Windows Factors Impact on Air Speed and Quality Inside Architectural Spaces

O. A. Masood[a], N.M. Guirguis[b], M. I. Abd Al-Hady[c], A. A. Fahmi[a].

[a] Faculty of Industrial Education, Suez University, Egypt.

[b] Housing & Building Research Center, P.O. Box 1770, Cairo, Egypt.

[c] Faculty of Industrial Education, Suez University, Egypt and SUC, Oman.

Abstract

Nowadays, healthy building is one of the contemporary architectural trends. Natural ventilation is one of the most important passive systems, which improves the building environment. Windows play very important role in this trend; windows factors like its position, shape, area, slop, wind direction and quantity are the variables, which affect on the level of comfortably inside buildings. The main objective of this paper is to determine the impact of these factors on the air quality and speed inside architectural spaces.

Keywords: Healthy Buildings, Natural Ventilation, Windows Variables.

INTRODUCTION

Normal air flow is the usage of wind and temperature variations to make airflows in and through buildings. This airflow can be used both for ventilation and for passive cooling strategies. Normal air flow is frequently strongly desired by using building occupants, specifically if they have a few controls over it, as with operable windows. Studies have proven that most occupants will conveniently tolerate a much wider range of ambient conditions in the event that they have such control [1]. As Givoni [2] mentioned that cooling of the building's mass at night by the enhanced ventilation enabled it to absorb heat from the ventilation air during the day-time hours, enough to reduce the indoor day time temperature and improve the comfort of the occupation even in buildings ventilated day and night.

Power call for in cooling-dominant climates may be decreased by means of implementation of normal air flow like a passive cooling method. Therefore, suitable assessment and prediction gear are necessities for correctly introducing normal ventilation in building design. Further, the quick appearance of multi-storey buildings may speed up power consumption, in particular in instances of beside the point building layout [3].

Normal air flow is a powerful method for lowering power use in buildings. The effect is particularly substantial for homes with excessive internal warmth generation, including business workplace buildings. Therefore, certainly ventilated office homes have become an increasing number of popular in Japan [4].

Herbal ventilation represents one of the demanding situations in inexperienced homes layout because the most important parameter that reflects the first-class of building design is the thermal consolation within the indoor surrounding conditions [5].

Normal ventilation is maximum appropriate for cooling and offering clean air in residential building in mild seasons for higher indoor air quality and thermal comfort. The natural ventilation overall performance is tormented by a combination of internaland external elements. Outside elements consist of the vicinity, the orientation, the winning wind speeds and the constructing forms of the residential improvement, which can be difficult to constraints past, manage of web page planners and designers. While for internal elements just like the openings designs and window sorts, web page planners and designers are usually given loose hand for a proper layout. They have an effect on of every internal elements at the herbal ventilation performance is therefore of interest to them.

There is some obstacles of using natural ventilation such as the outdoor temperature is higher than the indoor temperature, presence of external pollutions or dust, external noise and conflict with construction standards.

F. Muhsin, et al studied normal ventilation performance in multi-storey housing the use of roof void. This examine showed that the provision of void can improve normal ventilation functioning in multi-storey housing [7].

C.F. Gao and W.L. Lee studied have an impact on of window styles on normal ventilation of residential homes in Hong Kong using CFD software. The studied outcomes provide useful facts for destiny designs of residential homes for better normal ventilation [8].

L. Moosavi, et al, studied the effects of normal air flow on atria, indoor thermal conditions; they use distinct air flow strategies which may be applied in atria as helped air flow strategies. They decided that the outlet opening length is the maximum influential parameter that affects both indoor thermal condition and ventilation behaviors of atria and accordingly reduces energy usage load [9].

Guohui Gan, studied layout of natural air flow structures for lots kinds of building is based on buoyancy force. Consequences confirm that wind would adversely affect the air flow patterns within the building designed with buoyancy-pushed normal ventilation. Wind can at the same time help and oppose buoyancy in the windward and leeward wings, correspondingly [10].

A. B. Daemei, et al, they studied two types of Natural Ventilation using CFD mode. They concluded Natural Ventilation would help informed designing based on clean energies [11].

SIMULATION EXPERIMENTS

Software Program

A software computer package (ANSYS CFD Flotran) [6] was used to investigate the air velocity and temperature distribution inside the test model in the form of velocity contours and vectors. The program is a three dimensional, one of that makes use of the limited detail approach which makes use of the k-ε disturbance model and solves the Reynolds equations, the electricity equation and the equations for disturbance strength and its wastefulness.

In recent work, the boundary situations specified that the stream speed at all of the solids surfaces become zero (fulfilling the real viscous fluid configuration). Also the

imminent speed profiles have been preordained by logarithmic law. Three general assumption were assumed , the first that the fluid was Newtonian, the flow was a single-phase one and the solution domain was of constant geometry, in addition, the flow was steady incompressible and the body forces were neglected.

Study Hypotheses

A single room with dimensions (length 5m, width 4m, height 3m) has been studied. It has one inlet and one outlet openings. Different opening shape, size and locations were studied. The effect of model orientation to wind direction has been also studied. The average air velocity inside the room for all cases is calculated. The sill height is constant.

The dimensions of the solution domain are located around the room (length 50 m, width 20 m, height 12 m).

Note that, the air direction is fixed in all study cases as shown in Figure (1).
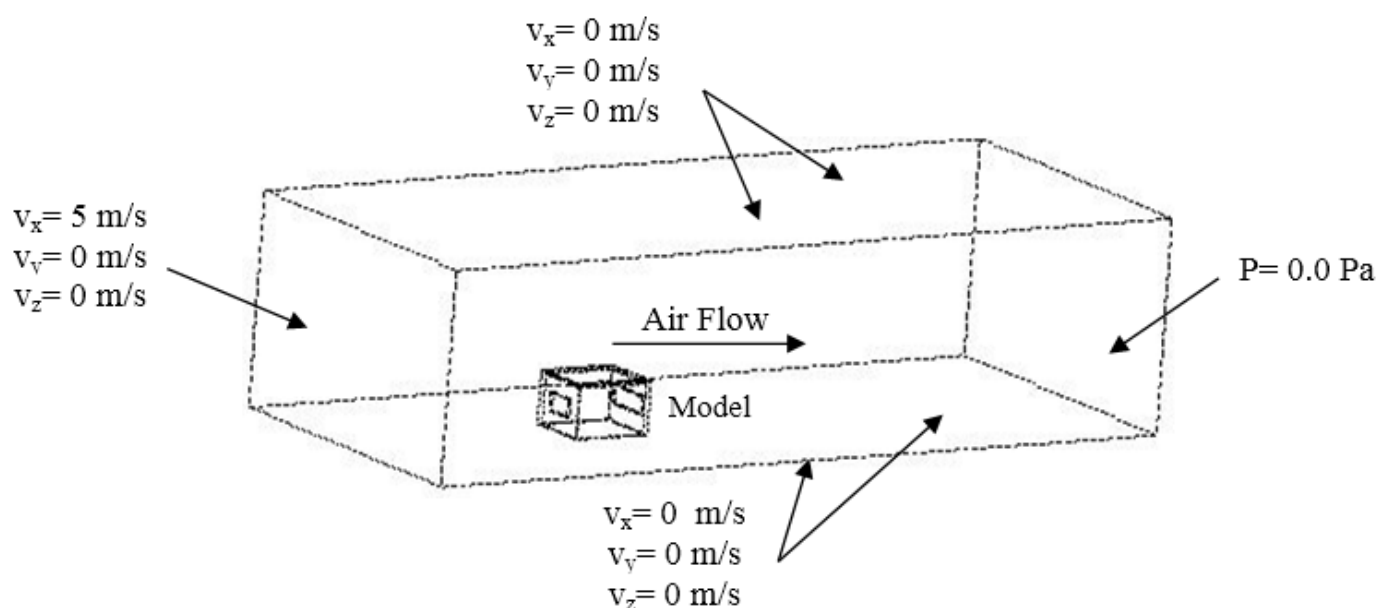
Figure 1: Boundary conditions applied on Model Domain.

Case Studied

Different model cases were simulated using the ANSYS CFD software program package as follows:
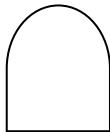
   a) Opening shape.

   b) Variation of inlet and outlet Window to Wall ratio (WWR).

   c) Inlet and outlet openings locations.

   d) Model orientation to air direction.

The air speed and the sill height are constant for all cases.

Opening shape

In these Cases, six different window shapes were used (circular – square – pentagon – rectangular – equal leg arch - peak pentagon). The inlet and outlet area, the sill height of the window and incoming air velocity 5m/s are fixed in all models.Table (1), show different openings shape for this simulation.

Table 1: Opening shape

Name	Circle	Square	Pentagon	Rectangle	equal leg arch	peak pentagon
Shape	⬤	▢	⬠	▭	⌂	⌓

Variation of inlet and outlet window to wall ratio (WWR)

In these cases, the windows to walls area ratio of the inlet and outlet openings were changed as follow:

   a) WWR_{in} =10%, WWR_{out} = 10%, 15%, 20% and 25%.

   b) WWR_{in} =15%, WWR_{out} = 10%, 15%, 20% and 25%.
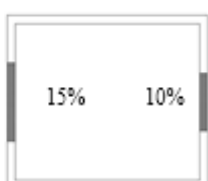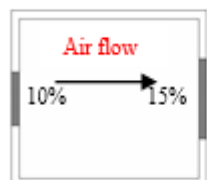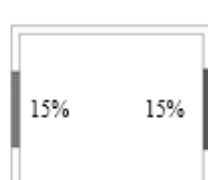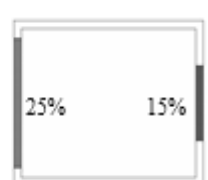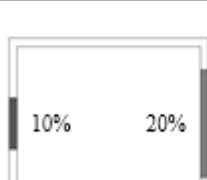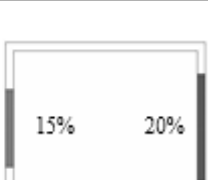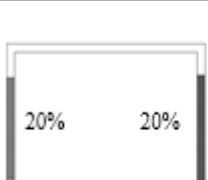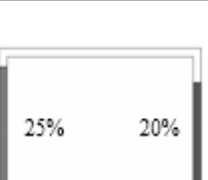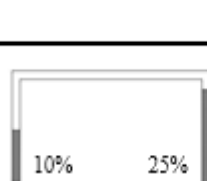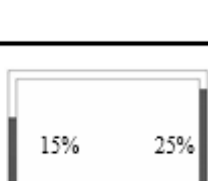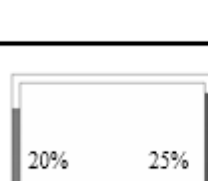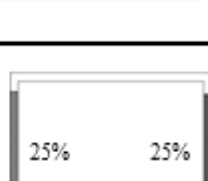
   c) WWR_{in} =20%, WWR_{out} = 10%, 15%, 20% and 25%.

   d) WWR_{in} =25%, WWR_{out} = 10%, 15%, 20% and 25%.

Where; WWR_{in} & WWR_{out} are the inlet and outlet are ratio respectively.

Table (2) shows different model cases for different WWR_{in} & WWR_{out}

Table 2: Inlet and outlet window to wall ratio (WWR) for different model cases.

Case NO.	Plan	Case NO.	Plan	Case NO.	Plan	Case NO.	Plan
1	WWR 10% 10%	5	15% 10%	9	20% 10%	13	25% 10%
2	Air flow 10% → 15%	6	15% 15%	10	20% 15%	14	25% 15%
3	10% 20%	7	15% 20%	11	20% 20%	15	25% 20%
4	10% 25%	8	15% 25%	12	20% 25%	16	25% 25%

Inlet and outlet openings locations

In this case, two types of different openings locations were examined, as shown in table (3). The first cases is (Two Openings in opposite Walls), in which, four models with different openings positions were examined. The second cases is (Two Openings in adjacent Walls), and in these cases eight models were examined with different openings positions. Note that, the inlet and outlet openings area and sill height are the same for all cases. The air velocity at the inlet domain is the same for all cases, 5 m/s which represent the average velocity during the year for Cairo city.

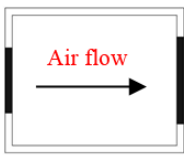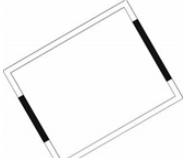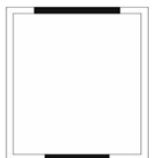**Table 3:** Locations of inlet and outlet openings in the room model, $WWR_{in}= 10\%$, $WWR_{out}=15\%$, (plan view).
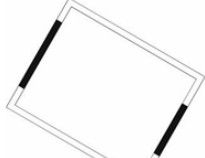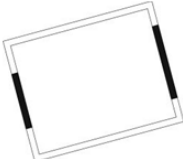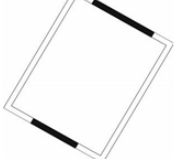
Case NO.	opposite walls	Case no.	adjacent walls	Case NO.	adjacent walls
1	Air flow →	5		9	
2		6		10	
3		7		11	
4		8		12	

Model orientation to air flow direction.

In this case, eight models orientations were simulated from angle 0º to 180°. Take into account that the horizontal projection is fixed in all the eight study models and the area of the incoming and outgoing air openings and the height of the window setting are fixed as well with change in the inclination of the incoming air opening.

Table 4: Models orientation cases relative to air flow direction, WWR$_{in}$= 15%, WWR$_{out}$=20%, air velocity= 5 m/s, (plan view).

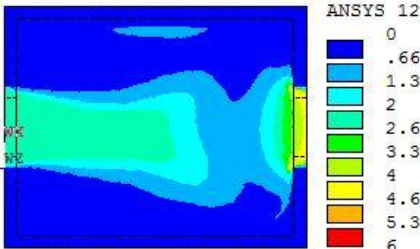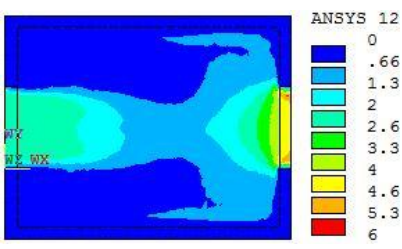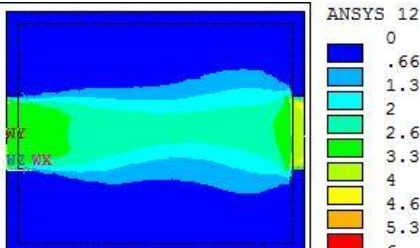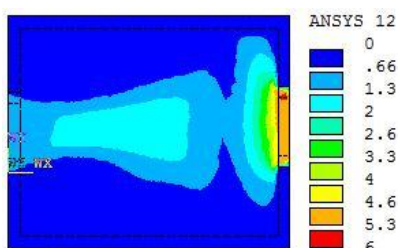| Case NO. &ang.	Plan	Case NO. &ang.	Plan	Case NO. &ang.	Plan	Case NO. &ang.	Plan
1 (0º)	Air flow →	3 (30º)		5 (90º)		7 (150º)	
2 (15º)		4 (60º)		6 (120º)		8 (180º)	

RESULTS AND DISCUSSIONS

Opening Shape

A simulation of the six different openings shape (3D) models with the same area were applied using the CFD ANSYS program, and calculating the average internal air velocity for each case study. Table (5) shows velocity contours inside the models of different opening shape and figure (2) represents average velocity inside the building models. The circular openings (case 1), achieve the best air distribution and maximum average air speed inside the room than other cases due to this circular shape reduces the air turbulence at the inlet and outlet openings. While the peak pentagon gives lower average air speed due sharp edges which increases the turbulence flow around it.

Table 5: Contours of air flow distribution inside the tested models cases for different openings shapes.

Case NO.	V$_{av}$/ V$_{air}$	Plan view	Case NO.	V$_{av}$/ V$_{air}$	Plan view
1	0.1983	Circular	4	0.1724	Rectangle
2	0.1742	Square	5	0.1565	Equal Leg arch

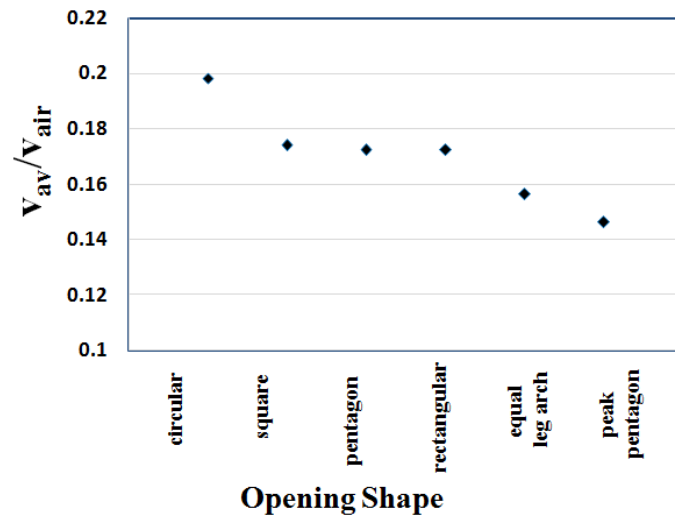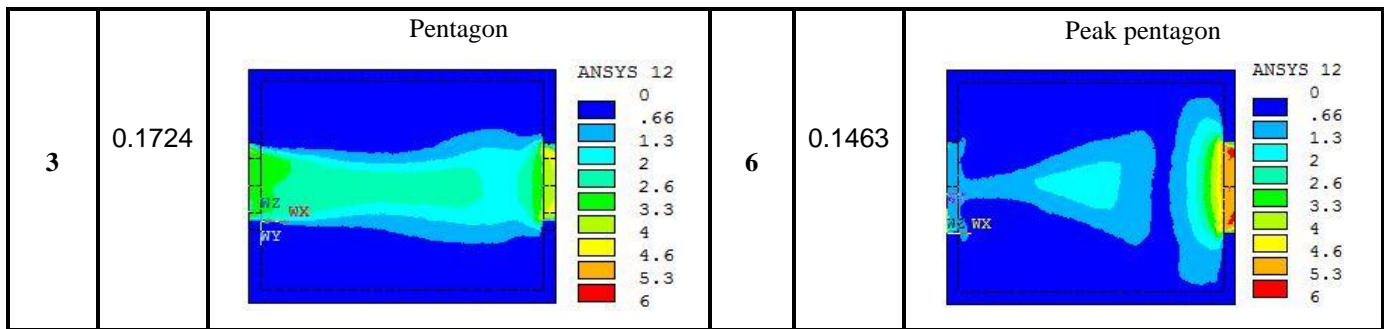|		Pentagon			Peak pentagon
3	0.1724		**6**	0.1463	

Figure 2: Average air velocity inside the room (v_{av}) relative to external air velocity (v_{air}) for different openings shapes.

Variation of Inlet and Outlet Window to Wall Ratio (WWR)

Different Window to Wall Ratio were simulated using the ANSYS software program. The WWR of the inlet and outlet openings were taking the values 10%, 15%, 20% and 25% as mentioned in table(2). Figure (3) shows the average indoor air velocity v_{av} relative to external air flow v_{air} for different inlet and outlet WWR. It is clear the as WWR_{out} increases the indoor velocity increases due to the reduction of turbulence flow through the outlet openings. The case WWR_{in} =15% and WWR_{out} = 25% gives the best air flow inside the room. Table (6) shows the velocity contours distribution inside the room. It is clear the as the outlet opening area increase the air flow distribution inside the room is better.
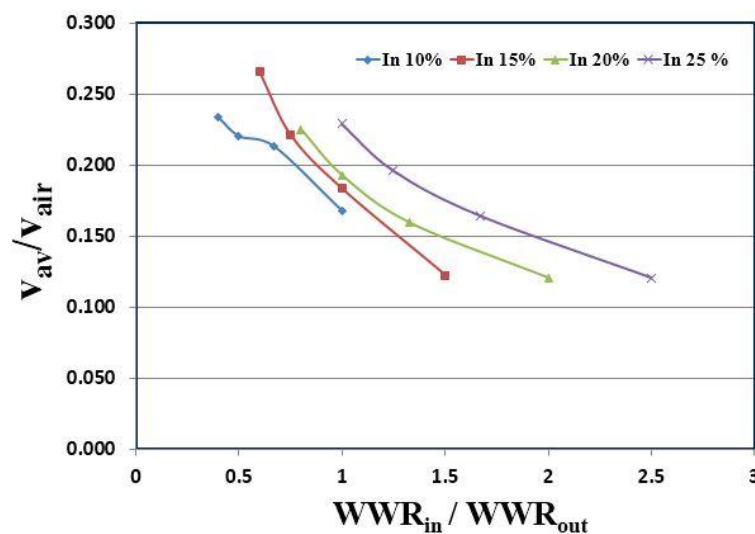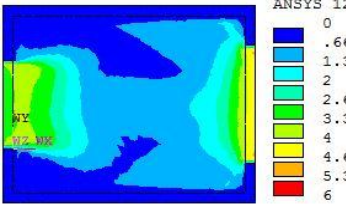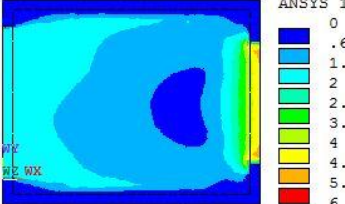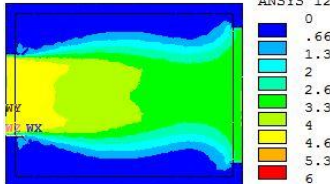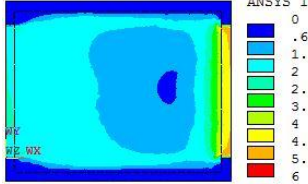

Figure 3: Variation of average indoor air velocity for different Inlet and Outlet Window to Wall Ratio (WWR)

Table 6: Contour of air flow distribution inside the tested models cases for different WWR.

Case NO.	V_{av}/V_{air}	Plan view	Case NO.	V_{av}/V_{air}	Plan view
1	0.168		9	0.121	
2	0.213		10	0.159	
3	0.221		11	0.193	
4	0.233		12	0.225	
5	0.122		13	0.120	
6	0.183		14	0.164	
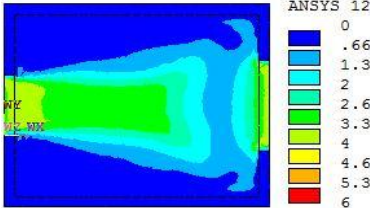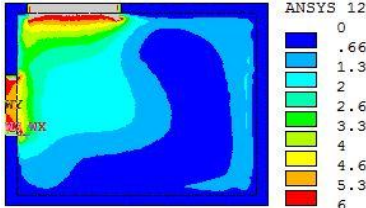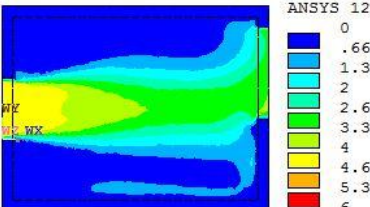
7	0.222		15	0.196	
8	0.266		16	0.229	

Inlet and Outlet Openings Locations

Different models with different openings locations were simulated (as mentioned in table (3). First, openings location at opposite walls and second openings locations at adjacent walls.

Table (7) shows Contour of air flow distribution inside the tested models cases for different inlet and outlet openings locations. It is clear that as the two openings location apart from each other there is good air flow distribution inside the room.
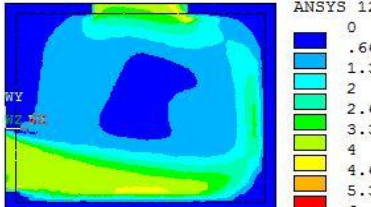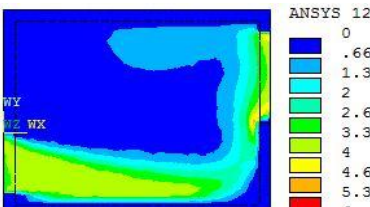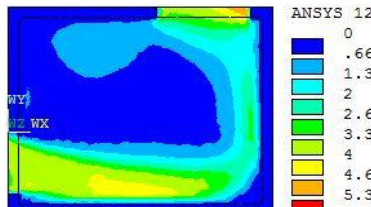
Table 7: Contour of air flow distribution inside the tested models for different inlet and outlet openings locations.

Case NO.	V_{av}/V_{air}	Plan view	Case NO.	V_{av}/V_{air}	Plan view
1	0.2099		7	0.3078	
2	0.1856		8	0.2231	
3	0.2021		9	0.2227	
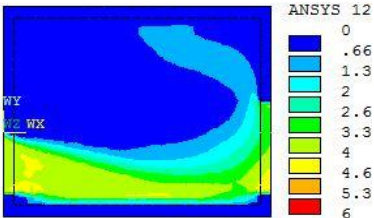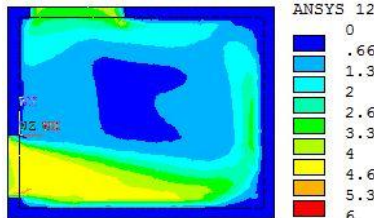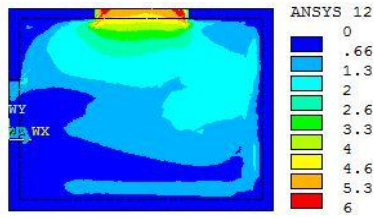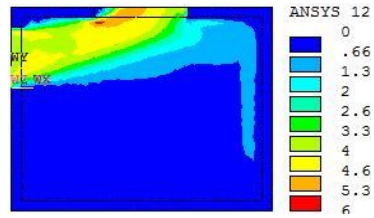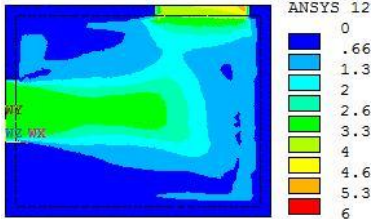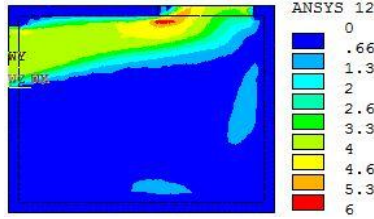
4	0.1732		10	0.2444	
5	0.2164		11	0.1822	
6	0.2102		12	0.1909	

Figure (4) shows the average air velocity inside the room (v_{av}) relative to external air velocity (v_{air}) for different openings locations. It is clear the case 7 gives the higher indoor air velocity but bad air distribution since the openings are in adjacent walls and very close to each other. Cases 2, 4, 11 and 12 give lower air speed since these openings locations increase the stagnation contours area.
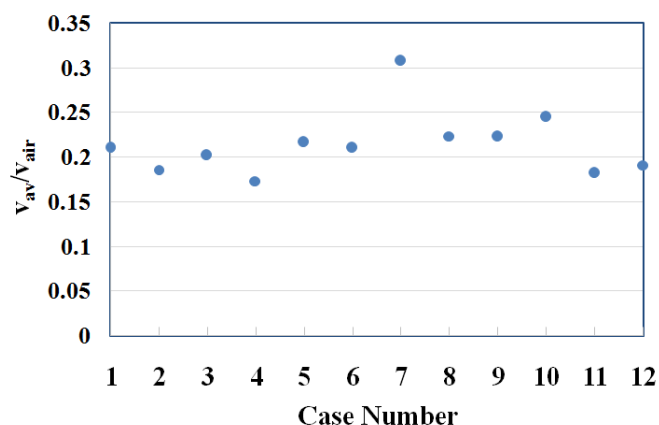

Figure 4: Average air velocity inside the room (v_{av}) relative to external air velocity (v_{air}) for different openings locations.

Model Orientation to Air Direction.

Figure (5) shows the average air inside the room for different model orientation, the WWR_{in} =15% and The WWR_{out}=20%.

The fig. shows that the air velocity increases slightly from 0º to 30º and decreases to a minimum air velocity value at angle 90º due to the openings is parallel to the air flow direction and increase slightly at angles more than 90º in which the outlet opening directed to the upwind. In comparison of average air velocities of cases 0º-180º, 30º-150º and 60º-120º the 0º , 30º and 60º angles gives higher velocity than angles 180º ,150º and 120º respectively, this results matches with results in fig. (3) that the smaller window area in the windward cases gives higher air velocity than cases the bigger area in the windward.
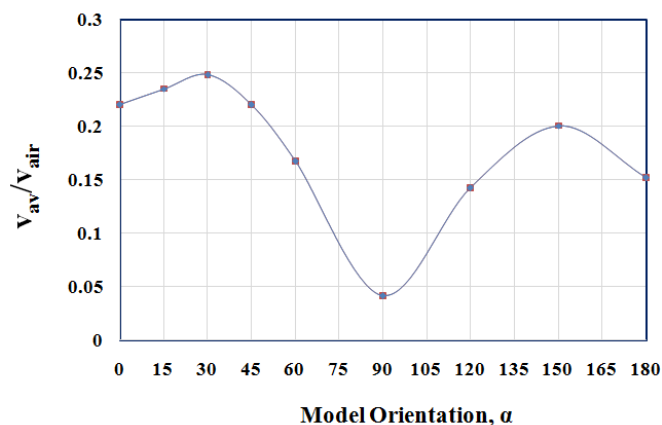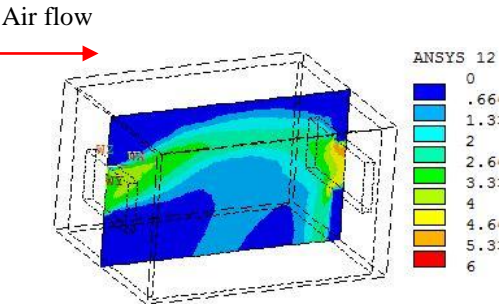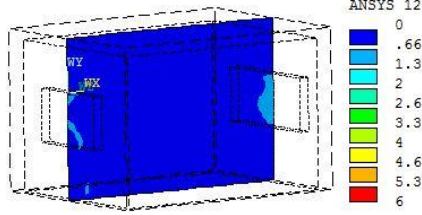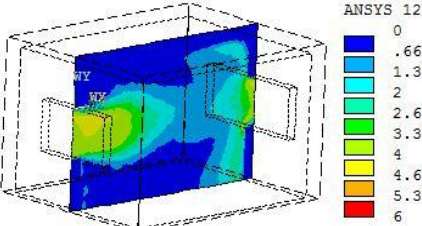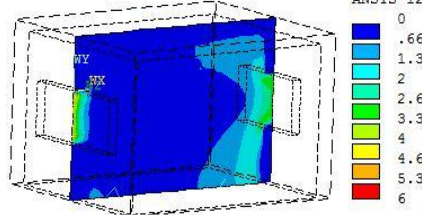

Figure 5: Average air velocity inside the room (v_{av}) relative to external air velocity (v_{air}) for different model orientations.

Table (8) illustrates air flow distribution contours inside the tested models for different models orientations. Cases 0º, 15º and 30º gives better air distributions than other cases. The air flow inside the room for orientation angle 90º gives the lower value since the air flow is parallel to the two openings.

Table 8: Contour of air flow distribution inside the tested models for different models orientations.

Case NO.	Angle	Velocity contour	Case NO.	Angle	Velocity contour
1	0º		5	90º	
2	15º		6	120º	
3	30º		7	150º	
4	60º		8	180º	

CONCLUSION

This study can provide a number of basic suggestions to activate the process of natural ventilation within spaces, which can be used to provide a good design for the ventilation within the buildings spaces.

- The ANSYS CFD software program can be used to evaluate the air flow around and inside buildings.
- The circular shape opening increases the air flow rate in the buildings due to the reduction of turbulence flow.
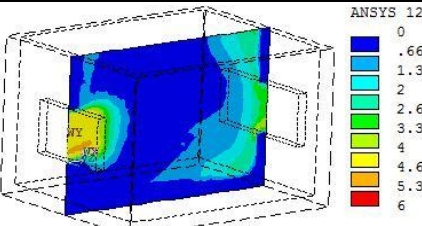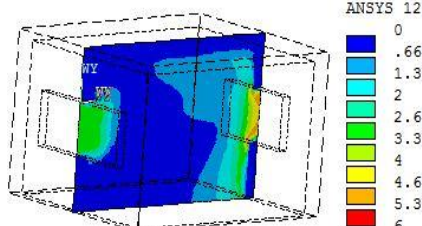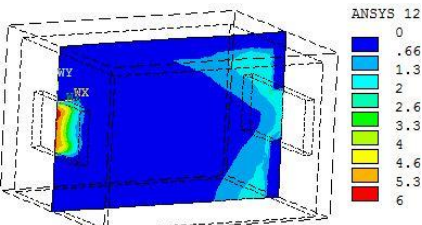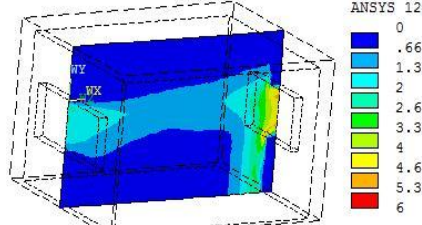- The air flow rate increases as the outlet area increases which reduces the effect of turbulence at the edge separation.
- The locations of openings at opposite walls give good air distributions inside building spaces, than the adjacent walls cases.
- The building orientation affect the air flow inside buildings in which the angles of 0º, 15º, 30º and 45º with the air flow direction give a good air flow than other air flow directions.
- The building orientation 90º in which the openings are parallel to the main air flow direction gives lower air flow than other directions that needs some types of vertical fences at the openings to direct the flow inside the building.

NOMENCLATURE:

Ratio of opening area to wall area, Window to wall ratio = $A_{window} / A_{wall} * 100$ % (non). : WWR

A_{window} :Window area, (m^2)

A_{wall} :wall area, (m^2)

Window to Wall area for inlet opening, (non) : WWR$_{in}$

Window to Wall area for outlet opening, (non) : WWR$_{out}$

Air velocity outside the room, (m/s). :V$_{air}$

Average air velocity inside the room, (m/s). : V$_{average}$

v_x, v_y, v_z: Air velocity components in x,y,z directions, (m/s).

CFD: Computational Fluid Dynamics

REFERENCES

[1]   Allard, Francis, ed., Natural Ventilation in Buildings, James and James, London, 1998, pp.45.

[2]   Baruch, Giovini, Climate consideration in building and urban design", Van Nostrand Reinhold, (1998), pp. 187.

[3]   S.Omrani, V.Garcia-Hansen, B.CapraR.Drogemuller, "Natural ventilation in multi-storey buildings: Design process and review of evaluation tools", ELSEVIER, Building and Environment, Volume 116, 1 May 2017, Pages 182-194.

[4]   Mika Nomura, Kyosuke Hiyama, "A review: Natural ventilation performance of office buildings in Japan, ELSEVIER, Renewable and Sustainable Energy Reviews, Volume 74, July 2017, Pages 746-754.

[5]   Ghada Elshafei, Abdelazim Negm, Mahmoud Bady, Masaaki Suzuki, Mona G lbrahim, Numerical and experimental investigations of the impacts of window parameters on indoor natural ventilation in a residential building, ELSEVIER, Energy and Buildings, Volume 141, 15 April 2017, Pages 321-332.

[6]   ANSYS FLOTRAN manual (2002), http//: www.ansys.com

[7]   F. Muhsin, W. F. Yusoff, M. F. Mohamed, A. R. Sapian, "The Effects of Void on Natural Ventilation Performance in Multi-Storey Housing", Buildings 2016, 6, 35; doi:10.3390/buildings6030035.

[8]   C.F.Gao and W.L. Lee, "Influence of window types on natural ventilation of residential buildings in Hong Kong", https://www.research.net/publication/2546256.

[9]   L Moosavi, N Mahyuddin, N A Ghaf, M A Ismail," Thermal performance of atria: An overview of natural ventilation effective designs", Renewable and Sustainable Energy Reviews, Contents lists available at ScienceDirect, ELSEVIER, Renewable and Sustainable Energy Reviews 34 (2014) 654–670.

[10]   Guohui Gan, "Interaction between Wind and Buoyancy Effects in Natural Ventilation of Buildings", The Open Construction and Building Technology Journal, 2010, 4, 134-145.

[11]   A. B. Daemei, A. K. Limaki, H. Safari, "Opening Performance Simulation in Natural Ventilation using Design Builder (Case Study: A Residential Homee in Rasht)", 3rd International Conference Power and Energy Systems Engineering, CPESE 2016, 8-12 September 2016, Kitakyushu, Japan, Science Direct, ELSEVIER, Energy Procedia 100 (2016) 412-422.