently large, i.e., there will be critical wall porosity when a straight stream tube is expected. For the Case B and C configurations, it is clear that the two openings always ‘‘see each other’’; thus a straight dominant stream tube should appear between the 2 windows, if the windows are large enough. In this study, the 10% porosity of walls (opening’s area) was assumed: a 2×2 m opening (5cm opening’s framework) in a 6×6m wall with 0.2 m thickness.

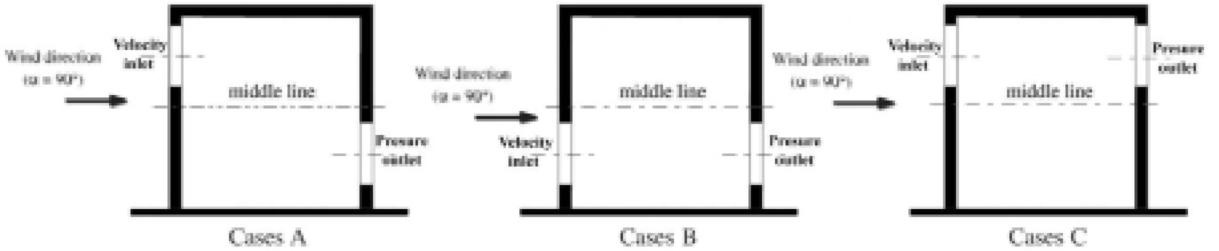


Fig. 6. Three Building Configurations with Two Identical Equal-Area Openings Located in the Opposite Walls Used for Studying the Effect of Building Envelope Porosity on Ventilation Flow Rates.

SIMULATIONS AND ANALYZES

The simulations of the 3 cases (A, B, and C in Fig. 6) are performed in Fluent (meshes with Gambit) and the results are illustrated (Figs. 7 and 8). Case A is simulated in 2 conditions, illustrated in Fig. 7. Following to Fig. 7, when the wind is blowing right to upper opening, the dominant current of indoor airflow is in the upper part of the room. In lower parts, an eddy current appears. This eddy current just circulates the indoor air around a point in about one third of the room's height (in lower part). So following Fig. 7 (a_1 in left) a cool calm protected current is established in lower parts, near the right wall (Fig. 7, a_1). In this condition the static pressure is like the outdoor's and the violence of the airflow is low. The stances of a, b, and c in Fig. 7 will occur when the outdoor wind blows toward the lower opening in case A. in this stances, the gradient of the wind pressure and its velocity in indoor airflow is unpredictable. The violence is high and the wind blows in lower parts of the room (in occupants' level). The differences of both conditions are illustrated in Figs 7 and 8 illustrated the different stances of cases B and C (in Fig. 6). The main current of indoor airflow in case B will occur in lower levels: the air current's velocity and the static pressure of this level, (tenant's level) is so violent. In case C there is a very calm circulating resident current in tenants' level (lower parts). The natural ventilation in this case is not good. Following the the simulations and the above-mentioned analyzes (for the cases of A, B, & C in 2d stances), following the conditions of indoor airflows, case C is refused; because the natural ventilation in this case is violence, air speed is so slow, and the indoor airflow's stance especially in people levels is self-circulating.

Some different conditions of the openings in assumed cube (a 6×6×6m cube with 2 opposite large openings) were simulated again, but in 3d modeling. In new simulations the conditions of the openings will differ in 3 dimensions. Following the Fig. 9, the 2 parallel openings in opposite walls are situated in the middle of the walls next to the floor level, right in front of each other (Fig. 9). Following the presumptions of this article (mentioned before), It's assumed that the outdoor wind is blowing right toward the opening with the velocity of 3m/s. following the models of Fig 9, the main indoor airflow is established between the 2 openings and the other parts of the room have calm close-circulated inner current of indoor air. The static pressure in all points of the inner space is almost the same (Fig. 9: c & c_1) and the indoor current cannot make a good natural ventilation. The simulated velocity magnitude and its vectors (a, a_1, b, b_1, & d_1 in Fig. 9) confirm this theory.

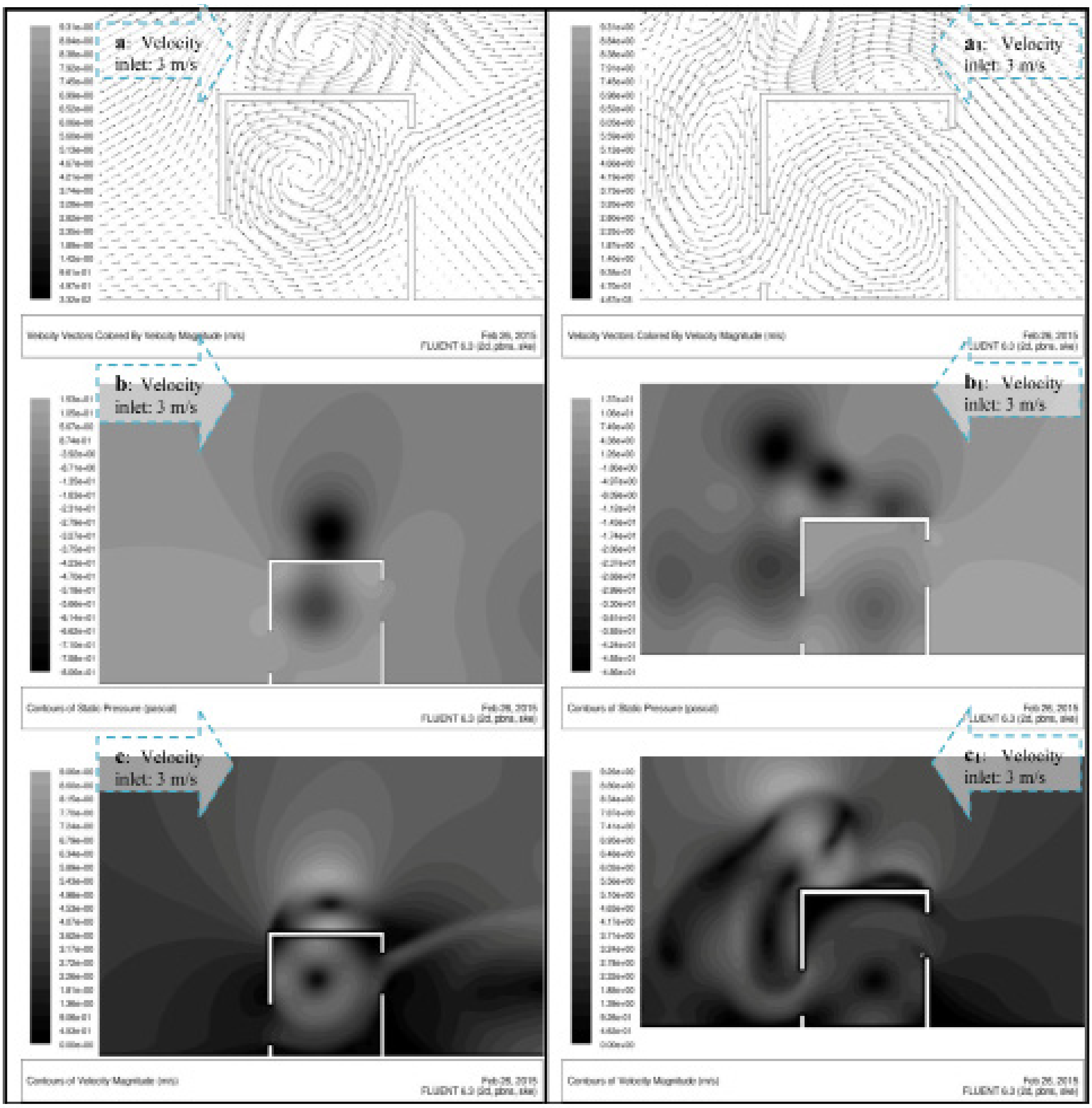


Fig. 7. Different Conditions of Airflow in Case A Depending on Wind's Directions (Left or Right, α=90oc); a, b, c in left: Wind's Direction is Left to Right (a. Velocity Magnitude's Vectors, b. Static Pressure's Contours, c. Velocity Magnitude's Contours). a_1, b_1, c_1 in Right: Wind's Direction is Right to Left (a_1. Veloctity Magnitude's Vectors, b_1. Static Pressure's Contours, c_1. Velocity Magnitude's Contours).

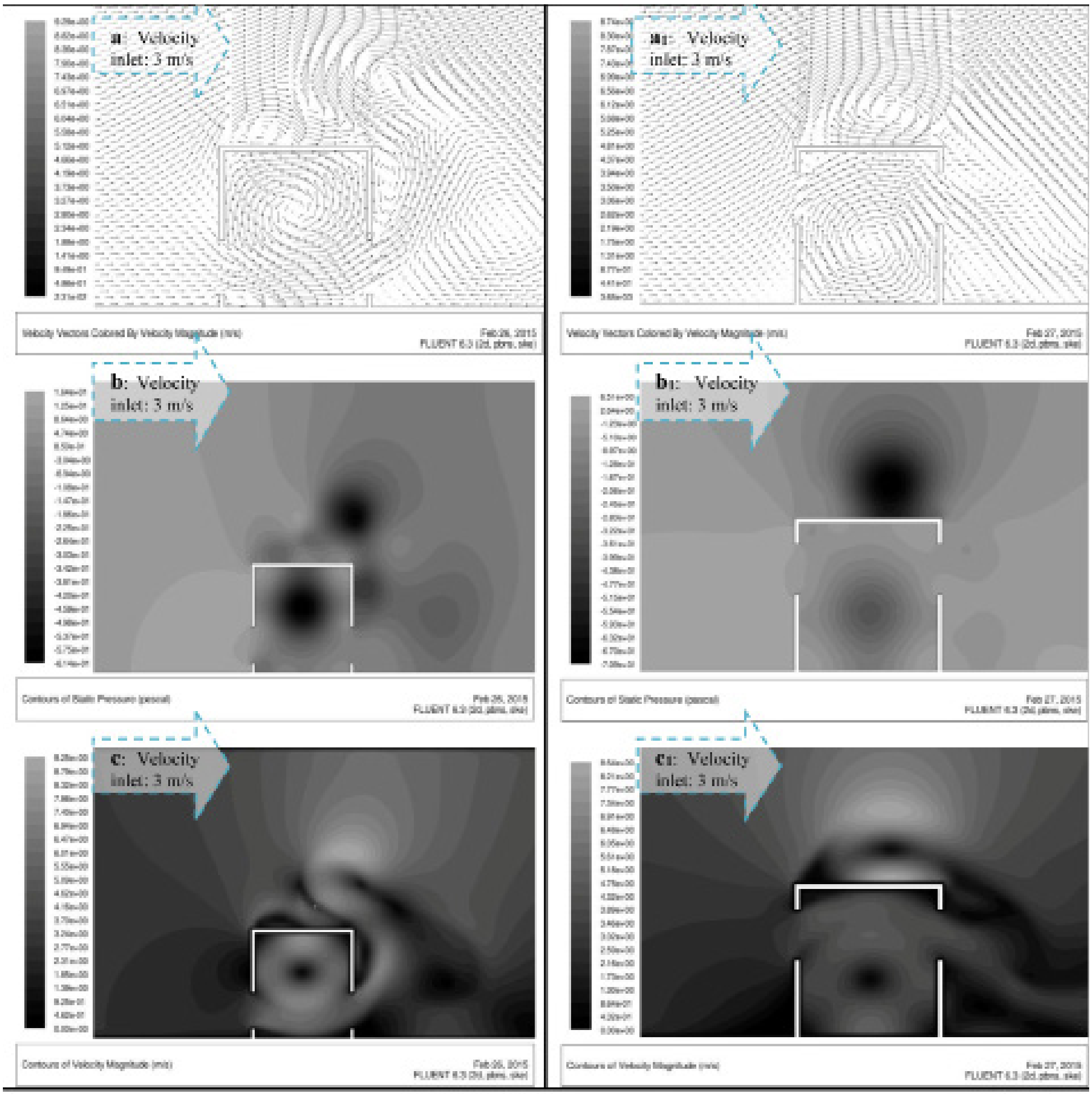


Fig. 8. Different Conditions of Airflow in Case B and C (Wind's Direction:Left to Right, α=90oc); a, b, c in Left: Case B (a. Velocity Magnitude's Vectors, b. Static Pressure's Contours, c. Velocity Magnitude's Contours). a_1, b_1, c_1 in Right: Case C (a_1. Velocity Magnitude's Vectors, b_1. Static Pressure's Contours, c_1. Velocity Magnitude's Contours).

Fig. 10 is dedicated to a condition which the openings are situated in the middle of the opposite walls, right in the center of the walls, in front of each other. Following the simulations of Fig. 10, the natural ventilation in this model is more effective in comparison with the illustrated models in Fig. 9. According to b and b1 in Fig. 10, the indoor airflow around the main stream of the indoor wind (established between 2 openings) is not self circulated. The problem is the vertical gradient of the velocity that is too high and the condition of the natural ventilation is uncontrollable. In this configuration the natural ventilation is completely depending on the outdoor wind's velocity and the indoor air is always replaced. A comparison between 2 graphs of b1 in Fig. 9 and Fig. 10, illustrates that the velocity of the main stream in Fig . 9, exceeds the main stream of Fig. 10. It means that the gradient of the velocity in Fig. 9 is more uncontrollable.

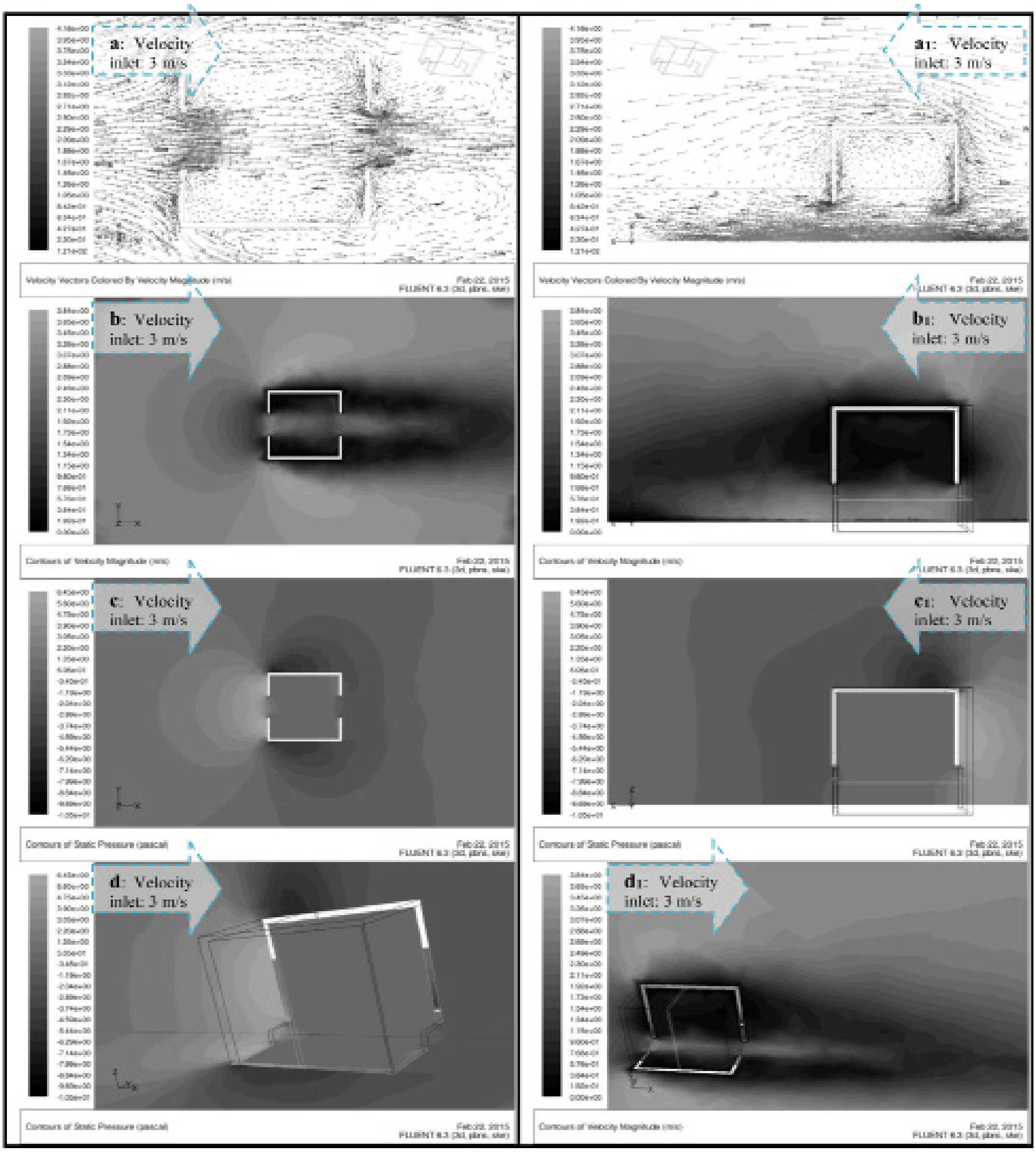


Fig. 9. 3d Simulations of Airflow in Assumed Model (6x6x6m cube). The Openings are Parallel, in the Middle Down. The Outdoor Wind Blows Toward the Opening (α=90oc). a, a_1: Velocity Vectors in Plan (Height of 1.5m) and Section, b,b_1: Contours of Velocity Magnitudes in Plan (Height of 1.5m) and section, c,c_1: Contours of Static Pressure in Plan (Height of 1.5m) and Section, d,d_1: Velocity Magnitude in 3d View. As Illustrated in This Picture, the Openings are Situated in Middle of the Parallel Walls Adjacent to Floor Level.

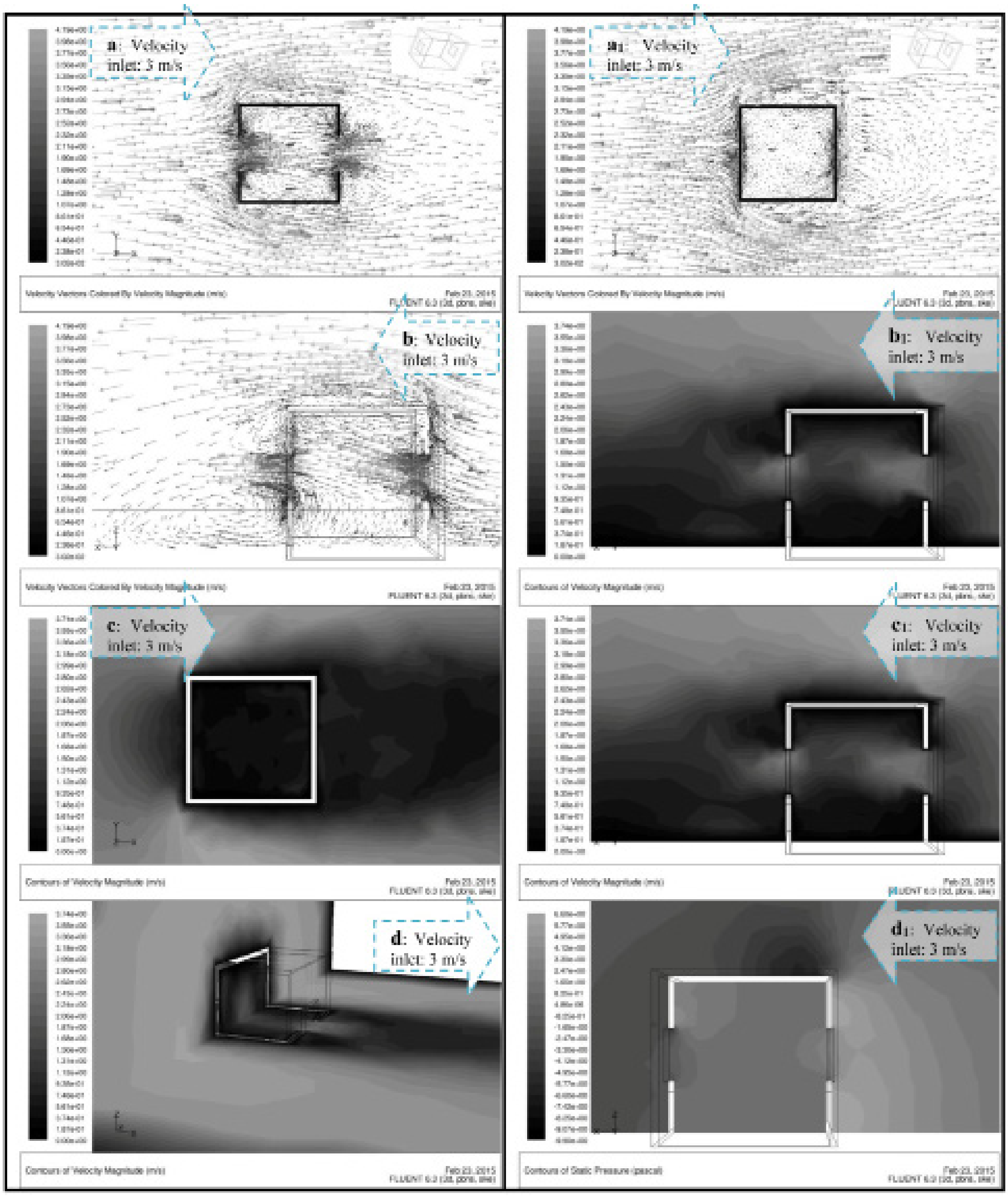


Fig. 10. 3d Simulations of Airflow in Assumed Model (6×6×6m Cube). The Openings are Parallel, in the Middle of Parallel Walls. The Outdoor Wind Blows Toward the Opening (α=90oc). a: Airflow Velocity Vectors at Level of 2.5m, a_1: Airflow Velocity Vectors at Level of 1.5m, b: Velocity Vectors in Section, b_1: Contours of Velocity Magnitudes in Section, c,c_1: Contours of Velocity Magnitude in Plan (Height of 2.5m) and Section, d:Velocity Magnitude in 3d View, d_1: Contours of Static Pressure in Section. As Illustrated in This Picture, the Rectangle Openings are Situated in Center of the Parallel Walls.

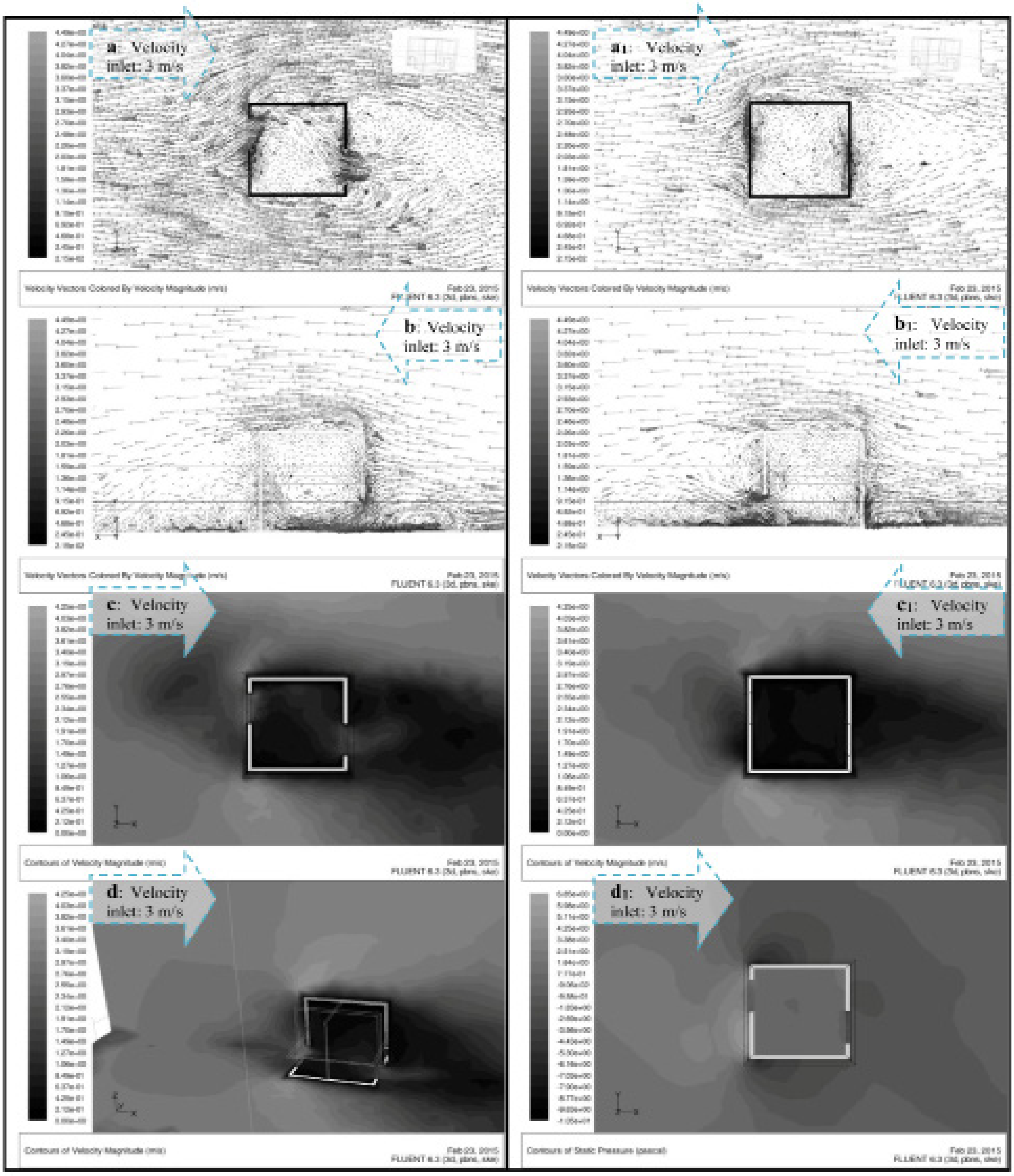


Fig. 11. The Configuration of This Model is Like Fig. 9. The Difference is in the Opening's Position. The Openings are Not in Front of Each Other. They are Parallel, but Situated in 2 Different Corners of Parallel Walls. The Outdoor Wind Blows Toward the Opening (α=90oc). a: Airflow Velocity Vectors at Level of 1.5m, a_1: Airflow Velocity Vectors at Level of 3.0m, b, b_1: Velocity Vectors in Section, c: Contours of Velocity Magnitudes at Level of 1.5m, c_1: Contours of Velocity Magnitude at Level of 3.0m, d: Velocity Magnitude in 3d view, d_1: Contours of Static Pressure in Section.

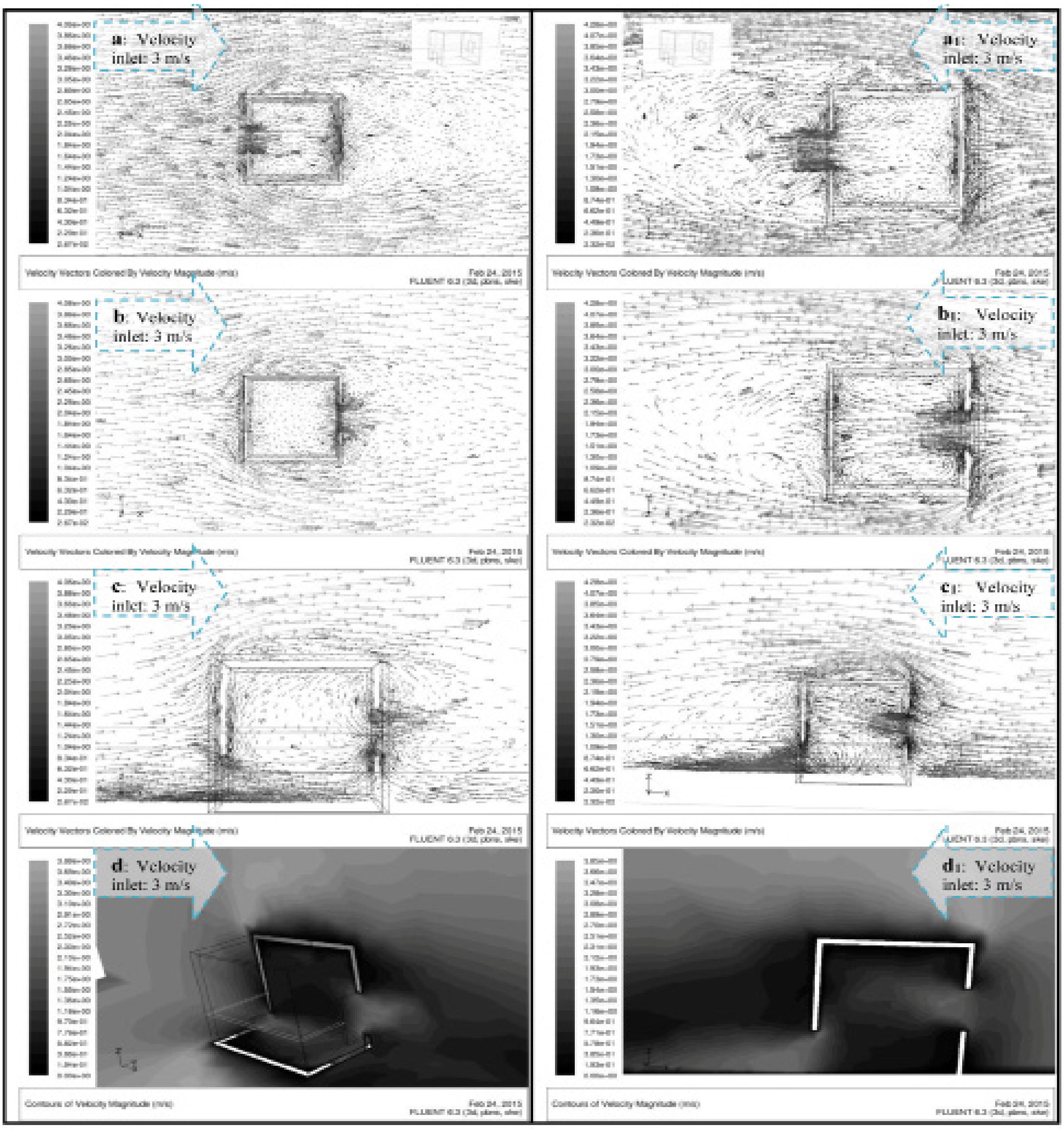
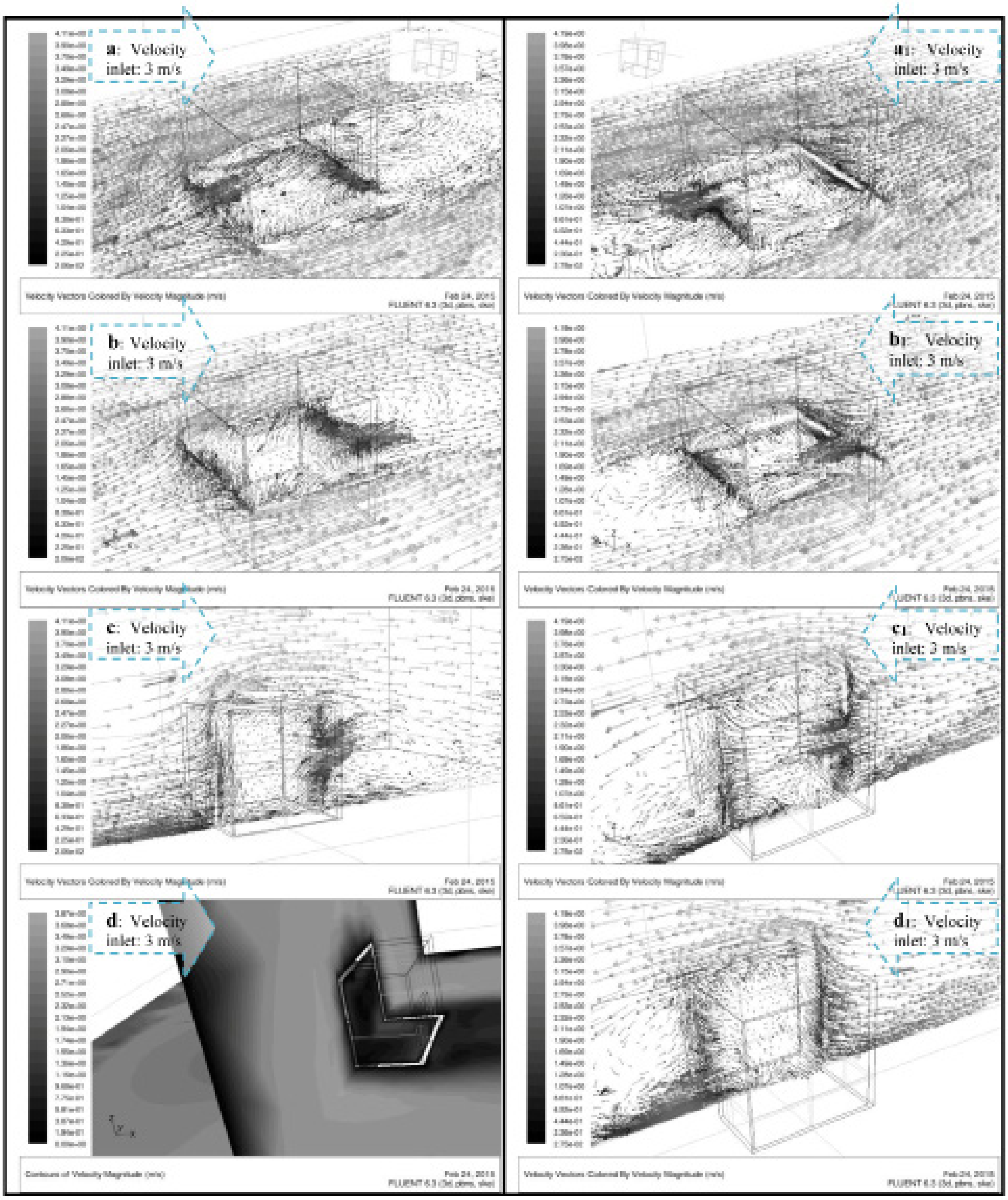


Fig. 12. The Openings in This Configuration are in 2 Parallel Walls. Both are in Front of Each Other at the Middle of the Opposite Walls, but One is Located at the Floor level (±0.0) and the Other is Situated at the Level of +2m. a, b , c, d: Airflow Velocity when the Outdoor Wind is Blowing (α=90oc) from Left to Right (a: Velocity Vectors at Level of 1.5m, b: Velocity Vectors at Level of 3.0m, c: Velocity Vectors in Section, d: 3d Contours of Velocity Magnitude). a_1, b_1, c_1, d_1: Airflow Velocity when the Outdoor Wind is Blowing (α=90oc) from Right to Left (a_1: Velocity Vectors at Level of 1.5m, b_1: Velocity Vectors at Level of 3.0m, c_1: Velocity Vectors in Section, d_1: 2d Contours of Velocity Magnitude in Section).



•
Fig. 13. The Difference Between this Configuration and the Configuration of Fig. 12 is the Location of the Openings: Both Openings are Situated Near the Wall (Distant to Nearest Wall=50cm), but not In front of Each Other. Other Settings are the Same as Fig. 12. a, b , c, d: Airflow Conditions when the Outdoor Wind is Blowing (α=90oc) from Left to Right (a: Velocity Vectors at Level of 1.5m, b: Velocity Vectors at Level of 3.0m, c: Velocity Vectors in Section, d: 3d Contours of Velocity Magnitude). a_1, b_1, c_1, d_1: Airflow Velocity Vectors when the Outdoor Wind is Blowing (α=90oc) from Right to Left (a_1: Velocity Vectors at level of 1.5m, b_1: Velocity Vectors at Level of 3.0m, c_1, d_1: Velocity Vectors in Section).

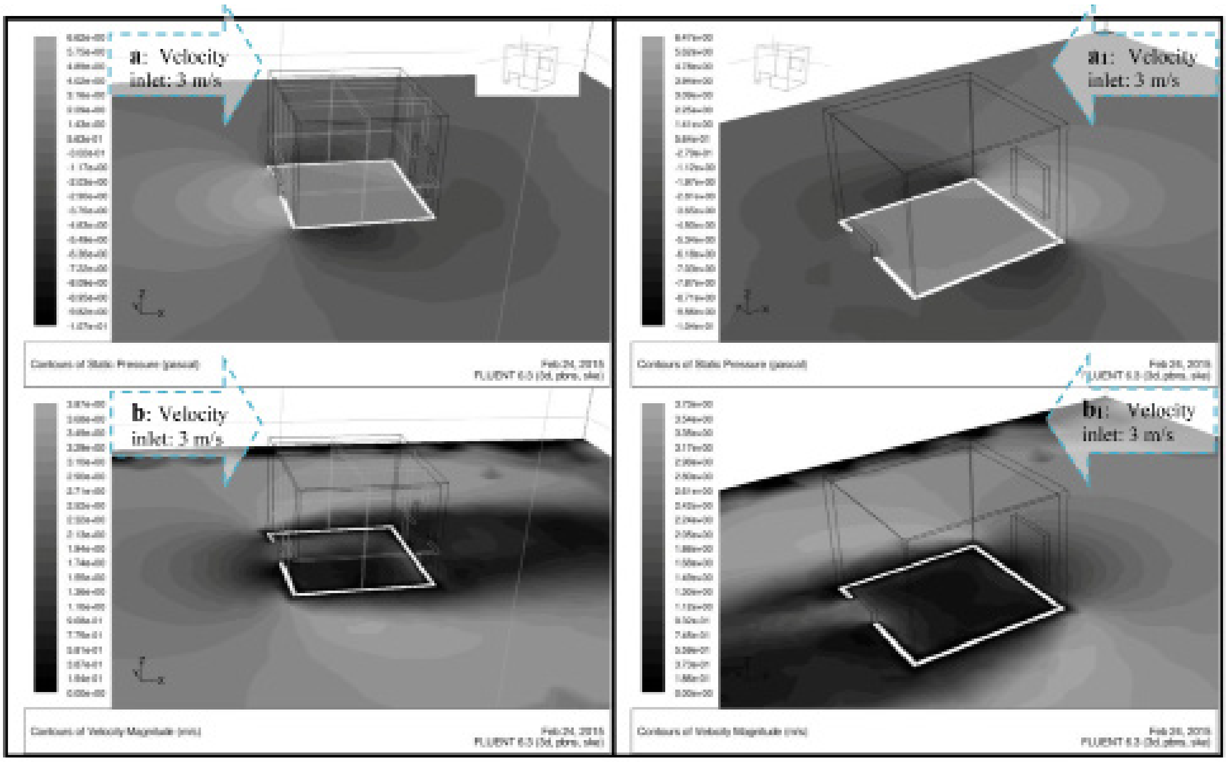


Fig. 14. Velocity Magnitude and Static Pressure in Level of 1.5m from the Configuration of Fig. 13 are Presented Here. a, b: Airflow Conditions when the Outdoor Wind is Blowing (α=90oc) from Left to Right, at Level of 1.5m (a: Contours of Static Pressure, b: Contours of Velocity Magnitude). a_1, b_1: Airflow Conditions when the Outdoor Wind is Blowing (α=90oc) from Right to Left, at level of 1.5m (a_1: Contours of Static Pressure, b_1: Contours of Velocity Magnitude).

The main stream in Fig. 9 is bending toward the floor, but in fig. 10 it is bending a little toward the roof. So the configuration of Fig 10 is more acceptable. The configuration of Fig. 11 is like Fig. 9, but the openings are not in front of each other. However they are parallel, they are situated in 2 different corners of opposite walls. In this condition, the indoor air current is violence: at level of the people, the circulating eddy airflow is established. The outdoor wind passes through the sample cube in the form of “L” shape (adjacent to the walls) and exhausts from the other opening in the opposite wall. An eddy circulating permanent indoor airflow will occur by this current, but the central points of the eddy current have stagnant airflow. Following Fig. 11, the upper parts of the assumed cube is still, and the static pressure is somehow low. So this model is not a good ventilated configuration. Configuration of Fig. 12 is better, but the problem is an uncontrollable stream and its undefined curvature near the side walls. Following the simulations there is a still current near the side walls, but in vertical section the conditions of indoor airflow are good and the acceptable. To solve the problem (the model in Fig. 12), configuration of fig 13 is proposed: the oppose openings of Fig. 12 stick to the side opposite walls (not in front of each other). In this configuration the indoor airflow, depending on the wind direction, has 2 different conditions: the CFD models (Figs. 13 & 14) illustrate that the natural ventilation process in this configuration is improved. This model was simulated in 2 different conditions:

- (windward opening: left to right in Figs. 13 and 14) and exhausted from the upper opening (leeward). a, b, c, and d in Fig. 13 illustrate that there is a calm permanent indoor airflow in almost all inner points of the assumed cube. The problem is the vertical gradient of indoor airflow velocity that is high and violence. Following c in Fig. 13 the main current is near the floor level and following b in Figs. 13 and 14 the indoor airflow near the side walls is circulating with the highest velocity, while the central points of the cube are calmer.
- The outdoor wind blows toward the upper opening (windward opening: right to left in Fig. 13 and 14) and exhaust from the lower opening (leeward). a1, b_1, c_1, and d_1 in Fig. 13 illustrated that there is a permanent indoor airflow in assumed cube and the vertical gradient of the indoor current's velocity is less than the previous condition. Following c_1 (Fig.13) and b_1 (Fig. 14) the indoor airflow is established in all inner points on the assumed cube with an acceptable velocity.

CONCLUSION

This paper provides an alternative approach to macroscopic airflow analysis. This approach is more accurate and more scientific than the traditional conjectural methods of many architects for cross ventilation flow rate in buildings with large openings. However in this research, it is assumed that the inner stream tube flow must be in alignment with the wind direction; in the other word, the wind direction is perpendicular to opening (porous walls) in a cube. Besides, the condition of a cubic cabin with just 2 parallel isometric opposite opening is examined and the other conditions must be considered in future, although nowadays suchlike approaches are very new and necessary to research (following the literature review) and the other aspects are currently under development by many other researchers. Nevertheless here a significant investigation was undertaken: in this method a detailed investigation of the static, dynamic and total pressure distribution for buildings with large openings was pursued. The investigation was performed by using experimental and CFD methods. However for architects, obtaining reliable wind pressure coefficients for sealed buildings can be so difficult, but it is necessary to find a true and reliable way to develop the natural ventilation in buildings. High accuracy in ventilation prediction is needed when the indoor air quality, energy consumption, or thermal comfort are concerned specially by architects.

When the building's openings are very large, it is obvious that an accurate prediction of ventilation flow rates may not be required. In this research, the ventilation flow rate was considered with CFD method, and wind blows with velocity of 3m/s toward the porous windward wall, for a wall porosity of 10%. Generalization of the results develops some better solutions for wind driven natural ventilation through buildings with large openings and establish useful design guidelines.

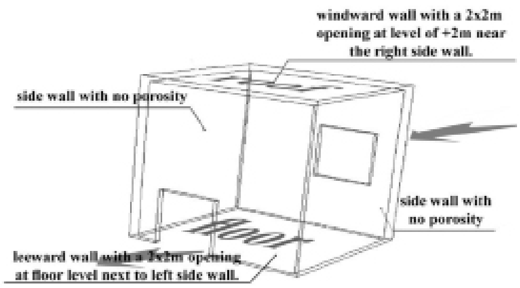


Fig. 15. A good model for overall natural ventilation in occupant's level when the upper opening is windward.

As mentioned before, in this article a cubic cabin was considered using CFD techniques, based on wind driven boundary conditions. Following the CFD models, configuration of fig 13 that the outdoor wind is blowing toward the upper opening (windward), is the best condition to establish an overall indoor airflow by natural ventilation (10% Porosity in each 2 opposite walls; the side walls and roof have no Porosity). This model is useful when the overall natural ventilation is needed particularly in occupants' level. Following this model (Figs. 13 and 14) the upper parts of the room (upper level above the upper opening) is circulating in a form of eddy current. So whatever the height of the sample cube is, the upper window's position is important, because the air current above the upper opening is circulating in an eddyThe outdoor wind blows toward the lower opening form. The upper air flow in this model is not important because it's not useful for the occupants to concern, but the level of the upper window is so important to establish efficient natural ventilation in each height's level (occupant's positions) that is needed. This model is demonstrated in a form of schematic figure in Fig. 15.

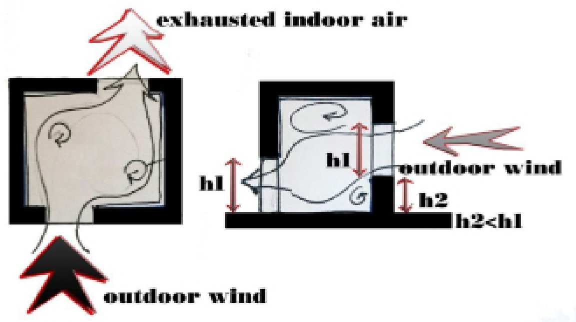


Fig. 16. Best Position of the Openings in Opposite Walls. Left: Plan Mode, and Right: Section Mode.

When the outdoor wind is too intensive, the use of the other model is proposed. The configuration of fig. 10 can be suitable, because of the indoor air velocity at the occupant's level: it is considerably calmer than the outdoor wind. All of the mentioned methods can be generalized for more porosity and other dimensions. It is schematically illustrated in a sketch of Fig. 16. So it is notable that to establish a good indoor airflow some below points must be regarded:

1. Both parallel openings must not be installed in front of each other.
2. The openings must be installed next to the side walls.
3. The openings must never be installed in the middle of the wall. This condition is worst.
4. Both parallel openings must not be installed in the same level.
5. The upper opening must not be installed near the ceiling. It is notable that the maximum difference between the levels of the installed openings (OKB) must be about the openings height.

After generalizations, in conclusion, to upgrade the natural ventilation in sealed cabins and depending on the necessity of the flow rate, each tested models can be useful. A designer can select the favorite model depending on the outdoor wind's velocity and the necessity of the project and the indoor airflow rate that is required.

NOMENCLATURE

A: effective area (m^2)

Cp: wind pressure coefficient (–)

CFD: computational fluid dynamic method

h: height of an opening (m)

p: pressure (Pa)

v: reference wind speed (m/s)

x, y, z: coordinate (m)

REFERENCES

Akins, R. E., & Cermak, J. E. (1976). *Wind Pressures on Buildings*, CER76-77EA-JEC15, Fluid Dynamics and Diffusion Laboratory, Colorado State University, CO.

Amidpoor, M. (2010). *Investigation of Domestic Flueless Space Heaters Effects on Indoor Air Quality,* Research Project, Iran: Khaje Nasir University, Department of Mechanical Engineering.

Arens, A. D. (2000). *Evaluation of Displacement Ventilation for Use in High-ceiling Facilities*, Master's Thesis, Department of Mechanical Engineering, Massachussets Institute of Technology, Cambridge, USA.

Aynsley, R., Melbourne, W., Vickery, B. (1977). *Architectural Aerodynamics*, London: Applied Science Publishers, pp. 254.

Castro, I. P., & Robins, A.G. (1977). The Flow around a Surface-Mounted Cube in Uniform and Turbulent Streams, *J. Fluid Mech*, 79 (2), 307–335.

Chan, C., & Li, Y. (2001). A Simple Design Method for Mixed-Mode Ventilation Systems in Australian Carparks, In: *Proceedings of the Fourth International Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings*, Changsha, China.

Chiu, Y., & Etheridge, D. W. (2007). External Flow Effects on the Discharge Coefficients of Two Types of Ventilation Opening, *Journal of Wind Engineering and Industrial Aerodynamics*, 95, 225-252

Foster, A., Barrett, R., James, S. J., & Swain, M. J. (2002). Measurement and Prediction of Air Movement through Doorways in Refrigerated Rooms. *International Journal of Refrigeration*, 25, 1102–1109.

Ghali, K., Ghaddar, N. & Ayoub, M. (2007). Chilled Ceiling and Displacement Ventilation System for Energy Saving: A Case Study, *International Journal of Energy Research*, 31, 743–759.

Ghobadian, V. (2012). *Tarahi Eghlimi*. Tehran: Tehran Uni. Ltd.

Groat, L., & Vang, D. (2004). *Research Methods in Architecture* (A. Einifar, Trans.), a. Tehran: TU Ltd.

Hoang, M. L., Verboven, P., De Baerdemaeker, J., & Nicolaı¨, B. M. (2000). Analysis of Air Flow in a Cold Store by Means of Computational Fluid Dynamics. *International Journal of Refrigeration,* 23, 127–140.

Hu, Z., & Sun, D. W. (2000). CFD Simulation of Heat and Moisture Transfer for Predicting Cooling Rate and Weight Loss of Cooked Ham During Air-blast Chilling Process. *Journal of Food Engineering*, 46(3), 189–198.

Hussein, M., & Lee, B.E. (1980). *An Investigation of Wind Forces on Three Dimensional Roughness Elements in a Simulated Atmospheric Boundary Layer*, BS 55, Department of Building Science, University of Sheffield, UK.

Jensen True, J.P. Sandberg, M., Heiselberg, P., V. Nielsen, P. (2003). Wind Driven Cross Ventilation Analysed as a Catchment Problem and as a Pressure Driven Flow, *Int. J. Vent*. 1, 89–101.

Kasmaei, M. (2011). *Eghlim & Memari*. Tehran: Khak Ltd.

Kato, S., Murakami, S., Takahashi, T., Gyobu, T. (1997). Chained Analysis of Wind Tunnel Test and CFD on Cross Ventilation of Large-Scale Market Building, *Journal of Wind Engineering and Industrial Aerodynamics*. 67–68, 573–587.

Knoll, B., Phaff, J.C., & de Gids, W.F. (1995). Pressure Simulation Program, in: *Proceedings of the 16th AIVC Conference,* Palm Springs, USA, 18–22 September, vol. 1, 233–242.

Kurabuchi, T. Ohba, M., Endo, T., Akamine, Y., & Nakayama, F. (2004). Local Dynamic Similarity Model of Crossventilation, Part 1: Theoretical Framework. *Int. J. Ventilation*, 2 (4), 371–382.

Liddament, M.W. (1986). *Air Infiltration Calculation Techniques: An Application Guide*, AIVC.

Loomans, M., & Mook, F. (1995). *Survey on Measuring Indoor Airflows FAGO*, Report 95.25.W., Eindhoven University of Technology Sweden.

Mariotti, M., Rech, G., & Romagnoni, P. (1995). Numerical Study of Air Distribution in a Refrigerated Room. *In Proceedings of the 19th International Congress of Refrigeration* (1st ed.), (98–105). Den Hague: International Institute of Refrigeration.

Mirade, P. S., & Daudin, J. D. (1998a). Numerical Simulation and Validation of the Air Velocity Field in a Meat Chiller. *International Journal of Applied Science and Computations*, 5(1), 11–24.

Mirade, P. S., Daudin, J. D., & Arnaud, G. (1995). Simulation En Deux Dimensions De L'ae´Raulique De Deux Tunnels De Re´Frige´Ration Des Viandes. *Revue Internationale du Froid*, 18(6), 403–412.

Mirade, P. S., Kondjoyan, A., & Daudin, J. D. (2002). Three-dimensional CFD Calculations for Designing Large Food Chillers. *Computers and Electronics in Agriculture*, 34, 67–88.

Novoselac, A., & Srebric, J. (2002). A Critical Review on the Performance and Design of Combined Cooled Ceilings and Displacement Ventilation Systems, *Energy and Building*, 34, 497–509.

Oakley, G. (2002). A Combined Day Lighting, *Passive Stack Ventilation & Solar Heating System*, Ph.D. Thesis, University of Nottingham.

Rahaei, O. (2014). Effects of Architectural Somatic Variables on Mixed Air Conditioning Systems' Efficiency in Industrial Buildings, *Armanshahr Architecture & Urban Development Journal*, 12, 69-81.

Rees, S.J., & Haves, P. (2001). A Nodal Model for Displacement Ventilation and Chilled Ceiling Systems in Office Spaces, *Building and Environment*, 36, 753–762.

Sandberg, M. (2003). An alternative view on the theory of cross ventilation, in: The Proceedings of the *First International Workshop on Natural Ventilation, Tokyo*.

Scott, G. (1994). Computational Fluid Dynamics for the Food Industry. *Food Technology International Europe*, 49–51.

Scott, G., & Richardson, P. (1997). The Application of Computational Fluid Dynamics in the Food Industry. *Trends in Food Science and Technology*, 8, 119–124.

Seifert, J. (2005). *Zum Einfluss Von Luftstro Mungen Auf Die Thermischen Und Aerodynamischen Verha Ltnisse In Und An Geba Uden*. Dissertation: TU Dresden.

Seifert, j. Li, Y., Axley, J., Rösler, M. (2006). "Calculation of Wind-driven Cross Ventilation in Buildings with Large Openings", *Journal of Wind Engineering and Industrial Aerodynamics*, 94, 925–947.

Xia, B., & Sun, D. W. (2002). Applications of computational fluid dynamics (CFD) in the food industry: a review. *Computers and Electronics in Agriculture,* 34(1–3), 5–24.

Xie, J., Qu, X. H., Shi, J. Y., & Sun, D. W. (2006). Effects of design parameters on flow and temperature fields of a cold store by CFD simulation. *Journal of Food Engineering*, 77(2), 355–363.