Enhanced wind-driven infiltration model by incorporating wind direction for design purpose

L G Perdamaian¹, Z J Zhai²

¹Department of Nuclear Engineering and Engineering Physics, Universitas Gadjah Mada, Jalan Grafika 2 Yogyakarta 55281 – Indonesia
²Department of Civil, Environmental, and Architectural Engineering University of Colorado Boulder, UCB 428, Boulder, CO 80309-0428 – USA

E-mail : ¹laksana_gp@mail.ugm.ac.id, ²john.zhai@colorado.edu

Abstract. Research has been conducted to improve and enhance existing LBNL infiltration model in order to incorporate wind direction effect on flow rate. Existing wind driven infiltration of LBNL model only consider terrain and shielding parameter as correction factor. Supplemental analytical model considered in this research was Swami-Chandra model, which the calculation was based on average wind pressure coefficient (Cp). It was evident that for same wind direction, Cp value will not change regardless of wind velocity. Process of simulation shows that infiltration modelling has to be performed in three-dimensional environment with transient calculation in order to achieve convergent result. Wind direction strongly affects infiltration flow rate. However, as the infiltration hole has relatively small size compared to wind velocity, flow rate variation is marginal for energy-based design. Nevertheless, for pollutant and particulate matter analysis, wind direction will give significant change in term of concentration and flow direction.

1. Introduction

Both infiltration and ventilation are vital in determining the indoor air quality. Ventilation, both mechanical and natural are most likely controlled either automatically or manually. Meanwhile, infiltration is unintentional and can introduce unwanted contaminants from outside and either hot air from the outside that increasing cooling load or bring cold air inside and increase the heating load. Study of infiltration has been conducted in decades and researcher has developed many kinds of infiltration model. However, the nature of airflow is very difficult to predict and model, because of the uncertainty in turbulence, vortex, and flow separation, for example. Therefore, from design purpose, the model is highly practical to use but prone to error and inaccuracies. An alternative to those analytical model is using computational fluid dynamics. Usage of CFD has increased along with the improvement of computing capabilities. CFD can model varieties of fluid flow with many physical parameter, therefore better result can be achieved.

2. Literature Review

2.1. Building Infiltration Model
Building infiltration model can be divided into single zone and multiple zone model. A zone is defined as fully mixed volume with a constant concentration level of the enclosed gas mixture [1]. Single zone model is used in a building which has virtually no partition, whereas multi zone model is suitable for multi-story building with complex partition.

Single zone infiltration can be used to determine impact of infiltration airflow on whole building energy balance. EnergyPlus used simplified single zone model to calculate infiltration flow rate. Two models are used, Effective Leakage Area which based on LBNL [2] model and Flow Coefficient which based on Walker and Wilson method [3]. Both of the models also can be found on ASHRAE Handbook as Basic and Enhanced method, respectively [4].

One of the infiltration calculation method that widely used is LBL infiltration method. Originally the LBL infiltration method was developed for residential building. LBL infiltration uses several parameters which are leakage area, height of the building, inside-outside temperature difference, terrain class of the structure, and wind speed [5]. Infiltration airflow rate through a leakage can be expressed by following equation.

$$Q = A \frac{2}{\rho} \Delta P$$  \hspace{1cm} (1)

Where

- $Q$ = infiltration rate (m$^3$/s)
- $A$ = effective leakage area (m$^2$)
- $\rho$ = density of air (kg/m$^3$)
- $\Delta P$ = pressure difference (Pa).

Based on its driving forces, there are two different type of infiltration, which are wind regime and stack regime. Wind regime infiltration dominated by dynamic pressure whereas stack regime dominated by pressure difference. Wind and stack regime infiltration are defined as follows.

$$Q_{wind} = f^*_{w}A_o v' \hspace{1cm} (2)$$
$$Q_{stack} = f^*_{s}A_o \sqrt{\Delta T} \hspace{1cm} (3)$$

Where

- $Q_{wind}$ = infiltration in the wind regime (m$^3$/s)
- $Q_{stack}$ = infiltration in the stack regime (m$^3$/s)
- $A_o$ = total leakage area of the structure (m$^2$)
- $f^*_{w}$ = reduced wind parameter
- $f^*_{s}$ = reduced stack parameter
- $v'$ = weather tower wind speed (m/s)
- $\Delta T$ = inside-outside temperature difference (K)

The ‘reduced’ term on reduced wind parameter is used to distinguish it from the original wind parameter ($f_w$) which does not incorporate shielding coefficient [2]. For reduced stack parameter, the case is the same with reduced wind parameter, however the term ‘reduced’ is not because of shielding coefficient, as stack infiltration is caused by temperature difference and has little to do with the terrain or geometry of surrounding environment. Rather, the reduced stack parameter includes ceiling-floor leakage difference. After $Q_{stack}$ and $Q_{wind}$ has been determined, the next step is combine two values using the following equation.

$$Q = \sqrt{Q_{stack}^2 + Q_{wind}^2} \hspace{1cm} (4)$$

2.2. Airflow around Buildings

Airflow around building also can exhibit variation of pressure as the result of fluctuation of wind direction and velocity. Pressure exerted by prevailing wind on a surface is called wind pressure (p$_w$),
which is a dynamic pressure. The value of wind pressure for every surface is different depending on the orientation of the surface and the surrounding environment. Hence a factor should be added to correct the actual pressure exerted by the airflow to the surface. This factor is called wind pressure coefficient \(C_p\). \(C_p\) can be defined as follows [6].

\[
C_p = \frac{P_x - P_0}{P_v}
\]  

(5)

Where

- \(C_p\) = wind pressure coefficient
- \(P_x\) = static pressure at a given point on the building façade (Pa)
- \(P_0\) = static reference pressure (Pa)
- \(P_v\) = wind pressure (Pa)

The value of \(C_p\) depends on many factors such as building geometry, façade, surface orientation, degree of exposure/sheltering, wind speed and wind direction. \(C_p\) data can be obtained from primary or secondary sources [6]. Primary sources usually in the form of experiment or measurement, whereas secondary source typically uses calculation and analytical method. Secondary sources can come in the form of database or analytical model. Database is prepared by research organization such as the Air Infiltration and Ventilation Centre (AIVC) and ASHRAE. Analytical model uses calculations to find \(C_p\) based on experiment using wind tunnel or full scale measurement. Using regressing technique, a set of formula can be generated. Example of analytical model include Swami and Chandra (1988) [7] and Walker and Wilson (1994) [8] methods.

Meanwhile, there is third method which is using computational fluid dynamics (CFD). CFD has been used in fluid flow modeling for decades. In the recent times, as computing performance improved, CFD is widely recognized as a valuable and reliable tool. However there are several challenges in using CFD to determine \(C_p\) value, which are include numerical accuracy, boundary condition, and turbulence models (Stathopoulos, 1997 in Cóstola, et. al, 2009) [6].

2.3. Importance of Wind Direction and Wind Pressure on Infiltration Calculation

Wind pressure distribution as an important parameter in infiltration calculation is incorporate in several airflow network (AFN) simulation program in the form of \(C_p\) value as mentioned by Cóstola et. al (2009) [6]. Majority of these simulation program, however, using average \(C_p\) value, which means there is only one \(C_p\) value for each surface of the building envelope, instead of distributed or local \(C_p\). Nevertheless, the incorporation of \(C_p\) in some AFN programs reflects the importance of wind direction in infiltration calculation, which plays a major role in determining \(C_p\) value.

Energy based infiltration models are formulated with the focus of energy efficient design i.e. heat transfer by the means of infiltration airflow. Air infiltration models for energy analysis purpose assume that the input values calculation are annual average. However, for mass transportation analysis of infiltration, such as for contaminant transport and indoor air quality analysis, instantaneous calculation is very important as the nature of airflow fluctuation caused by wind.

COMIS, an international collaboration of multizone airflow and contaminant transport modeling, incorporates the importance of wind pressure distribution on building envelope to correctly determine the contaminant distribution inside the building [9]. The wind pressure distribution on building envelope is calculated by COMIS using pressure coefficient as follows [9].

\[
c_p(x, y, z) = \frac{p(x, y, z) - p_0(z)}{p_{dyn}(z)}
\]  

(6)

with

\[
p_{dyn}(z) = \frac{1}{2} \rho_{out}(z) v^2(z)
\]  

(7)
where

\[ c_p(x,y,z) = \text{pressure coefficient at coordinates x, y, and z, respectively [nondimensional]} \]
\[ p(x,y,z) = \text{surface pressure at coordinates x, y, z [Pa]} \]
\[ p_o(z) = \text{atmospheric pressure at height z [Pa]} \]
\[ p_{dyn}(z) = \text{dynamic pressure in the undisturbed flow at height z [Pa]} \]
\[ \rho_{out}(z) = \text{density of outside air at height z [kg/m}^3\text{]} \]
\[ v^2 = \text{wind speed at height z [m/s]} \]

The practice of using average pressure distribution represented by one Cp value in AFN program caused uncertainties compared to multiple local Cp values that form a wind pressure distribution on a particular surface.

2.4. Improvement of Existing Air Infiltration Model

L.C. Ng, et.al. (2015) [10] have proposed a model to improve the accuracy of building air infiltration calculation on building energy analysis. Improved infiltration model was intended to be implemented on building energy simulation, namely EnergyPlus. The Strategy implemented was coupling and incorporating EnergyPlus with other software specialized in multiple airflow modeling and calculation, which is CONTAM. Research was focused to improve the formulation used in EnergyPlus Result shows that each building has unique value of empirical factors.

3. Methodology

3.1. CFD Case Setup

In this simulation, a simple, square blockage is considered as a model. Study will be focused on the infiltration through the building. Any flow outside the building surface will not be considered but will be valuable for verifying factor. Two dimensional simulation will be considered in this research. This is because for low rise building, the difference of wind between the base and the top of the building is very small, therefore for simplicity and practicality, two dimensional simulation is adequate to provide reasonable result that can be used in preliminary design purpose. The terrain roughness category is comparable to that of LBNL model, therefore direct comparison can be done. The roughness is set at its lowest level, a representative of open sea which has the roughness of 0.0002 m.

3.2. Study of Cp around Building

The first step of the research is conducted a numerical experiment to study the influence of varying wind speed and direction on Cp value on each surfaces. With the advantage of CFD, study of Cp value can be conducted based on small area (dA) rather than surface average, which the experiment shows there are unique pressure and Cp distribution based on measurement location. The size of the area is of course depends on the size of cells near the wall. Numerical experiment will be conducted with a wind speed of 1, 2, 3, and 4 m/s and the wind direction will be varied from north (0 degrees) clockwise to the east (90). Experiments will only cover a quarter of a full circle because the model is square in shape and will exhibit symmetry.

3.3. Study of Air Infiltration through a Lump Sum Hole

On the second step, the experiment will start incorporating infiltration feature which is leakage in the wall. This is in accordance with existing infiltration model, which based on lump sum leakage or effective leakage area, particularly LBNL model and its derivative in EnergyPlus and ASHRAE. As the research is intended the existing LBNL model, this step is very important although it is not realistic in most cases. Implementation of the numerical experiment is by adding a hole in two opposite surfaces to let the air going through. The size of the hole is set at 1/20 or 0.05 of building length for each two surface.
3.4. Study of Infiltration through Multiple Small Hole
The purpose of the third step is to study the effect of individual and distributed Cp around the surface. Another purpose of this step also useful to study the influence of Cp, wind speed and wind direction flow rate of infiltration air either incoming or outgoing which is important factors in formulating improvement of existing building infiltration model. Numerical simulation (CFD) case setup will be identical with previous step. The only difference is now the hole will be distributed but total area is constant. Size of distributed hole depend on the cell size used in the case. The smaller the cell size the smaller possible size of the hole and more information can be obtained.

3.5. Proposed Improved LBNL-Based Infiltration Model
To incorporate the effects of Cp factor into existing LBNL model, this research will utilize regression method. Ideally, the difference between variations of observed variables, in this case wind velocity, wind direction, Cp factor, and infiltration flow rate, is as small as possible so that the trendline can be smooth and accurate regression can be obtained. As CFD can only handle one case at a time only a handful of representative value is observed and then be regressed. The regression analysis result is then used as a correction factor in existing LBNL model. As a complement the Cp factor will be added to accommodate effect of wind direction to infiltration flow rate.

4. Study of $C_p$ around Building

![Figure 1. Cp of west-facing wall with various wind velocity.](image1)

![Figure 2. Cp of north-facing wall with various wind velocity.](image2)

![Figure 3. Cp of east-facing wall with various wind velocity.](image3)

![Figure 4. Cp of north-facing wall with various wind velocity.](image4)
Figure 1 through 4 show the relationship between velocity, wind direction, and surface averaged coefficient of pressure (Cp). For each four surfaces, the pattern is just rotated 90° or a quarter of full circle. This is because the model is in the form of quarter shape. Also, there is no obstruction whatsoever in every direction. In real cases, the shape usually come not in the form of square and there are variety of obstruction and blockages around the building, therefore the surface averaged Cp will be different. Experiment results show that there is little or no impact of velocity of the value of Cp. This is because Cp is basically a ratio between pressure at the surface of the building and to that of prevailing wind without obstruction. Increasing wind velocity will increase both wind pressure at the surface (Pw) and wind pressure without obstruction (Pv) hence the ratio, which is Cp, will stays the same. The relatively small variation of Cp value for the same wind direction with different wind velocity is caused by the profile of the flow itself.

5. Study of Air Infiltration through a Lump Sum Hole
Simulation was conducted using wind direction variation from 0° to 90° with increment every 5° and fixed wind velocity of 3 m/s. The largest Y-velocity occurred when the wind is blowing from 0 to 10°. At this wind direction, the north surface act as the windward side. The resultant of velocity vector is maximum since the angle between the velocity vector and normal axis of the surface is zero.

As the wind approach the building, wind vector will be deflected and turbulence is created. This creates the resistance around the building. When the air is inside the building, it will be trapped. The pressure builds up until it reaches some point when it can overcome outside resistance and drive the air outward. The phenomenon shows the importance of analysing velocity component separately from the resultant velocity. The resultant velocity will always positive and in the case of wind blowing parallel with the opening, the velocity can be large and use it directly to determine inflow or outflow will be misleading.

To obtain the correct infiltration airflow rate using Swami-Chandra model, a series of steps have to be followed, which are depending on the case. For this case, the building has two openings, one at the north surface and the other at the south surface. The opening is a representative of infiltration leakage accumulated in one large opening. Before calculating the infiltration airflow through the opening, Cp value for the surfaces has to be calculated. The result of Cp calculation for wind direction of 0° to 90° for both north and south wall is presented by Figure 5.

Figure 5 shows that for the north surface, which is windward side, the Cp is decreasing when the angle of the wind is getting larger relative to the normal of north wall. Cp for north wall is going negative after the wind direction is about 70°. As for the south wall, which in this case is leeward side, the Cp is always negative for wind direction from 0° to 90°.
After the Cp, or to be more precise, the average Cp has been determined, the next step is selecting the method of calculation based on window or opening configuration of the building. As mentioned earlier, the building leakage and crack is assumed only can be found in north and south wall. The cracks and leakage are represented by one large opening each at the north and south wall. Size of both opening is 0.1 m in width and 0.1 m in height, which is also the height of the building. As the height of the building is relatively short, the building can be classified as low-rise building.

The result of calculation is as following, which is relatively small compared to the CFD result or LBNL infiltration model. Figure 6 shows that the flow rate relatively stable and unaffected by the change of wind direction. However, after 60° of wind direction, the flow rate start decreasing with larger wind direction. Eventually, the flow rate becomes zero when the wind is blowing parallel with the north and south opening.

The reason of the unaffected flow rate until 60° of wind direction is that the calculation of Swami-Chandra model is based on the value of ΔCp. The value of ΔCp itself relatively constant until the wind direction is larger than 60°. After 60° the Cp value of north surface is close to zero and becomes negative after 70°. Meanwhile, the Cp value of south wall is relatively constant at 0.3 but increasing after 45° and reaches its largest magnitude (0.7) at around 65° then decreasing again.

The small flow rate produced by calculation of Swami-Chandra model is because there are several correction factor applied in the calculation. There are discharge coefficient, flow coefficient, and effective window area. The calculation is as such because of the nature of application of Swami-Chandra model which is for ventilation via the large opening. As opposite, infiltration flow rate will always incorporating small opening. Therefore, the correction factors introduced in the calculation might not suitable for calculation of infiltration flow rate.

Calculation using LBNL infiltration model for lump-sum opening case is straightforward. However, as the LBNL model does not incorporate wind direction and made no distinction between inlet and outlet opening, some adjustment should be made. Instead, the LBNL focusing on location of the leakage, i.e. at the floor or at the ceiling. This is will be irrelevant with this case, which is a two-dimensional case. Therefore, the effect of leakage location will be eliminated in the calculation. As expected, Figure 7 show that the flow rate is constant and unaffected by wind direction change, as the formula itself does not include wind direction term.

The aforementioned three different methods will be compared to evaluate and determine their characteristics. Figure 7 shows that the infiltration flow rate from Swami-Chandra model is relatively small compared to two other models. Meanwhile at the wind direction of zero degree (wind blowing from the north), there is small difference between LBNL model and CFD simulation. This is indicating that the LBNL model is basically calculate the infiltration airflow of wind blowing...
perpendicular to the opening. Based on correlation evaluation, this is a good start because the LBNL model is ready to be corrected by including some terms that will correlate with the change of wind direction.

Swami-Chandra model is an innovative method because of the incorporation of Cp terms. However, the calculation method should be adjusted for infiltration calculation, since correction factors applied diminish the infiltration flow rate.

6. Study of Infiltration Flow Rate from Multiple Holes

6.1 Distributed Holes at Front and Rear Sides (North and South)

In this case, wind direction variation was from 0° to 90° with 5° difference. In all cases, air pressure inside the building is positive and relatively larger compared to ambient temperature. This made air velocity vector is pointing outward from the building. However, mass flow rate analysis showed that there were air mass inflow as well as outflow.

Figure 8 shows the mass flow rate pattern of air flowing through distributed holes located at the front and rear. It is evident that the highest mass flow rate is achieved around 45 to 55 degrees, which is diagonal wind direction. Meanwhile, the lowest mass flow rate is achieved at 0°. This is because the size of the hole is far too small compared to the velocity of the wind. The difference of order between hole size (1.25 × 10⁻³ m width) and wind velocity (3 m/s) is more than 1000. Small hole makes resistance, as a result the entering air is partially blocked and makes the flow rate small.

On the other hand, at the diagonal wind direction, the direction of the airflow is not perpendicular but rather parallel with the airflow inside the building, which tends to flow in circular pattern. Based on the Bernoulli law, the difference between air velocity flowing around the building wall and air velocity inside the building produce pressure difference. As a result, airflow is forced to flow.

6.2 Distributed Holes at All Four Sides

![Figure 8. Mass flow rate of multiple located at the front and rear of the building.](image-url)
Figure 9. Mass flow rate of multiple located at all four sides of the building.

Figure 10. Mass flow rate of multiple located at all four sides of the building simulated in 3D.

Figure 9 shows the mass flow rate pattern of air flowing through distributed holes located at the front and rear. Again, almost similar with the previous front and rear case, it is evident that the highest mass flow rate is achieved around 45 degrees, which is diagonal wind direction. Meanwhile, the lowest mass flow rate is achieved at 0°. This is because the size of the hole is far too small compared to the velocity of the wind. The difference of order between hole size (1.25 × 10^-3 m width) and wind velocity (3 m/s) is more than 1000. Small hole makes resistance, as a result the entering air is partially blocked and makes the flow rate small.

On the other hand, at the diagonal wind direction, the direction of the airflow is not perpendicular but rather parallel with the airflow inside the building, which tends to flow in circular pattern. Based on the Bernoulli law, the difference between air velocity flowing around the building wall and air velocity inside the building produce pressure difference. As a result, airflow is forced to flow.

The difference between the four direction and front-rear cases is that the mass flow rate difference between the lowest and highest value is rather smaller. This is obviously because the nature of the hole location. More distributed hole means more airflow can flow inside. Although the size of the holes is smaller, the location has a significant effect in introducing air. As explained in the front-rear case, hole faced directly to the wind tend to has less airflow compared to hole located not facing directly.

6.3 Three-Dimensional Simulation

Figure 10 shows the mass flow rate of wind direction from 0° to 15° is almost constant, about 0.00216 kg/s. The change of mass flow rate because of wind direction is almost negligible. Mass flow rate start to increase when the wind is coming from 15° and steadily moving up until the wind is coming from 35°. At wind direction of 40° the magnitude of mass flow rate is somewhat decreasing and decreasing further in 45° wind direction.

It is interesting that the 3D case provide lower mass flow rate compared to 2D cases. This is because in 2D simulation, convergence is very hard to achieve, mainly because when the airflow ‘hit’ the hole and blockage around it, it should flows to other direction, including Z direction. As Z-direction is not available in 2D case, the flow profile is somewhat inaccurate.

From the simulation it is can be seen that wind direction do have a role in determining mass flow rate. Similar with previous scenarios, wind direction between 35 to 55° degrees will produce maximum mass flow rate. However, if the mass flow rate is seen as a whole, there is only 12% difference between the lowest (0.00216 kg/s) and the highest (0.00242 kg/s) mass flow rate magnitude. In term of ACH, the infiltration will cause around 1.01 to 1.13 ACH.

One of the interesting phenomenon in the simulation is that the pressure inside the building will always higher than the environment. With the time of simulation is 60 seconds, it is sufficient enough for the wind to sweep the entire domain (80 m x 80 m). Therefore it can be said that in steady condition, the pressure of air inside the building will be higher than the ambient environment. Velocity vector analysis also shows that airflow is tend to flow outward in every direction, however they tend to flow through hole in leeward side rather than through the windward side.
Meanwhile, mass flow rate analysis shows that the airflow as a mass rate always enters through west and south holes, whereas airflow exits through holes located and east and north. This is because the wind direction is moving gradually to the northeast in a clockwise manner.

In order to determine whether placement of the hole in a building will affect mass flow rate or not, a full 1-story building (3 meter height) has been simulated. In the 3-m case, the hole is located in the middle of building height (1.5 meter). The size and configuration remain the same. There is slight difference between the two, with the 3-m result is relatively lower compared to its 0.1-m counterpart.

### Table 1. Comparison between mass flow rate of 0.1 m and 3 m height.

| Wind Direction | 0.1 m height | 3 m height |
|----------------|-------------|------------|
| 0°             | 0.00218     | 0.00215    |
| 45°            | 0.00227     | 0.00218    |

7. Proposed Improved LBNL-Based Infiltration Model

To determine the correction term to be applied in the existing LBNL infiltration model, the first step that has to be done is find the mathematical formula of the flow rate pattern obtained from simulation result. As shown in the previous chapter, the flow rate has a repetitive pattern based on the angle of the wind direction. In mathematics, one equation that exhibit repetitive (cyclic) pattern is sinusoidal wave, which in fact is also based on the angle. Therefore, as the base of the mathematical formula is sinusoidal equation.

The pattern of the flow rate is not purely sinusoidal because there are multiple peak and valley in one cycle, in contrast with a pure sinusoidal wave which there is only one peak and one valley in every cycle. However, this does not mean that mathematical formula cannot be used to track the pattern of the flow rate result. There will be some modification to the sinusoidal wave.

The proposed formula is actually a sum of more than one sinusoidal wave. This is partly inspired by the creation of square wave in Fourier analysis by summation of odd harmonics of original sinusoidal wave equation. In this case, the constituent of the wave will not following nth harmonics as prescribed to create a square wave, because the pattern of the flow rate is not a square wave.

To make an improvement on the existing LBNL model, the flow rate value of the experimental model is not important. Rather, the more important thing is the factor determining how the flow rate of experimental model correlated to that of the existing LBNL model. The original LBNL infiltration model only produce one infiltration flow rate value, as the formula does not include wind direction.

The first step is to look at the shape of the pattern. There are several key characteristics of the pattern that is very important to determine the constants of the sinusoid function. Sinusoid function has frequency and amplitude. To determine the frequency, one can look at the wave cycle. The pattern can be divided into two parts, the large and small wave.

Based on the pattern of the wave, the large wave is assumed starts at its lowest value (valley). Sinusoid function that starts at its lowest value (when there is no phase shifting) is negative cosine. The large wave reach its peak on 45°, 135°, 225° and 315°. On the other hand, the large wave will have its lowest value at 0°, 90°, 180° and 270°. Therefore, one cycle of the large wave requires 90°, which means the frequency is four times larger than ordinary cosine function.

Meanwhile, the amplitude can be determined by observe the difference between the largest and smallest value, then divided by two. Thus, the mathematical formula of the large wave is shown below.

\[-0.055 \cos(4\theta)\]  

(8)

For the small wave, the method to determine the amplitude and the frequency is similar with that of the large wave. The difference is that the small wave is a positive cosine wave rather than the negative...
one. This is because the small wave is assumed starts at the largest value and then decreases as the wind angle is getting larger.

The small wave is required to adjust the large wave so that it can closely tracks the pattern of correction factor. The small wave is set to complete a full cycle by 30°. Ordinary sinusoid function needs 360°, therefore the small wave has the frequency 12 times larger than ordinary cosine function. Therefore the mathematical formula for the small wave is shown below.

\[-0.03 \cos(8\theta)\]  \hspace{1cm} (9)

Finally to obtain the formula for improvement of the LBNL Model, both of the formula of large and small waves have to be summated and added with another value which is 1.36. Therefore, the final formula is shown below.

\[f_d = -0.055 \cos(4\theta) - 0.03 + 1.36\]  \hspace{1cm} (10)

The formula then compared with the pattern of correlation factor to determine the error. Error of the curve fitting is between 0.4 to 3.7%, which is under 10%. Therefore, the final correction factor will be as following.

\[Q_{\text{wind}} = f^*w f_d A_o v'\]  \hspace{1cm} (11)

where

\[Q_{\text{wind}}\] = infiltration in the wind regime (m³/s)
\[A_o\] = total leakage area of the structure (m²)
\[f^*w\] = reduced wind parameter
\[f_d\] = wind direction parameter
\[v'\] = weather tower wind speed (m/s)

Figure 11 shows the curve fitting between infiltration volumetric flow rate obtained by CFD simulation and volumetric flow rate obtained by adding \(f_d\) factor into the simulation. As the value of \(f_d\) is obtained by combining two sinusoidal function, the line is rather smooth. The error of the curve fitting is shown by Figure 12. The highest error value is around 7%, which is well under 10% value, therefore the formulated value can be applied for design.

Testing of the improved model is essential that the formulation is ready to be applied in similar condition with different wind speed and wind direction. There will be five test condition, each will use
random wind velocity and wind direction. However, all of the terrain parameter will be the same. Result of the simulation will be compared with the calculated result to determine the error.

| Wind Speed | theta | f_w | A_θ | Q | f_d | Q_{enhanced} | Q_{CFD} | Error  |
|------------|-------|-----|-----|---|-----|--------------|---------|--------|
| 3          | 45    | 0.34| 0.002| 0.0544| 1.385| 0.002825     | 0.002746 | 2.90%  |
| 4          | 45    | 0.34| 0.002| 0.0544| 1.385| 0.003767     | 0.003821 | 1.42%  |

8. Conclusion
Research has been conducted to improve and enhance existing LBNL infiltration model in order to incorporate wind direction effect on flow rate. From the step one of the research, which is studying wind pressure coefficient (Cp) factor based of numerical simulation result, it is evident that Cp for the same wind direction, Cp value will be the same regardless of wind velocity.

Meanwhile, step two of the research shows that wind direction will affects infiltration flow rate so that wind direction should be considered in infiltration calculation. It is also evident that overall (resultant) of wind velocity is not accurate for infiltration calculation, since it cannot have negative sign, which is very important to determine the flow direction of the airflow.

In distributed hole cases, it is evident that largest flow occurred when the wind direction is 45°. There is huge difference of order between hole size and wind velocity, so that virtually the hole is relatively almost nonexistent. This creates resistance at the hole. Meanwhile, while the outdoor air flow is parallel with the indoor, more flow rate can be generated because of Bernoulli Law.

In term of relationship between wind direction and infiltration flow rate, it is indeed there is strong relationship between the two. Given the nature of infiltration flow rate which is very small, for thermal design purpose it will be marginal. Therefore, for simplicity, it can be ignored. However, in other infiltration flow analysis, particularly in pollutant or particulate matter study, it is very important.

Air inside the building is tend to flow in circular pattern. For particulate matter and pollutant analysis, more investigation should be done, especially in term of pollutant distribution.

For future infiltration analysis, it should be noted that 3D cases is better in term of convergence achievement and obviously closer to reality. More building shape and geometry should also be studied, in order to achieve more understanding.

9. Acknowledgement
This paper is first author’s master project conducted at University of Colorado Boulder under supervision from the second author. The first author would like to thank Indonesia Endowment Fund for Education (LPDP) for providing funding for this project and the master study as a whole. Also, the first author would like to thank to his wife and family for providing tireless support. Both authors would like to appreciate for the support provided by CDSR - SHERA (Center for Development of Sustainable Region – Sustainable Higher Education Research Alliances).

References
[1] Feustel, H. E., & Rayner-Hooson, A. 1990. COMIS Fundamentals. Report LBL-28560, US Department of Energy, Lawrence Berkeley Laboratory.
[2] Sherman, M.H. and Grimsrud, D.T. 1980b. “Infiltration-Pressurization Correlation: Simplified Physical Modeling”, 1980 Annual ASHRAE Conference, June 22-26, Denver, CO.
[3] Department of Energy. 2016. EnergyPlus™ Version 8.5 Documentation. Engineering Reference.
[4] ASHRAE. 2009. Fundamentals Handbook, Ventilation and Infiltration, Chapter 16, Atlanta,
GA: American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.

[5] Sherman, M.H. and Grimsrud, D.T. 1980a. “Measurement of Infiltration Using Fan Pressurization and Weather Data”, 1st Symposium of the Air Infiltration Centre on Instrumentation and Measurement Techniques, October 6-8, Windsor, England.

[6] Cóstola, D, B. Blocken and J.L.M. Hensen. 2009. “Overview of pressure coefficient data in building energy simulation and airflow network programs”. Building and Environment 44 (2009) 2027–2036.

[7] Swami, M.V. and S. Chandra. 1987. Procedures for calculating natural ventilation airflow rates in buildings. Final Report FSEC-CR-163-86. Florida Solar Energy Center, Cape Canaveral.

[8] Walker, I.S. and D.J. Wilson. 1994. “Practical methods for improving estimates of natural ventilation rates”. Proceedings of the 15th IEA Conference of the Air Infiltration and Ventilation Centre, Buxton, U.K., pp. 517-526.

[9] Feustel, H.E. 1999. “COMIS—an international multizone air-flow and contaminant transport model”. Energy and Buildings 30 (1999) 3–18.

[10] Ng L.C., Persily A.K., Emmerich S.J. “Improving infiltration modeling in commercial building energy models”. Energy and Buildings 88 (2015) 316–23.