

MODELING NATURAL VENTILATION IN EARLY AND LATE DESIGN STAGES: DEVELOPING THE RIGHT SIMULATION WORKFLOW WITH THE RIGHT INPUTS

Maria Alejandra Menchaca Brandan¹, and FA Dominguez Espinosa²

¹Thornton Tomasetti, United States of America

²Arup, New York, United States of America

ABSTRACT

Designing natural ventilation systems involves different levels of sophistication in calculations and software tools according to the system complexity and the stage of the design process. This paper provides a general workflow to model these systems, while describing the types of resources available for different complexities and design stages. Inputs needed at each step of the workflow are discussed, and, when not available anywhere else, ways to compute them are also presented. Particularly, this paper shows how to account for the ventilation efficiency of various window types and pressure losses of obstructions along the airflow path when their discharge coefficient is unknown.

NOMENCLATURE

Q	Volumetric flowrate
C_d	Opening discharge coefficient
A	Cross-sectional area to airflow, reference area
ΔP	Pressure difference between intake and outlet
P_{ref}	Reference pressure at CFD domain intake
P_{gauge_x}	Relative pressure of the air at a given point x in the façade with respect to ambient reference pressure
C_w	Wind pressure coefficient
U	Upstream wind speed
Z	Opening height measured from ground level
a	Exponent associated with building terrain
γ	Exponent associated with building terrain
δ	Atmospheric boundary layer thickness of building terrain
ΔT	Temperature difference between indoors and outdoors

T_{ave}	Average absolute temperature between indoors and outdoors
q	Heat gains in the space
ΔH	Height difference between the mid-heights of a top and a bottom opening
H	Height of the window from sill to head
f_{contr}	Contraction friction factor
f_{exp}	Expansion friction factor
ρ	Air density
C_p	Specific heat capacity of air
g	Gravity constant

Underscores

in	Variable at intake / inlet
out	Variable at outflow / outlet
N	Variable at n^{th} flow element
eff	Effective value
x	At a given point x on the building façade
z	At an opening of height z
met	Value measured at meteorological station

INTRODUCTION

Natural ventilation systems, when adequately designed, can lead to significant energy savings in building energy use, increase occupant productivity by increasing space ventilation (Loftness et al., 2007), and act as a resiliency strategy for both cooling and ventilation when the HVAC power goes out.

Some of the challenges to design a well-functioning natural ventilation system include its high dependency on building and opening geometry, constantly changing ambient conditions, a lower level of control of indoor thermal comfort levels, and a lack of tools to adequately inform the design process from its early stages.

This paper aims to clarify the simulation workflow for a natural ventilation system, no matter how simple or

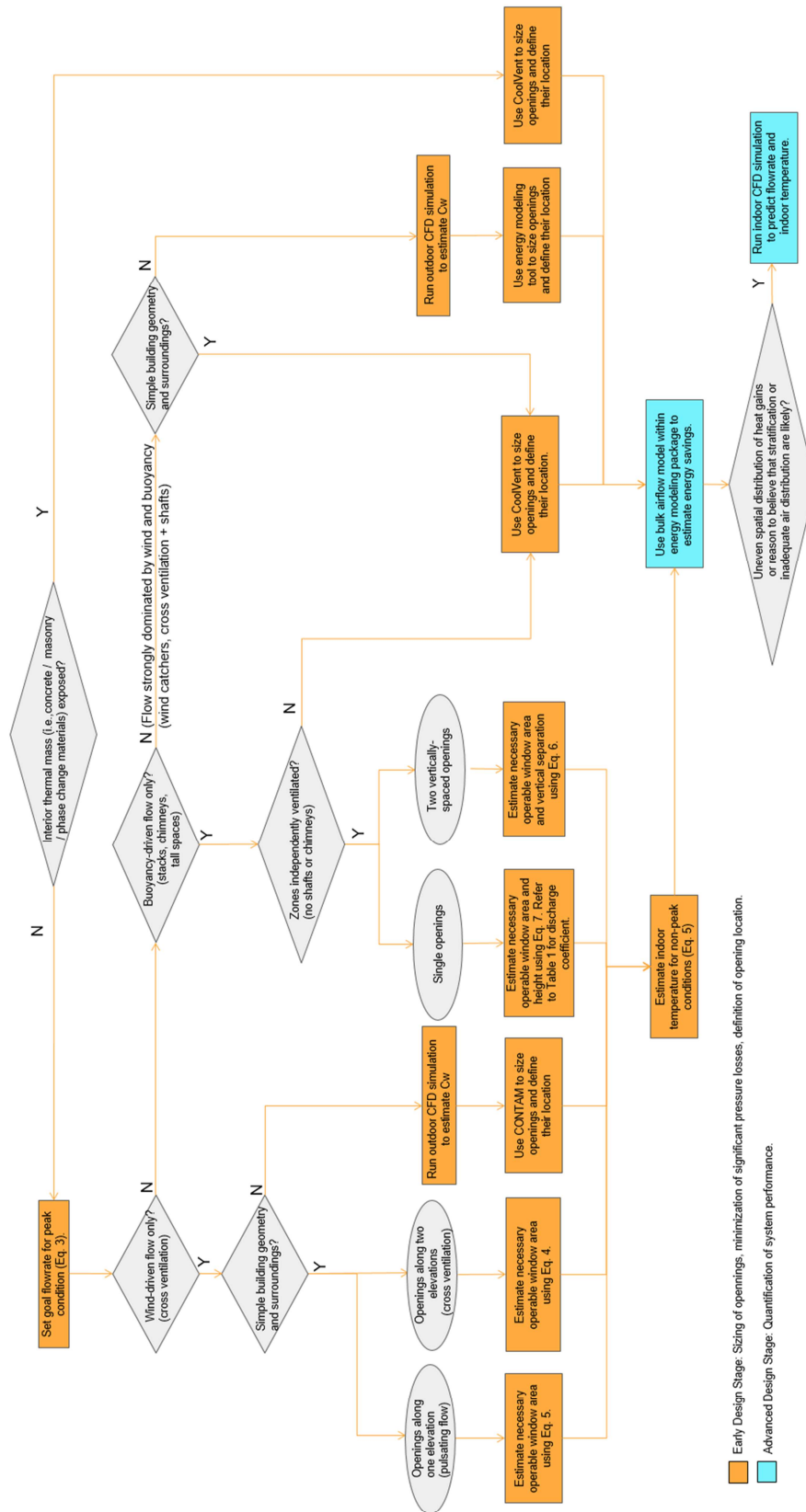


Figure 1 Natural Ventilation Simulation Workflow

complex, as well as to provide a concise summary of the most critical inputs needed for different simulation strategies to obtain reliable predictions from them.

DESIGN WORKFLOW

The first task when designing a natural ventilation system is to perform a climate analysis based on hourly outdoor data, such as the one proposed by Axley (2001), and implemented in the Climate Suitability Tool (Emmerich, 2011).

The climate analysis should also be used to decide whether the natural ventilation system will be mostly driven by wind (locations with consistent wind directions year-round, and a building program that allows for cross-ventilation), by buoyancy (locations with unpredictable wind patterns, weak wind speeds, with programs where cross ventilation is not an option, or where a solar chimney is an option), or by both.

Once the climate has been deemed favorable and a driving force has been selected, the designer has to evaluate whether the building typology and design are adequate for natural ventilation, and if it can be used year-round or must be mixed-mode. The design workflow provided by CIBSE Guide AM10 (CIBSE, 2005) can be useful to perform this evaluation.

Finally, simulation tools can aid in sizing the natural ventilation system to maximize its potential during the expected hours of operation. For this, the following simulation workflow is proposed.

Simulation Workflow

The goal in early design stages is to *size* the natural ventilation system correctly. Running complex and time-consuming energy simulations to estimate savings should be avoided at this stage. Rather, the design team should focus on optimizing the size and location of openings by using the right simulation tools.

Figure 1 shows a simulation workflow that guides the designer through the tools and assumptions required to size a natural ventilation system early in the design process and to quantify its performance later in the design. The different tool types and calculations listed in the chart will be described in the following sections.

Depending on the situation, one of three types of tools are recommended for use: hand calculations, multi-zone airflow network tools and computational fluid dynamics (CFD).

The rest of this document will focus on outlining the characteristics of each of these tools as well as how to obtain the most critical inputs so that their outputs are reliable to the modeler.

MODELING TOOLS

Three main sets of tools can be used to model the impact of natural ventilation on energy savings, thermal comfort and indoor air quality: hand calculations, multi-zone airflow models and computational fluid dynamics. Each of these tools has its advantages and disadvantages and are more appropriate to be used in certain stages of building design.

Hand Calculations

The driving forces for natural ventilation can originate from wind or buoyancy, which create a driving pressure difference between the space's inlet and outlet. To quantify the amount of air entering through a *series* of openings, e.g. windows and vents, given an overall pressure difference, the orifice equation is used:

$$Q = (A C_d)_{eff} \sqrt{\frac{2}{\rho} \Delta P} \quad (1)$$

where:

$$(A C_d)_{eff} = \frac{1}{\sqrt{\sum_1^N \frac{1}{(A_i C_{d_i})}}} \quad (2)$$

Once the flowrate is known, interior temperatures can be estimated using the so-called “well-mixed temperature” assumption, via a simple energy balance:

$$T_{zone} = T_{out} = T_{in} + \frac{q}{\rho Q C_p} \quad (3)$$

Guidance on selecting the right inputs for each equation outlined in this section is provided in the “Model Inputs” section of this paper.

Wind-Driven Ventilation

If ventilation is driven purely by steady wind speeds impinging on a façade, the resulting flow is expressed as:

$$Q_{wind} = (A C_d)_{eff} \sqrt{U^2 |C_{win} - C_{wout}|} \quad (4)$$

When ventilation is driven by wind-originated flow pulsations on a single-side opening, the flowrate can be estimated by:

$$Q_{wind\ puls} = \frac{C_d l \sqrt{C_w} \alpha U}{z_{ref}^\gamma} \int_{z_0}^h \sqrt{z^{2\gamma} - z_0^{2\gamma}} dz \quad (5)$$

where z_{ref} is the height at which U is measured, and z_0 is the vertical location of the neutral plane. Note that the impact of turbulent eddies on the flow along single-sided openings can also be relevant for some wind conditions.

More details on calculations can be found in Wang and Chen (2012).

Buoyancy-Driven Ventilation

When natural ventilation is driven by the temperature (and hence density) difference between indoor and outdoors, the flowrate can be estimated through two different expressions, depending on the nature of the openings.

For a zone with two vertically-spaced openings:

$$Q_{\text{buoy, 2openings}} = (A C_d)_{\text{eff}} \sqrt{2g \Delta H \frac{\Delta T}{T_{\text{ave}}}} \quad (6)$$

For a zone with a single opening (bi-directional flow):

$$Q_{\text{buoy, bi-directional}} = A C_d \sqrt{2g H \frac{\Delta T}{T_{\text{ave}}}} \quad (7)$$

In the case of buoyancy-driven flows, the temperature difference between indoors and outdoors (energy balance, Eq. 3) and the flowrate (Eqs. 6 and 7) are coupled and cannot be solved separately. Therefore, for single zones ventilated through buoyancy and where thermal mass effects are not significant, the following approximations can be used to size and locate openings:

For two vertically-spaced openings:

$$Q = \left((A C_d)_{\text{eff}} \sqrt{2g \Delta H \frac{q}{\rho C_p T_{\text{ave}}}} \right)^{2/3} \quad (8)$$

For a single opening (bi-directional flow):

$$Q = \left(A \frac{C_d}{3} \sqrt{2g H \frac{q}{\rho C_p T_{\text{ave}}}} \right)^{2/3} \quad (9)$$

The minimum window area to meet a comfortable indoor temperature can be obtained by solving either Eqs. 6 or 7 along with Eq. 3.

It is not uncommon to find designs that rely on a combination of wind and buoyancy forces (and, often, the additional pressure of an assisting fan) to drive the flow, and through multiple zones and openings at a time. In those cases, the simple hand calculations outlined in this section cannot be used to predict flowrates, instead, the designer relies on simulation tools based on numerical methods to adequately predict the flow through one or more spaces.

Airflow Network Solvers

When there is a need to design natural ventilation systems that involve multiple stories, rooms with several

openings, or heavily massive construction, airflow network solvers (also known as bulk airflow models, or multi-zone models) are the ideal tools to use. Multi-zone solvers model each space as a node of uniform pressure and temperature in a network, and find the pressure differences that result in flowrates that ensure mass and energy conservation between the nodes. These solvers are relatively fast, thus making them suitable for the early design stages of complex natural ventilation systems.

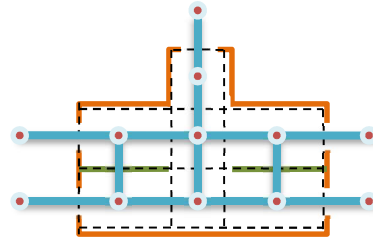


Figure 2 Representation of nodes and flowpaths of a bulk airflow network model

Note that while some solvers are able to jointly solve an energy and a mass balance (Dominguez Espinosa et al., 2014), some others require to be coupled to an energy modeling tool to iteratively solve the flow equations.

These solvers have several advantages: they can use hourly weather data to assess annual ventilation performance as well as associated energy savings and indoor comfort levels. They can also quantify the effect of thermal mass on the performance of the system, accounting for variable occupancy, equipment and solar loads on a zone-by-zone basis. Finally, they can simulate the effect of fan-assisted ventilation.

Amongst their limitations are that most solvers require being coupled to an energy modeling tool, and often they do not allow the user to visualize the flow through the system (and thus understand critical aspects of the design such as the location of the neutral pressure plane or any risks of backflow in certain zones). Thus, designers relying on these solvers are often constrained to using them as a verification tool rather than a design tool. On the other hand, solvers that allow to model the physics of natural ventilation without the need to build an energy model, cannot be easily used to quantify the annual energy performance of the system.

Another limitation of these tools is that their ability to estimate appropriate wind pressure coefficients is rather limited, since they—at best—estimate wind pressure coefficients using the methodology by Swami and Chandra (1988). Therefore, they neglect the impact of the surroundings in the pressure distribution along the building's façade, and are only valid if the building has a simple shape. Moreover, because wind pressure coefficients depend on wind direction, running an annual

simulation accounting for wind-driven flow would require providing the model with coefficients for each wind angle, currently an intractable task. Consequently, the authors recommend caution with results from these models that take wind data from a weather file, since the solvers' inability to adequately calculate wind pressure coefficients can lead to misleading results.

Finally, these solvers typically take as an input the free operable area for each opening and do not account for pressure losses associated with insect screens, fire dampers or transfer ducts. Most frequently the only way to account for these pressure losses in the simulation is by providing an "effective area" for each opening instead. Eq. 2 can be used for that purpose.

Two examples of airflow networks tools are CoolVent (Dominguez Espinosa et al., 2014) and CONTAM (Walton and Dolls, 2006). CoolVent can account for thermal mass effects as well as buoyancy-driven flow, however it is not ideal for wind-driven scenarios where the building geometry or surroundings are complex. CONTAM is used mainly for wind-driven flow, because it can be used to simulate complex wind pressure conditions when the wind pressure coefficients along each façade are obtained through CFD. It cannot model buoyancy-driven flow, however, unless coupled to an energy modeling engine.

Computational Fluid Dynamics (CFD)

CFD is a powerful modeling tool that divides a domain into thousands or millions of elements and solves the mass, momentum and energy conservation equations between them. Due to their higher level of discretization, CFD results are significantly more detailed than those of multi-zone models, so information unavailable in typical multi-zone results, including smaller areas of uncomfortable temperature or of stagnant flow in a room, can be resolved using CFD. Moreover, CFD can also be used to study transport of different species, including contaminants and rain.

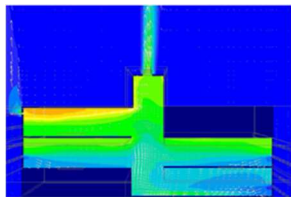


Figure 3 Indoor CFD simulation

Nevertheless, CFD simulations can be computationally intensive, so, currently, it is not feasible to use them to model hourly variations and quantify annual performance of a building. Instead, simulations are typically run for a particular point in time that is relevant to the design.

Natural ventilation design can benefit from CFD simulations throughout the entire process. In the earlier design stages, it can be used to model flow outside of a building to determine wind pressure coefficients and to assess the impact that favorable and unfavorable wind directions can have on the flow patterns inside the building.

In the later design stages, CFD can be instrumental to quantify the pressure losses associated with specific elements along the flowpath, such as windows, louvers or sound attenuators. CFD simulations are also crucial to quantify the performance of complex natural ventilation systems which airflow network tools are not designed to do, such as solar chimneys. Finally, CFD can be used to ensure that the airflow and temperature are properly distributed, thereby providing adequate thermal comfort in the space.

Properly setting up and running a CFD simulation requires a relatively high level of technical expertise to make sure that the correct boundary conditions, mesh size, and turbulence and radiation models have been used.

The user of CFD tools needs to generate high quality meshes and must perform grid studies (Stern et al., 2001). Great care should be taken when meshing the region near solid surfaces such as walls, occupants and furniture, where viscous effects are important.

Selecting the most appropriate turbulence model for the CFD simulation is also critical to obtain accurate results. While the RNG k-ε model is commonly used for both indoor and outdoor simulations (Chen, 1995; Ray, 2012), modeling turbulent eddies to replicate pulsating flow requires more computationally-intensive models such as large-eddy simulation (Jiang et al, 2003).

Finally, radiative heat transfer should be modeled in all indoor CFD simulations. Failing to account for radiation in CFD has been demonstrated to result in temperature profiles, air flow patterns and velocities that are not realistic (Menchaca-Brandan et al., 2017).

MODEL INPUTS

While the equations driving natural ventilation flow are well known, the inputs to these calculations, applicable to hand calculations and airflow network models, are often loosely defined in the literature. This section provides a guide to obtain or compute these values. Note that how to quantify some of these variables is, in some cases, very well understood and standardized in the simulation community. In other cases, however, the variables have been left to the interpretation of the user and are often defined differently by various consultants and academics.

Wind Pressure Coefficient (C_w)

Plenty of literature is available on how to obtain wind pressure coefficients along façades. Swami and Chandra (1988) provide a comprehensive model for rectangular building of various dimensions without surroundings, which most energy modeling packages coupled with an airflow network tool use.

Wind pressure coefficients can be obtained more accurately, but not without a large computational effort, through outdoor CFD simulation using Eq. 10.

$$C_{wx} = \frac{P_{gauge}}{\frac{1}{2} \rho U_x^2} = \frac{P_x - P_{ref}}{\frac{1}{2} \rho U_x^2} \quad (10)$$

Upstream Wind Speed (U_z)

Estimating the local upstream wind speed for an opening at height Z from meteorological weather data is typically done using the following power-law model:

$$U_z = U_{met} \left(\frac{\delta_{met}}{Z_{met}} \right)^{\alpha_{met}} \left(\frac{Z}{\delta} \right)^{\alpha} \quad (11)$$

Where α is an exponent and δ is the boundary layer thickness, all associated with the terrain type, and the values of which can be found in Table 1 of chapter 24 of the ASHRAE Handbook of Fundamentals (2017).

Window free area (A_f)

When dealing with a sliding window, the free area of the opening to use in calculations is simply the width times the height of the opening. When using hinged / pivoting windows (casement, awning, center pivot, etc), however, the free area must be calculated more carefully. A common calculation for free area of pivoting windows found in the literature (von Grabe, 2013), involves the sum of the triangular openings growing from the hinge towards the open edge of the window, and that of the rectangular plane connecting the open edge of the window to the frame, until this sum reaches the area of the window in fully open mode (Figure 4, left), as given by Eq. 12.

$$A = \min(LW, L^2 \sin\theta + LW\sqrt{2(1 - \cos\theta)}) \quad (12)$$

Where L is the length of the window, perpendicular to its hinged edge, W is the length of the hinged edge, and θ is the opening angle of the window.

An alternative calculation is used by Wang et al (2015), where the sum of A_1 , A_2 and A_4 result in the operable area of the window (Figure 4, right).

In practice, the difference in predicted area between each approach is small enough that whichever model is selected, it does not significantly affect early design calculations of flowrate.

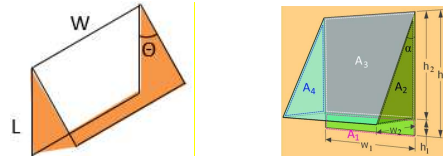


Figure 4 Two approaches to estimating opening area of pivot windows. Left: von Grabe (2013), right: Wang et al. (2017)

Note that the area reduction due to additional flow restrictions is typically accounted for by modifying the discharge coefficient of the opening (see next section), and not the free unobstructed area.

Discharge coefficient (C_d)

The discharge coefficient quantifies the relation between air flowrate and the pressure drop suffered by the air stream as it flows through an orifice.

Window elements with unidirectional flow

The discharge coefficient for a typical window opening with unidirectional flow has been estimated to be in the range of 0.60 to 0.65 (CIBSE, 2005; Etheridge and Sandberg, 1996). Using a discharge coefficient of 0.60 is recommended as a conservative approach.

Window elements with bi-directional flow

When dealing with bi-directional flow, the standard discharge coefficient is typically multiplied by a reduction factor that is associated with both the nature of the flow and the geometrical vertical area distribution of the window. For a sliding non-pivoting window with buoyancy-driven bi-directional flow, the discharge coefficient is 0.21 (Etheridge and Sandberg, 1996).

For pivoting windows under buoyancy-driven ventilation, the discharge coefficients can be inferred from the results provided by vonGrabe (2013) and summarized in Table 1. Note that these results were obtained for a single width-to-height opening ratio, and may not be applicable to windows with geometries that deviate significantly from it. Alternatively, CIBSE Guide 3 (CIBSE, 2006) provides a factor that modifies both the window area and discharge coefficient of pivoting windows.

Table 1 Discharge coefficient comparisons for different window types (inferred from van Grabe, 2013)

Window type				
C_d	0.13	0.09	0.11	0.16

In the case of bi-directional flow associated with wind-driven pumping ventilation, Wang et al. (2015) provide a semi-empirical model to quantify the impact of window type (awning, hopper and casement) on the resulting discharge coefficient.

For non-window elements

Traditionally, in naturally ventilated buildings the most common connection between two zones has been a window, however other connecting elements, such as shaft inlets and noise filters, have become more prevalent in modern buildings with hybrid natural-mechanical ventilation systems (Ray et al., 2014).

While collections of discharge coefficients for different types of openings and connections are available in the relevant literature (e.g. CIBSE, 2006), the modeler of naturally ventilated buildings faces two challenges when using these tables. The first one is to ensure that the definition of the discharge coefficient found in tables is compatible with Eq. 1. The second challenge is that these tables are not exhaustive, and rarely include the discharge coefficient of common elements in modern naturally ventilated buildings.

Fortunately, other metrics of flow resistance can be transformed into an equivalent discharge coefficient to be used in multi-zone tools. The Handbook of Hydraulic Resistance (Idelchik, 2005) is an example of an extensive collection of flow resistance measurements for different types of flow elements that can be used in this manner.

In the Handbook of Hydraulic Resistance (Idelchik, 2005) the flow resistance for a flow element is expressed in terms of the friction factor based on *total* pressure, f . This friction factor can be related to the discharge coefficient commonly used in multi-zone tools by computing the *static* pressure drop along a connecting path between two zones and recasting the resulting expression in the form of Eq. 1. The result is given by the following expression:

$$C_d = \left(\sqrt{\frac{A}{A_{in}} f_{in} + \sum_n \frac{A}{A_n} f_n + \frac{A}{A_{out}} f_{out}} \right)^{-1} \quad (13)$$

where f_{in} is the friction factor related to the contraction of the air in the zone of origin of the airflow as it enters the connecting path, f_{out} is the friction factor related to the expansion of the air as it enters the zone where it is discharged, f_n is the friction factor of the n th element in the path (e.g. a ducted section with sizeable viscous losses). A_{in} , A_{out} and A_n are cross-sectional areas to airflow of the inlet and outlet of the connecting path, and of the n th element in the path, respectively. Finally, A is an area of reference, which can be selected arbitrarily.

This reference area is also the same one used in the multi-zone tool to represent the path.

The friction factor associated with the expansion, f_{out} , is by definition, equal to 1, since it was assumed that all the dynamic pressure of the air jet is transformed into static pressure. This holds true regardless of the shape of the orifice (White, 2008). The contraction friction factor, f_{in} , however, depends on the shape of the entrance to the connecting path. If no other information is available, a first order estimate of the discharge coefficient can be obtained using $f_{in} \approx 0.35$.

Eq 13 can also be used to estimate the discharge coefficient of a connecting path that includes an element with unknown friction factor, but when experimental measurements of pressure drop versus flowrate are available, by making use of the mathematical definition of the friction factor.

Fan-driven pressure

Frequently natural ventilation is assisted by exhaust fans. In general, a fan causes a pressure rise in the flow, which is related to the volumetric flowrate through the fan by its characteristic curve. It is possible to fit a polynomial function to the fan curve, such that the flowrate of the fan is expressed as a function of its pressure rise. Generally, this is possible only for a limited, albeit important, range of pressures.

The volumetric-pressure rise function changes when operating the fan at different velocities. The manufacturer of the fan could provide a characteristic curve for different operating velocities, which can then be fitted to polynomials. If this is not possible, then fan laws can be used to estimate such curves at different fan speeds.

Accounting for a fan in otherwise naturally ventilated buildings is challenging. Thus, these situations are generally handled with multi-zone models.

SUMMARY / CONCLUSIONS

Designing, modeling and analyzing a natural ventilation systems require the use of very diverse hand calculations and modeling tools. Knowing how and when to use them to properly size a natural ventilation system is critical. A simulation workflow that coherently organizes the most important design tool types, their advantages and disadvantages as well as the key inputs that they require to produce accurate results is introduced in this paper. This workflow is a roadmap that engineers and designers can use to define the best tool to aid them to design naturally ventilated buildings. The main inputs needed for each tool type are also discussed. Even though most of these inputs are well understood, some of them are not defined clearly and consistently in the relevant literature

or are available only in a limited manner. This paper introduces strategies to compute or estimate some of these inputs.

In particular, detailed guidance is provided regarding determining the right window free area and discharge coefficient for different window types. Finally, the derivation of a simple equation to estimate the discharge coefficient of flow elements, such as noise filters and insect screens, that are becoming more prevalent in hybrid mechanically- and naturally-ventilated buildings is presented in this paper.

While the research on how to best model the physics of natural ventilation is ever-evolving, this paper offers up-to-date information on which modeling tool to rely on, how to define their inputs and when in the design process to use them to allow for greater design iterations and maximize system performance.

REFERENCES

- American society of heating, refrigerating and air-conditioning engineers (2017). *Handbook of HVAC Fundamentals. Inc.: Atlanta, GA, USA.*
- Axley, James W. (2001) Application of Natural Ventilation for US Commercial Buildings—climate Suitability, Design Strategies & Methods, Modeling Studies. Gaithersburg, MD, *NIST*.
- CIBSE. (2005). Natural ventilation in non-domestic buildings. *The Chartered Institution of Building Services Engineers, London, UK.*
- CIBSE. (2006). Environmental design. *The Chartered Institution of Building Services Engineers, London.*
- Dominguez Espinosa, F.A., Menchaca-Brandan, M. A., Ray, S., Glicksman, L. (2014). CoolVent, a simple robust simulation tool for natural ventilation design and analysis. *Proceedings of Roomvent 2014*, 1, 550-557.
- Etheridge, D. W., & Sandberg, M. (1996). *Building ventilation: theory and measurement* (Vol. 50). Chichester, UK: John Wiley & Sons.
- Idelchik, I. E. 2005. *Handbook of Hydraulic Resistance*, 3rd Edition.
- Jiang, Y., Alexander, D., Jenkins, H., Arthur, R., & Chen, Q. (2003). Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, 91(3), 331-353.
- Loftness, V., Hakkinen, B., Adan, O., Nevalainen, A. (2007). Elements That Contribute to Healthy Building Design. *Environmental Health Perspectives*, 115, 965-970
- Menchaca Brandan, María Alejandra. (2012) *Study of airflow and thermal stratification in naturally ventilated rooms*. Diss. Massachusetts Institute of Technology.
- Menchaca-Brandan, M. A., Dominguez Espinosa, F.A., Glicksman, L. R. (2017). The influence of radiation heat transfer on the prediction of air flows in rooms under natural ventilation. *Energy and Buildings*, 138, 530-538.
- Ray, S., Menchaca-Brandan, M. A., Dominguez Espinosa, F. A., Fukuda, M., Glicksman, L. 2014. Measured performance of naturally ventilated commercial building compared to analysis from CoolVent design tool. In *Proceedings of Roomvent 2014*.
- Stern, F., Wilson, R. V., Coleman, H. W., Paterson, E. G. (2001). Comprehensive Approach to Verification and Validation of CFD Simulations-Part 1: Methodology and Procedures. *Journal of Fluids Engineering*, 123, 793-802.
- Swami, M. V., & Chandra, S. (1988). Correlations for pressure distribution on buildings and calculation of natural-ventilation airflow. *ASHRAE transactions*, 94(3112), 243-266.
- von Grabe, J. (2013). Flow resistance for different types of windows in the case of buoyancy ventilation. *Energy and Buildings*, 65, 516-522.
- Walton, George, and William S. Dols. (2006) *CONTAM 2.4 user guide and program documentation*. No. NIST Interagency/Internal Report (NISTIR)-7251. 2006.
- Wang, H., & Chen, Q. (2012). A new empirical model for predicting single-sided, wind-driven natural ventilation in buildings. *Energy and Buildings*, 54, 386-394.
- Wang, H., Karava, P., & Chen, Q. (2015). Development of simple semiempirical models for calculating airflow through hopper, awning, and casement windows for single-sided natural ventilation. *Energy and Buildings*, 96, 373-384.
- Wang, J., Wang, S., Zhang, T., & Battaglia, F. (2017). Assessment of single-sided natural ventilation driven by buoyancy forces through variable window configurations. *Energy and Buildings*, 139, 762-779.
- White, F. (2008). *Fluid Mechanics*. 6th Ed