CFD-Based Analysis of the Discharge Coefficient for Buoyancy-Driven Ventilation in a Full-Scale Operational Building

Lup Wai Chew*, Chen Chen, Catherine Gorlé
Stanford University, Stanford, USA
*Corresponding author: lupwai@stanford.edu

Abstract
Natural ventilation can play a key role in reducing building energy consumption for ventilation and cooling. The ventilation rate across an opening can be characterized using a still-air discharge coefficient, \( C_d \), which is often assumed to be constant for a given opening. We aim to evaluate this constant \( C_d \) assumption for the buoyancy-driven natural ventilation process in Stanford’s Y2E2 Building. Computational fluid dynamics (CFD) simulations, specifically large eddy simulations, are used to calculate the flow rate through each window and determine the corresponding \( C_d \). The validated CFD results show that \( C_d \) decreases linearly with Reynolds number (\( Re \), calculated based on the flow rate across the windows) when \( Re < 10,000 \) but stays constant around 0.51 when \( Re \geq 10,000 \). As a result, assuming a constant \( C_d \) to estimate natural ventilation rate can introduce significant errors. Widespread use of CFD would support a more accurate characterization of the dependency of \( C_d \) for \( Re \) for specific building geometries and support robust natural ventilation system design.

Key Innovations
- Large eddy simulation of a real operational building of complex geometry with validation from measurements.
- Discharge coefficient is not a constant when the Reynolds number is below a critical value (\( Re = 10,000 \) in the present case).
- Natural ventilation rates could be overestimated by a factor 2 if we assume a constant discharge coefficient.

Practical Implications
Assuming a constant discharge coefficient, \( C_d \), in designing natural ventilation systems could lead to large errors, especially when the ventilation flow rate is low. CFD simulations can provide a more accurate characterization of \( C_d \).

Introduction
Buildings made up 32% of the total global energy consumption in 2010 (Intergovernmental Panel on Climate Change, 2014). In tropical countries such as India, air conditioning can account for more than half of the building energy consumption (Berardi, 2015). Natural ventilation does not consume energy and could mitigate the unsustainable energy demand for space cooling. However, the design of natural ventilation systems is extremely challenging with many unpredictable parameters including the outdoor conditions and occupant behavior (Chen, 2009; Etheridge, 2011). The ventilation rate across an opening is characterized by the still-air discharge coefficient, \( C_d \), which can be estimated experimentally. Once determined, \( C_d \) is often specified as a constant in empirical models used for the designs of natural ventilation systems.

Several studies have investigated \( C_d \) associated with wind-driven ventilation, as summarized in the review by Karava et al. (2004). However, there are fewer works focusing on buoyancy-driven ventilation. Among these works, the validity of the constant \( C_d \) assumption remains a subject of debate. On one hand, Flourentzou et al. (1998) measured buoyancy-driven ventilation flow across a three-story building and concluded that \( C_d = 0.6 \pm 0.1 \). Wilson and Kiel (1990) conducted experiments in a full-scale room with single doorway opening and concluded that \( C_d \) remains a constant around 0.6. Partridge and Linden (2013) performed experiments in reduced-scale models and reported that \( C_d \) is insensitive to the strength of the buoyant driving force. On the other hand, Holford and Hunt (2001) found that \( C_d \) is a function of Reynolds number (\( Re \)) at \( Re < 4000 \) and only becomes a constant at \( Re \geq 4000 \). A series of controlled experiments in Heisellberg et al. (2001) reported that \( C_d \) is dependent on \( Re \) as well as window type. Despite these works showing that the constant \( C_d \) assumption is not always valid, \( C_d \) is often assumed to be a constant in natural ventilation design (Chartered Institution of Building Services Engineers, 2015; Etheridge, 2011).

The brief literature review above highlights that whether \( C_d \) can be assumed constant is likely to be case-dependent. This study aims to evaluate whether the constant \( C_d \) assumption is valid for the buoyancy-driven ventilation (i.e., with low outdoor wind speeds) process in Stanford’s Y2E2 Building, and whether CFD can be used to characterize its value. To achieve this, a large eddy simulation (LES) is conducted to obtain the unsteady indoor temperature field. The LES results are then validated using high spatial-temporal resolution temperature data collected in a full-scale measurement campaign in the Y2E2 Building. Subsequently, the flow rate through each window is obtained from the validated LES results to determine the corresponding \( C_d \) values.
The Y2E2 Building and Experimental Setup

The Jerry Yang and Akiko Yamazaki Environment and Energy (Y2E2) Building was built in 2008 on the Science and Engineering Quad in Stanford University, USA. It is a certified LEED platinum building with high-performance systems to minimize energy consumption, to values up to 42% less than a traditional building of similar size (Stanford University Office of Sustainability, 2015). One of the building’s energy saving design features is that it uses natural ventilation for passive cooling of the common spaces and hallways. This is achieved through 4 atria that connect the common spaces on floors 1 to 3 and have louvers below the roofs. When the natural ventilation system is turned on, cool air enters through windows at floor 1 to floor 3 and exits through the louvers. A detailed description of the natural ventilation system is outlined in Lamberti and Gorlé (2018). In this analysis, we focus on Atrium D only, shown in Figure 1, as the symmetry of the layout allows modeling this atrium individually.

A simplified model of the building is shown in Figure 2, where the indoor area is separated into five sections, S1-S5. A network of thermistors is distributed to cover S1-S5 on floor 1, floor 2 and floor 3, respectively, to measure the indoor air temperatures. These sensor locations are chosen based on a preliminary CFD study to represent the volume-averaged air temperature in each floor-section (Chen and Gorlé, 2019). In addition, 34 sensors are attached to ceilings, floors or sidewalls to measure their temperatures. All sensors are calibrated in an air chamber with an accuracy of ±0.3 °C. The sampling frequency is 1 Hz, and the averaging period is 30 s. The measurements started at 8 pm local time on 8th October 2020 and ended at 6 am on 9th October 2020. During this measurement period, the windows and louvers stayed open by overriding the building management system. Our analysis focuses on the first three hours of the measurement period, i.e., between 8 pm and 11 pm on 8th October 2020.

The averaged outdoor wind speed during this period is low at 0.5 m/s, so the ventilation is predominantly buoyancy driven.

Computational Model

Figure 2 shows the computational model of the Y2E2 Building and the top view of the second floor. There are three floors and an atrium with a slanted roof, where the louvers are located. On floor 1, there is one window (f1) indicated by the arrow. On floor 2, there are four windows (f2 left, f2 right, f2 top and f2 bottom). Floor 3 also has four windows (f3 left, f3 right, f3 top and f3 bottom). The windows at the back (“back windows”) have a left-right configuration while the windows at the side (“side windows”) have a top-bottom configuration. The areas, heights (from the ground of floor 1) and (width-to-length) aspect ratios of the windows and louvers are summarized in Table 1.
The building geometry is complex and highly asymmetric (e.g., the location of the floor 1 window is different from those of floor 2 and floor 3). This complexity makes the building an interesting case study for evaluating the performance of CFD. When the natural ventilation system is activated, the windows and louvers are opened, and cool outdoor air enters through the windows while warmer indoor air exits through the louvers at the top. For the purpose of this analysis, the indoor area is divided into five sections, S1 to S5, as indicated in Figure 2. Offices and stairways are closed during the experiments and are not included in the computational model. There is no physical division between sections S1-S5, but based on the building geometry, these different zones can be expected to have different cooling rates. For example, S1 and S4 which do not have windows can be expected to have slower temperature decreases, and studying the different zones instead of taking the volume-averaged temperature of the entire floor, will support exploring this spatial variability.

Table 1: The area, height (from the ground of floor 1), and (width-to-length) aspect ratio of the windows and louvers.

<table>
<thead>
<tr>
<th>Opening</th>
<th>Area (m²)</th>
<th>Height (m)</th>
<th>Aspect ratio (-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Window f1</td>
<td>1.11</td>
<td>3.33</td>
<td>2.5</td>
</tr>
<tr>
<td>Window back f2 left</td>
<td>0.60</td>
<td>7.95</td>
<td>4.7</td>
</tr>
<tr>
<td>Window back f2 right</td>
<td>0.60</td>
<td>7.95</td>
<td>4.7</td>
</tr>
<tr>
<td>Window side f2 top</td>
<td>0.32</td>
<td>7.95</td>
<td>4.7</td>
</tr>
<tr>
<td>Window side f2 bottom</td>
<td>0.35</td>
<td>6.17</td>
<td>2.4</td>
</tr>
<tr>
<td>Window back f3 left</td>
<td>0.60</td>
<td>12.85</td>
<td>4.7</td>
</tr>
<tr>
<td>Window back f3 right</td>
<td>0.60</td>
<td>12.85</td>
<td>4.7</td>
</tr>
<tr>
<td>Window side f3 top</td>
<td>0.32</td>
<td>12.85</td>
<td>2.4</td>
</tr>
<tr>
<td>Window side f3 bottom</td>
<td>0.35</td>
<td>10.93</td>
<td>2.4</td>
</tr>
<tr>
<td>Louvers</td>
<td>18.37</td>
<td>15.13</td>
<td>7.0</td>
</tr>
</tbody>
</table>

Figure 3 shows the mesh of the computational domain, which is built and meshed in the ANSYS Workbench package. The simulation domain consists of the target building and its surrounding. The mesh has 2.6 million grid points and is finest near the building, and gradually coarsened away from the building. The maximum grid expansion ratio is 1.2. To verify that the mesh is sufficiently fine, another simulation with refined mesh of 8.6 million grid points is run. The difference of volume-averaged temperature between the normal and fine mesh models is negligibly small at 0.06 °C. Therefore, the normal mesh model with 2.6 million grid points is verified to have achieved mesh-independent results, and we use it for subsequent analysis.

The boundary conditions (BC) are as follows. The symmetry plane has a symmetric BC. The side and top (not shown) faces have a zero gradient BC for velocity, constant pressure BC of 1 atm and are prescribed the temperature obtained from outdoor temperature measurements (see Figure 6 for the outdoor temperature trend). The ground has a no-slip BC for the velocity, zero gradient BC for pressure (adjusted for gravity) and zero gradient BC for temperature. The building walls, floors and ceilings have similar velocity and pressure BC as the ground, but the temperatures are prescribed using measurement data. Wall functions are employed to reduce the total number of grid points. The windows are open at an inclined angle of 42°, the resulting resistance is modeled with a porous pressure jump $\Delta p = 0.5C\rho U^2$, where $C = 1.65$ from previous studies of the same windows (Lamberti and Gorlé, 2018; Hult et al., 2011), $\rho$ is the air density and $U$ is the air velocity orthogonal to the porous surface. To simulate the internal load in the building, a constant volumetric heat source is added. Based on the lighting, equipment and occupant loads during the measurement period, the total internal load is estimated to be 0.37 W/m² (Hult et al., 2011). The initial conditions are prescribed based on the measured temperatures at the start of the experiment, i.e., at 8 pm on 8th October 2020. The initial outdoor temperature is 16.9 °C. The initial indoor air temperatures are 20.5 °C, 21.0 °C and 21.0 °C for floor 1, floor 2 and floor 3, respectively.

The open-source, finite volume solver Open Field Operation and Manipulation (OpenFOAM) version 7.0 is used for the CFD simulations. The simulations are run in the Texas Advanced Computing Center (TACC) servers at The University of Texas at Austin. The dynamic-k-equation subgrid model is used. Second order differencing schemes are used for both time and space discretization. The time step size can vary such that the maximum Courant number $< 1.0$ to maintain stability. This results in a time step size of about 0.1 s, with an average Courant number of 0.1 (the maximum is 1.0). The Boussinesq approximation is employed to model the density difference since the temperature difference is small ($< 5$ °C in our case). The convergence criteria for all variables are set to be smaller than 10⁻⁶. Post-processing is done with the open-source software ParaView version 5.7.0.

![Figure 3. The mesh of the computational domain.](image-url)
Experimental Results and Model Validation

Figure 4 shows the comparison between simulated and measured temperatures for the different floors and sections. Note that there is no measurement at Floor3-Section5 (F3S5) due to a faulty sensor, but the simulated temperature is included for completeness. The measurements show that temperatures drop relatively fast in the first 15 minutes in most sections. This is expected because at time = 0, the temperature difference between indoor and outdoor is the largest, and therefore the driving force for buoyancy-driven ventilation is the strongest. The simulation correctly predicts the measured temperature trends, showing a fast decrease in the first 15 minutes. After the first 15 minutes, both simulated and measured temperatures continue to drop at a lower rate. Another key observation from the measurement is that the sections with windows (F1S5, F2S3, F2S5, F3S3) have lower temperatures, as expected, due to cold air entering through these windows. This is evidence from the instantaneous temperature contours in Figure 5(a) obtained from LES at 60, 120 and 180 minutes. Cold air enters through the windows (see Figure 2 for the locations of windows) and cools S3 and S5, while S1, which is furthest away from the windows, has the highest temperature. This non-uniform indoor temperature distribution is maintained even after 180 minutes of natural ventilation, where S3 and S5 remain as the coldest sections while S1 is still the warmest section, which justifies the need to deploy a network of sensors to measure the temperatures at difference sections. Figure 5(a) also shows how the overall indoor air temperature decreases with time, consistent with the temperature

Figure 4: Comparison between measurement and simulated (CFD) temperatures at each floor-section. For example, F1S1 means Floor1-Section1, F2S3 means Floor2-Section2, and so on. There is no measurement in F3S5 but the simulated temperature is included for completeness.

Figure 5: (a) Instantaneous temperature contours on the second floor showing cold air enters through the windows. (b) Instantaneous horizontal velocity magnitude on the second floor showing jets of cold air enters through the windows and areas away from the windows have low horizontal velocity.
trends observed in Figure 4. Figure 5(b) shows the contours of horizontal velocity magnitude obtained from LES at 60, 120 and 180 minutes. The velocity is the highest near the windows, while most other areas have relatively low velocity.

Overall, Figure 4 shows that the simulated temperature agrees well with the measurements, with discrepancies not exceeding 1 °C. The good agreement in the temperature trend indicates that the flow rates are predicted with acceptable accuracy, thereby supporting the use of the results for further analysis of the discharge coefficient.

Calculation of $C_d$ and Discussion

The discharge coefficient, $C_d$, is calculated following Equation (1):

$$C_d = \frac{Q}{\sqrt{2gH(T_{in} - T_{out})/T_{out}}}$$  \hspace{1cm} (1)

where $Q$ is flow rate across a window, $A$ is opening area of a window, $g$ is gravitational constant, $H$ is height difference between inflow and outflow (i.e., between a window and the louver in our case, see Table 1 for the height of windows and louvers), $T_{in}$ is volume-averaged indoor temperature of the entire building, and $T_{out}$ is outdoor temperature. Among these parameters, $A$, $g$ and $H$ are constants and known; $T_{out}$ is obtained from measurement (which is prescribed as the boundary condition of the outermost faces in the CFD domain so simulated $T_{out}$ is the same as measured $T_{out}$); $T_{in}$ and $Q$ are obtained from simulations.

Figure 6 shows the time series of $T_{in}$ and $T_{out}$. $T_{out}$ decreases almost linearly with some fluctuations, while $T_{in}$ decreases rapidly during the first 15 minutes and the rate of decrease is lower after that. Figure 7 shows the five-minute averaged flow rate $Q$ for each window. $Q$ is the highest during the first 15 minutes due to the large indoor-outdoor temperature difference. After the first 15 minutes, $Q$ remains relatively constant. The window at floor 1 has the highest $Q$ at about 0.80 m$^3$/s, while the windows at floor 2 have $Q$ between 0.18 and 0.40 m$^3$/s and the windows at floor 3 have $Q$ between 0.08 and 0.20 m$^3$/s. As expected, higher floors have lower $Q$ because the height difference $H$ between the inflow (window) and the outflow (louver) is smaller. This trend is also observed looking at the side windows (with top-bottom configuration) on each floor, where the bottom window has a higher $Q$ than the top window. For example, on floor 2, the bottom side window (plotted as red open triangles in Figure 7) has $Q$ of about 0.22 m$^3$/s, while the top side window (plotted as red open diamonds) has lower $Q$ of about 0.18 m$^3$/s. In contrast, the back windows (with left-right configuration) on each floor have the same $Q$ because they have the same $H$.

To non-dimensionalize $Q$, the Reynolds number, $Re$, is defined in Equation 2, where $D_h$ is hydraulic diameter of the window and $v$ is kinematic viscosity of air.

$$Re = \frac{D_h Q}{v}$$  \hspace{1cm} (2)

Figure 8 plots $C_d$ against $Re$. The range of $Re$ is between 6000 and 47,000. The window on floor 1 has the largest $Re$ between 36,000 and 47,000, with $C_d$ nearly constant at 0.51. The windows on floor 2 have a lower $Re$ between 15,000 and 24,000, but $C_d$ also stays constant at around 0.51. The windows on floor 3, with $Re$ between 6000 and 16,000, show a different trend. At $Re$ above 10,000, $C_d$ stays constant around 0.50, similar to those of floor 1 and floor 2 windows. At $Re$ below 10,000, there is a linear decrease with decreasing $Re$. The lowest $C_d$ is 0.32. Overall, there is a critical $Re$ (equal to 10,000 in our case) where $C_d$ becomes constant. Below this critical $Re$, the assumption that $C_d$ is a constant is no longer valid.
In the design or evaluation of natural ventilation systems, $Q$ is often estimated with analytical or empirical equations such as Equation 1 by assuming a constant value for $C_d$. For example, $C_d = 0.6 \pm 0.1$ is suggested if $Re > 4000$ (Holford and Hunt, 2001; Acred and Hunt, 2014). However, Figure 8 shows that $Re$ should be above 10,000 in our studied building for the constant $C_d$ assumption to be valid. If we assume $C_d = 0.6$ for all windows, the flow rate across the floor 3 back windows can be overestimated by a factor of 2. In addition, the suggested critical $Re$ of 4000 (Holford and Hunt, 2001; Acred and Hunt, 2014) is based on a simple rectangular geometry. Our study of a real operational building with complex geometry suggests a higher critical $Re$ of 10,000, indicating that the critical $Re$ could be geometry-dependent. As such, we emphasize that the critical of $Re = 10,000$ should not be considered a universal value, and different building geometries or flow conditions may have a higher or lower critical $Re$. It is suggested that CFD provides an invaluable tool for assessing appropriate values for $C_d$ for a specific building configuration. These values can then be used in envelope models implemented in building thermal models or energy models to support efficient investigation of the building’s thermal response and energy savings under a range of operating and weather conditions.

Lastly, it is important to note that the current analysis considered a purely buoyancy-driven natural ventilation flow. With assisting wind (i.e., the wind strengthens the buoyancy-driven ventilation), the flow rates and the resulting $Re$ would increase, potentially reducing the risk of being in the $Re$-dependent flow regime. In contrast, with opposing wind (i.e., the wind weakens the buoyancy-driven ventilation), the flow rates and resulting $Re$ would decrease, which could further decrease $C_d$ and consequently, introduces a larger error in the estimation of $Q$. In wind-driven conditions, the outdoor wind direction could also result in installation effects that can further modify $C_d$; the use of CFD for these mixed buoyancy- and wind-driven flow conditions is the subject of ongoing research.

**Conclusion and Future Work**

Buoyancy-driven natural ventilation in the Y2E2 Building at Stanford University is analyzed in the present work. Temperature measurements are used to validate an LES model, where the agreement between natural models and predictions is within 1 °C. Using the validated LES results, the discharge coefficients, $C_d$, across the nine windows in the building have been calculated. $C_d$ maintains a constant value of 0.51 when the Reynolds number ($Re$) exceeds a critical value of 10,000. Below this critical $Re$, $C_d$ decreases almost linearly with $Re$ and can be as low as 0.32. In the design of natural ventilation systems, $C_d$ is often assumed to be a constant. Our result shows that this assumption becomes invalid for relatively low flow rates. The critical $Re$ for the constant $C_d$ assumption to be valid is likely to be case-dependent. Therefore, we suggest using CFD to assess appropriate values for $C_d$ for a specific building configuration.

An ongoing work is to employ the results in this study in an integral model to predict the indoor temperature. Integral models are simplified models that solve for the average indoor air temperature, often assuming a constant $C_d$. Though not capable of providing the spatial temperature distribution, e.g., in Figure 5(a), integral models are fast (can be solved in seconds, compared to hours or even days required by CFD simulations) and are therefore useful in the early design phase. We aim to improve the accuracy of the integral model by employing $C_d$ as a function of $Re$.

As mentioned in the Discussion Section, outdoor wind, either assisting or opposing wind, can affect $C_d$. The results reported in this paper focus only on buoyancy-driven ventilation. The measurement campaign included a few days with high outdoor wind speeds, and the results and analysis will be reported in a future work.

**Acknowledgement**

The authors acknowledge the Stanford University Land, Buildings & Real Estate, Stanford University Science & Engineering Quad Building Management and Stanford University Fire Marshal’s Office for their assistance during the measurement campaign. This work used the Extreme Science and Engineering Discovery Environment (XSEDE, https://www.xsede.org) HPC resources provided by the Texas Advanced Computing Center (TACC, http://www.tacc.utexas.edu) at The University of Texas at Austin. XSEDE is supported by National Science Foundation grant number ACI-1548562.

**References**


Chartered Institution of Building Services Engineers (2015). *Guide A: Environmental design.*


