INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 7, Issue 1, 2016 pp.37-48
Journal homepage: www.IJEE.IEEFoundation.org

CFD model using for natural ventilation study in the local building of hot-humid climate in Vientiane Capital, Lao PDR

Pakasith Phonekeo[1], P. Amparo López-Jiménez[2], Ignacio Guillén Guillamón[1]

[1] Applied Physics Department. Universitat Politécnica de Valencia. Spain.
[2] Hydraulic and Environmental Engineering Department. Universitat Politécnica de Valencia. Camino de Vera s.n. 46022. Valencia, Spain.

Abstract
In Laos, the research of the relationship between local climate and the building is still less, therefore the attentiveness of passive design in building is also rare and depreciates. This study will be some basic guideline for studying about air movement that impact to the building in Laos which has the hot-humid weather all year. This study analyzes how the physical characteristics of air flow affect to the local accommodation building in Vientiane Capital of Laos by using Computational Fluid Dynamics (CFD) techniques. Furthermore, the study aims to investigate the air movement appearance inside the room influenced by the environment around building, particularly the main factors of human comfort including the air movement, the age of air, temperature and pressure.

Keywords: Passive ventilation; Air movement; Computational Fluid Dynamics (CFD); Hot-humid climate.

1. Introduction
The design limitation of the accommodation building in Vientiane capital confronts the area design problem. As an example the room will be add in the building as much as possible and influence them cannot achieve good ventilation that impact to use the air-conditioner all the time to gain the comfort zone in the room, however it spends a lot of electricity. Most of buildings in Vientiane Capital do not meditate for the energy conservation as well, they always were handle only active ventilation and do not try to adopt any passive ventilation.
Therefore, giving emphasis in building design for energy saving is very important, particularly in the primary design for building space planning to use the best natural ventilation. However, the weather in Vientiane Capital is mostly hot and humid, but we can design any room for good air movement and find the way to reduce the heat inside the room on suitable time or in the hottest period. Moreover this will be the way to improve the good life quality for the residents and also help to raise about the energy saving in the building.
The study analyzes of natural ventilation, mainly in 2 factors including the wind and buoyancy [1], and there are many standard and guideline to define the fundamental theory and criteria of natural ventilation in building [2, 3]. The main reason of ventilation in room is the comfort zone for the occupants, replacing the old environment by the fresh air from the outside building and cool down the heat inside the room

[4]. The good ventilation is to provide the air movement in all activity area in the room, considered as the age of air in the particular location and display as the average time elapsed [5, 6]. However, the air movement depends upon many factors in both of the outside and inside the room such as the obstacle wall. The relative position of walls or furniture may increase the amount of the age of air efficiency [7]. In Addition, the environment parameter are also important including the natural wind velocity outside affect to the air behavior and influence to the thermal comfort zone of the occupants in the room [8-10], and also relate to the different pressure and the thermal mass that occur in whole room [11-13].
The prediction of air movement appearance inside the room by using computational fluid dynamic (CFD) can be described in many research studies, it can simulate and demonstrate the relationship of the environment factors [14], it can clearly perform the air flow direction and its behavior [15]. The CFD codes can effectively improve the speed and accuracy in predicting the air flow and thermal environment for natural ventilation studies [16].

1.1 Objective

This research is about the study of the environment affect around a particular building, including the wind direction, the wind velocity, temperature and pressure. The models observe the wind appearance effective to each room in each building. Consecutively, to study the physical characteristic of air movement inside the room, to investigate and to find the room for the passive ventilation. As the objective we will also investigate the air movement occurred when change some factors such as the bathroom position inside the room, the wind direction and velocity that flow through the opening inside the room.

1.2 Research limitation

This study is the experiment for the natural ventilation simulation related to the building, the main factor is the characteristic of the natural wind affecting to the outside and inside building (do not consider the energy consumption). The experiment uses the computer software to simulate the general environment around the building in Vientiane Capital (Laos) by using the climate data in Nong Khai province (Thailand). As the software does not provide the micro-climate data in Vientiane Capital, therefore in the simulation, the software will use Nong khai weather database instead Vientiane Capital (the distance from Vientiane Capital is 30 kilometers far from Nong Khai and located in the same longitude and latitude). However, the research still uses some micro-climate data in Vientiane Capital such as the temperature, the wind direction and velocity.

1.3 The climate of Vientiane Capital

Vientiane Capital is located in latitude 17°58'N and the longitude 102°33'E and 121 meters elevation above sea level, the seasonal wind have 2 directions including the wind from north in winter and the wind from south in summer, the average of velocity is about 5mph to 10mph. Vientiane Capital has a tropical wet and dry(savanna climate, Koppen-Geiger classification) and hard rainfall from July to September with annual precipitation average 1648.7mm which is equivalent to 1648.7 liters/m2. The mean of annual temperature is 25.9°c, the hottest is in April and May with the max temperature 40'c and the coldest is in December and January with the min temperature 16°c. The amount of sunshine is average 2420 hours per year.
For the comfort zone of southeast Asian people, it is possible that the topography and the different culture of dressing impact to the different comfort feeling [17]. Based on the results of [17], the Asian people have the thermal comfort range between 24°c - 27°c with humidity 50% - 70 % for air velocity 0.2 m/s [18] and they will feel more cooler when the wind velocity increase.

2. Research methodology

2.1 General objective and CFD simulation program

The study of indoor air movement can quantify the airflow rate in the real environment condition. Many researches are conducted in the real site and the real model, however they spend a long time to study with more expenses. Computational Fluid Dynamics (CFD) is used and has agents worldwide; therefore it is a reliable program. DesignBuilder is a very common use for design model. To calculate energy saving, it can calculate the environmental performance data such as: energy consumption, carbon emissions, comfort conditions, daylight luminance, maximum summertime temperatures and HVAC component sizes. Moreover, it can predict the result experiment and provide the report saving in electric lighting due to use of natural daylight, calculating heating and cooling equipment according to the environmental climate data. Furthermore, it can compute and simulate the detailed design of HVAC and natural ventilation systems including the impact of supply air distribution on temperature and velocity distribution within a room using CFD by providing in the result reports simple to observe and distinguish.

2.2 Numerical method

The CFD technique is used to solve and analyze the fluid dynamics problem. The basic concept is to consider the continuity of fluid flow by defining the model experiment to the spatial domain as small elements, known as the volume mesh or grid, then use the solution algorithms to solve the dynamics equation by using Euler Equation for inviscid flow and Navier-Stokes Equation for viscid flow. Furthermore, the mesh element has been defined in 2 term including the unstructured mesh and structured mesh, however when the model problem has the high dynamic and large scale, it will be used the adaptive mesh refinement method. Generally, the navier-stokes laws can solve both laminar flow and turbulent flow but for the large eddy flow using the Reynolds-Averaged Navier-Stokes Equation (RANS) with model test k-ε or the Reynolds Stress Model to solve them [17].

The basic conservation equations are provided to solve the numerically in CFD software, based on three equations conservation [18], including the equation of mass conservation, known as the continuity equation (1) is

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

The equation for momentum conservation (2) is

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \tag{2}$$

And the equation for total energy conservation (3) is

$$\frac{\partial}{\partial x_i}(\rho u_i h) = \frac{\partial}{\partial x_i}(k\frac{\partial T}{\partial x_i}) + u_i \frac{\partial p}{\partial x_i} + \tau_{ij}\frac{\partial u_i}{\partial x_j} \tag{3}$$

where: ρ is the density, u_i is the velocity component in the i direction, p is the static pressure, x_i is a Cartesian coordinate, τ_{ij} is the stress tensor, g_i is the gravitational acceleration in the I direction, h is the static enthalpy, k is the thermal conductivity, and T is the temperature [13].

The CFD numerical method used by DesignBuilder software which relate to the set of those equation above, particularly describe the main of the environment factors including the conservation of heat, temperature, mass, momentum and where k-ε turbulence model is used. The equations comprise a set of coupled non-linear second-order partial differential equations having the following general form, in which ϕ represents the dependent variables (4)

$$\frac{\partial}{\partial t}(\rho\phi) + div(\rho u\phi) = div(\tau\ grad\phi) + S \tag{4}$$

(A) (B) (C) (D)

Term *(A)* represents the rate of change, term *(B)* represents convection, term *(C)* represents diffusion and S term *(D)* is a source term [19].

3. Mathematical model

3.1 Geometry

The case study is located in Vientiane capital, the building is a hotel resort and be the local Laos style. The whole hotel has 3 main buildings consist of the rooms for tourists and the necessary facilities for hotel (Figure 1). The initial experiment will be conduct outside the building to investigate the environment around the building, particularly the wind direction that influences toward the each room in each building. Afterwards, according to the wind result data in the initial test. Subsequent the models are done to see the atmosphere inside the room and investigates the physical air-movement and summarize them to find the best room position and the best natural ventilation.

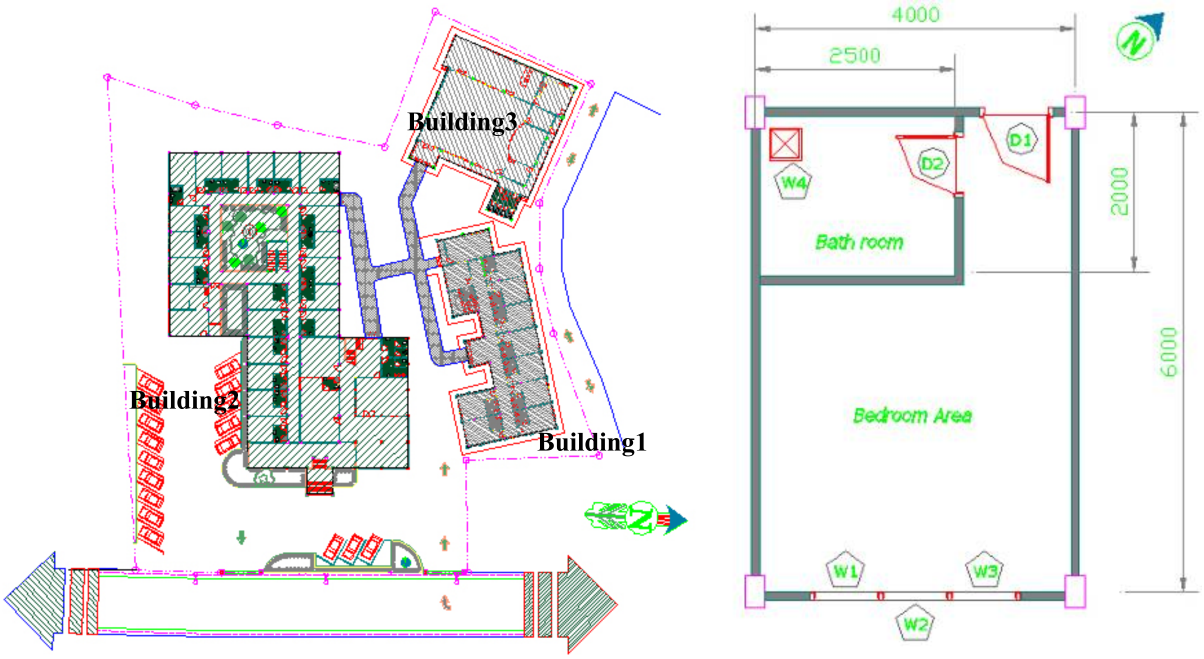


Figure 1. The building and the room will use for the virtual model in CFD simulation

3.2 Boundary condition and physics

Firstly, the whole building will be created as the virtual model in DesignBuilder software for the initial experiment to see the environment achieved surrounding whole building. The location has set in the program is located in Vientiane capital, Laos. However, in DesignBuilder does not provide the climate data in Laos, therefore the experiment will use the climate data in Nong Khai Province, Thailand (according both province are so close each other and belong to the same latitude and longitude). However some macro-climate data still need gathered from the real site such as wind direction and the velocity. Due to micro-climatic condition, the prevailing wind comes from the north in winter and come from south in summer (Figure 2).

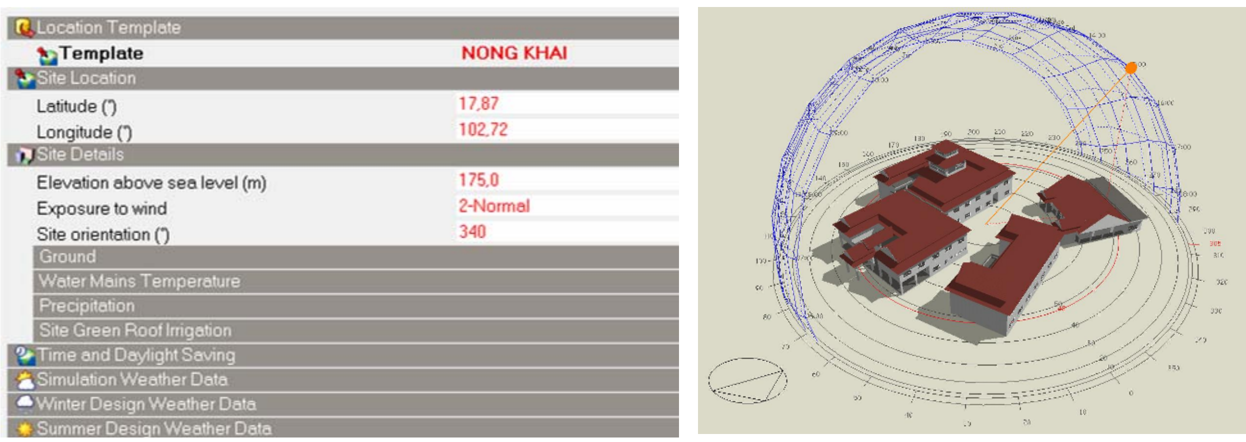


Figure 2. The software parameter inputs and the virtual model in CFD DesignBuilder

Therefore, the CFD boundary, the wind velocity is set for 7mph, moreover the temperature in the hottest day has an average amount around 35°c (Data from the meteorological section).
The next step consists on testing the air-movement in the each room by creating virtual model of 2 rooms that were selected to represent as all of room in hotel building. The room size is 4m wide, 6m long and 3.5m high include the bathroom inside, the room have big window 2.40m x1.35m, was divided for 3 small window (3 x 0,80m x 1,35m) and the middle one cannot open. There is a door in the innermost wall inside the room size 0.90m x 2.10m. On the wall partition between the bedroom area and bathroom has the door interpose them and in model was assumed as a hole (the door is always open). The last opening is the ceiling ventilation fan size 0.30m x 0.30m inside the bathroom. For the environment parameters, some data is required such as temperature of the internal surfaces of the window, the average air temperature in the area, the driven air temperature, the angle of wind come in to the room (discharge angle), the opening position, etc. (Figure 3).

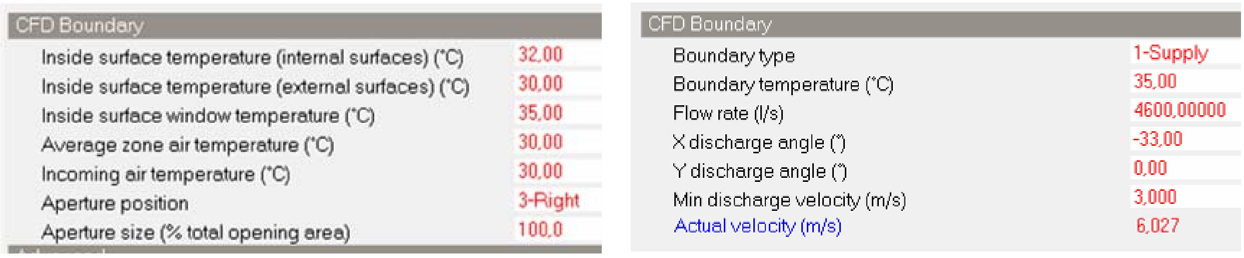


Figure 3. The parameters used in DesignBuilder to determine the environment for the CFD simulation

The experiment will conduct in four options, following the opening inlet and outlet of air -movement, as depicted in Table 1.

Table 1. The testing options following by the opening characteristic

Opening	Option1	Option2	Option3	Option4
W1	Open (for inlet)	Open (for inlet)	Open (for inlet)	Open (for inlet)
W2	Closed	Closed	Closed	Closed
W3	Open (for outlet)	Open (for inlet)	Open (for inlet)	Open (for inlet)
W4	Closed	Closed	Open (for outlet)	Open (for outlet)
D1	Closed	Open (for inlet)	Closed	Open (for outlet)
D2	Open (Hole)	Open (Hole)	Open (Hole)	Open (Hole)

Apart from the testing by changing the factors of the opening, the experiment also was conducted more by adjust the other factors including the velocity value and the changing of the bathroom position inside the room (Figure 4). All of the modeling result will be investigated to make conclusion to find the best option of room design for this resort-hotel in Vientiane capital.
As shown in Figure 4, the room will be simulated following the bathroom is changed inside and the wind direction go through the room opening. Therefore, we conduct the experiment in 4 different ways, as indicated in Table 2.

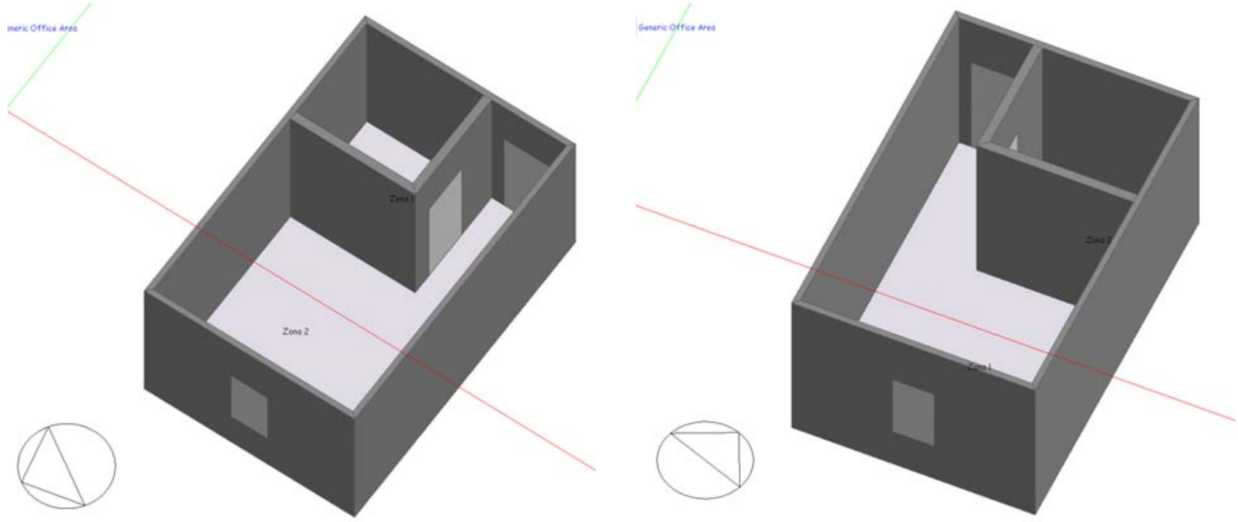
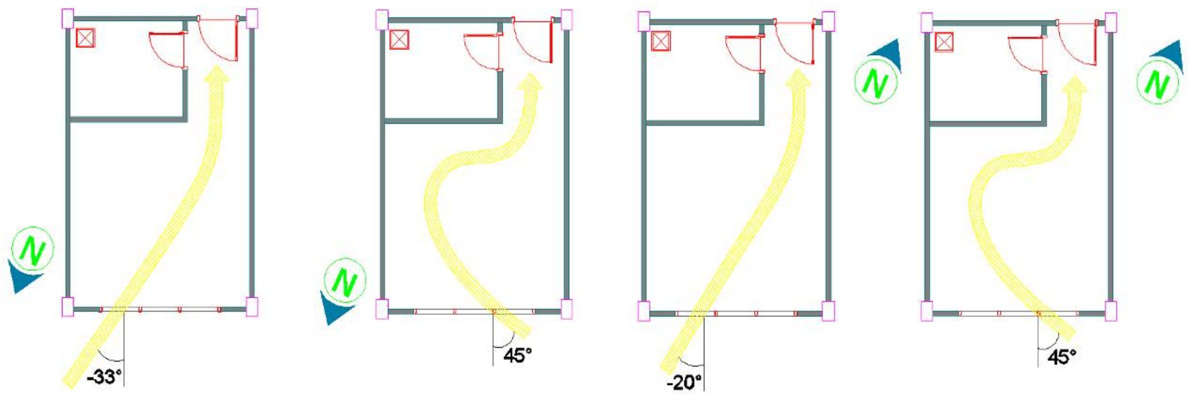


Figure 4. The virtual model for testing about the different position of the bathroom

Table 2. The physical of wind direction that flow into the room

Type of Room			
Building 1		Building 2	
α = -33°	**α = 45°**	**α = -20°**	**α = 45°**



4. Analysis of results

4.1 The simulation results

The initial experiment is the observation of the air movement appearance around whole building, the simulation will illustrate the air flow stream, the velocity, temperature and pressure. Moreover the testing also indicates the characteristic of wind influence to each room in the building. the simulation result display the prevailing wind direction from north and south can flow through the building gap and every building can achieve the wind due to the each buildings have enough distance between them. As the Figure 5 indicates the wind vectors flow from the front building to the building behind. The velocity simulation shows that the wind velocity from north is a little bit faster than the wind from south and the wind velocity that impact to the room on second floor is faster that the wind velocity that impact to the room on first floor.

The subsequent experiment is providing the primary test result above to study the characteristic of air movement inside the room in hotel building, the testing are conducted by changing many factors and many options. The simulation result explanation will clarify in four options, as detailed in Table 3 and Table 4.

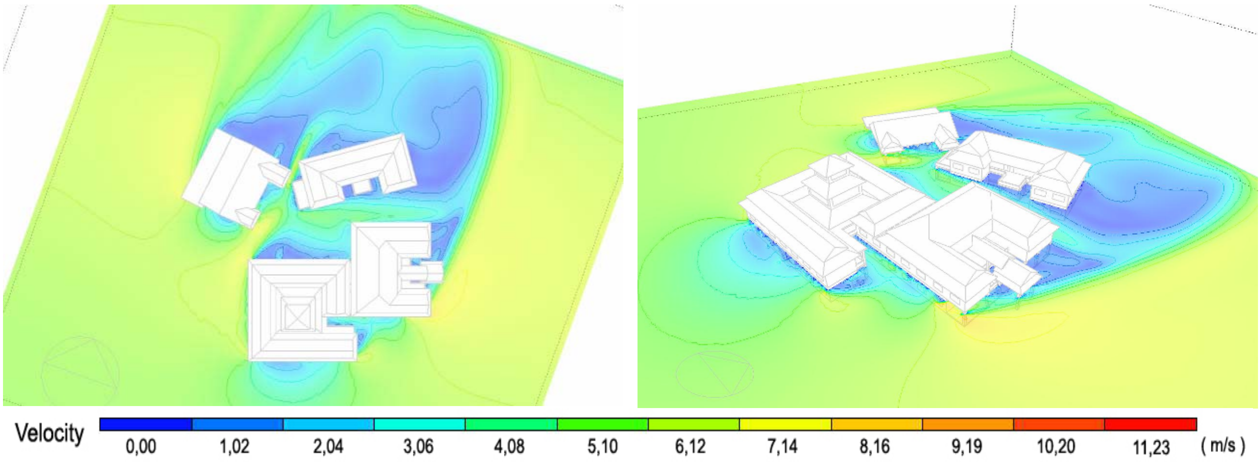


Figure 5. The simulation of the wind flow stream affect to whole building

Table 3. The simulation result for the age of air in each option experiment in building 1

Building 1	Age of Air			
	Room plan as Table2		Change bathroom position	
	BED	WC	BED	WC
1. Room in BD1 at 1st Floor (V=5m/s, α = -33°)				
- Option1	20	180	30	213
- Option2	13.5	90	7	42
- Option3	3.5	5	2.2	5.2
- Option4	3.5	5	1.8	5.4
2. Room in BD1 at 1st Floor (V=5m/s, α = 45°)				
- Option1	18.51	185	29.2	261
- Option2	7	36	13	126
- Option3	4.2	6	3.6	4.5
- Option4	4.0	6	3.4	6.8
3. Room in BD1 at 2nd Floor (V=6m/s, α = -33°)				
- Option1	19	173	24	167
- Option2	13.8	83	6	35
- Option3	3	4.2	1.2	4.3
- Option4	2.5	4.2	1.2	4.5
4. Room in BD1 at 2nd Floor (V=6m/s, α = 45°)				
- Option1	18	179	33.5	268
- Option2	6.3	32	10	80
- Option3	3.5	4.78	3.5	4.5
- Option4	3	4.78	2.7	5.0

Table 4. The simulation result for the age of air in each option experiment in building 2

Building 2	Age of Air			
	Room plan as Table2		Change bathroom position	
	BED	WC	BED	WC
1. Room in BD2 at 1st Floor (V=4m/s, α = -20°)				
- Option1	30	257	22	210
- Option2	11	75	7	52
- Option3	3	5.8	2	7
- Option4	3	5.8	2	6
2. Room in BD2 at 1st Floor (V=4m/s, α = 45°)				
- Option1	25	210	22	221
- Option2	9	42	15	156
- Option3	4	7.5	4.5	7
- Option4	4	8	4.2	7
3. Room in BD2 at 2nd Floor (V=5m/s, α = -20°)				
- Option1	30	266	22	202
- Option2	10	60	7	40
- Option3	2.5	4.5	1.5	5.2
- Option4	2.2	4.5	1.5	5
4. Room in BD2 at 2nd Floor (V=5m/s, α = 45°)				
- Option1	22	204	24	240
- Option2	8	40	14	133
- Option3	3.3	5.8	4	6
- Option4	3.2	6.3	4	6

4.2 Analyses

The simulation result in *the option 1* show the airflow diffuse only in the sleeping area (Figure 6), according to the openings inside deeply the room are closed such as the door and the opening in the bathroom ceiling. Less air comes inside the room can flowing to the door due to there are high pressure in front of the door and the bathroom. However, when the bathroom position is changed to obstruct the wind direction from outside flow through the window inside the room, the air movement will be better, as the wind from outside confront to the bathroom wall, the air will be more diffuse and flow around the room then the bathroom wall which does not obstruct the flow stream.

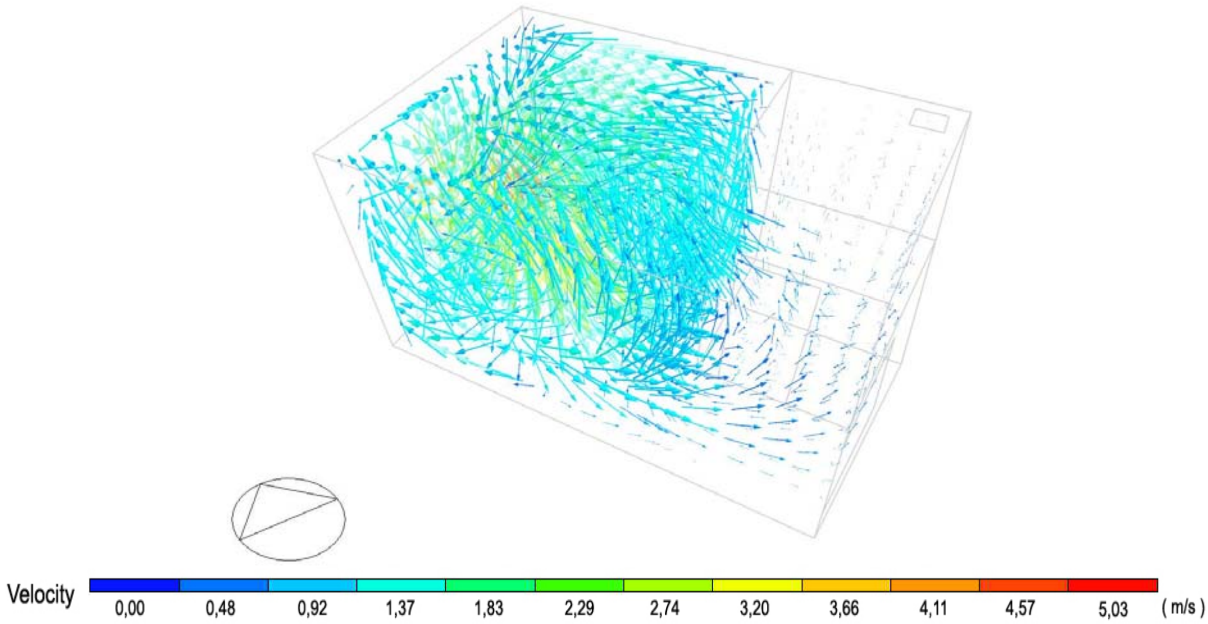


Figure 6. The simulation result in option 1 is illustrated for the velocity in the virtual model testing

In *the option 2*, when the door inside the room is opened, a particular air movement can be observing passing the sleeping area and flowing out by the door outlet, the age of air number is lower than the option 1 because there is not the door obstruction that allows the air move out and the pressure is low. The same as option 1, the bathroom position influences to the air movement, when the bathroom wall confront to the wind direction that flow inside the room, the air will diffuse and spread around the room, conversely if the bathroom wall is not obstruct the wind direction, the age of air will higher than the bathroom wall obstruct, this event occur due to the obstructed wall will effect to the air movement which make the air more diffuse, it also impacts to the air movement inside the bathroom, the air flow stream will flow into the bathroom through the bottom part of the bathroom door and flow out from the bathroom through the top of the bathroom door (Figure 7), particularly the age of air will decrease more if the wind direction comes from the north and south.

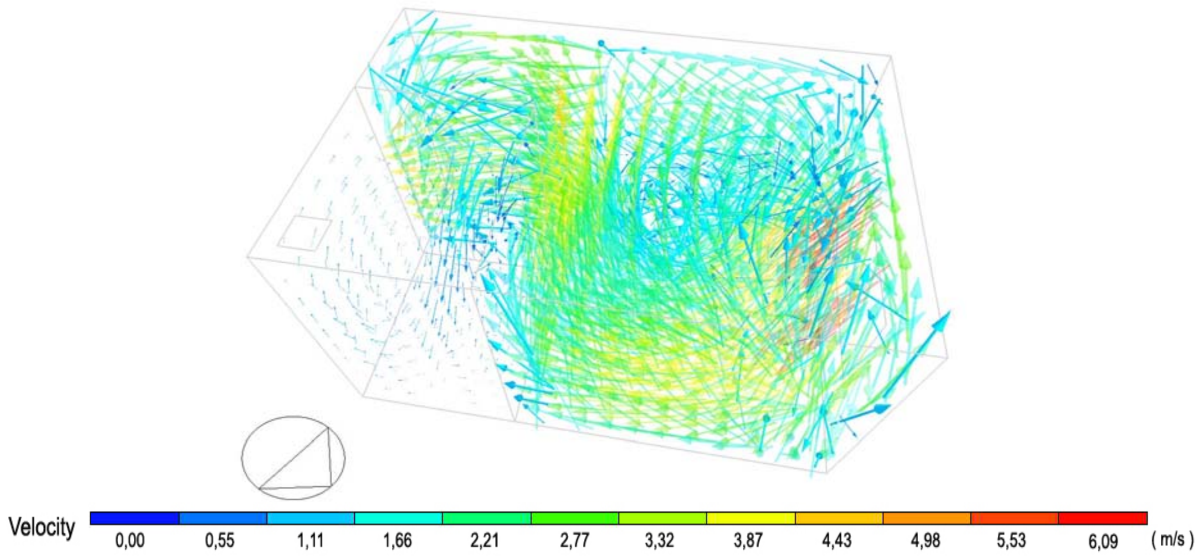


Figure 7. The simulation result in option 2 is illustrated for the velocity in the virtual model testing

In *the option 3*, when the ceiling of the bathroom is opened , the amount of age of air and the pressure inside whole room will decrease, that means the air movement flow better. The air flow stream displays that when the air come inside the room and pass until confront to the closed door, the air flow will suddenly change direction to go inside the bathroom and flow out through the opening in the ceiling. Noteworthy, the velocity of air in front of the outlet opening in the ceiling is very high, (as the Figure 8, the velocity will be higher as much as the outlet opening size will be smaller). The comparison result when changing the bathroom position as the above testing; that is the bathroom wall which confront to the air flow direction will be more diffuse and spread around the room than the case of the bathroom wall which do not obstruct the air flow.

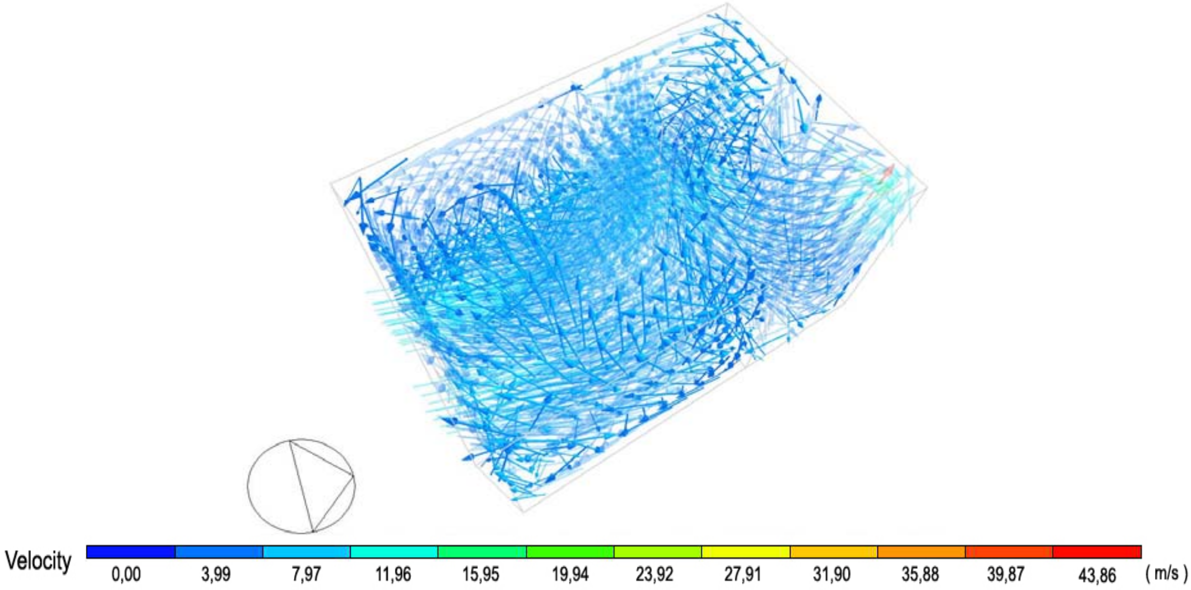


Figure 8. The simulation result in option 3 is illustrated for the velocity in the virtual model testing

In the *option 4* shows that when all outlet opening are opened as the Figure 9, the air flow velocity in the sleeping area is a little bit increased, it is sometimes similar to the option 3, and it will increase more when the bathroom wall confront to the wind direction that flow through inside the room. For the air movement inside the bathroom will be lower than the option 3 if the wind that flow inside the room has direction from the different way of the prevailing wind direction, however if it comes from the same direction the air movement inside the bathroom will be very similar between both case the option 3 and the option 4. Furthermore, as the case of the bathroom position changed is still the same appearance as the previous test in each option above.

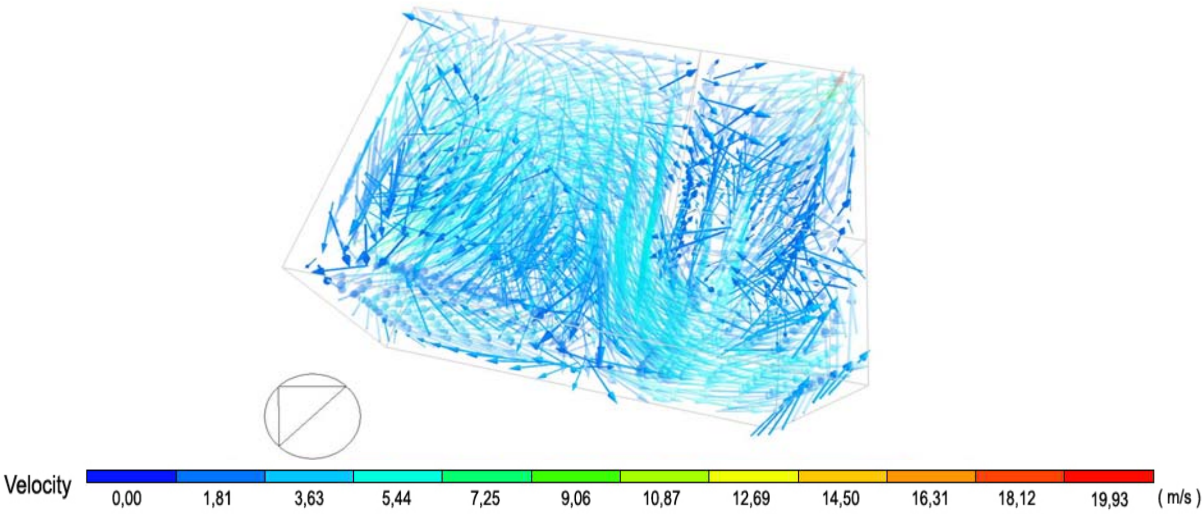


Figure 9. The simulation result in option 4 is illustrated for the velocity in the virtual model testing

After comparing the four options, the results found that the air movement in *option 3* and *option 4* are the best and both are very similar results, representing the best air-flow depend on the outlet opening. Even though the *option 3* will use the small outlet opening in the ceiling, and it can provide good ventilation, however if the door is opened more for the outlet opening will improve the air movement more (*option 4*). Furthermore, the results also indicate the outlet opening in *option 3* provides better air movement than the outlet opening in *option 2* which it has bigger size (the outlet opening in option 2 is the door), it cause the small opening will increase the velocity when the air flow through it [20, 21]. Therefore, in the reality it's not necessary to open the door all the time, just use the outlet opening in the bathroom ceiling can get the good ventilation, however the outlet opening must open to work all time (it's possible the ventilation fan or exhaust fan).

5. Conclusion

The research study about the air movement outside and inside building is very significant, it will be the guideline for the real building design and also solve the problem about the building environment problem relate the energy saving. This study result will provide the solution of the wind appearance that effect to the building in Vientiane Capital in Laos where the climate is hot and humidity, by using CFD model simulation technique. The result of experiment show the good air movement in the room influence from the wind direction from the north and south which are the prevailing wind in this location, and the good natural ventilation will be better if the room obtain the natural wind from the north than the wind from the south and from other side in this location. For the opening providing in the room, the passive ventilation inside the room will be benefit when using the outlet opening in the bathroom ceiling will provide the better air movement in the sleeping area than the case of opening the door, and if open both it will surely provide the best. However, it is not usually happen for the air movement inside the bathroom, which depend on the bathroom position relate to the wind direction flow through inside the room. Furthermore, the ventilation testing followed by changing factors including wind direction, bathroom position, air flow velocity and air flow rate differentiation, can finally resume that the room position providing the best ventilation in this case study is the room at the second floor of building 1.

References

[1] A. Walker. Natural Ventilation, 2014 Whole Building Design Guide (WBDG). https://www.wbdg.org/resources/naturalventilation.php. (Access: 30 May 2015).

[2] ASHRAE. 2013. ASHRAE Handbook : Fundamental. Chapter 25 Heat, Air and Moisture control in building assemblies. Atlanta.

[3] ASHRAE. 2013. ASHRAE Standard 62.2.2013. Ventilation and Acceptable indoor air quality in low-rise residential building. Atlanta.

[4] S.A.Rahman. K.S. Kannan. Air flow and thermal comfort simulation studies of wind ventilated classroom in Malaysia. 1996 World Renewable Energy Congress. Vol.8, pp. 264-267.

[5] Commercial Building Ventilation and Indoor Environmental Quality. Ventilation rates and technologies. http://energy.lbl.gov/ied/viaq/v_rates_11.html. (Access: 23 May 2015).

[6] G. C. Graça, Q. Chen, L.R. Glicksmam, L.K. Norford. Simulation of wind-driven ventilative cooling systems for an apartment building in Beijing and Shanghai. 2002 Energy and Building. Vol.34, pp. 1-11

[7] S. Sadrizadeh, S. Holmberg, A. Tammelin. A numerical investigation of vertical and horizontal laminar airflow ventilation in an operating room. 2014 Building and Environment. Vol.82, pp. 517-525.

[8] M.H. Kim, J. H. Hwang. Performance prediction of a hybrid ventilation system in an apartment house. 2009 Energy and Building. Vol.41, pp. 579-586.

[9] L. Peeters, R. de Dear, J. Hensen, W. D'haeseleer. Thermal comfort in residential buildings: Comfort values and scales for building energy simulation. 2009 Applied Energy. Vol.86, pp. 772-780.

[10] H.M. Taleb. Using passive cooling strategies to improve thermal performance and reduce energy consumption residential building in U.A.E. buildings. 2014 Frontiers of Architectural Research. Vol.3, pp.154-165.

[11] G.R. Hunt, P.F. Linden. The fluid mechanics of natural ventilation displacement ventilation by buoyancy-driven flows assisted by wind. 1999 Building and Environment. Vol.34, pp. 707-720.

[12] Y. Li, A. Delsante, J. Symons. Prediction of natural ventilation in buildings with large openings. 2000 Building and Environment. Vol.35, pp. 191-206.
[13] R. Letan, V. Dubovsky, G. Ziskind. Passive ventilation and heating by natural convection in a multi-storey building. 2003 Building and Environment. Vol.38, pp. 197-208.
[14] J.D. Posner, C.R. Buchanan, D.D. Rankin. Measurement and prediction of indoor air flow in model room. 2003 Energy and Building. Vol.35, pp. 515-526.
[15] A. Stamou,I. Katsiris. Verification of a CFD model for indoor airflow and heat transfer. 2006 Building and Environment. Vol.41, pp. 1171-1181.
[16] L. Wang, N.H. Wong. Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment. 2009 Building and Environment. Vol.44, pp. 95-112.
[17] J. Waewsak. Computertional Fluid Dynamics. 2006 Thaksin Science Journal. Vol.1, pp. 32-42. Songkhla, Thailand.
[18] V. Dubovsky, G. Ziskind, S. Druckman, E. Moshka, Y. Weiss, R. Letan. Natural convection inside ventilated enclosure heated by downward-facing plate: experiments and numerical simulations. 2001 International Journal of Heat and Mass Transfer. Vol.44(16), pp. 3155-68
[19] DesignBuilder Simulation and CFD training guide. CFD calculations and convergence in Chapter 19 Computational Fluid Dynamics (CFD). pp. 162-163. http://www.designbuildersoftware.com/docs/designbuilder/DesignBuilder_2.1_Users-Manual_Ltr.pdf. (Access: 23 May 2015).
[20] M. DeKay and G. Z. Brown. Sun, Wind & Light: Architectural design strategies, 2nd edition. New York, Wiley, 2001.
[21] F. Allard, M. Santamouris,. Natural ventilation in buildings : A design handbook. London, James & James, 2002. http://books.google.com/books?hl=en&lr=&id=1tdQMyhPA2gC&oi=fnd&pg=PR9&dq=Natural+ventilation+theory&ots=mFzmfd4mct&sig=XA3zksH_OBkkS8tILbxmwJqbWyo (Access: 23 May 2015).

Pakasith Phonekeo is Ph.D student in Faculty of Architecture, the University Polytechnic of Valencia and currently doing his doctoral thesis here. He graduated the Master degree in Faculty of Architecture Chiangmai University (Thailand) with the thesis topic related about the Computer-Aided and software application using for on the environmental study for the building. He is also a lecturer in the field of building and environmental design in Faculty of Architecture, National University of Laos (NOUL).
E-mail address: pakasith@gmail.com; papho@alumno.upv.es

P. Amparo López-Jiménez is PhD in Industrial Engineering, Associate Professor in the Hydraulic and Environmental Engineering Department at the Universitat Politècnica de València. She is currently the Associate Director of the Hydraulic and Environmental Engineering Department of Universitat Politècnica de València. She has more than a decade of experience in research and teaching in Engineering fields, always related to hydraulic topics. She is author and editor of several publications about Hydraulic an Environmental Engineering and Flow Dynamics. She has participated in national and international R&D projects and co-organized International Seminars and Networks. She is an experienced University Teacher, an active researcher and a former practicing engineer.
E-mail address: palopez@upv.es

Ignacio Guillén Guillamón is Ph.D. Architec on Architecture Acoustics from the University Polytechnic of Valencia on 1999. Assistant Professor of Applied Physics on the Architectural School since 1998 and currently Researcher at the Physical Technologies Center. Director of several Master Thesis on LCA for building, Energy Building Simulation and Energy and Sustainable Housing Renovation. He is currently IP of an R+D+I project founded from CDTi together with a pool of Enterprises. The E3 Project "EdificaciónEcoEficiente" search through the development and construction of high efficient building prototypes new way to improve the efficiency and industrialization on housing, both for news or renovations. And also has been working as technical consultant on projects on topics like : Acoustic Design of Concert and Opera Halls and theaters, Acoustic Characterization of listening rooms, Sound insulation, Passive solar architecture and natural ventilation, Energy Efficiency and building simulation.
E-mail address: iguillen@fis.upv.es

