MASTER

Analysis of the effect of building geometry modifications on pedestrian-level wind speed

van Druenen, T.

Award date:
2016

Link to publication

Disclaimer
This document contains a student thesis (bachelor's or master's), as authored by a student at Eindhoven University of Technology. Student theses are made available in the TU/e repository upon obtaining the required degree. The grade received is not published on the document as presented in the repository. The required complexity or quality of research of student theses may vary by program, and the required minimum study period may vary in duration.

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
Analysis of the effect of building geometry modifications on pedestrian-level wind speed

Eindhoven University of Technology

Thijs van Druenen
0649217

21/03/2016

Chairman: Prof.dr.ir. B. Blocken
Daily Supervisor: dr.ir. T. van Hooff
Co-supervisor: dr.ir. H. Montazeri
ACKNOWLEDGEMENTS

In front of you lies my Graduation Report of the Master program Building Physics and Services at the department of the Built Environment at the Eindhoven University of Technology. I would like to thank Twan van Hooff for his help and co-supervision during this project. I would also like to thank Bert Blocken as Chairman of my graduation committee, Hamid Montazeri (co-supervisor) and all people who gave me feedback and advice during the project meetings.

Thijs van Druenen

Student Building Physics and Services

Eindhoven University of Technology

21st of March 2016
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>2</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>5</td>
</tr>
<tr>
<td>Abstract</td>
<td>6</td>
</tr>
<tr>
<td>Introduction</td>
<td>7</td>
</tr>
<tr>
<td>Research question</td>
<td>9</td>
</tr>
<tr>
<td>Methodology</td>
<td>9</td>
</tr>
<tr>
<td>Validation study</td>
<td>11</td>
</tr>
<tr>
<td>Validation criteria</td>
<td>11</td>
</tr>
<tr>
<td>Wind tunnel measurements</td>
<td>13</td>
</tr>
<tr>
<td>CFD</td>
<td>13</td>
</tr>
<tr>
<td>Steady RANS</td>
<td>14</td>
</tr>
<tr>
<td>Turbulence models</td>
<td>14</td>
</tr>
<tr>
<td>Computational settings and parameters</td>
<td>17</td>
</tr>
<tr>
<td>Computational domain and grid</td>
<td>17</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td>17</td>
</tr>
<tr>
<td>Other computational parameters</td>
<td>19</td>
</tr>
<tr>
<td>Results</td>
<td>20</td>
</tr>
<tr>
<td>Grid-sensitivity analysis</td>
<td>20</td>
</tr>
<tr>
<td>Validation study</td>
<td>23</td>
</tr>
<tr>
<td>Conclusion</td>
<td>29</td>
</tr>
<tr>
<td>Parameter study</td>
<td>30</td>
</tr>
<tr>
<td>Parameter study setup</td>
<td>30</td>
</tr>
<tr>
<td>Building type</td>
<td>30</td>
</tr>
<tr>
<td>Canopy</td>
<td>31</td>
</tr>
<tr>
<td>Podium</td>
<td>31</td>
</tr>
<tr>
<td>Permeable floor</td>
<td>31</td>
</tr>
<tr>
<td>Computational grid</td>
<td>32</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td>33</td>
</tr>
<tr>
<td>Other computational parameters</td>
<td>35</td>
</tr>
<tr>
<td>Grid-sensitivity analysis</td>
<td>36</td>
</tr>
<tr>
<td>Canopy/Podium</td>
<td>36</td>
</tr>
</tbody>
</table>
Results ........................................................................................................................................................................38

Canopy ............................................................................................................................................................................39
0° Wind direction ............................................................................................................................................................39
45° Wind direction ............................................................................................................................................................41
90° Wind direction ............................................................................................................................................................42
Average ............................................................................................................................................................................42

Podium ............................................................................................................................................................................43
0° Wind direction ............................................................................................................................................................43
45° Wind direction ............................................................................................................................................................45
90° Wind direction ............................................................................................................................................................46
Average ............................................................................................................................................................................46
Canopy vs. podium ............................................................................................................................................................47

Opening ............................................................................................................................................................................50
Ground floor .....................................................................................................................................................................51
First floor .........................................................................................................................................................................52
Second floor .....................................................................................................................................................................53
higher located floors ........................................................................................................................................................55
Wind velocity through the opening ................................................................................................................................55
Average ............................................................................................................................................................................56

Conclusion ........................................................................................................................................................................57

How can building geometry modifications improve wind comfort around high-rise buildings? .........................57
What will be the influence of different building geometry modifications applied to a high-rise building? ........57
Does the analysis allow a derivation of general guidelines? .......................................................................................59

Discussion .......................................................................................................................................................................60

Further research .............................................................................................................................................................60

References ......................................................................................................................................................................61
## NOMENCLATURE

### Table 1: Nomenclature

<table>
<thead>
<tr>
<th><strong>Roman symbols</strong></th>
<th><strong>Greek symbols</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_s$</td>
<td>$\alpha$</td>
</tr>
<tr>
<td>$H$</td>
<td>$\varepsilon$</td>
</tr>
<tr>
<td>$h$</td>
<td>$\kappa$</td>
</tr>
<tr>
<td>$k$</td>
<td>$\varphi$</td>
</tr>
<tr>
<td>$k_s$</td>
<td>$\omega$</td>
</tr>
<tr>
<td>$n$</td>
<td></td>
</tr>
<tr>
<td>$Re$</td>
<td></td>
</tr>
<tr>
<td>$t$</td>
<td></td>
</tr>
<tr>
<td>$U$</td>
<td></td>
</tr>
<tr>
<td>$U_h$</td>
<td></td>
</tr>
<tr>
<td>$U_{ref}$</td>
<td></td>
</tr>
<tr>
<td>$U_{var}$</td>
<td></td>
</tr>
<tr>
<td>$U_x$</td>
<td></td>
</tr>
<tr>
<td>$U_y$</td>
<td></td>
</tr>
<tr>
<td>$U_z$</td>
<td></td>
</tr>
<tr>
<td>$U_{ABL}$</td>
<td></td>
</tr>
<tr>
<td>$y_0$</td>
<td></td>
</tr>
<tr>
<td>$y_p$</td>
<td></td>
</tr>
<tr>
<td>$y_{ref}$</td>
<td></td>
</tr>
<tr>
<td>$x, y, z$</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td></td>
</tr>
<tr>
<td>F</td>
<td></td>
</tr>
</tbody>
</table>
ABSTRACT

The wind speed at pedestrian level can be significantly increased by high-rise buildings, which can lead to uncomfortable and even dangerous conditions. Therefore, knowledge of building aerodynamics is of significant importance in the design of a building. Yet, most designers have little notion of the relation between urban geometry and pedestrian comfort, and are not aware of how the pedestrian-level wind environment can be affected by geometrical building modifications.

This study tries to limit the wind velocity in the standing vortex and corner streams around a high-rise building. For pedestrians, these are the two most important wind flow zones. The following building geometry modifications, and their effect on the pedestrian-level wind environment, are investigated:

- The use of a canopy around the building.
- The application of a podium to the base of the building.
- The introduction of a permeable floor to the building.

To investigate the influence of the proposed building geometry modifications a parameter study is carried out. Using Computational Fluid Dynamics (CFD), the wind flows around different building configurations are simulated and the effects on the pedestrian wind environment are analysed. Grid-sensitivity analyses are performed and sub-configuration validation is applied using existing wind tunnel measurements.

The highest wind speeds at pedestrian level are found through passages and in the corner streams. These corner streams partly result from the wind flowing downwards in front of the windward façade. By limiting the amount of wind flow that reaches this area, the wind speed in the corner streams can also be reduced. The building geometry modifications investigated in this study are focused on the reduction of these downward wind streams. It is found that especially the application of a canopy or a podium to the building can significantly reduce the average and maximum occurring wind speeds around a single high-rise building. In general, the observed velocities decrease with increasing canopy or podium size. The introduction of a permeable floor to the building can also reduce the maximum and average wind speed. The lowest average wind speed on pedestrian level are seen when a building layer halfway the building is removed while the removal of lower floors decreases the maximum occurring wind speed. However, when building layers too low are removed, adverse effects are noted. Moreover, the removal of the ground floor, as well as passages through buildings, must be avoided. This study resulted in guidelines for the application of the mentioned building geometry modifications. However, different buildings in different urban environments ask for different solutions. Therefore wind comfort studies will still be necessary to investigate the effect of a high-rise building design on the pedestrian wind environment.
INTRODUCTION

The wind speed at pedestrian level can be significantly increased by high-rise buildings. This can lead to uncomfortable and even dangerous conditions. Examples are given by Wise (1970) who reports about discouraged shoppers due to a windy environment, leaving shops unattended, and by Lawson and Penwarden (1975). They report the death of two old ladies because of an unfortunate fall caused by high wind speeds at the base of a high-rise building.

The wind speed at pedestrian level results from the wind flow pattern around a building. Figure 1 provides a schematic view of a wind flow pattern around a high-rise building. The approaching wind flow diverges over and around the building. A stagnation point is created at approximately 70% of the building height, which causes a considerable amount of air to flow downwards indicated by number 5. This results in a so-called standing vortex at ground level. The standing vortex is indicated by number 6 and by sweeping around the building corners, flow separation causes the creation of high wind speed corner streams. These are visualized by number 8 in Figure 1. Together with the standing vortex, these corner streams are the highest wind speed zones at pedestrian level for a single high-rise building (Blocken & Carmeliet, 2004).

Blocken and Carmeliet (2004) described the importance of knowledge of building aerodynamics in the design of a building. Especially the outdoor wind environment around buildings has received relatively little attention in the Building Physics community, while the consequences of an unfavourable pedestrian wind environment near high-rise buildings can be detrimental to the success of new buildings. When it comes to the design of high-rise buildings, previously named authors recommend the avoidance of building entrances, walkways and bicycle routes near corners. In addition, passages through buildings and passage-ways that are led through narrow passages between buildings should be avoided (see Fig. 2) as well as the creation of recreational areas around high-rise buildings unless specific attention is given to the design of these features (Beranek & van Koten, 1982). By combining wind tunnel experiments with literature information, Stathopoulos and Wu (1995) established generic models and empirical relations for the wind conditions over streets. For example, when a tall building is surrounded by uniform blocks, the maximum speed amplification is a function of the height difference of the tall building and the surroundings, the measurement height and the blockage ratio of surrounding blocks.

Figure 1: Schematic illustration of the wind flow pattern around a single wide high-rise building slab (Beranek & van Koten, 1982).

Figure 2: Schematic illustration of increased wind velocities in passage-ways and passages through buildings (Beranek & van Koten, 1982)
Although some urban authorities only grant a building permit for a new high-rise building after a wind comfort study has indicated that the consequences for the pedestrian wind environment remain limited (Blocken & Carmeliet, 2008), it is stated that most designers have little notion of the relation between urban geometry and pedestrian comfort (Willemsen and Wisse, 2007). The perception of urban space by the public is not always correctly understood. Sociological, cultural and psychological influences are of importance as well as exposure to rain, sunshine and wind. Moreover, alterations or additions in favour of the pedestrian wind comfort are found to be ugly, unpractical, expensive and their effects are rarely investigated (Willemsen and Wisse, 2007).

There are some wind tunnel studies, which investigated the effect of geometrical modifications of the building on the pedestrian wind environment. Uematsu et al. (1992) carried out a series of wind tunnel experiments on the effects of the corner shape of high-rise buildings on the pedestrian-level wind environment around them. It was found that the wind environment can be improved by slightly changing the corner shape, especially for a wind angle of 0°. The effect of the corner shapes becomes smaller as the wind angle is increased from 0° to 45°. Jamieson et al. (1992) carried out wind tunnel experiments of a representative city model including different building designs, while Beranek & van Koten (1982) also investigated building geometry modifications on single high-rise buildings. These designs included the application of a canopy and a podium. It was found that there is a beneficial effect on the wind conditions directly beneath the canopy, in front and around the frontal corners of the building. As can be seen in Figure 3 the downward wind streams are deviated from the building, moving the areas of wind discomfort away from the corners. An enclosure on top of the canopy had a positive influence. The use of a podium reduced the extreme speeds at the base of a large building (Fig. 4). However, the size of the areas of wind discomfort increased. Enlarging the offset size of the podium did not result in more positive results. Another possible solution, investigated by Lam (1992), is the introduction of a permeable floor at an intermediate level. This would provide a bleed path for the upper-level winds to pass through the building before they are brought down to the ground level by the windward façade of the building. With the use of wind tunnel measurements it was shown that the introduction of a permeable floor reduces the spatial extent where the high-rise building could bring about unpleasant windy conditions in the corner streams on ground level. It does not, however, lower the peak values of wind speeds that are occurring.

The previous studies confirm that geometrical modifications can improve the pedestrian wind environment. Yet for designers, it is often unclear how these modifications have to be applied and what the consequences are. This study elaborates further on this previous research. After investigating different building geometry modifications and their effect on the pedestrian wind environment, an attempt is made to derive guidelines from these results.

![Figure 3: Schematic illustration of wind flow around a building with a canopy (Beranek & van Koten, 1982)](image1)

![Figure 4: Schematic illustration of wind flow around a building with a podium (Beranek & van Koten, 1982)](image2)
The following research questions are answered in this investigation:

- How can building geometry modifications improve pedestrian-level wind comfort around high-rise buildings?
- What will be the influence of the following building geometry modifications applied to a high-rise building?
  - Canopy
  - Podium
  - Permeable floor
- Does the analysis allow a derivation of general guidelines for the application of canopies, podiums and building openings for different building configurations?

**METHODOLOGY**

To investigate the influence of the proposed building geometry modifications a parameter study is carried out. Using Computational Fluid Dynamics (CFD), wind flows around different building configurations are simulated and the effects on the pedestrian wind environment are analysed. To be able to rely on the CFD results, the simulations need to be validated by measurements. The methodology used in this study is based on the framework described by Blocken et al. (2012).

The process starts with sub-configuration validation, Figure 5. This implies the decomposition of the actual building geometry into a simpler building configuration which exhibits similar features as the flow around the actual building configuration (Blocken & Carmeliet, 2008). For generic sub-configurations, several high-quality experimental data sets are available which are then used for CFD validation. In this study, a data set by the Architectural Institute of Japan is used (Chapter 3.2), including a comparable building with the one used in this study (Chapter 4.1.1). Next, a grid-sensitivity analysis is performed to reduce the discretization errors and the computation time. When the grid resolution is satisfactory, the validation test case is computed using several different turbulence models. Validation is performed by comparing the results with the available measurements to determine their validity.

![Figure 5: Methodology – Sub-configuration validation](image-url)
When the sub-configuration is validated, the grids required for the parameter study are constructed and a grid-sensitivity analysis is performed (Figure 6). Because the canopy and podium solution can use the same grid, only two grids are created and two grid-sensitivity studies are performed.

When the grid-sensitivity analysis is successful, the parameter study is performed. Different building configurations are tested, Figure 7, and the effect on the pedestrian wind environment is analysed. The canopy/podium study starts with the investigation of a base case situation. Next, the offset of the applied building geometry modification is extended to half the height of the building (0.5H). The investigation of the opening through the building also starts with the base case situation. Subsequently, the floors shown in Figure 7 are removed from the building. Each building configuration is simulated with three different wind directions. Finally, the results are analysed and possible guidelines are proposed for the application of the measures.
Traditionally, the studies of wind environment around buildings are performed in a wind tunnel, since it is the most well-established way to simulate the natural wind. In addition to wind tunnel testing, CFD has been applied in many industries and its application of studying street-level winds has been growing for several decades (Hu & Wang, 2005). In the past, CFD has been frequently used to study wind speed conditions at pedestrian level in urban areas (e.g. Murakami et al., 1990; Stathopoulos & Baskaran, 1990; Gadihi et al., 1993; He & Song, 1999; Yoshie et al., 2007; Blocken et al., 2007, 2008, 2014; Janssen et al., 2013; Montazeri et al., 2013)

Generally, wind tunnel measurements are only performed at a few selected points in the model. CFD, on the other hand, provides data on the relevant parameters in all points of the computational domain. Compared to wind tunnel testing, CFD has some more important advantages which are important in this study. CFD simulations easily allow parametric studies to evaluate alternative design configurations, especially when the different configurations are all embedded within the same computational model (van Hooff & Blocken, 2010).

The establishment of Best-Practice Guidelines concerning the use of CFD has been an important step towards more accurate and reliable CFD simulations. Several of these guidelines have been developed specifically to support the evaluation of pedestrian-level wind conditions (Franke et al., 2004; Blocken et al., 2007; Tominaga et al., 2008).

These guidelines describe that whenever possible, on-site measurements should be made to provide validation data for the CFD simulations of the existing configuration. If such measurements are not possible, which is the case in this generic study, an alternative validation procedure called “sub-configuration validation” can be employed. It consists of subdividing the actual configuration into a number of generic sub-configurations, each of which contains one or several of the main physical flow features that occur in the actual configuration (e.g. Oberkampf et al., 2004; Franke et al., 2007; Blocken and Carmeliet, 2008). When a given combination of computational parameters and settings provides accurate simulation results for each of the sub-configurations, it can reasonable be assumed that the same will also provide accurate results for the actual configuration (Blocken et al., 2012). The actual building configuration used in this study includes the proportions 4:4:1 (H:W:D). For the sub-configuration validation a building with the proportions 2:1:1 is used.

At the base of high-rise buildings, uncomfortable and even dangerous conditions may occur. Particularly these negative conditions result from high wind velocities in the corner streams around a building. The velocities within and the size of these corner stream are highly dependent from the building’s dimensions and geometry. In general, larger buildings lead to a greater change of negative wind effects. Such effects are for example cooling of the body, flapping of clothes, hairs or a newspaper one is trying to read, flying dust and wind gusts requiring extra efforts to walk against the wind or to keep balance (Bottema, 2000). Bottema (2000) defines pedestrian discomfort as:

“Pedestrian discomfort occurs when wind effects become so strong and occur so frequently, that people experiencing those wind effects will start to feel annoyed, and eventually will act in order to avoid these effects.”

By this definition, the occurrence of wind effects does not necessarily imply discomfort. The emphasis in this study, as is conventional in the wind engineering community (Metje et al., 2008), will be on mechanical wind effects. Thermal wind effects, which can have both positive or negative effects depending on the climate and weather conditions, will not be considered.
In the Netherlands, NEN 8100 (NEN, 2006) describes a standard for wind comfort. The standard states that in general, comfortable conditions cannot always be met and uncomfortable conditions must be accepted for a certain percentage of time. Discomfort probability and danger probability are defined as the percentage of hours (during a year) in which the thresholds are exceeded, which is 5 m/s at pedestrian level. Five grades of wind comfort A-E are defined as function of this probability. In addition, for three different activities of the public (traversing, strolling and sitting) these grades of wind comfort are defined in terms of a poor, moderate or good local wind climate (Table 2) (Willemsen & Wisse, 2007). The safety criterion has a threshold of 15 m/s and a maximum allowed exceedance probability of 0.3% (Table 3).

<table>
<thead>
<tr>
<th>P(U_THR &gt; 5 m/s) (in % hours per year)</th>
<th>Grade</th>
<th>Activity</th>
<th>Traversing</th>
<th>Strolling</th>
<th>Sitting</th>
</tr>
</thead>
<tbody>
<tr>
<td>&lt; 2.5</td>
<td>A</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
<td>Good</td>
</tr>
<tr>
<td>2.5 – 5.0</td>
<td>B</td>
<td>Good</td>
<td>Good</td>
<td>Moderate</td>
<td>Moderate</td>
</tr>
<tr>
<td>5.0 – 10</td>
<td>C</td>
<td>Good</td>
<td>Moderate</td>
<td>Poor</td>
<td>Poor</td>
</tr>
<tr>
<td>10 – 20</td>
<td>D</td>
<td>Moderate</td>
<td>Poor</td>
<td>Poor</td>
<td>Poor</td>
</tr>
<tr>
<td>&gt; 20</td>
<td>E</td>
<td>Poor</td>
<td>Poor</td>
<td>Poor</td>
<td>Poor</td>
</tr>
</tbody>
</table>

Table 2: Criteria for wind nuisance according to NEN 8100 (2006)

<table>
<thead>
<tr>
<th>P(U_THR &gt; 15 m/s) (in % hours per year)</th>
<th>Grade</th>
<th>Activity</th>
<th>Traversing</th>
<th>Strolling</th>
<th>Sitting</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05 – 0.3</td>
<td>Limited risk</td>
<td>Acceptable</td>
<td>Not acceptable</td>
<td>Not acceptable</td>
<td>Not acceptable</td>
</tr>
<tr>
<td>≥ 0.3</td>
<td>Dangerous</td>
<td>Not acceptable</td>
<td>Not acceptable</td>
<td>Not acceptable</td>
<td>Not acceptable</td>
</tr>
</tbody>
</table>

Table 3: Criteria for wind danger according to NEN 8100 (2006)

According to the above criteria for wind nuisance, the level of mechanical wind discomfort depends on the velocity of the wind, rather than other parameters like pressure and turbulent kinetic energy. Therefore, the effect of the different building configurations tested in this study on the pedestrian wind environment are assessed based on the wind velocity. Because this is a generic study in which different building configurations are compared with each other, with a regular approaching wind flow, the results are not assessed over time but in a steady state. Finally, due to the fact that only an isolated building is studied, the wind velocities are expected to be high for each case. Therefore, the wind criteria according to NEN 8100 (2006) are not applied in this study.
For generic sub-configurations validation, several high-quality experimental data sets are available which can be used for CFD validation. In this study, a data set by the Architectural Institute of Japan is used (Tanaka et al., 2006). The used wind tunnel experiments are actually performed for the validation of a CFD model for the prediction of pollution levels in urban locations. It provides data such as average wind velocity, temperature and concentration, various turbulence intensity data, shear stresses, heat fluxes, concentration fluxes etc., and is therefore suitable for the current investigation. Note that only velocities will be used in this validation study.

The experiment was conducted in the wind tunnel (cross section at measurement part: 1.2 m x 1.0 m) of Tokyo Polytechnic University. The model building has a height (H) of 0.2 m, a width (W) of 0.1 m, and a depth (D) of 0.1 m (H:W:D = 2:1:1) and was located in a turbulent boundary layer. The inflow condition can be described with a power-law profile with a power index of about 0.25. Measurement points were placed at four planes as shown in Figure 8. The wind velocity is measured using a split film probe. The sampling frequency was set at 1,000 Hz, to obtain 120,000 data values in 120 seconds. Figure 9 gives a schematic illustration of the wind tunnel study.

Figure 9: Schematic illustration of wind tunnel study.

CFD

Computational Fluid Dynamics (CFD) uses numerical analysis and algorithms to solve problems involving fluid flows. The basis of CFD modelling is formed by the Navier-Stokes equations, the fundamental equations of fluid dynamics. In addition, the indoor and outdoor flow is generally turbulent. During the process of CFD modelling, one of the important decisions is to select the appropriate turbulence models which are solved to describe the turbulent flow. Three modelling approaches are developed, which are Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS). DNS solves the Navier-Stokes equations directly down to the smallest length and time scales without implementing any assumptions or sub-models and is therefore extremely time consuming and limited to be applied to relatively low Reynolds numbers. The Reynolds number is defined as the ratio of momentum forces to viscous forces. Laminar flow occurs at low Reynolds numbers where viscous forces are dominant, and is characterised by smooth, constant fluid motion. Turbulent flow occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices and other flow instabilities. Because the outdoor wind flow around a building is turbulent, and due to its extreme time consumption, DNS is not suitable for the current investigation. LES uses a filter on the Navier-Stokes equation, in which eddies larger than a certain filter size are directly resolved while the effect of the eddies smaller than the filter size is approximately modelled by subgrid modelling. However, LES is still too computationally expensive to be used in a parameter study, which requires a considerable amount
of simulations. RANS resolves only the mean flow and the turbulence scales are modelled. RANS is the most widely applied modelling method and generally the CFD simulations in comparable cases are isothermal RANS simulations (Blocken et al., 2004, 2012; Blocken & Carmeliet, 2008; van Hooff & Blocken, 2010). Isothermal simulations are justified because the criteria for wind comfort and safety generally refer to high wind speed conditions, in which mechanical turbulence generation dominates and thermal effects are small or absent (Blocken et al., 2012).

### STEADY RANS

#### 3.3.1

Steady RANS is the time average leading to a statistically steady description of the turbulent flow and is the method that has been most widely applied and validated in the field of numerical computation of wind flow around buildings and air flow inside building (Blocken, 2012). However, due to the unsteady meteorology it is questionable whether the resulting equations, eliminated from the time dimension, are a useful model. However, for a comparison with wind tunnel experiments these RANS equations are an adequate representation of the wind tunnel’s reality as the time averaged approach flow conditions of the tunnel do not change (Franke et al., 2007). Moreover, time averaged results provide good conditions for the comparison of the different building configurations as this leads to well comparable results.

The RANS equations are obtained by decomposing the solution variables as they appear in the exact Navier-Stokes equations into a time-averaged and a fluctuating component. The decomposition process generates additional unknown called Reynolds stresses which represent the influence of turbulence on the mean flow. The Reynolds-averaged Navier-Stokes equations do not form a closed set of equations due to the presence of these Reynolds stresses. Closure must therefore be obtained by modelling the Reynolds stresses. The modelling approximations are called turbulence models. Requirements for turbulence models for use in general-purpose CFD codes are: wide applicability, accuracy and economy (Blocken, 2012).

### TURBULENCE MODELS

#### 3.3.2

The most common turbulence models for the RANS equations are the k-ε model and the k-ω model (two-equation models), the Reynolds Stress equation model (RSM) and the Algebraic Stress Model (ASM). In wind engineering, the k-ε turbulence model has most often been used (Blocken, 2012).

The Best Practice Guidelines (Franke et al., 2007) give no exact advice for the choice of the turbulence model. However, a validation strategy is proposed to evaluate the performance of the different turbulence models. The validation test cases should be computed with several different turbulence models for the Reynolds stresses. The results should be compared with available measurements of velocities around buildings. In the absence of reliable measurement data, no direct validation can be made. In such situations, it is recommended to look for simpler test cases with reliable reference data (Oberkampf et al., 2004). The test cases to be chosen must include the critical basic features of the actual problem of interest and is described later earlier this report (Chapter 3.2). Four different turbulence models are tested, these are the Realizable k-ε turbulence model, the RNG k-ε turbulence model, the Shear-Stress Transport (SST) k-ω turbulence model and the Reynolds Stress Model (RSM).
The main deficiencies of the tested turbulence models mainly result from the inaccurate prediction of turbulent kinetic energy levels (Blocken, 2012):

1. The models generally overestimate the production of turbulent kinetic energy $k$ around the frontal (and the side) corners, yielding excessive levels of turbulent diffusion. As a result, it typically underestimates the separation and recirculation regions (Fig. 10) that occur over the windward roof and the sides or, in the worst case, does not predict them at all.

2. The models underestimate the value of $k$ in the recirculation region behind the building and therefore typically overestimate the velocity in and the size of the recirculation vortex.

![Figure 10: Schematic illustration of the wind-flow pattern over a building including the regions where the standard k-ε model shows deficiencies (Blocken, 2012).](image)

### K-EPSILON TURBULENCE MODEL

#### STANDARD

The standard k-ε model was developed in the early 1970s. Its strengths as well as its shortcomings are well documented. The most important advantages over other models are the fact that it is the most widely used and validated turbulence model, the fact that it has a good performance for many industrially relevant flows and the fact that it is the simplest turbulence model for which only initial and/or boundary conditions need to be supplied. However, next to the already mentioned deficiencies it performs poorly for complex flows involving separation and strong streamline curvature (Blocken, 2012).

#### REALIZABLE

The previously named flaws of the standard k-ε model are mainly due to the equation of the dissipation rate ($\varepsilon$). The Realizable k-ε model, proposed by Shih et al. (1995), shares the same turbulent kinetic energy equation as the standard k-ε model, but includes an improved equation for $\varepsilon$. Compared to the standard k-ε model, the Realizable k-ε model has improvements on the prediction for rotation, recirculation and boundary layer under strong adverse pressure gradients or separation (ANSYS Fluent Theory Guide, 2013).
The RNG k-ε model was derived using a statistical technique called renormalization group theory. It is similar in form to the standard k-ε model, but includes an additional term in the equation of ε for interaction between turbulence dissipation and mean shear. Also, the effect of swirl on turbulence is taken into account. In contrast to the standard k-ε model, the RNG k-ε model uses a differential formula for effective viscosity. RNG has improvements on prediction for separated flow, wall heat and mass transfer (ANSYS Fluent Theory Guide, 2013).

Another two-equation model is the k-ω model. In this model, the specific dissipation rate (ω) is an inverse time scale that is associated with the turbulence, and can also be thought of as the ratio of ε to k. Its numerical behaviour is similar to that of the k-ε models (ANSYS Fluent Theory Guide, 2013).

The shear-stress transport (SST) k-ω model, developed by Menter (1994), is an eddy-viscosity model which main novelty is its combination of using a k-ω model in the inner boundary layer and the use of a k-ε model in the outer region and outside of the boundary layer. The switch between the two models is controlled by blending functions. (ANSYS Fluent Theory Guide, 2013).

The Reynolds Stress Models (RSM) closes the Reynolds-Averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the dissipation rate. This results in five additional transport equations in 2D flows and seven additional transport equations in 3D. Compared to two equation models, the RSM has a greater potential to give accurate prediction for complex flows. A major drawback of the RSM is its slow convergence. On top of that, the additional computational expense might not always lead to superior results compared to two equation models (ANSYS Fluent Theory Guide, 2013).
COMPUTATIONAL SETTINGS AND PARAMETERS

3.4

COMPUTATIONAL DOMAIN AND GRID

Because the simulations are to be compared with boundary layer wind tunnel measurements, it is recommended to use the cross section of the wind tunnel’s test section for the computational domain, in this case a height of 1 meter (5H) and a width of 1.2 meters (6H). In this way the computational model accurately replicates the geometry of the wind tunnel test section (Franke et al., 2007). The distance from the model to the inlet of the domain, 0.4 meters (2H), also represents the wind tunnel experiments. The extension of the computation model behind the building is 2 meters (10H) and is based on a comparable study (Yoshie et al., 2011).

The grid was constructed using the grid-generation technique presented by van Hooff & Blocken (2010), which allows a large degree of control over the quality of the grid and its individual cells. It consists of only hexahedral cells. The use of hexahedral cells avoids the well-known convergence problems that are associated with hybrid grids with tetrahedral cells, especially when the required second-order discretization schemes are used. This advantage is considered very important, as first-order schemes should not be used due to their excessive contribution to numerical diffusion. This corresponds to the Best Practice Guidelines by Franke et al. (2004), which discommends using first-order discretization schemes. The computational grid consists of 557,840 cells. The grid is based on a grid-sensitivity analysis, outlined in Chapter 3.5.1.

BOUNDARY CONDITIONS

3.4.2

In order to achieve a good agreement between CFD and measured flow fields, the approach flow in CFD has to match the measured approach flow profiles. The atmospheric boundary layer inflow at the inlet of the domain consists of the profiles of mean wind speed, turbulent kinetic energy and turbulence dissipation rate. To describe the mean wind speed profile, mainly the logarithmic law (Equation 1) and the power-law (Equation 2) are used.

\[ U(y) = \frac{u^*_{ABL}}{\kappa} \ln \left( \frac{y + y_0}{y_0} \right) \]  

(Eq. 1)

\[ \frac{U(y)}{U_{ref}} = \left( \frac{y}{y_{ref}} \right)^{\alpha_p} \]  

(Eq. 2)

With \( U(y) \) the mean wind speed at height \( y \), \( u^*_{ABL} \) the friction velocity, \( \kappa \) the Von Karman constant and \( y_0 \) the aerodynamic roughness length. In Equation 2, \( U_{ref} \) is a reference wind speed at height \( y_{ref} \) and \( \alpha_p \) is the power-law exponent which expresses the roughness of the terrain. The logarithmic law has a physical background and results from a theoretical derivation, while the power-law is an empirical law in which the constant \( \alpha_p \) is determined from measurements. Currently, in the field of wind engineering the logarithmic law is slightly preferred due to its physical background and because the power-law does not describe the flow near the ground very well (Blocken et al., 2012).
In the wind tunnel experiment the approaching flow was modelled with a power-law profile with an exponent of \( \alpha = 0.25 \). The reference wind speed is 4.2 m/s at a height of 0.2 m \((y/H = 1)\). Incident measurement data are available. A comparison of these data with corresponding logarithmic law and power-law mean wind speed profiles can be found in Figure 11. The mean wind speed profile prescribed by the logarithmic law is defined with \( y_0 = 0.005 \) meter. The reference wind speed is 4.2 m/s at 0.2 m \((y/H = 1)\). In general, the measurements show a better agreement with the logarithmic law. For this reason the mean wind speed profile used in the validation study is prescribed by the logarithmic law.

The profiles of turbulent kinetic energy \( k \) are created from the measurement data from the wind tunnel (Appendix 1.2.3). ANSYS Fluent will interpolate between these data points to obtain values over the entire height at the boundary faces. The inlet profiles of the turbulence dissipation rate \( \varepsilon \) \((k-\varepsilon \) turbulence model) and the specific dissipation rate \( \omega \) \((k-\omega \) turbulence model) are calculated using Equations 3 and 4, respectively.

\[
\varepsilon(y) = \frac{u^*_{ABL}^3}{k(y + y_0)} \\
\omega = \varepsilon \frac{k C_\mu}{\varepsilon U_{ref}} \\
u^*_{ABL} = \kappa \frac{U_{ref}}{ln(y + y_0/y_0)}
\]

Where \( y \) is the height co-ordinate, \( u^*_{ABL} \) the friction velocity, \( \kappa \) the von Karman constant \((0.42)\) and \( C_\mu \) a model constant \((0.09)\) (Blocken & Carmeliet, 2008). In Equation 5, \( U_{ref} \) is the reference wind speed at height \( y \), where \( y_0 \) is the aerodynamic roughness length.

At the outlet, previously named turbulence profiles are used and zero static pressure is specified. At the walls, the standard wall functions by Launder and Spalding (1974) with the sand-grain roughness modification by Cebeci and Bradshaw (1977) are used. With the use of ‘wall functions’ empirical information is added to the bottom of the domain. Wall functions are used to avoid the inability of turbulence models to provide flow predictions in the near-wall region. The wall functions provide the boundary conditions for the RANS equations and the transport equation for \( k \).

No extra roughness was imposed on the bottom of the wind tunnel floor. Therefore, similar to the wind tunnel experiment, no roughness is imposed on the bottom of the domain resulting in a roughness height of 0 m and a default roughness constant of 0.5. The same conditions have been applied at the building surfaces, and at the top and lateral boundaries. Because the computations are compared with wind tunnel measurements, the top and lateral boundaries are also treated as solid walls.
In case of a homogeneous equilibrium boundary layer flow, the prescribed profiles should not change until the built area is reached. Therefore, for every CFD simulation it is advisable to assess the extent of horizontal inhomogeneity by a simulation in an empty computational domain prior to the actual simulation with the building model present. Sensitivity tests in an empty computational domain are of critical importance. In addition, it is of importance to always report not only the inlet profiles but also the incident flow profiles obtained from the simulation in the empty domain because they characterize the real flow to which the building model is subjected (Blocken et al., 2007).

The results are found on the right in Figures 12, 13 & 14. Wind speed ratios are defined by dividing the local velocity in x-direction by the reference speed ($U_{ref}$), which is 4.2 m/s at building height $H$. Note that simulations are performed with both the Realizable k-ε turbulence model and the SST k-ω turbulence model. However, because the results of both simulations were practically similar, only the results with respect to the Realizable k-ε turbulence model are shown. Due to the fact that there is no roughness imposed on the bottom of the wind tunnel/computational domain it was expected that the wind flow would accelerate in the lower regions. Figure 14 confirms this expectation, although it can be seen that the measured wind speed accelerates slightly more than the wind speed belonging to the simulations. Because the roughness height of the bottom wall is already set to 0 m, this difference is taken as it is.

OTHER COMPUTATIONAL PARAMETERS

3.4.3

The CFD simulations are performed using the commercial CFD code ANSYS Fluent 15.0.7 and using the 3D steady RANS equations. Four different turbulence models are tested. Pressure velocity-coupling is taken care of by the SIMPLE algorithm. Pressure interpolation is second order. Second-order discretization schemes are used for both the convection terms and viscous terms of the governing equations. The iterations were terminated when the scaled residuals did not show any further reduction with increasing number of iterations. The minimum values that were reached are found in Appendix 6. An overview of the used settings can be found in Appendix 1.
GRID-SENSITIVITY ANALYSIS 3.5.1

A grid-sensitivity analysis is performed to reduce the discretization errors and the computation time. The resolution of a grid is fine enough if a finer grid produces comparable results.

For the grid-sensitivity analysis, five grids are constructed which are shown in Figure 15. Grid sizes are found in Table 4. Coarsening and refining was performed with an overall linear factor $\sqrt{2}$. An overview of the used simulation settings is shown in Appendix 1.1. Wind speed ratios are defined by dividing the local velocity in x-direction by the reference speed ($U_{\text{ref}} = 4.2$ m/s). The results are compared on 4 planes for the points displayed in Figure 16.

Table 4: Amount of cells of the grids used in the grid-sensitivity analysis

<table>
<thead>
<tr>
<th>Grid</th>
<th>Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarser</td>
<td>25,540</td>
</tr>
<tr>
<td>Coarse</td>
<td>70,896</td>
</tr>
<tr>
<td>Medium</td>
<td>201,328</td>
</tr>
<tr>
<td>Fine</td>
<td>557,840</td>
</tr>
<tr>
<td>Finer</td>
<td>1,610,624</td>
</tr>
</tbody>
</table>

Figure 15: Grids used in grid-sensitivity analysis; (a) coarser grid; (b) coarse grid; (c) medium grid; (d) fine grid; (e) finer grid.

Figure 16: points for comparison in (a) the horizontal plane $y/H = 0.0625$; (b) the vertical plane $z/H = 0$; (c) points for comparison in the vertical plane $x/H = 0.25$; (d) points for comparison in the vertical plane $x/H = 0.5$. 
The results of the simulations are compared with each other by calculating the Root Mean Squared Error (RMSE). The RMSE is the square root of the average of the square of all the errors (Equation 6).

$$\text{RMSE} = \sqrt{\frac{1}{n} \sum_{i=1}^{n} (y_i - \bar{y}_i)^2} \quad \text{(Eq. 6)}$$

With $y_i$ and $\bar{y}_i$ the wind speed ratios at equal points in the two grids to be compared. Table 5 shows the errors of the comparison between the results from the Coarse and the Coarsest grid. It can be seen that the total RMSE is 0.0458. The minimum errors are found on the xy-plane and the zy-plane ($x=0.25H$). The highest error is calculated on the xz-plane.

Table 5: RMSE - Coarsest vs. Coarse grid

<table>
<thead>
<tr>
<th>Plane</th>
<th>RMSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>xz-plane ($y = 0.0625H$)</td>
<td>0.1073</td>
</tr>
<tr>
<td>xy-plane ($z = 0.0H$)</td>
<td>0.0237</td>
</tr>
<tr>
<td>zy-plane ($x = 0.25H$)</td>
<td>0.0229</td>
</tr>
<tr>
<td>zy-plane ($x = 0.5H$)</td>
<td>0.0895</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>0.0458</strong></td>
</tr>
</tbody>
</table>

Figure 17 shows discrepancies between the results obtained on the coarsest and coarse grid, especially in the xz-plane. Also, some measurement points on the xy-plane show significantly different results between the two grids.

Table 6 shows the RMSE calculated between the medium and coarse grid. Here the total RMSE is reduced to 0.0086. Figure 18 displays the wind speed ratios calculated on the medium and on the coarse grid. The results obtained on the coarse grid are pretty well in agreement with those belonging to medium grid. In contrast with the previous comparison the deviations in the xz-plane are significantly reduced. However, there are some differences in the xy-plane including high wind velocity values. Because the higher wind velocity values are most important in this study, the coarse grid does not provide totally satisfying results.

Table 6: RMSE - Medium vs. Coarse grid

<table>
<thead>
<tr>
<th>Plane</th>
<th>RMSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>xz-plane ($y = 0.0625H$)</td>
<td>0.0144</td>
</tr>
<tr>
<td>xy-plane ($z = 0.0H$)</td>
<td>0.0095</td>
</tr>
<tr>
<td>zy-plane ($x = 0.25H$)</td>
<td>0.0060</td>
</tr>
<tr>
<td>zy-plane ($x = 0.5H$)</td>
<td>0.0169</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>0.0086</strong></td>
</tr>
</tbody>
</table>
Next, Table 7 and Figure 19 show the RMSE and wind speed ratios calculated using the medium and fine grid. Although there are no measurement points which show results that do significantly deviate, the total RMSE has slightly increased. Therefore, an even finer grid is tested.

Table 7: RMSE – Medium vs. Fine grid

<table>
<thead>
<tr>
<th>Plane</th>
<th>RMSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>xz-plane (y = 0.0625H)</td>
<td>0.0181</td>
</tr>
<tr>
<td>xy-plane (z = 0.0H)</td>
<td>0.0089</td>
</tr>
<tr>
<td>zy-plane (x = 0.25H)</td>
<td>0.0078</td>
</tr>
<tr>
<td>zy-plane (x = 0.5H)</td>
<td>0.0191</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>0.0101</strong></td>
</tr>
</tbody>
</table>

The results of the finest grid simulation are shown in Table 8 and Figure 20. The total RMSE is the lowest error calculated so far. Moreover, the small deviations that were noticed before are within an acceptable range now. Especially the higher wind velocity values are in good agreement. This implies that the fine grid is fine enough to use, and therefore will be used for the remainder of this validation study.

Table 8: RMSE – Finer vs. Fine grid

<table>
<thead>
<tr>
<th>Plane</th>
<th>RMSE</th>
</tr>
</thead>
<tbody>
<tr>
<td>xz-plane (y = 0.0625H)</td>
<td>0.0119</td>
</tr>
<tr>
<td>xy-plane (z = 0.0H)</td>
<td>0.0063</td>
</tr>
<tr>
<td>zy-plane (x = 0.25H)</td>
<td>0.0075</td>
</tr>
<tr>
<td>zy-plane (x = 0.5H)</td>
<td>0.0140</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>0.0079</strong></td>
</tr>
</tbody>
</table>
The results of the wind tunnel experiments are compared to the CFD simulations on the points displayed in Figure 21. An overview of the used settings is found in Chapter 3.4 and Appendix 1.2. Wind speed ratios are defined by dividing the local velocity in x-direction by the reference velocity ($U_{ref} = 4.2$ m/s).

Figure 21: points for comparison in (a) the horizontal plane $y/H = 0.0625$; (b) the vertical plane $z/H = 0$; (c) points for comparison in the vertical plane $x/H = 0.25$; (d) points for comparison in the vertical plane $x/H = 0.5$.

In Figure 22, the Root Mean Squared Error (RMSE) of the different simulation results with respect to the wind tunnel experiments is calculated. This figure indicates that the results belonging to the Realizable k-ε turbulence model shows the best agreement in 3 of the 4 planes. Only the Reynolds Stress Model and RNG k-ε model show a better agreement in the xz-plane. The use of the SST k-ω turbulence model results in the highest deviation in every measurement plane. Next, the results obtained with each turbulence model will be discussed in detail. More results are found in Appendix 3.
Figure 23 shows the wind speed ratios obtained in the xz-plane. By breaking down the plane into three different sections, it becomes clear that the results in front of the building are in better agreement than the results above and behind the building. This confirms one of the known deficiencies of using RANS with two equation models, which is the calculation of highly anisotropic wind-flow patterns around buildings. These wind-flow patterns include the separation and recirculation regions that are found above, besides and behind the building. Therefore, especially in these regions the simulation results deviate from the results obtained in the wind tunnel experiments.

As observed for the xz-plane, also in the xy-plane (see Fig. 24) the values in front of the building show the best agreement with the measurements. The total RMSE calculated in the plane is 0.129. The higher wind velocities are in a good agreement with the measurement results. In the yz-planes behind the building (see Fig. 25 on the next page), the values are less in agreement with the measurements, but this difference decreases when the wind speed increases.
With exception of some values behind the building, the results for the RNG k-ε turbulence model in the xz-plane, Figure 26, are well in agreement with the wind tunnel experiments. The RNG k-ε turbulence model owes its higher RMSE to the region above and behind the building as visible in Figure 27. In front of the building, the RNG model performs even better than the Realizable k-ε turbulence model.
SST k-ω TURBULENCE MODEL

3.5.2.3

Figure 28 and 29 shows that the high error values for the SST k-ω turbulence model are mainly caused by a poor prediction of the results behind the building. The results in front of the building have a fair agreement with the measurement results.

Figure 28: Scatterplot of the results obtained in the xz-plane (y=0.0625H) using the SST k-ω turbulence model versus the results obtained in the wind tunnel experiments with regards to the dimensionless wind velocity in x-direction.
Figure 29: Scatterplot of the results obtained in the xy-plane (z=0.0H) using the SST k-ω turbulence model versus the results obtained in the wind tunnel experiments with regards to the dimensionless wind velocity in x-direction.

In front of building | RMSE = 0.051
Above the building | RMSE = 0.227
Behind the building | RMSE = 0.294

Figure 30: Scatterplot of the results obtained in both yz-planes and confirms the poor prediction of the results behind the building. The results improve with higher wind speeds.

Figure 30: Scatterplot of the results obtained in the yz-planes (x=0.25H; x=0.5H) using the SST k-ω turbulence model versus the results obtained in the wind tunnel experiments with regards to the dimensionless wind velocity in x-direction.

- In front of building | RMSE = 0.205
- Above the building | RMSE = 0.234
- Behind the building | RMSE = 0.294
The Reynolds Stress Model shows comparable results to those of the Realizable k-ε turbulence model, but scores a better RMSE in the xz-plane and worse RMSE in the other planes. In Figure 31 the results for using the Realizable k-ε turbulence model and the RSM are compared in the xz-plane. It can be seen that the Realizable k-ε turbulence model predicts lower wind speed values than the RSM. By analysing the values at two lines (Fig. 32 & 33) it is noted that near the ground surface the RSM results are in better agreement with the wind tunnel experiments directly behind the building (Fig. 32). This difference reduces as the measurement points move away from the building and the wind speed increases. This advantage of the RSM, however, only appears near the ground surface. Figure 33 shows that at higher locations behind the building, the Realizable k-ε turbulence model is in better agreement with the wind tunnel experiments. This difference reduces as the measurement points are above the height of the building and the wind speed increases.

Figure 31: Scatterplot of the results obtained in the xz-plane (z=0.0625H) using the Realizable k-ε turbulence model and the RSM versus the results obtained in the wind tunnel experiments with regards to the dimensionless wind velocity in x-direction

Figure 32: Dimensionless wind velocity in x-direction for the line x/H = 0.5 in the horizontal plane xz (y=0.0625H)

Figure 33: Dimensionless wind velocity in x-direction for the line z/H = 0.25 in the vertical plane 2y (x=0.5H)
The results obtained using the Realizable $k$-$\epsilon$ turbulence model show the best agreement with the wind tunnel experiments in three of the four measurement planes. The Reynolds Stress Model (RSM) has a better score for the $xz$-plane. The use of the RNG $k$-$\epsilon$ turbulence model results in a higher error compared to the RSM and, with exception of the $xz$-plane, also the Realizable $k$-$\epsilon$ turbulence model. The use of the SST $k$-$\omega$ turbulence model results in the highest error in every measurement plane.

Looking at the performance of the Realizable $k$-$\epsilon$ turbulence model in all 4 planes it can be concluded that the results are best in agreement in front of the building and show some deviations above, besides and behind the building. However, the higher wind speeds are in a good agreement in every plane. This is a good thing, as these are the wind flows which cause wind danger.

The Realizable $k$-epsilon model is well described in literature and it is known that it can underestimate the separation and recirculation regions on top and besides the building and may overestimate the velocity and the size of the recirculation area behind the building (Blocken, 2012). The results seem to confirm this statement as the results in front of the building are much more in agreement than the results above and behind the building. Also some measurement points show a higher wind velocity on top of the building compared to the measurements. Because these values represent the velocity in $x$-direction, this may indicate that the recirculation area is indeed being underestimated. The same holds for the recirculation area behind the building.

The results from the simulations show a lower wind velocity in $x$-direction than the measurements. This may indicate an overestimation of the recirculation area which is in line with the known flaws of the $k$-$\epsilon$ turbulence models. In order to avoid the buildup of turbulent kinetic energy in the stagnation regions, the production term in the turbulence equations can be limited by a Production limiter (Menter, 1994) and the Kato-Launder modification (Kato & Launder, 1993). A small study was carried out to investigate whether the application of these options would improve the validation results. This study is found in Appendix 2. Although the application of the limiters improved the approximation of $k$, it does not lead to a significant change in velocities and is therefore not taken into account in the remainder of this investigation.

The RNG $k$-$\epsilon$ model performs better than the Realizable $k$-$\epsilon$ turbulence model in front of the building and near the bottom of the domain, but results in a larger overall error due to worse predictions besides and behind the building. The Realizable $k$-$\epsilon$ turbulence model produces the best results in three of the four planes. But, when it comes to the most important plane when investigating the pedestrian wind environment, the RSM results in the lowest error. However, because the Reynolds Stress Model converges slowly, and sometimes even has difficulties to converge at all, it is not very applicable for a parameter study. Therefore, and due to the fact that the Realizable $k$-$\epsilon$ turbulence model is well known, described and validated extensively in literature, this turbulence model will be used for the parameter study.
The geometry modifications are applied on a single building (isolated building). The definition of the building geometry is based on a study of Beranek & van Koten (1982), who investigated wind discomfort around buildings. He stated that the height of the building has a dominant influence, the width of the building has a strong influence and the depth of the building is of small importance. He also concluded that there is significant wind discomfort if the height and the width of the building exceed 50 meters. In addition, Willemsen and Wisse (2007) created a decision scheme to bring some order in the field by giving decision rules to discriminate between building plans which do or do not require action with regards to wind comfort. This decision scheme is based on experience and on research from Beranek (1980) and Bottema (1993). Here it is stated that buildings higher than 30 m should always require wind comfort assessment bases on a CFD or wind tunnel study. Together with the recommendations from Beranek (1982) this resulted in a building of 60 x 60 x 15 m$^3$ (H x W x D) that will be used in this study:

![Figure 34: Dimensions of the building used in parameter study](image_url)
As mentioned before, the investigated building has a height of 60 meters. The canopy is attached at a height of 4 m (1/15 H). The parameter study investigates the following offset sizes of the canopy: 3, 6, 12, 18, 24 and 30 meter (0.05H, 0.1H, 0.2H, 0.3H, 0.4 and 0.5H). Simulations are performed for three wind directions: \( \phi = 0^\circ \) (x direction), \( \phi = 45^\circ \) (xz direction) and \( \phi = 90^\circ \) (z direction). The results with regards to pedestrian-level wind velocities are evaluated on the sampling plane indicated in red in Figure 35, and which is situated at a height of 1.75 m. In the computational grid, the canopy is constructed as a zero-thickness wall.

For the investigation of the podium solution, approximately the same setup is used with respect to the canopy parameter study. As shown in Figure 36 the simulations are performed for three wind directions and the following offset sizes are investigated: 3, 6, 12, 18, 24 and 30 meter (0.05H, 0.1H, 0.2H, 0.3H, 0.4 and 0.5H).

The investigation of introducing a permeable floor (see Fig. 37) uses the same building as the previous studies. A building of 60 meters is used including fifteen floors of four meters each. The analysis focuses on the pedestrian level which is 1.75 m above the ground. The parameter study is conducted again for three wind directions and investigates in succession the omission of floor 0, 1, 2, 3, 4, 5, 7, 9, 10, 11 and 13.
For a single building the top of the computational domain should be at least 5H above the roof of the building, where H is the building height (Franke et al., 2007; Tominaga et al., 2008). This results in a domain height of 360 meters. After having chosen the height of the computational domain the lateral extension of the domain can be determined. Hall (1997), Cowan et al. (1997), Scaperdas & Gilham (2004), and Bartzis et al. (2004), Franke et al. (2007), Tominaga et al (2008), all recommend using a distance of 5H, in this case 300 meters, between the building’s sidewalls and the lateral boundaries of the computational domain. Concerning the longitudinal extension of the domain in the region in front of (approach flow) and the region behind (wake) the building have to be discerned. For a single building a distance of again 5H between the inflow boundary and the building is recommended if the approach flow profiles are well-known. The region behind the built area is terminated by the outflow boundary. In the case of a single building this boundary should be positioned at least 15H behind the building to allow for flow-redevelopment behind the wake region, as fully developed flow is normally used as a boundary condition in steady RANS calculations (Cowan et al., 1997; Scaperdas & Gilham, 2004; Bartzis et al., 2004; Franke et al., 2007; Tominaga et al., 2008). This results in a region behind the building of 900 meters. The simulation of different wind directions is taken into account in the construction of the computational domain, as shown in Figure 38a.

The grid has to be designed in such a manner that it does not introduce discretization errors that are too large. This means that the resolution of the grid should be fine enough to capture the important physical phenomena like shear layers and vortices with sufficient resolution. Also the quality of the grid should be high: Standard automatic or semi-automatic generation of an unstructured grid allows insufficient control of local grid resolution, grid stretching, control volume skewness and aspect ratio. To allow a large degree of control over the quality of the grid, it is constructed using the grid-generation technique by van Hooff & Blocken (2010). Ideally the grid is equidistant. Therefore, grid stretching/compression should be small in regions of high gradients, to keep the truncation error small. The expansion ratio between two consecutive cells should be below 1.3 in these regions (Franke et al., 2007).

In the area of interest, at least 10 cells per cube root of the building volume should be used and 10 cells per building side to simulate wind flow fields (Tominaga et al. 2008; Franke et al. 2004). This must be understood as an initial minimum grid resolution. The necessary resolution is analysed by the performance of a grid-sensitivity study (Chapter 4.2). As a result the two grids that are used in the parameter study consist of 1,066,080 cells (canopy/podium grid) and 2,049,348 cells (opening grid).
In contrast with the validation study, here symmetry boundary conditions are imposed on the lateral and top boundaries. Also, roughness is considered for the bottom of the domain. With regards to the bottom of the domain, the relationship between the aerodynamic roughness length $y_0$ and the parameters $k_s$ and $C_s$ was derived by Blocken et al. (2007):

$$k_s = 9.793 \frac{y_0}{C_s}$$  \hspace{1cm} (Eq. 7)

In Fluent 15.0.7 the value of $k_s$ is limited to the value of $y_p$, which is the height of the centre point of the wall-adjacent cell. The values for $y_0$ are estimated based on the updated Davenport-Wieringa roughness classification (Wieringa, 1992). For the first part of the domain, a roughness of 0.5 m is imposed to the bottom of the domain. A roughness of 0.5 m is indicated as very rough and is described as (Wieringa, 1992):

“Old cultivated landscape with many rather large obstacle groups separated by open space of about 10 obstacle heights.”

This type of terrain is chosen because a high-rise building, as investigated in this study, is rarely built in wide open spaces. Also other investigators impose a roughness of 0.5 m to the bottom of the domain in similar studies (Blocken & Carmeliet, 2008; Blocken et al., 2004). Because a roughness of 0.5 meters will be used there are some problems related to the construction of the grid. It is impossible to satisfy all requirements with respect to pedestrian-level wind analysis and the use of $k_s$-type wall functions. Pedestrian-level wind analysis requires a high-resolution grid near the bottom of the domain. At least three cells should be provided below pedestrian height which is 1.75 m (Janssen et al., 2013). On the other hand, the use of $k_s$-type wall functions within ANSYS Fluent requires a distance from the centre of the cell near the wall that is larger than the physical roughness ($k_s < y_p$) (ANSYS Fluent User’s Guide, 2013).

Various remedial measures are proposed by Blocken et al. (2007). However, it is known that none of them can be considered totally satisfactory. The measure applied in this study is described by Franke et al. (2004) and implies the use of a non-conformal grid. Here, the usual approach of using a constant height of all the cells adjacent to the bottom of the computational domain is abandoned. Instead, different initial cell heights are used in different areas of the domain to satisfy Equation (7) at every position in the domain. This includes using higher cells in the upstream and downstream regions and lower cells in the central region of the domain. To smoothen the transition between these separate grids an extra intermediate grid is created. A vertical cross section of the transition of the grids is displayed in Figure 39. Figure 38b indicates the location of the cross section. The inner grid includes the building and is limited to an offset of 36 meters around the building. The intermediate grid starts at the end of the inner grid and ends at an offset of 43 meters from the building. The outer grid starts at the end of the intermediate grid and covers the area to the boundaries of the domain. As can be seen, the cells near the bottom of the domain in the outer grid are larger than the cells near the bottom in the inner grid, which allow a roughness of 0.5 meters to be used.

**Figure 39:** Vertical cross section grid transition: (a) Canopy / podium grid (b); opening grid
The area near the building, the inner grid, is described by a roughness of 0.03 m, which implies open terrain (Wieringa, 1992):

“Level country with low vegetation (e.g. grass) and isolated obstacles with separations of at least 50 obstacle heights.”

The intermediate grid is used to smoothen the transition between the inner and outer grid and therefore an intermediate terrain roughness of 0.25 m is used. An overview is found below in Table 9.

<table>
<thead>
<tr>
<th></th>
<th>$y_0$ [m]</th>
<th>$k_3$ [m]</th>
<th>$C_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer grid</td>
<td>0.5</td>
<td>0.85</td>
<td>5.76</td>
</tr>
<tr>
<td>Intermediate grid</td>
<td>0.25</td>
<td>0.4</td>
<td>6.12</td>
</tr>
<tr>
<td>Inner grid</td>
<td>0.03</td>
<td>0.1</td>
<td>2.35</td>
</tr>
</tbody>
</table>

**WIND PROFILE**

4.1.6.1

The inflow at the inlet of the domain consists of the profiles of mean wind speed, turbulent kinetic energy and turbulence dissipation rate. The mean wind speed profile is prescribed by the logarithmic law with $y_0 = 0.5$ m (Eq. 1).

$$ U(y) = \frac{u^*_{ABL}}{\kappa} \ln \left( \frac{y + y_0}{y_0} \right) $$

(Eq. 1)

In this equation, $U(y)$ is the mean wind speed at height $y$, $u^*_{ABL}$ the friction velocity, $\kappa$ the Von Karman constant and $y_0$ the aerodynamic roughness length. The reference wind speed ($U_{ref}$) is 5 m/s at 10 m height.

The turbulent kinetic energy $k$ is calculated using Equation 8, where $u^*_{ABL}$ is the friction velocity and $C_\mu$ a model constant (0.09). The inlet turbulence dissipation rate is described by Equation 3. At the outlet, Equation 8 and 3 are used for respectively the turbulent kinetic energy $k$ and the turbulence dissipation $\varepsilon$. Zero static pressure is specified at the outlet.

$$ k = \frac{u^*_{ABL}^2}{\sqrt{C_\mu}} $$

(Eq. 8)

$$ u^*_{ABL} = \frac{\kappa U_{ref}}{ln \left( \frac{y + y_0}{y_0} \right)} $$

(Eq. 5)

$$ \varepsilon(y) = \frac{u^*_{ABL}^3}{\kappa(y + y_0)} $$

(Eq. 3)
First, a simulation in the empty domain is performed to ascertain whether the chosen grid and boundary conditions are consistent and there is no substantial change in the specified inflow boundary profiles. The result of the simulation in the empty domain can be seen in Figure 40. The velocity wind profiles in x-direction at four positions (Fig. 41) in the domain are compared; $x = -0.25H$ displays the velocities at the inlet, $x = -1.5H$ is shown to see how the velocity profile develops over a continuous roughness, $x = -0.9H$ is located in the intermediate grid and $x = -0.25H$ indicates the front of the building. The results are normalized by the inlet velocity at pedestrian height $U_h = 2.5 \text{ m/s (y = 1.75 m)}$.

A slight increase in velocity near the bottom is noticed in the first part of the domain. In the intermediate grid ($x = -0.9H$) the velocity has increased a bit more, but this is still a small difference. However, this difference increases in the area around the building. As expected, the velocity profile increases near the bottom of the domain as the roughness decreases. In front of the building ($x = -0.25H$) this difference has reached a value of approximately 150% of the inlet velocity on pedestrian level (1.75 m). This increase seems inevitable due to the different roughness values imposed on the bottom of the domain. However, in reality such acceleration can also occur since around high-rise buildings in the Netherlands quite often relatively open terrain like streets, grass, parks or sidewalks is present.

![Velocity profile](image)

**Figure 40:** Dimensionless wind velocity profile at different distances from the inlet.

![Figure 41](image)

**Figure 41:** Positions of velocity wind profiles in cross section through computational grid.

---

**OTHER COMPUTATIONAL PARAMETERS**

The CFD simulations are performed using the commercial CFD code ANSYS Fluent 15.0.7. The 3D steady RANS equations are solved. Pressure velocity-coupling is taken care of by the SIMPLE algorithm. Pressure interpolation is second order. Second-order discretization schemes are used for both the convection terms and viscous terms of the governing equations. An overview of the used settings is found in Appendix 1.3. The iterations were terminated when the scaled residuals did not show any further reduction with increasing number of iterations. The minimum values that were reached can be found in Appendix 6. Some simulations showed fluctuating residuals and the results have therefore been averaged. In order to average the results, 10, 20 or 40 data files with the results are exported from Fluent after a certain amount of iterations. After processing these data files, average results have been achieved.
Performing a grid-sensitivity analysis is important to reduce the discretization errors and the computation time. The grid-sensitivity analysis is best performed by comparing relevant parameters at relevant locations. In this study, the wind speed at pedestrian level (1.75 m) will be evaluated. Because the different geometrical variations required for the parameter study are taken into account in the model, this allows to easily generate a range of different models and grids without having to rebuild them all from the start. By defining extra meshed volumes in the model, one can change the building geometry by just deleting the volumes that have to become solid, i.e. part of the building (van Hooff & Blocken, 2010). This leads to a single model to perform the parameter studies for both the canopy and podium solution. Simulations are performed for $\phi = 0^\circ$ (x-direction).

**CANOPY/POD IUM**

A grid-sensitivity analysis is performed for the 9 m (1.5H) canopy configuration by making a finer grid with 2,130,664 cells. The medium and fine grid are shown in Figure 42 and 43.

When the simulations are completed, node values are extracted from ANSYS Fluent and interpolated in MATLAB for further analysis. For the interpolation, a grid spacing of 1 meter in x- and z-direction is used. The normalized results for the wind velocity in x-direction ($U_x/U_h$) are compared in Figure 44 and Figure 45. Figure 44 displays the location of the differences between the medium and the fine grid. The white contour lines give the dimensionless wind velocity in x-direction ($U_x/U_h$) to indicate the high and low velocity areas. The largest deviations occur in the recirculation areas near the corner streams in which the wind velocity is relatively low. Here differences of maximum 0.19 are noticed. Figure 45 shows that the differences between the two grids are small and confirms that higher wind velocities values are well in agreement. A RMSE of 0.016 is calculated within the measurement area with an offset of 50 meters (5/6 H) around the building at a height of 1.75 m. The use of a grid spacing of 0.5 m and thereby more measurement points results in an equal error (Appendix 4.1). This seems to indicate that the medium grid is a good compromise between computational accuracy and computational costs. This grid is therefore selected for further analysis.
For the investigation of the permeable floor the following models are created; a medium grid with 2,049,348 cells (Fig. 46) and a finer grid with 3,539,910 cells (Fig. 47). A grid-sensitivity analysis is performed for the building configuration with the third floor (fourth building layer) removed.

The comparison of the normalized wind velocity in x-direction ($U_x/U_h$) shown in Figure 48 indicates the location of possible differences between the results of the two grids. Like the previous grid-sensitivity study of the canopy/podium model, the largest differences of maximum 0.14 are found in the low-velocity recirculation areas near the corner streams. The scatter plot in Figure 49 indicates that there are no large deviations between the medium and finer grid. Moreover, a RMSE of only 0.01 is calculated and therefore the medium grid is retained for further analysis.
Because CFD produces data for different parameters in every point in the domain, a wide range of results is available. The results shown in the next paragraphs are a selection of all produced results found in Appendix 5.

First, the results of the canopy solution are shown, followed by the results with respect to the podium and the opening. The results are displayed by figures and graphs. The concerning case is found in the title of the figure. Table 10, 11 and 12 give an overview of the investigated building geometry modifications.

The resulting node values are extracted from ANSYS Fluent and interpolated in MATLAB. For the interpolation, a grid spacing of 1 meter is used within a sampling plane with an offset of 50 m (5/6H) around the building at a height of 1.75 m. Because this area decreases with an increasing podium size, the area inside the podium is left out of the calculation with respect to the determination of the average wind velocity. The results are normalized by the inlet velocity at pedestrian level $U_h = 2.5$ m/s ($y = 1.75$ m).

### Table 10: Parameter study – Canopy

<table>
<thead>
<tr>
<th>Canopy</th>
<th>Offset [m]</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>(base case)</td>
<td>0</td>
<td>B_0H</td>
</tr>
<tr>
<td>0.05H</td>
<td>3</td>
<td>C_0.05H</td>
</tr>
<tr>
<td>0.1H</td>
<td>6</td>
<td>C_0.1H</td>
</tr>
<tr>
<td>0.2H</td>
<td>12</td>
<td>C_0.2H</td>
</tr>
<tr>
<td>0.3H</td>
<td>18</td>
<td>C_0.3H</td>
</tr>
<tr>
<td>0.4H</td>
<td>24</td>
<td>C_0.4H</td>
</tr>
<tr>
<td>0.5H</td>
<td>30</td>
<td>C_0.5H</td>
</tr>
</tbody>
</table>

### Table 11: Parameter study - Podium

<table>
<thead>
<tr>
<th>Podium</th>
<th>Offset [m]</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>(base case)</td>
<td>0</td>
<td>B_0H</td>
</tr>
<tr>
<td>0.05H</td>
<td>3</td>
<td>P_0.05H</td>
</tr>
<tr>
<td>0.1H</td>
<td>6</td>
<td>P_0.1H</td>
</tr>
<tr>
<td>0.2H</td>
<td>12</td>
<td>P_0.2H</td>
</tr>
<tr>
<td>0.3H</td>
<td>18</td>
<td>P_0.3H</td>
</tr>
<tr>
<td>0.4H</td>
<td>24</td>
<td>P_0.4H</td>
</tr>
<tr>
<td>0.5H</td>
<td>30</td>
<td>P_0.5H</td>
</tr>
</tbody>
</table>

To investigate whether a possible correlation exists within the results, the Pearson product-moment correlation coefficient is calculated. This correlation coefficient for two sets of values, $x$ and $y$, is given by Equation 9, where $\bar{x}$ and $\bar{y}$ are the sample means of the two arrays of values.

$$r = \frac{\sum(x - \bar{x})(y - \bar{y})}{\sqrt{\sum(x - \bar{x})^2 \sum(y - \bar{y})^2}}$$

(Eq. 9)
0° WIND DIRECTION

For the 0° wind direction the average wind velocity in the sampling plane (Fig. 50) remains approximately equal with increasing canopy size. The maximum wind velocity (Fig. 51) does decrease with increasing canopy size until the size of the canopy reaches an offset of 0.2H. By then, the maximum wind velocity is decreased by 11% compared to the base case.

The decrease in maximum wind velocity is caused by a reduction of the corner wind streams. This can be seen in Figure 53 where the base case results are subtracted from the canopy solution with size 0.3H. With increasing canopy size, the high velocities in the corner streams are moved away from the building. In the region in front of the building, below the canopy, the wind velocity is reduced the most, whereas further upstream (before the canopy) the velocities are higher with a canopy than in the base case. Furthermore, the area right below the side edges of the canopy shows increasing wind velocities compared to the base case (yellow/red areas). Directly behind the building, no large effects are noticed caused by the attachment of a canopy, but in general the wind velocities are lower behind the building when a canopy is present (blue area).

![Figure 50: Relative percentage difference of the average wind velocity in the measurement plane with respect to the base case](image1)

![Figure 51: Relative percentage difference of the maximum occurring wind velocity in the measurement plane with respect to the base case](image2)

![Figure 52: Contours of the dimensionless wind velocity (U/Uₜₜ) in a horizontal plane at a height of 1.75 m for C₀.₃H and φ = 0°](image3)

![Figure 53: Contours of the subtraction of B₀H from C₀.₃H regarding the dimensionless wind velocity (U/Uₜₜ) in a horizontal plane at a height of 1.75 m for φ = 0°](image4)
Figure 54 displays velocity contours and velocity vectors in a vertical plane 0.5 m in front of the windward building façade. It shows the deviation of the wind flow in front of the building without canopy attached. From the stagnation point at approximately 0.75H the wind diverges in all directions. The wind flowing downwards contributes to the high-velocity vectors near the ground corner streams. When the canopy is attached, Figure 55, these high-velocity vectors cannot reach the bottom corners and move around the corner above the canopy.

With a larger canopy, Figure 56, the wind leaves the area above the canopy horizontally. In the situation with the smaller canopy in Figure 55, the wind flow just outside the canopy is directed slightly downwards to pedestrian level.

The effect of the canopy on the wind flowing downwards in front of the building is also visible in the vector plots at pedestrian level. In the base case the wind flows in negative x-direction in front of the building which indicates a standing vortex (Fig. 57). When a canopy is attached (Fig. 58), the wind flows in positive x-direction in front of the building. This implies the absence of a standing vortex which reduces the velocity in the corner streams.
Also in the 45° base case situation (Fig. 59), the corner streams produce the highest velocities, especially near the corner at the end of the long windward façade.

With increasing canopy size, these maximum occurring wind velocities decrease significantly. In Figure 60, the red areas near the corner have almost disappeared when a canopy of 0.3H is applied. Figure 61 displays the differences with the base case. It can be seen that in the high-velocity areas near the corner, reductions in wind velocity occur within a large area. However, there are also some areas in which the wind velocity increases (red areas), mainly near the windward corner of the building.

Looking at the wind directions around the building, differences can be seen in front of the long windward façade. Without a canopy attached (Fig. 59) the wind flow is directed away from the building, a possible result from the wind flowing downwards along the building. In the situation with the canopy attached (Fig. 60), the wind flows to the building, resulting in a more gradual separation of the wind flow around the building at pedestrian level.
The largest reduction is found in the 90° wind direction situation. A canopy offset of 0.05H already reduces the maximum occurring wind velocity by 10%. Moreover, with increasing canopy size the corner streams seem to disappear and in the region behind the building below the canopy, the wind velocity reduces to almost zero.

Figure 62, 63 and 64 explain the reason for this low velocity area. Figure 62 displays the base case situation, in which high-velocity corner streams are noted (orange/red areas). When a small canopy is attached (Fig. 63), the wind is directed to the lateral edges of the canopy and the velocity in the corner streams is reduced (orange area). The larger the canopy, the larger the low velocity area will be. As a result, in Figure 64 it can be seen that the velocity in the corner streams is significantly reduced and the red/orange areas have disappeared.

**AVERAGE**

When the three wind directions are averaged, an approximately linear correlation can be seen in Figure 65 between the average and maximum occurring wind velocity around the building. The average wind velocity has a correlation of -0.99 (Eq. 9) with the offset size of the canopy. The correlation of the maximum occurring wind velocity with the offset size of the canopy is -0.96. Both the average and the maximum occurring wind velocity decrease with increasing canopy size. Note that this is not a weighted average, here it is assumed that the wind comes equally from all three wind directions.
5.2.1 0° WIND DIRECTION

With a small podium attached to the building, the average (Fig. 66) and maximum wind velocities (Fig. 67) slightly increase. Better results are seen if the size of the podium is further increased. Positive results with respect to the average wind velocity are noticed with a podium size of 0.3H and larger. The maximum occurring wind velocity is reduced for all wind directions with the application of a podium with a size of 0.1H and larger.

The increase of wind velocity, compared to the base case, is visible near the corner streams in Figure 68. The red areas indicate an increase of the maximum wind velocity and the yellow areas behind the building are responsible for the increase of the average wind velocity.
If the size of the podium is increased to 0.3H (Fig. 69), it can be seen that the corner streams are significantly reduced in terms of maximum velocity and the higher wind velocities are located further away from the building. However, the vectors indicate that the wind flow at pedestrian level still contributes to the corner streams. Therefore it is assumed that the reduction of the maximum velocities in the corner streams is due to the decrease of vertical wind streams flowing downwards in front of the building. Vertical cross sections are created to investigate these wind streams. The location of the cross sections is found below in Figure 70.

Figure 70: Illustration of the location of the cross sections visible in Figure 71 and 72.

Figure 71 shows that the wind streams flowing downwards in front of the building partly flow to pedestrian level around the frontal edge of the podium. However, these are not the high-velocity wind flows, and thus do not contribute much to the corner streams.

The high velocity wind flows, flowing down in front of the building, wrap around the building corner on top of the podium, as can be seen in Figure 72. In this way, these high-velocity wind streams are effectively held away from pedestrian level.

Figure 71: Contours of the dimensionless wind velocity (U/U_h) and velocity vector field in a vertical section at z/H = 0.51 for P_0.3H and φ = 0°

Figure 72: Contours of the dimensionless wind velocity (U/U_h) and velocity vector field in a vertical section at z/H = 0.51 for P_0.3H and φ = 0°
45° WIND DIRECTION

5.2.2

In the 45° wind direction situation the average and maximum wind velocity decrease with an increasing podium size. A podium of 0.05H (Fig. 73) already results in a significant reduction of the corner streams (Fig. 74). A further reduction is noticed when the size of the podium is increased to a size of 0.2H (Fig. 75) and 0.4H (Fig. 76). In Figure 76 for instance, the high velocity areas displayed in red have almost disappeared.

Because the measurement area decreases with an increasing podium size, the area inside the podium is left out of the calculation with respect to the determination of the average wind velocity. However, the actual performance of larger podiums may be different than represented by the results. This is the result of the acceleration of the wind within the measurement area, outlined in chapter 4.1.6. Because an increase in podium size reduces the measurement area, the wind may reach the podium with a less accelerated wind velocity.
The 90° situation shows the most reduction of the maximum wind velocity. An increase in podium size leads to a decrease of the high velocity corner streams. Figure 77, 78 and 79 show a decrease of the red high-velocity area with increasing podium size. In the area behind the podium, the wind velocity reduces to almost zero.

**AVERAGE**

When the results of the three different wind directions are averaged, the graph in Figure 80 can be seen. Here, the average wind velocity gradually decreases with an increasing podium offset. This also holds for the maximum occurring wind velocity. The correlation of the average as well as the maximum occurring wind velocity with the offset size of the podium is -0.97 (Eq. 9).
Although similarities between both solutions exist, they show different results with different wind velocities. With a small podium, the average and maximum wind velocities slightly increase in the 0˚ wind direction. Better results can be seen if the size of the podium is increased. From a podium size of 0.3H the average wind velocity (Fig. 81) is significantly reduced by 12% for a podium size of 0.5H. When looking at the maximum occurring wind velocity in the measurement region (Fig. 82), the podium solution shows better results than the canopy solution for an offset larger than approximately 0.15H. For a podium size of 0.5H the maximum wind velocity is decreased by 28%.

It was expected that the wind velocity on top of the roof of the podium/canopy would be higher with the podium solution, because the lower wind flows cannot go through the podium as they can in the canopy solution. This would result in extra wind flow over the podium. However, the expectation is not supported by this study as the canopy solution shows an higher average and maximum wind velocity (Table 13).

Table 13: Average and Maximum dimensionless wind velocity in a horizontal plane at a height of 5 m for \( \phi = 0^\circ \)

<table>
<thead>
<tr>
<th></th>
<th>((U/U_h)_{avg})</th>
<th>((U/U_h)_{max})</th>
</tr>
</thead>
<tbody>
<tr>
<td>Canopy</td>
<td>1.39</td>
<td>2.8</td>
</tr>
<tr>
<td>Podium</td>
<td>1.31</td>
<td>2.74</td>
</tr>
</tbody>
</table>

Differences are seen in Figure 83, here the velocity magnitude of the canopy simulations are subtracted from the velocity magnitude of the podium simulations in a plane at 5 m height (1 m above the roof). Values above zero indicate that the podium solution results in higher wind velocities than the canopy solution. Values below zero show where the canopy solution results in a higher wind velocity.
For the 45° wind direction both solutions show a decrease of the average wind velocity with an increasing offset. Both solutions show approximately linear results. For the maximum offset simulated in this study, 0.5H, the canopy solution shows a decrease of 7% in average wind velocity and the podium solution results in a decrease of 22% (Fig. 84). The development of the maximum wind velocity (Fig. 85) shows a similar pattern for both solutions, resulting in a maximum decrease of 20% for the canopy solution and 26% for the podium solution.

In contrast to the other wind directions, the 90° wind direction shows better results for the canopy solution than for the podium solution when it comes to the average wind velocity (Fig. 86). The largest difference is noticed with an offset of 0.3H, where the average wind velocity for the canopy solution is almost 10% lower than the average wind velocity in the podium solution. In line with the results of the 45° simulations, both solutions show similar results for the development of the maximum wind velocity (Fig. 87). For an offset of 0.5H the canopy solution shows a decrease of 36% in maximum wind velocity and the podium solution shows a decrease of 34% in maximum occurring wind velocity in the sampling plane.
With help of Figure 88 and 89, differences between the two solutions are analysed. In the podium solution, higher wind velocities can be seen at the side of the building. In the canopy solution, these corner streams seem more diffused, resulting in a lower maximum wind velocity in the corner streams. The large difference with regards to the average wind velocity can be declared by the fact that the area within the podium is not taken into account in the calculation, in contrast with the canopy situation where very low wind velocities occur in this area. When the offset increases this difference decreases, resulting in a total decrease in average wind velocity for the canopy solution of 29% and 26% for the podium solution with respect to the base case.

Although the individual wind directions show differences between the application of a canopy or a podium, when averaged (Fig. 90 and 91) both solutions show similar trends. In this situation, the podium is preferred as a solution when the offset is minimal 0.3H. For a small offset of 0.05H, best scores are shown for the canopy solution.
Figures 92 and 93 show the average and maximum occurring wind velocity in the measurement area with respect to the base case in percentages, respectively. Figure 94 is added to give more insight in the area over which the high wind velocity values are spread. Here the values of the 60 highest occurring wind velocities $\left(60 \text{ m}^2\right)$ around the building on pedestrian level are averaged. The x-axis in all three figures describes the floor which is eliminated from the building. In the next paragraphs the results with respect to the opening solution are described per floor that is removed.

The $0^\circ$ and $45^\circ$ situation show comparable results, the removal of lower floors can lead to an increase of the average and maximum occurring wind velocity. With respect to the maximum wind velocity, best results are seen when the second floor is removed from the building. The lowest average wind velocity in the area is noticed when there is a permeable floor at approximately half the height of the building. Different results are seen by looking at the $90^\circ$ situation. For this wind direction the maximum occurring wind velocity is found by removing the lower floors of the building, and increases with the removal of every higher located building layer. While the removal of the lower floors leads to a reduction with a maximum of 10% of the highest occurring wind velocity, the average wind velocity in the sampling plane increases with respect to the base case. This indicates that, although the maximum wind velocity may be lower, these higher wind velocities are distributed over a larger area than for the base case.
The figures on the right confirm that it is not advisable to convert the ground floor of the building into a permeable floor. On average the wind velocity around the building is 20% higher when the ground floor is open in the 0° wind direction and 15% higher in the 45° wind direction. However, the maximum wind velocities show an increase of only 3% in the 45° situation and even a decrease of 3% in the 0° situation. This indicates that the higher wind velocities stretch out over a larger area (Fig. 95 and 96). This is also confirmed by Figure 94, which states that there are more high wind velocity values than in the base case. To a larger extent this effect is noticeable in the 90° situation (Fig. 97). Here, the maximum wind velocities are reduced by 10%, while on the other hand the average wind velocity is 3% higher than the base case.

Looking at the section through the building in Figure 98 (φ = 0°), it can be seen that the high velocity wind flow flowing down in front of the building is directed through the opening, creating high wind velocities in the opening and eliminating the wake region on pedestrian level behind the building.

The idea of the introduction of a permeable floor in the building includes that, instead of creating a standing vortex at ground level, the wind flowing downwards in front of the building is directed through the building. The removal of the ground floor does not contribute to this idea, as the wind flow is being directed to pedestrian level and even increases the maximum wind velocities at pedestrian level.
Removing the first floor of the building increases the average wind velocity around the building by 8% compared to the base case in the 0° wind direction and by 4% in the 90° wind direction. The average wind velocity in the 45° situation is approximately equal to the base case. The maximum occurring wind velocities, on the other hand, are reduced by 5% and 10% in the 45° and 90° situation respectively. In the 0° situation the maximum wind velocity is increased by 2%, but these high velocity values cover only a small area compared to the base case. Figure 99 shows the wind velocity around the building at pedestrian level when the first floor of the building is removed. The differences with the base case are visualized in Figure 100. The red areas make clear that the removal of the first floor causes a significant increase in corner stream velocity and wind velocities behind the building.

A section of the 0° wind direction makes clear that the high velocity wind flow through the opening is directed downwards on the back side of the building (Fig. 101), and thereby increasing the wind velocities at pedestrian level. In Figures 102 and 103 it can be seen that the wind flowing downwards in front of the building is redirected and/or reduced in velocity.

Figure 99: Contours of the dimensionless wind velocity \( \frac{U}{U_h} \) in a horizontal plane at a height of 1.75 m for \( F_1 \) and \( \varphi = 0° \)

Figure 100: Contours of the subtraction of \( F_0 \) from \( F_1 \) regarding the dimensionless wind velocity \( \frac{U}{U_h} \) in a horizontal plane at a height of 1.75 m for \( \varphi = 0° \)

Figure 101: Contours of the dimensionless wind velocity \( \frac{U}{U_h} \) and velocity vector field in a vertical section at \( z/H = 0 \) for \( F_1 \) and \( \varphi = 0° \)

Figure 102: Contours of the dimensionless wind velocity \( \frac{U}{U_h} \) and velocity vector field in a vertical plane at a distance of 0.5 m in front of the windward façade of the building for \( F_1 \) and \( \varphi = 0° \)

Figure 103: Contours of the dimensionless wind velocity \( \frac{U}{U_h} \) and velocity vector field in a vertical plane at a distance of 0.5 m in front of the windward façade of the building for \( F_1 \) and \( \varphi = 90° \)
In this case, the wind flow through the opening barely causes an increase in wind velocity in the wake region of the building (Fig. 104). In contrast to the results for the permeable first floor, the wind flow through the permeable second floor is now directed upwards behind the building (Fig. 105). On top of that, the wind velocity directly in front of the building at pedestrian level is reduced because the high velocity wind velocities flowing downwards in front of the building flow through the opening, and thereby do not participate in the creation of a standing vortex at ground level. In Figure 106 it can be seen that the high-velocity wind streams in front of the building are significantly reduced after they have passed the height of the opening. Although higher velocity corner streams are noticed compared to the base case (Fig. 104), here (φ = 0˚) the average wind velocity is reduced by 2% and the maximum wind velocities by 6%.

For the 0˚ and 45˚ wind direction, the removal of the second floor shows the lowest occurring maximum wind velocity, while also the average wind velocity in the measurement area is lower than the average wind velocity in the base case. For the 90˚ wind direction, the average wind velocity is higher than the base case, but the maximum occurring wind velocity is further reduced compared to the 0˚ and 45˚ situation.

Figure 104: Contours of the subtraction of F_00 from F_2 regarding the dimensionless wind velocity (U/U_h) in a horizontal plane at a height of 1.75 m for φ = 0˚.

Figure 105: Contours of the dimensionless wind velocity (U/U_h) and velocity vector field in a vertical section at z/H = 0 for F_2 and φ = 0˚.

Figure 106: Contours of the dimensionless wind velocity (U/U_h) and velocity vector field in a vertical plane at a distance of 0.5 m in front of the windward façade of the building for F_2 and φ = 0˚.
In this study, the removal of the second floor leads to the lowest maximum wind velocities. The view on both windward sides in the 45° wind direction gives a good example.

If the permeable floor is too high, more wind flows down due to the extra surface area below the permeable floor on which the wind deviates. If the permeable floor is too low, the high wind velocities within the permeable floor affect the pedestrian level wind velocities.

In the base case (Fig. 107), the wind is directed downwards along the building. This results in excessive wind velocities at the end of the building at pedestrian level.

When the second floor is removed, Figure 108, this permeable floor acts like a barrier. Below the second floor, mainly horizontal vectors are noted. This indicates that less wind from above is directed downwards, resulting in a reduction of the corner wind velocities at the end of the building.

The removal of a higher floor (Fig. 109) leads to a higher located barrier. Now the surface area below this barrier is increased, allowing more wind to be directed downwards.
HIGHER LOCATED FLOORS

By removing higher located floors, the corner streams move to the velocities observed at the base case and the average wind velocity is reduced further to approximately 5% by the removal of the seventh floor. In general, the maximum observed wind velocities increase with the removal of higher located floors compared to the removal of lower floors.

In the base case building a stagnation point is created at approximately 0.75H. This height corresponds with the tenth floor of the building. It is investigated whether the removal of this building layer would result in different effects on the wind flow but this is not the case. As was the case in the other situations, a new stagnation point is created in the region above and below the opening from which the wind flow is deviated (Fig. 110).

WIND VELOCITY THROUGH THE OPENING

Figure 111 displays the average wind velocity through the opening in the 0° wind situation. This data is collected on the plane at half the height of the opening. It is striking that the lowest velocity through the opening is in agreement with the lowest maximum velocity noticed on pedestrian level, namely with the removal of the second floor. In Figure 112, these values are normalized by the inlet velocity at the height of the permeable floor (U_var). It can be seen that the wind is reduced in all cases with respect to the inlet velocity at the same height. This reduction is less present in the lower floors of the building, but increases to a velocity of only ⅓ of the inlet velocity.

![Figure 110: Contours of the dimensionless wind velocity (U/U_h) and velocity vector field in a vertical plane at a distance of 0.5 m in front of the windward façade of the building for F_10 and φ = 0°](image1)

![Figure 111: Average wind velocity through the building at half the height (2 m) of the permeable floor for φ = 0°](image2)

![Figure 112: Average dimensionless wind velocity through the building at half the height (2 m) of the permeable floor for φ = 0°](image3)
The removal of the second floor of the building leads, on average, to the largest reduction of the maximum occurring wind velocity on pedestrian level (Fig. 113). For the lowest average wind velocity, best results are seen when a floor half the height of the building is removed (Fig. 114), although the differences are not very large between floors two and fourteen.

**Figure 113:** Relative percentage difference of the average wind velocity in the measurement plane with respect to the base case, averaged for all wind directions

**Figure 114:** Relative percentage difference of the maximum occurring wind velocity in the measurement plane with respect to the base case, averaged for all wind directions
CONCLUSION

HOW CAN BUILDING GEOMETRY MODIFICATIONS IMPROVE PEDESTRIAN-LEVEL WIND COMFORT AROUND HIGH-RISE BUILDINGS?

The level of mechanical wind discomfort depends on the wind velocity, so by reducing high-velocity wind streams at pedestrian level the pedestrian-level wind comfort can be increased. The highest wind velocities at pedestrian level are found in the corner streams. These corner streams partly result from the wind flowing downwards in front of the windward façade. By limiting the amount of wind flow that reaches this area, the velocity in the corner streams can be reduced. Therefore, the building geometry modifications investigated in this study all focused on these vertical wind streams.

The application of a canopy and podium to the building results in a physical barrier which causes the high-velocity wind velocities to wrap around the building corners directly above the canopy/podium. In general, it can be concluded that the larger the offset of the measure is, the smaller the amount of high-velocity wind flows that reach pedestrian level is.

With the introduction of a permeable floor in the building, the wind flowing downwards in front of the building is directed through the building before reaching pedestrian level. In this way, the creation of a standing vortex at ground level is limited and thereby reducing the resulting high-velocity corner streams. Note that the removal of a too low building layer will result in adverse effects, as it will direct the high-velocity wind streams to pedestrian level.

WHAT WILL BE THE INFLUENCE OF FOLLOWING BUILDING GEOMETRY MODIFICATIONS APPLIED TO A HIGH-RISE BUILDING?

CANOPY

For all wind directions, the application of a canopy leads to a decrease in the maximum occurring wind velocity. The higher velocity wind streams are located further away from the building and reduce the maximum wind velocity with increasing canopy size. A maximum reduction of 11% is noted in the 0˚ wind direction, while the application of the largest canopy investigated reduces the maximum wind velocity by 20% for the 45˚ wind direction and by 36% for the 90˚ wind direction. The vector plots confirm that the high-velocity corner streams near the building are reduced by the application of the canopy. The wind flowing downwards in front of the building does not longer reach pedestrian level and wraps around the corners on top of the canopy. However, with a relatively small canopy the wind is diverged downwards over the edges of the canopy. Compared to the base case, the area directly below the edge of the canopy shows increasing wind velocities, while for all wind directions, the area in front of the long façade shows a reduction in wind velocity.

The average wind velocity over the measurement area remains approximately equal in the 0˚ wind direction. A decrease of 6% is noted in the 45˚ wind direction, while in the 90˚ situation the average wind velocity is reduced by a maximum of 29%.
PODIUM

The maximum and average wind velocities in the measurement area decrease with increasing podium size for all wind directions. An exception to this is the smaller podium in the 0° wind direction. The application of the podium causes the corner streams to reduce in velocity. Also, the high wind velocity areas are now located further away from the building instead of near the corners. The maximum reduction of the average and maximum wind velocity are found with the application of the largest canopy investigated. Concerning the average wind velocity a maximum reduction of 12% is found in the 0° wind direction, 22% for the 45° wind direction and 26% for the 90° wind direction. For the maximum wind velocity the maximum reduction is 34% in the 90° situation, while in both other wind directions a reduction of approximately 27% is achieved.

The vector plots indicate that the wind flow below the edge of the podium still contributes to the corner streams. There is almost no wind flow passing the podium vertically, instead these wind flows wrap around the side corners. Nevertheless, the velocities in the corner streams are reduced. This is because of the high-velocity vertical wind flows in front of the building, which wrap around the corners on top of the podium and therefore do not reach pedestrian level.

PERMEABLE FLOOR

The removal of a building floor can reduce the maximum and average wind velocity around a building at pedestrian level, but to a lesser extent than the canopy and podium solution. The wind flowing downwards in front of the building (partly) flows through the permeable floor and thereby does not participate in the creation of a standing vortex at ground level.

In general, the maximum occurring wind velocity decreases with a lower building layer to be removed. However, when a too low building layer is removed, adverse effects on the pedestrian wind environment are observed. In this case, the high wind velocities within the permeable floor affect the pedestrian-level wind velocities. If the permeable floor is too high, the more wind flows down due to the extra surface area below the permeable floor, which acts like a kind of wind catcher.

Another important condition seems to be the direction of the wind after it leaves the opening at the back of the building. With the removal of the bottom floors of the building, the wind flow is directed downwards on the leeward side of the building and therefore eliminating the wake region on pedestrian level behind the building. When higher floors are removed the wind is directed upwards, away from the pedestrian wind environment.

The removal of the second floor of the building leads, on average, to the largest reduction of the maximum occurring wind velocity on pedestrian level. For both 0° and 45° situations this results in a maximum reduction of 6%. In the 90° wind direction, the removal of the ground and first floor leads to a maximum reduction of 10%. For the lowest average wind velocity, best results are seen when a floor halfway the building is removed. This results in a maximum reduction of 6% for both 0° and 45° wind directions. No significant reduction is noted in the 90° situation.

IN GENERAL

The canopy and podium solution can have a much larger effect on the pedestrian wind environment than the introduction of a permeable floor to the building. However, it is difficult to determine the best measure. Not only does the size and choice of the applied measure affect the pedestrian wind environment, also the wind direction has a significant impact on the results. Therefore, to define the best measure for a building (design) it is important to know which wind directions produce the highest wind velocities. By linking wind statistics to the parameter study results, a weighted average can be calculated which will give a better insight in the effect of a possible building geometry modification, e.g. canopy, podium and opening.
DOES THE ANALYSIS ALLOW A DERIVATION OF GENERAL GUIDELINES FOR THE APPLICATION OF CANOPIES, PODIUMS AND BUILDING OPENINGS FOR DIFFERENT BUILDING CONFIGURATIONS?

Different buildings in different urban environments ask for different solutions. It is therefore inconvenient to create guidelines which are applicable for all situations. Wind comfort studies will still be necessary to investigate the effect of a high-rise building on the pedestrian wind environment. However, in the early stage of the design, the following guidelines can be taken in mind:

- “Building entrances near corners of especially high-rise buildings should be avoided, as well as walkways or bicycle routes.” (Blocken & Carmeliet, 2004)
- “Recreational areas around high-rise buildings should be avoided unless specific attention will be given to the design of these areas.” (Blocken & Carmeliet, 2004)
- “The effectiveness of specific design features has to be ascertained by the use of wind tunnel or numerical modeling.” (Blocken & Carmeliet, 2004)
- “Passages through buildings and passage-ways that are led through narrow passages between buildings should be avoided because of pressure short-circuiting effects.” (Blocken & Carmeliet, 2004)

**CANOPY**

- The application of a canopy can lead to a decrease in the maximum occurring wind velocity. The decrease in maximum velocity becomes larger with increasing canopy size.
- After the application of a canopy the higher velocity wind streams are usually located further away from the building.
- Especially the pedestrian-level region in front of a long building façade can decrease significantly in wind velocity by the application of a canopy.

**PODIUM**

- After the application of a podium, the maximum and average wind velocities around the building can decrease. This decrease of the velocities will be larger with increasing podium size.
- The velocities in the corner streams can be significantly reduced. With increasing podium size the high velocity areas move away from the building corners. Yet, a small podium may increase the velocity in the corner streams.

**PERMEABLE FLOOR**

- Avoid the use of the ground floor as a permeable floor in the building. As known from literature and as shown again in this study this can lead to large high wind velocity areas on pedestrian level.
- A permeable floor can decrease the maximum wind velocity at pedestrian level. Best results are noted when the lowest building layer not negatively affecting the pedestrian wind environment is removed.
  - However, if a too low building layer is removed from the building, this will result in an increase in the average and maximum wind velocity at pedestrian level and the wake region behind the building will disappear.
- For the lowest average wind velocity, best results can be achieved when a floor halfway the building is removed.
The conclusion provides some useful guidelines for the application of measures to reduce the pedestrian-level wind velocities. However, these recommendations are based on the mean wind velocity and do not take into account sudden changes in wind direction. Together with the mean wind velocity, these sudden wind gusts also contribute to pedestrian wind discomfort. In this study steady RANS is used, but when insight in sudden wind gusts is required the use of unsteady RANS or Large Eddy Simulation (LES) is recommended/needed.

The numerical simulations of the wind flow pattern introduce a number of errors and uncertainties. It is tried to reduce these uncertainties by the performance of a validation study and grid-sensitivity analyses. Although a good agreement was achieved between CFD simulations and wind-tunnel measurements in front of the building, larger deviations were obtained for wind velocities on the side and back of the building. Also, the grid-sensitivity analysis made clear that deviations can occur in the recirculation area near the corner streams. Another limitation of the study is that only an isolated high-rise building was considered.

Due to different surface characteristics, outlined in chapter 4.1.6, the wind accelerates when it enters the area of interest within the computational domain. Different tests are performed to handle this problem but eventually it seemed inevitable that a certain level of acceleration occurs. This is not a problem if the distance from the inlet of the computational domain to the building is the same with every building configuration tested, hence the wind would have accelerated by an equal amount. However, because an increase in podium size reduces this distance, the wind may reach the podium with a less accelerated wind velocity. Therefore, the actual performance of larger podiums may be different than represented by the results. In general, the absolute average wind velocity in the 90° wind direction is higher than in the other two wind directions due to a smaller wake region. However, there is also more acceleration near the ground surface because the area of interest is larger in z-direction (φ = 90°) than in x-direction (φ = 0°).

FURTHER RESEARCH

Further research may focus on a permeable floor above the canopy/podium. This combination of solutions may further decrease the high wind velocities on pedestrian level. Also, the application of an enclosure on the edge of the canopy possibly further reduces the wind flowing downwards. An investigation on varying heights of the canopy/podium might also contribute to the reduction of pedestrian-level wind velocities. The results with respect to the permeable floor may be improved by the application of Venturi-shaped openings to guide the wind flow to a greater extent through the opening. Likewise, the empty space created by the permeable floor can be used for the placement of wind turbines for the generation of electricity (Li et al., 2016). Other possible building geometry modifications that can have a positive effect on the pedestrian wind environment are the attachment of wings to the sides of the building to create a low velocity area in front of the building (Fig. 115), or a widened building layer above pedestrian level (Fig. 116). Another question is how the performance of the investigated solutions for a high-rise building in a modelled urban environment will be.

This study focused on the mean wind velocity, so the use of steady RANS was a justified way. Pedestrian wind discomfort is also caused by the existence of sudden changes in wind direction occurring near high-rise buildings. To investigate this nuisance the use of unsteady RANS is recommended.
REFERENCES


