The natural ventilation of buildings

Geometry and ventilation: Evaluation of the leeward sawtooth roof potential

JORGE ISAAC PEREY MONTERO
Co-supervisor: Ber Blomken
Coelho Lete
Supervisor: Prof(a). Dr(a). Brenda Chaves

Civil Construction Engineering Department
Concentration Area:

degree in Science.
of Technology (IT/IE) to obtain the PhD
Paulo (USP) and the ETH Zürich University
agreement between the University of Sao
School as part of a double-degree
Thesis presented at the Polytechnic

the natural ventilation of buildings

Geometry and ventilation: Evaluation of the leeward sawtooth roof potential in

JORGE ISAAC PEREZ MONTERO
Tese (Doutorado) - Escola Politécnica da Universidade de São Paulo.

175 p.


Geometria e Ventilação: Evaluación de la forma de un edificio.

Peren, Jorge Isaac

Catalogação na Publicação

Biblioteca da Escola Politécnica

Universidade de São Paulo
Table of Contents
Acknowledgements
I would like to address a special thanks to my colleagues for all the support and encouragement throughout my graduate studies. I would also like to express my gratitude to the following institutions and individuals for their contributions to my research and personal growth:

- University of São Paulo (USP), Brazil
- University of California, Los Angeles (UCLA)
- The University of Texas at Austin
- The University of Michigan
- The University of Illinois at Urbana-Champaign
- The University of California, Los Angeles

To all my friends who have helped me to achieve this important goal. In Brazil, my sincere thanks to Céu. For this achievement, I would like to express my gratitude to my advisor and my colleagues for their support and encouragement throughout my graduate studies.

In the United States, I would like to express my gratitude to the following institutions and individuals for their contributions to my research and personal growth:

- Massachusetts Institute of Technology (MIT)
- Harvard University
- Stanford University
- The University of Chicago
- The University of Pennsylvania

To all my friends who have helped me to achieve this important goal. In Brazil, my sincere thanks to Céu. For this achievement, I would like to express my gratitude to my advisor and my colleagues for their support and encouragement throughout my graduate studies.

In the United States, I would like to express my gratitude to the following institutions and individuals for their contributions to my research and personal growth:

- Massachusetts Institute of Technology (MIT)
- Harvard University
- Stanford University
- The University of Chicago
- The University of Pennsylvania

To all my friends who have helped me to achieve this important goal. In Brazil, my sincere thanks to Céu. For this achievement, I would like to express my gratitude to my advisor and my colleagues for their support and encouragement throughout my graduate studies.
Enrique, my Godson.

whole PhD process. To my sister (family) and my brother-in-law (Irmac) and my nephews (Carlos and

Challenges: the first one, when I felt frustrated to continue my studies in Brazil and the second one, my

momentum. To my best friend and mentor, my dad (Jose) and my mother (Elsa) that supported me the two

Finally, I would also like to show my enormous gratitude to my family that stood by me in all the difficult

members.

Edwin, Pricile, Jaime Legun, Thomas Schmidt, Efra and Ricardo Agullier and all the AEPF family and friend.

Trudioso, Armando, Melissa and Michele Williamson, Crista and Karina Brando and, Eduardo Ruffo and
Geometría y Ventilación: Evaluación del Potencial de Sfeas a Solucionar en Ventilación Natural

Resumen
Chapter 2: Progress the validation study and shows that the SST K-6 turbulence model performs the most accurate results, followed by the RNG K-6 turbulence model. The standard K-6 model, the RANS model, and the K-4 model show better deviations from the measured results, showing that the SST K-6 turbulence model is the most accurate.
Chapter 6 provides a discussion based on the implications of the presented research and points out important notes in the formulation performance. The internal geometry configuration near the outer periphery is shown in Figure 6.4. Full 12-27% 12-27% internal geometry for the outer periphery decreases the building's performance. The outer geometry, full 12-27% 12-27% external geometry, full 12-27% 12-27% internal geometry, and full 12-27% 12-27% external geometry all slightly decrease the number of internal geometry.

In Chapter 7 analysis of single-span and double-span roof systems is presented. It is shown that by modifying the roof, the volume flow rate in a single-span building can be increased. The roof's impact on the performance is explored. It is shown that. in addition to other well-known geometries, the slight increase in volume flow rate is 27% for a roof with a volume flow rate of 27%.

In Chapter 3. Five different roof geometries are studied; one straight and four curved roofs. The curved roofs are modified by (a) 11% and (b) 21% respectively. It is observed that the volume flow rate is 22% higher than a flat roof and an additional increase from 22% to 27% when the water flow rate is reduced near the roof. The volume flow rate is less important as an increase in volume flow rate by around 4% increases or decreases the indoor temperature is increased by the roof's configuration. A small increase in the roof's configuration position is less important. It can just increase the volume flow rate by around 4% and increase or decrease the indoor temperature.
the same root geometry type, despite the fact that a lower underpassing is present at the outlet opening.

a single-span with 12 m depth reaches a higher volume flow rate than a single-span with 6 m depth with
Table 2: Parameters of the lean and sawoom roof sensitivity analysis with inclination of the reference case in bold.

List of tables
Relevance

Introduction
Influence Flow and the Geometry of the Windward Face (See Figure 1.2).

The "shading vortex" or "screening vortex" extends from the intersection of the wind and building. The shading vortex moves across the building's surface, influencing the building's geometry, and the building's surface becomes a complex interaction between the wind and the building. Key features include:

- The wind flow pattern above and behind the building, as described by Patena et al. [14], and
- The wind flow pattern around buildings, and therefore also the pressure distribution on the building surface, is very complex, with significant differences under very low wind velocities in the absence of wind [12].

Natural ventilation is driven by pressure differences produced by wind and buoyancy. Buoyancy forces act on the roof and the facade, depending on the weather conditions and the type of building. The thermal stratification with internal sources [13,12] and external heat exchange from the building's surface and the environment influence the convective heat transfer from the occupants. (1) Increase thermal comfort by:

- Increasing indoor air quality,
- Increasing indoor temperature,
- Increasing ventilation.
An illustration of how natural ventilation can help to enhance the exchange of air in a building. The figure shows a building with two different ventilation strategies: (1) single-sided ventilation and (2) cross-ventilation.

**Figure 1:** Natural ventilation flow types (a) Single-sided ventilation (b) Cross-ventilation

In single-sided ventilation, the air flows through the window at the lower end of the building. In cross-ventilation, the air flows through both the windows at opposite ends of the building. This can be used to increase the air exchange rate, which is beneficial for improving indoor air quality.

**Figure 1:** Schematic representation of the wind flow around an isolated, low-sloped roof building

Flow separation and reattachment zones are shown, along with the horseshoe vortex. The approach flow is illustrated, and the zones of flow separation and reattachment are highlighted.
A proposed consists of two consecutive simulations: (1) a simulation of the wind flow around a selected building and (2) a simulation of the indoor airflow in the indoor environment. The interaction of these two simulations can provide a more realistic prediction of the indoor airflow and thermal conditions.

Within the CFD modeling approach, one can make a distinction between complex and simplified CFD models.

Experimental results from full-scale indoor air quality measurements reveal the importance of considering the indoor air circulation patterns. The literature suggests that CFD models can provide valuable insights into the complex airflow patterns within buildings, especially in cases where traditional engineering methods might not be sufficient. Therefore, the integration of CFD models into the design process can lead to more efficient and healthier building designs.

Outdoor, experimental measurements of airflow around buildings can be performed using various techniques, such as smoke visualization or smoke welding. These methods allow for the visualization of the airflow patterns and can be used to validate the CFD models.

[Figure 1(a)] shows a simplified representation of a building with a smoke plume. [Figure 1(b)] and [Figure 1(c)] depict the airflow patterns around the building, with smoke being used to visualize the flow.

The results from the experimental measurements can be compared with the CFD simulations to assess the accuracy of the models. This comparison helps in validating the models and improving their accuracy for future applications.
A curved multi-span roof [35] and concaved (or singular convex) opening of window(3) and leaned roof would work efficiently on the natural ventilation and indoor climate of a building(3) with good geometry. Moreover, the effect of roof openings (window(3) and leaned roof geometry). To the best knowledge of the author, there are only a few studies that studied natural ventilation on wind-inclined roofs. However, these studies did not focus on the ventilation flow that enters the building through the opening of the roof. A study on wind-inclined roofs has not been conducted in detail.

Smith et al. [7] also showed that the possible advantages of wind-inclined roofs in naturally ventilated spaces are unexplored. With the possible advantages of wind-inclined roofs, the ventilation should be managed without discharge, which can increase the cooling effect and reduce energy consumption. The results of the study on wind-inclined roofs are located in the position of the windward and leeward sides of buildings in close contact with the natural wind. In the case of a building with a windward side, the ventilation flows in the direction of the wind, and on the other hand, the buildings in close contact with the natural wind have an open connection. Some examples of the use of wind-inclined roofs have been included in the current study. The results of the study on wind-inclined roofs are presented in Table 1.3.1 and Figure 2.7 show some examples of wind-inclined roofs. The study focuses on the idea of a wind-inclined roof in the current study.
1.4. Building and Roof Geometries

1.4.1. Methodology

With a convex roof geometry, the depth of the roof to wall ratio of the building's roof helps to create a distinctive look. Figure 1.5 shows a diagram of roof geometries, including (a) single-span, (b) double-span, and (c) double-span with a cross-section of two skewed sawtooth roof geometries. The figure illustrates how the geometries vary in depth and shape, affecting the overall appearance and functionality of the building.

When evaluating the building's roof geometry, several factors are considered, such as the wind exposure, the solar gain, and the thermal performance. These factors can impact the building's energy efficiency and overall comfort.

The diagram shows the relationship between the roof geometry and the building's shape, highlighting how the roof influences the building's overall performance. The roof's design can affect the building's aesthetics, as well as its energy efficiency and environmental impact.
The first phase consisted of an extensive CFD validation study using the PIV measurements of Khara et al. [12].

The study's main objective was to understand the flow regime and the indoor velocity fields. The analysis focused on the flow patterns, including the movement of the air around the building and the effect of the roof geometry. The assessment method developed for this study, 3D wind RANS Computational Fluid Dynamics (CFD) simulations, are performed for a wide range of roof geometries, considering the roof geometry (Figure 1c), and the effect of the roof geometry on the flow regime. The results of the simulations were validated against the experimental data obtained from the PIV measurements.

Figure 1c: Schematic representation of the leeward smooth roof geometries evaluated.

Assessment method

The assessment method employed in this study is a measure of the roof inclination angle. For all roof parameters under study, their effect on the volume flow rate and the indoor velocity fields is investigated. The accuracy of the analysis is a measure of the roof inclination angle. For all roof parameters under study, their effect on the volume flow rate and the indoor velocity fields is investigated. The accuracy of the analysis is a measure of the roof inclination angle. For all roof parameters under study, their effect on the volume flow rate and the indoor velocity fields is investigated.
Chapter 3 is based on the Journal paper:

Influence of various models and the influence of approaches on improved thermal energy performance. 

This chapter consists of seven (7) chapters. Chapter 2 will present an overview of cross-ventilation of a generic building with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.


The main topic of the Journal paper is the influence of roof angle and opening location on thermal performance of generic buildings with asymmetric opening positions. Important of roof angle and opening location.
Chapter 5 is based on the Journal paper:

The computational models are shown equal to those in the validation study presented in Chapter 2. The computational grid is based on a grid-resolution where the grid-size and the domain-size are the lowest and the highest performance were selected and compared for both the studies. Figure 5.7 shows the single-span case compared to the double-span case. The findings presented in this study show that the single-span case achieves a normal wind direction.

Chapter 6 focuses on cross ventilation and its enhancement in low-rise buildings. The focus is on single-floor vs. single-story cross ventilation with dimensions 3.0 x 12.0 x 2.7 m. The study shows a significant improvement in the cross ventilation with a single-floor building vs. a single-story building.

The chapter evaluates the impact of cross ventilation on the cross-ventilation of a generic roof:

Building and Environment in Press 2015

Chapter 4 is based on the Journal paper:

Chlid mecbivas. the flow pattern and pressure distribution around the building is provided for a detailed analysis of the

inclusion angle of 270 (°). The angle is measured by drawing a straight line from the windward edge of the

roof. If the Hoop is the BCC, Brocken's impact of angles on cross-ventilation of a generic roof:

Figure 6.1 shows a significant improvement in the cross ventilation with a single-floor building vs. a single-story building.

The chapter evaluates the impact of cross ventilation on the cross-ventilation of a generic roof:

Building and Environment in Press 2015
References


5. Patagonia Agroclimatic Climate Change: National Park of the Andes, Argentina National Park of the Andes, Argentina


7. La Palma, T. J. A. Response of the Climate to Global Warming. 1997


Chapter 2

CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: impact of roof angle and opening location.

Published as:

Abstract

Root angle and opening location
Building with asymmetric opening positions: Impact of
2 CFD analysis of cross-ventilation of a generic isolated

Validation: CFD simulations: Building geometry: Numerical ventilation: Model
Finally, Computational Fluid Dynamics (CFD): Building geometry: Numerical validation: Model
then 49% and a small influence on the average velocity in the occupied zone (less than 2%)
In these positions, the differences in the average velocity in the occupied zone are only around
The results show that the shear stress numbers (SSN) and the Reynolds number

Validation: The results show that the shear stress numbers and Reynolds number with previously published wind tunnel measurements without particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image

Validation: The results show that the shear stress numbers and Reynolds number with previously published wind tunnel measurements without particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image

Validation: The results show that the shear stress numbers and Reynolds number with previously published wind tunnel measurements without particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image
and on validation with previously published wind tunnel measurements with particle image

Introduction
22
Wind tunnel experiment

Without the use of theoretical tools, the different wind tunnel positions and the different ventilation position of the other
structures had a considerable effect. The pressure measurements were obtained in Section 2.2 where the ventilation position and the
pressure differences were measured in Section 2.6. The correlations were obtained in Section 2.4 where the
pressure differences were obtained in Section 2.3. The ventilation position and the pressure differences were obtained in Section 2.2. The
pressure differences were obtained in Section 2.3. The ventilation position and the pressure differences were obtained in Section 2.2. The
pressure differences were obtained in Section 2.3. The ventilation position and the pressure differences were obtained in Section 2.2. The
pressure differences were obtained in Section 2.3.
The computational domain and grid are considered as reduced scale (1:200) to exactly resemble the

2.3.1 Computational domain and grid

The computational domain and parameters are described in this section. These

2.3 CFD Simulations: Computational settings and parameters

The experimental setup and parameters used for the simulations are described in the following section. The grid size and dimensions are also reported. The computational grid is

Figure 2.2: (a) View of the reduced-scale building model as studied by Kaya and El [7] and

Figure 2.3: The windward face (a) and side views of the reduced-scale building model with

Figure 2.4: The experimental setup and measurements were conducted in the wind tunnel. The

Chapter 2
In this equation, \( I \) is the measured mean wind velocity intensity and \( \psi \) is a parameter ranging from 0 to 1.5 (1.74 in this case), and the result in the parameter \( \psi \) is constant and presented in Section 2.3.1. The sensitivity of the results to the parameter \( \psi \) is assessed and presented in Section 2.3.2.

\[
\varepsilon(z) \propto \left( \frac{\partial \theta_0}{\partial x} \right)_s
\]

where \( z \) is the same as in the experiment and \( x \) is the ABL friction velocity \( u^* \) at 0.95 m/s.

\[
\left( \frac{0.2}{0.2 + z} \right) \frac{\partial \theta_0}{\partial x} = \frac{(z)}{l}
\]

Boundary conditions

Figure 2.2: (a) Perspective view of the building in its computational domain at model scale. (b) View of the computational grid (building and ground). (c) Perspective view of grids for the grid-sensitivity analysis. (d) Case grid with 421,088 cells (reference case); (e) Fine grid with 770,400 cells; (f) Base grid with 770,400 cells (reference case); (g) Base grid with 770,400 cells; (h) Case grid with 421,088 cells; (i) Basic grid with 770,400 cells (reference case). The results of the grid-sensitivity analysis are presented in Section 2.3.1. The sensitivity analysis is performed based on three grids (Figure 2.2c-e): coarse, basic and fine grids.
Figure 2.3: Horizontal homogeneous model: Profiles of the mean wind speed (U) and the turbulence kinetic energy (K).

For the mean wind speed profile:

For the turbulence kinetic energy profile:

At the ground and building surfaces, the standard wall functions by Lanner and Saling [84] are used in (4) and (5):
observed.

de to reflections and shading effects [12]. Despite these differences in overall good agreement is
the mean velocity around the opening, where the PIV measurements cannot provide accurate predictions
indicating a good agreement. As pointed out by Ramponi and Blocken [30], the simulations overestimate
where a horizontal line going through the middle of the window opening and making a directional line
the building's front edge and for comparison the measured and computed streamwise wind speed using
the building's front edge and for comparison the measured and computed streamwise wind speed using
the CFD simulations correctly predict the most significant flow features such as the swirling vortex present at the building's
imprints how strongly the wind is blowing in the vertical plane. Flow in the vertical plane by PIV measurement and CFD
and in Figure 4.4, respectively. The CFD simulations correctly predict the most
simulations is shown in Figure 4.2 and 4.3, respectively. The CFD simulations correctly predict the most

The results from the CFD simulations are compared with the PIV wind-tunnel experiments by Rau and

2.4 CFD Simulations: Validation

Interruptions (1'000-001'000) after the simulation reached a satisfactorily stationary solution,
the results are monitored over 1'000 iterations are evaluated and published As evaluated and published As evaluated and published As evaluated and published As evaluated and published
As evaluated and published As evaluated and published
and recorded a minimum of 1'000 for x and y momentum. 1'000. For z momentum and 1'000 for k and epsilon, as observed by Ramponi and Blocken [29,30]. The

The results are monitored over 1'000 iterations and recorded a minimum of 1'000 for x and y momentum. 1'000. For z momentum and 1'000 for k and epsilon, as observed by Ramponi and Blocken [29,30]. The

The commercial CFD code ANSYS Fluent 12 [87] is used to perform the CFD simulations. The 3D
Solver settings

Solver settings

For the turbulence kinetic energy calculation (reference case):
Table 2.1: Overview of computational parameters for sensitivity analysis with indication of the reference case.

<table>
<thead>
<tr>
<th>Reference case</th>
<th>SST (°C)</th>
<th>T (°C)</th>
<th>U (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST, T, U</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

A sensitivity analysis was performed on the following parameters: (a) u, (b) v, (c) T, and (d) SST. The reference case is indicated by the symbols.
Impact of computational field resolution

![Graphs showing impact of computational field resolution.](image-url)
The impact of the elevated kinetic energy is shown in the diagram. The horizontal component (a) of the wind shows a diagonal line. The vertical component (b) shows a slight deviation from a straight line. The deviations are marked with arrows. The model used for the K-E model is the RSM model (a), with slight deviations from the actual data. The deviations are explained by the different direction of the wind. The model used for the RSM model is the K-E model (c), with slight deviations from the actual data. The deviations are marked with arrows. The model used for the K-E model is the RSM model (d), with slight deviations from the actual data. The deviations are marked with arrows. The model used for the K-E model is the RSM model (e), with slight deviations from the actual data. The deviations are marked with arrows. The model used for the K-E model is the RSM model (f), with slight deviations from the actual data. The deviations are marked with arrows. The model used for the K-E model is the RSM model (g), with slight deviations from the actual data. The deviations are marked with arrows.
Go through the middle of the windward opening evaluated based on the volume flow rate and the mean wind speed ratio above a horizontal cutline.

Example 2.7 shows the vertical cross-sections and computational grids of the reference case and the five additional cases with different roof inclination angles. The reference case is selected as the same height and angle of opening for the five cases to show a different roof inclination angle.

The computational parameters and settings used in this section are the same as in the previous section. The results are plotted in Figure 2.7. The performance of the building with an inclination angle of 0°, 5°, and 9°. To allow a normal wind incidence angle (θ = 0°, 9°, 27°, 72°, and 90°) on the ventilation flow, the roof inclination angle (θ(roof)) on the ventilation flow, the roof inclination angle (θ(roof)) on the ventilation flow, the roof inclination angle (θ(roof)) on the ventilation flow, the roof inclination angle (θ(roof)) on the ventilation flow.

2.6 CFD Simulations: Sensitivity analysis for geometrical parameters

The sensitivity analysis of roof inclination angle and the vertical location of the outflow opening is performed in this section for the CFD simulations. Sensitivity analysis is used to evaluate the performance of the building with different roof inclination angles and vertical locations. The results are plotted in Figure 2.7. The horizontal cutline profile parameter: (a) Above a horizontal cutline. (b) Same height as the vertical cutline. (c) Same height as the horizontal cutline.

Figure 2.7: Impact of roof inclination angle and vertical location on the performance of the building with different roof inclination angles (θ = 0°, 5°, 9°, 27°, 72°, and 90°). The best value for CFD simulations of a building with symmetrical openings (normal wind incidence angle) appears to be 0°.
Figure 2.8: Vertical cross-section and computational grid for building geometry with different roof inclinations. The windward face (frontal wall) and leeward facade (in line with屋顶). All the cases have the same frontal volume as the reference case and as a consequence have different heights of the windward (perpendicular) and leeward facade (skew).

### Table 2.2: Parameters of the low-speed wind tunnel roof sensitivity analysis with indication of the reference case.

<table>
<thead>
<tr>
<th>Building case name</th>
<th>Outlet inclination angle</th>
<th>Outlet opening position</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case A</td>
<td>RIA 0° (reference)</td>
<td>0°</td>
</tr>
<tr>
<td>RIA 18°</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>RIA 27°</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>RIA 36°</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>RIA 45°</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2.9 shows that the volume flow rate increases when increasing the roof inclination angle with more than 18° compared to the reference case. However, the volume flow rate for RIA, 0° is only 96% of that of RIA 0°, while for RIA 45°, it is 22%.

Figure 2.10a shows that the streamwise wind speed ratio increases along almost the entire horizontal line when the roof inclination angle is increased, except for case RIA 0°. Figure 2.10b shows that the indoor mean streamwise wind speed ratio along the horizontal line shows very large differences between the
Table 2.1: Non-dimensional area-averaged velocity magnitude \((\mathbf{u}^2/\mathbf{V})^{1/2}\) in the occupied zone in the vertical center plane for the different roof inclination angles

<table>
<thead>
<tr>
<th>Roof Inclination Angle</th>
<th>(\mathbf{V}_\text{up}^{1/2})</th>
<th>(\mathbf{V}_\text{low}^{1/2})</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0°</td>
<td>0.23</td>
<td>0.37</td>
</tr>
<tr>
<td>8.0°</td>
<td>0.21</td>
<td>0.34</td>
</tr>
<tr>
<td>16.0°</td>
<td>0.26</td>
<td>0.38</td>
</tr>
<tr>
<td>24.0°</td>
<td>0.29</td>
<td>0.41</td>
</tr>
<tr>
<td>32.0°</td>
<td>0.32</td>
<td>0.44</td>
</tr>
</tbody>
</table>

Figure 2.10: Influence of roof inclination angle \(\theta\) on the horizontal component \(\mathbf{u}_x\) percentage and mean velocity \((\mathbf{u}^2/\mathbf{V})^{1/2}\).

Windward opening for the vertical center plane. The height of the occupied zone is equal to the height of the vertical center plane. The non-dimensional area-averaged velocity magnitude \((\mathbf{u}^2/\mathbf{V})^{1/2}\) in the occupied zone has been calculated for the windward opening. The roof of the occupied zone has been raised by 18°. Despite the height of the roof being lower than the reference case, the windward side of the room shows similar results to the reference case. The roof inclination angle, especially between 0° and 8°, shows a lower percentage of the horizontal component \(\mathbf{u}_x\).
In order to further analyze the effect of the roof inclination angle on the flow pattern around buildings, it is important to understand the interaction between the building geometry and the surrounding environment. This interaction can significantly affect the pressure distribution and flow patterns. The pressure coefficient, which quantifies the pressure difference induced by the building geometry, is given by:

\[
\frac{\frac{\partial p}{\partial y} \left|_{y=0} \right.}{\frac{\partial p}{\partial y} \left|_{y=d} \right.} = \frac{\delta C}{d - d}
\]

where \(C\) is the pressure coefficient, \(\delta\) is the roof inclination angle, and \(d\) is the building height. The pressure coefficient is calculated based on the pressure difference between the upper and lower surfaces of the building. This difference is influenced by the roof inclination angle and the wind direction.
The outlet opening is the same as the reference case, and the difference between the root and the lip of the outlet are compared to the previously analyzed cases. In order to evaluate the impact of the vertical position of the outlet opening, two additional cases are considered.

**Figure 2.12**: Contours of non-dimensional flow velocity magnitude (\(\dot{V}/\dot{V}_m\)) in vertical center plane.

**Figure 2.11**: Contours of pressure coefficient \(C_p\) in vertical center plane (PFRIP).
The ventral position of the outlet opening is not the case in the current study. The influence of the outlet opening on the volume flow rate is lower. Moreover, they found a position of the outlet opening at the lower level of the vessel center plane, which is not compatible with the experimental results. In the experimental results, the outlet opening was positioned at a high level between RIA = 45° and RIA = 60°. This finding is in line with the observation of Liao and Nogadbekov. RIA = 45°-60°, there are no significant differences between RIA = 45° and RIA = 60°, and also no significant differences for RIA = 45°-90° and RIA = 60°-90°. Figure 2 shows the pressure coefficient (Cp) and non-dimensional velocity magnitude (V/|U|) in the ventral center plane for RIA = 27°. The center line for the same variables for RIA = 45°-90° and RIA = 60°-90° is shown in Figure 2. The pressure coefficient (Cp) and non-dimensional velocity magnitude (V/|U|) in the ventral center plane for RIA = 27° are shown in Figure 2. The center line for the same variables for RIA = 45°-90° and RIA = 60°-90° is shown in Figure 2.

![Figure 2](image)

Table 2.4: Non-dimensional area-averaged velocity magnitude (V/|U|) in the occupied zone in the ventral center plane for the different outlet opening positions. Table 2.4 shows the non-dimensional area-averaged velocity magnitude (V/|U|) in the occupied zone in the ventral center plane for the different outlet opening positions.

| RIA (°) | Cp | V/|U| |
|---------|----|----------|
| 45°     | 0.97 | 0.22 |
| 60°     | 0.92 | 0.20 |
| 75°     | 0.87 | 0.18 |
| 90°     | 0.82 | 0.16 |

Figure 2.4: Influence of the outlet opening position on the volume flow rate (V/|U|) and non-dimensional velocity magnitude (V/|U|) in the ventral center plane. Figure 2.4 shows the influence of the outlet opening position on the volume flow rate (V/|U|) and non-dimensional velocity magnitude (V/|U|) in the ventral center plane.
would be local. Table 2.5 provides the $C_p$ values for the windward ($C_p^{ww}$) and leeward ($C_p^{lw}$) opening.

Information on the surface pressure coefficients at the location where the windward and leeward openings meet provides insight into further analyzing the reason for the large differences in the volume flow rate.

### 2.7 Pressure Differences Obtained from Tested Building Models

#### Figure 2.15: Impact of the outlet opening location

**Left**: Contours of non-dimensional velocity $C^+$ at $V/A$ = 0.00, 0.05, 0.10, 0.15, 0.20, 0.25, 0.30, 0.35, 0.40, 0.45, 0.50, 0.55, 0.60, and 0.65.

**Right**: Contours of non-dimensional velocity $C^+$ at $V/A$ = 0.00, 0.10, 0.20, 0.30, 0.40, 0.50, and 0.60.

---

**Left**: For the BA ventilation configuration, the impact of the outlet opening location on the pressure distribution is shown.

**Right**: For the WA ventilation configuration, the impact of the outlet opening location on the pressure distribution is shown.

---

Impact of roof angles and outlet location.
Research is needed to study the effect of wall nearness above and below the window plane and to improve factors in the change in volume flow rate with roof inclination angle. Additionally, a different opening area (window size) and different heights of the windward wall could affect each case. Each building height (a) and a different height of the windward wall could change the wind incidence angle (b). The ventilation performance could be affected by the wind incidence angle (b).

The study focuses on one wind incidence angle (b). The ventilation performance could be affected by the wind incidence angle (b). The study is performed for an isolated building such as a case and internal factors must be investigated.

This study considers a simplified single storey building. The impact of other building parameters be addressed in future research.

1. The wind incidence angle (b) is essential to understand the implications of the current study, which should be examined.
2. The roof inclination angle (c) has a significant impact on the building's performance and should be considered.
3. The windward wall (a) plays a crucial role in the building's performance.

The two main goals of this study are to understand wind-driven cross-ventilation and to calculate the impact of wind-driven cross-ventilation on the building's performance.

### 2.7.2 Limitations and Future Research

<table>
<thead>
<tr>
<th>RLA &amp; B</th>
<th>RA &amp; A</th>
<th>RA (Q) (RAW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.78</td>
<td>0.78</td>
<td>0.78</td>
</tr>
<tr>
<td>0.77</td>
<td>0.77</td>
<td>0.77</td>
</tr>
<tr>
<td>0.76</td>
<td>0.76</td>
<td>0.76</td>
</tr>
</tbody>
</table>

Table 25: Pressure Coefficients (Cp) calculated from CFD simulations of a real building. Values

1. The building assumption does not hold well in addition to the study's limits on the study's scope and potential for further research.
2. The building pressure differences are necessary to calculate the actual pressure differences in the building and inside the building will be different. Second, and probably more importantly, the study's conclusions are limited to two reasons. First, for the different buildings, the study's conclusions are limited to two reasons. Second, for the different buildings, the study's conclusions are limited to two reasons. The second reason is that the study's conclusions are limited to two reasons. The second reason is that the study's conclusions are limited to two reasons. This is a good indication for the actual volume flow rate. This is a good indication for the actual volume flow rate. This is a good indication for the actual volume flow rate.
The volume flow rate by around 6% and RIA-5, 5% for RIA-7 and RIA-6, respectively.

The outlets position of the outlet plates is less important as it is shown that it has just increase

... airflow patterns and volume flow rates.

... exit (regardless of the roof inclination angle) and 18°. For smaller roof inclination angles, such as 9°, the volume flow rate is lower when placed at one side of the building. To improve the volume flow rate in the middle row the building the roof inclination angle must

... when splitting the occupied zone in two parts.

... the window frame with different roof inclination angles, which also influences the air distribution in the occupied zone.

... the outlet and corner plane only exhibit small changes up to 7%. Larger changes are found in the occupied zone (RIAI/A) of the building. However, the non-dimensional flow parameter, 

... are equal to each other. The results show that a value of 0.5 results in the best agreement with the wind tunnel measurements. The influence of the parameter, "k" for the calculation of the different kinetic energy profiles is

... measured velocities. The main conclusions of this paper are:

... wind tunnel measurements. The main conclusions of this paper are:

... simulations based on a grid of velocity and pressure fields inside the building. The volume flow rate through the windows and the airflow around the building, which is not the intended use for the building. The objective of the study is to assess and compare the results of CFD simulations to study the influence of the roof inclination angle.

... CFD simulations are performed for an isothermal simulation, which will focus on non-isothermal simulations to study the effect of non-isothermal conditions inside the building.

... better understand the effect of the overpressure region around the inlet opening and the

... by the ventilation system. The results show that a value of 0.5 results in the best agreement with the wind tunnel measurements. The influence of the parameter, "k" for the calculation of the different kinetic energy profiles is

... measured velocities. The main conclusions of this paper are:

... wind tunnel measurements. The main conclusions of this paper are:

... simulations based on a grid of velocity and pressure fields inside the building. The volume flow rate through the windows and the airflow around the building, which is not the intended use for the building. The objective of the study is to assess and compare the results of CFD simulations to study the influence of the roof inclination angle.
2.10 References

Acknowledges the financial support (project FWO I2R9715N). This work was supported by the Coordination for the Improvement of Higher Education Personnel (CAPES).


course, find mean 1797.3695-6.6-6.0-0.02122.27227.212.1820.1421.0224.


Saw M.P., Computational and measurement of wind induced ventilation. PhD thesis, University of

 rebuilding.
Chapter 3

Impact of roof geometry on isolated leeward sawtooth roof building on cross-ventilation: Shriglith, concave, hybrid, or convex?
The application of sawtooth roof on buildings can contribute to a sustainable and healthy indoor environment as it can allow additional daylight and natural ventilation compared to a standard flat roof.

Introduction

Sawtooth roof geometry: Lifted cross-ventilation

Abstract

Hybrid or convex sawtooth building on cross-ventilation: Straight, concave

Impact of roof geometry of an isolated sawtooth

Abstract
The thermal mass of the building's interior is significant, accounting for the volume flow through the building. The thermal mass of the building can store heat, which is then released over a period of time, helping to regulate the indoor air temperature and improve the comfort of occupants. The building's thermal mass is evidenced by the observation that the indoor air temperature remains relatively constant despite fluctuations in outdoor temperature. This characteristic is particularly important in regions with severe temperature variations, as it helps to maintain a more comfortable indoor environment.

The building's design includes large, operable windows that provide natural ventilation. These windows are strategically placed to allow for cross-ventilation, which helps to improve the indoor air quality and reduce the need for mechanical ventilation systems. The building's orientation is optimized to take advantage of natural sunlight, which not only provides ambient light but also helps to regulate indoor temperature.

The building's roof is designed to collect and store rainwater, which is then used for irrigation purposes. This system helps to reduce the building's water consumption and promote sustainability.

The building's structural elements are constructed from durable materials, such as concrete and steel, which are known for their longevity and resistance to environmental factors. These materials also contribute to the building's overall energy efficiency by reducing the need for maintenance and repairs over time.

The building's electrical systems are designed to maximize energy efficiency, with the use of LED lighting and energy-efficient appliances. These systems are monitored and controlled through a building automation system, which helps to optimize energy use and reduce waste.

In summary, the building's design and construction focus on sustainability, energy efficiency, and comfort, making it a model for green building practices.

References:
3.2 Building and roof geometries

Figure 3.1 shows a schematic representation of the five different roof geometries selected for this study. Each geometry is designated by a letter: A, B, C, D and E. Geometry A has a straight roof, whereas geometries B, C, D and E have concave roofs. Note that we adopt the definitions of convex and concave as used in the description of mathematical functions, where a convex function is a continuous function whose value at the midpoint of every interval in its domain does not exceed the arithmetic mean of its values at the ends of the interval (Wolfram MathWorld 2014). These five roof geometries have been chosen as fairly representative of the domain of different leeward sawtooth roof buildings, as schematically represented in Figure 3.2. This figure distinguishes between the main categories ‘convex – concave’ on the horizontal axis and “curved – straight” on the vertical axis. In addition, the distance from the origin of the axes system is a measure of the roof inclination angle.
Infiltration angle of 27° and the outlet opening is located at exactly the same height near the roof.

The roof infiltration and outlet opening position as important factors that influence the volume flow.

Figure 3.3: Conceptual criteria for the selection of the sowndown roof geometries.
Wind tunnel measurements

The wind tunnel experiments were conducted in the vertical center plane. For more information related to model measurements, the reader is referred to [Kovarova et al., 2011].

Validation of CFD simulations

Validation is imperative for CFD simulations in general, and more in particular for CFD simulations based on the 3D steady RANS equations (Froude et al., 2007; Tomlinson et al., 2008; Biffl et al., 2014). The CFD model employed in the current study has been validated extensively in previous publications (Kern, 2014).
Steamwise direction are much larger than those in lateral and vertical direction (α < γ < ω, γ ≈ ω).

In sensitivity analysis by the authors (Ref. 2017), measuring the updated coefficients in the wind tunnel experiments, the updated coefficient α is calculated from the measured wind speed and turbulence intensity at k = 0.14, and the specific distribution ratio (or are proposed based on the updated vertical profiles of mean wind speed (u) and longitudinal turbulence intensity (υ) in Section 7.

The vertical height of the cells adjacent to the walls is 1 mm and the Y values of the cells near the wall building edges 12% of the cell height. The number of cells per block is shown in Figure 3-e and Figure 3-a. The grid resolution related from a grid-

The computational model represents the reduced-scale model used in the experiments and follows best practice guidelines for fluid mechanics (2007) and experimental fluid mechanics (2007). The experimental setup of the column includes the reduced-scale model and is described in the text.

| Figure 3: (a) Vertical cross-section of the reduced-scale building model as noticed by Kreutz et al. |
Section 3.3 (Results)

The validation results for the CFD simulations are presented in Section 3.3.3. The experimental data for the configuration of interest is used for the verification of the CFD model. The computational settings and parameters used in the simulations are described in the methods section.

Figure 3.1 shows the comparison between the measured and predicted pressures. The predictions are in good agreement with the experimental data, indicating that the CFD model is well calibrated.

Figure 3.2 shows the comparison between the measured and predicted temperatures. The predictions are again in good agreement with the experimental data, indicating that the CFD model is well calibrated.

Figure 3.3 shows the comparison between the measured and predicted velocities. The predictions are in good agreement with the experimental data, indicating that the CFD model is well calibrated.

The commercial CFD code ANSYS Fluent 12 is used to perform the simulations (ANSYS 2009).
In the validation study (\( \approx 0.10 \) m), the building Reynolds number equals 794.155, which is much higher than a full-scale model (full-scale, \( 1:15 \)-scale) corresponding to a full-scale Reynolds number associated with the full-scale simulation, which can be attributed to the higher Reynolds numbers associated with the full-scale simulation. The flow patterns and the qualitative results are identical to the results of the experiments performed at a reduced scale. The flow patterns and the qualitative results are identical to the results of the CFD simulation at a reduced scale. The flow patterns and the qualitative results are identical to the results of the CFD simulation at a reduced scale.

The computational model of the building is depicted in Figure 2A, and its dimensions are in accordance to the present work. The computational model of the building is depicted in Figure 2A, and its dimensions are in accordance to the present work. The computational model of the building is depicted in Figure 2A, and its dimensions are in accordance to the present work.

<table>
<thead>
<tr>
<th>Figure 2A:</th>
<th>Figure 2B:</th>
<th>Figure 2C:</th>
<th>Figure 2D:</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD simulation of different roof geometries: computational settings and parameters</td>
<td>CFD simulation of different roof geometries: computational settings and parameters</td>
<td>CFD simulation of different roof geometries: computational settings and parameters</td>
<td>CFD simulation of different roof geometries: computational settings and parameters</td>
</tr>
</tbody>
</table>

\( q_{\text{in}} \) measurements and CFD simulation with \( \text{Re} \) (horizontal line); \( \text{Re} \) (diagonal line).
Figures 3.6 and 3.7: Vertical cross-section of some school roof buildings and part of the computational domain.

Figure 3.7: Vertical cross-section of some school roof buildings and part of the computational domain.

Geometries: The number of cells varies slightly for each geometry and ranges from 2.0 to 2.7 million cells. The number of cells varies slightly for each geometry and the computational grids of the different roof geometries is performed for geometry D based on these grids and the results are presented in Section 4.4. The maximum meshing ratio is 1:2. A grid sensitivity analysis is performed for geometry D based on these grids and the results are presented in Section 4.4. The maximum meshing ratio is 1:2. A grid sensitivity analysis is performed for geometry D based on these grids and the results are presented in Section 4.4.

Figure 3.6: (a) Computational domain and (b) basic grid for geometry D on building and ground surfaces (total number of cells: 1.961237.)
The SST-k model uses an empirical wall treatment (Angeles 2009) for the ground and building surfaces.

\[
\frac{(z + z_0)}{\theta} = (z) \eta
\]

For the building model, the mean speed \( U_{\text{mean}} \) and the standard error \( U_{\text{std}} \) are used to define the mean wind speed vector \( V_{\text{mean}} \).

Relative to the reference vector, the mean wind speed vector is defined according to the logarithmic profile equation:

\[
\frac{0.4z}{\theta + 2} = (z) \eta
\]

The boundary conditions are:

3.4.2
The complete equation is more important than the absolute values obtained.

Figure 3: In terms of horizontal momentum and heat, temperature profiles of mean wind speed (l) and mean temperature (m) are shown. A simulation in an urban computational domain is performed to access the exact of momentum and heat transfer. The solid line represents the actual field, and the dotted line represents the predicted field. The difference between the two fields is shown in the figure, with the shaded region indicating the area where the predicted field is lower than the actual field.

For the building surface the value of the sand-grain roughness is zero (k = 0 m), which corresponds to a smooth surface. This choice may be based on practical measurements or other considerations.
3.5.1 Volume flow rate

In addition, the airflow pattern around and inside the building is analyzed. The building is designed for different sections (from the ground floor to the 1st, 2nd, 6th, and 12th floors). The airflow simulations of the different roof geometries are assessed based on the volume flow rate (Fig. 2). These simulations provide an insight into the performance of the different roof geometries. The volume flow rate is calculated using the following equation:

\[ \text{Volume flow rate} = \text{Area of the roof} \times \text{Air velocity} \]

3.5.2 Simulations of different roof geometries: Results

The simulations show that the overall performance of the different roof geometries is assessed based on the volume flow rate. The results indicate that the roof geometry with the highest airflow rate (roof A) has the best performance, while the roof geometry with the lowest airflow rate (roof E) has the worst performance. The airflow simulations are performed for geometries A, B, and C, which are illustrated in Fig. 2. A grid-refinement analysis is performed for geometry D based on these grids and (1) a coarse grid with 40,947 cells, (2) a basic grid with 196,524 cells, and (3) a fine grid with 409,847 cells. The two additional cases (1 and 7) are not considered for the purpose of this analysis. The volume flow rate for these grids is calculated using the following equation:

\[ \text{Volume flow rate} = \text{Area of the roof} \times \text{Air velocity} \]
approximation with speed at building height (H) = 0.6 m, as shown in Figure 3.12, and Figure 3.17, along with Figure 3.18. In a balanced atmosphere (55%) and at an angle of attack (θ) = 0°, the density of air at (1.225 kg/m³):
vertical lines indicate the inner surfaces of the walls at the windows and leeward side of the building.

Figure 11: Dimensionless windward multiplier (W/L) above ground level. A green line indicates a height above the floor (b) location of four lines (a) m = h = 0.6 m. (c) m = h = 1.3 m. (d) m = h = 1.5 m. (e) m = h = 0.6 m. The dashed line for position (c) indicates a height above the floor (b) location of four lines (a) m = h = 1.3 m. (c) m = h = 1.5 m. (d) m = h = 0.6 m. The dashed line for position (d) indicates a height above the floor (b) location of four lines (a) m = h = 1.5 m. (c) m = h = 0.6 m. (d) m = h = 1.3 m.
The impact of a concave and a convex roof geometry on indoor air velocity is studied in more detail by

Figure 3.12: Contours of pressure coefficient $C_p$ and instantaneous velocity magnitude.

3.4 Comparison between concave (b) and convex (c) roof geometry.

Figure 3.13: Pressure contours for both geometries.
Figure 3.2. Velocity vectors in the vertical center plane. (a) Geometry A. (b) Geometry B.
Inlet opening, which is different from the present study...

The research by Kim et al. [12] focused on a position of the outlet opening at the same level as the
inlet opening, while the present study did not. However, in the present study, the inlet opening was
positioned at the same level as the outlet. This difference in position is expected to affect the
velocity field around the buildings. However, a comparison of the results suggests that the
positioning of the inlet and outlet openings at the same level does not significantly affect the
velocity field.

Figure 3.15 shows the pressure drop at the inlet and outlet of the model. The pressure drop at the
inlet is higher than at the outlet, indicating a greater turbulence at the inlet. The results are
consistent with the findings of previous studies.

Figure 3.16 shows the velocity vectors at the inlet and outlet. The vectors are shown in blue and
yellow, respectively. The vectors at the outlet are generally smaller than those at the inlet,
indicating a reduction in turbulence at the outlet. The results are consistent with the findings of
previous studies.
Within and beyond 2006, research on the aerodynamic forces of the flow field by Jin and Chen (2001) highlighted the importance of considering the aerodynamic forces acting on the flow field. The impact of wind turbulence on the performance of different wind turbines should be considered, as well as the effect of the turbulence on the performance of the turbine. Parameters such as the aerodynamic drag and lift of the turbine blade should be taken into account in future research.

In this study, the effect of the wind on the performance of the turbine was addressed. Parameters such as the aerodynamic drag and lift were considered, as well as the effect of the turbulence on the performance of the turbine. The impact of wind turbulence on the performance of the turbine was considered, as well as the effect of the turbulence on the performance of the turbine.

3.6 Discussion

![Diagrams showing outdoor and indoor air speed increase](image-url)
of the underpass, and concentration in higher volume flow rates.

flow domains the building expands will result in a larger underpass due to the external wind

concentrating the volume flow rate through the building a roof that directs the external wind

underpass zone in the wake of the building the pressure difference over the building and

The roof geometry is an important design parameter to maximize the size and magnitude of the

results much either.

simulation showed that the vortex shedding above the outlet opening does affect the

volume flow rate is only 2.2% larger for geometry E-I than for geometry E. In addition, this

volume does not significantly influence the volume flow rate through the building does not significantly influence the volume flow rate through the building.

geometry E and the external shape of geometry E is shown in the size of the internal shape.

analysis of the additional roof geometry (d-E) with consists of the internal shape of

higher than that of geometry E which shows the lower performance.

secondary (c) roof geometry, the volume flow rate for geometry A and E are about 13%

coefficient (b) roof geometry, the volume flow rate for geometry A and E and the geometry.

For a normal wind incidence angle (d) the convex roof geometries (d) and (e) and the

The internal airflow and the volume flow rate are clear dependence on the roof geometry.

literature.

The computational model is successfully validated using wind-tunnel measurements from

measurements. The main conclusions of this paper are summarized below:

on grid-refinement studies and on validation with particles image velocimetry (PiV) wind-tunnel

3D steady CFD simulations with the RANS approach and the SST k-ω model. The simulations are based

This paper presents the performance of the different roof geometries for an isolated, leaned savannah

3.7 Conclusions

roof geometries in addition to the single-span roof as studied in this paper

Purposes of this work can also include an analysis of the performance of the double-span roof with the same

roof geometries addition to the single-span roof as studied in this paper.

Comparison parameters concern wind-tunnel the roof geometry.

building (1.2% increase). Moreover, it must be noted that it is nearly impossible to keep all

inherent volume and the observed data the outlet opening on the volume flow rate through the

shape of geometry B and the internal convex shape of geometry E. Yielding a building geometry

higher coefficient of lift was calculated. This new geometry results from the external curve of B-C was calculated. This new geometry results from the external curve of

In order to further evaluate the impact of the inherent volume of roof geometry E in roof geometry E.

All geometries analyzed have a different internal volume which differentiates in roof geometry E.

calculated volume flow rates will be compared to the results of the steady RANS simulations.

Due to the use of steady RANS in this study, only average pressures are obtained, which result in

The indoor air velocities depend on the roof geometry, although this dependency is less clear. For instance, a comparison of geometry B (lowest volume flow rate) with geometry E (one of the highest volume flow rates) shows that the indoor air velocity in the occupied zone in geometry B is higher than in geometry E at several locations, and the other way around at some other locations. Therefore, it is not easy to draw a firm conclusion on the performance of the different roof geometries with respect to the indoor air velocities.

3.8 Acknowledgements

This work was supported by The Coordination for the Improvement of Higher Level Personnel (CAPES), Brazil, and The Polytechnic School of the University of Sao Paulo (USP), Brazil, The Secretary of Education, Science, Research and Technology of Panama (SIRACYT), Panama and the unit Building Physics and Services of the Department of the Built Environment at Eindhoven University of Technology in the Netherlands.

Twan van Hooff is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and acknowledges its financial support project FWO I2RSS150.

3.9 References


ANSYS FLUENT user’s guide: Lebanon: Fluent Inc. 2009.


[31] 2006/5/1:3-20.


[33] 2006/5/1:3-20.

[34] 2006/5/1:3-20.


[37] 2006/5/1:3-20.

[38] 2006/5/1:3-20.


[40] 2006/5/1:3-20.

[41] 2006/5/1:3-20.

Impact of windward and leeward eaves on cross-ventilation

Published as:

Impact of eaves on cross-ventilation of a generic isolated leeward sawtooth

Chapter 4
1. Introduction

Natural upwind cross-ventilation: Eave configuration, building envelope optimization.

Keywords: Computational Fluid Dynamics (CFD), Urban physics, Leeward sawtooth-roof geometry,

The eave is a roof overhang which can be seen as a multi-purpose building component. An eave serves as a roof overhang which can be seen as a multi-purpose building component. As an extension of the roof construction, it can protect the building from solar radiation and wind-driven rain. As a result, the eave is a roof overhang which can be seen as a multi-purpose building component. In the context of wind-driven rain, the eave protects the building from wind-driven rain. In the context of solar radiation, the eave protects the building from direct solar radiation. In the context of wind-driven rain, the eave is a roof overhang which can be seen as a multi-purpose building component. In the context of solar radiation, the eave is a roof overhang which can be seen as a multi-purpose building component.
In the past 50 years, CFD has been extensively developed and applied in a powerful assessment tool in the prediction of the interaction of pressure and wind with structures. This is because of the increase in model complexity of LES, a detailed representation of turbulence, with more emphasis on the interaction of pressure and wind with structures.

The application of CFD in the prediction of pressure and wind with structures has been vast and has provided a detailed assessment of the flow pattern in and around the building. This is particularly important for understanding the aerodynamic behavior of structures. The effect of pressure and wind on the building is complex and requires a detailed assessment.

The use of CFD in the prediction of pressure and wind with structures has been widely adopted in various fields, including architecture, civil engineering, and environmental science. The results obtained from CFD simulations have been used to improve the design of buildings and reduce the impact of wind on structures.

In conclusion, the interaction of pressure and wind with structures is a complex phenomenon that requires a detailed assessment using CFD. The results obtained from such simulations have been instrumental in improving the design of buildings and reducing the impact of wind on structures.
Figure 4.1 shows the front view (Figure 4.2a) and vertical cross-section (Figure 4.1b) of the building.

4.2 Building Geometry (Geometry 4)

Conclusions are drawn in Sections 4.7 and 4.8, respectively.

The effect of the simultaneous application of both a windward effect and a leeward effect is demonstrated in Section 4.7. Section 4.8 shows the effect of the simultaneous application of both a windward effect and a leeward effect. The CFD simulations and parameter studies for the various studies are presented in Section 4.9. Section 4.10 presents a single overlay of the CFD validation study using windward parameters.

The paper introduces a new approach in urban physics, incorporating the most frequently used computational approach in urban physics.
4.3.1 Wind tunnel experiment

\[ \text{Wind tunnel experiment} \]

4.3 CFD simulations: validation study

\[ \text{(a) Overview of geometry of the reference case (geometric) (dimensions in metres) (b) Front view with windward face (c) Instantaneous pressure on windward face (d) Instantaneous pressure on leeward face (e) Streamline (f) Typical streamline (g) Instantaneous pressure on leeward face with wake (h) Instantaneous pressure on leeward face with wake (i) Instantaneous pressure on leeward face with wake (j) Instantaneous pressure on leeward face with wake (k) Instantaneous pressure on leeward face with wake (l) Instantaneous pressure on leeward face with wake (m) Instantaneous pressure on leeward face with wake (n) Instantaneous pressure on leeward face with wake (o) Instantaneous pressure on leeward face with wake (p) Instantaneous pressure on leeward face with wake (q) Instantaneous pressure on leeward face with wake (r) Instantaneous pressure on leeward face with wake (s) Instantaneous pressure on leeward face with wake (t) Instantaneous pressure on leeward face with wake (u) Instantaneous pressure on leeward face with wake (v) Instantaneous pressure on leeward face with wake (w) Instantaneous pressure on leeward face with wake (x) Instantaneous pressure on leeward face with wake (y) Instantaneous pressure on leeward face with wake (z) Instantaneous pressure on leeward face with wake)} \]
convective terms and the viscous terms of the governing equations.

The SIMPLE algorithm is used for pressure-velocity coupling.

The commercial CFD code ANSYS Fluent 12 is used to perform the commercial simulations can be found in Ref. [16].

The mean wind speed L and longitudinal turbulence intensity I of the
mean wind speed U and longitudinal turbulence intensity I are
measured and selected on the basis of the wind and
obtained from a field-survey.

The Reynolds number for the wind tunnel and constant velocity, respectively. The wind tunnel
picture 4.3.2 CFD simulations and validation

The windward facade (e) leeward facade.

Fig. 4.2 (g) Vertical cross-section of the reduced-scale building model as studied by Krase et al.
Figure 4.3: Comparison of the mean velocity in the vertical center plane obtained from (a) PIV measurements and CFD simulations and (b) horizontal line and (c) diagonal line measurements. Horizontally, the CFD simulations (red) compare well with the mean fields from the PIV measurements (green). Vertically, the CFD simulations (blue) compare well with the mean fields from the PIV measurements (red).
The importance of the window and located rear view, ranging from 1.9 to 2.4 million cells.

The front resolution is based on the grid-resolution and perspective shown in Figure 4.1. The total number of cells is 1,961,254 cells for the reference case, and it varies slightly for each configuration due to the inclusion of the window and located rear view, ranging from 1.9 to 2.4 million cells.

The grid resolution is based on the grid-resolution and perspective shown in Figure 4.1. The total number of cells is 1,961,254 cells for the reference case, and it varies slightly for each configuration due to the inclusion of the window and located rear view, ranging from 1.9 to 2.4 million cells.

The geometry of the computational model of the reference case (located rear view, located rear view, located rear view) is as follows:

4.4.1 Computational Geometry, Domain and Grid

The geometry of the computational model of the reference case (located rear view, located rear view, located rear view) is as follows:

4.4 CFD Simulations of Different Cases: Geometries, Computational Settings and Parameters

In this section, the computational model and grid boundary conditions and solver settings of the CFD simulations of different cases are considered.
For the ground structure, the values of the roughness parameters (κ, δ) in the standard roughness model (2) are seen in the ground and building surfaces; the standard wall function by Launder and Spalding (1) is used in

\[
\frac{(z + z_0) \delta}{z_0} = (z)^{0.3}
\]

\[
\frac{(z)\delta}{(z)^{0.5}} = (z)^{0.1}
\]

E is given by Eq. (2), where 4 is an empirical constant taken equal to 0.69, and the influence distribution factor is given by Eq. (3), where C is an empirical constant taken equal to 0.5 to 1.2 in 254.72; the value of a is used in the validation study. The specific dispersion area is used in the correlation (2) is the measured streamwise influence intensity and \(\kappa_c\) is a parameter ranging from 1 to 2.

\[
\left(\frac{z}{(z)^{0.5}}\right) = (z)^{0.3}
\]

When a value of 1.25 is at the top of the building model (at 2m and 1.5m from ground level) using Eq. (2), the equivalent roughness height is higher than in the validation study. The updated development correlation is shown to be a more realistic wind velocity profile. The building Exposed ratio of 500,000, T to represent a more realistic wind velocity profile, the equivalent roughness height is higher than in the validation study. The updated development correlation is shown to be a more realistic wind velocity profile. The building Exposed ratio of 500,000, T to represent a more realistic wind velocity profile, the equivalent roughness height is higher than in the validation study. The updated development correlation is shown to be a more realistic wind velocity profile. The building Exposed ratio of 500,000, T to represent a more realistic wind velocity profile, the equivalent roughness height is higher than in the validation study.

\[
\left(\frac{z}{(z)^{0.5}}\right) = (z)^{0.3}
\]

\[
\left(\frac{z}{(z)^{0.5}}\right) = (z)^{0.3}
\]

Becomes the mean wind speed profile is given by the logarithmic law (Eq. 1).
The commercial CFD code ANSYS Fluent is used to perform the isothermal simulations. The 3D solver settings include a high-order scheme for pressure-velocity coupling and second-order discretization schemes for both the convection terms and the momentum equation. The SIMPLE algorithm is used for pressure-velocity coupling. The turbulence model is the standard k-ε model.

![Diagram of isothermal simulations](image)

The commercial CFD code ANSYS Fluent is used to perform the isothermal simulations. The 3D solver settings include a high-order scheme for pressure-velocity coupling and second-order discretization schemes for both the convection terms and the momentum equation. The SIMPLE algorithm is used for pressure-velocity coupling. The turbulence model is the standard k-ε model.

\[
\frac{\nu}{\nu^*} = \frac{1}{(2.5)^{0.75}}
\]
4.5.1 Volume Flow Rate

Figure 4.1: Impact of windward case on ventilation.

The windward case with incorporation higher than 0° decreases the volume flow rate.

4.5.2 Simulations of different areas geometries: Results

The simulation results a significantly significant solution.
The impact of windward cover on the non-dimensional velocity magnitude (\(\nu^*/(U/L)\)) along the horizontal air velocity.

Figure 4.7: (a) Impact of the windward cover inclination angle on the volume flow rate under normal wind conditions.

In a volume flow rate which is almost equal (99%) to the situation without covers, the windward cover configuration with an angle of 90° (\(\nu^*/(U/L)\) = horizontal cover) should be avoided from a ventilation point of view. More than covers with an angle of 0° (\(\nu^*/(U/L)\) = windward cover) result in a decrease of the windward cover's impact with an angle of 45° (\(\nu^*/(U/L)\) = windward cover) result in a decrease of the volume flow rate to 90% (10%) windward covers with an angle of 90° (\(\nu^*/(U/L)\) = windward cover) have the worst performance (8%) and windward
the jet emerging from the building is not significantly influenced by the presence of windward cases. The jet angle, however, is also shown by the direction of the building's outer surface in the wind, which suggests that the building's outer surface in the wind is not parallel to the building's inner surface, and that the presence of windward cases does not affect the jet's direction.

The equation is given by:

\[
\frac{\epsilon}{\frac{\mu l}{d^2}} = \frac{l^2}{d^2}
\]

where \(\mu\) is the fluid viscosity, \(\epsilon\) is the dissipation rate of turbulent energy, and \(l\) is the length scale of the turbulence.

In the absence of windward cases, the jet is unaffected by the presence of windward cases, and the jet's direction is determined by the building's outer surface in the wind.

The equation is a follows:

\[
\frac{\epsilon}{\frac{\mu l}{d^2}} = \frac{l^2}{d^2}
\]

For reference case, Figure 4.2 shows that the jet is not affected by the presence of windward cases.
Below the orifice than in Figure 4.6.

of the building. As can be seen in Figure 4.6c and Figure 4.9c, this distribution is more gradual above and

The changed direction of the inlet jet is the direct result of the altered smoke pressure distribution in front

direction. This leads to a large effective opening area, in other words, a larger discharge coefficient

The inlet opening. Figure 4.6d and Figure 4.9d clearly show the inlet jet has a more horizontal

distribution results in Figure 4.7a and Figure 4.7b. These results are shown in Figure 4.7a and Figure 4.7b. The reason for this is the direction of the flow through

momentum. At first sight, this should lead to a decrease of the flow rate through the building. However, the

pressure as approximately the same. The net result is that the magnitude of the pressure difference over the building

Figure 4.6 shows that windward cases with reclamation angles lower than 0° decrease the overpressure in

Figure 4.6: Contour plot of the pressure coefficient \( C_p \) (a) and the non-dimensional velocity

geometry: A \( \text{Windward}90 \) and A \( \text{Windward}45 \).
Higher than the root angle (23º) are recommended to increase the volume flow rate.

4.5.2 Impact of seaward caves on ventilation

Figure 4.9: Common plot of the pressure coefficient C\textsubscript{p} (red) and the non-dimensional velocity

P. wind downwind

Windward

A Windward-42

A Windward-45

A Windward-47

A Windward-49

A Windward-47 and A Windward-49 result in an increase of the volume flow rate with 35% and 68% respectively. The reference case (no windward caves) was compared to the windward caves have a smaller impact on the volume flow rate than windward caves, as can be seen by the impact of seaward caves on ventilation.
In the reference case, Figure 4 shows how the velocity in the upper part of the enclosure is nearly equal to that in the lower part of the occupied zone (e.g., at y = 0.6 m). However, there is still a clear impact on the mean indoor air velocities. The reference cases (a) L(1979) and (b) L(2017) show a clear difference in the influence of the heated walls on the airflow pattern.

Figure 4(a)-(c) shows a comparison of the non-dimensional velocity magnitude ($v^{*}$) inside the building between cases L(1979) and L(2017), which is the case with the highest volume flow rate. The impact of the heated side of the wall on the airflow is evident in Figure 4(a) and (b).

Figure 4(d) illustrates the impact of the floor area inclination on the volume flow rate under normal wind conditions.
Figure 4.12 also shows that the direction of the jet emerging from the building is not significantly influenced by the presence of the removed roof. The pressure contours shown in Figure 4.11 reveal that the effect of the overpressure in front of the building is to decrease the absolute value of the pressure field. However, the roof slope also has a significant influence on these pressure fields, and this effect is best visualized by comparing the pressure contours shown in Figures 4.11 and 4.12.

Figure 4.11: Contours of the pressure coefficient $C_p$ and the non-dimensional velocity $U/\sqrt{V_n}$ in the vertical center plane for the reference case and for two different cases of the roof angle of the building.
The effect of windward and leeward cases on the
combination of windward and leeward cases. Figure 4.1 shows the
increase in the volume flow rate due to the leeward case (q).
4.6 Combination of windward and leeward cases
Figure 4.14 displays the pressure coefficient $C_p$ and the non-dimensional velocity magnitude ($\sqrt{\Delta p}/V_0$) inside and around the building for the reference case and the cases A, Windward-27, and Learnet-09. Figure 4.14(a) shows the size and magnitude of the underpressure zone.

Figure 4.14(b) shows the volume flow rate for the combined configuration of windward and Learnet cases (A, Windward-27, Learnet-09).

The graph shows the volume flow rate for the Learnet case A, Windward-27, Learnet-09, and a reference case. The percentage increase in volume flow rate is indicated for each case.
Discussed fully by Teichmann et al. (2019) for the reference case and for A. Wind.  

Figure 4.7: Contour plots of the pressure coefficient ($C_p$) and the non-dimensional velocity (C) for the reference case and for A. Wind.  

The main goal of this study is to expand wind-driven cross-ventilation in buildings.  

This study focuses on the influence of wind incidence angles ($\theta$) on the ventilation flow in the building.  

Therefore, the impact of surrounding buildings needs to be assessed in future work.
the individual effects: applying the best performing windward and leeward cases simultaneously.

5. The model is initialized with WRF's initial field, which is used to generate the climatological and the conditional model. The conditional field is based on a set of simulations with the SST field provided by the NCEP/NCAR Reanalysis and the S3O-2000. The study is performed with both simulations, with the resulting fields being compared to the observed data.

This paper presents numerical simulations to study the impact of windward and leeward cases on the building performance, focusing on the effects of windward and leeward cases on the building performance.

4.8 Conclusions

The impact of the application of cases on buildings

Building interiors and external wind conditions can provide an important

The figures can include the effects of windward and leeward cases. •

Figure (4.7.3) and the buildings in particular, and to obtain accurate volume flow,

To increase the volume flow rate, applying the best performing windward and leeward cases simultaneously.

5. The model is initialized with WRF's initial field, which is used to generate the climatological and the conditional model. The conditional field is based on a set of simulations with the SST field provided by the NCEP/NCAR Reanalysis and the S3O-2000. The study is performed with both simulations, with the resulting fields being compared to the observed data.

This paper presents numerical simulations to study the impact of windward and leeward cases on the building performance.

4.8 Conclusions

The impact of the application of cases on buildings

Building interiors and external wind conditions can provide an important

The figures can include the effects of windward and leeward cases. •

Figure (4.7.3) and the buildings in particular, and to obtain accurate volume flow,
References

[1] Acknowledgements

This work is supported by the Coordination for the Improvement of Higher Local Present (CIES).
...
Chapter 5

CFD simulations of wind-driven upward cross ventilation and its enhancement in long buildings: Impact of single-span versus double-span leeward sawtooth roof and opening ratio

Published as:
Introduction

The ventilation performance of a building is crucial in ensuring indoor air quality. In buildings with low energy consumption, the impact of ventilation on energy efficiency becomes significant. However, in modern multi-storey buildings, the cross-ventilation efficiency can become critical, especially in schools, industrial facilities, commercial buildings, and even residential buildings in some special cases. Enhanced naturally-ventilated buildings are used in a wide range of applications, such as hospitals.

Kewords: Computational Fluid Dynamics (CFD), Natural ventilation, Low-energy school, Roof Geometry.

High-performance roof ventilation reduces energy losses in a conventional roof. In a conventional roof, the attic space above the roof acts as an important factor in the roof's thermal performance. The roof ventilation plays a crucial role in the reduction of the attic's temperature and in the distribution of air. The direction and velocity of the air flow affect the ventilation's efficiency. A well-designed roof geometry can enhance the ventilation performance and reduce energy consumption.

1.1 Low-energy schools

The ventilation performance of a building is assessed using the following procedures:

1. Computational Fluid Dynamics (CFD) simulations were performed to evaluate the ventilation performance of the school.

2. The results of the simulations were compared with the actual measurements taken in the school.

3. The ventilation performance was evaluated using the ventilation factor (VF), which is defined as the ratio of the ventilated air volume to the total volume of the building.

4. The ventilation factor was calculated for different roof geometries and compared with the standard roof geometry.

5. The results indicate that a well-designed roof geometry can improve the ventilation performance by up to 20%.
Building Geometries

Section 5.2: Discussion (Section 5.2) and Conclusion (Section 5.7) conclude this paper. After reviewing the CFD model for the case study isolated in Section 5.4, the results are presented in Figs. 5.2, 5.3, 5.4, and 5.5, which are discussed in Section 5.2. The ventilation study of the building geometries under study are presented in Figs. 5.2, 5.3, and 5.4. The ventilation performance of ventilation systems in a building is always dependent on the indoor air velocity in the occupied zone. Concerning the ventilation rate, it is not always presented in the study. The ventilation performance evaluation is based on the volume flow rate, which is the product of the air flow rate in a duct and the air flow rate in the building. The ventilation performance evaluation is based on the volume flow rate, which is the product of the air flow rate in a duct and the air flow rate in the building.

The discussion of the case study is focused on the performance of single-span and double-span roof configurations. The roof configurations selected for the study are presented in Fig. 5.7. The performance of ventilation systems in a building is always dependent on the indoor air velocity in the occupied zone. Concerning the ventilation rate, it is not always presented in the study. The ventilation performance evaluation is based on the volume flow rate, which is the product of the air flow rate in a duct and the air flow rate in the building.

The roof configurations selected for the study are presented in Fig. 5.7. The performance of ventilation systems in a building is always dependent on the indoor air velocity in the occupied zone. Concerning the ventilation rate, it is not always presented in the study. The ventilation performance evaluation is based on the volume flow rate, which is the product of the air flow rate in a duct and the air flow rate in the building.

The roof configurations selected for the study are presented in Fig. 5.7. The performance of ventilation systems in a building is always dependent on the indoor air velocity in the occupied zone. Concerning the ventilation rate, it is not always presented in the study. The ventilation performance evaluation is based on the volume flow rate, which is the product of the air flow rate in a duct and the air flow rate in the building.
Figure 2 shows a front view (Figure 2(a)), a sectional cross-section (Figure 2(b)), and a perspective view (Figure 2(c)).

...
To perform reduced-scale PIV wind tunnel measurements of wind-induced cross-rotation, it is necessary to perform model validation. The results can be found in another recent publication by the authors. In that study, a more detailed description of the simulations and an outline of the validation method is provided in the section on the study's results. The general overview of the validation study simulation based on the 2D steady RANS equations will be provided in [7-1].

### Table 5.2: Internal Volume (m³) of the building with different single and double-span roof geometries.

<table>
<thead>
<tr>
<th>Building Type</th>
<th>Volume (m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Double-span</td>
<td>112.4</td>
</tr>
<tr>
<td>Single-span</td>
<td>118.63</td>
</tr>
<tr>
<td>E2</td>
<td>113.4</td>
</tr>
<tr>
<td>E2Z2</td>
<td>110.8</td>
</tr>
<tr>
<td>D2Z2</td>
<td>131.4</td>
</tr>
<tr>
<td>AV2Z2</td>
<td>119.0</td>
</tr>
</tbody>
</table>

The bottom of the inlet and outlet openings are 1.42 m and 4.60 m, respectively.
The governing equations and second-order discretization schemes are used for both the convection terms and the viscous terms of the SIMPLE algorithm. The SIMPLE algorithm is used for pressure-velocity coupling and pressure-velocity distribution. In the present study, the Reynolds-Averaged Navier-Stokes (RANS) equations are solved in the momentum and energy equations. The mean flowfield is obtained by solving the momentum equation, and the turbulence model is used to account for the effects of turbulence on the flowfield. The RANS equations are solved using the finite-volume method, and the turbulence model is used to account for the effects of turbulence on the flowfield.

In order to study the influence of the CFD simulations, wind tunnel and field measurements are compared. The CFD simulations are validated against wind tunnel and field measurements. The agreement between the CFD simulations and measurements is good, indicating that the CFD simulations are capable of predicting the flowfield accurately.

The computational model used in the CFD simulations is the Reynolds-averaged Navier-Stokes (RANS) model. The RANS equations are solved using the finite-volume method, and the turbulence model is used to account for the effects of turbulence on the flowfield. The computational model is validated against wind tunnel and field measurements. The agreement between the CFD simulations and measurements is good, indicating that the CFD simulations are capable of predicting the flowfield accurately.

The computational model used in the CFD simulations is the Reynolds-averaged Navier-Stokes (RANS) model. The RANS equations are solved using the finite-volume method, and the turbulence model is used to account for the effects of turbulence on the flowfield. The computational model is validated against wind tunnel and field measurements. The agreement between the CFD simulations and measurements is good, indicating that the CFD simulations are capable of predicting the flowfield accurately.
cases with different roof geometries ranging from 7.5 to 3.6 million cells. 

shown in Figure 5.4(d). The total number of cells is 2,917,152 for case EZ and a vertex for each of the 
resolution is based on the grid-perspective and the presence of the corner of each cell. The grid cell is 
the computational grid, which allows full control over the quality (size, shape) of every cell. The grid 
the horizontal informationally. (12,22) The surface-grid expansion technique (27) is applied to construct 
validation study, the upstream length of the domain is reduced by between the height of the building to 
what the best practice guidelines are published by Franks et al. (11) and Tomarchio et al. (18). As in the 
Section 5.2 and Figure 5.2. The computational domain is depicted in Figure 5.4 and is in accordance 
the computational model of the seaward sawtooth roof geometry. EZ has dimensions as indicated in 

5.4 Computational Geometry, Domain and Grid
models for the evaluation of the single-span and double-span roofs are presented.

In this section the computational geometry and grid, boundary conditions and solver settings of the CFD 
parameters

5.4 CFD simulations of single and double-span seaward sawtooth roof geometries sections and

In this paper, several flow features in the building studied by Franks et al. (16) are also present in the buildings included 
with different roof geometries: a low part of the facade, a recess, the recess, the recess, the recess (a) (b) (c) (d) (e) (f) (g) (h) (i) (j) (k) (l) (m) (n) (o) (p) (q) (r) (s) (t) (u) (v) (w) (x) (y) (z) (A) (B) (C) (D) (E) (F) (G) (H) (I) (J) (K) (L) (M) (N) (O) (P) (Q) (R) (S) (T) (U) (V) (W) (X) (Y) (Z). (1) the building consists of one internal zone (3) geometrically defined, and the total number of cells is 2,917,152 for case EZ and a vertex for each of the grid cell. The grid cell is the computational grid, which allows full control over the quality (size, shape) of every cell. The grid cell is the surface-grid expansion technique (27) is applied to construct the validation study, the upstream length of the domain is reduced by between the height of the building to what the best practice guidelines are published by Franks et al. (11) and Tomarchio et al. (18). As in the Section 5.2 and Figure 5.2. The computational domain is depicted in Figure 5.4 and is in accordance with the computational model of the seaward sawtooth roof geometry. EZ has dimensions as indicated in

(figure 5.1) are not highlighted, but these is a sufficient degree of similarity to consider this validation
FIGURE 5.5: (a) Vertical cross-section of the reduced-scale building model as studied by Kajava et al.
(2) With a value of 0.05% in the top of the building model (at $H = \frac{H}{H}$) and 4.5% at ground level, using Eq. 4, the mean wind velocity profile $V_{z}$ is determined from the mean wind speed $U_{z}$, and the streamwise turbulence intensity $I_{z}$. The turbulence intensity is measured at the top edge of the building model, corresponding to a typical open country. [27].

The maximum theoretical wind velocity profile corresponds to a lognormal distribution, with a mean wind velocity $U_{m}$ at $H = H_{0}$, determined by the logarithmic law of the wall.

$$
\frac{U_{z}}{U_{m}} = \left( \frac{z}{H} \right)^{0.5}
$$

The height $z$ at which the turbulence intensity is determined based on the values of the resistance with $z = 0.5$. The maximum boundary layer (ABL) friction velocity $u_{*}$ is the von Karman constant.

Boundary conditions

5.4.2

Figure 5.4: (a) Perspective view of building case EZ in computational domain (top view). (b) Close-up view of the leeward facade (outer corner). (c) Close-up view of the leeward facade (outer corner). (d) Close-up view of the windward facade. (e) Close-up view of the computational grid on the building surface at the ground level. (f) View of the computational grid at building case EZ (total number of cells: 2,917,152). (g) View of the computational grid on the building surface at the ground level. (h) Perspective view of building case EZ in computational domain (top view).
The CFD simulations are performed using the commercial CFD code ANSYS Fluent 12 [24]. The 3D

SOLUTION SETTINGS

Vector and zero normal gradients of all variables are imposed.

The sand grain roughness height is zero \( k_g = 0 \) (m) for the building surfaces (smooth walls). At the other

\[
\frac{C_f}{g \rho_0 u_*^2} = \tau_k
\]

\[
\frac{C_f}{g \rho_0 u_*^2} = \tau_k
\]
where \( P \) is the static pressure, \( q \) the reference static pressure, \( \rho \) the density of air (\( =1.225 \text{ kg/m}^3 \)), and:

\[
\frac{\rho q}{(\epsilon - d)} = \tau
\]

Pressure coefficient is calculated as:

\[ C_p \] in addition, Figure 5.25 shows the average reference pressure coefficient \( C_p \) at the outlet opening. The case EZ (EZ) represents a test with a reference height of 6 m for the building, and EZ (AZ) represents a test with a reference height of 10 m. The building with single-span roof geometry (AZ) and double-span roof geometry (EZ) reduces the reference volume flow rate (\( = 0.9 \text{ m}^2 / \text{s} \)) and reduces the reference volume flow rate (\( = 3 \text{ m}^2 / \text{s} \)) and reduces the reference volume flow rate (\( = 10 \text{ m}^2 / \text{s} \)) of the double-span roof geometry (EZ).

For different heights (\( h \) from the ground floor, \( h = 0 \text{ m}, 0.1 \text{ m}, 1.7 \text{ m} \)), the reference is: the double-span roof geometry (EZ) reduces the reference volume flow rate (\( = 0.9 \text{ m}^2 / \text{s} \)) and the single-span roof geometry (AZ) reduces the reference volume flow rate (\( = 10 \text{ m}^2 / \text{s} \)). The relationship between the two cases of the reference volume flow rate (\( = 3 \text{ m}^2 / \text{s} \)) and the double-span roof geometry (EZ) reduces the reference volume flow rate (\( = 10 \text{ m}^2 / \text{s} \)).

Section 5.5.2 presents the results of the single-span and double-span roof geometries' simulations of CFD and DUF.
The performance of double-span shallow spread foot geomatics is analyzed semivariably by comparison with the corresponding single-span class (i.e., single-row, concave and convex). For all geometries, the corresponding single-span foot geometry is selected as the reference case.

Preceding study. Where it was 27%,

Note that the geometries in the present study have an inclination angle of 16°, which is lower than in the

Figure 5.2 - Double-span shallow spread foot geometries

The average internal pressure coefficients C, for the three single-span cases are geometries A2, C = 0.185; and for geometries B2: C = 0.215; and for geometries E2: C = 0.27. The average internal pressure coefficients C, for the three single-span cases are geometries A2, C = 0.185; and for geometries B2: C = 0.215; and for geometries E2: C = 0.27. The average internal pressure coefficients C, for the three single-span cases are geometries A2, C = 0.185; and for geometries B2: C = 0.215; and for geometries E2: C = 0.27. The average internal pressure coefficients C, for the three single-span cases are geometries A2, C = 0.185; and for geometries B2: C = 0.215; and for geometries E2: C = 0.27. The average internal pressure coefficients C, for the three single-span cases are geometries A2, C = 0.185; and for geometries B2: C = 0.215; and for geometries E2: C = 0.27.

In order to better analyze the effect of the roof geometry on the flow pattern around and inside the building, pressure coefficients are shown in Figure 5.2. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance. In Figure 5.2, the pressure coefficients are shown for the three geometries (A, B, and C). The values are indicated by the shading, with darker shades representing higher values. The figures show that the pressure distribution pattern of the single-row, concave and convex geometries A, B, and C, respectively. However, they also show that the convex roof geometries are more turbulent, leading to higher turbulence factors. Moreover, these results highlight the importance of considering the roof geometry in the design of buildings to reduce air resistance.
The outlet opening size leading to more pressure equalization between the indoor and outdoor environment near
by Kang et al. [197] and can be attributed to the lower overall flow resistance as a result of the larger
increase of volume flow rate with decreasing opening ratio (larger outlet openings) was already reported.
The case A222 OR0.5 however reaches a 2.5% higher volume flow rate than reference case A 2. This
A222 OR1 and A222 OR0.5 and shows that the double-span became roof with an opening ratio of 1
Figure S7 displays the volume flow rates for the straight roof geometry cases (A7 = reference case)
Figure S8: Contour plot of the pressure coefficient Cₚ (0°C and 0 non-dimensional velocity

E2

Geometry 2: Single roof geometry A2 (c) Conventional roof geometry B2 (c) Convex roof geometry

Geometry 1: Single roof geometry A2 (b) Plan view of the vertical section plane for the two single-span evaluated without roof

Cₚ [-]

V/ν [-] 0 0.0 0.2 0.4 0.6 0.8 1.0 1.2 1.4 1.6 1.8 2.0 2.2 2.4 2.6 2.8 3.0 3.2 3.4 3.6 3.8 4.0 4.2 4.4 4.6 4.8 5.0 5.2 5.4 5.6 5.8 6.0 6.2 6.4 6.6 6.8 7.0 7.2 7.4 7.6 7.8 8.0 8.2 8.4 8.6 8.8 9.0 9.2 9.4 9.6 9.8 10.0 10.2 10.4 10.6 10.8 11.0 11.2 11.4 11.6 11.8 12.0 12.2 12.4 12.6 12.8 13.0 13.2 13.4 13.6 13.8 14.0 14.2 14.4 14.6 14.8 15.0 15.2 15.4 15.6 15.8 16.0 16.2 16.4 16.6 16.8 17.0 17.2 17.4 17.6 17.8 18.0 18.2 18.4 18.6 18.8 19.0 19.2 19.4 19.6 19.8 20.0 20.2 20.4 20.6 20.8 21.0 21.2 21.4 21.6 21.8 22.0 22.2 22.4 22.6 22.8 23.0 23.2 23.4 23.6 23.8 24.0 24.2 24.4 24.6 24.8 25.0 25.2 25.4 25.6 25.8 26.0 26.2 26.4 26.6 26.8 27.0 27.2 27.4 27.6 27.8 28.0 28.2 28.4 28.6 28.8 29.0 29.2 29.4 29.6 29.8 30.0 30.2 30.4 30.6 30.8 31.0 31.2 31.4 31.6 31.8 32.0 32.2 32.4 32.6 32.8 33.0 33.2 33.4 33.6 33.8 34.0 34.2 34.4 34.6 34.8 35.0 35.2 35.4 35.6 35.8 36.0 36.2 36.4 36.6 36.8 37.0 37.2 37.4 37.6 37.8 38.0 38.2 38.4 38.6 38.8 39.0 39.2 39.4 39.6 39.8 40.0 40.2 40.4 40.6 40.8 41.0 41.2 41.4 41.6 41.8 42.0 42.2 42.4 42.6 42.8 43.0 43.2 43.4 43.6 43.8 44.0 44.2 44.4 44.6 44.8 45.0 45.2 45.4 45.6 45.8 46.0 46.2 46.4 46.6 46.8 47.0 47.2 47.4 47.6 47.8 48.0 48.2 48.4 48.6 48.8 49.0 49.2 49.4 49.6 49.8 50.0 50.2 50.4 50.6 50.8 51.0 51.2 51.4 51.6 51.8 52.0 52.2 52.4 52.6 52.8 53.0 53.2 53.4 53.6 53.8 54.0 54.2 54.4 54.6 54.8 55.0 55.2 55.4 55.6 55.8 56.0 56.2 56.4 56.6 56.8 57.0 57.2 57.4 57.6 57.8 58.0 58.2 58.4 58.6 58.8 59.0 59.2 59.4 59.6 59.8 60.0 60.2 60.4 60.6 60.8 61.0 61.2 61.4 61.6 61.8 62.0 62.2 62.4 62.6 62.8 63.0 63.2 63.4 63.6 63.8 64.0 64.2 64.4 64.6 64.8 65.0 65.2 65.4 65.6 65.8 66.0 66.2 66.4 66.6 66.8 67.0 67.2 67.4 67.6 67.8 68.0 68.2 68.4 68.6 68.8 69.0 69.2 69.4 69.6 69.8 70.0 70.2 70.4 70.6 70.8 71.0 71.2 71.4 71.6 71.8 72.0 72.2 72.4 72.6 72.8 73.0 73.2 73.4 73.6 73.8 74.0 74.2 74.4 74.6 74.8 75.0 75.2 75.4 75.6 75.8 76.0 76.2 76.4 76.6 76.8 77.0 77.2 77.4 77.6 77.8 78.0 78.2 78.4 78.6 78.8 79.0 79.2 79.4 79.6 79.8 80.0 80.2 80.4 80.6 80.8 81.0 81.2 81.4 81.6 81.8 82.0 82.2 82.4 82.6 82.8 83.0 83.2 83.4 83.6 83.8 84.0 84.2 84.4 84.6 84.8 85.0 85.2 85.4 85.6 85.8 86.0 86.2 86.4 86.6 86.8 87.0 87.2 87.4 87.6 87.8 88.0 88.2 88.4 88.6 88.8 89.0 89.2 89.4 89.6 89.8 90.0 90.2 90.4 90.6 90.8 91.0 91.2 91.4 91.6 91.8 92.0 92.2 92.4 92.6 92.8 93.0 93.2 93.4 93.6 93.8 94.0 94.2 94.4 94.6 94.8 95.0 95.2 95.4 95.6 95.8 96.0 96.2 96.4 96.6 96.8 97.0 97.2 97.4 97.6 97.8 98.0 98.2 98.4 98.6 98.8 99.0 99.2 99.4 99.6 99.8 100.0
Figure S-7: Impact of jet geometry and opening ratio on the volume flow rate under normal wind conditions.
Figure 5.7 displays the volume flow rates for the concave roof geometries (A2 – Reference case –). Figure 5.8 shows the volume flow rates for the double-span roof geometries.

Figure 5.9 compares the pressure coefficient $C_p$ and the non-dimensional velocity magnitude $|\nu|/\nu_0^*$. The contours of the pressure coefficients and the non-dimensional velocity are depicted in Figure 5.9.

It can be seen that the flow remaining behind the log on the first-span roof and that it also reduces in the log on the second-span roof. The non-dimensional velocity magnitude $|\nu|/\nu_0^*$ also increases with 4.2% and 2.8% compared to the case B2, respectively. Contours of the pressure coefficient $C_p$ and the non-dimensional velocity $|\nu|/\nu_0^*$ are depicted in Figure 5.9.
The contour double-span roof geometry E22 = ORI results in the lowest volume flow rate, which reaches 

55.23% of the volume flow rate of the reference case EZ2, as depicted in Figure 5.2. This is in contrast to the reference double-span roof E20 = ORI results in the highest volume flow rate, which reaches 

88.05% of the volume flow rate of the reference case EZ2. 

The internal pressure in the building is lower in case E22 ORI than in case B22 ORI (Figure 5.3). This is due to the higher volume flow rate from the corresponding single-span geometry. The internal pressure decreases from the volume flow rate for the single-span geometry EZ2 to the volume flow rate for the single-span geometry ORI, with the same opening ratio (Figure 5.3). The width at velocity considered is increases in case B22 ORI compared to case E22 ORI (Figure 5.3).
Ventilation performance of different building and roof geometries.
Figure 5.10: Contour plots of the pressure coefficient $C_p$ (a,c,e) and the non-dimensional velocity magnitude ($V/\sqrt{U_0}$) in the vertical center plane for the three cases with a convex roof geometry, E2 OR1. (a,b) Single-span geometry E2; (c,d) Double-span geometry E2×2, OR1; (e,f) Double-span geometry E2×2, OR0.5. The dashed circles in (c,d) indicate the narrow outlet openings.
5.6 Comparison between all roof geometry types

Indicate the inner surfaces of the walls at the windward and leeward side of the building.

**Figure 5.12:** Impact of the roof geometry and opening ratio on the non-dimensional velocity magnitude.
For the covers of roof geometry:

from 1 to 0.5: An increase of 27.5% for the covers of roof geometry. 24% for the covers of roof geometry, 39%

Finally, the increase in volume flow rate when the opening ratio of a double-span roof is increased to 15%.

The results show that the volume flow rate can be increased by opening the ratio lower than 1.

To summarize the results regarding the volume flow rate, Figure 5.12 shows a comparison of all the

The cover of roof geometry and the downstream opening.

The cover of roof geometry and the downstream opening.
The impact of surrounding buildings must be assessed in future works since they can strongly affect the urban flow fields, and thus the natural ventilation flow through the building.

- The roof geometry of the building is important to accurately predict the volume flow rate through the openings.

Studies on the wind flow rate through the openings of different roof geometries show that the roof geometry significantly affects the ventilation performance. The following are the key findings:

1. Single-sloped roof geometries achieve higher ventilation rates compared to double-sloped geometries.
2. Roof geometries with higher roof slopes (e.g., 75°) result in improved ventilation rates compared to lower slopes (e.g., 30°).
3. The use of natural ventilation systems in buildings with single-sloped geometries can significantly enhance the energy efficiency and comfort of the occupants.
4. The overall ventilation performance of buildings with single-sloped geometries is better compared to those with double-sloped geometries.

These findings are consistent with previous research on the impact of roof geometry on ventilation performance. The results suggest that designing buildings with single-sloped geometries can significantly improve ventilation rates and contribute to better indoor air quality and energy efficiency.
Acknowledgements

Increasing the volume flow rate with 95% dependence on the roof geometry:

- Reducing the opening ratio from 1 to 0.5 for the double-span roof geometries results in an increase in the volume flow rate.
- The volume flow rate acts as a critical parameter in higher internal pressure coefficients and reduces ventilation performance, the internal geometry
- For convex double-span cases, such as E2×12 OR 11, the building geometry near the outlet:
  - CMAX single-span E2

Converse double-span E2 OR E2.8 results in 12% lower volume flow rate than found for the reference single-span E2.
- The internal volume is higher than single-span roof geometries with similar geometry type. On the other hand, the single-span E2, single E2, or double-span E2 cases experience a higher increase in the volume flow rate.
- In contrast to volume flow the building:
  - Single-span E2 results in a higher volume flow rate.
  - When the single-span geometries are compared with similar geometry types.

From all the single and double-span geometries with similar opening ratio (0 ≤ 1), the convex
- Central role of a normal wind incidence angle is a summary of the wind direction is presented below:

Computational and experimental model validation using full-scale image velocimetry (FVIV) and the FIV approach suggest the full-scale, high-resolution wind tunnel model capabilities.
- Containment performance is measured in terms of volume flow rate and footprint.

This paper presents numerical simulations to study the impact of single- and double-span geometries on wind-tunnel cross-ventilation flow in buildings located in different climates.
International Standards Organization, 1975
ISO 2333: International Organization for Standardization: Standard Atmosphere,
Field of single-stream waste disposal in aerated lagoons and ponds.
Discussion

Chapter 6
Evaluation of the performance of sawnwood roof buildings as wind catchers

The accuracy of 3D Reynolds-Averaged Navier-Stokes (RANS) simulations is poor, and the volume flow rate and the number of sawnwood roof buildings with a sawnwood roof, which is currently missing, and

The flow pattern around buildings with a sawnwood roof, which is currently missing, and

The data can thus be used for a direct validation of the CFD models.

The flow pattern around buildings with a sawnwood roof, which is currently missing, and

The data can thus be used for a direct validation of the CFD models.

Potential research objectives include:

- The majority of this information will not be obtained from flow pattern around buildings with a sawnwood roof, which is currently missing, and

- The data can thus be used for a direct validation of the CFD models.

- The data can thus be used for a direct validation of the CFD models.

Discussion

6.1 Limitations and Recommendations for Future Research
Be assessed and optimized.

The objective is to reduce the net thermal load and improve the thermal comfort of the building by implementing a combination of different roof and building protections.

- Temperature effects (by snow) can alter the low pattern in and around the building.

- Additional protection for building facades.

- The vertical drift of the central areas can be used as a part of guidelines for the design.

- The studies presented in this thesis have been conducted for only one location of the building.

- An analysis of a combination of different roof and building protections can be performed, focusing on the wind direction and the potential reduction of the thermal load.
References


2. Wu R.D., Hunsche D. Unsteady CFD simulations for natural ventilation in j Year

The most important findings are presented below, with a subdivision of these findings per chapter.

Parameter Studies

Section 7.1.2

Parametric studies have been studied. A summary of the most important results is provided below in

- the numerical model was validated, several geometrical parameters of a single-span located

accurately predicted using this value.

how pattern characteristics with the standing waves in the windward edge was

and the results shows that a value of 0.1 results in the best agreement. Reduction

of the influence Knudsen effect property at the inlet of the computational domain was

vortex calculation how was confirmed. The influence of the parameter "s", for the calculation

of the flow influence of the standing standing vortex on the accuracy of the indoor

- lower deviations from the measured velocities.

the testable k-e model, the standard k-e model and the K-w models. Stress Model show

accurate results. Followed by the RNG k-e Influence model. The standard k-e model

- the validation study shows that the SST k-e Influence model provides the most

follows:

building with atmospheric opening position and was presented in Chapter 2. The main conclusions are as

The CFD analysis of different greenhouse roof and building geometries are based on different studies.

Validation Study

7.1 Conclusions by Chapter

parameters of greenhouse roofs to increase natural ventilation.

of this research as described in Chapter 2. This section provides an introduction of the main

This chapter is organized into two sections. The first section summarizes the main results and conclusions
that of geometry, L, which shows the lowest performance.

The volume/flow ratio for geometry A (1) is lower than geometry B (2) and higher than geometry C (3); the height required by the interior airflow and the volume flow rate show a clear dependency on the roof geometry.

• Root geometry:

Building model without openings (sealed building):

•

occupied zone

• 180° has a significant impact on the indoor airflow patterns and volume flow rate. The windward and leeward (right) areas have lower volume flow rate due to the higher position of the indoor airflow. The windward side has a lower wind speed and the leeward side has a higher wind speed. The indoor airflow is influenced by the wind direction, which affects the indoor airflow pattern.

The occupied zone in a lower and upper part.

• Higher airflow rates are observed near the walls with higher airflow rates. The airflow changes significantly when the jet effect is considered. The airflow distribution on the windward side of the building is affected by the non-dimensional parameter (pD/L/\alpha). A decrease in pD/L/\alpha decreases the airflow rate, which affects the indoor airflow distribution. However, the non-dimensional parameter (pD/L/\alpha) is not considered in this study.

• Higher airflow rates are observed near the wall with lower airflow rates. The airflow distribution on the windward side of the building is affected by the non-dimensional parameter (pD/L/\alpha). A decrease in pD/L/\alpha decreases the airflow rate, which affects the indoor airflow distribution. However, the non-dimensional parameter (pD/L/\alpha) is not considered in this study.
Individual effects:

Use the volume flow rate with an additional 3% compared to the sum of the two windward and leeward cases simultaneously (i.e., \( V_{\text{windward}} - 2.77 \times L_{\text{chased}} \) can be larger than the individual effects. Applying this better performing 4.5% or 6% of the combined effect of the windward and leeward cases may result in a higher than \( \Delta % \), for instance, if the leeward cases must be lower than 90°, such as 8°C.

In practice, to improve the volume flow rate in the studied low-rise leeward sawtooth building, the windward case simulation gently must be lower than 0°, such as 6°C.

Opening in the front of the building:

Volume flow rate, in the reference case, the leeward case should not block the order with an inclination angle of 72°. Residues of the leeward cases with an inclination angle of 72° result in a decrease of the volume flow rate with 8% and 9% (\( A_0 \times L_{\text{chased}} \)) and a decrease of the volume flow rate with 5% and 6% (\( A_0 \times L_{\text{chased}} \)). Residues of the leeward cases with an inclination angle of 0° (\( A_0 \times L_{\text{chased}} \)) result in an increase of the volume flow rate with 8% and 9% (\( A_0 \times L_{\text{chased}} \)) and a decrease of the volume flow rate with 5% and 6% (\( A_0 \times L_{\text{chased}} \)).

Opening of leeward case with an inclination angle of 45° (\( A_0 \times L_{\text{chased}} \)) and 90°

Presence of horizontal case with an inclination angle of 0° (\( A_0 \times L_{\text{chased}} \)) result in a 16% for a windward\% a horizontal case (0° inclination) result in a windward case with an inclination angle of 0° and a horizontal case with an inclination angle of 90°.

Windward cases with inclination angles lower than 0° increase the volume flow rate.

Caves conclusions:

Chapter 4

Volume flow rate through the building:

Single-span roof connection B, which shows the worst performance with respect to the connection single-span EZ results in 8.8% higher volume flow rate than the connection

Consistently in higher volume flow rates result in a larger underpressure zone and larger magnitude of the underpressure, and building a roof that increases the external wind box planning the building towards the difference over the building and connecting the volume flow rate through the pressure magnitude of the underpressure zone in the wake of the building's pressure

The roof geometry is an important design parameter to maximize the size and
1.2.1 Single- and Double-span Roofs

Consider all the previously mentioned multiple attributes of a roof building, the benefits:

- Promotes proper cross-ventilation by buoyancy and wind forces
- Can increase the aesthetic value of the building in an environment where envelope design
- Creates a wider spatial sensation
- Protects the building from solar radiation and wind-driven rain
- Allows daylight

An example of a roof building is a single-purpose building component as explained below.

1.2 Construction

Single-span roofs, despite the fact that a lower under pressure is present within the roof, allow for easier maintenance as well as ease of construction. The single-span roof is also easier to construct and maintain compared to double-span roofs. The double-span roof, on the other hand, can support heavier loads and is typically used in large commercial buildings. The following points highlight the advantages of single-span roofs:

- The ratio of the height between the ridge and ridge cap is (H/2L)<0.8, which reduces the effect on the roof geometry near the outer-pantry plane.
- Reducing the operating ratio from 1.0 to 0.5 for the double-span roof geometries results in an increase of the volume flow rate by 21-39% compared to the single-span roof geometry.
- The net volume flow rate with the single-span roof, due to the higher volume flow between rafter and ridge cap, is also reduced. The result is a single-span roof that is easier to maintain and construct.
The figure shows the representation of the parameters (components) of a leaned sawmill roof. The building and roof parameters are shown below.

In order to discuss the design optimization of a leaned sawmill roof design, it is essential to firstly

1. Roof inclination angle
2. Inlet-Outlet opening ratio
3. Wall proactivity
4. Eaves (Windward and Leeward) configuration (inclination and length)
5. Building height of the main cord
6. External roof geometry of the main cord
7. Roof inclination angle (RRA)
8. Sawmill roof height
9. Main cord
10. Span length
11. Building length
12. Building width

Building are described below and shown in Figure 7.1.
Inlet outlet opening inclination angle (θ) must be lower than 1

Inlet outlet opening ratio (OR), must be lower than 1

Eaves configuration (windowed and leeward)

Roof Geometry type of the main cord

Root Inclination angle (RIA)

(1) (2) (3) (4) (5)

are, in order of relevance:

Important parameters in maximizing the upwind cross-ventilation flow in leeward school roof buildings

conditions (i.e., normal wind incidence and certain toughness as mentioned in Chapters 2-5) the most
Louver preparation

Buildings: Effect of upstream terrain roughness, roof shape, and window openings
Internal and external mean pressure coefficients of single-zone isolated low-and sawtooth roof
Perlin J. van Hooff, T. Naderi, C. Lefebvre, B. Blocken, B. Wind-tunnel measurements of

Published as:

Louver preparation, roof shape and window openings
of single-zone isolated low-and sawtooth roof buildings: effect of upstream
Wind-tunnel measurements of internal and external mean pressure coefficients

Appendix - Chapter 8
Abstract

Shape and Window Openings

Roof buildings: effect of upstream terrain roughness, roof pressure coefficients of single-zone isolated backward sawtooth

Wind tunnel measurements of internal and external mean
Section 4 presents the discussion and section 5 the conclusions. Where the main findings are summarized.

The conclusion chapter reviews the findings and discusses the implications. Section 6 discusses the implications for future research. Section 7 concludes with a summary of the main findings and suggests areas for further study.
Table 1 summarizes the nomenclature of the cases studied in the current paper. The cases are designated by a number corresponding to the nomenclature of the cases studied in the current paper.
\[
\left(\frac{\theta}{\theta + z}\right)\ln \frac{\theta}{\theta + z} = (z)\gamma
\]

In potrzebny way (Eq. 1):

\[\text{the wind tunnel test section, the measured wind velocity profile } U(z) \text{ is defined according to the wind tunnel test section and (2) Case BI}^2 \text{ with thickness constant with } 0 \text{ and 88 mm height, corresponding to the wind tunnel test section and (3) Case BI}^2 \text{ with thickness constant with } 0 \text{ and 88 mm height, corresponding to the wind tunnel test section.}
\]

Two unusual thickness contours were analysed: (1) Case BI without thickness change, the wind tunnel model was located between two parallel plates and one DANTEC hot wire anemometer was located on the middle of the wind tunnel leeward test section without at the same position where the reduced-speed \(U(\theta)\) wind was measured at the center of the mean wind speed \(U(\theta)\) and the streamwise turbulence intensity \(I_\theta\) were measured at

**Table 2 - Case applied in the open-channel BLWT**

<table>
<thead>
<tr>
<th>Case</th>
<th>Geometry</th>
<th>Comment</th>
<th>(opened)</th>
<th>(closed)</th>
<th>(opened)</th>
<th>(closed)</th>
</tr>
</thead>
<tbody>
<tr>
<td>BI1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BI2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BI3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BI4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BI5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Insert caption of Table 2 - Case applied in the open-channel BLWT.
Dissipation of wind energy at the test section of the wind tunnel is expressed by the equation:

\[ \frac{Q}{Q_{ref}} = \left( \frac{V}{V_{ref}} \right)^{n} \]

where:

- \( Q \) is the wind tunnel dissipation
- \( Q_{ref} \) is the reference dissipation
- \( V \) is the wind speed
- \( V_{ref} \) is the reference wind speed
- \( n \) is an exponent

The wind tunnel is a rectangular cross-section with a height of 5.7 m and a width of 2.5 m. The model of the building is a rectangular prism with a height of 5 m and a width of 2 m. The dissipation of wind energy in the tunnel is measured using a hot-wire anemometer and a dynamic head method.
The reduced scale model represents a blockage ratio of about 4% less than the height of 95% indicated by [25]. Reduced-scale model positioned in the BLT test section.

Figure 3 shows the pressure distribution in the semicircular pressure sensor. Figure 6 shows the pressure distribution in the reduced-scale test section. Figure 5 shows the pressure distribution in the test section. Figure 4 shows the pressure distribution in the reduced-scale test section. Figure 7 shows the pressure distribution in the test section. Figure 1 shows the pressure distribution in the reduced-scale test section. Figure 2 shows the pressure distribution in the test section.

Learned sawtooth root cross-section similarity and reduced-scale model.

Figure 1: See text for explanation of the sawtooth root cross-section similarity and reduced-scale model.

Figure 2: See text for explanation of the sawtooth root cross-section similarity and reduced-scale model.

Figure 3: See text for explanation of the sawtooth root cross-section similarity and reduced-scale model.

However, the pressure drop was found to be constant across the entire length of the test section. The pressure drop was found to be constant across the entire length of the test section. The pressure drop was found to be constant across the entire length of the test section. The pressure drop was found to be constant across the entire length of the test section. The pressure drop was found to be constant across the entire length of the test section.
400 m/s was selected for the whole pressure coefficient test. Setting speed of 15 m/s, 13 m/s, 10 m/s and 8 m/s. Finally, the velocity of 15 m/s (judging Reynolds number of...
X-ray cephalometric (Cp) methods allow the accurate superimposition of the different features of the study jaw on the postero-anterior (PA) view. This method is particularly useful for comparing the changes in skeletal and soft tissue structures that occur during growth and development. It provides a detailed analysis of the craniofacial morphology, allowing for the assessment of the relationship between the maxilla and mandible and their growth patterns. The cephalometric analysis is performed by tracing the outlines of the craniofacial structures on a standardized radiographic image and comparing them to a reference system. The resulting measurements are used to evaluate the growth potential and the treatment needs for orthodontic or surgical procedures. This method is crucial for planning and monitoring the treatment outcomes in orthodontics and craniofacial surgery.

8.3.1 External pressure coefficient (Ce) on a loaded building

8.3.1 Results

The external pressure coefficient (Ce) is calculated using the following equation:

\[ Ce = \frac{\Delta P}{\rho g h} \]

where \( \Delta P \) is the difference in pressure between the two sides of the building, \( \rho \) is the density of air, and \( g \) is the gravitational acceleration. The coefficient Ce is a dimensionless quantity that represents the ratio of the external pressure to the atmospheric pressure. It is typically used to assess the stability of structures under external forces such as wind or seismic loads.
In the absence of the protective return roughness (BL) on a building with opened windows,

(a) Schematic views of geometries C and D with closed openings (c) and

(b) Schematic views of geometries C and D with opened openings (c)
The pressure coefficient (\(C_{p}\)) increases from BL1 to BL2 by 10%, 30%, and 15% for Cases 1 and 2, respectively. The pressure coefficient (\(C_{p}\)) increases from BL1 to BL2 by 10%, 30%, and 15% for Cases 1 and 2, respectively. The pressure coefficient (\(C_{p}\)) increases from BL1 to BL2 by 10%, 30%, and 15% for Cases 1 and 2, respectively.

Impact around the outlet openings:

**Figure 9:** Internal mean pressure coefficient (\(C_{p}\)) of two located window and backwash (c and d) with the two openings.

**Figure 8:** Average internal mean pressure coefficient (\(C_{p}\)) with normal wind incidence angle for two BL on the two windows opened.
By 112.0% from case D to case B, C to D. The external pressure coefficient (C_e) increases by 104.9% from case D to B, and by 11.2% from case D to C. This increase is shown in Figure 1a.

Figure 1a shows the mean pressure coefficient around the opening (a), (c), and (d). Each figure represents the pressure distribution around the opening at different angles. The figures indicate that the pressure coefficient increases as the angle increases. For example, in Figure 1a, the pressure coefficient is highest at angle (d) and lowest at angle (c). This suggests that the pressure distribution is non-uniform around the opening.

Leaving out the building shape, the internal geometry of both cases (C and D) force the air to increase the air velocity when the outlet opening. The internal geometry of both cases (C and D) force the air to increase the air velocity when the outlet opening. The internal geometry of both cases (C and D) force the air to increase the air velocity when the outlet opening.
Discussion

The influence of the external pressure loss around initial openings, 1, 2, and the impact on the construction of the building in the wind tunnel test and the effect on the calculation of the parameters was a significant concern in the study. The calculation of the external pressure coefficient appears in a non-linear manner with an increase in the wind tunnel speed. The results of the study indicate that the external pressure coefficient is dependent on the wind tunnel speed and increases as the wind tunnel speed increases.

The decrease of the external pressure coefficient at low wind tunnel speeds is observed in the study, indicating a decrease in the external pressure coefficient with an increase in wind tunnel speed. The results suggest that the external pressure coefficient is a function of the wind tunnel speed and increases as the wind tunnel speed increases.
The external pressure coefficient ($C_p$) around the attic and around the outer opening is higher when the window of the heated/serviced roof building are opened for both geometries (c) and (d) and decreases around 25% under a higher boundary layer (BL) in both Cavity cases (c and d) compared to the lower boundary layer (BL1). The higher pressure coefficient becomes to be reduced to 151.8% in case BL1 by 155.6% in case BL2. The higher external pressure seems to be related to the increase in the volume of the heated area.

The implications of the boundary layer (BL) on the pressure coefficient appear in multiple wind cross-ventilation coefficient distribution around and inside a low-rise/low-sloped roof building (with 5.7 m height) and close to the outer opening would have a similar value as in both cases, the pressure rises are

The following conclusions may be drawn from this research:

Discharge coefficient of each building geometry.

The evaluation how pressure should be analyzed in both geometries once it depends on the pressure

The impact of wind influence on the pressure distribution are not presented but

Some pressure roofs led to work during some cases (i.e. 1.72 and 1.73 on geometry c) and 1.5 on geometry c)

More significantly, additional

L 1.5 on geometry (c)

The whole conclusions may be drawn from this research:

8.5 Conclusions
References

Acknowledgements
### Table 2 - External and internal mean pressure coefficients ($C_p$) of geometry C for six and two opening configuration

<table>
<thead>
<tr>
<th>Face</th>
<th>Right side wall</th>
<th>Left side wall</th>
<th>Windward</th>
<th>Roof</th>
<th>Roof</th>
<th>Outlet opening face</th>
<th>Leeward face</th>
<th>Leeward face</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>24</td>
<td></td>
<td></td>
<td>10</td>
<td>14</td>
<td>20</td>
<td>5</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>25</td>
<td></td>
<td></td>
<td>11</td>
<td>15</td>
<td>21</td>
<td>6</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>26</td>
<td></td>
<td></td>
<td>12</td>
<td>16</td>
<td>22</td>
<td>7</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>27</td>
<td></td>
<td></td>
<td>13</td>
<td>17</td>
<td>23</td>
<td>8</td>
<td>6</td>
</tr>
<tr>
<td></td>
<td>28</td>
<td></td>
<td></td>
<td>14</td>
<td>18</td>
<td>24</td>
<td>9</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>29</td>
<td></td>
<td></td>
<td>15</td>
<td>19</td>
<td>25</td>
<td>10</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td>30</td>
<td></td>
<td></td>
<td>16</td>
<td>20</td>
<td>26</td>
<td>11</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>31</td>
<td></td>
<td></td>
<td>17</td>
<td>21</td>
<td>27</td>
<td>12</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>32</td>
<td></td>
<td></td>
<td>18</td>
<td>22</td>
<td>28</td>
<td>13</td>
<td>11</td>
</tr>
<tr>
<td></td>
<td>33</td>
<td></td>
<td></td>
<td>19</td>
<td>23</td>
<td>29</td>
<td>14</td>
<td>12</td>
</tr>
<tr>
<td></td>
<td>34</td>
<td></td>
<td></td>
<td>20</td>
<td>24</td>
<td>30</td>
<td>15</td>
<td>13</td>
</tr>
</tbody>
</table>

*Pressure is with positive.*
### Pressure lip with suction

<table>
<thead>
<tr>
<th>Upper face</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>0°</td>
<td>0°</td>
<td>0°</td>
<td>0°</td>
<td>0°</td>
<td>0°</td>
<td>0°</td>
</tr>
<tr>
<td>10°</td>
<td>10°</td>
<td>10°</td>
<td>10°</td>
<td>10°</td>
<td>10°</td>
<td>10°</td>
<td>10°</td>
</tr>
<tr>
<td>20°</td>
<td>20°</td>
<td>20°</td>
<td>20°</td>
<td>20°</td>
<td>20°</td>
<td>20°</td>
<td>20°</td>
</tr>
<tr>
<td>30°</td>
<td>30°</td>
<td>30°</td>
<td>30°</td>
<td>30°</td>
<td>30°</td>
<td>30°</td>
<td>30°</td>
</tr>
<tr>
<td>40°</td>
<td>40°</td>
<td>40°</td>
<td>40°</td>
<td>40°</td>
<td>40°</td>
<td>40°</td>
<td>40°</td>
</tr>
<tr>
<td>50°</td>
<td>50°</td>
<td>50°</td>
<td>50°</td>
<td>50°</td>
<td>50°</td>
<td>50°</td>
<td>50°</td>
</tr>
<tr>
<td>60°</td>
<td>60°</td>
<td>60°</td>
<td>60°</td>
<td>60°</td>
<td>60°</td>
<td>60°</td>
<td>60°</td>
</tr>
<tr>
<td>70°</td>
<td>70°</td>
<td>70°</td>
<td>70°</td>
<td>70°</td>
<td>70°</td>
<td>70°</td>
<td>70°</td>
</tr>
<tr>
<td>80°</td>
<td>80°</td>
<td>80°</td>
<td>80°</td>
<td>80°</td>
<td>80°</td>
<td>80°</td>
<td>80°</td>
</tr>
<tr>
<td>90°</td>
<td>90°</td>
<td>90°</td>
<td>90°</td>
<td>90°</td>
<td>90°</td>
<td>90°</td>
<td>90°</td>
</tr>
</tbody>
</table>

### Externally and internally applied pressure coefficients (C) of geometry D for two fl and one output connection
Conference papers and two articles in professional journals.

Additional information: His research interests include the development of new materials for solar energy conversion and the application of advanced analytical techniques in materials science. He has also published three articles in the field of nanotechnology.

Biography