Using CFD simulation to improve estimation of wind pressure coefficient for naturally-ventilated buildings in tropical climate

Matthieu Zubialde-Elzaurdia, Franck Lucas, Alain Bastide

To cite this version:

Matthieu Zubialde-Elzaurdia, Franck Lucas, Alain Bastide. Using CFD simulation to improve estimation of wind pressure coefficient for naturally-ventilated buildings in tropical climate. 40th AIVC conference, Oct 2019, Ghent, Belgium. hal-03180305

HAL Id: hal-03180305
https://hal-upf.archives-ouvertes.fr/hal-03180305

Submitted on 24 Mar 2021

Using CFD simulation to improve estimation of wind pressure coefficient for naturally-ventilated buildings in tropical climate

Matthieu Zubialde-Elzaurdia*[1], Franck Lucas[1], Alain Bastide[2]

1 Laboratoire GEPASUD
University of French Polynesia
98702 Faa'a - Tahiti
French Polynesia
**Corresponding author:*
matthieu.zubialdeelzaurdia@doctorant.upf.pf

2 Laboratoire PIMENT
University of Reunion Island
117 rue du Général Ailleret
97430 Le Tampon – Reunion Island
France

ABSTRACT

Building energy simulation (BES) and Airflow network (AFN) programs generally incorporate wind pressure coefficients (C_p) estimated from secondary sources, namely data bases or analytical models. As these coefficients are influenced by a wide range of parameters, it is difficult to obtain reliable C_p data. This leads to uncertainties in BES-AFN models results, especially for naturally ventilated building studies, where air change rate which strongly depends on C_p, is a key value for thermal comfort and energy consumption results. This study focuses on naturally ventilated buildings in tropical climate and presents an alternative approach to estimate wind pressure coefficient. Computational fluid dynamics (CFD) simulations are performed to calculate wind-driven airflow rates at building opening level. Then wind pressure coefficient difference ΔC_p is calculated from large opening equation to be used as input data in BES-AFN program. Numerical simulations are performed for various wind directions on a typical cross-ventilated isolated building and a more complex building with opposite large openings. CFD results of wind pressure coefficient difference and airflow rate are compared to those obtained from AFN model using two different C_p sources. The results show that the calculated values vary greatly depending on the method used and highlight that an accurate estimation of wind pressure coefficient is a key parameter for evaluating natural ventilation in buildings.

KEYWORDS

Natural ventilation, computational fluid dynamics, airflow network model, wind pressure coefficient

1 INTRODUCTION

Tropical climate is characterized by relatively constant high temperature and humidity throughout the year, especially during summer (wet season). Providing thermally comfortable indoor environment in such conditions is seriously challenging. In most of the cases, building design response is based on active strategies to mechanically control indoor air temperature and humidity, thus resulting in a significant increase in the energy consumption of the buildings. Therefore passive strategies of sustainable architecture must be developed to reduce energy consumption in particular in tropical insular regions such as French Polynesia, where electricity production is highly polluting due to importation of fossil energy. In 2014, 279 000 tonnes of fossil fuels were imported, representing nearly 94% of the primary energy consumed by the country (ADEME, 2015).

Natural ventilation is a key strategy for the design of sustainable buildings in tropical climate. It addresses three distinct issues: thermal comfort, indoor air quality and energy savings. In order to quantify the impact of natural ventilation on thermal comfort and energy consumption, it is necessary to know the naturally driven ventilation rates. At building design stage, it is often done by using Building Energy Simulation (BES) tools coupled with Airflow Network Models (AFN) which allow to perform annual simulation of the entire building, calculating coupled airflow and heat transfer.

Airflow network models are generally based on the large opening equation describing a steady incompressible flow through an opening (Bernoulli's assumption). This equation is valid under the following assumptions:

- the turbulent flow is fully-developed;
- the pressure distribution on the building facades is not affected by the presence of openings;
- the pressure drop across the inflow and outflow opening is equal to the static pressure difference, i.e. the dynamic pressure in the room can be neglected.

For wind-driven cross-ventilation, the large opening equation can be written as follows:

$$Q = C_D . A . U_{ref} . \sqrt{C_{p_W} - C_{p_L}} \tag{1}$$

Where A is the equivalent opening area (m^2), C_D is the discharge coefficient, U_{ref} is the reference wind speed at building height (m/s), C_{p_W} and C_{p_L} respectively the pressure coefficient on windward and leeward facade.

Wind pressure coefficient describes the pressure distribution on the building surfaces. It is defined by the equation:

$$C_p = \frac{P_x - P_0}{P_d} \; ; \; P_d = \frac{\rho . {U_{ref}}^2}{2} \tag{2}$$

Where P_x is the static pressure at a given point on the building facade (Pa), P_0 is the static reference pressure (Pa), P_d is the dynamic pressure (Pa), and ρ is the air density (kg/m3)

Wind pressure coefficient is influenced by a wide range of parameters, including building geometry, facade detailing, position on the facade, sheltering elements, wind speed, wind direction and turbulence intensity. Therefore it is difficult to obtain reliable C_p data. Cóstola et al. (Cóstola et al., 2009) identify two main sources of C_p data.

Primary sources including full-scale measurements, wind-tunnel measurements and Computational Fluid Dynamics (CFD) simulations are considered to be the most reliable but also the most expensive and complex to realize. As wind tunnel experiments can rarely be carried out during building design stage, CFD can be an interesting alternative.

Secondary sources including databases and analytical models, are most often used in BES–AFN programs. For example, EnergyPlus uses the analytical model proposed by Swami and Chandra. Databases compile C_p data generally obtained from wind-tunnel measurements with the main objective of providing an indication on the range of C_p values for various basic buildings geometries and orientations. Analytical models are also developed based on wind-tunnel and full-scale experiments, using regression techniques to analyse a large amount of C_p data. They consist of a set of equations to calculate averaged C_p values but are still limited to a narrow range of parameters taken into account. So, influence of environment is not considered by these sets of equations.

Comparing pressure coefficients from different data sources, Cóstola et al. (Cóstola et al., 2009) concluded that for the same building in the same conditions, C_p values show large variations, even for simple configurations like fully exposed cubic buildings. Pressure coefficients are generally calculated considering buildings without openings and averaging pressure surface values over the building facades. Cóstola et al. (Cóstola et al., 2010) highlighted that the use of surface-averaged pressure coefficients can lead to uncertainty in airflow rate calculations. Karava et al. (Karava et al., 2006) investigated internal pressure coefficient in naturally ventilated building and found that this parameter has a significant impact on airflow prediction.

Discharge coefficient is also a source of uncertainty in the calculation of airflow rates. It characterizes the local contraction of the flow at opening level and shear stress forces and therefore depends on the characteristics of the fluid and on the shape and dimensions of the opening. For small sharp-edged openings, typical values for discharge coefficient are within the range 0.60–0.65 and are considered independent of the Reynolds number (Re). These typical values are generally used as constant in BES-AFN programs. However Karava et al. (Karava et al., 2004) concluded after reviewing the literature that the discharge coefficient shows considerable variation with opening porosity, configuration (shape and location in the facade), wind angle and Reynolds number. Yi et al. (Yi et al., 2019) using CFD method found that discharge coefficient depends on the wind direction and its values show large variations in the case of large openings.

The use of the large opening equation for the study of naturally ventilated buildings shows limitations due to coefficients evaluation. The underlying assumptions of this equation are not compatible with the large openings encountered with naturally ventilated buildings. Large openings affect pressure distribution and kinetic energy may be not fully-dissipated at the downstream of the opening.

The objective of this study is to present an alternative approach to estimate pressure coefficients for use in BES-AFN programs. The traditional method consists in calculating pressure coefficients as intermediate values from primary or secondary sources and using them as input data in BES-AFN programs to calculate airflows rates. The approach proposed in this study is based on the opposite reasoning and on the use of conservative quantities: airflow rates at building openings level are directly calculated form the CFD. From the large opening equation, the artificial pressure field can be deduced. This value will be used as input data in AFN models. The objective of this method is to eliminate the various coefficients and their associated uncertainties in order to construct only one coefficient adapted to the geometry of the studied building.

Using numerical simulations, three methods to calculate airflow rates of two different cross-ventilated buildings are compared: (I) directly calculated from CFD; (II) calculated from AFN using C_p values from CFD; (III) calculated from AFN using C_p values from Swami and Chandra analytical model.

2 COMPUTATIONAL METHODS

Whole-domain CFD approach is considered in this study, as outdoor and indoor airflow are modelled simultaneously and within the same computational domain in order to calculate airflow through the ventilation openings.

Steady-RANS computations are performed for various wind speeds and wind incident angles using the open-source library OpenFOAM, with a finite volume method for solving the flow equations. RNG k-e turbulence model is chosen based on previous work by Evola and Popov (Evola and Popov, 2006) who concluded that RNG model show good agreement with experimental data and thus “can be considered a useful tool for the study of air flow inside and around building when dealing with wind driven natural ventilation”. Standard k-e model and Realizable k-e model are also tested on two configurations to quantify discrepancies between the different models. SIMPLE algorithm is used to solve pressure-velocity coupling. Second order schemes are employed until residuals decrease by at least four order of magnitude.

For this study, various wind incident angles are considered in the range of 0°-90°. In order to analyse the possible impact of wind speed on the CFD results, simulations are performed for different wind speeds corresponding to a Reynolds number range from 50,000 to 500,000.

Numerical simulations are carried out on two different cross-ventilated buildings. The first one is a typical cubic building with reduced scale dimensions W x D x H = 0.1 x 0.1 x 0.8 m^3. Openings on opposite facades represent 10% wall porosity. Details of the geometry can be found in Figure 1. The second studied building is a typical school building that can be encountered in tropical climate, particularly in French Polynesia, with gable-roofed and large openings on opposite facades (representing more than 30% wall porosity) to favour natural ventilation. Building geometry is based on a rectangular floor plan with full-scale dimensions W x D x H = 9 x 7 x 4 m^3.

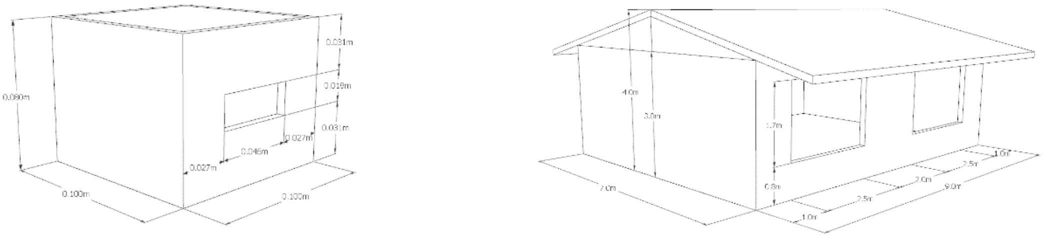

Figure 1: Building geometry description with on the left the cubic building and on the right the school building

Computational domain is constructed following the best practices proposed by Franke et al. (Franke et al., 2004): considering that H represents the height of the building, inlet, lateral and top boundary are 5H away from the building and outlet boundary is positioned at 15H behind the building.

Inlet boundary condition is imposed in order to generate an Atmospheric Boundary Layer (ABL). Velocity, turbulent kinetic energy and turbulence dissipation rate profiles are described by equations recommended by Richards and Hoxey (Richards and Hoxey, 1993):

$$U = \frac{u_*}{\kappa} ln\left(\frac{z+z_0}{z_0}\right) \tag{3}$$

$$k = \frac{{u_*}^2}{\sqrt{C_\mu}} \tag{4}$$

$$\varepsilon = \frac{{u_*}^2}{\kappa(z+z_0)} \tag{5}$$

Where κ is von Karman’s constant, z_0 is surface roughness length ($z_0 = 0.03$m), C_μ is a model constant and u_* is the friction velocity usually calculated from a specified velocity U_h at a reference height h as:

$$u_* = \frac{\kappa\, U_h}{ln\left(\frac{h+z_0}{z_0}\right)} \tag{6}$$

Symmetry boundary conditions are imposed at the sides and the top of the domain, implying zero normal velocity and zero gradients for all the variables at these boundaries. The outlet of the domain is described by a zero static pressure boundary condition. Building surfaces are treated with non-slip boundary condition and specific wall function for ABL provided by OpenFOAM is used for ground boundary.

For the cubic building, the computational grid is composed of about 800 000 cells. As the school building geometry is complex and in order to describe accurately airflow at building openings level, the computational grid is composed of more than 4 200 000 cells, of which more than 90% are hexahedral cells. Special care is taken to limit the distance from the centre point of the wall-adjacent cell on the ground and building surfaces. For the entire range of simulated wind speed, the maximum value of y^+ does not exceed 300.

To calculate the volume flow rate through the building openings, the opening surfaces are decomposed into triangulated surfaces. Then velocity is interpolated onto the triangles and integrated over the surface area. Triangles are smaller than mesh cell size for an accurate result. Using the large opening equation, artificial ΔC_p can be deduced as:

$$\Delta C_p = \left(\frac{Q}{C_d * A_w * u_{wind}}\right)^2 * sgn(Q) \tag{7}$$

With:

$C_d = 0.65$

$$\frac{1}{{A_w}^2} = \frac{1}{(A_1 + A_2)^2} + \frac{1}{(A_3 + A_4)^2}$$

The previous calculated volume flow rate is compared to the one calculated by airflow network (AFN) using the large opening equation and mean pressure coefficient obtained from two different sources: (i) CFD results analysis of building with openings and (ii) the analytical model proposed by Swami and Chandra.

$$C_{p,n} = 0.6\, ln \begin{bmatrix} 1.248 - 0.703\sin(\alpha/2) - 1.175\sin^2(\alpha) + 0.131\sin^3(2\alpha G) \\ +0.769\cos(\alpha/2) + 0.07G^2\sin^2(\alpha/2) + 0.717\cos^2(\alpha/2) \end{bmatrix} \tag{8}$$

Where:

$C_{p,n}$ = C_p value at a given angle between wind direction and the outward normal of the surface under consideration [dimensionless];

α = Angle between wind direction and outward normal of wall under consideration [deg];

G = Natural log of the ratio of the width of the wall under consideration to the width of the adjacent wall [dimensionless];

3 RESULTS OF NUMERICAL SIMULATIONS

The three different methods to calculate airflow rate and wind pressure coefficient differences ΔC_p are compared in this section according to the wind incident angle for the cubic building case and the school building case:

- method (1) corresponds to CFD results of airflow rate by integrating velocity over the opening surface of the building and calculating artificial pressure coefficient with equation (7);
- method (2) corresponds to AFN results of airflow rate with surface-averaged pressure coefficients calculated from CFD considering building with openings;
- method (3) corresponds to AFN calculation of airflow rate with surface-averaged pressure coefficients calculated from Swami and Chandra analytical model.

Figure 2 presents the variations of ΔC_p values and non-dimensional airflow rates as a function of wind direction for the cubic building case. Significant differences appear between the results of methods (1) and (2), particularly for wind direction normal to building openings where the discrepancy is about 40% on the ΔC_p value. As the angle of incidence increases the difference decreases to a minimum of 5% for a 45° wind direction and then increases again beyond this value. The consequence on the calculation of the airflow rate is an underestimation of the value calculated with method (2) compared to the actual flow rate calculated by the CFD for wind directions below 45° and on the contrary an overestimation of the flow rate for wind directions above 45°. Swami and Chandra's analytical model estimates a ΔC_p very similar to that of method 2 for 15° and 30° wind directions. Beyond a 30° wind direction the value of the ΔC_p is larger and therefore the air flow rate is greatly overestimated compared to the results of the other two methods.

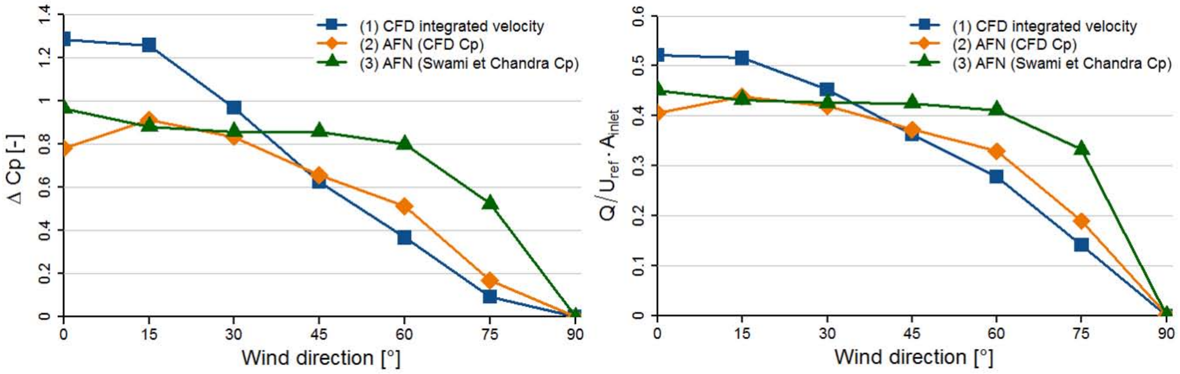

Figure 2: Wind pressure coefficient difference and non-dimensional airflow rate as a function of wind direction – results based on CFD simulation for the cubic building case

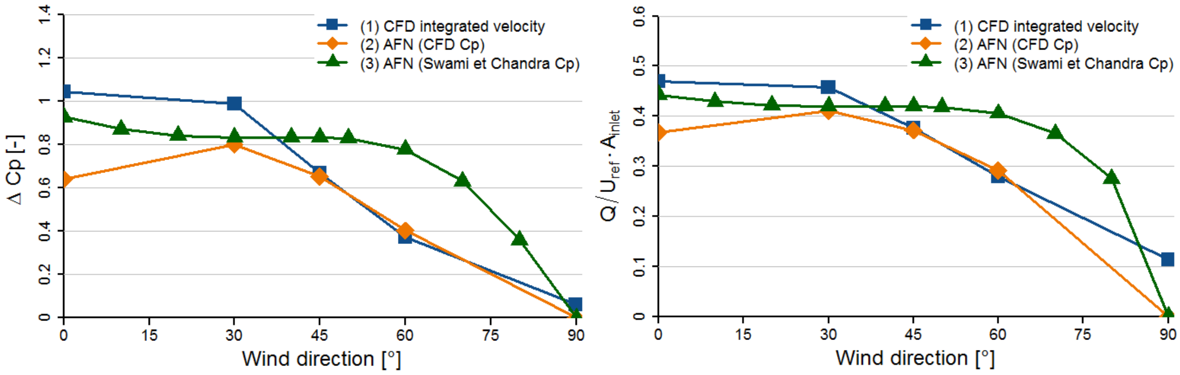

Figure 3: Wind pressure coefficient difference and non-dimensional airflow rate as a function of wind direction – results based on CFD simulation for the school building case

Figure 3 presents the results for the school building case. The same overall behaviour is observed for the evolution of the ΔC_p value and airflow rate as a function of wind direction. The differences between the results of methods (1) and (2) are of the same order for wind directions below 30°. Compared to method (1), method (2) still underestimates airflow rate by 20% for a 0° wind direction and by 10% for a 30° wind direction. For 45° and 60° wind directions, the results of these two methods are almost identical. For wind direction normal to the openings, results of method (1) are closer to those of method (3) for this building configuration compared to the cubic building configuration. For wind direction parallel to the openings, only method (1) is able to predict airflow rate through the building. Method (3) still calculates larger ΔC_p values for wind directions above 30° and thus overestimates airflow rates.

Turbulence models are compared in order to quantify the discrepancies about the calculation of the airflow rate through the school building openings. Table 1 presents the value of the non-dimensional airflow rate for a reference wind velocity corresponding to a Reynolds number Re ~ 50 000 and for two wind directions: 0° and 45°. For wind direction normal to the building openings, the three models calculate similar airflow values with a maximum difference of less than 5% between Realizable k-e model and RNG k-e model. For a 45° wind direction, Standard k-e and Realizable k-e calculate the same airflow value. This value differs by 5% from the value calculated by RNG k-e model.

Table 1: Comparison of results from various turbulence models for the calculation of non-dimensional airflow rate through the school building openings for a 0° and 45° wind direction.

		RNG k-e	**Std k-e**	**Realizable k-e**
Wind direction 0°				
Q_{tot} / U_{ref} . A_{inlet}	[-]	0.686	0.677	0.719
Discrepancy w/ RNG k-e	[%]		-1.3%	4.9%
Wind direction 45°				
Q_{tot} / U_{ref} . A_{inlet}	[-]	0.566	0.536	0.536
Discrepancy w/ RNG k-e	[%]		-5.3%	-5.3%

The influence of wind speed on the calculated artificial pressure coefficient ΔC_p from equation (7) is investigated for the school building case and shown on Figure 4. The results indicate that for all simulated wind directions the Reynolds number does not affect the value of the artificial pressure coefficient. Only very slight variations can occur for low Re values. This result supports what other studies have mentioned: pressure coefficient is normally assumed to be independent of wind speed.

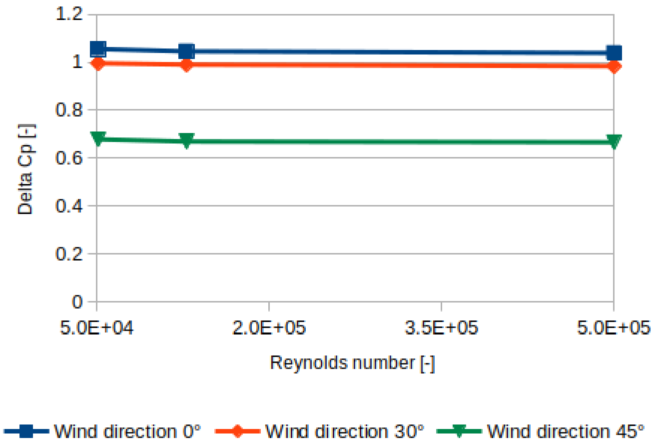

Figure 4: Wind pressure coefficient difference as a function of Reynolds number for three different wind directions – results based on CFD simulation

4 CONCLUSIONS

Large opening equation used in AFN models is not suitable for studying naturally ventilated buildings with large openings. In this equation, wind pressure coefficient is a key parameter and its estimation is a source of uncertainty in the calculation of airflow rates. This study proposes an alternative approach for estimating pressure coefficients that will be used as input data in BES-AFN programs. CFD RANS simulations are carried out on two different cross-ventilated buildings for various wind directions in order to calculate airflow rates at building opening level and wind pressure coefficient differences. The results are compared to AFN model calculation using C_p data from primary and secondary sources.

It can be concluded that airflow rates calculated from AFN model using surface averaged C_p values from CFD analysis (considering building with openings) are different from those directly calculated by CFD. In the case of wind direction normal to building opening, the discrepancy is more than 20% between the two calculated airflow rates. Such a difference can be significant on the estimation of thermal comfort of a naturally-ventilated building.

The alternative method proposed in this study has the advantage of calculating a ΔC_p value directly related to the flow rate calculated by the CFD. Therefore it avoids the uncertainties associated with the estimation of wind pressure coefficient and discharge coefficient. Future work will focus on experimental validation with PIV measurements to assess airflow rate calculation from CFD.

5 ACKNOWLEDGEMENTS

The author would like to acknowledge the *Service de l'énergie du gouvernement de la Polynésie française (SDE)* for the financial support of his PhD thesis.

6 REFERENCES

ADEME, 2015. Plan Climat Energie de la Polynésie française.

Cóstola, D., Blocken, B., Hensen, J.L.M., 2009. Overview of pressure coefficient data in building energy simulation and airflow network programs. Build. Environ. 44, 2027–2036. https://doi.org/10.1016/j.buildenv.2009.02.006

Cóstola, D., Blocken, B., Ohba, M., Hensen, J.L.M., 2010. Uncertainty in airflow rate calculations due to the use of surface-averaged pressure coefficients. Energy Build. 42, 881–888. https://doi.org/10.1016/j.enbuild.2009.12.010

Evola, G., Popov, V., 2006. Computational analysis of wind driven natural ventilation in buildings. Energy Build. 38, 491–501. https://doi.org/10.1016/j.enbuild.2005.08.008

Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W., Schatzmann, M., 2004. Recommendations on the use of CFD in wind engineering.

Karava, P., Stathopoulos, T., Athienitis, A.K., 2006. Impact of Internal Pressure Coefficients on Wind-Driven Ventilation Analysis. Int. J. Vent.

Karava, P., Stathopoulos, T., Athienitis, A.K., 2004. Wind Driven Flow through Openings – A Review of Discharge Coefficients. Int. J. Vent. 3, 255–266. https://doi.org/10.1080/14733315.2004.11683920

Richards, P.J., Hoxey, R.P., 1993. Appropriate boundary conditions for computational wind engineering models using the k-e turbulence model. J. Wind Eng. Ind. Aerodyn., Proceedings of the 1st International on Computational Wind Engineering 46, 145–153. https://doi.org/10.1016/0167-6105(93)90124-7

Yi, Q., Wang, X., Zhang, G., Li, H., Janke, D., Amon, T., 2019. Assessing effects of wind speed and wind direction on discharge coefficient of sidewall opening in a dairy building model – A numerical study. Comput. Electron. Agric. 162, 235–245. https://doi.org/10.1016/j.compag.2019.04.016