Asia Pacific Research Initiative for Sustainable Energy Systems 2012 (APRISES12)

Office of Naval Research
Grant Award Number N00014-13-1-0463

Computational Fluid Dynamics (CFD) Applications at the School of Architecture, University of Hawaii: Literature Review of External CFD

Task 7

Prepared For
Hawaii Natural Energy Institute

Prepared By
Sustainable Design & Consulting LLC, UH Environmental Research and Design Laboratory & UH Sea Grant College Program

August 2013

Computational Fluid Dynamics (CFD) Applications at the School of Architecture, University of Hawaii

Project Phase 1 – 7.A

Develop Skill Set for External CFD Analysis and Verification at the Building

Project Deliverable No. 1: Literature Review of External CFD

Prepared for Hawaii Natural Energy Institute

in support of

Contract #N000-14-13-1-0463

August 4, 2013

Prepared by:

Manfred J. Zapka, PhD, PE (Editor) (*)

Sanphawat Jatupatwarangkul, D.Arch (**)

Tuan Tran, D.Arch (**)

(*) Sustainable Design & Consulting LLC

(**) Environmental Research and Design Laboratory (ERDL), School of Architecture, University of Hawaii at Manoa

Summary and Objective

This review of scientific and technical literature is conducted to identify previous published research in areas that pertain to predicting wind phenomena around buildings by means of Computational Fluid Dynamics (CFD) investigations. The use of CFD programs to assess wind phenomena around buildings has become an important design and analysis tool in fields and their special applications, such as:

- Wind engineering with an emphasis on urban wind comfort
- Wind loading on buildings
- Dispersion of gaseous matter or laden air around building
- Wind induced natural ventilation performance for buildings
- Convective heat transfer processes from exterior building envelope

The present review of scientific and technical literature is arranged in accordance with three previously identified areas of concern, which are as follows:

1. Review of basic processes of wind movements around buildings, including but not limited to topics addressing atmospheric boundary layer, mechanism of wind induced pressures, effects of building geometry on wind regime, basics of urban wind comfort.
2. Review of assessment methods of air movement around building, including an assessment of the effectiveness of CFD compared with other prediction methods
3. Review of the state of the art and trends in external CFD applications, including but not limited to basics consideration of CFD calculation processes , verification and validation, tasks required for CFD pre-processing, solver and post processing.

The present literature review is presented in three parts:

A) Part A: A discussion of various topics concerning previous research in external CFD; these topics have been previously identified as being important for the present project work. The part of discussing the various topics is organized in three chapters reflecting the above mentioned areas of concern 1 through 3.
B) Part B: A list of primary literature reviewed
C) Part C: A short summary for each primary literature reviewed; the summaries provide bullets and short descriptions of the topics found in the literature that will be helpful in the present research

Table of Contents:

Part 1 – Discussion of Previous Research in External CFD:

Part 2 – Listing of Primary Literature Reviewed

Part 2 – Summary Review for each Primary Literature Document

Part 1 – Discussion of Previous Research in External CFD

This part of the literature review discusses previously published research which provides important application knowledge on how to prepare, conduct, analyze and validate CFD simulations for wind induced phenomena around buildings.

1. Fundamentals of Wind Movement around Buildings and Urban Wind Comfort

This section reviews literature about wind movement phenomena around buildings and at a neighborhood level. The literature cited and reviewed contains pertinent information for the present CFD project.

1.1 Wind Induced Pressure Differentials on the Building Envelope

Kleiven (2003) describes the relationship between building design and natural ventilation. The author first elaborates architectural consequences of natural ventilation. The author then establishes the extent of natural airflow being an important design criterion, thus contributing significantly to the design of energy efficient and sustainable buildings. The primary goals of this study is offering a better understanding of the architectural premises for utilization of natural ventilation and as a result identify potentials associated with the utilization of natural driving forces in better building designs.

Kleiven (2003) suggests that there are three essential aspects of natural ventilation to describe and classify various concepts. The first aspect is the natural force utilized to drive ventilation. The driving force can be wind, buoyancy or a combination of both. The second aspect is the ventilation principle used to exploit the natural driving forces to ventilate a space. This can be done by single-sided ventilation, cross ventilation, or stack ventilation. The third aspect is the characteristic ventilation element used to realize natural ventilation. The most important characteristic design elements are wind towers, wind scoops, chimneys, double façades, atria, and embedded ducts.

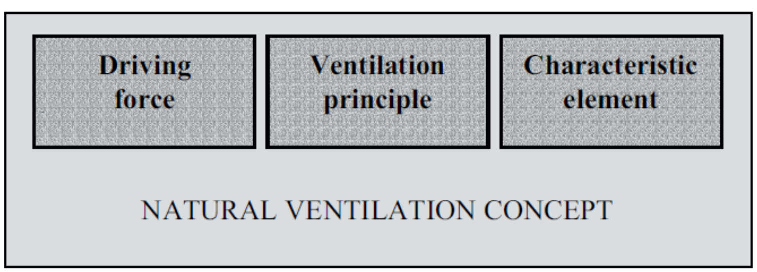


Figure 1.1A: To the notion natural ventilation concept we assign the driving force that is utilized to drive a ventilation principle with the aid of certain characteristic ventilation elements. (Kleiven, 2003)

There are only two fundamentally different types of natural driving forces available; thermal buoyancy and wind. The properties of these two are elaborated in the following section. Both their individual effect and their combined effect are described.

Thermal buoyancy driven ventilation occurs when there is a density difference between the internal and external air, which again is caused by temperature differences between the inside and outside. Thermal buoyancy is sometimes referred to as the stack effect or the chimney effect. The difference in density creates pressure differences that pull air in and out of a building through suitably placed openings in the building envelope. When the indoor air temperature exceeds the outdoor temperature, an over-pressure is built up in the upper part of the building and an under-pressure is formed in the lower part. At a certain height, the indoor and outdoor pressure equals each other, and this level is referred to as the neutral plane. An over-pressure above the neutral plane drives air out through openings in the building envelope, and an under-pressure under the neutral plane pulls air in through openings in the building envelope.

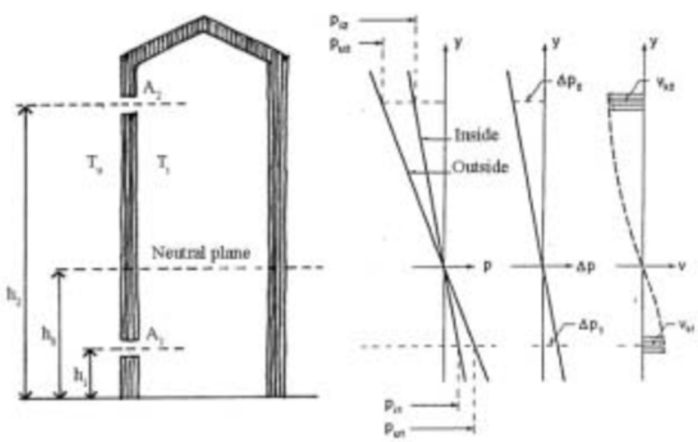


Figure 1.1B: Thermal buoyancy in a space with two openings. (Kleiven, 2003)

Wind driven ventilation occurs as a result of pressures gradients created on the building envelope by the wind. These pressure differences drive air into the building through openings in the building envelope's windward side, and drive air out of the building through openings in the building envelope's leeward side. The wind pressure on a surface in the building envelope is the dynamic pressure given by:

$$p_v = C_p (1/2 \rho_u V_{ref}^{\;2}) \text{ [Pa]}$$

Figure 1.1C: Wind pressure formation

Where p_v is the wind pressure [Pa], C_p is the static pressure coefficient, V_{ref} is the wind speed at reference height [ms^{-1}], and u is the outdoor air density [kgm^{-3}]. (Kleiven, 2003)

Thermal buoyancy and wind in combination: The two driving forces can occur separately but most likely they occur at the same time. Thermal buoyancy will typically be the dominating driving force on a calm cold day with practically no wind, whereas pressure differentials created by wind will typically be the dominating driving force on a windy hot day. Their forces can oppose or complement each other depending on the placement of the inlet and outlet openings in relation to the wind direction.

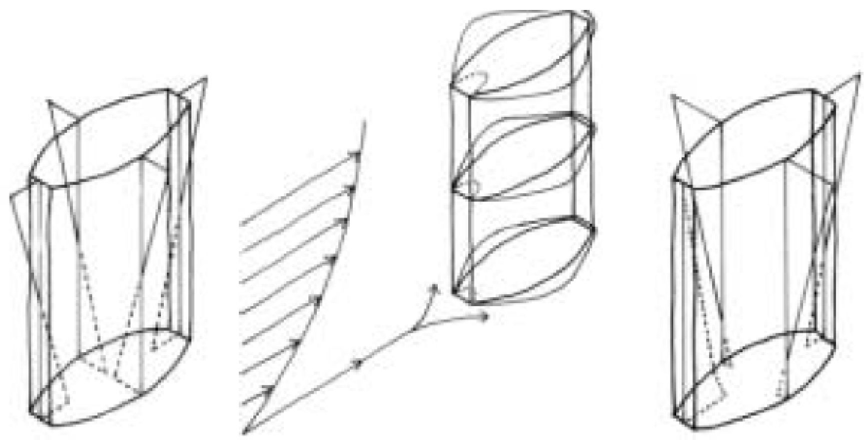


Figure 1.1D: The buoyancy-induced pressure distribution upon the envelope of a high-rise building with an oval shape. (Kleiven, 2003)

The schematic drawing in Figure 1.1D (left image) shows the buoyancy-induced pressure distribution upon the envelope of a high-rise building with an oval shape. The inward-pointing dotted lines indicate under pressure, and the outward-pointing solid lines indicate over pressure. Because of the differences in interior and exterior temperatures, a pressure differential over the building envelope is created. The schematic drawing (middle) shows the wind-induced pressure distribution upon the same high-rise building. The inward-pointing dotted lines indicate the positive pressure created on the windward side of the building envelope, and the outward-pointing solid lines indicate the under pressure created on the building envelope on the leeward side. Both the positive and the negative pressure increase towards the top of the high-rise building as a result of the illustrated wind profile. The schematic drawing (right) illustrates the combined effect of wind and buoyancy, and the distribution of pressure differentials on the building envelope. The figure illustrates that the pressure gradients derived from buoyancy and wind forces can be added. They either strengthen or neutralize each other.

The shape of a building together with the location of the ventilation openings dictates the natural ventilation's manner of operation. One usually differentiates between three different ventilation principles for natural ventilation. The ventilation principle indicates how the exterior and interior airflows are linked, and hence how the natural driving forces are utilized to ventilate a building. Furthermore, the ventilation principle gives an indication on how the air is introduced into the building, and how it is exhausted out of it.

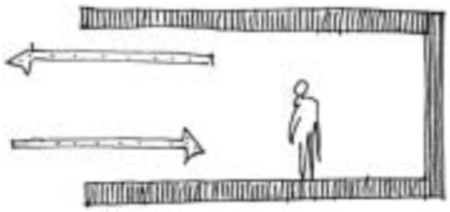


Figure 1.1E: Sketch of single sided ventilation. As a rule of thumb, single-sided ventilation is effective to a depth of about 2 – 2.5 times the floor to ceiling height. (Kleiven, 2003)

Single sided ventilation relies on opening(s) on only one side of the ventilated enclosure. Fresh air enters the room through the same side as air leaves the room. A typical example is the rooms of a cellular building with open windows on the windward side and closed internal doors on the leeward side. With a single ventilation opening in the room, the main driving force in summer is wind turbulence. In cases where ventilation openings are provided at different heights within the façade, the ventilation rate can be enhanced by the buoyancy effect. The contribution from thermal buoyancy depends on the temperature difference between the inside and the outside, the vertical distance between the openings, and the area of the openings. The greater vertical distance between the openings, and the greater temperature difference between the inside and the outside, the stronger is the effect of buoyancy. Compared with other strategies, lower ventilation rates are generated, and the ventilation air does not penetrate far into the space.

Cross – ventilation

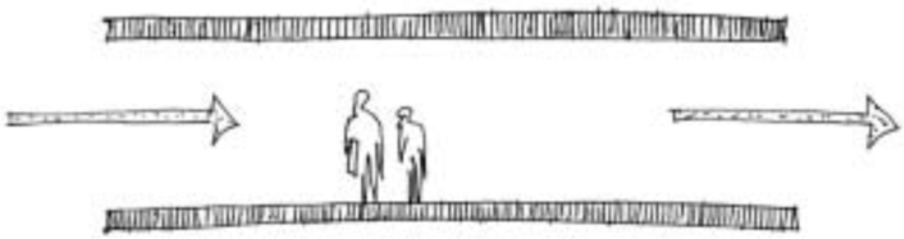


Figure 1.1F: Sketch of cross ventilation. As a rule of thumb, cross-ventilation is effective up to 5 times the floor to ceiling height. (Kleiven, 2003)

Cross-ventilation occurs when air flows between two sides of a building envelope by means of wind-induced pressure differentials between the two sides. The ventilation air enters and leaves commonly through windows, hatches or grills integrated in the façades. The ventilation air moves from the windward side to the leeward side. A typical example is an open-plan office landscape where the space stretches across the entire depth of the building. The airflow can also pass through several rooms through open doors or overflow grills. The term cross ventilation is also referred to situations when considering a single space where air enters one side of the space and exits on the opposite side. In this case the ventilation principle on the system level can be either cross- or stack ventilation. As the air moves across an occupied space, it picks up heat and pollutants. Consequently, there is a limit to the depth of a space that can be effectively cross-ventilated.

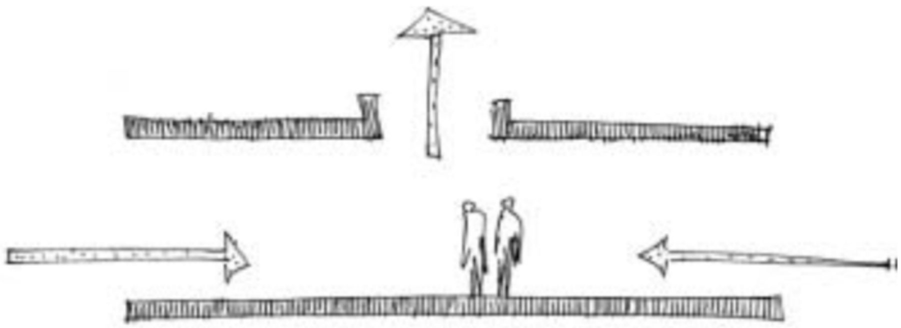


Figure 1.1G: Sketch of stack ventilation. As a rule of thumb, stack ventilation is effective across a width of 5 times the floor to ceiling height from the inlet to where the air is exhausted. (Kleiven, 2003)

Stack ventilation occurs where the buoyancy driving forces promote an outflow from the building, thereby drawing fresh air in via ventilation openings at a lower level. Fresh air typically enters through ventilation openings at a low level, while used and contaminated air is exhausted through high level ventilation openings (a reversed flow can occur during certain conditions). Designing the outlet to be in a region of wind-induced under pressure can enhance the effectiveness of stack ventilation. A typical example is a building with an elevated central part, in which warm and contaminated air from the surrounding spaces rises to be exhausted through wind towers located on the roof.

Due to its physical nature, the stack effect requires a certain height between the inlet and the outlet. This can be achieved by e.g. increasing the floor to ceiling height, tilting the profile of the roof, or applying a chimney or an atrium. By its nature, stack ventilation resembles cross ventilation as far as some individual spaces are concerned, in that air enters one side of the space and leaves from the opposite side. The air may flow across the whole width of the building and be exhausted via a chimney, or it may flow from

ASHRAE 2005 (SI) reports that airflow around buildings affects worker safety, process and building equipment operation, weather and pollution protection at inlets, and the ability to control indoor temperature, humidity, air motion, and contaminants. Wind causes variable surface pressures on buildings that change intake and exhaust system flow rates, natural ventilation, infiltration and exfiltration, and interior pressures. The mean flow patterns and turbulence of wind passing over a building can recirculate exhaust gases to air intakes. This ASHRAE handbook provides basic and fundamental information for evaluating wind flow patterns, estimating wind pressures, and identifying problems caused by the effects of wind on intakes, exhausts, and equipment.

Flow patterns: Buildings having even moderately complex shapes, such as L- or U-shaped structures formed by two or three rectangular blocks, can generate flow patterns too complex to use general design guidelines. In order to determine flow conditions influenced by surrounding buildings or topography, wind tunnel or water channel tests of physical scale models or tests of existing buildings are required. However, if a building is oriented perpendicular to the wind, it can be considered as consisting of several independent rectangular blocks. Only isolated rectangular block buildings are discussed here.

As wind impinges on a building, airflow separates at the building edges, generating recirculation zones over downwind surfaces (roof, side and downwind walls) and extending into the downwind wake (see Figure 1.1H). On the upwind wall, surface flow patterns are largely influenced by approach wind characteristics. Figure 1.1H shows that the mean speed of wind UH approaching a building increases with height H above the ground. Higher wind speed at roof level causes a larger pressure on the upper part of the wall than near the ground, which leads to downwash on the lower one-half to two-thirds of the building. On the upper one-quarter to one-third of the building, wind flow is directed upward over the roof (upwash). For a building with height H three or more times width W of the upwind face, an intermediate stagnation zone can exist between the upwash and downwash regions, where surface streamlines pass horizontally around the building, as shown in Figure 1.1H (inset) and in Figure 1.1I, the upwind building surface is “folded out” to illustrate upwash, downwash, and stagnation zones.) Downwash on the lower surface of the upwind face separates from the building before it reaches ground level and moves upwind to form a vortex that can generate high velocities close to the ground (“area of strong surface wind” in Figure 1.1H, inset). This ground-level upwind vortex is carried around the sides of the building in U shape and suspends dust and debris that can contaminate air intakes close to ground level.

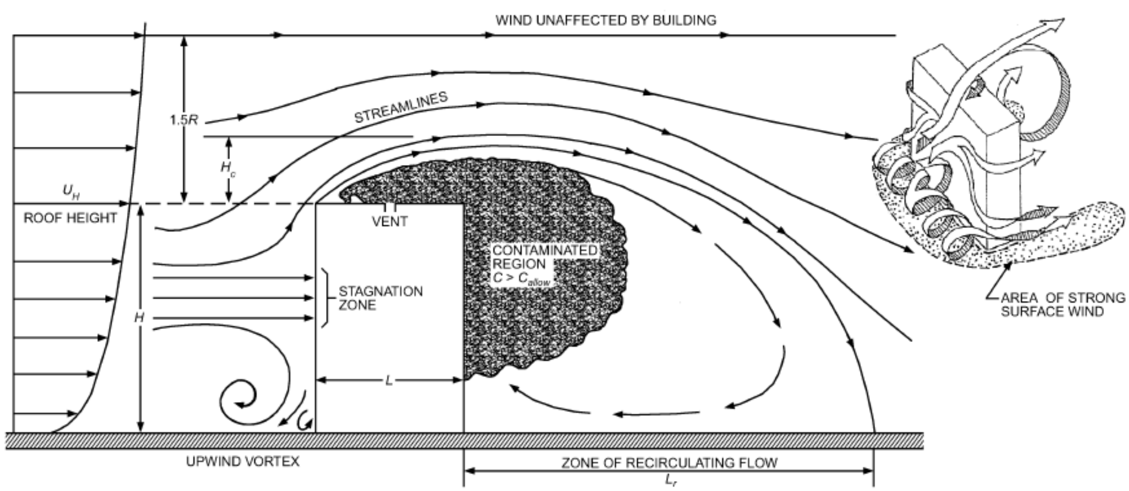


Figure 1.1H: Flow Patterns Around Rectangular Building. (ASHRAE, Chapter 16, 2005)

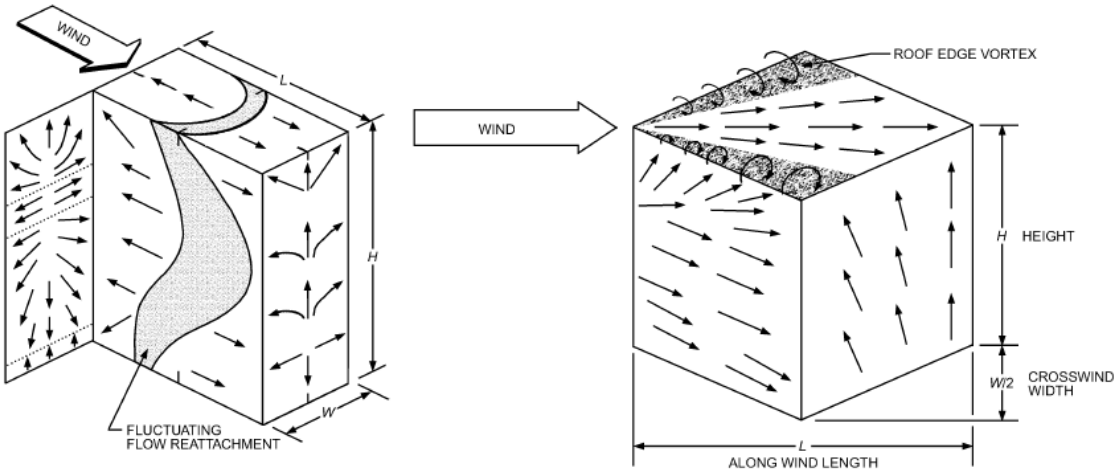


Figure 1.1I: Surface Flow Patterns and Building Dimensions (ASHRAE, Chapter 16, 2005)

The downwind wall of a building exhibits a region of low average velocity and high turbulence (i.e., a flow recirculation region) extending a distance Lr downwind. If the building has sufficient length L in the windward direction, the flow reattaches to the building and may generate two distinct regions of separated recirculation flow, on the building and in its wake, as shown in Figure 1.1I and Figure 1.1J also illustrates a rooftop recirculation cavity of length Lc at the upwind roof edge and a recirculation zone of length Lr downwind of the rooftop penthouse. Velocities near the downwind wall are typically one-quarter of those at the corresponding upwind wall location. Figures 1.1H and 1.1I show that an upward flow exists over most of the downwind walls.

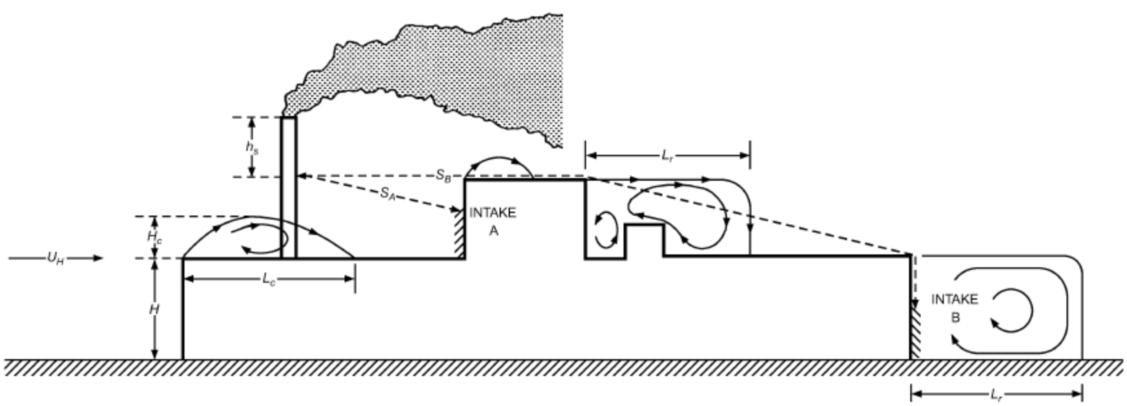


Figure 1.1.J: Flow Recirculation Regions and Exhaust-to-Intake Stretched-String Distances (ASHRAE, Chapter 16, 2005)

Streamline patterns are independent of wind speed and depend mainly on building shape and upwind conditions. Because of the three-dimensional flow around a building, the shape and size of the recirculation airflow are not constant over the surface. Airflow reattaches closer to the upwind building face along the edges of the building than it does near the middle of the roof and sidewalls (Figure 1.1.I). Recirculation cavity height Hc (Figures 1.1H and 1.1J) also decreases near roof edges.

Local Wind Pressure Coefficients: Values of the mean local wind pressure coefficient Cp used in Equation (3) depend on building shape, wind direction, and influence of nearby buildings, vegetation, and terrain features. Accurate determination of Cp can be obtained only from wind tunnel model tests of the specific site and building. Ventilation rate calculations for single, unshielded rectangular buildings can be reasonably estimated using existing wind tunnel data.

$$p_s = C_p p_v \qquad (3)$$

Equation (3) is the proportional relationship is shown in the following equation, in which the difference ps between the pressure on the building surface and the local outdoor atmospheric pressure at the same level in an undisturbed wind approaching. (ASHRAE, Chapter 16, 2005)

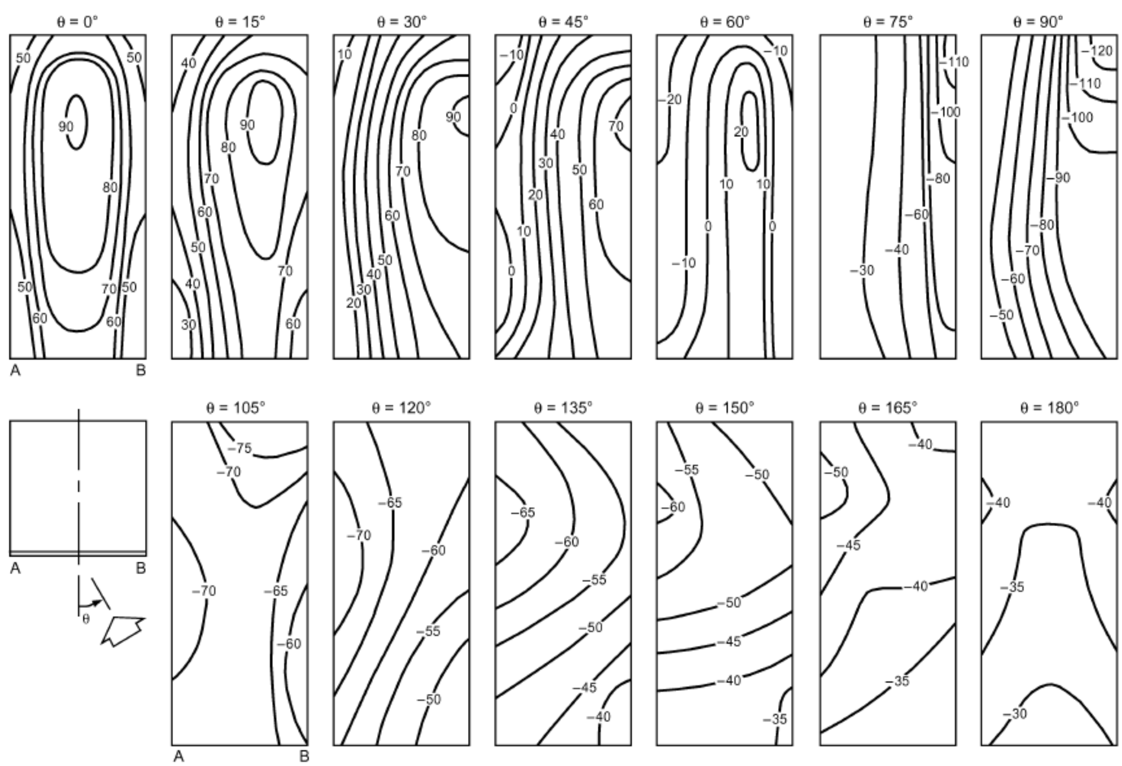


Figure 1.1.K: Local Pressure Coefficients (Cp × 100) for Tall Building with Varying Wind Direction (ASHRAE, Chapter 16, 2005)

Figure 1.1.K shows pressure coefficients for walls of a tall rectangular cross section building (high-rise) sited in urban terrain. Figure 1.1L shows pressure coefficients for walls of a low-rise building. Generally,

for high-rise buildings, height H is more than three times the crosswind width W. At a wind angle θ = 0° (e.g., wind perpendicular to the face in question), pressure coefficients are positive and their magnitudes decrease near the sides and the top as flow velocities increase. As seen in Figure 1.1.K, Cp generally increases with height, which reflects increasing velocity pressure in the approach flow as wind speed increases with height.

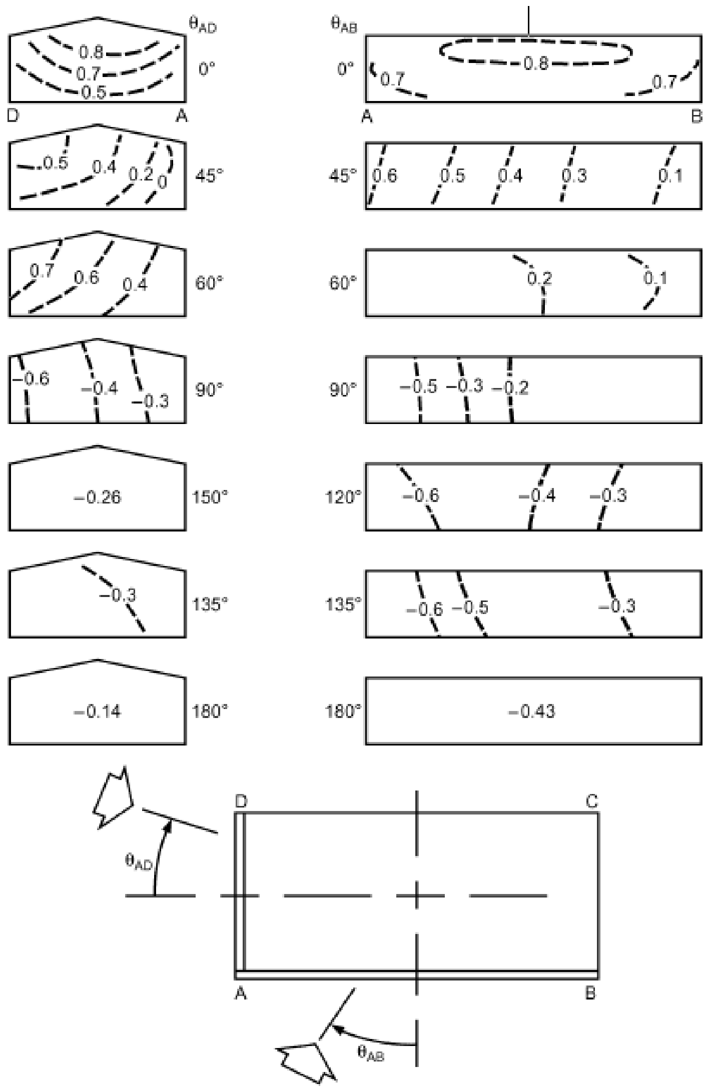


Figure 1.1.L: Local Pressure Coefficients for Walls of Low-Rise Building with Varying Wind Direction (ASHRAE, Chapter 16, 2005)

As wind direction moves off normal (θ = 0°), the region of maximum pressure occurs closer to the upwind edge (B in Figure 1.1K) of the building. At a wind angle of θ = 45°, pressures become negative at the downwind edge (A in Figure 1.1K) of the front face. At some angle θ between 60° and 75°, pressures become negative over the whole front face. For θ = 90°, maximum suction (negative) pressure occurs near the upwind edge (B in Figure 1.1K) of the building side and then recovers towards Cp = 0 towards the downwind edge (A in Figure 1.1K). The degree of this recovery depends on the length of the side in relation to the width W of the structure. For wind angles larger than θ = 100°, the side is completely within the separated flow of the wake and spatial variations in pressure over the face are not as great. The average pressure on a face is positive for wind angles from θ = 0° to almost 60° and negative (suction) for θ = 60° to 180°.A similar pattern of behavior in wall pressure coefficients for a low-rise building is shown in Figure 1.1L. Here, recovery from strong suction with distance from the upwind edge is more rapid.

1.2 Effects of Wind Movement Patterns around Buildings and on Pedestrian Comfort

The literature describes concept study and observation of wind-driven rain (WDR) as well as methodology to infer valuable velocity and pressure information on building design. Wind-driven rain research is of importance in a number of research areas including wind movement pattern around building in both pedestrian and elevated level, earth sciences, meteorology and building science.

Blocken and Carmeliet (2004) define wind-driven rain (WDR) or driving rain as rain that is given a horizontal velocity component by the wind and that falls obliquely. WDR over unlevel ground such as hills or valleys results in a redistribution of raindrops by local wind flow deformations that can cause large precipitation variations. WDR is also an important research subject in building science. It is the most important moisture source affecting the hygrothermal performance and durability of building facades. Blocken and Carmeliet (2004)'s suggest that WDR causes an oblique rain intensity vector (see Figure 1.2A) between rain vector and the vertical building facade. On the other hand, the component of the rain intensity vector causing rain flux through vertical plane is horizontal rainfall intensity.

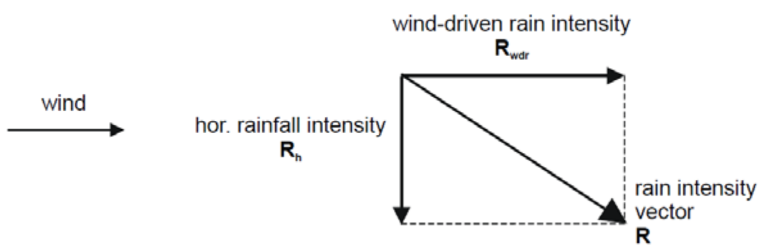


Fig. 1.2.A: Rain intensity vector R and its components: wind-driven rain intensity Rwdr and horizontal rainfall intensity Rh (Blocken and Carmeliet, 2004)

Figure 1.2.B presents wind flow pattern around a building and of raindrop trajectories in the wind flow pattern. When wind approaches a building, a disturbance is generated and a specific flow pattern develops around it, including a frontal vortex, separation at building corners, corner streams, recirculation zones, shear layers and a far wake. When rain is added to the flow field, it will be driven against the windward facade of the building. As a result of the specific flow features, the course of the raindrop trajectories is changed which results in a non-uniform wetting of the facade.

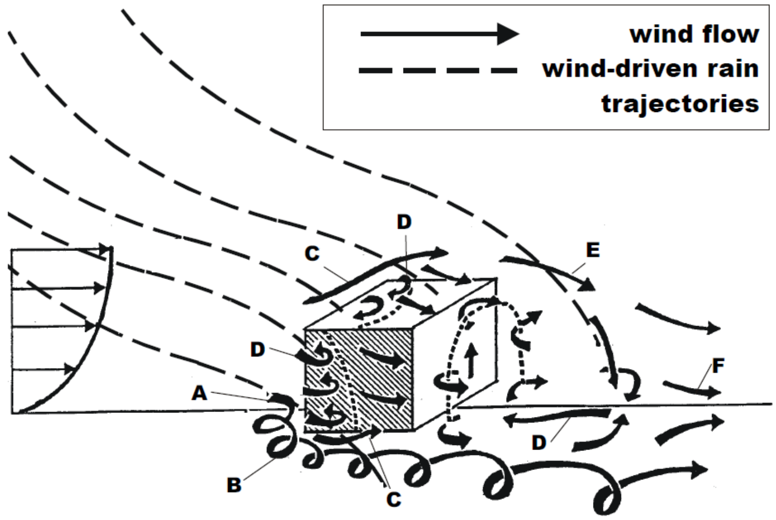
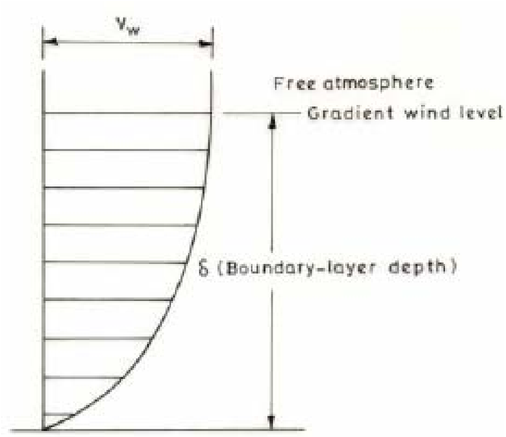


Fig. 1.2.B: A representation of the wind flow pattern around a building and of raindrop trajectories in the wind flow pattern. The flow pattern includes a frontal vortex (A), corner streams (B), separation at building corners (C), recirculation zones (D), shear layers (E) and a far wake (F) (Blocken and Carmeliet, 2004)

Ahuja et al. (2006) describes that wind velocity close to the earth surface is close to zero and increases with height. However, construction of tall buildings in a locality of low-rise buildings can alter street level wind environment. Wind striking tall building is deflected downwards to the ground causing high speed winds on the windward side as well as near the corners of buildings at street / pedestrian level. This leads to discomfort to the pedestrian walking and can even represent a safety concern for cyclists or other two-wheel vehicle drivers. Ahuja et al. (2006) describes the comfort criteria for pedestrians within a built environment and also enumerates the recent research work done in this area.

Pedestrian Level Wind: As can be seen in Figure1.2.C, wind velocity close to the ground approaches zero and increases with height. The height above which there is a constant velocity is called boundary layer depth and corresponding velocity of wind above the boundary layer is referred to as free stream velocity. However, construction of tall buildings in a locality of low-rise buildings alters the street level wind environment.



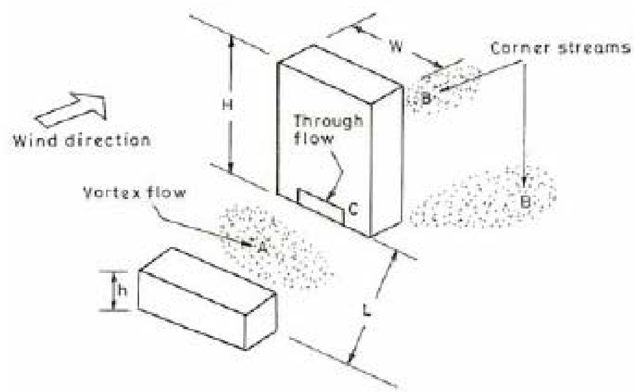


Figure 1.2.C (Left): The atmospheric boundary layer and
Figure 1.2.D (Right): Regions of high surface wind speeds around a tall building (Ahuja and etc. ,2006)

Comfort Criteria : As mentioned above, presence of one or few tall buildings near groups of low-rise buildings result in high speed wind in the passages and streets around tall buildings. This leads to pedestrian discomfort and the referred to safety concerns for cyclists. Comfort or discomfort is an abstract as well as relative phenomenon. It is not possible to exactly define comfort or discomfort level of various physical quantities like temperature, humidity and wind speed. Whereas an individual may feel quite comfortable at a temperature and humidity condition, another individual may complain for his/her discomfort. Similarly wind speed, which may cause discomfort to one person may be acceptable to other people. It is not only a particular value of wind speed, which may cause discomfort, but there are other parameters which combine result in discomfort. They parameters include gustiness, duration, mean wind speed, frequency of occurrence and also the fact that the mean wind speed is achieved in a short duration, i.e. relatively sudden or over a long duration. Generally a wind speed (V) above 5m/s is considered as uncomfortable wind speed. However wind speed above 10 m/s almost always causes discomfort and a wind speed above 20 m/s is dangerous (Table 1.2.A).

Wind tunnel studies: Review of codes of practice of various countries dealing with wind load indicates that there is no such provision to consider an unwind speed in the streets and passages due to the construction of tall buildings in the vicinity of low rise buildings. However, some researchers have carried out experimental study in the wind tunnels in order to suggest certain reproducible design standards in order to avoid pedestrian discomfort. These design features describe wind movements around buildings in general and for high rise buildings in particular, which might cause discomfort to the pedestrians around such buildings. The author has also suggested the remedial measures, as detailed below, so as to minimize the discomfort.

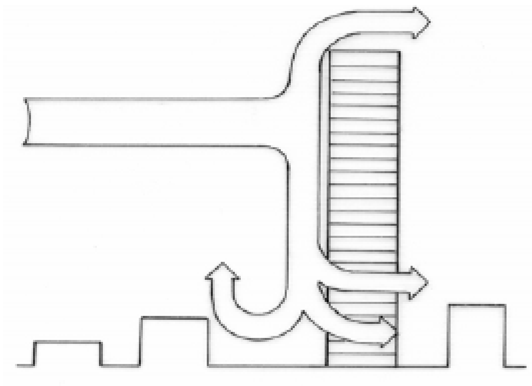
Table 1.2.A: Summary of wind effect (Ahuja and etc. ,2006)

Description of wind	Speed (m/sec)	Description of wind effects
Calm	Less than 0.4	No noticeable wind.
Light airs	0.4-1.5	No noticeable wind.
Light breeze	1.6-3.3	Wind felt on face.
Gentle breeze	3.4-5.4	Wind extends light flag.
		Hair is disturbed.
		Clothing flaps.
Moderate breeze	5.5-7.9	Wind raises dust, dry soil, and loose paper. Hair disarranged.
Fresh breeze	8.0-10.7	Force of wind felt on body.
		Drifting snow becomes airborne.
		Limit of agreeable wind on land.
Strong breeze	10.8-13.8	Umbrellas used with difficulty.
		Hair blown straight.
		Difficulty to walk steadily.
		Wind noise on ears unpleasant.
		Windborne snow above head height (blizzard).
Moderate gale	13.9-17.1	Inconvenience felt when walking.
Fresh gale	17.2-20.7	Generally impedes progress.
		Great difficulty with balance in gusts.
Strong gale	20.8-24.4	People blown over by gusts.

<u>Local topography:</u> Local topography is one of the factors, which will have an impact on the wind conditions around a building.

<u>Downwash:</u> When a stream of wind strikes the surface of a tall building, a significant portion of the incident wind moves downward thus reaching of street / pedestrian level; such a mechanism being referred to as downwash (Figure 1.2.E). Ground-level corners of the building (Figure 1.2.F) are also subjected to high wind due to this reason.

<u>Effect of canopy:</u> In high wind areas, once the wind reaches the ground it is accelerated around the ground-level corners. A large canopy may interrupt the flow as it moves down the windward face of the building. This will protect the entrances and sidewalk area by deflecting the downwash at the second story level. But, this approach may have the effect of transferring the breezy conditions to the other side of the street. The large canopies are a common feature near the main entrance of office buildings (Figure 1.2.G).



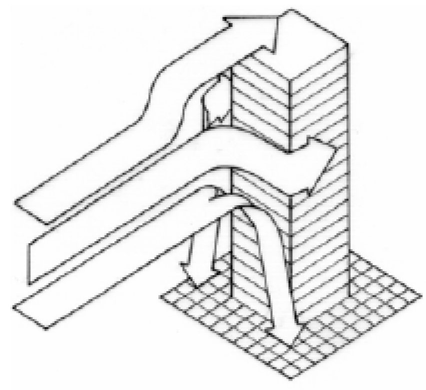
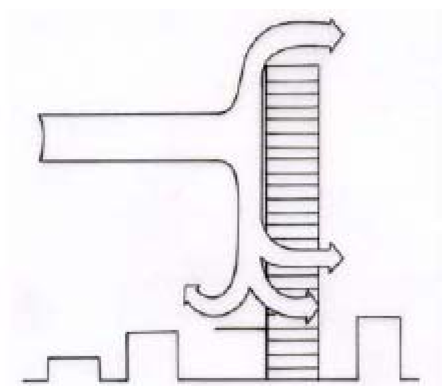


Figure 1.2.E (Left): Downwash to street level
Figure 1.2.F (Right): High wind at the ground-level (Ahuja and etc. ,2006)

Effect of podium: High-rise buildings might have a raised platform, i.e. a podium (Figure 1.2.H), if the development site conditions allow it and the design complies with the design mandate. This will reduce wind speed at the pedestrian level. However, it may be counterproductive if the architect wishes to use the podium roof for long-term activity such as pool or tennis court.



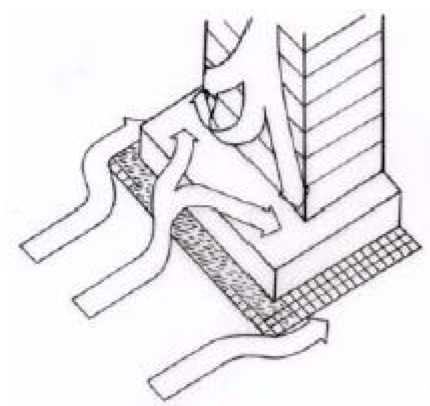
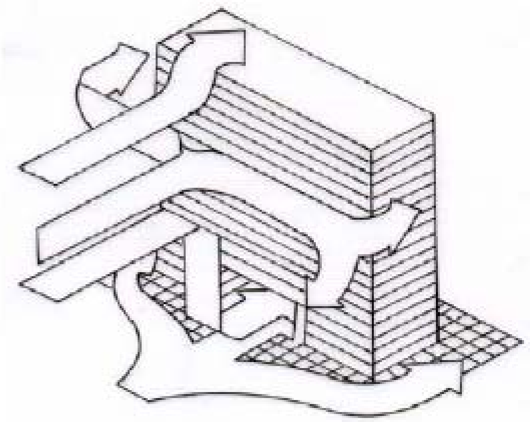


Figure 1.2.G (Left): A large canopy as a solution to the pedestrian-wind problem
Figure 1.2.H (Right): The Tower-on-podium (Ahuja and etc. ,2006)

Effect of arcade: An arcade or thoroughfare opening from one side of the building to the other (Figure 1.2.I) effectively connects a positive pressure region on the windward side with a negative pressure region on the leeward side. It often results in a strong flow through the opening. Similar phenomenon occurs with a high-rise building is raised up on columns.

Effect of alcove entrance: An entrance alcove behind the building line at a mid-building location (Figure 1.2.J) will generally produce a calmer entrance area. In some cases a canopy may not be necessary with this scenario, depending on the local geometry and directional wind characteristics.



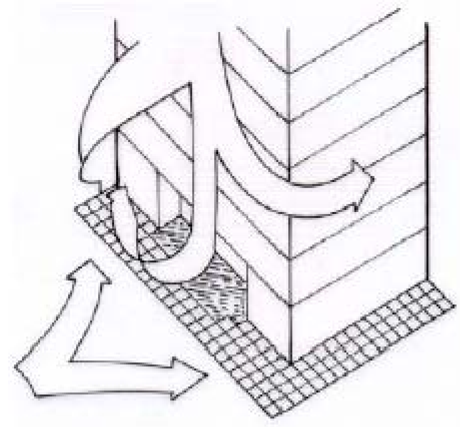
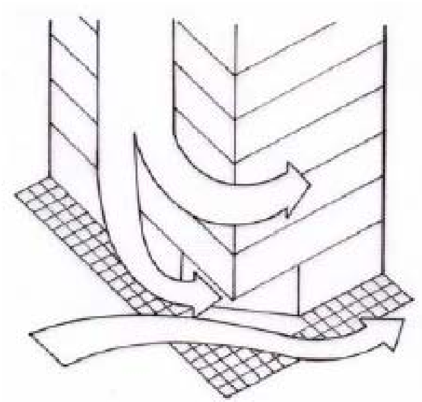


Figure 1.2.I (Left): An arcade in a building
Figure 1.2.J (Right): A mid-building alcove entrance (Ahuja and etc. ,2006)

Effect of corner cut: The building designer might prefer to provide entrance door to a building fronting on two adjacent streets by making a cut at the corner of the building (Figure 1.2.K). If there is strong directional wind preference at the site and the corner door is shielded from those common stronger winds, then the corner entrance can be an effective design solution. However, it is more common for a corner entrance to be adversely impacted by the local building geometry and the strong winds that may occur in the city, both influencing the exposed corner entrance.

Effect of landscaping: Horizontally accelerated flows between two tall towers may cause an unpleasant windy ground level pedestrian environment (Figure 1.2.L), which could also be aggravated by ground topography. By inspection of the available wind data, the designer may find a dominant wind direction that can be used to align the building on the site so as minimize these accelerated flows in highly populated pedestrian areas. Use of porous screens and proper plantation can also improve wind environment.



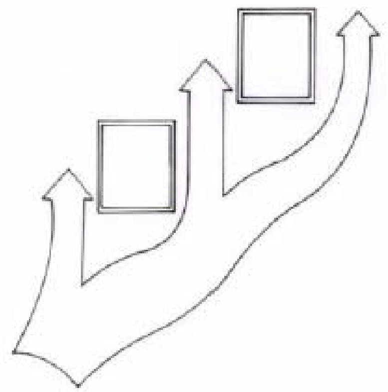


Figure 1.2.K (Left): Accelerated corner flow from downwash
Figure 1.2.L (Right): Accelerated horizontal wind between the building (Ahuja and etc. ,2006)

1.3 Atmospheric Boundary Layer

ASHRAE 2005 (SI) reports that airflow around buildings can affect worker safety, process and building equipment operation, weather and pollution protection at inlets, and the ability to control indoor temperature, humidity, air motion, and contaminants. Wind causes variable surface pressures on buildings that change intake and exhaust system flow rates, natural ventilation, infiltration and exfiltration, and interior pressures. The mean flow patterns and turbulence of wind passing over a building can recirculate exhaust gases to air intakes. ASHRAE (2005) provides basic and fundamental information for evaluating windflow patterns, estimating wind pressures, and identifying problems caused by the effects of wind on intakes, exhausts, and equipment.

The wind boundary layer thickness δ and exponent a for the local building terrain and a_{met} and δ_{met} for the meteorological station are determined from Table 1.3.A. Typical values for meteorological stations, generally measured in flat, open terrain (category 3 in Table 1.3A), are a_{met} = 0.14 and met = 270 m. The values and terrain categories in Table 1.3.A are consistent with those adopted in other engineering applications (e.g., ASCE Standard 7). Equation (4) gives the wind speed at height H above the average height of local obstacles, such as buildings and vegetation, weighted by the plan-area. At heights at or below this average obstacle height (e.g., at roof height in densely built-up suburbs), speed depends on the geometrical arrangement of the buildings, and Equation (4) is less reliable.

Mendis et al (2007) describe methods of simple quasi-static treatment of wind loading. These methods can be universally applied to the design of typical low to medium-rise structures, can they can be unacceptable, or too conservative, for the design of very tall buildings. The authors report that such simple treatment can easily lead to erroneous results and under-estimations of wind induced phenomena. More important, such a simplified treatment for deriving lateral loads does not address key design issues including dynamic response (effects of resonance, acceleration, damping, structural stiffness), interference from other structures, wind directionality, and cross wind response, which are all important factors in wind design of tall buildings.

Table 1.3.A: Atmospheric Boundary Layer Parameters (ASHRAE, Chapter16, 2005)

Terrain Category	Description	Exponent a	Layer Thickness δ, m
1	Large city centers, in which at least 50% of buildings are higher than 21.3 m, over a distance of at least 0.8 km or 10 times the height of the structure upwind, whichever is greater	0.33	460
2	Urban and suburban areas, wooded areas, or other terrain with numerous closely spaced obstructions having the size of single-family dwellings or larger, over a distance of at least 460 m or 10 times the height of the structure upwind, whichever is greater	0.22	370
3	Open terrain with scattered obstructions having heights generally less than 9.1 m, including flat open country typical of meteorological station surroundings	0.14	270
4	Flat, unobstructed areas exposed to wind flowing over water for at least 1.6 km, over a distance of 460 m or 10 times the height of the structure inland, whichever is greater	0.10	210

$$U_H = U_{met}\left(\frac{\delta_{met}}{H_{met}}\right)^{a_{met}}\left(\frac{H}{\delta}\right)^{a} \quad (4)$$

The paper provides an outline of advanced levels of wind design, in the context of the Australian Wind Code, and illustrates the exceptional benefits it offers over simplified approaches. Wind tunnel testing, which has the potential benefits of further refinement in deriving design wind loading and its effects on tall buildings, is also emphasized.

Wind Speed: At great heights above the surface of the earth, where frictional effects are negligible, air movements are driven by pressure gradients in the atmosphere, which in turn are the thermodynamic consequences of variable solar heating of the earth. This upper level wind speed is known as the gradient wind velocity. Different terrains can be categorized according to their associated roughness length.

Table 1.3.B shows the different categories specified in the Australian/New Zealand wind code, AS/NZS1170.2 (2002). Closer to the surface the wind speed is affected by frictional drag of the air stream over the terrain. There is a boundary layer within which the wind speed varies from almost zero, at the surface, to the gradient wind speed at a height referred to as the gradient height. The thickness of this boundary layer may vary from 500 to 3000 meter, depending on the type of terrain, as depicted in Fig. 1.3.A. As can be seen the gradient height within a large city center is much higher than it is over the sea where the surface roughness is less.

In practice, it has been found useful to start with a reference wind speed based on statistical analysis of wind speed records obtained at meteorological stations throughout the country. The definition of the reference wind speed varies from one country to another. For example in Australia/New Zealand, it is the 3-sec gust wind speed at a height of 10 m above the ground assuming terrain category 2 (Table 1.3B). Contour maps of reference wind speeds that apply for nominated statistical Return Periods in various countries are usually available. An engineering wind model for Australia has been developed in Melbourne from the Deaves and Harris model (1978). This model is based on extensive full-scale data and on the classic logarithmic law in which the mean velocity profile in strong winds applicable in non-cyclonic regions (neutral stability conditions).

Table 1.3.B: Terrain category and roughness length (zo) (Mendis and etc, 2007)

Terrain category	Roughness length, z_0, (m)
1. Exposed open terrain with few or no obstructions and water surfaces at serviceability wind speeds.	0.002
2. Water surfaces, open terrain, grassland with few, well scattered obstructions having heights generally from 1.5 to 10 m.	0.02
3. Terrain with numerous closely spaced obstructions 3 to 5 m high such as areas of suburban housing.	0.2
4. Terrain with numerous large, high (10.0 m to 30.0 m high) and closely spaced obstructions such as large city centres and well-developed industrial complexes.	2

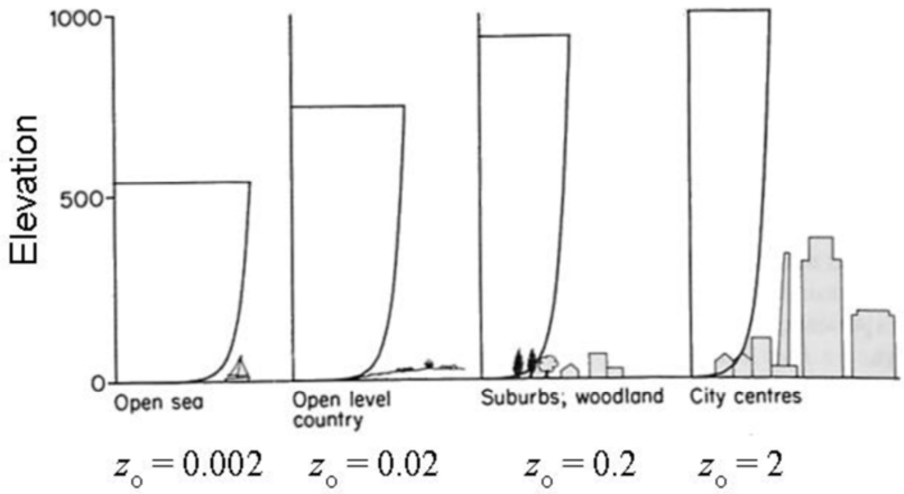


Figure 1.3.A: Mean wind profiles for different terrains (Mendis and etc, 2007)

Table 1.3C: Roughness length, friction velocity and gradient height (Mendis and etc, 2007)
.

Terrain Category	z_o (m)	u* (m/s)	Z_g (m)
1	0.002	1.204	2006
2	0.02	1.385	2308
3	0.2	1.626	2710
4	2	1.963	3272

As given in Table 1.3.C, there is an interaction between roughness length and terrain category, so it is necessary to define a terrain category to find the design hourly wind speeds and gust wind speeds.

For design, the basic wind speed is classified into three different speeds as follows:

- Vs = V20yr = serviceability limit state design speed having an estimated probability of exceedence of 5 % in any one year, that is adopted for the serviceability limit states.
- Vp = V50yr = permissible, or working stress design wind speed which can be obtained directly from Vu using the relation Vp = Vu/(1.5)0.5

- Vu = V1000yr = ultimate limit state design wind speed having an estimated probability of exceedence of 5 % in a lifetime of 50 years, for the ultimate limit states.

Basic design wind speeds for different directions and different return periods can be derived using a rigorous analysis incorporating probability distributions for wind speed and direction. For example AS/NZS1170.2 standards provide wind direction multipliers, which vary between 0.80 for wind from the East to 1.0 for wind from the West. The standards consider wind speeds up for a 2000 year recurrence period.

Crastto (2007) has focused on numerical simulations of the Atmospheric Boundary Layer (ABL). Investigations of the ABL can be seen as part of a more general framework of the Computational Wind Engineering (CWE). The author suggests that the drastic increase of computational capacities and the relatively high price of wind tunnel facilities promote a shift in wind engineering to a wider usage of computational resources rather than experimental facilities. The numerical simulations of the ABL are applied to the evaluation of the wind loads on structures, pollutants dispersion, the natural ventilation of buildings, the assessment of the pedestrian comfort in an urban environment, aerodynamics of sails and many other topics. For instance, CFD computations on given terrains coupled with the meteorological wind data are becoming indispensable design and planning tools for projects that are significantly dependent on the wind environment, such as planning of wind farms.

Specific numerical tools that can evaluate the expected Annual Energy Production (AEP) of a wind farm are typically based on empirical relations. The empirical relations commonly used for the computation of the speed-up due to the topography are prone to fail or loose accuracy when dealing with complex terrains. A terrain can be considered complex when its steepness exceeds a certain threshold and above this level the empirical relations don't guarantee applicability. In fact, in the flow around steep terrains, hills separation air flow region are very likely to occur and the level of turbulence in these regions can be high. Furthermore also unsteady phenomena like shedding of vortices can occur. This kind of flows leads the commonly used empirical models to fail and the numerical solution of the RANS equations remains therefore only analysis able to reproduce the correct behavior of the flow.

Crastto (2007) also focuses on the application of two commercial finite volume CFD codes for the numerical simulations of the ABL flow. The first one is the more general finite volume solver Fluent and the second one, WindSim, is devoted to wind energy assessment using the general RANS finite volume solver PHOENICS.

Atmospheric Boundary Layer (ABL): Crastto (2007) suggests that the Atmospheric Boundary Layer (ABL) or Planetary Boundary Layer (PBL), can extent to the lowest 1-2 km of the atmosphere. This region is most directly influenced by the exchange of momentum, heat, and water vapour at the earth's surface. The author defines the ABL as "that part of the atmosphere that is directly influenced by the presence of the earth's surface, and responds to surface forcings with a timescale of about an hour or less". Moreover there are several physical occurrence that act on the boundary layer, such as frictional drag, evaporation, heat and pollutants transfer, modifications of the flow due to topography. The ABL depth ranges from hundreds of meters to a few kilometers, depending on the physical parameters involved. In regard to the time-based flows and phenomena inside an ABL a typical time scale is one hour while a typical spatial scale is few kilometers. KNoeledge of these average time and spatial scales are important when numerical simulations of the ABL have to be performed

Vertical Velocity Profile in ABL: A neutral ABL, where heat transfer is negligible, can be subdivided into several sub-layers, as illustrated in Figure 1.3.B: a canopy layer attached to the ground surface, where the obstacles constituting the roughness elements are displaced; above the canopy layer there is the surface layer, where the effects of Coriolis force are still negligible. The surface layer is typically considered to extend upwards to one tenth of the total height of the ABL. The outer layer of the ABL, the so-called Ekman layer, is affected by the rotation of the Earth by means of Coriolis forces. A rotation of the wind direction is therefore observed, gradually passing from a crossing-isobars wind in the boundary layer to a parallel isobars wind (Geostrophic wind) in the free atmosphere where frictional forces are negligible. The crossing isobars flow and the Geostrophic wind are due to the balance of friction, pressure gradient and Coriolis forces. Inside the surface layer the only important forces are friction and pressure gradient therefore there are no observable or significant variation of horizontal wind direction within it. Geostrophic wind and wind in the Ekman layer are sketched in Figure 1.3.C.

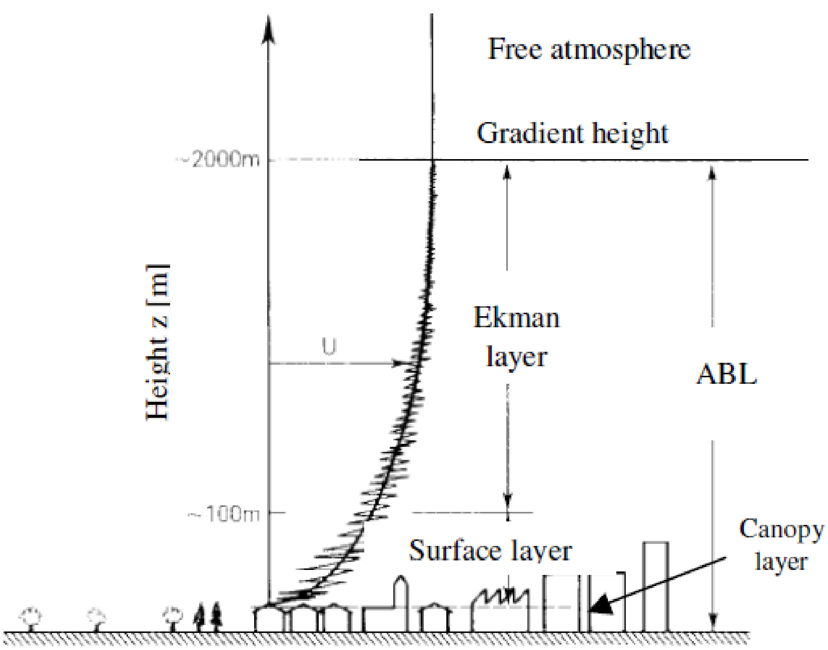


Figure 1.3B: Subdivision of the ABL (or PBL) into further sub-layers (Crastto, 2007)

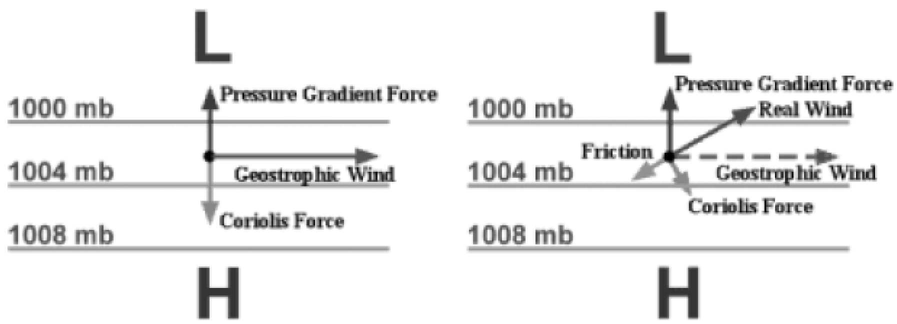


Figure 1.3C: Geostrophic wind and wind in the Ekman layer (Crastto, 2007)

If Coriolis force, as well as friction and pressure gradients are responsible for the wind flow in the outer layer and free atmosphere, in the surface layer the Coriolis force loses its importance while the roughness of the ground becomes a more significant parameter, affecting both the velocity profile and the angle of incidence of wind at the ground level and the isobars. The roughness of the terrain influences the depth of the ABL as it's sketched in Figure 1.3D, the rougher the terrain and the higher the ABL.

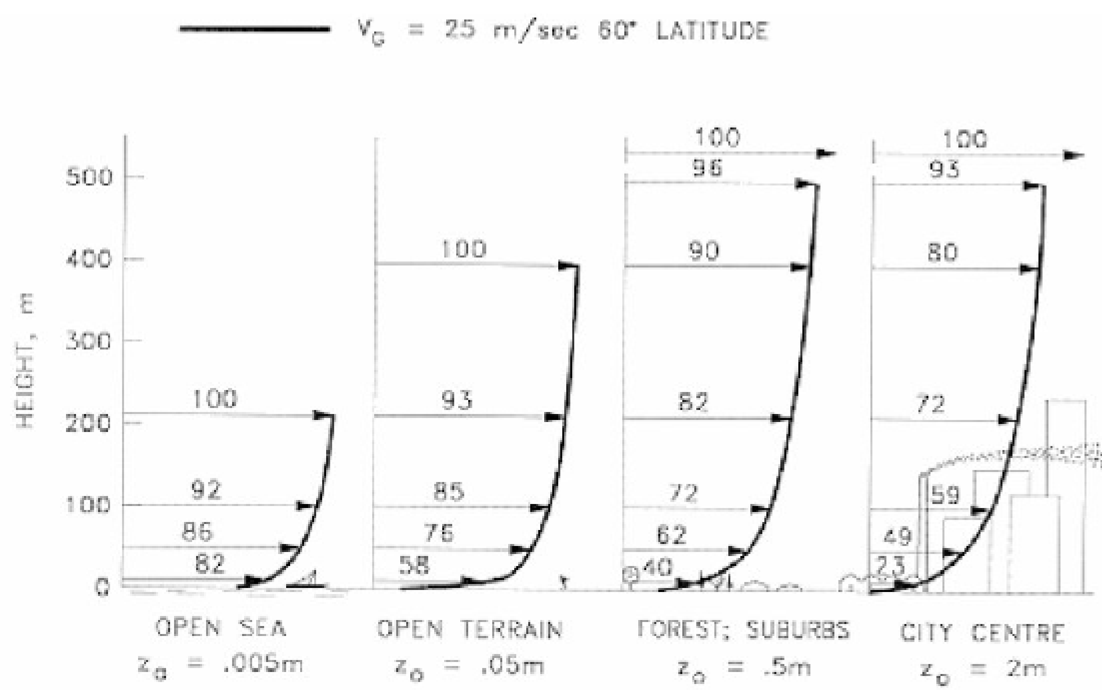


Figure 1.3.D: Different velocity profiles for four kinds of terrain typologies. (Crastto, 2007)

2. Assessment Methods of Air Movements around Buildings

This section reviews literature about assessment methods of wind movement phenomena around buildings and at a neighborhood level. The literature cited and reviewed contains pertinent information for present CFD project.

2.1 Approaches to Predict Air Movement and Pressure Distribution around Buildings

Kotani and Yamanaka (2007) described wind pressure coefficients as the basic driving force of wind-induced natural ventilation. The wind velocity along the building wall is important for wind environment and ventilation driving force at openings in the building envelope. Experimental investigation have revealed that the façade type, the shape of veranda and wind direction affect the wind movement adjacent to the building. However, wind pressure coefficient on the building envelope is not significantly influenced by the type of veranda, parapet or fence. Basically, the distribution of wind pressure coefficient on the building façade depends mainly on the direction of the incident wind vector.

Kotani and Yamanaka (2007) have prepared values for wind pressure coefficient for the various building shapes and wind directions. The authors compiled the results of the experiments into proposed regression models for the wind pressure coefficients. These experiments were conducted for the buildings with and without façade elements. Therefore the influence of the building façade elements on the wind pressure coefficients and the wind velocities along the building wall could be investigated. These two set so values of a five-storied apartment building with/without balcony (veranda) are measured by wind tunnel test using a scaled model.

Experiment Method: Kotani and Yamanaka (2007) report on a 1/60 scaled apartment building wind tunnel test as shown in Figure 2.1A. An atmospheric boundary layer wind tunnel with approaching air flow of 1/5 power law was used in the experiments. Figure 2.1B shows the profiles of velocity and turbulent intensity of it. Here the height of the building model is 25 cm. Large-scale turbulence is generated by a windward lattice, and roughness elements on the tunnel floor generate the small-scale turbulence and the velocity profile of boundary layer. The reference external wind velocity is 10 m/s at 900 mm height above the tunnel floor. The building model is assumed as a five-storied apartment building with ten rooms on each floor. Figure 2.1C shows the building model and its measurement points. Veranda and partition wall as the building façade elements can be installed and removed according with the experimental conditions.

Wind pressure coefficient and Wind velocity along building wall: Wind pressure coefficients were measured at 50 points on the building wall (see Figure 2.1C). The pressures were measured by the pressure transducer during 30 seconds with its sampling frequency of 100 Hz and those averaged values at each measurement point were calculated. The wind pressure coefficients were obtained from dividing these values by the dynamic pressure at the building height that was calculated from measured reference wind velocity.

Table 2.1A shows the experimental conditions. The façade types, the shapes of veranda and external wind directions are changed. For the shapes of veranda, parapet means the blocked wall and fence has continuous vertical gap with its porosity of about 90 % (see Figure 2.1C). Partition wall are assumed between neighbors located inside the veranda. The definition of wind direction is shown in Figure 2.1D.

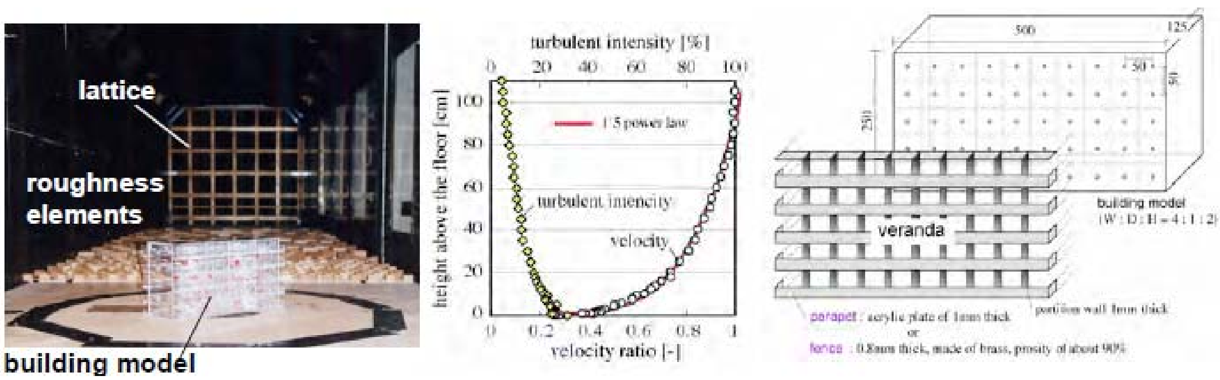
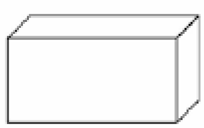
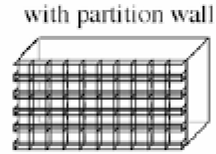


Figure 2.1A (Left): Wind tunnel set up Figure 2.1B (Mid): Approaching flow
Figure 2.1C(Right): Building model and measurement points (Kotani and Yamanaka, 2007)

Table 2.1A: Experiment condition (Kotani and Yamanaka, 2007)

facade	plane 	veranda 	veranda with partition wall 
veranda		parapet, fence	
wind direction	0, 22.5, 45, 67.5, 90, 135, 180 degrees		

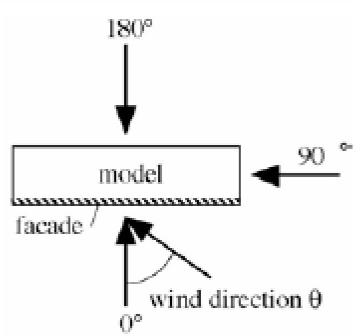


Figure 2.1D: Definition of wind direction (Kotani and Yamanaka, 2007)

Wind velocities along the building walls were measured at 50 points in front of the wind pressure measurement points. The velocities were measured by the hot-wire anemometer during 30 seconds with its sampling frequency of 100 Hz and those averaged values at each measurement point were calculated. Figure 2.1.E shows the measurement setup and measurement point of wind velocity. The hot-wire anemometer has I-type probe and the wire is always set perpendicular to the wall to measure the wind velocity parallel to the wall. From the previous detailed velocity measurements in the vicinity of wall, the boundary layer of the wall was determined as less than 5 mm, so the measurement point of wind velocity was set at 5 mm from the wall. In all cases with/without veranda, this measurement point was used. Obtained wind velocities were normalized by the wind velocity at the building height that was calculated from measured reference wind velocity.

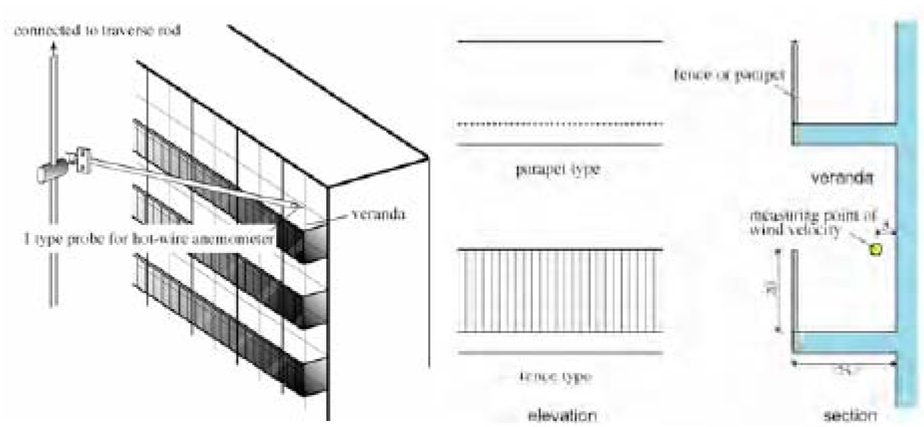


Figure 2.1.E: Setup and measurement point of wind velocity (Kotani and Yamanaka, 2007)

Example of results and discussions

Wind pressure coefficient: Figure 2.1.F shows the distribution of wind pressure coefficient in the case of wind direction of 0 degrees with veranda of parapet type. The view of figures is the elevation from the front of the building and white horizontal lines indicate the bottom height of the parapet for the case without partition, and the bottom height of the parapet and partition position for the case with partition. Significant differences cannot be seen among cases. There is a slight difference in the case without partition. The wind pressure coefficient in the upper corner shows rather small value, because the impinging air with high velocity around the stagnation area tends to flow to the side of the building along the parapet slab.

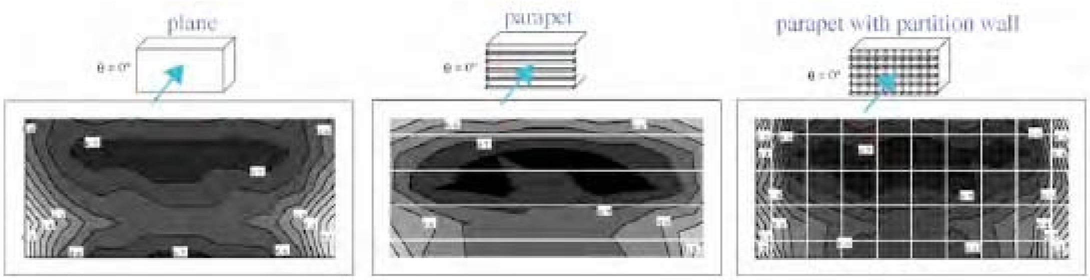


Figure 2.1.F: Distribution of wind pressure coefficient (wind direction: 0 deg. Veranda type: parapet) (Kotani and Yamanaka, 2007)

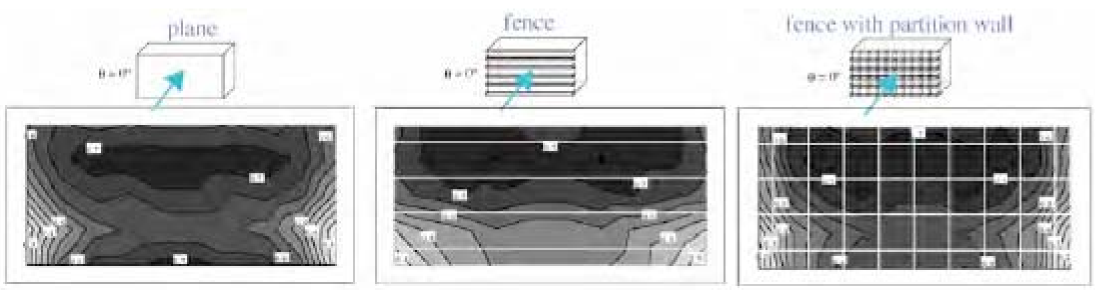


Figure 2.1.G: Distribution of wind pressure coefficient (wind direction: 0 deg. Veranda type: fence) (Kotani and Yamanaka, 2007)

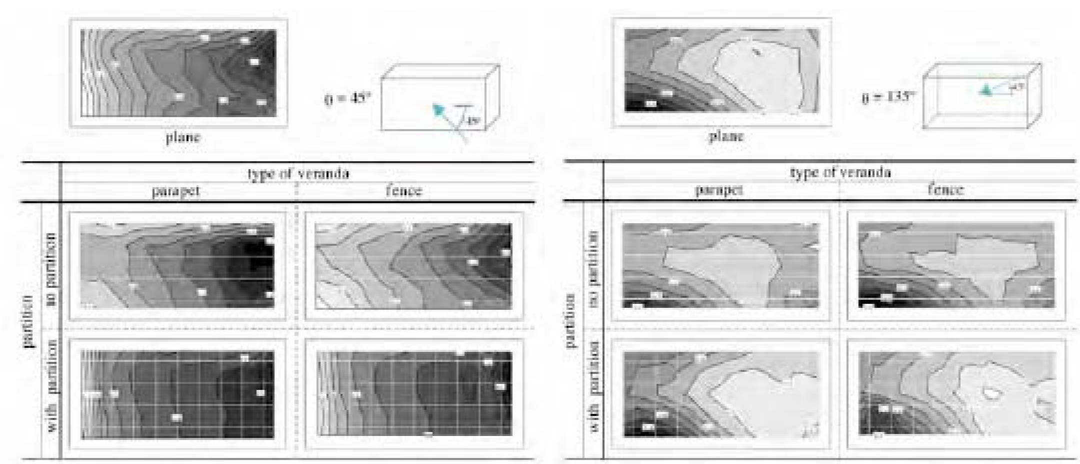


Figure 2.1.H (Left): Distribution of wind pressure coefficient (wind direction: 45 deg)
Figure 2.1.I (Right): Distribution of wind pressure coefficient (wind direction: 135 deg) (Kotani and Yamanaka, 2007)

Wind velocity along building walls: Figure 2.1.J shows the distribution of normalized wind velocity in the case of wind direction of 0 degrees. The left images in the figure show the distribution of coefficients on the façade, the right one shows the horizontal distribution on each floor. The distributions are changed by veranda type and with/without partition. In the case of parapet type without partition, the velocity at the highest floor decreases in half, though that of the fence type without partition does not change. At the lower floors, both types show the velocity decrement. The partition cause the large velocity decrement to about one fifth compared to the wall without any façade elements. Also the vertical velocity distribution becomes more uniform.

These tendencies do not correspond to the results of wind pressure coefficient, because the wind velocity indicates the local wind velocity inside the veranda. The wind pressure of the wall is dominated by the surrounding pressure distribution generated by building itself, so the façade elements are not so important. Outside the veranda, the wind velocity may be explained by the Bernouilli's equation, that is the more wind pressure the less wind velocity.

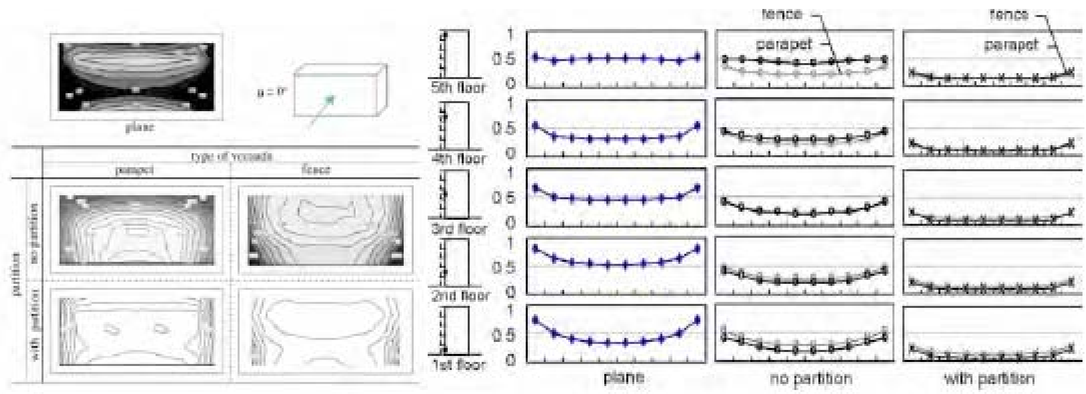


Figure 2.1J: Distribution of wind velocity along wall (wind direction: 0 deg) (Kotani and Yamanaka, 2007)

The authors conclude that the distribution of wind pressure coefficient is not significantly affected by the type of veranda, such as parapet or fence. Basically, the distribution of wind pressure coefficient on the building façade depends mainly on the wind direction. Also, the existence of veranda has little effect on the wind pressure distribution. The wind pressure of the wall is dominated by the surrounding pressure distribution generated by building itself, so the façade elements are not so important. At wind direction of 0 degrees, the local wind velocity along the façade wall is much affected by the type of veranda. Parapet type tends to diminish the local velocity. The partition causes the large velocity decrement to about one fifth compared to the wall without any façade elements and makes the vertical velocity distribution more uniform.

Costola et al (2009) have addressed the importance of wind pressure coefficients (Cp) which are influenced by a wide range of parameters, including building geometry, facade detailing, position on the facade, the degree of exposure/sheltering, wind speed and wind direction. It is practically impossible to take into account the full complexity of pressure coefficient variations. Building Energy Simulation (BES) and Air Flow Network (AFN) programs generally incorporate a standard and simple distribution of coefficients in their numerical analyses. The authors developed an overview of pressure coefficient data and the extent to which they are currently implemented in BES-AFN programs.

A distinction is made between primary sources of Cp data, such as full scale measurements, reduced-scale measurements in wind tunnels and computational fluid dynamics (CFD) simulations, and secondary sources, such as databases and analytical models. The comparison between data from secondary sources implemented in BES-AFN programs shows that the Cp values are quite different depending on the source adopted. The two influencing parameters for which these differences are most pronounced are the position on the facade and the degree of exposure/sheltering. The comparison of Cp data from different sources for sheltered buildings shows the largest differences, and data from different sources even present different trends. The author concludes that quantification of the uncertainty related to such data sources is required to guide future improvements in Cp implementation in BES-AFN programs.

Primary sources: Primary sources are considered to be the most reliable Cp data sources. In this section, a brief description of the main primary sources is full-scale measurements, wind-tunnel measurements, and CFD.

Full-scale measurements: Costola et al (2009) suggest that on-site full-scale measurements at real building facades provide the most reliable source of Cp predictions. In those measurements there is no need to reproduce boundary conditions, uncertainties stemming from scaling do not exist and no physical models are required. However, full-scale measurements are complex and expensive, and are therefore mainly used for validation purposes. Early full-scale experiments used sensors with high uncertainty for the pressure measurements, such as manometers, and for the wind speed, such as cup anemometers. More recent experiments using ultrasonic anemometers and pressure transducers provide a large amount of high-quality data about the pressure at the building facade. It can be concluded that full-scale experiments are the primary data source that provides the most representative information; however the use of these experiments for BES-AFN is restricted to research and validation purposes. Full-scale experiments should also focus on urban environments and on low wind speed conditions, for which the analysis of pressure coefficient data is particularly challenging. Also measurement uncertainty requires further attention.

Wind-tunnel measurements: According to Costola et al (2009) wind-tunnel experiments are generally considered the most reliable source of pressure data for buildings in the design phase. Structural engineering uses custom wind-tunnel experiments to assess the wind loads on a specific building, while considering geometry, immediate surroundings and appropriate approach-flow profiles of mean wind speed and turbulence. The use of wind-tunnel measurements to provide BES input data however is limited due to cost, time and know-how involved in this type of experiments. Based on the sample of studies provided by the author, it can be concluded that wind-tunnel experiments present specific challenges. The quality of wind-tunnel results is directly affected by the history of calibration in the wind tunnel, quality assurance procedures, and the know-how of the personnel involved in the test set-up and execution.

CFD: Computational fluid dynamics (CFD) has been used to study air flow around buildings for more than 30 years, while simulations focused on wind pressure on the building facade emerged about 20 years ago (Costola et al, 2009). Those studies were clearly exploratory, with no direct application in building design or industry. In the recent years, the application of CFD has increased significantly due to improvements in computer performance, price reduction, and the availability of more affordable yet powerful commercial CFD software products. The author suggests a trend of "past achievements and future challenges" in Computational Wind Engineering (CWE) in a paper from 1997 which is still equally valid today in many respects. The paper expresses concern about the misuse of CFD for problems that cannot be approached using this technique, which is still the case today considering the lack of validation in many CFD applications. The review indicates some areas for improvement in the future. The list is reproduced below;

- **Numerical accuracy** by using higher-order approximations coupled with grid independence checks
- **Boundary conditions,** which depend on the specific problem under consideration so that they require good physical insight and high level of expertise.
- **Refined turbulence models** although ad hoc turbulence model modifications are unlikely to perform well beyond the specific flow conditions for which they have been made.

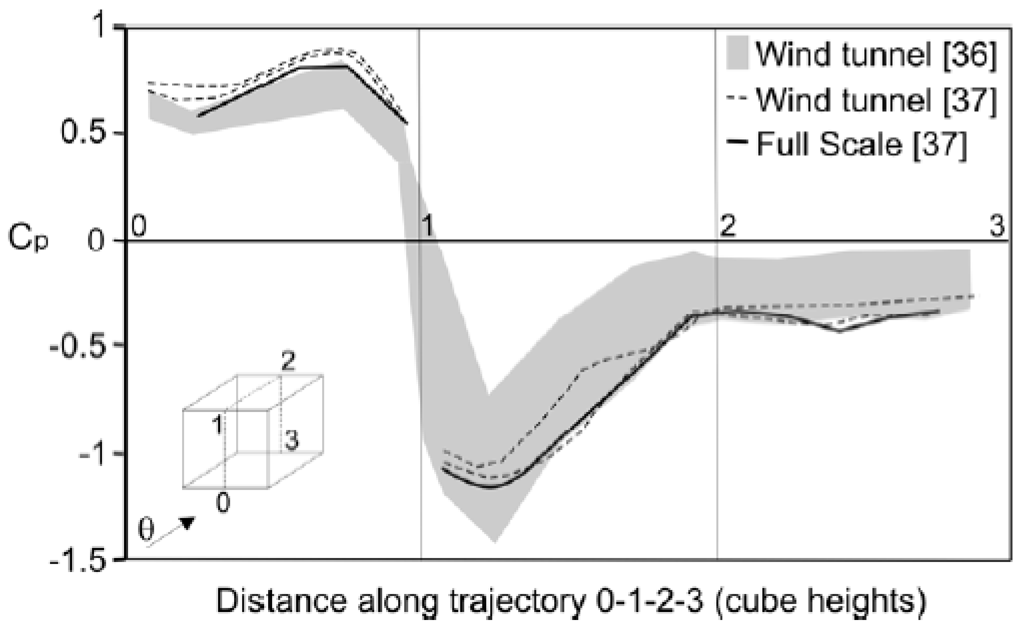


Figure 2.1.K: Comparison if different wind tunnel experiments (Cóstola and etc., 2009)

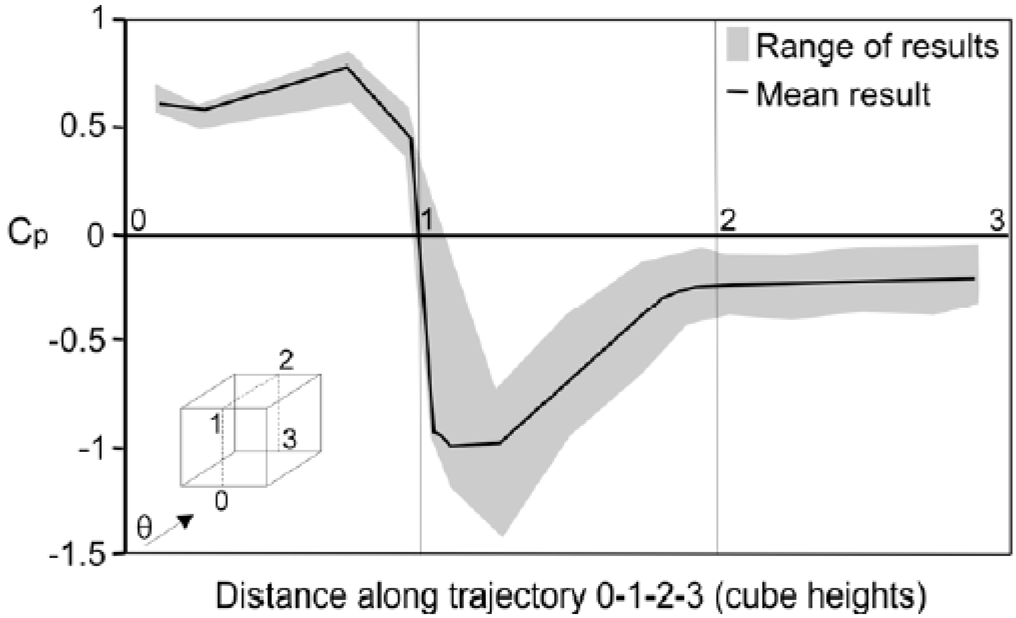


Figure 2.1.L: Comparison if different wind full scale results (Cóstola and etc., 2009)

2.2 The Role of Computational Fluid Dynamics (CFD) in the Study of Wind Phenomena

The reviewed literature in this section describes observation of wind-driven rain (WDR) as well as a methodology to conduct valuable velocity and pressure information on building design. Wind-driven rain research is of importance in a number of research areas including wind movement pattern around building in both pedestrian and elevated level, earth sciences, meteorology and building science.

Blocken and Carmeliet (2004) present the state-of-the-art of wind-driven rain research in building science. Wind-driven rain is the most important moisture source affecting the performance of building facades. There are three distinguished quantification methods have used, such as experimental methods, semi-empirical methods, and numerical methods. The quantity of WDR impinging on building facades is governed by a diversity of parameters: building geometry, environment topology, position on the building facade, wind speed, wind direction, turbulence intensity, rainfall intensity, raindrop size distribution and rain event duration.

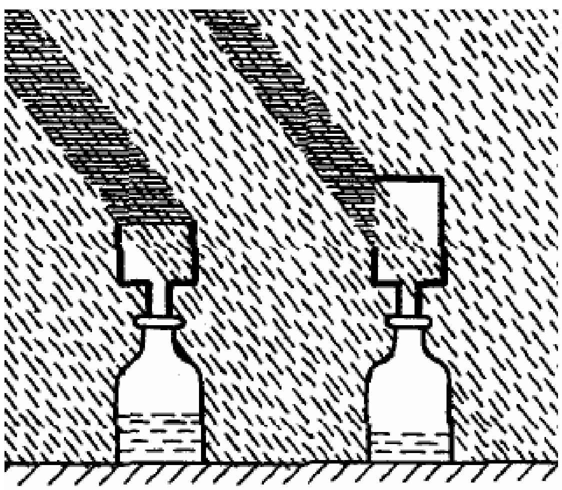
Experimental methods

<u>Wind-driven rain gauges and measurements</u>

WDR measurement is using both of WDR horizontal rainfall gauge with a horizontal aperture to measure horizontal rainfall (figure 2.2A left) and wind-driven rain gauge with a vertical aperture to measure wind-driven rain (figure 2.2A right).

Free-standing wind-driven rain gauges

Blocken and Carmeliet (2004) suggest that free WDR gauges (Figure 2.2B) are placed in "free field conditions" on a post to obtain a general idea of the WDR conditions, whereas wall-mounted WDR gauges are intended to obtain specific information of the WDR exposure at certain positions of the facade. Figure 2.2B-(a) is free-standing wind-driven rain gauge with eight vertical apertures and one horizontal aperture. Figure 2.2B-(b) is free-standing wind-driven rain gauge with four vertical apertures and one horizontal aperture. The purpose of the free-standing gauges are, first to obtain directional information from the catches of the apertures facing different directions and second to obtain free-standing WDR amounts as an indication of the amounts impinging on the building facade. This author also address the completion process there are two special gauges in figure 2.2C are (a) Circular free-standing wind-driven rain gauge and (b) funnel-shaped free-standing wind-driven rain gauge



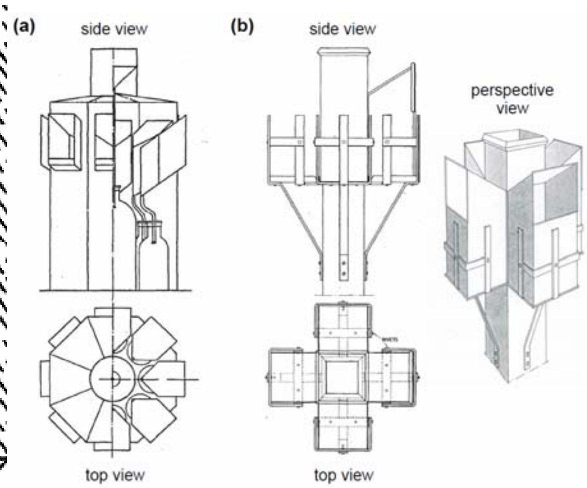


Fig. 2.2A (Left) Fig. 2.2B (Mid), (Blocken and Carmeliet, 2004)

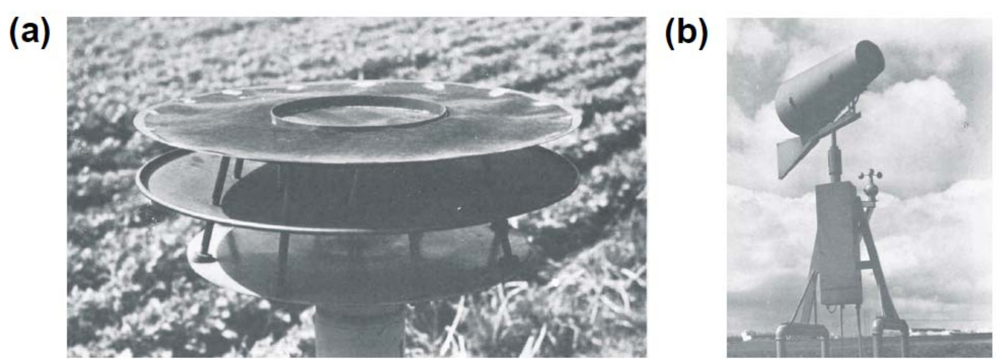


Fig. 2.2C, (Blocken and Carmeliet, 2004)

<u>Wall-mounted wind-driven rain gauges:</u> The wind-driven rain gauges used were all of a similar basic design (Fig. 2.2D). It was plate-type gauges consisting of a collection area and a reservoir. The collection area is made up of a shallow tray (collection plate or catch area) of some material, shape and size and is fixed at the building surface. It has a raised rim around the perimeter to prevent the collection of water from outside the plate. The lowest point of the tray is drilled and tapped to accept a tube leading to the reservoir. The volume or weight of the collected rainwater in the reservoir is manually or automatically registered at regular intervals. For recommended wall-mounted measurements use gauges in Figure 2.2E based on the international Council for Building Research (CIB) as recessed plate-type gauge, meaning that the collection area is to be built into the wall. The advantage of this type of gauge is that the disturbance of the wind field around the gauge is limited. The recession however is responsible for some inconvenience, as it imposes restrictions on the locations where it can be installed. The recessed wall-mounted gauges can be used when there are windows in suitable parts of the buildings or if holes can easily be cut. When this is impossible, gauges must be mounted on the surface of the building.

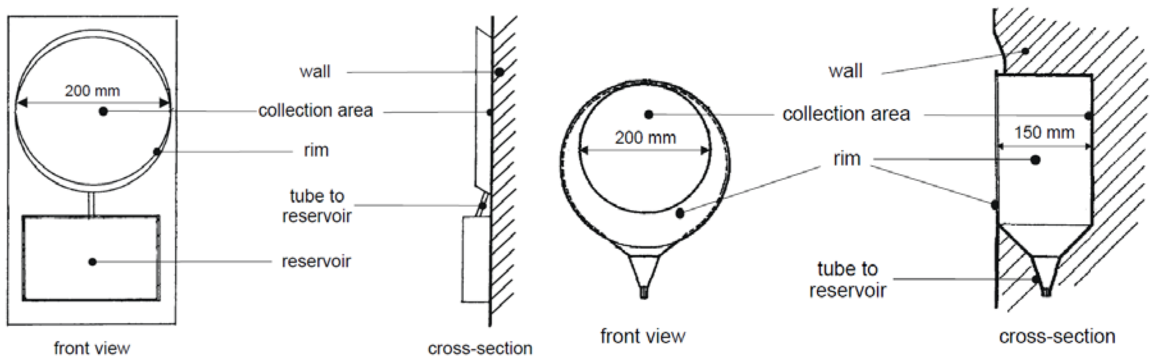


Fig. 2.2D (Left) Wall-mounted plate-type wind-driven rain gauge where the collection area fits flush into the vertical façade. Fig 2.2E (Right) Wall-mounted plate-type wind-driven rain gauge where the collection area is recessed in the wall. (Blocken and Carmeliet ,2004)

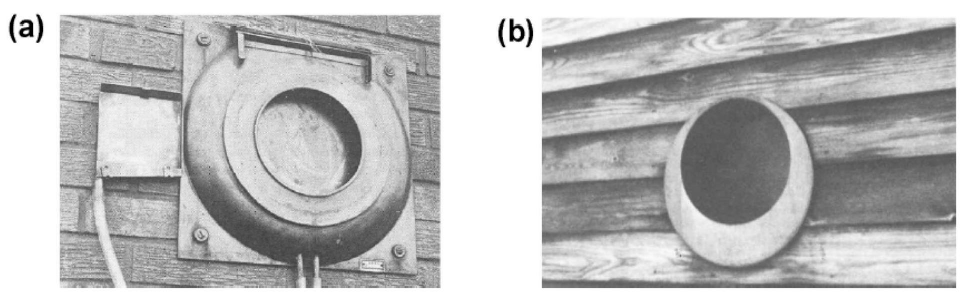


Fig. 2.2F: (a) Two gauges with the collection area fitting flush into the wall and (b) The recessed gauge designed (Blocken and Carmeliet ,2004)

Accuracy of wind-driven rain measurements: Blocken and Carmeliet (2004) have discussed on WDR gauges are not industrially manufactured and there exists no standard on their design. As a result, there are almost as many types of WDR gauges as there are researchers using them. The present discussion is focused on the plate-type WDR gauges that are used for measurements on buildings. The reasons are; first the fact that most gauges used are plate-type gauges, whether they are wall-mounted or free-standing, and second the fact that the discussion of these gauges will for a large part be extendable to the traditional 4-way, 8-way and - to a lesser extent - circular and tube-shaped WDR gauges.

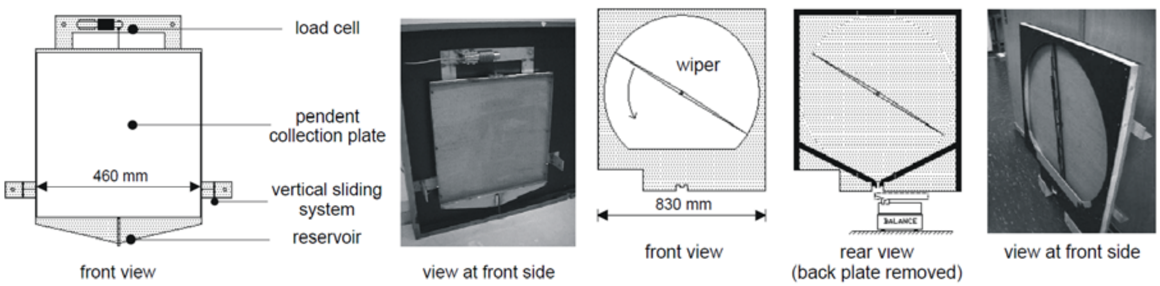


Figure 2.2.G (Left): Special wind-driven rain gauge designed to measure adhesion water. The collection plate and the reservoir are suspended from a load cell. Figure 2.2 G (Right): Special wind-driven rain gauge designed to measure adhesion water. The collection area is equipped with an automated wiper. (Blocken and Carmeliet ,2004)

Application: Blocken and Carmeliet (2004) performed WDR measurements which have been conducted on the south-west facade of the low-rise. The building consists of two main modules, the flat roof module and the sloped roof module. Figure 2.2.H, in between the main modules, shows a small terrace module building dimensions including roof overhang length. Roof overhang varies along the length axis of the building. The direction of the prevailing winds at the test site is south-west. For the purpose of WDR studies, one of the longitudinal facades of the building was constructed facing this direction. The building is situated in a suburban area. For the south-west facade, the only elements providing some shielding from wind and rain are a row of poplars at the north-west side and some low agricultural constructions that are situated about 80 m in front of the south-west facade. Meteorological

<u>Physical simulation of wind-driven rain in wind tunnels</u>

Blocken and Carmeliet (2004) discuss the possibility of wind tunnel modeling of WDR on buildings which had been considered by other researchers (the paper list s series of research projects). The authors report on the difficulties involved in these types of wind tunnel tests. Flower and Lawson concluded that it should be possible to predict impingement rates on buildings by suitable laboratory tests. Rayment and Hilton visualized the movement of raindrop trajectories around a building model using bubbles. Only two actual attempts of WDR quantification tests are reported and described to the authors. An elaborate scaled wind tunnel simulation has been performed at the Boundary Layer Wind Tunnel Laboratory by Inculet and Surry. Modeling of WDR on a full-size building has been attempted in the large Jules Verne wind tunnel at the CSTB-Nantes.

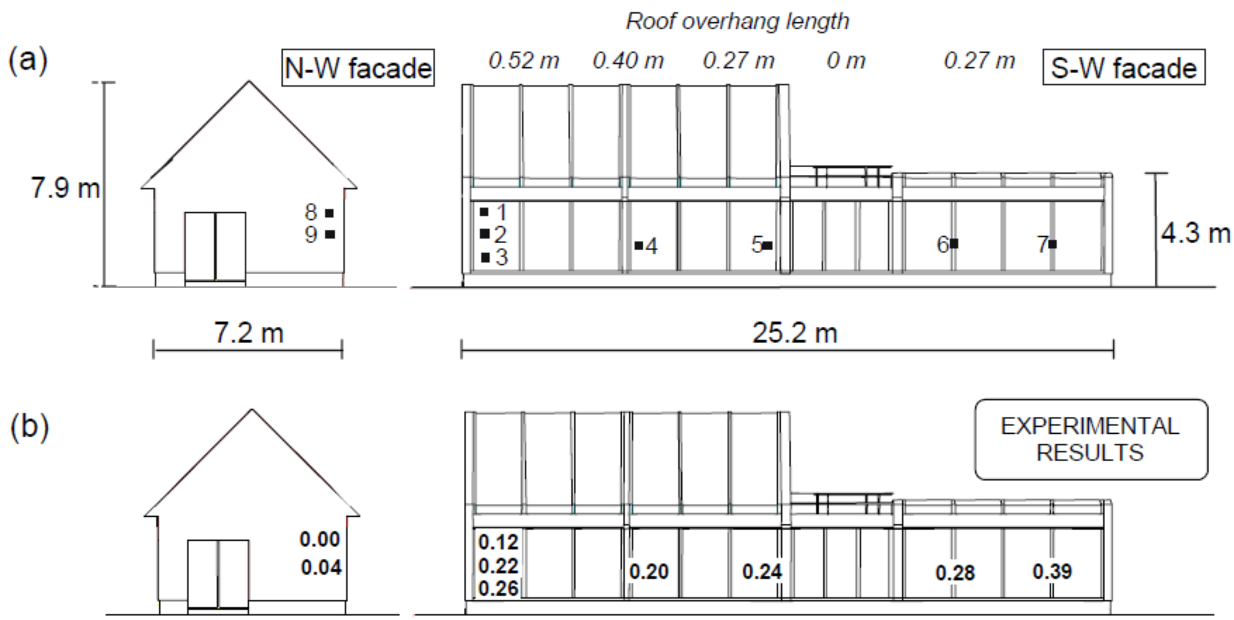


Fig. 2.2.H: (a) North-west and south-west facade. Building dimensions, roof overhang length, positions and numbers (1-9) of wind-driven rain gauges. (b) Spatial distribution of measured catch ratio values at the end of the rain event given in estimated measurement error: 0.04. Multiply catch ratio values with Sh = 24.8 mm to obtain the wind-driven rain amount. (Blocken and Carmeliet ,2004)

The wind tunnel simulation by Inculet and Surry will be briefly discussed here. Nozzle arrays were installed in a boundary layer wind tunnel. Building models at a scale of 1:64 were constructed and placed in the wind tunnel. The wind speed and the raindrop sizes were scaled and WDR on the buildings was physically simulated. An important problem was determining the amount of WDR falling onto different positions on the models.

Measuring WDR on small models requires the use of special techniques such as the electrostatic-sensor technique or the water-sensitive paper method, Inculet and Surry (1994). In the experiments mentioned, the water-sensitive paper method was used. This method consists of positioning pieces of water-sensitive paper on the building model. Each drop that falls on the paper leaves a stain. This way, a visual picture of the wetting pattern is obtained. These tests clearly reproduced the “classical” wetting pattern of building facades. The accuracy of wind tunnel data would be higher than field data, the two wind tunnel experiments mentioned above indicated that this is not necessarily true. Problems that will require specific attention in future wind tunnel modeling are the simulation of a spatially uniform drop size distribution and the development of techniques for determining the impinging quantity of WDR on small-scale building models.

Wind-driven rain gauges and measurements discussion

- The methods for field measurements of WDR in building science have practically remained unchanged since the first measurements were made in the 1930s. In contrast to the very simple measurement principle, determining the error of the measurements appears to be complicated. Evaporative loss of adhesion water from the collection area is considered to be the most important error source. This error can be very large. The relative adhesion-water-evaporation error of WDR measurements decreases as the collected amount of WDR increases.
- Measurements of both free WDR and WDR on buildings have indicated that the intensity of WDR increases approximately proportionally with wind speed and horizontal rainfall intensity. Measurements of WDR on buildings have revealed part of the complex wetting pattern of a facade: top corners, top and side edges are most exposed to WDR.
- A systematic experimental approach in WDR assessment is not feasible. WDR is usually not measured at meteorological stations and databases of WDR field measurements are not commonly available. Furthermore, WDR measurements usually only provide limited spatial and temporal information and measurements at a particular station have very limited application to other sites. Wind tunnel measurements are very demanding. They are labor intensive, expensive and difficult and the latter factor negatively influences their accuracy. Note that this is the present status of wind tunnel modeling reported in literature and that future research efforts might lead to new techniques yielding improved simulations with a higher accuracy.
- Despite all drawbacks mentioned, the experimental methods have proven vital in gaining knowledge on the interaction between WDR and buildings. Field measurements serve as a basis for the development and validation of semi-empirical methods and for the validation of numerical methods. The restriction, which has to be kept in mind when using and interpreting such measurement results is that – depending on gauge type and duration, intensity and type of the rain event - errors can be very large (any value up to 100%).

Semi-empirical methods discussion

- Standard meteorological data measured at weather stations are wind speed, wind direction and horizontal rainfall intensity. WDR is usually not measured. Therefore, it would be interesting if the WDR exposure of building facades could be obtained from semi-empirical relationships between standard weather data and WDR exposure.
- The pioneering work of Hoppestad (1995) in Norway and Lacy in the United Kingdom has provided two semi-empirical methods: the WDR index and the WDR relationship.
- The WDR index is calculated as the product of wind speed and horizontal rainfall amount and is approximately proportional to the WDR amount. In its original form, it is a qualitative measure of the exposure of a location to the “free” WDR. Further developments and improvements in the past decades have transformed the index into a quantitative measure of the exposure of a building wall to WDR.
- The WDR relationship is a formula relating WDR intensity to the standard variables wind speed, wind direction and horizontal rainfall intensity by a WDR coefficient. The main problem when using this relationship is obtaining a reliable WDR coefficient, as it depends on a large number of parameters and is different for each situation and for each WDR spell. A WDR coefficient can be

obtained by short-term measurements or by long-term measurements. The former will yield a WDR coefficient that is only representative for the WDR spell measured, the latter will yield a coefficient that is only representative for the average of the measured situations.

- The European Standard Draft provides a procedure that is based on both the WDR index and the WDR relationship. It inherently uses an adapted WDR coefficient α that is determined as the product of the free WDR coefficient (0.222 s/m) with four empirically determined correction factors, which can be selected by the user. The Standard Draft is the result of good research work over many years and it is based on a wide range of WDR measurements on different buildings and at various positions on these buildings. It is clearly superior to the traditional practice of using the WDR relationship with a single WDR coefficient that does not take into account the variation of WDR with the building geometry and with the position on the building facade. However, the Standard Draft also has a number of drawbacks: (1) It can only be applied for the building configurations shown in Fig. 2.2I. (2) The wall factors in Fig. 2.2I. only provide limited information about the spatial variation across the facade. (3) The WDR coefficient α is assumed to be constant for a fixed position on the building (i.e. constant in time). (4) It is assumed that the effect of varying wind direction can be taken into account by the factor cosθ (cosine projection).
- The accuracy of semi-empirical methods has up to now not been investigated. It is not clear to what extent the underlying assumptions of the WDR relationship and the European Standard Draft (α constant in time, cosine projection) are justified.
- Semi-empirical methods can provide rough estimates of the WDR quantity to be expected on buildings. Such estimates may be sufficient in some cases, but they are insufficient when more detailed information is requested, e.g. the complete WDR distribution on buildings or the effect of building details such as roof overhang. In such cases, one has to resort to numerical methods.

<u>Numerical methods discussion</u>

- The complexity of the interaction between WDR and buildings has led to numerical modeling. The main part of numerical WDR research has been conducted in the past decade and has been made possible by the increasing computing performance.
- Numerical modeling has significantly increased the understanding of the interaction between wind, rain and buildings. The numerical method allows a detailed and high resolution quantification of WDR to be made, both spatially and temporally.
- Drawbacks of the numerical method are a very large amount of preparation work, the need for high computing performance and long calculation times.
- Accuracy requirements have been listed and must carefully be adhered to: the choice of the turbulence model, the spatial resolution of the computational grid, the raindrop size distribution, the drag coefficient formulae, whether or not to include turbulent dispersion of raindrops and the time resolution of the meteorological input data.
- The results of the few validation studies that have been performed are encouraging. Further validation studies for different building geometries and environment topologies are needed. Rain events for validation purposes must be carefully selected, in such a way that

measurements errors are limited (e.g. high WDR amount, few evaporation periods, avoiding glancing wind angles, etc).

Description of wall	Average value	Distribution
Two storey gable	0.4	0.5 0.4 0.3 0.3 0.3 0.2
Three storey gable	0.3	0.5 0.4 0.3 0.3 0.3 0.2
Multi storey flat roof[1]	0.2 for e.g. ten storey, but higher intensity at top	0.5 for top 2.5 m 0.2 for remainder
two storey eaves wall	0.3	Pitched roof (20° or over) typical overhang 350 mm 0.3 0.3 0.3
three storey eaves wall	0.4	Pitched roof typical overhang 350 mm 0.4 0.4 0.4
two storey flat roof (pitch <20°)	0.4	0.5 0.4 0.2
1) These data apply to multi-storey blocks of normal aspect width; no data are available for exceptionally narrow buildings		

Fig. 2.2 I: Wall factors as provided by the European Standard Draft to take into account the type of wall (height, roof overhang) and the variation of the WDR across the surface of the wall. (Blocken and Carmeliet ,2004)

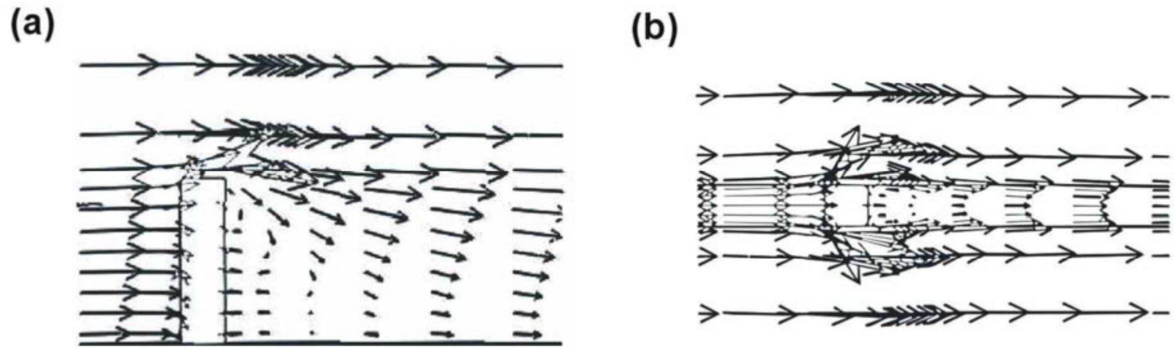


Fig. 2.2 J: Wind flow pattern around a building (10x10x40 m³) calculated with Stead-state CFD simulation by using standard k-ε turbulence model.
(a) Longitudinal flow in the center plane of the building.
(b) Flow in a horizontal plane at mid-height of the building (Blocken and Carmeliet ,2004)

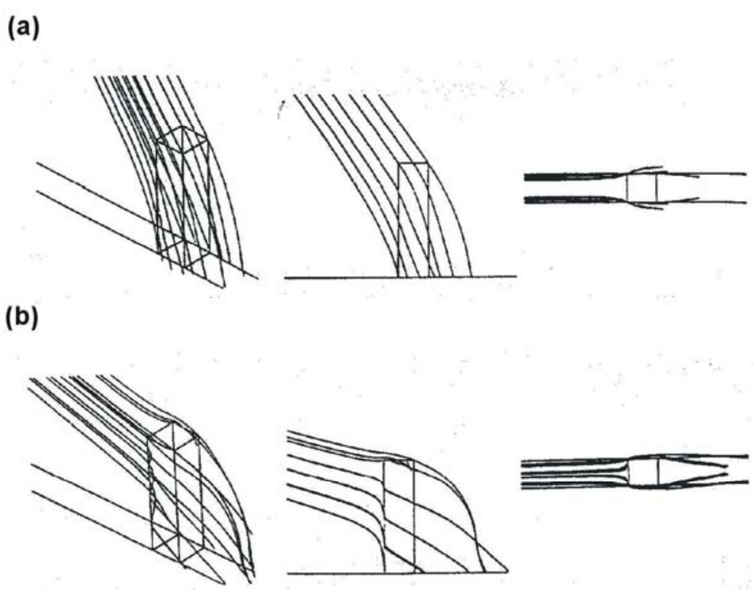


Fig. 2.2.K: Perspective, side and plan view of raindrop trajectories for (a) 5 mm radius raindrops and (b) 0.25 mm radius raindrops both in the 10 m/s flow field. (Blocken and Carmeliet ,2004)

Moeseke and etal (2005) conclude that the natural ventilation is of increasing interest in building industry because of recent focus on environmental concern, occupant comfort and economic performance criteria. The objective of the paper was to investigate how wind may induce natural ventilation, with focus on wind incidence and large scale environment density influences. These parameters modify flows inside and outside buildings. Numerical dynamic simulations are achieved for a standard office building using pressure coefficients obtained from a parametrical model. Simulations allow description of inside building air flow for three incidences: 0^0, 45^0, 90^0 and three theoretical environments: open, suburban and urban. The study worked with both vertical and horizontal pressure coefficient gradients. Results show that horizontal velocity gradients are important aspects in the description of air movement around buildings. Urban wind driven ventilation potential is also discussed. The need for further studies is illustrated in order to obtain handy pressure coefficients prediction tools and to optimize openings mechanical regulation.

Computational fluid dynamics (CFD) simulations have similar benefits as wind tunnel tests. Moeseke and etal (2005) suggest that readily available computer resources will drastically improve CFD investigations to a broad range of architecture design problems. The authors suggest that accuracy and availability of such CFD tools must still be perfected. This is especially true when working on complex turbulent air flows situation such as wind movement that interacts with the buildings. Therefore, architectural applications of CFD's developments are still at the beginning.

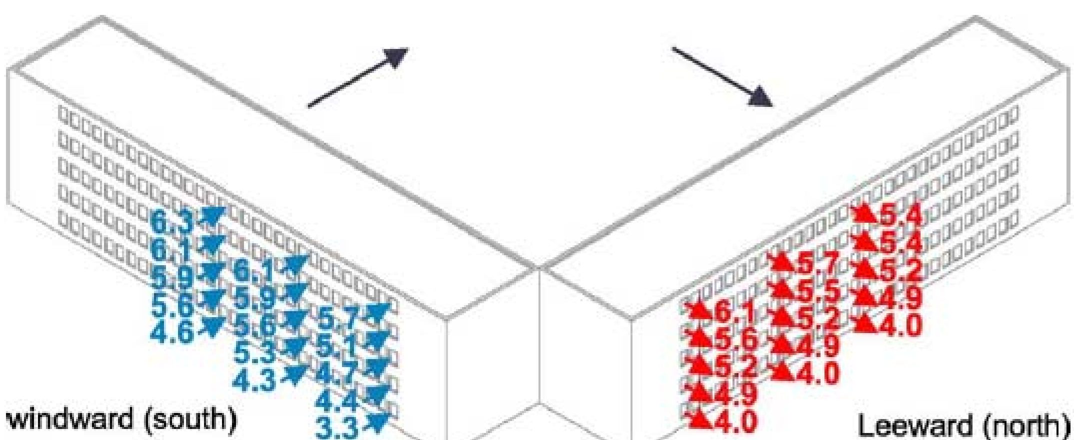


Fig. 2.2.L: ach results for a 4 m s_1 normal wind in an open environment: whole building. (Moeseke and etal, 2005)

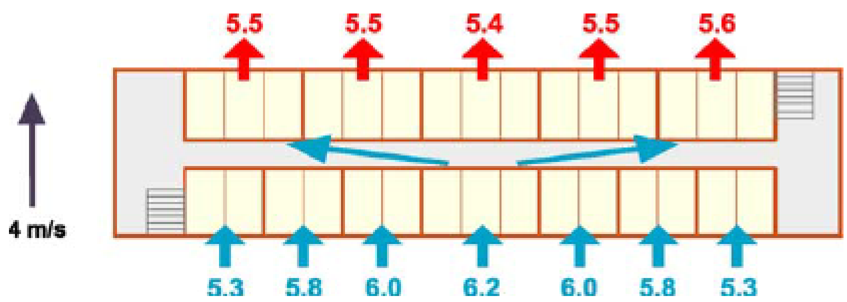


Fig. 2.2.M: ach results for a 4 m s_1 normal wind in an open environment: fourth storey, basic opening regine. (Moeseke and etal, 2005)

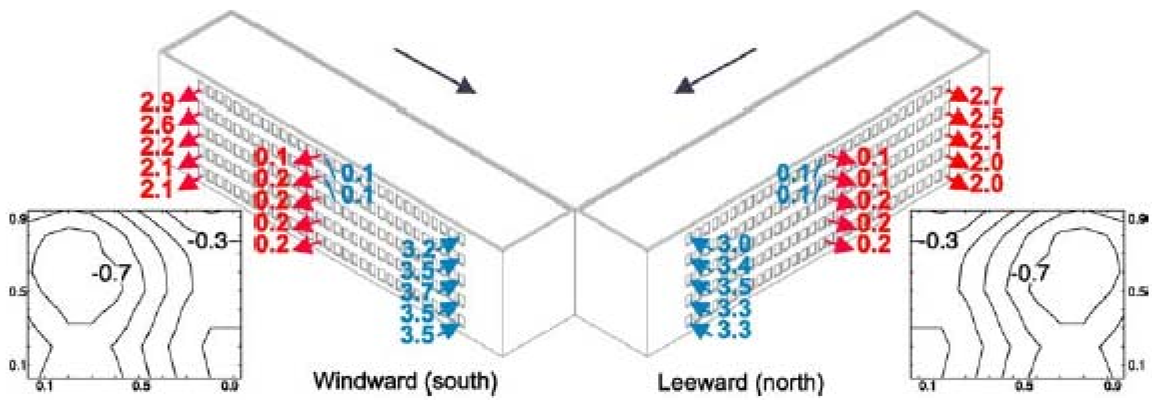


Fig. 2.2.N: Pressure coefficients obtained with M. Grosso's model (left and right) and ach results (centre) for a 4 m s_1 908 incident wind in an open environment (Moeseke and etal, 2005)

3. External CFD – Numerical Assessment of Air Movement around Buildings

This section reviews literature about CFD based assessment methods of wind movement phenomena around buildings and at a neighborhood level. The literature cited and reviewed contains pertinent information for the present CFD project.

3.1 Basic Considerations of CFD

3.1.1 Solutions Methods of Partial Differential Equations

Fluid problems can be represented by mathematical models described by partial different equations (PEDs). For example, the Navier-Stokes equations is used to describe fluid motion. In order to solve PEDs, there are three classical choices for numerical solution: the finite different method (FDM), the finite volume method (FVM and the finite element method (FEM).

According to description from Autodesk Simulation (2013), FDM is based on the differential approach by which "the partial derivatives are replaced with a series expansion representation, usually a Taylor series. The series is truncated usually after the first two terms. The more terms are included, the more accurate the solution. However, more terms in the series expansion causes the complexity and number of discrete points or nodes of the solution to increase significantly. Applying this method to a regularly shaped geometry is straightforward. However, for irregularly shaped geometries, the equations must be transformed before the Taylor series can be applied. This transformation introduces all sorts of problems in terms of additional cross-coupling of equations, mesh generation and general convergence."

FVM is based on the integral approach by which "the governing equations are integrated over a volume or cell assuming a piece-wise linear variation of the dependent variables (u, v, w, p, T). Again the piece-wise linear variation determines both the accuracy and the complexity. Using these integrations, one essentially balances fluxes across the boundaries of the individual volumes. The flux is calculated at the mid-point between the discrete nodes in the domain. Hence, you must calculate a flux between all neighboring nodes in the domain. In a topologically regular mesh (same number of divisions in any one direction), this flux calculation is quite straightforward. In an irregular mesh (as in an automatically generated tetrahedral mesh), this calculation will lead to an excruciating amount of fluxes and a major bookkeeping effort to make sure all the fluxes have been calculated properly."

FEM is also based on the integral approach. However, "the governing partial differential equations are integrated over an element or volume after having been multiplied by a weight function which generally uses Galerkin's method of weighted residuals. The dependent variables are represented on the element by a shape function, which is the same form as the weight function. The shape function may take any of several forms. The main advantage as well as the main disadvantage of finite elements is that it is a mathematical approach that is difficult to put any physical significance on the terms in the algebraic equations. In the finite volume method, you are always dealing with fluxes - not so with finite elements.

However, the application of finite elements on any geometric shape is the same. Also, the boundary conditions which must be added after the fact for finite volume methods are an integral part of the discretized equations.”

Advantages and disadvantages: Molina-Azi et al. (2010) suggested that FDM is rarely used in engineering flow due to its difficulties in handling of complex geometry. The FVM is both versatile and easy to understand and FVM programs have become commonly used in most commercial CFD application. FEM is suitable to intricate geometry or boundary conditions but is found few commercial CFD applications for its programing and implementation difficulties of this techniques.

In this study, Molina-Azi et al. (2010) compared the efficiency of two different discretization methods (FVM and FEM) used CFD solvers for simulating the natural ventilation in greenhouses. By using measured data for validation, the authors have not found significant differences in accuracy of prediction between the two methods. For most of the cases, they shows more similar results of temperatures than those of velocities. They provided similar qualitative descriptions of airflow. Molina-Azi et al. (2010) also concluded that FEM allowed easier meshing than FVM for complex geometries. FEM is the method with more computational efficiency in terms of computational speed, physical and memory computer storage. Specifically, the FVM required twice as much computing time per cell and step as FEM and the amount of required memory storage was approximately 10 times greater for the FEM. The table below summarizes the advantages and disadvantages of the various methods:

Table 3.1.1.A Advantages and disadvantages comparison between FVM and FEM (Autodesk Simulation, 2013, modified)

Method	Advantages	Disadvantages
Finite Difference	- Less and simple mathematics involved	- Irregular geometries require far more effort
Finite Volume	- Fluxes have more physical significance - Common commercial availability (ANYS/FLUENT, STAR-CCM+)	- Irregular geometries require far more effort
Finite Element	- More mathematics involved - Natural boundary condition (for fluxes) - Master element formulation - Any shaped geometry can be modeled with the same effort	- Less physical significance - Few commercial availability (Autodesk Simulation CFD, ANSYS/FLOTRAN, Comsol)

McBride et al. (2007) discussed a coupled finite volume method which called a combined vertex-based-cell-center discretization technique, which solves the flow field at the cell or element vertexes, and all other variables are solved at cell center. This technique allows for improvement of the purely cell-centered approach in the conventional CFD tools which often fails on distorted meshes, capable to deal with arbitrarily complex geometrical structures. This technique can be embedded within generic center-cell CFD tools for wider application.

McBride et al. (2007) mentioned that CD-Adapco has introduced a multi-physics solvers based on polyhedral meshes in STAR-CCM+ application to employ the vertex-based discretization schemes for the polyhedral control volumes, creating more neighboring points to give higher accuracy in computation and provide additional flexibility in mesh generation for very complex geometries.

3.1.2 Various Applications of External CFD

Blocken et al (2011) provides an overview of the application of CFD in building performance simulation for the outdoor environment, focused on four topics: (1) pedestrian wind environment around buildings, (2) wind-driven rain on building facades, (3) convective heat transfer coefficients at exterior building surfaces, and (4) air pollutant dispersion around buildings. For each topic the author delineate the need for CFD. The author provides several CFD cases studies as well as a discussion about accuracy and some perspectives for practical application are provided. For all four topics, CFD offers considerable advantages compared to wind tunnel modeling or (semi-)empirical formulae because CFD analysis can provide detailed whole-flow field data under fully controlled conditions, without similarity constraints from model scaling. The author suggests shortcomings of CFD such as limitations of using steady RANS modeling approach, the increased complexity and computational expense of LES and the requirement of systematic and time-consuming CFD solution verification and validation studies.

Pedestrian wind environment around buildings: High-rise buildings can introduce high wind speed at pedestrian level, which can lead to uncomfortable or even dangerous conditions. Wind discomfort and wind danger can be detrimental to the success of new buildings. Today, many urban authorities only grant a building permit for a new high-rise building after a wind comfort study has indicated that the negative consequences for the pedestrian wind environment remain limited. A wind comfort study is generally performed by a combination of three types of information/data: (1) statistical meteorological information; (2) aerodynamic information; and (3) a comfort criterion. CFD or wind tunnel data can be used to provide part of the aerodynamic information.

The author describes that one of the main advantages of CFD in pedestrian-level wind comfort studies relates to avoiding the time-consuming two-step approach by providing whole-flow field data. In spite of its deficiencies, steady RANS modeling with the k-e model or with other turbulence models has become the most popular approach for pedestrian-level wind studies. Two main categories of studies can be distinguished: (1) fundamental studies, which are typically conducted for simple, generic building configurations to obtain insight in the flow behavior, for parametric studies and for CFD validation, and (2) applied studies, which provide knowledge of the wind environmental conditions in specific and often much more complex case studies.

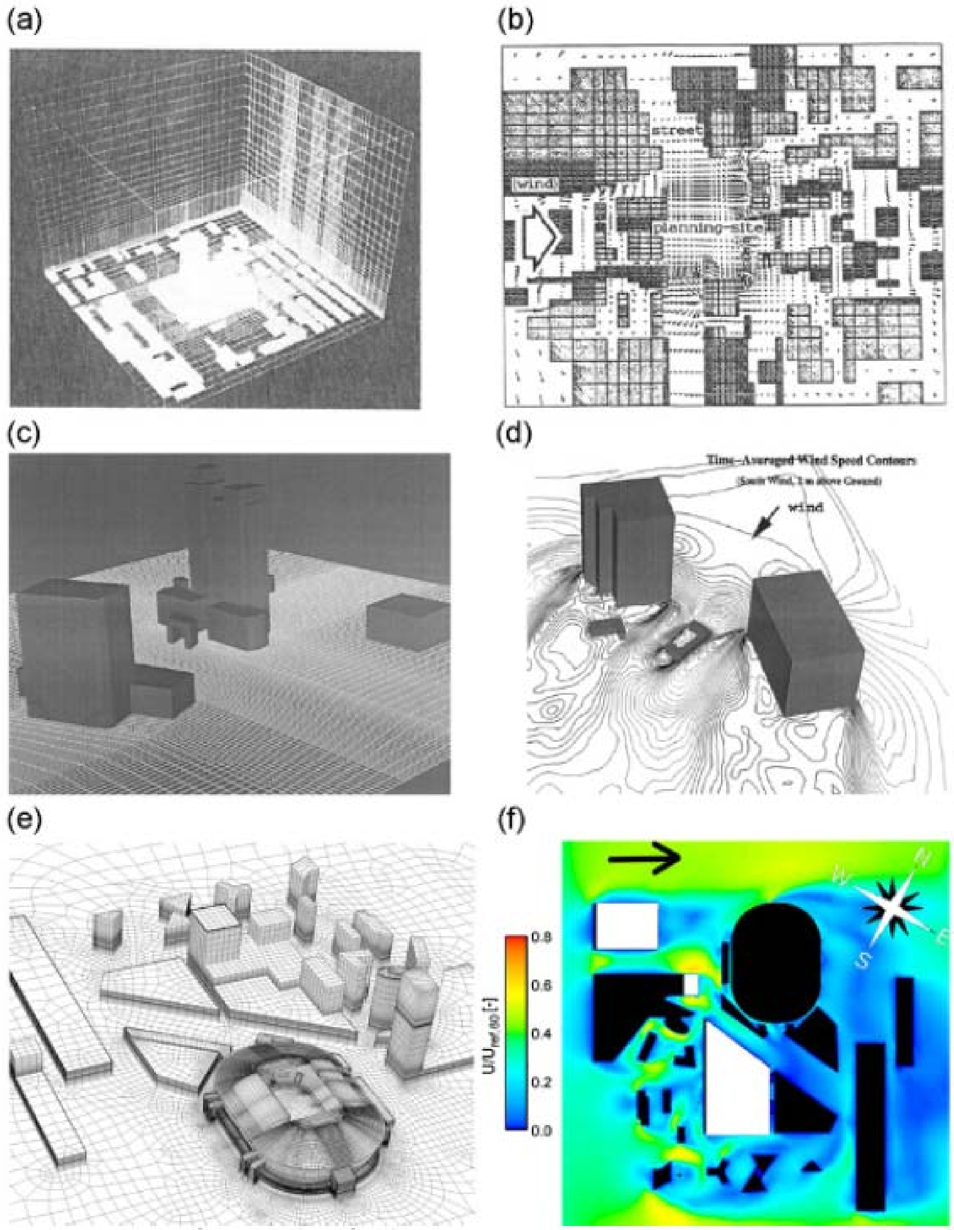


Figure 3.1.2.A:. Examples of CFD studies of pedestrian-level wind environment in urban areas: (a-b) Grid (finite elements) and wind-velocity vectors based on steady RANS simulations , (c-d) Grid (total cell count unknown) and wind speed contours based on, (e-f) Grid (2.8 million cells) and wind speed ratio contours, based on steady RANS (Blocken and etal, 2011).

3.1.3 Best Practices Guidelines for CFD Applications Related to Wind Engineering:

The expanding use of CFD in the prediction of wind effects in the urban built environment and wind movement around buildings around building has raised the demand for standards to ensure the quality assurance. Assuring quality and precision of the analysis and resulting repeatability of prediction of pertinent phenomena in conjunction with wind is being accomplished by an expanding need in this area of CFD applications. A number of Best Practice Guidelines (BPG) for CFD simulation of air flows in the urban environment have been published by different authors and organization.

The main objective of establishing BPGs for the urban wind environment is the improvement and quality assurance of microscope obstacle-accommodating meteorological models and their application to the prediction of flow and transport processes in urban or industrial environments (Frank et al, 2007). The term microscope obstacle-accommodating models suggest that the covers CFD application in the BPGs are describing wind induced processes around buildings, at neighborhood and street level.

According to Frank et al (2007) the required quality assurance of the application is closely related to the users' knowledge of modeling process. The require knowledge is most effectively transferred by the formulation of a BPG for the intended application. However, even for this well-defined application the formulation of BPGs faces the problem of providing general advice for specific problems that may vary substantially although belonging to the same field. The authors of the most frequently quoted general BPCs for CFD applications used in the industry (Casey & Wintergerste, 2000) acknowledge that the use of the BCPs offer “roughly those 20% of the most important general rules of advice that cover roughly 80% of the problems likely to be encountered”. The authors that that BPG are therefore not exhaustive but cover as many aspects of the proper usage of CFD for the prediction of urban flows as possible.

Tominaga (2008) indicate that CFD describing the wind environment around buildings has been applied with insufficient information about the influence of many factors related to the computational condition on prediction results. While there have been numerous studies on the wind environment around actual buildings using CFD, the influence of the computational conditions, i.e., grid discretization, domain sizes, boundary conditions, etc., on the prediction accuracy has not been systematically investigated. Therefore, a set of guidelines is required that summarize important points in using the CFD technique for appropriate prediction of pedestrian wind environment.

Franke et al (2008) list he different sources of errors and uncertainties that are known to occur in numerical simulation results are listed and defined. The sources of error that can be controlled and quantified by the user are discussed in detail and best practice guidelines for their reduction and quantification are given. The evaluation of CFD codes requires that all the errors and uncertainties that cause the results of a simulation to deviate from the true or exact values are identified and treated separately if possible. Several classifications of these well-known errors and uncertainties exist. The authors suggest that most general discrimination divides them into two broad categories:

Errors and uncertainties in modeling the physics, which arise from the assumptions and approximations made in the mathematical description of the physical process:

- Simplification of physical complexity
- Usage of previous experimental data
- Geometric boundary conditions
- Physical boundary conditions
- Initialization

Numerical errors and uncertainties result from numerical solution of the mathematical model:

- Computer programming
- Computer round-off
- Spatial discretization
- Temporal discretization
- Iterative convergence

Frank et al (2010) indicate that BPGs should avoid or at least reduce what is known as user errors, which errors typically originate from the incorrect use of CFD and related codes due to either a lack of experience or a lack of resources. In the course of a simulation the user may make mistakes or unwise choices, which then manifest themselves as one, or more, of the above mentioned user errors. In addition the authors suggest that the user should be aware of the uncertainties that exist in the simulation of flow in the urban wind environment.

Blocken et al (2012) describes that wind studies require combining statistical meteorological data with aerodynamic information. The aerodynamic information is needed to transform the statistical meteorological data from the weather station (meteorological site) to the location of interest at the building site. As illustrated in Figure 3.1.3.A the statistical wind information at site can be expressed as specific wind speed profile which results from the specific terrain aerodynamic roughness conditions, as expressed with aerodynamic roughness length $z0 = 0.03$ m. The aerodynamic information consists of two parts: the terrain related contribution and the design related contribution. The terrain related contribution represents the change in wind statistics from the meteorological site to a reference location near the building site. The design related contribution represents the change in wind statistics due to the local urban design. The design related contribution can be obtained by either wind tunnel modeling or CFD.

Blocken et al (2012) emphasize that CFD has some important advantages over wind tunnel testing. Wind tunnel tests generally only produce specific results at a few selected points in the urban model. They do not provide a whole image of the flow field. CFD on the other hand provides whole-flow field data, this means CFD produces data on the relevant parameters in all points of the computational domain. Unlike wind tunnel testing, CFD does not suffer from potentially incompatible similarity requirements because simulations can be conducted at full scale. This is particularly important for extensive urban areas such as the case study in this paper. CFD simulations easily allow parametric studies to evaluate alternative design configurations, especially when the different configurations are all a priori embedded within the same computational domain and grid

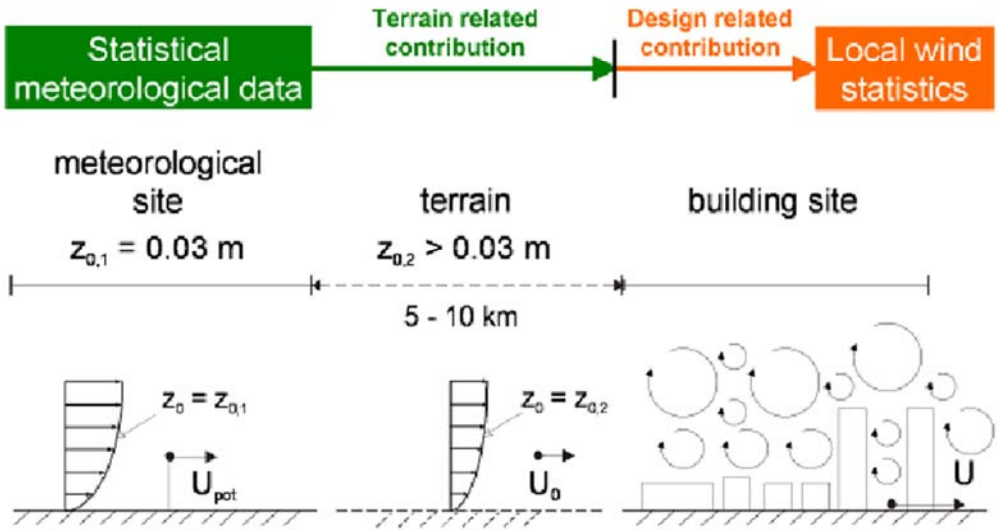


Figure 3.1.3.A: Transformation of statistical meteorological data from the meteorological site to the building site (Blocken et al, 2012)

<u>Some main aspects of the best practice guidelines:</u>

Blocken et al (2012) provide a summary of main elements of these guidelines:

- The computational geometry should contain all buildings and obstacles that have a significant effect on the flow at the location of interest. These objects should be modeled with their main shape.
- The computational domain should be large enough to avoid artificial acceleration of the flow. Its size can be based on the height of the tallest building in the urban configuration and/or on the blockage ratio
- The computational grid should preferably consist of hexahedral or prismatic cells near solid boundaries; grid resolution should be based on a grid-sensitivity or grid-convergence analysis on at least three different grids
- The boundary conditions (for inflow, ground roughness and top and lateral) should be consistent; Iterative convergence should be monitored and the simulation termination criterion should be established
- While most of the BPGs are consistent with these main elements, some BPCs do not give recommendations on the turbulence model.

<u>Simulation and decision framework for urban wind studies:</u>

Blocken et al (2012) integrates BPGs with more basic guidelines in terms of verification and validation. From the three cases that describe recommended processes of new developments within an urban wind configuration one case is depicted in Figure 3.2.3.B. Under the sample case it is recommended to evaluate both the existing and the new urban configuration with CFD. Here on-site measurements will be made to provide validation data for the CFD simulations of the existing configuration. The reason is that on-site measurements, in spite of their disadvantages, represent the complex reality without simplifications and are therefore the true validation data for numerical models.

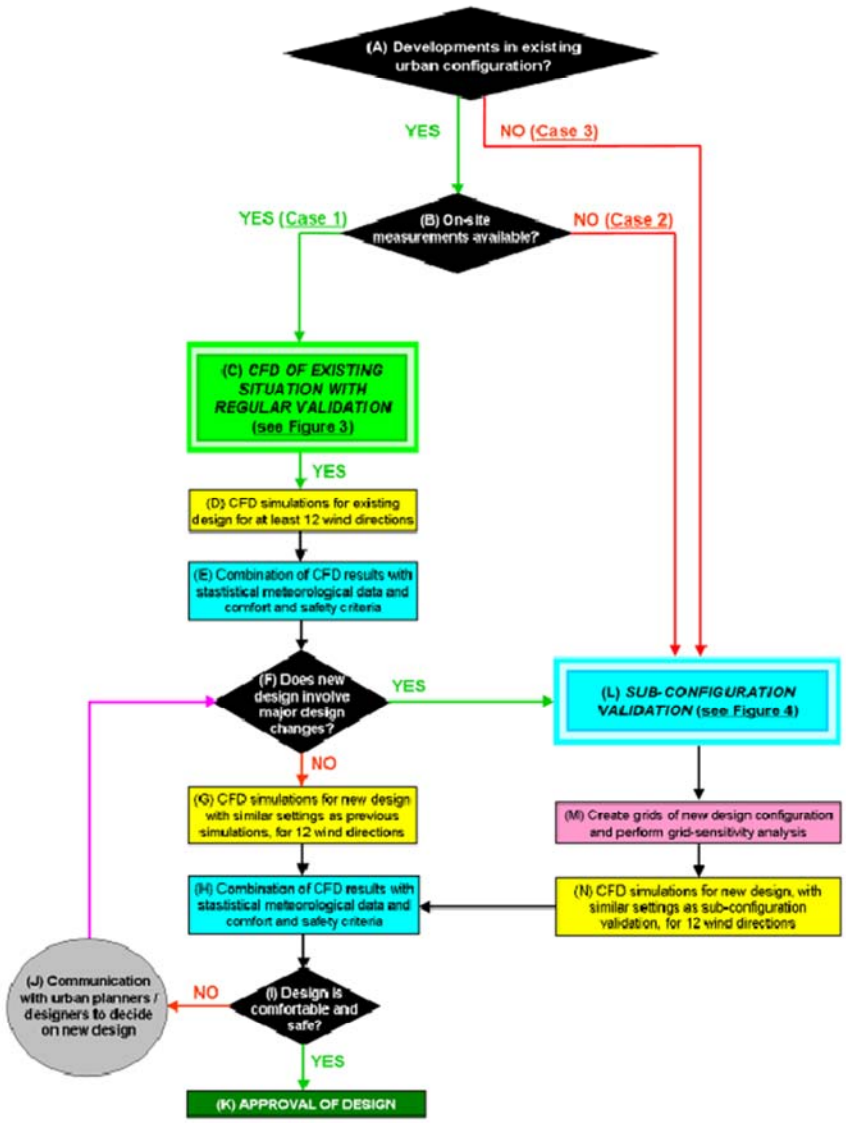


Figure 3.1.3.B: Flowchart illustrating the framework for the assessment of pedestrian wind comfort and safety with CFD for Case 1 (Blocken et al, 2012)

The following describes more detailed elements of several published BPGs, which are frequently cited in the reviewed literature. (Casey and Wintergerste, 2000), (Franke et al, 2004, 2008, 2012), (Blocken at al 2011 and 2012) (Tominaga et al, 2008)

Choice of target variables:

The first step in a validation simulation should be the definition of the target variables. These should include the variables that are representative of the goals of the simulation and those that can be compared with the corresponding experiments.

Choice of approximate equations describing the physics of the flow:

The potentially largest impact on errors and uncertainties arises from the choice of the basic equations. The turbulent flow within urban wind environments is in general modeled by the Navier-Stokes equations using one of the following closures for the turbulence:

- Steady Reynolds Averaged Navier-Stokes (RANS) - The guidelines focus on steady RANS simulations as they are the most common approach
- Unsteady RANS (URANS)
- Large Eddy Simulation (LES) and hybrid RANS-LES approaches

It should be noted that best practice advice is given for the choice of the turbulence models. Rather 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.

Choice of the geometrical representation of obstacles

Distribution of buildings has the greatest impact on wind flow patterns. Secondary influence factors in the urban area include vegetation, orography and surface characteristics (e.g. roads, grass, sand). The central area of interest should be reproduced with as much detail as possible.

Tominaga et al (2008) investigated several basic geometric representation of investigated structures and evaluated the simulation performance for these geometries. Figure 3.1.3.C shows several of the basic geometries.

Choice of the computational domain:

The size of the entire computational domain in the vertical, lateral and flow directions depends on the area that shall be represented and on the boundary conditions that will be used. If large eddy simulation (LES) is used the overall size of the computational domain is that it is large enough to contain also the largest, energetically relevant flow structures.

The different BPGs suggest slightly different vertical and lateral extensions of the domain as well as for the extension of the domain in flow direction, divided in the region in front and behind the built area. The recommendations for the different extensions are multiples of the representative building height investigated or a certain blockage ratio.

Choice of boundary conditions:

The boundary conditions represent the influence of the surroundings on the computational domain whereby the surrounding has been cut off. As the boundary conditions determine to a large extent the solution inside the computational domain, their appropriate selection is of high importance. Often, however, these boundary conditions are not fully known. Therefore the boundaries of the computational domain should be far enough away from the region of interest to not contaminate the solution there with the approximate boundary conditions.

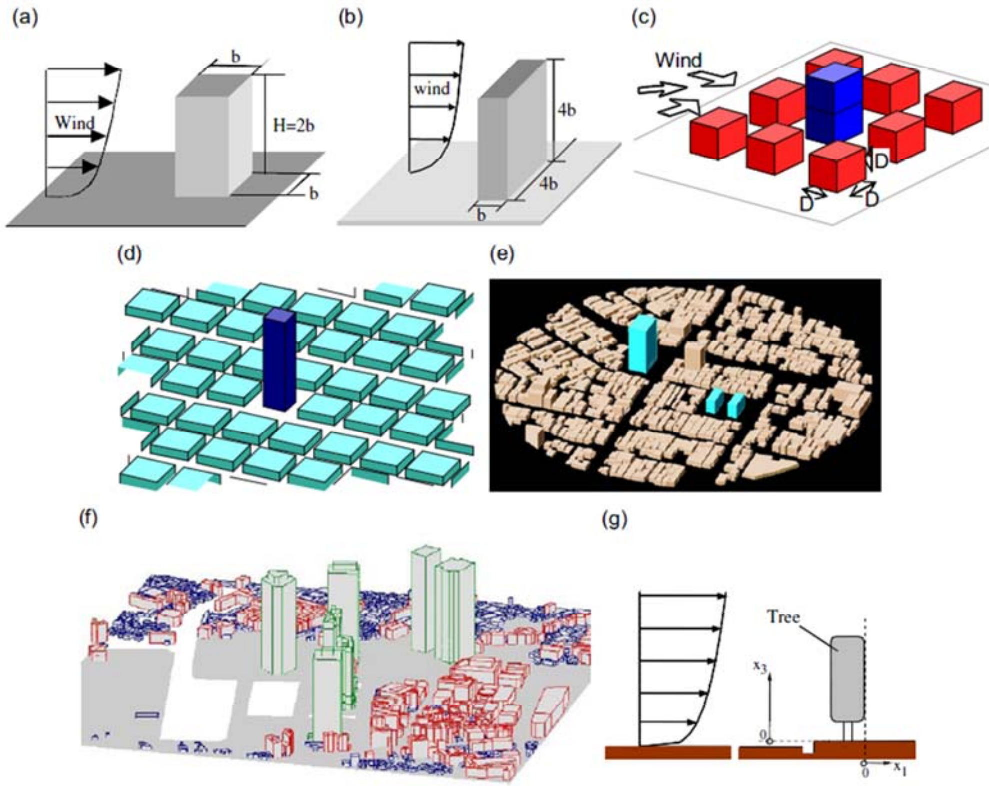


Figure 3.1.3.C: Seven test cases for cross-comparison. (a) Test case A (2:1:1 square prism); (b) Test case B (4:4:1 square prism); (c) Test case C (Simple city blocks); (d) Test case D (High-rise building in city); (e) Test case E (Building complexes with simple building shapes in actual urban area); (f) Test case F (Building complexes with complicated building shapes in actual urban area); (g) Test case G (Two-dimensional pine tree).

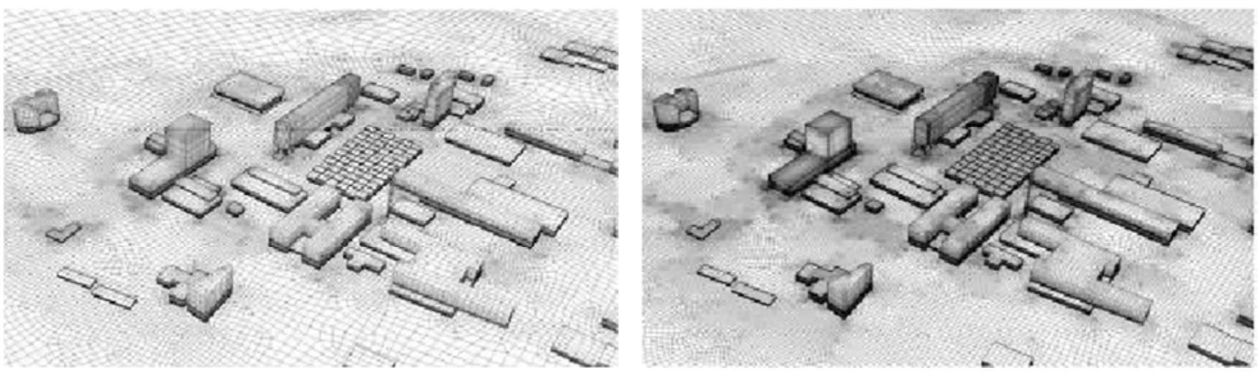
The following boundary conditions are considered:

- Inflow boundary conditions: equilibrium boundary layer at the inlet plane
- Wall boundary conditions: Solid boundaries with no-slip boundary conditions
- Lateral boundary conditions: important applications is the symmetry boundary conditions when approach flow direction is parallel to the lateral boundaries.
- Outflow boundary conditions: At the downwind boundary plane of the computational domain

<u>Choice of the computational grid:</u>

The main criteria for the selection of the computational gird is the discretization method that is used for solving the basic equations. In the CFD analysis of urban wind applications the closely related Finite Volume Method (FVM) and the Finite Difference Method (FEM) are used, whereby most commercial CFD codes used for computational wind analysis use FVM. . For both the FVM and FEM the computational results depend on the grid that is used to discretize the computational domain. The grid has to be designed in such a manner that it does not introduce 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. Ideally the grid is equidistant. Therefore, grid stretching/compression should be small in regions of high gradients, to keep the truncation error small.

The grid resolution plays a significant role in the quality of the solution and the convergence. Basically a fine grid results in a more accurate solution. However, a good compromise between computational accuracy and computational cost has to be selected. Significant computational resources are required for fine grids whereas coarse girds can be solve with less computational resources. The convergence however is less prominent in coarse grids than in fine grids. Figure 3.1.3.D shows a comparison of a fine and a coarse grid for a university campus. The fine grid has about five time as many cells as the coarse grid.



Coarse computational grid (~ 2.6 million cells)   Fine computational grid (~ 12.4 million cells)

Figure 3.1.3.D: Example of grid resolution for a coarse and a fine grid (Blocken et al, 2012)

The preferred shape of the computational cells is hexahedra preferable over tetrahedral. This results in smaller truncation errors and a better iterative convergence. It is necessary to ensure that the aspect ratios of the grid shapes do not become excessive in regions adjacent to coarse girds or near the surfaces of complicated geometries. For improved accuracy, it is desirable to arrange the boundary layer elements (prismatic cells) parallel to the walls or the ground surfaces. Figure 3.1.3.E shows the arrangement of grid elements near solid surface in unstructured grid. When a global systematic grid refinement is not possible due to resource limitations, then at least a local grid refinement should be used in the area of the main interest.

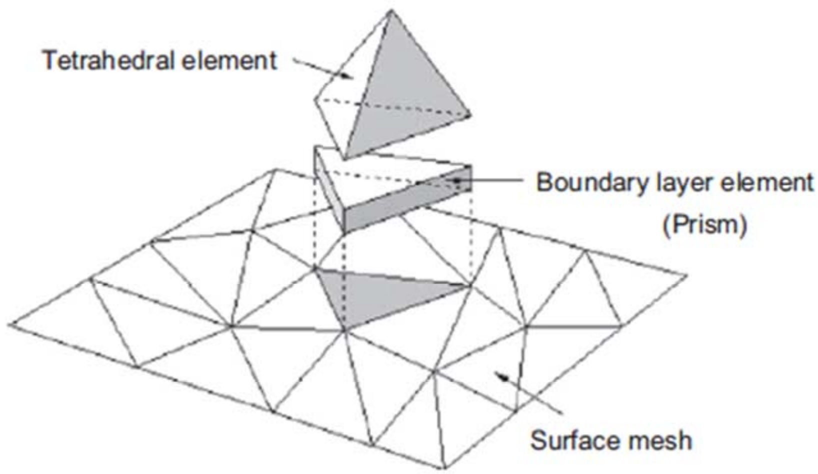


Figure 3.1.3.E Arrangement of grid elements near solid surface in unstructured grid. (Tominaga at al, 2008)

Choice of initial data:

In RANS, URANS and LES models a boundary and initial value problem has to be solved numerically. The larger the model domain or the smaller the wind speed, the more relevant the initial data become. For steady RANS stationary solutions are searched thus the iteration is stopped as soon as the solution is not changing any more or the iterative solution converges. In these cases mainly the boundary values influence the model solution and the impact of the initial data is small. When selecting initializing data for the flow filed with values that are close to the final solution the computational efforts needed to reach stationary solutions will be reduced. Generally speaking, to obtain the converged solution quickly, an appropriate physical property of initial condition should be given. The inflow profiles extended to the whole domain or the results obtained by laminar flow computation are often used for the initial condition.

Criteria for convergence criteria:

After sufficient convergence of the solution is achieved calculation needs to be finished. For this purpose, it is important to confirm that the solution does not change by monitoring the variables on specified points or by overlapping the contours among calculation results at different calculation steps. Default values for convergence of commercial codes might not be appropriate. Therefore, stricter convergence criteria are required to check that there is no change in the solution. The termination criterion is usually based on the residuals of the corresponding equations. Residuals should tend towards zero. Especially in complex geometries of urban areas the residuals may not drop below a user specified error. Typically, the scaled residuals should be dropped 4 orders of magnitude.

Measures to mitigate calculation divergence or convergence problems could include the following:

- The aspect ratio and the stretching ratio of the grids may be too large.
- The relaxation coefficient of the matrix solver may be too small.
- Periodic fluctuations such as a vortex shedding may be occurring.

3.1.4 Coupled Versus De-coupled Modeling of External Windflow

Van Hoff and Bocken (2010a) report that wind flow in urban environments is an important factor for a large number of physical processes that can affect human health and comfort and the durability of man-made constructions. Wind related phenomena that are subject to extensive investigation for the design of structures and planning of neighborhoods are atmospheric transport and dispersion of solid, liquid and gaseous air pollutants, wind loading, pedestrian wind comfort and natural ventilation of buildings. The following describes articles describe wind induced processes that relate to natural ventilation performance of buildings.

In regard to the use of terms authors use different terms to describe the same CFD domain calculation. The term "full-domain" is synonymous with "coupled flow". The terms "recoupled domain" and domain decomposition" or "decomposed domain" is synonymous with "un-coupled flow. Furthermore, it should be noted that the literature also refers to "coupled" CFD models when CFD calculations are coupled with energy simulations. This type of coupling is performed to increase the accuracy of ventilation prediction with reduced computing efforts. When coupling CFD with multizone airflow programs a more accurate ventilation performance can be evaluated. A coupled CFD -multizone system provides more accurate ventilation predictions over a longer duration and achieve a time dependent building ventilation performance, than when using multi-zonal approaches alone. While CFD provide only a snapshot in time of a stead state ventilation scenario, the multi zone results typically extend over a longer time, such as an annual prediction of ventilation. In the following discussion "coupled CFD" only refers to the domains used in CFD calculations.

Nore et al (2010) reports on a investigation of narrow (23 mm) ventilated facade cavities of a low-rise building by coupled and decoupled CFD simulations. In the coupled simulations, the wind-flow pattern around the building and the resulting air flow in the cavities are calculated simultaneously and within the same computational domain. In the decoupled simulations, two separate CFD simulations are conducted: a simulation of the wind flow around the building (with closed cavities) to determine the surface pressures at the cavity inlet and outlet openings, and a simulation of the cavity airflow, driven by these surface pressures. The author indicated that the results of the coupled simulations compared favorably with those from past experimental studies. Comparing the results from the coupled and decoupled simulations allowed to assess the local losses (entrance and exit losses) of the cavities.

Ramponi and Blocken (2012c) reports on a series of coupled 3D steady RANS simulations for a generic isolated building. The CFD simulations were validated based on detailed wind tunnel experiments with Particle Image Velocimetry (PIV). The authors describe that in in CFD simulations of cross-ventilation involving large openings, a major issue of concern is the accurate modeling of the interaction between the outdoor wind flow around the buildings and the indoor air flow inside the buildings, which interact with each other at the ventilation openings. A distinction can be made between a coupled and a decoupled approach. Figure 3.1.4 A indicates the difference between the coupled and uncoupled domains as used in CFD calculations.

In the coupled approach, there is a single computational geometry and computational domain, that includes both the outside and the inside environment of the building (Fig. 3.1.4.A (a)). In this approach, the ventilation openings are considered open, the outdoor wind flow and indoor air flow are solved within the same computational domain and the interaction (coupling) between the outdoor wind flow and indoor air flow is resolved in detail using the appropriate governing equations. Contrary to this, in the decoupled approach, there are two different computational geometries and two different computational domains: one for the outdoor environment and one for the indoor environment of the building (Fig. 3.1.4.A(b)). In this approach, the wind flow simulation is conducted for the building as a sealed body, i.e. the openings are “closed”. This simulation yields the pressure coefficients at the positions of the openings and these coefficients are subsequently used as boundary conditions for the CFD simulation of the indoor air flow.

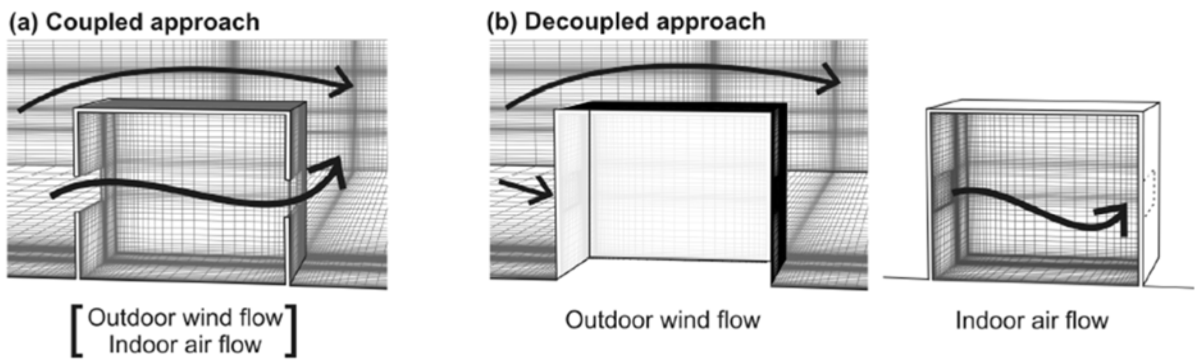


Figure 3.1.4.A: (a) Coupled and (b) decoupled approach for analysis of wind-induced cross-ventilation of buildings (Ramponi and Blocken (2012)

Ramponi and Blocken (2012) suggest, based on an detailed literature review that, by far, most CFD research on wind induced cross-ventilation has applied the coupled approach. An extensive, but not necessarily exhaustive, overview of coupled outdoor-indoor CFD studies included indicates the type of study (generic or applied), the type of building and surroundings (isolated, building group, urban), the turbulence modeling approach (RANS, LES, DES) and turbulence models used, whether validation was performed, and whether and for which parameters a sensitivity analysis was performed.

The authors suggest that the main reason for the extensive use of the coupled approach is the knowledge that, in case of large ventilation openings, the decoupled approach can introduce important errors. The decoupled domain approach, which includes the so-called sealed-body assumption, implies that the pressure distribution on the building envelope is not affected by the presence of the openings. It assumes that the turbulent kinetic energy is dissipated at the windward opening and that the effect of the dynamic pressure on the air flow passing through the opening is negligible. However, in case of wind flow through large ventilation openings, the turbulent kinetic energy is rather preserved and the sealed-body assumption might therefore not longer valid.

Figure 3.1.4.B illustrates a comparison between measured velocity vector (e.g. “PIV”) and a coupled CFD reference case for the he vertical center plane and the horizontal plane at mid-height through the

openings (Ramponi and Blocken ,2012). The figure suggest good general correlation of the velocity flow fields between measured (PIV) and coupled CFD reference case. It should be noted that the experimental data obtained by PIV cannot resolve the flow filed in close proximity to openings. Figure 3.1.4.C shows the comparison of experimental (PIV) and reference CFD case for the streamwise wind speed ratio U/Uref along the centerline. Figure 3.1.4.C suggests a very good correlation between the measurements and CFD predictions. Furthermore an increase in wind speed downstream of the openings can be observed with both the measure and CFD predictions.

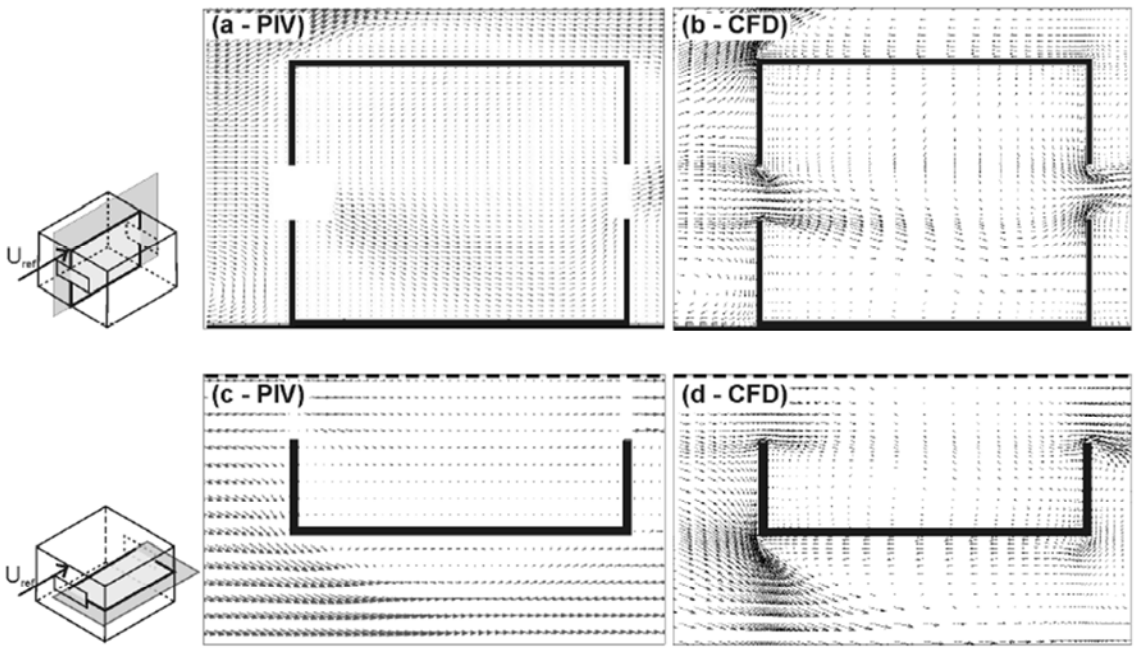


Figure 3.1.4.B: Comparison of the velocity vector fields obtained with the PIV measurements and the CFD reference simulation case in the vertical centerplane and in the horizontal plane at mid-height through the openings (Ramponi and Blocken (2012)

VanHoff and Blocken (2010a) report on a coupled CFD modeling approach for urban wind flow and indoor natural ventilation applied to a large sports arena. The authors suggest that, to their knowledge, this CFD study was the first of a coupled CFD simulations of urban wind flow and indoor natural ventilation in complex urban environments and for complex building geometries have not yet been performed. The authors point out that wind flow is very complex and that appropriate tools are required for characterization of the flow and the related processes. They name three main approaches: (1) on-site full-scale experiments; (2) reduced-scale wind tunnel measurements; and (3) numerical modeling with Computational Fluid Dynamics (CFD). As opposed to experiments, the main advantages of CFD are that it provides information on the relevant flow variables in the whole calculation domain (whole-flow field data), under well-controlled conditions and without similarity constraints. However, the accuracy of CFD is an important matter of concern. Care is required in the geometrical implementation of the model and in grid generation, and solution verification and validation studies are imperative.

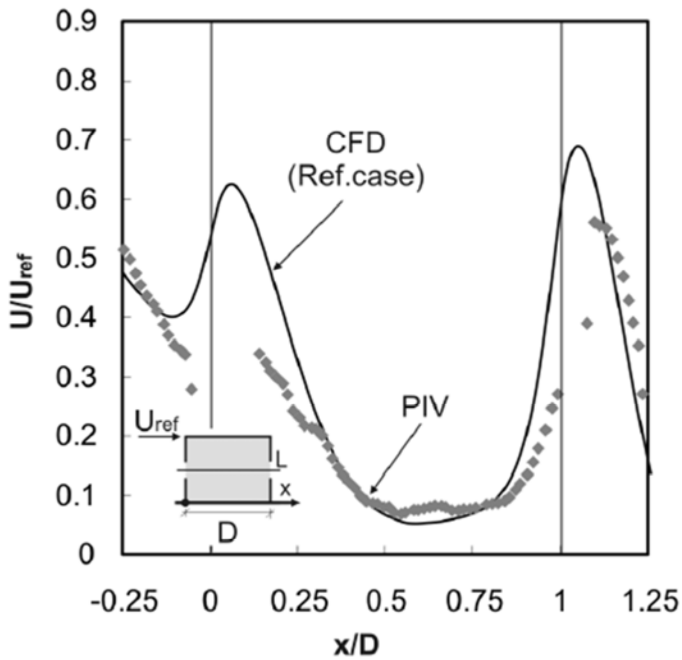


Figure 3.1.4.C: Comparison of experimental (PIV) and reference CFD case for the streamwise wind speed ratio U/Uref along the centerline.

The authors point out advantages and disadvantages of coupled and decoupled CFD domains. In the coupled CFD simulation which both the outdoor and indoor air flow are modeled simultaneously and within the same (therefore coupled) computational domain. This method allows the proper calculation of air flow in the proximity of and through ventilation openings. The main disadvantage of this method in urban applications is the large difference in geometrical length scales between the outdoor (urban) environment (1-5 km) and the ventilation openings (e.g., 0.01 – 1 m), resulting in a large and high-resolution grid, and thus in a relatively high computational cost. The authors stipulate that this might probably be the reason why the coupled CFD approach method has only been used for relatively simple outdoor and indoor environments and relatively large ventilation openings.

In the case of the decoupled simulation the authors suggest that very large grids that would be required to include the complex outdoor and indoor environments and/or for small ventilation openings can be avoided. In decoupled simulations two separate simulations are conducted, one for the outdoor flow and one for the indoor flow, each in their own computational domain. In the outdoor flow simulations, the ventilation openings are closed. The information obtained from the simulation of the outdoor flow (generally pressure coefficients at the positions of the openings) can be used as boundary condition for the simulation of the indoor flow. Although this is the standard approach for indoor ventilation studies, its accuracy can easily be compromised because of the simplifications involved. Often, only pressure is passed from outdoor to indoor environment by means of pressure coefficients at the boundary, and assumptions are made in terms of discharge coefficients and expansion coefficients.

The main objective of performing the coupled CFD ventilation study of the stadium was to determine alternatives in the ventilation openings in the envelope. There have been instances where the thermal (overheating) and ventilation performance, the latter being quantified as air changes per hour (ACH) of the arena was insufficient. Coupled CFD simulations are performed in the stadium study was to verify present natural ventilation configuration of the entire building and evaluate four alternative ventilation configurations to improve thermal and ventilation performance. The alternative configurations consist of one specific geometrical change made to the current stadium geometry with closed roof, except for the last configuration, which has an open roof.

The resulting grid for the stadium is depicted in Figure 3.1.4.D. A coarse and a fine grid were produced with 3 and 9.2 million cells respectively. The different grid sizes were used in sensitivity tests. The results of the CFD calculations were compared with air flow measurements conducted at various representative points at the stadium envelope. Alternatives of larger ventilation openings in the building envelope were investigated by modifying the grid. The different ventilation configurations were evaluated and compared based on the overall ventilation rate/ACH of the entire indoor air volume. The ACH for the alternatives were calculated based on the mass flow rates through the ventilation openings. Therefore, the most relevant locations are the ventilation openings, and the most relevant parameter is the mass flow rate through these openings. Figure 3.1.4.E shows contours of non-dimensional velocity magnitude U/U10 in four horizontal planes, for the representative wind direction at different heights. The lower wind speed ratios around the stadium indicate that it is situated in the wake of the surrounding buildings, which causes the lower air exchange rates for this wind direction.

The CFD based design optimization of the stadium ventilation openings in which four alternatives to the current ventilation configuration were evaluated resulted in a significantly improved ventilation performance for two of the alternatives. (see Table 3.1.4.F)

Table 3.1.4.F: Calculated air change rate per hour (ACH) for the current situation and for four alternative ventilation configurations, for reference wind speed U10 = 5 m/s, for wind directions = 16º, 151º, 196º and 331º and for fixed indoor surface temperatures. (VanHoff and Blocken, 2010a)

Ventilation configuration	ACH (h^{-1})				
	φ (°)				
	16°	151°	196°	331°	Average
Current situation	1.51	1.33	1.11	1.49	1.36
Configuration 1	1.56	1.52	1.12	1.33	1.38
Configuration 2	1.91	1.61	1.29	1.54	1.59
Configuration 3	2.19	2.28	1.61	1.72	1.95
Configuration 4	4.57	3.40	2.66	3.41	3.51

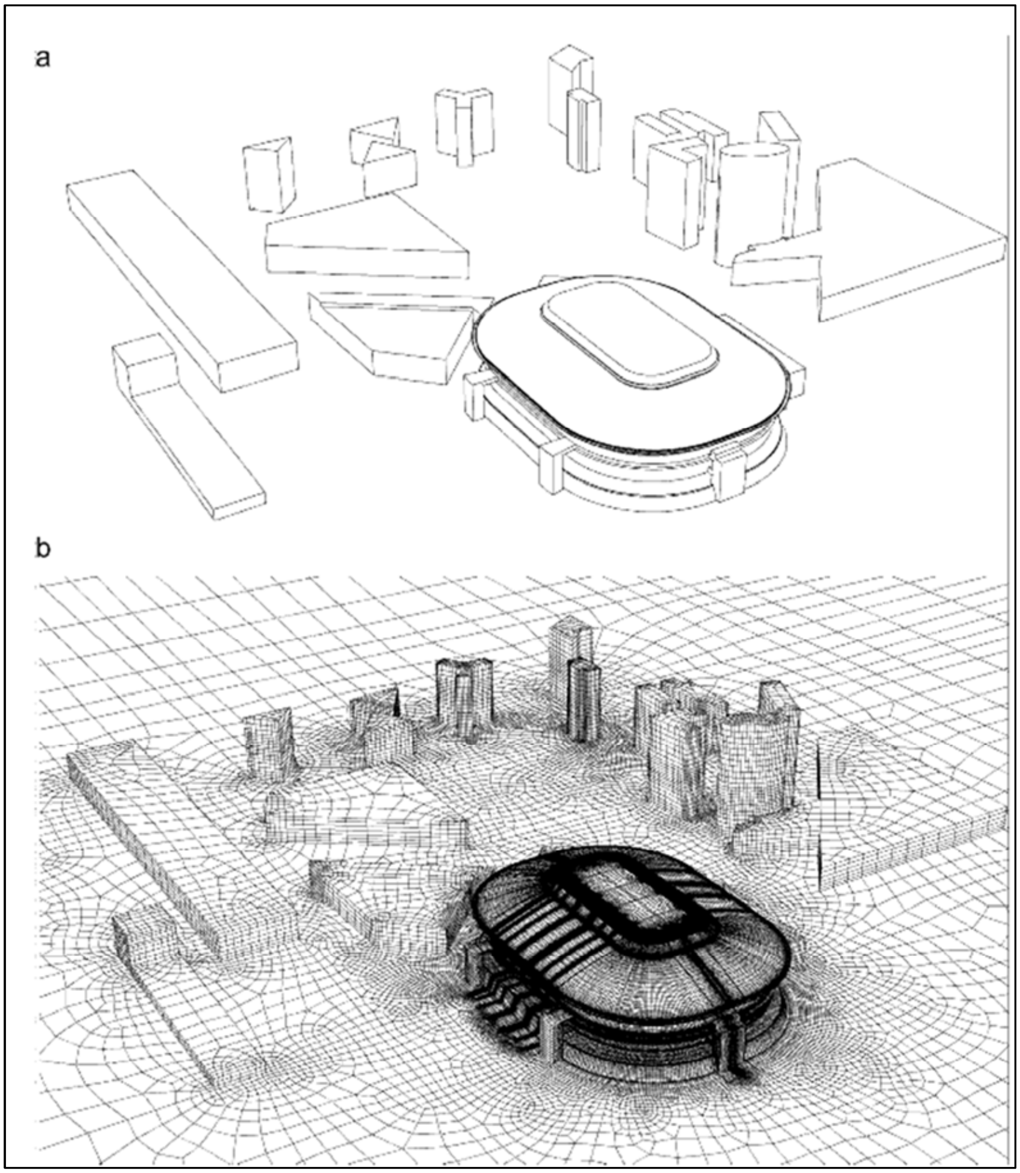


Figure 3.1.4.D: (a) Computational model geometry, view from northeast; (b) Computational grid on the building surfaces and part of the ground surface. A high resolution is used in the proximity of the stadium, and a lower resolution at a larger distance from the stadium. (VanHoff and Blocken, 2010a)

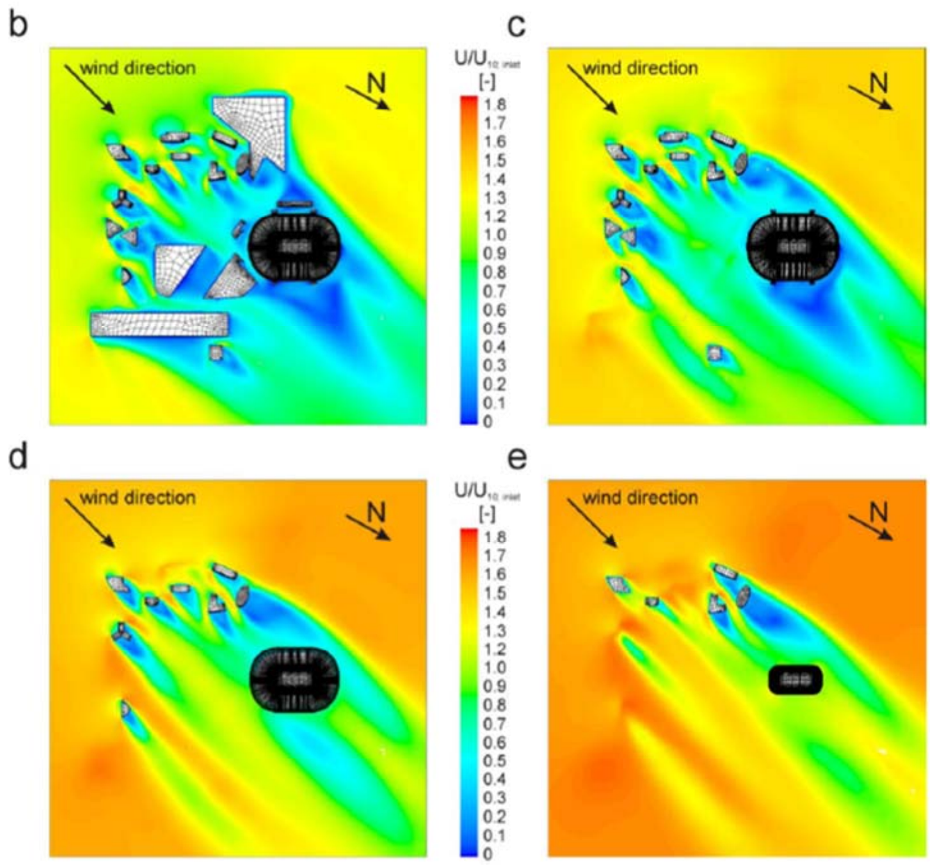


Figure 3.1.4.E: Contours of non-dimensional velocity magnitude U/U10 in four horizontal planes at different heights (VanHoff and Blocken, 2010a; modified)

Meroney (2009) reports on studies that compare results of internal flow evaluated from full domain (coupled) and decomposed (de-coupled) domain investigations. The author points out that CFD analysis is first performed for the flow field outside the building, from which boundary conditions are extracted for the analysis of the indoor situation. Boundary conditions of wind pressures and flow orientation at proposed opening locations are used to estimate internal pressure coefficients based on cross-building surface pressures, and opening discharge coefficients related to a dimensionless internal room pressure to predict opening flow rates. Although the different turbulence models produced significantly different external flows, it appears that since the internal flows are primarily driven by cross building surface pressures which did not vary strongly, the cross ventilation air flow values are very similar.

Meroney (2009) compared internal flow characteristics, expressed as a flow vector field and internal pressure coefficients, between the full and decomposed domain. Both pressure fields were also compared to measured values. The author reports very similar internal flow values for both the full and decomposed domains. Furthermore the values compare well with measured internal pressures. Figure 3.1.4.E shows a comparison between values of external and internal air flow for different cases examined measured in the wind tunnel (1), in the full domain (2) and in the decomposed domain. The filed measurements were unable to provide values near the openings due to generic limitations of data equations . The full-domain predictions are only dependent on domain entrance conditions and building dimensions. The decomposed domain prediction is only dependent on boundary conditions extracted from the full-field sealed building surface results. Both the filed data and full-domain CFD calculations reproduce the *vena-contracta* that occurs downwind of the inlet opening, that results in maximum concentrations occurring about one half opening dimension within the room. The domain decomposition approach does not produce a vena-contracta, and maximum velocities tend to occur at the inlet opening.

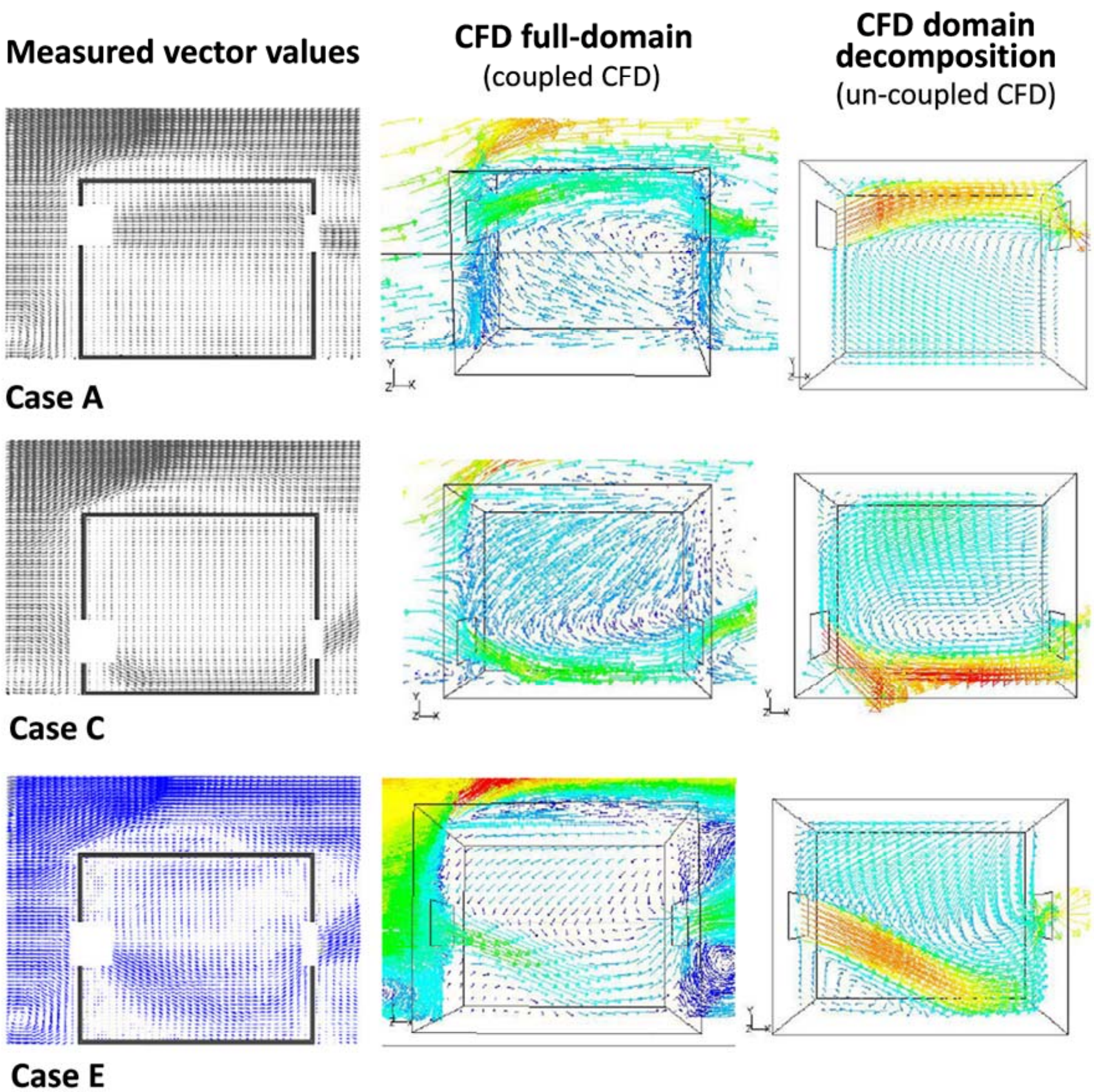


Figure 3.1.4.E: Velocity vectors on building vertical midplane for three case Cases, obtained from measured air flow and coupled and uncoupled flow domains (Meroney, 2009; modified)

Meroney (2009) further describes the internal flow field by means of the ratio of x-component velocity to reference velocity as a function of the distance along the centerline of the space. Figure 3.1.4.F shows an example of the ratio of x-component velocity to reference velocity for Case C (compare Figure 3.1.4.E). The given data suggest measured values and full-domain CFD calculations display a maximum local velocity just downstream of the inlet and outlet openings, but the domain decomposition method does not, although it otherwise reproduces flow variations quite closely. Figure 3.1.4.G shows the variation of static pressures along the same line joining the openings for full-domain and domain decomposition CFD calculations. The author suggests that this data again shows that full-domain (coupled) and decomposed domain can reproduced the variations of internal pressures within the building very closely.

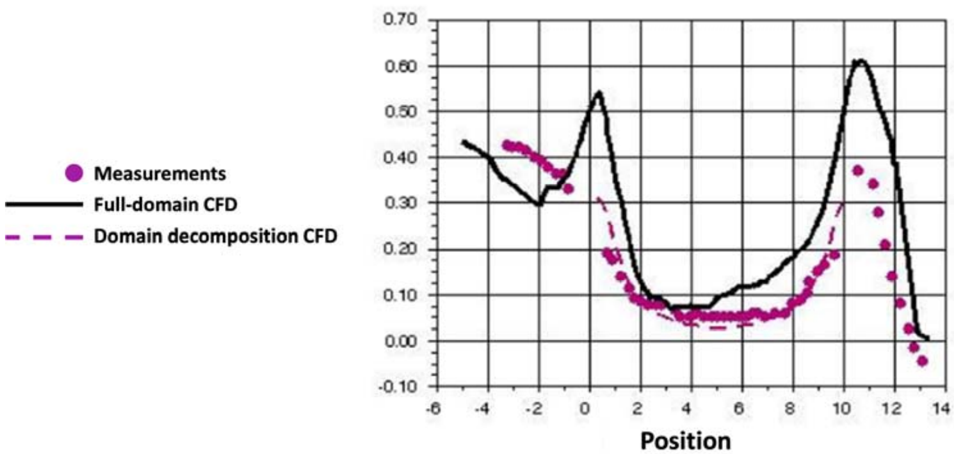


Figure 3.1.4.F: Velocity ratio VX/Vref along line joining inlet and outlet openings for Case C (Meroney, 2009) modified

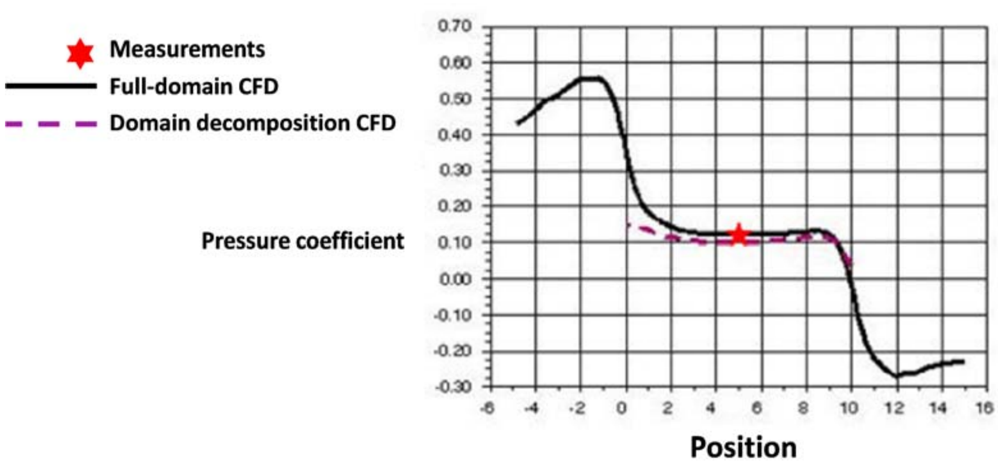


Figure 3.1.4.G: Static pressure coefficient, Cp, along line joining inlet and outlet openings for Case C

3.2 CFD Pre-Processing

3.2.1 Computational Domain Geometry, Grid and Mesh Design

Representation of the surroundings: Regarding to choosing geometrical representation of obstacles, Franke et al. (2007) recommended the required levels of detail in CFD modeling of buildings or features dependent on their distant to the central area of interest. That means the central of interest should be reproduced with highly detailed while the buildings far away can be represented as simple blocks.

For the actual urban area, Tominaga et. al (2008) suggested that the buildings in the region to be assessed (generally 1–2H radius from the target building) should be clearly modeled. Moreover, at least one additional street block in each direction around the assessment region should also be clearly reproduced. In addition, the authors recommend using simplified geometries of a cluster of buildings or to specify appropriate roughness lengths z_o for the ground surface boundary condition to represent the roughness of the outer region (from the outer edge of the additional street blocks to the boundary of the computational domain).

To simulate the aerodynamic effects of small-scale obstacles such as small buildings, sign boards, trees and moving automobiles, etc., canopy models are normally added additional terms to the basic flow equations in order to decrease wind velocity but increase turbulence. In particular, tree planting is one of the most popular measures for improving the pedestrian wind environment. Tominaga et. al (2008) provided citation on various tree canopy models and comparison of the predictions of various canopy models with field measurements of flows around trees.

Computational domain size: Regarding to the necessary extension of the computational domain, Tominaga et. al (2008) recommended the blockage ratio should be less than 3%. For a single-building model, the lateral and the top boundary should be set at least 5H away from the building, where H is the building height. The distance between the inlet boundary and the building should be set to correspond to the upwind area covered by a smooth floor in the wind tunnel. The outflow boundary should be set at least 10H behind the building. Where the building surroundings are included, the height of the computational domain should be set to correspond to the boundary layer height determined by the terrain category of the surroundings (Architectural Institute of Japan, 2004). The lateral size of the computational domain should extend about 5H from the outer edges of the target building and the buildings included in the computational domain should not exceed the recommended blockage ratio (3%).

Franke et al. (2007) summarized on the overall size of the computational domain which depends on the represented area and the boundary condition that will be used. Particularly, the extent of the built area (e.g. buildings, structures or topography) that is represented in the computational domain depends on the influence of the features on the region of interest. A building with building height H causes a minimal effects on an area which is away from 6-10H. It is also recommended to perform simulations with and without the features (i.e. larger and smaller built areas) when there is uncertain about the influence of these features on the flows and dispersion in the area of interest. The extension of the computation domain is recommended as follows:

a. The vertical extension of the domain:

- The vertical extension of the domain is the distance from the roof of a building to the top of the computational domain should be at least 5H, where H is the building H.
- Another recommendation relating to the blockage ratio (the ratio between the projected area of the building in the flow direction to the free cross section of the computational domain), the vertical extension should be 4H for small blockage ratio and 10H for a larger blockage ratio. The maximum blockage ratio is quite different between wind tunnel modeling guidance (below 10%) and CFD (3%).
- In case of urban area with multiple buildings, the vertical extension should be $5H_{max}$, where H_{max} is the height of the tallest building in the area of interest.

b. The lateral extension of the domain:

- The lateral extension should be 2.3H for a computational domain having the height of 6H and the blockage requirement of 3%, where H is the building height. Another recommendation suggested using the domain height of 5H and the blockage ratio of 1.5%.
- In case of urban area with multiple buildings, the lateral extension can be placed closer than 5Hmax.

c. The extension of the domain in flow direction:

- The approach flow distance of the domain (the distance of the region in the front of the domain) should be 5H- 8H if the approach flow profile are well known or even larger if this profile is not available.
- If blockage ratio is minimal, the approach flow distance of the domain should be as small as 2H while a larger blockage ratio (e.g. 10%) requires a distance of 8H.
- The wake flow distance of the domain ((the distance of the region behind the domain) should be 15H for a single building.

Boundary conditions

According to Franke et al. (2007), boundary conditions, represent the cut-off surrounding and determine the solution inside the computational domain. Selecting the proper boundary conditions is very important and suggested as follows:

<u>The inflow boundary condition:</u> For the inflow boundary condition, Franke et al. (2007) suggests to use the mean velocity profile which is usually obtained from the logarithmic profile corresponding to the upwind terrain via the roughness length or from the profiles of the wind tunnel simulations. Available information from nearby meteorological stations is used to determine the wind speed at the reference height. Steady RANS solvers also requires both mean velocity profile, information and turbulence

quantities. To obtain these profiles, different strategies such as the equilibrium boundary layer or turbulent kinetic energy from wind tunnel studies are also are provided.

While the recommended inflow profile by Franke et al. (2007) are based on an assumed value of the roughness parameter z_o, Tominaga et. al (2008) suggests a different method to calculate the vertical profile based on the assumption of using the power-law exponent and consistent with the wind load estimation method in Japan. And therefore, the height of the computational domain recommended by Franke et al. (2007) is assumed much lower than the atmospheric boundary layer height, because the assumption of constant shear stresses is only valid in the lower part of the atmospheric boundary layer. Therefore, it is necessary to pay attention to the relationship between the height of the computational domain and the atmospheric boundary layer.

Franke et al. (2007) also recommends "one method to generate inflow profiles, which will not change within the computational domain in front of the built area is to fist perform a simulation in the empty domain with the same grid and periodic boundary conditions to obtain constant profiles that match the velocity measurements at the meteorological station."

For Large Eddy Simulation (LES) and other unsteady simulation approach, the boundary conditions are dependent to time. Franke et al. (2007) recommends several methods such as artificial stochastic inflow data generation methods based on statistical description of turbulence, the usage of a fetch method comparable to ones used in wind tunnels and the method of using periodic simulations over roughness elements.

An example of a study of wind-induced pedestrian comfort around the large football stadium in an urban environment (Blocken and Person 2009) delineates how to obtain the mean velocity profile for inflow boundary conditions, implemented the estimation of the aerodynamic roughness length y0,dav which was used for the wall-function. For an heterogeneity of surrounding terrain resulting different roughness levels from different directions, a good estimate for yo,dav is difficult and therefore a representative aerodynamic roughness length was calculated based on the NPR 6097 software using high-resolution roughness map.

Wall boundary conditions: The solid walls will be assigned as non-slip boundary conditions for velocity. For shear stress at smooth walls, Franke et al. (2007) discussed two different approaches for RANS simulation and LES: the low-Reynolds number approach and the wall function approach. The low-Reynolds number approach resolves the viscous sub-layer and computes the wall shear stress from the local velocity gradient normal to the wall, requiring a very fine mesh resolution in wall-normal direction. In order to reduce the number of grid points in the wall-normal direction and therefore the computational costs, the wall function approach is can be used as an alternative approach to compute the wall shear stress based on the assumption of a logarithmic velocity profile between the wall and the first computational node in the wall-normal direction. Beside these approaches, there is also an alternative called the distributed roughness approach to model the roughness elements with a porous region with prescribed losses.

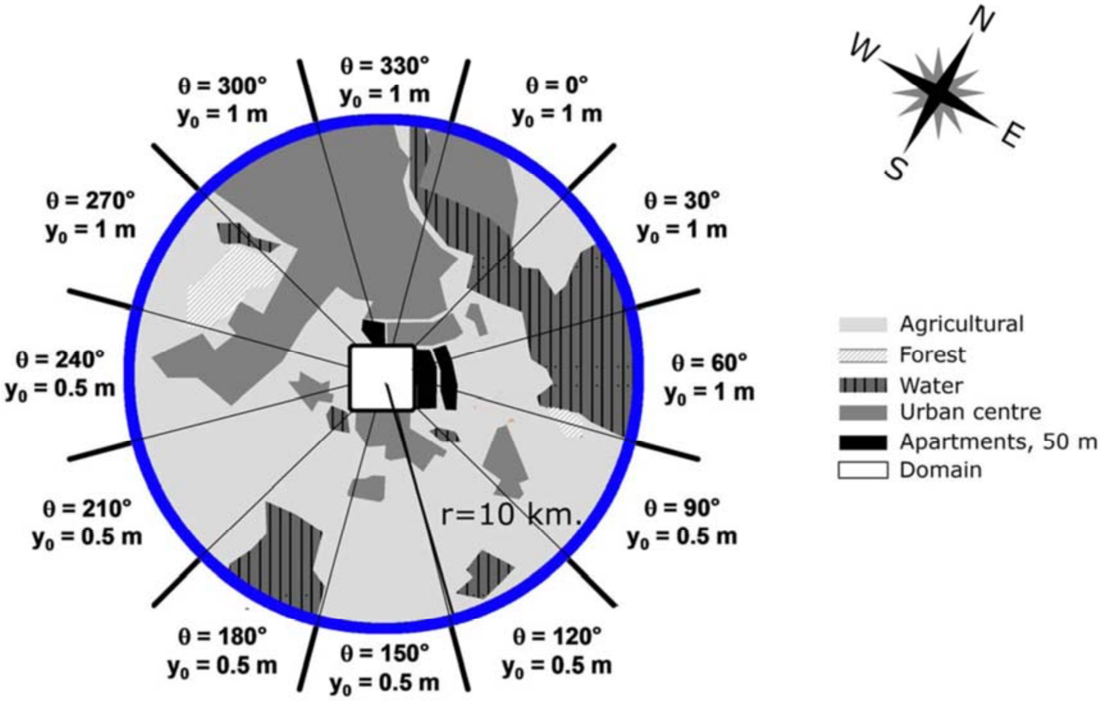


Figure 3.2.1A: Terrain surrounding the stadium with a radius of 10 km and estimated aerodynamic roughness lengths y0. The white square represents the computational domain used in this study (Blocken and Person, 2009).

For shear stress at rough walls, such as for of urban areas, the wall function approach can be used. In this case, the roughness is included by the hydrodynamic roughness length and the corresponding modifications of the turbulence quantities at the first computational node off the wall are derived assuming an equilibrium boundary layer flow. Franke et al. (2007) summarized the wall roughness which is implemented in commercial general purpose CFD codes.

<u>Top and lateral boundary conditions:</u> The top boundary layer is assumed to have a constant sheer stress and therefore is normally prescribed a constant value for the top of the boundary condition. Another method is to provide the values for the velocities and the turbulence quantities of the inflow profile at the height of the top boundary over the entire top boundary Franke et al. (2007).

In case different wind directions are to be simulated with the same computational domain, the lateral boundaries become inflow and outflow boundaries with corresponding boundary conditions, the boundary should be positioned far enough away from the built area of interest to not lead to an artificial acceleration of the flow in the region of interest. In micro-scale obstacle-accommodating meteorological models, open lateral radiation boundaries are frequently used at the lateral boundaries. When using this boundary condition, the model domain can be smaller in the lateral direction and it can be sufficient to include only a few grid points between a building that is close to the boundaries and the boundary Franke et al. (2007).

According to Tominaga et. al (2008), If the computational domain is large enough, the boundary conditions for lateral and upper surfaces do not have significant influences on the calculated results around the target building. Using the inviscid wall condition (normal velocity component and normal gradients of tangential velocity components set to zero) with a large computational domain will make the computation more stable.

Outflow boundary conditions*:* The outflow boundary conditions, which should be ideally far enough away from the built areas to avoid any fluid entering into the computational from this boundary condition, are normally defined as open boundary condition in commercial CFD and are assigned as either outflow or constant static pressure boundary conditions Franke et al. (2007).

Tominaga et. al (2008) suggested to used common practices to set the normal gradients of all variables to zero for the outflow boundary condition. The outflow boundary needs to be placed far from the target building in order to create a negligible influence on flow conditions.

3.2.2 Selection of Type of Mesh and Cells

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The attributes associated with mesh quality are node point distribution, smoothness, and skewness. Depending on the cell types in the mesh (tetrahedral, hexahedral, polyhedral, etc.), different quality criteria are delineated in guidelines (ANSYS, 2009):

- Cell squish on all meshes. Cell squish is a measure used to quantify how far a cell deviates from orthogonality with respect to its faces. *Squish index* is computed for cells using the vector from the cell centroid to each of its faces and the corresponding face area vector. A poorly generated cell has a cell squish index close to 1, with better cells having smaller indices, ideally close to zero. For tetrahedral meshes, either skewness or cell squish index can be used to measure the mesh quality. Skewness information is not available for polyhedral meshes therefore must be relied on the cell squish index and an additional index for the face squish (which is computed using the vector connecting the centroids of adjacent cells). A good rule of thumb is that the maximum skewness for tetrahedral cells should less be than 0.95. The maximum cell squish index for all types of cells should be less than 0.99.
- *Skewness* is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. As a general rule the maximum skewness for a triangular/ tetrahedral mesh in most flows should be kept below 0.95, with an average value that is less than 0.33. A maximum value above 0.95 may lead to convergence difficulties and may require

changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

- The aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the distances between the cell centroid and face centroids, and the distances between the cell centroid and nodes. For a unit cube (Fig.), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

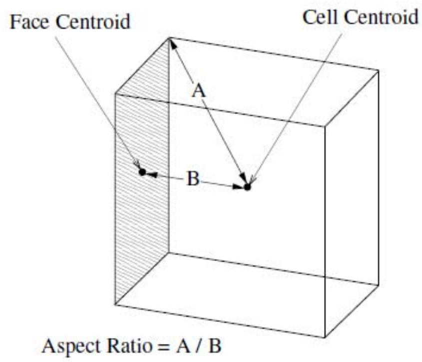


Figure 3.2.2.A: The aspect ratio for a unit cube (ANSYS, 2009)

Other best practices on computational grids by Franke et al. (2007) summarized several recommendations on the choice of computation grid which affects the computational expense and accuracy. According to these best practices, the grid has to be constructed to: minimize the errors, have adequate resolution and capture the important physic phenomena.

Here are some recommendations:

- Ideally the grid is equidistant.
- The expansion ratio between two consecutive cells should be below 1.3 in these regions.
- The angle between the normal vector of a cell surface and the line connecting the midpoints of the neighboring cells, ideally these should be parallel.
- Hexahedra are preferable to tetrahedral, as the former are known to introduce smaller truncation errors and display better iterative convergence.
- On walls the grid lines should be perpendicular to the wall.
- Prismatic cells should be used at the wall with tetrahedral cells away from the wall.
- In the area of interest, at least 10 cells per cube root of the building volume should be used and 10 cells per building separation to simulate flow fields as an initial minimum grid resolution.
- When a global systematic grid refinement is not possible due to resource limitations, then at least a local grid refinement should be used in the area of the main interest.

Hefny and Ooka (2009) assessed two discretization methods relating to mesh generation: tetrahedral-based meshing and hexahedra-base meshing. Hexahedral-based meshing requires significant time and effort while tetrahedral-based meshing can be constructed much faster in complex geometries. To do so, the authors carried out a CFD analysis of pollutant dispersion around a buildings using two different cell shapes with four different resolutions ranging from 150,000 to 1,000,000 control volumes. The quality of their solution was determined on the observed gas concentration as well as the quantitative grid convergence, which was calculated based on a grid convergence index (GCI).

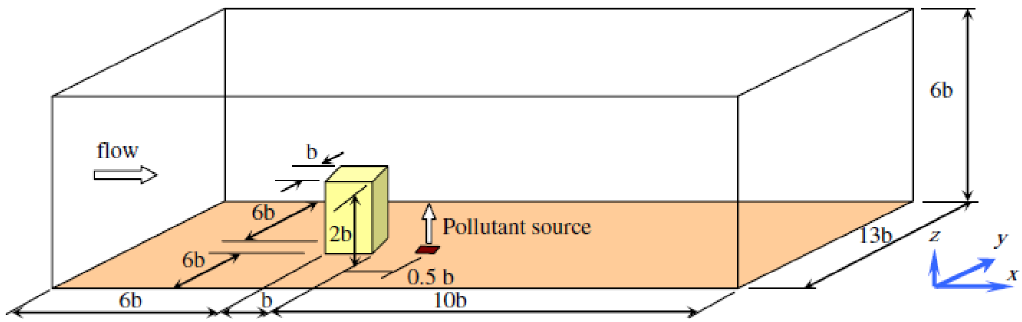


Figure 3.2.2.B: Computational domain (Hefny and Ooka, 2009)

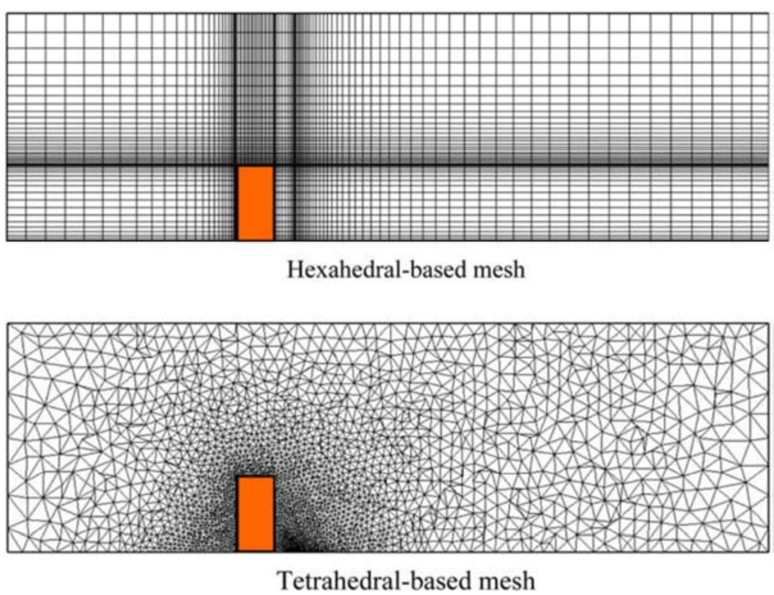


Figure 3.2.2.C: Hexahedral-based and Tetrahedral-based mesh (Hefny and Ooka, 2009)

Table 3.2.2.D: Details of computational cases (Hefny and Ooka, 2009)

Table 1
Details of computational cases.

Grid resolution	Subdivisions ($x \times y \times z$)	Mesh		Min. grid spacing
		Cells (approximate)	Vertexes (approximate)	
Hexahedral-based meshes				
Coarse (H1)	$90 \times 48 \times 35$	150,000	163,800	0.07
Medium (H2)	$114 \times 94 \times 44$	300,000	322,015	0.05
Fine (H3)	$143 \times 118 \times 55$	600,000	634,932	0.035
Very fine (H4)	$169 \times 140 \times 65$	1,000,000	1,048,675	0.028
Tetrahedral-based meshes				
Coarse (T1)	–	150,000	27,210	0.07
Medium (T2)	–	300,000	53,780	0.05
Fine (T3)	–	600,000	105,342	0.035
Very fine (T4)	–	1,000,000	175,885	0.028

Hefny and Ooka (2009) found that hexahedral-based meshing produced higher quality solution than tetrahedral meshing. More specifically, the hexahedral-based approach is well able to capture the pollutant concentrations around buildings even in the case a coarse is used while the tetrahedral-based approach failed to adequately capture the pollutant concentrations for most mesh cases considered. The authors points out that where there is no symmetry in the contours around the centerline and inaccurate predictions near the pollutant source. The hexahedral-based approach produced the GCI values (also mean the truncation error values) below those from tetrahedral-based approach for all resolutions. This study results to special consideration when employing an unstructured tetrahedral-based mesh to ensure that the mesh is fine enough and any numerical errors should be documented to assess the quality of the numerical solution.

Biswas and Strawn (1998) presented their work on two unstructured mesh adaptation schemes for CFD problem: the tetrahedral and the hexahedral mesh adaptation procedures using edge-based data structures for efficient subdivision. Tetrahedral mesh adaptation causes poor mesh quality as deploying repeated refinement. Hexahedral meshes have the advantage of allowing for anisotropically subdivided repetition. To eliminate the “hanging vertices” in the hexahedral meshes, pyramids, prisms and tetrahedral mesh adaptation uses as tetrahedral as buffer elements for refinement. Biswas and Strawn (1998) also asserted that adaptive hexahedral meshes and tetrahedral meshes showed good agreement but adaptive hexahedral meshes is more computational efficiency than that of tetrahedral meshes in terms of computer resources and calculating time.

Table 3.2.2.E: Grid convergence index (GCI) for all mesh schemes (Hefny and Ooka, 2009)

Table 2
Grid convergence measures for all mesh schemes.

Cell geometry	Grid size ($\times 10^3$)	*r* value	ε_{rms} (%) c_g	GCI (%) c_g
Hexahedron	300–150	1.26	1.71	2.91
	600–300	1.26	0.29	0.49
	1000–600	1.26	0.19[a]	0.33[a]
Tetrahedron	300–150	1.26	4.54	7.73
	600–300	1.26	2.88	4.90
	1000–600	1.26	0.98	1.68

[a] Minimum value for all cases.

3.2.3 Effects of Scaling

Browne et al (2007) discusses porous screens which as widely used as external cladding on the façades of tall buildings. This paper presents results of wind tunnel testing of porous screens attached to tall building façades with a focus on the geometric scale effects on the wind loading on porous screens. A number of tests, with geometric scale varying between 1:400 and full scale were carried out and the results are provided in this paper. CFD simulations were also performed for comparison purposes. Overall, it was found from this study that the geometric scale effects were minor in nature, and reliable results can be obtained if proper care is taken during the selection of the model-scale screens.

Example of wind tunnel experiments and CFD simulations: Case StudyI – 1: 400 VS. 1:50

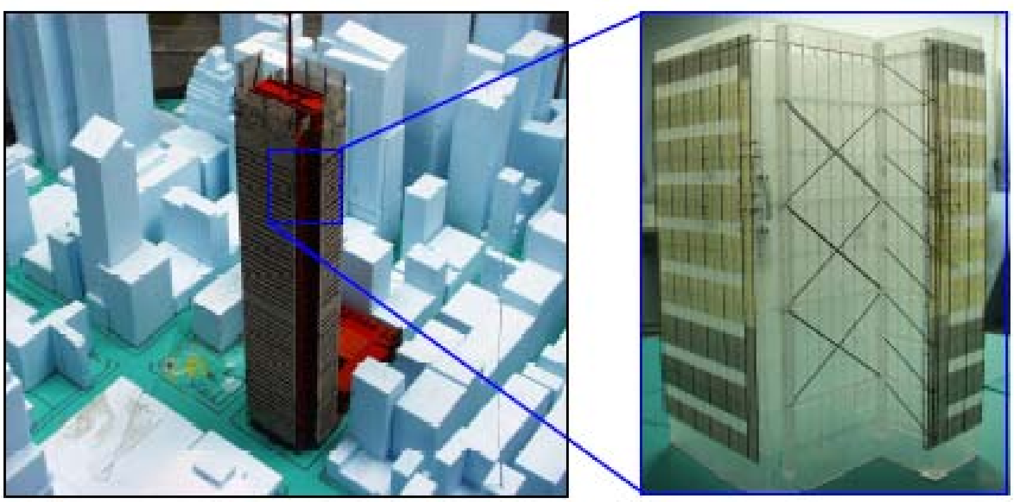


Figure 3.2.3A: Case Study I Wind Tunnel Models (a) 1:400 Scale (left) and (b) 1:50 Scale (right). (Browne and etal, 2007)

For the tested section, the mean wind speed and turbulence properties were made to be consistent between the 1:400 [Figure 3.2.3A-(a)] and 1:50 scale [Figure 3.2.3A-(b)] model tests. As a result, a direct comparison of the pressure coefficients could be made. A total of 92 pressure taps were installed on the front and back surfaces of the ceramic tube screen and on the façade of the 1:50 scale model of the building. The model was tested for selected wind directions (180°-320° clockwise from North), which were chosen to minimize the effects of the immediate surroundings in the 1:400 tests.

Example of wind tunnel experiments and CFD simulations: Case Study II – 1:200 vs. Full Scale

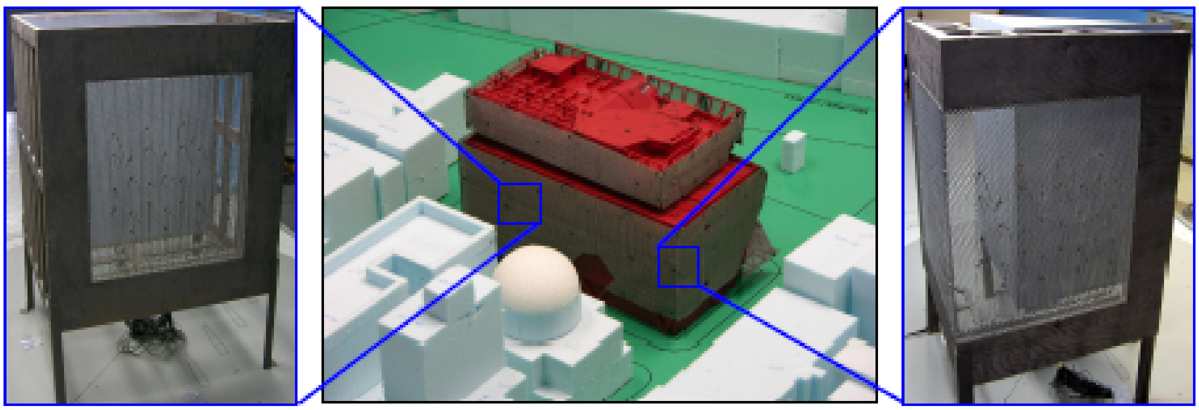


Figure 3.2.3B: Case Study II Wind Tunnel Models (a) 1:200 Scale (centre), (b) Full-Scale Prototype of Central Wall Condition (left), (c) Full-Scale Prototype of Corner Condition (right). (Browne and etal, 2007)

In this case study, the building was 45m in height, located in primarily urban terrain, with the majority of the exterior of the building covered with a perforated plate screen set off the façade between 0.3m and 1.5m. Pressure Drop test and Wind Tunnel test of prototype screens has been testing.

Computational Fluid Dynamics (CFD) Simulations: For comparison purposes a CFD simulation was conducted on a solid cube with the vertical faces surrounded by a screen (0.15m gap) at a scale similar to the full-scale prototype wind tunnel model. The commercial software FLUENT 6.2 was utilized for this simulation, and the governing equations employed were the Reynolds Averaged Navier-Stokes (RANS) equations, together with the RNG k-turbulence model.

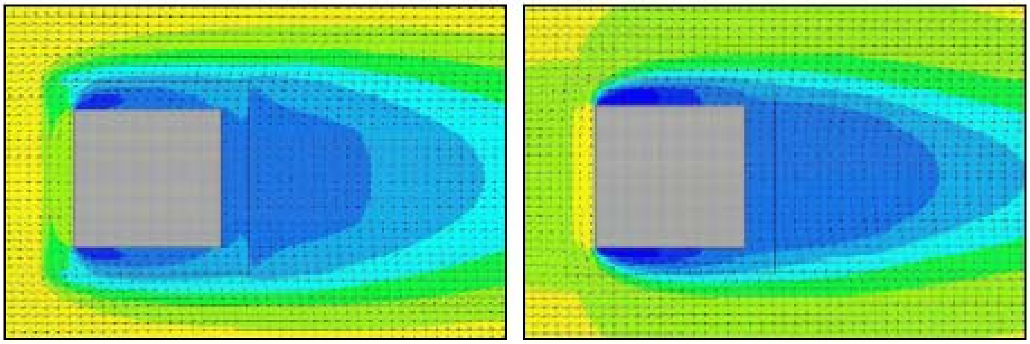


Figure 3.2.3C: Velocity Field and Pressure Contours from CFD Simulation (a) with a 50% porous screen (left), and (b) without screen (Browne and etal, 2007)

The dimensions of the cube were 0.8m wide by 0.8m long by 1m high (where the cube height H = 1m). The inlet, top, outlet, and two sides of the computational domain were set to 6H, 5H, 16H, and 3H from the centre of the cube, respectively. The screen was represented with a pressure drop (or loss) coefficient (k=3.5), which was consistent with the coefficient of the screen used in the wind tunnel tests. In addition, the inlet velocity and turbulence intensity were similar to the profile data used for the high-turbulence wind tunnel tests. A roughness length of 0.02m was applied to the bottom surface of the computational domain.

<u>Result and conclusion:</u> This investigation examined two case studies that compared wind tunnel results of porous screens from scale models ranging between 1:400 and full scale. CFD simulations were also performed for comparison purposes. The study concluded that geometric scale effects were minor in nature. If proper care is taken during the selection of the model-scale screens, reliable results can be obtained.

Lieberman et al (2012) describes a Hessian-based model reduction that was previously proposed as an approach in deriving reduced models for the solution of large-scale linear inverse problems by targeting accuracy in observation outputs. A control theoretic view of Hessian-based model reduction hinges on the equality between the Hessian and the transient observability of the underlying linear system. The model reduction strategy is applied to a large-scale (O(106) degrees of freedom) three-dimensional transport problem in an urban environment, an application that requires real-time computation. In addition to the inversion accuracy the paper addresses the ability of reduced models of varying dimension to make predictions of the contaminant evolution beyond the time horizon of observations. Results indicate that the reduced models have a factor O(1000) speedup in computing time for the same level of accuracy.

Before investigating the convergence of the reduced models with increasing dimension, the author presents a side-by-side comparison of the flow conditions of dispersing contaminants at time steps k = 0, 60, 120 (initial condition, end of observation period, end of prediction period). Figure 3.2.3D, illustrates a horizontal slice of the state at each time for three cases: (i) the actual initial condition; (ii) inversion and prediction using the full finite element model with n D 1, 007, 412 (p = 1); and (iii) inversion and prediction using a reduced model of dimension m D 800. Although the full and reduced model inversions do not recover the initial condition exactly, they do locate the most concentrated regions with the contaminant. The figure suggests that the reduced model has picked up all of the features of the full inversion with only 800 basis vectors, whereas the full model contains over 1 million degrees of freedom. This large reduction in dimension translates directly to computational savings in the inversion (see table 3.2.5A)

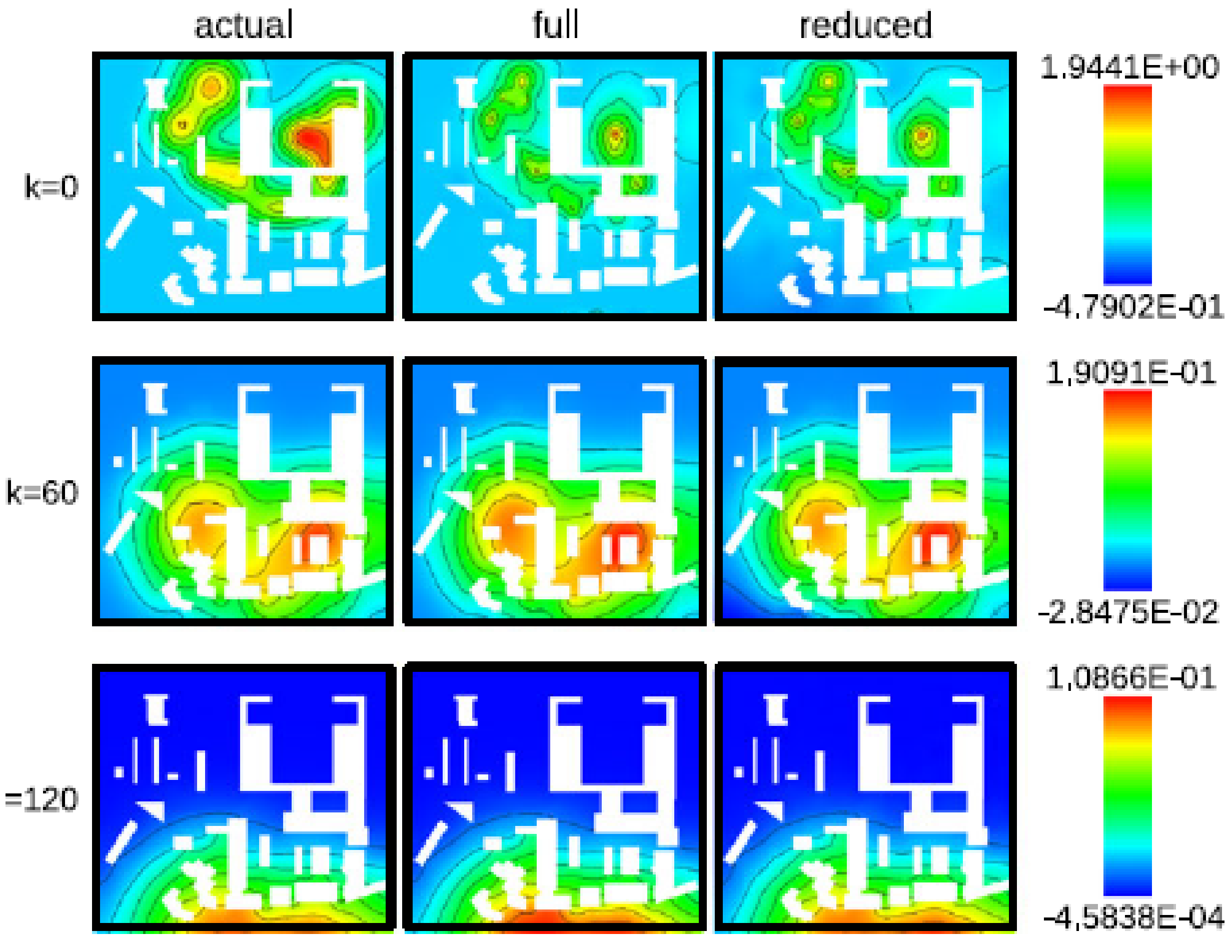


Figure 3.2.3D: Comparison between the actual contamination event (left column), inversion and prediction by the full model (center column), and inversion and prediction by the reduced model of dimension m = 800 (right column) at the initial condition k =0 (top row), at the end of the observation period k D 60 (middle row), and at the end of the prediction period k = 120 (end row). All plots are cuts through z = 0.1, a representative slice. (Lieberman and etal, 2012)

Table 3.2.3F: CPU timings and percent relative errors for the full and reduced model inversions and predictions.

	Inversion		Prediction	
Model	CPU time	Error (%)	CPU time	Error (%)
Full $n = 1\ 007\ 412$	3300 h	57	7.8 h	5.3
Reduced $m = 800$	68 min	62	3.7 s	3.1
Reduced $m = 600$	47 min	55	1.7 s	3.9
Reduced $m = 400$	3.5 min	58	1.4 s	11
Reduced $m = 200$	16 s	171	0.6 s	58

Section 3.2.4: Domain Inflow and Outflow Air Flow Modeling

The reviewed literature indicates the importance of considering the upwind velocity wind profiles for CFD simulations. The following lists several authors which have addressed the wind velocity profile and related issues of fetch as they affect the quality of the CFD simulations.

The CFD analysis requires the appropriate setting of the upwind condition. Upwind conditions here are referred to as relative to the structure to be tested in the numerical wind tunnel (e.g. CFD application domain). Good et al (2008) suggests that boundary conditions for the atmospheric boundary layer have to be applied. The authors indicate that the atmospheric boundary layer profile has to be appropriate for the upwind terrain. Goode et al (2008) suggest that this typically means a power-law representation for the velocity profile.

Li and Ward (2007) suggest that the urban wind profile at the inlet boundary is the most important issue, which determines development the whole domain turbulent flow. The authors describe that there are two types of methods available to describe it in CFD. The most common one is the power law (exponent decided by terrain types) or logarithmic law (friction velocity and roughness length) determines from wind tunnel experiments or field studies. The authors describe that applications of the power and logarithmic laws may be useful for the region above the urban roughness sublayer, but they fail to describe the region below the building height, which is much more important for urban natural ventilation.

Li and Ward (2007) point out that an alternative approach in CFD is to directly simulate the roughness elements. The authors point out the challenges of determination of the velocity profile through simulating roughness elements since it results in a significant amount of computational time being devoted to simulating these elements rather than concentrating on the buildings under consideration.

Li and Ward (2007) suggest that an appropriate approach to define the wind velocity profile can be found by subdividing the simulation into two steps. The first step is to acquire the urban wind profile after the infinite length of urban roughness elements, which is then used as input to the CFD domain.

The CFD domain size consists of the fully developed flow region and the area in which the building models are situated. Figure 3.2.4.A below illustrates the division of the domain into the two regions, the fully developed flow region followed by the working area, e.g. the area where CFD is used to determine the pressure coefficients for natural ventilation effectiveness.

Li and Ward (2007) suggest to reduce computing time by establishing units of surface roughness elements rather than simulating individual roughness elements. Using this approach the authors suggest a good correlation of achievable wind velocity profiles with established methods as well as wind tunnel experiments. Figure 3.2.4.B illustrates the correlation of the ration of wind speed at a certain height to the free stream wind velocity. (U= wind velocity at certain height; UG = free stream wind velocity).

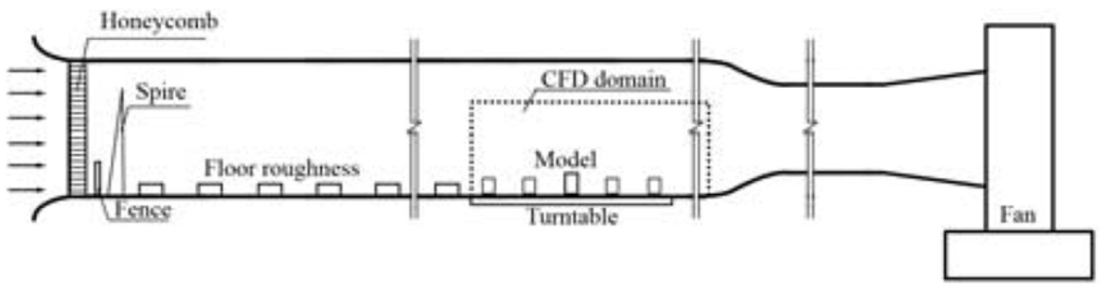


Figure 3.2.4.A Schematic of CFD domain in wind tunnel (Li and Ward, 2007)

Li and Ward (2007) provide citations of recommendations for the required upwind, lateral and downwind fetch by different researchers. The magnitudes of the fetch is hereby dependent on the several parameters, such as the layout, density and height of the building models. The authors suggest that a typical example of an appropriate upwind fetch is about 5 to 6 times the height of the buildings. The authors point out that the resulting pressure coefficients are significantly affected by the ratio of the fetch to the building height. An example of influence of the fetch to building height ration on the resulting pressure coefficients calculated of the tested buildings can be seen in Figures 3.2.4.C. The figure suggests minimum values for upwind fetch expressed as R/H (R=fetch, here upwind fetch; representative H=building height) as a function of layout pattern types of the investigated buildings as well as and the density of the layout.

Figure 3.2.6.C indicated the significant effect of flow regimes on the wind-induced ventilation potential. The %-values shown in Figure 3.2.6.C indicate 5% as “isolated flow regime” 10% as “wake interference flow regime” and 20% as “skim flow regime”.

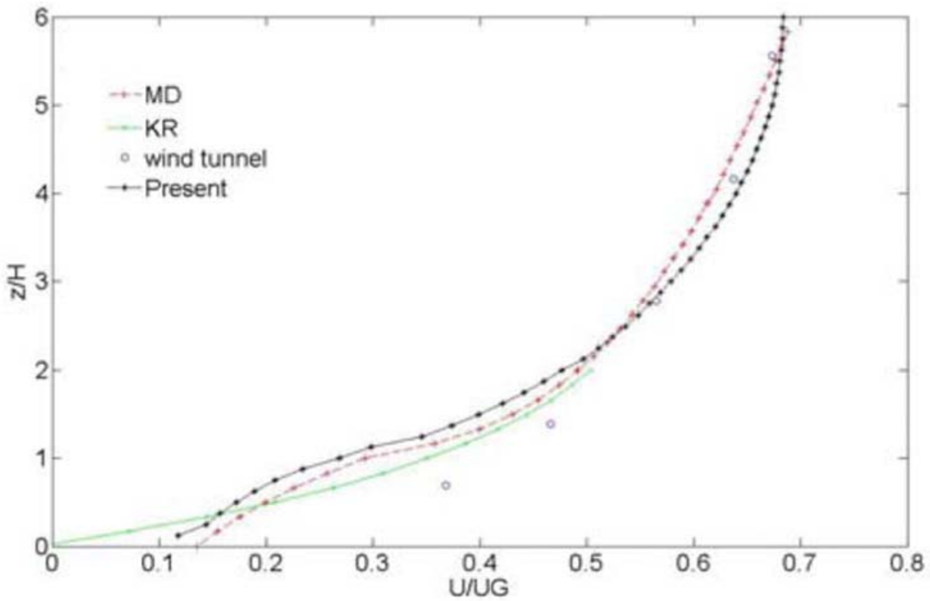


Figure 3.2.4.B Mean velocity profile in urban roughness elements region(Li and Ward, 2007)

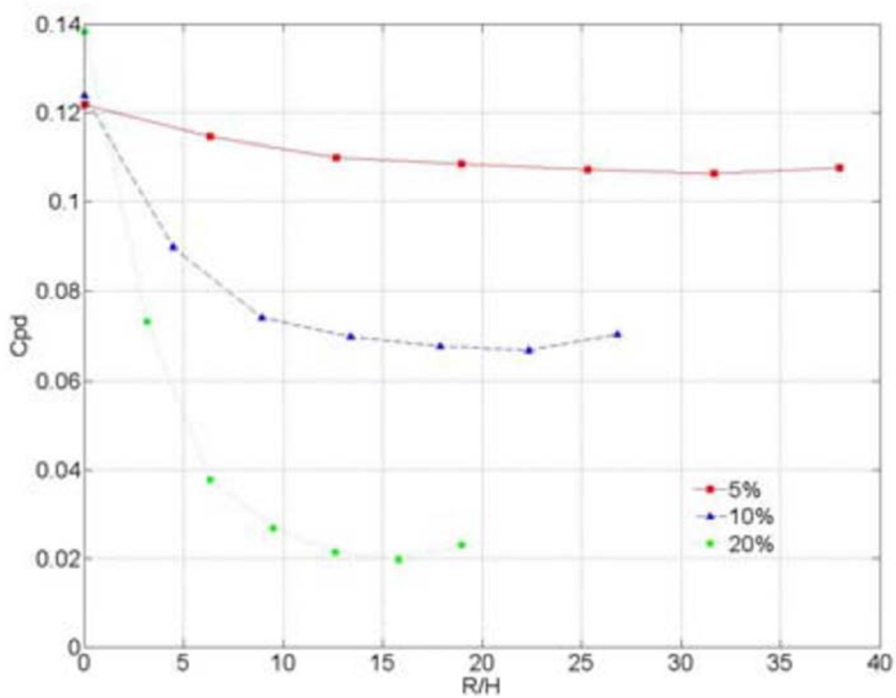


Figure 3.2.4.C Example of the effect of fetch on the pressure coefficient difference– staggered pattern (Li and Ward, 2007)

Li and Ward (2007) provide a recommendation about the required longitudinal and lateral fetch for CFD investigations and compare the CFD fetch with values used in wind tunnels (Figure 3.2.4.D).

Blocken et al (2007) suggest that commercial CFD products might have challenges to correctly describe the ABL. The authors suggest that the accuracy of such simulations can be seriously compromised when wall function roughness modifications based on experimental data for sand grain roughened pipes and channels are applied at the bottom of the computational domain. The unintended consequences are stream wise gradients in the vertical mean wind speed and turbulence profiles as the travel though the computational domain.

Authors	Method	Longitudinal Fetch upstream	Longitudinal Fetch downstream	Lateral Fetch
Cook (1972)	Wind tunnel	5 rows		
Soliman (1976)	Wind tunnel	12H		
Lee, et al (1979)	Wind tunnel	19H	11H	8H
Kiefer & Plate (1998)	Wind tunnel	5H	5H	5H
Franke et al. (2004)	experience	6~10H	6H	6H
Present	CFD	Normal 10H	5H	5H
		Staggered 15H	6H	6H

Figure 3.2.4.D Neighborhood scale for urban natural ventilation(Li and Ward, 2007)

The article indicates that accurate simulation of ABL flow in the computational domain is imperative to obtain accurate and reliable predictions of the related atmospheric processes. In a CFD simulation, the flow profiles of mean wind speed and turbulence quantities that are applied at the inlet plane of the computational domain are generally fully developed, equilibrium profiles. These profiles are representative of the terrain outside the computational domain, e.g. terrain upstream of the inlet plane. Usually the profiles are described as an appropriate roughness or power law.

Brocken et al (2007) suggest that it is important to distinguish between three regions in the computational domain as depicted in Figure 3.2.6.E. The authors suggest that wall functions can be applied to the bottom of the domain to characterize the appropriate terrain roughness. The authors describe the importance of horizontally homogeneous ABL over uniformly rough terrain, which refers to the absence of streamwise gradients in the vertical profiles of the mean wind speed and turbulence quantities, i.e. these profiles are maintained with downstream distance. The flow types are distinguished between inlet flow, approach flow and incident flow (see figure 3.2.6.E). The authors suggest that while simulating a simple case of a horizontally homogeneous ABL flow suggest that similar or maybe even more serious problems can be expected when more complex cases of ABL flow have to be simulated, e.g. the development of internal boundary layers (IBL) over terrains with roughness changes.

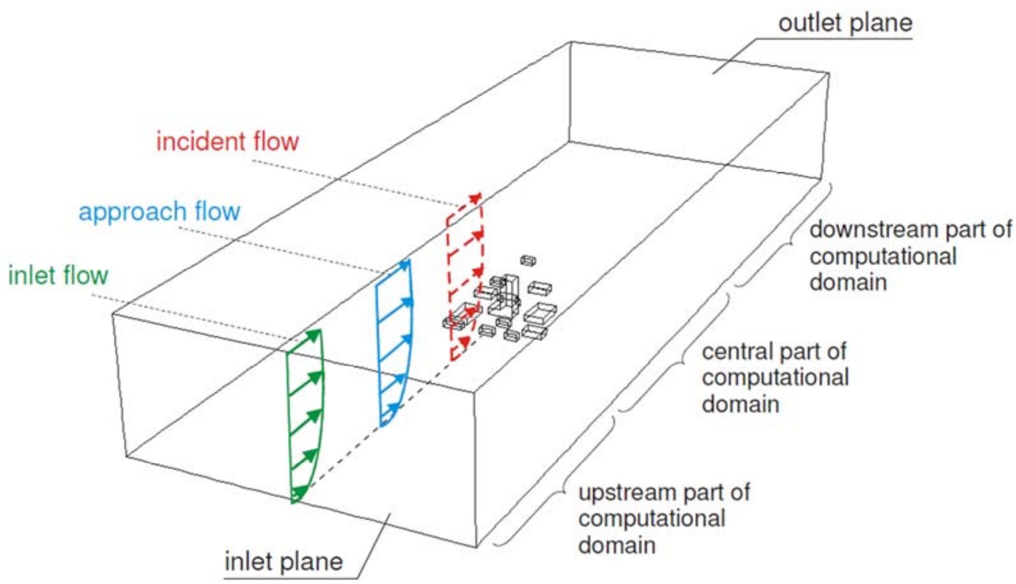


Figure 3.2.4.E Computational domain with building models for CFD simulation of ABL flow – definition of inlet flow, approach flow and incident flow and indication of different parts in the domain for roughness modeling (Brocken et al, 2007)

Brocken et al (2007) indicates that in almost all CFD simulations of the lower part of the ABL, an accurate description of the flow near the ground surface is required. The authors indicated that in cases, where the wall roughness is expressed by an equivalent sand-grain roughness kS in the wall functions, four requirements should be simultaneously satisfied.

Brocken et al (2007) describe profiles of mean streamwise wind speed and turbulent kinetic energy which are used when no actually measured values of these parameters are not known. These profiles are not only used for simulations with the standard k-e model but also with other types of turbulence models: RNG k-e, realizable k-e, standard k-w, SST k-w, , the Reynolds Stress model, etc.. Therefore they are universal.

The authors suggest several remedial measures and wall functions to improve the quality of the ABL description. One of which is the variable height of wall-adjacent cells. This means that cells adjacent to the bottom of the computational domain have different initial vertical cell heights in different areas of the domain. When a coarse mesh in employed at the inlet plane the analytical, continuous inlet profiles imposed at the inlet are discretized for input into the simulation. The result is that the continuous inlet profiles will be converted into rough discrete profiles (as depicted in Figure 3.2.6.F)

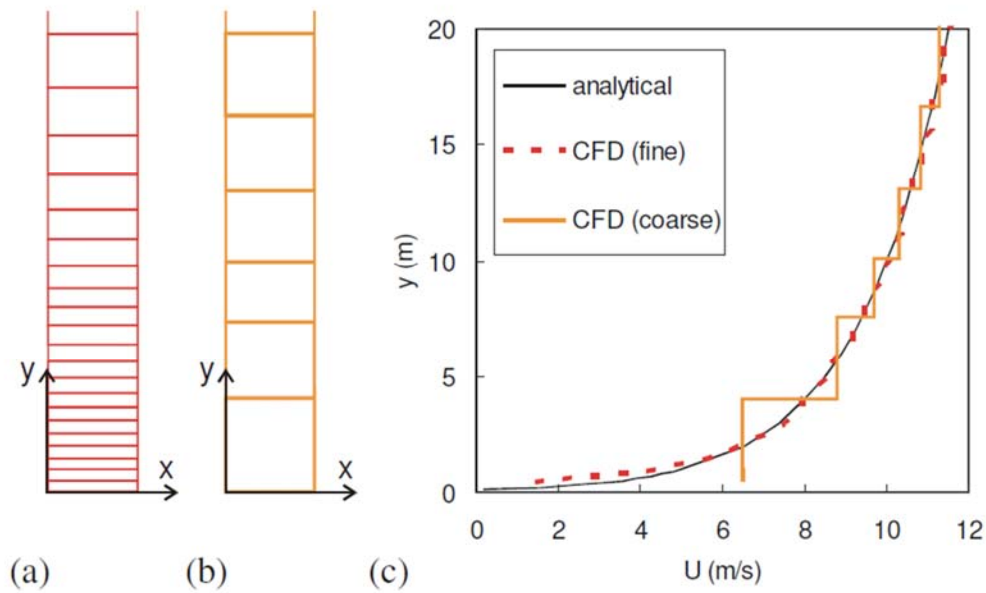


Figure 3.2.4.F (a-b) Fine and coarse vertical near-ground mesh distribution at the inlet plane. (c) Corresponding mean wind speed profile at the inlet of the domain: the analytical profile (imposed boundary condition) and the CFD profiles (one variable value per cell) for the fine and the coarse mesh distribution.

Brocken et al (2007) suggest an approach to mitigate unintended changes (streamwise gradients) in thevertical mean wind speed and turbulence profiles as they travel through the computational domain. These gradients can result in discrepancies sometimes found between seemingly identical CFD simulations performed with different CFD codes and between CFD simulations and measurements. The authors suggest a mitigation measure that is not supported by most commercial CFD codes and which would require changes to their codes.

Brocken et al (2007) point out that irrespective of the type of simulation, the inlet profiles, turbulence model, wall functions and near-wall grid resolution used, it is advisable to always assess the extent of horizontal inhomogeneity by a simulation in an empty computational domain prior to the actual simulation with the obstacle models present. Sensitivity tests in an empty computational domain are of critical importance. In addition, for every CFD simulation it is advisable to always report not only the inlet profiles but also the incident flow profiles obtained from the simulation

3.2.5 Validation of CFD Simulations

In the numerical natural ventilation study of indoor air quality of a large semi-enclosed stadium, Hooff and Blocken (2012) used a coupled CFD approach which consists of modeling the urban wind flow and indoor air flow simultaneously and within the same computational domain, presenting both the wind flow pattern around the buildings as well as the indoor airflow. The validation of this study therefore required high quality experimental data from either wind tunnel or on-site measurement. The advantages and disadvantages of reduced-scale wind tunnel and full-scale on-site measurement are described as follows (Table 3.2.7.1).

Method	Advantages	Disadvantages
Reduced-scale wind tunnel	- Repeatability - Controlled boundary condition	- Small ventilation openings - Similarity requirements (Reynolds, Grashof and Richardson number)
Full-scale on-site measurement	- Real condition to be measured - Insight understanding the physical process indoor and outdoor - Feasibility of measuring complex physical phenomena (rain, heat, moisture, combined of wind and heat…)	- Lack of repeatability - Uncontrolled boundary condition due to heterogeneous surrounding terrain

Table 3.2.5.A: Comparison between reduced-scale wind tunnel and. full-scale on-site measurements

Hooff and Blocken, (2012) presented their solutions on the limitations of on-site measurement for validation. For example, the number of positions for measuring temperature, indoor air speed, RH and CO_2 concentration was limited due to practical and financial constraints. However, the locations of these four measurement units were selected carefully, in such a way that possible difference dues to solar irradiance and due to possible thermal stratification inside the stadium could be detected.

One important notice from on-site data measurement mentioned by Bloken and Person (2009) is that to eliminate the local thermal effect on measured wind data values. To do so, only measured wind speed values at the desk larger than 2m/s were retained and only measurement data that showed sufficiently stable wind direction values of at least an hourly interval were retained and transformed into hourly values. That is because this measurement will be used for CFD validation, and the CFD simulation will performed for isothermal conditions.

<u>An example of CFD validation with full-scale on-site measurement and reduced-scale wind tunnel experiment:</u> Tominaga et. al (2005) compared the results of wind environments at pedestrian level within high-rise building area in Shinjuku (Fig. 3.2.7.1) with numerical analysis obtained by different CFD codes ad validated by full-scale on-site measurement and reduced-scale measurement from wind tunnel experiment. Different grid discretization methods were applied: unstructured grid and structured grid. Wind data with velocities at least 5m/s were extracted from on-site data measurement.

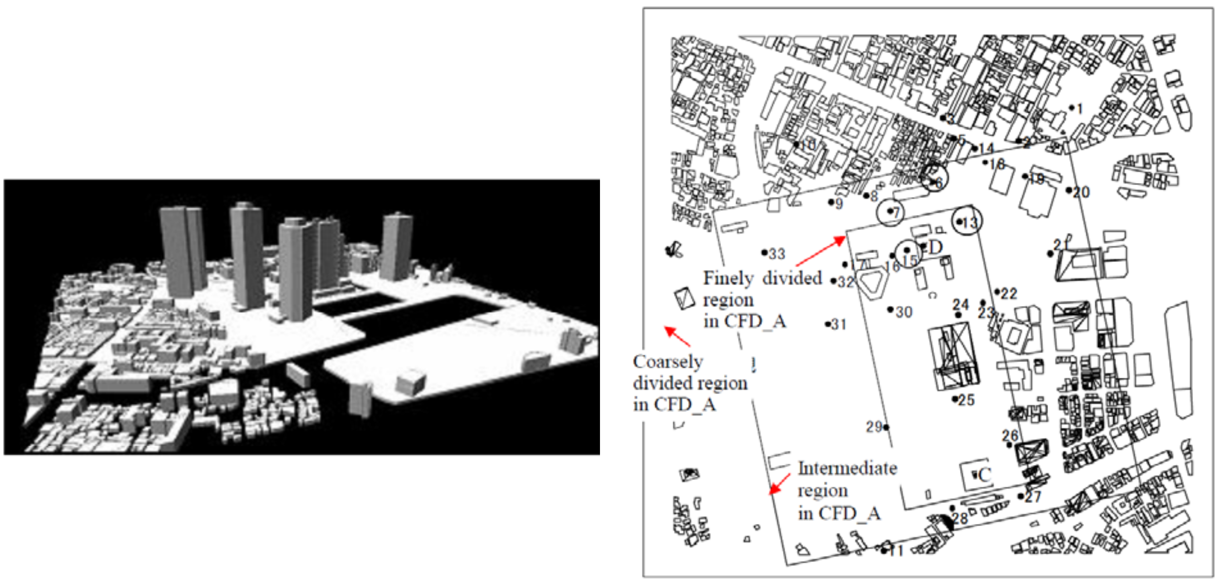


Figure3.2.5.B: CAD data of urban area and measuring points Tominaga et. al (2005)

The validation of CFD results was based on wind speed ratios at 16 representative measuring points from wind tunnel experiment and on-site measurement. The study found a general agreement between CFD results with those of the wind tunnel experiment and field measurement can be achieved with sufficient grid resolution around target buildings (Fig. 3.2.7.2). However, in regions such as around low-rise houses where the grid resolution strongly influences the reproducibility of configurations, differences were observed in the CFD results due to differences in grid resolution Tominaga et. al (2005).

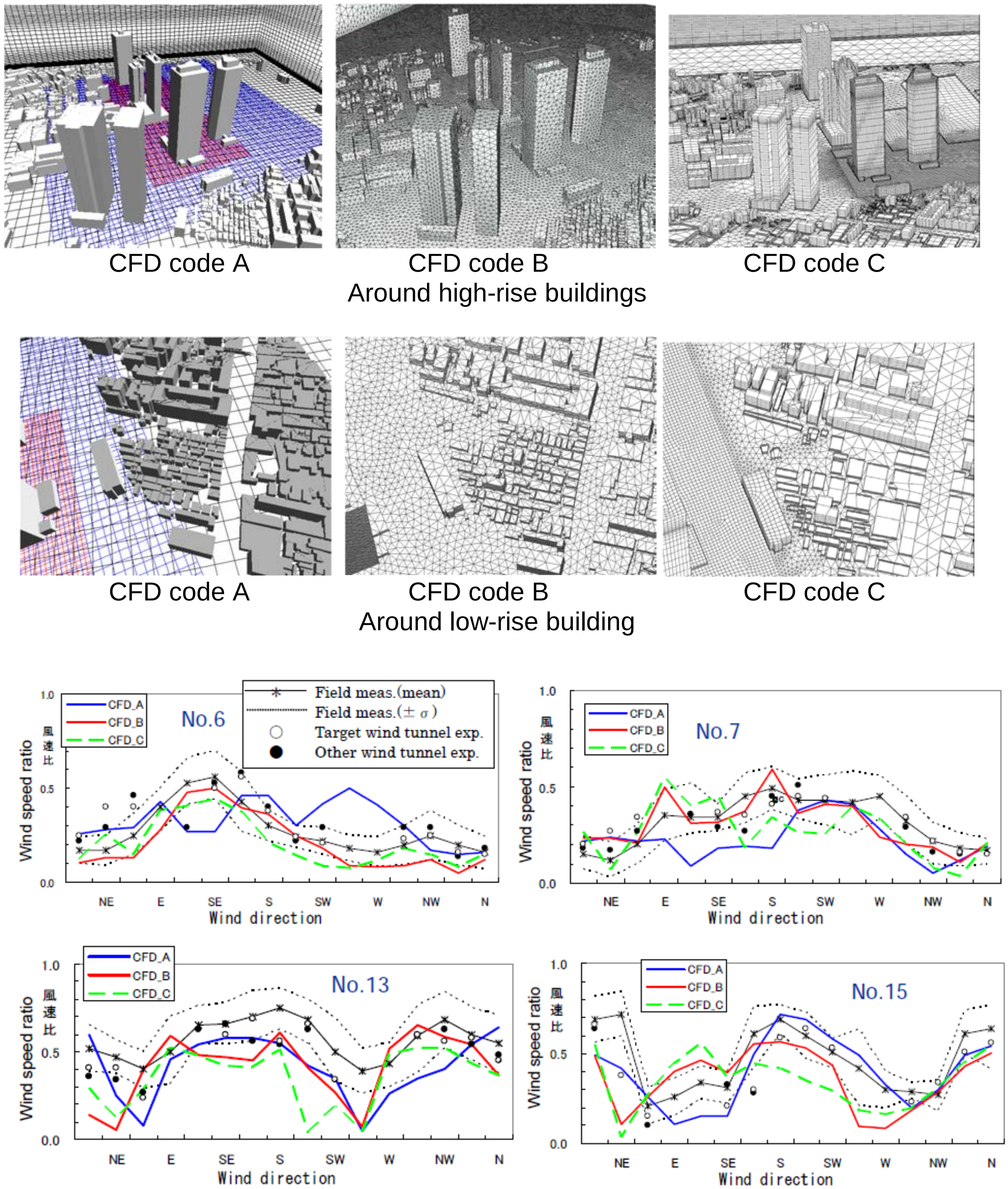


Figure 3.2.5.C: Comparison of wind speed ration for 16 wind directions Tominaga et. al (2005).

Accurate measurement of wind velocity, temperature in wind tunnel experiment: In wind tunnel experiment examining the target pollutant diffusion or heat island phenomena in a non-isothermal flow in urban areas influenced by solar and nocturnal radiation, it is very difficult to measure wind velocity and temperature and pollutant concentration simultaneously by using a Laser Doppler Velocimeter (LDV) (Yoshie et. la, 2007). The use of combination of an X probe and a cold wire is limited due to the X probe's limitation in distinguishing positive and negative state (reverse) of a considerably fluctuating flow. To overcome this, Yoshie et. la (2007) proposed a new technique, the measurement system is composed of a split film (S.F.) of DANTEC (55R55) and CTA Module (90C10), a cold wire (C.W.) (55P31) Temperature Module (90C20) for temperature measurement, and a high speed flame ionization detector (FID), and enable the following:

- Simultaneous measurement of instantaneous wind velocity, temperature, and concentration.
- Measurement of a flow field with a large turbulence intensity accompanied by positive and negative (reverse) flows.
- Appropriate temperature compensation for a flow with a large temperature fluctuation.

3.3 CFD Solver

3.3.1 Turbulence Modeling

As described by Versteeg and Malalasekera (2007) turbulence causes the appearance of eddies in a wide range of length and time that interact in a dynamically complex way. Three groups methods of numerical methods that can capture the important effect of turbulence. These are;

Turbulence models for Reynolds-average Navier- Stokes (RANS) equation: For this method attention is focused on the average or mean flow and the effects of turbulence on mean flow properties. The nature of the method is that extra terms appear in the time-averaged (eg. Reynolds-averaged) equations due to the interactions with various turbulence models. These extra terms are modeled with turbulence models and Reynolds stress models. The RANS turbulence modeling approach requires only modest computational resources and therefore this approach represents the overwhelming portion of engineering CFD applications.

Turbulence models for Large Eddy Simulations equation (LES): This method represents an intermediate form of turbulence calculations which considers large eddies. The methods uses space filtering procedures for the unsteady Navier-Stokes equations prior to calculations to reject smaller eddies and only use larger eddies. Since unsteady flow equations have to be solved the demand on computational resources is large. LES is staring to be accepted in the industry to address CFD problems involving complex geometries

Direct Numerical Simulations (DES): This method computes the mean flow as well as all turbulent velocity fluctuations. This method requires very significant computational resources and at the present DES is not used for industrial floe computations.

Reynolds-average Navier- Stokes (RANS) equation and classical turbulence models: In most applications it is not required to resolve the details of turbulent fluctuations. Instead for most CFD calculations it is sufficient to provide information about time-averaged properties of the flow, such as mean velocities, mean pressures. While the details of turbulent fluctuations is not calculated the effects of turbulence on the mea flow is needed. For this purpose turbulence models are applied to predict the Reynolds stresses and scalar transport terms and provide closure of the mean flow equations. The most common RANS turbulence models are classified on the basis of the number of additional transport equations that need to be resolved along with the RANS flow equations.

The following list the most common turbulence models. It is noteworthy to point out that there are generalized and specific turbulence models. As the terms suggest the more generalized turbulence models can be used to solve a broader range of flow problems as the more specific models. The following present several turbulence models that are in use in general CFD applications:

Mixing length model: Basically, the mixing length model is typically not used on its own in general purpose CFD, but is found embedded in more sophisticated turbulence models to describe near-wall flow behaviors. The advantages of the mixing length model sis the ease of use which requires little computational resources and good predictions of this shear layers. The disadvantages include that the model is incapable to describe flows with separation and recirculation.

The k-ε turbulence model: The k-ε model a more generic, yet sophisticated and effective, description of turbulence. The model allows for the effects of transport of turbulence properties by means of convection and diffusions and also for the production and destruction of turbulence. The model includes solving for the turbulent kinetic energy k and the rate of dissipation of turbulent kinetic energy ε. The k-ε model is the most widely used and validated turbulence. The model performs well in confined flows where the Reynolds shear stresses are most important, which includes many engineering flow problems. The model does not perform equally well in unconfined flows. Advantages of the k-ε model include that it is the simplest turbulence model for which only initial and/or boundary conditions are required. Other advantages include excellent performance in many industrial CFD applications and a wide application. Disadvantages include poor performance in some unconfined flows, flows with large strains (i.e. curved boundary layers) and other special flow applications.

Reynolds stress equation model (RSM): The RSM is the most complex classical turbulence and is an improvement of the k-ε model. RSM is termed the "most general of all classical turbulence models. There are indications the RSM model might be more widely used in the future after some of the difficulties in handling this models have been solved for a wide application. For example, the RSM model outperforms the k-ε model in terms of pressure distributions predictions in windward and leeward of flow obstructions. Disadvantages of RSM include very large computing costs and the fact that RSM is not as much validated as the k-ε model. In addition, and important to the application of air movement around buildings, RSM shares with the k-ε model a rather poor performance to describe unconfined recirculating flow.

In addition to the more classical turbulence models more advanced turbulence models have been developed, which promise good application potential for the simulation of wind movement around buildings. The development of more advanced turbulence models was triggered by the need to produce models with good performance while at the same time reduce the computational cost in comparison to high performance model such as the classical RSM.

<u>RNG k-ε model:</u> The model represents an expansion if the classical k-ε model. RNG stands for renormalization group. The RNK k-ε model represents the effects of small-scale turbulence by means of random forcing functions of the governing Navier-Stoke equations. Basicallt th RNG procedure removes small scales of motion from the governing equation.   The RNG model performs better foer certain flow applications than the classical k-ε model.

<u>The Wilcox k-ω model:</u> This model uses a turbulence frequency ω as an additional equation to describe a length scale determining variable. At the inlet k and ω have to specified. The model has limited use in industrial CFD applications.

<u>Menter SST k-ω model:</u> This turbulence model has advantages over the k-ε model in regrd to near-wall performance with adverse pressure gradients. The model is actually a hybrid model by first transforming the k-ε model into a k-ω model in the near-wall region and using the standard The k-ε model in the fully turbulent region. The model SST k-ω provide a general models with specific good performance of presser gradient boundary layers.

Various researchers have tested the validity of different turbulence models for applications describing wind flow phenomena. Some of the reviewed articles are discussed below .

Glover et al (2006) reports that the standard k-ε model exhibits problems predicting flow separation and underpredicting turbulent kinetic energy values within street canyons. The practicality of the standard k-ε model explains that the mode is still widely used in industry and research, thus creating a demand for improved performance from the model. The authors present a comprehensive study in which constants contained within the standard k- ε model were investigated on their validity to predict flow situations for which experimental data was available. The authors suggest that often industry leave the constant unchanged from the original values suggested by the developer of the k- ε model, Launder and Spalding.

The authors performed CFD calculations with varying constants and they were able to assess their influence on the CFD models capability to simulate flow within and above a street canyon. The authors found that there was a large spread of results shown in Figure 3.3.1.A which suggests these values have a significant impact on the turbulence values. Figure 3.3.1..A illustrates the wide spread of results identified by the authors. Furthermore the authors show that the constants can be adjusted to improve model predictions as seen by the use of modified parameters shown in Figure 3.3.1.B. The authors suggest that they were able to use statistical methods to emulate the CFD model, providing a much improved prediction of the turbulent kinetic energy values within the street canyon by integrating over the range of possible values for the parameters.

Franke et al (2007) recommend that simulations should always be started with the standard k-ε model, due to its very good stability. Depending on the application different models are then additionally tested:

- Realizable k-ε model for pedestrian wind environment
- Renormalization group k-ε model for pressures on buildings, see
- Reynolds stress model(s) (RSM) with and without wall damping for both applications

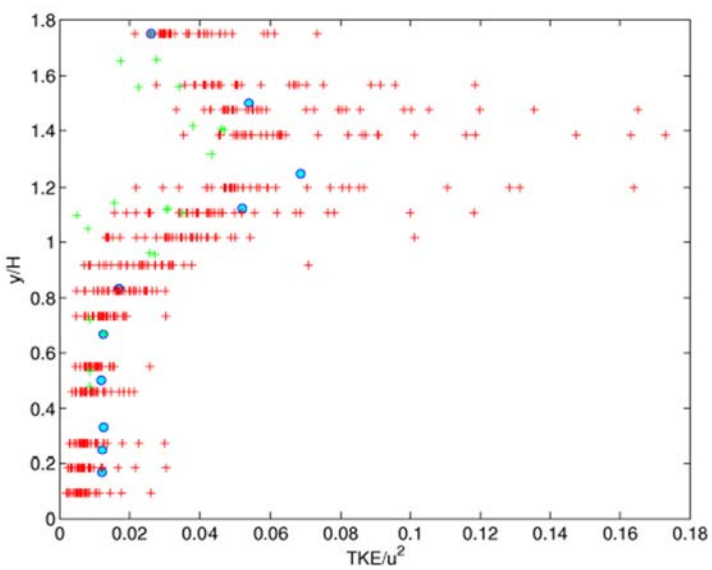


Figure 3.3.1.A: Normalized TKE (e.g. turbulence kinetic energy) data from wind tunnel (blue circles) and CFD simulation (crosses) against normalized by the height h of the buildings. (Glover et al, 2006)

Ramponi and Blocken (2012c) compared the results of CFD for the same building and wind conditions using different turbulence models. In their test they used 3D steady RANS simulations with the following turbulence models:

- Standard k-ε model (Sk-ε)
- Realizable k-ε model (Rk-ε)
- Renormalization Group k-ε model (RNG k-ε)
- Standard k-ω model (Sk-ω)
- Shear-stress transport k-ω model (SST k-ω)
- Reynolds Stress Model (RSM)

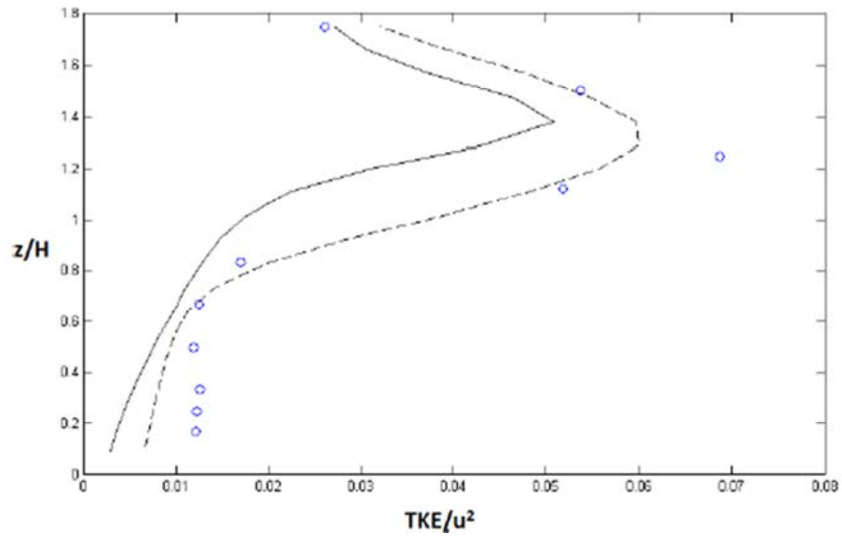


Figure 3.3.1.B: Normalized TKE profiles from CFD model with un-modified constants (black line), modified constants (dashed line) and wind tunnel data (circles)

The authors report that the effects of the turbulence models on the indoor air flow are illustrated in terms of streamwise wind speed ratio along the centerline of the openings (Figure 3.3.1.C). Measured ventilation flow rate (referred to as PIV in Figure 3.3.1.C) compared very well with the SST k- ω model (also referred to as the Ref. case) , followed by the RNG k-ε model, which also provides a fairly good performance. The discrepancies by the other models are very large. In fact, Fig. 3.3.1.C shows that the indoor streamwise wind speed obtained with the other k-e models is up to 6 times higher (at x/D = 0.75) than the one obtained using the SST k- ω model (Ref. case). The result suggest that RSM and the standard k-ε models tend to over predict the experimental values by up to 9 times (x/D = 0.6). The authors indicate that in several previous studies, authors pointed out the superior performance of the RNG k-ε model for indoor air flow modeling, especially compared to the standard k-model. In the cited study, however, the SST k- ω model clearly outperforms the RNG k-ε model.

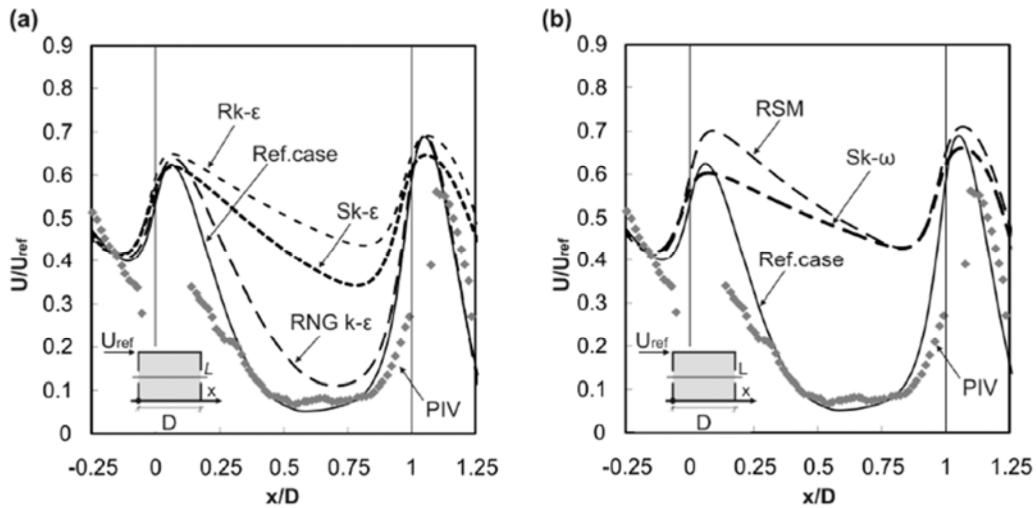


Figure 3.3.1.C: CFD simulation results for sensitivity analysis: impact of turbulence models on the streamwise wind speed ratio along the centerline. Comparison between the reference case (Ref.case = SST k- ω model) and (a) the k-ε models and (b) the RSM and standard k- ε model.

3.3.2 Steady Flow vs. Unsteady Flow

Steady flow is flow that has its properties (e.g. velocity, temperature, pressure, and density) are independent of time. That is,

$$\frac{\partial \Phi}{\Phi t} = 0$$

Where ⍰ represents a fluid property. The properties, however, may vary from point to point, which means that they could be a function of space (i.e., T = T(x, y, z), p = p(x, y, z) and ρ = ρ(x, y, z)). It should be noted, steady flow does not mean the velocity and accelerations are constant. Flow in a curved pipe or through a nozzle may be steady, but the velocity and/or acceleration is not constant.

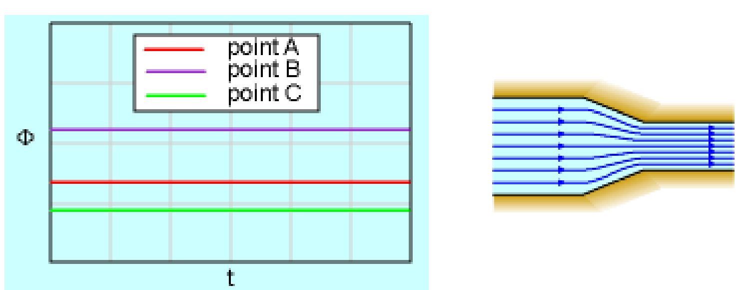


Figure 3.3.2.A: Steady flow: (a) Steady Flow (b) Flow with changing velocity

Steady flow is the simplification of a flow regime for certain types of analysis. However, most real-world flow problems are unsteady in nature. For unsteady flow, the fluid properties are time dependent (i.e., $T = T(x, y, z, t)$, $p = p(x, y, z, t)$ and $\rho = \rho(x, y, z, t)$). Unsteady flows can be further divided into periodic flow, non-periodic flow and random flow (Fig. a,b,c respectively) (Ngo and Gramoll, 1998)

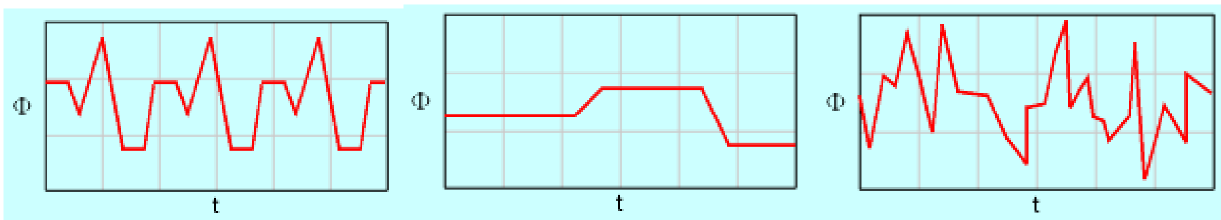


Figure 3.3.2.B: Time dependency of unsteady flow: (a) Periodic Flow, (b) Non-Periodic Flow and (c) Random Flow

<u>Suitable analysis for external wind flow in CFD</u>**:** Since working with turbulent atmospheric boundary layer flow, the principle is the choice of unsteady treatment for the approximate equations (Franken, 2007). There are three common averaging modelling approaches of the Navier-Stokes equations: Steady RANS, Unsteady RANS (URANS), Large Eddy Simulation (LES) and the hybrid RANS-LES (or Detached Eddy Simulation or DES) approaches.

- <u>Steady RANS:</u> this approach uses the time (infinite) average leading to a statistically steady description of the turbulent flow and therefore there are some limitations for applications that model the inherently unsteady meteorology. RANS is adequate representation of the wind tunnel's reality. RANS also can be applicable in studying certain turbulent flows but it requires closure by using one of several available turbulence models.
- <u>Unsteady URANS:</u> the basic equations of URANS are formally derived by applying ensemble averaging in which the resulting equations comply with the RANS equations now containing the partial time derivatives. URANS also allows for simulating temporal changes in the flow field caused, for example, by different surface temperatures. For an approach requiring a high spatial resolution, the LES or DES is recommended.
- <u>Large Eddy Simulation (LES) and Detached Eddy Simulation (DES):</u> LES provides an alternative approach in which large eddies are explicitly computed (resolved) in a time-dependent simulation using the "filtered'' Navier-Stokes equations for reducing the error resulted from turbulence modeling. The DES models, often referred to as the hybrid LES/RANS models combines RANS modeling with LES, in which RANS is used for near-wall region while LES is associated with the core turbulent regions where large unsteady turbulence space play a dominant role.

Advantages and disadvantages: According to Hertwig et. al (2012), Reynolds-averaged Navier Stokes equations (RANS models) allows fast computation of mean flow and dispersion even in very complex environments, especially applying studying urban ventilation, wind comfort assessment and pollutant dispersion predictions in cities. However, RANS models are only steady state solution for an inherent unsteadiness of wind phenomena in wind engineering. Particularly within the canopy region (i.e. up to roof level) this can lead to inaccurate predictions of mean flow fields-especially for situations in which unsteady effect like flow separation, impinging, reattachment or vortex shedding play a predominant role in the dynamics of the flow.

For example, Montazeri and Blocken (2013) used 3D steady RANS simulation in their studying of the pressure coefficients of a building with and without balcony and compared with five turbulence models: the standard k-ε (Sk-ε), the realizable k-ε (Rk-ε), the renormalization group k-ε (RNG k-ε), the standard k-ω (Sk-ω) and the Reynolds Stress Model (RSM). The result unveiled that the RNG k-ε model tends to overestimate the pressure variations near the ground level while the standard k- ω generally provides a slight overestimation. According to author the k-ω, Rk-ε and RSM models depict some agreement.

Using more detailed models such as models with or without balconies, Montazeri and Blocken (2013) found that the presence of balconies into building causes flow separation, recirculation and detachment on the building façade and therefore has impacted dramatically on the calculation of the pressure coefficients on the windward building façade. Also, the limitation of the steady RANS CFD has been found that it only can be reliable when predicting cases of perpendicular approach flow wind direction. In case of oblique flow wind direction, CFD analysis using RANS depicted large discrepancies against validated from wind-tunnel measurement.

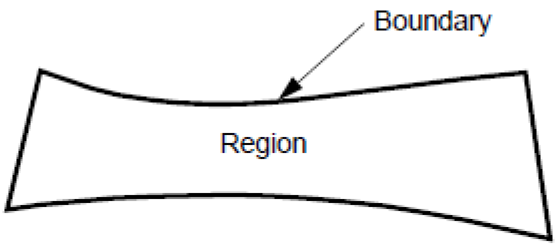
3.3.3 Boundary Conditions for External Flow

The CD-Adapco user guide documentation describes the concept of boundary conditions specifically on the external flow. The documentation illustrates basic principle and definition of boundary conditions, inlet boundaries, outlet boundaries, no-slip wall, and free stream which are the significant elements for the CFD external flow simulation. Boundaries are surfaces (or lines in a two-dimensional case) that completely surround and define a region. Each boundary has a corresponding node in the simulation tree, and has its own properties and pop-up menu. The figure below shows a simple region where the lines surrounding the region represent the boundaries. In this example, the region could be surrounded by a single boundary or multiple boundaries. The choice depends on what conditions and values need be assigned.

The mathematical model of the CFD simulation requires specifying conditions on the solution domain boundaries. Conditions related to start time are called Initial Conditions. Conditions related to space are called Boundary Conditions.

<u>Boundary types and definition for the external flow:</u> Pressure boundary conditions can be specified at boundaries where the pressure distribution is known.

Boundary velocities are obtained from Neumann Boundary condition for velocity. All dependent variables are either specified or extrapolated from the inside using zero gradient assumption. At the outflow, all variables are extrapolated. The velocity at the pressure boundary where the flow comes in has to be sub-sonic, or else the upstream velocity needs to be specified, thus violating the Neumann boundary condition.



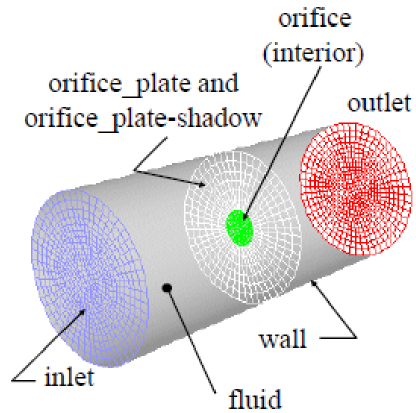
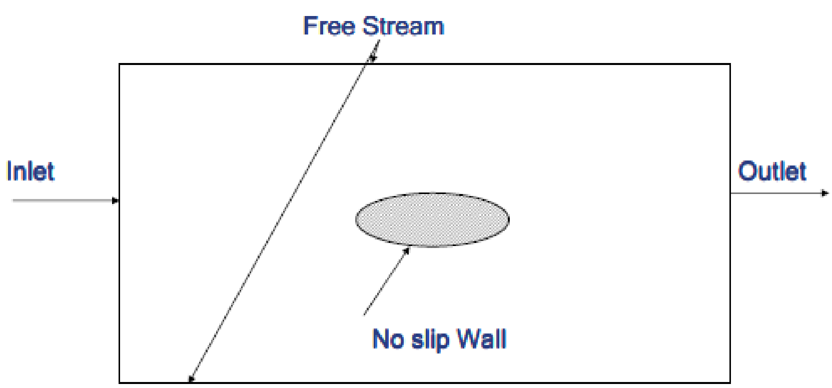


Figure 3.3.3A: Concept of boundaries (CD-adapco user guide 8.02)
Figure 3.3.3B: Concept of boundaries (Bakker, 2006)



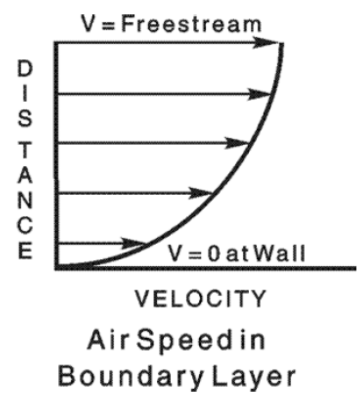


Figure 3.3.3B (Left): Concept and composition of boundaries (CD-adapco training document)
Figure 3.3.3C (Right): Free Stream in air speed in boundary layer(CD-adapco training document)

Inlet Boundaries can be specified at portions of boundary where the fluid enters the solution domain, and where the velocity and scalar (temperature, species concentration, turbulence quantities) distributions is known. Mass Flow Inlet is defined when a mass flow inlet boundary represents an inlet for which the mass flow rate is known. The stagnation inlet boundary is an inlet condition for compressible flows, although it is equally valid for incompressible flows. The stagnation conditions refer to the conditions in an imaginary plenum, far upstream, in which the flow is completely at rest. For incompressible flows, Bernoulli's equation is used to relate total pressure, static pressure, and velocity magnitude. For compressible ideal-gas flows, isentropic relations are used, and characteristic variables determine the propagation properties of the flow. A velocity inlet boundary represents the inlet of a duct at which the flow velocity is known.

Outlet boundaries can be specified at that portion of the solution domain, where flow leaves the domain. It assumes zero gradients of all dependent variables in the flow direction. Flow Split Outlet refers to an outlet boundary represented by a duct. When multiple flow split outlets bound a fluid continuum, you specify the fraction of the total mass flow through each of the outlet boundaries. The sum of all flow split fractions from all of the flow split outlet boundaries in a simulation must equal 1.0. When only one flow split boundary bounds the continuum, it is assumed that all the flow exits through this boundary. This boundary condition cannot be used with compressible flows or for recirculation inflow. A pressure outlet boundary is a flow outlet boundary at which the pressure is specified, such as the boundary surrounding a jet emanating into a large chamber.

No-Slip Wall / Symmetry plane requires prescription of velocity at the wall (e.g. zero velocity for a stationary wall). (see Figure 3.3.3.D). A symmetry plane boundary represents an imaginary plane of symmetry in the simulation. The solution is obtained with a symmetry plane boundary is identical to the solution that would be obtained by mirroring the mesh about the symmetry plane (in half the resulting domain).

A wall boundary represents an impermeable surface.

The free stream boundary represents the conditions at a far-field location. This condition generally applies to external flows when the boundary is placed sufficiently far from the body. Internal flows often have walls, possibly of irregular shape, immediately next to the flow boundary. These walls can produce boundary layers, vortices, or other multi-dimensional flow structures such that the irrotational, quasi-1D flow assumption breaks down.

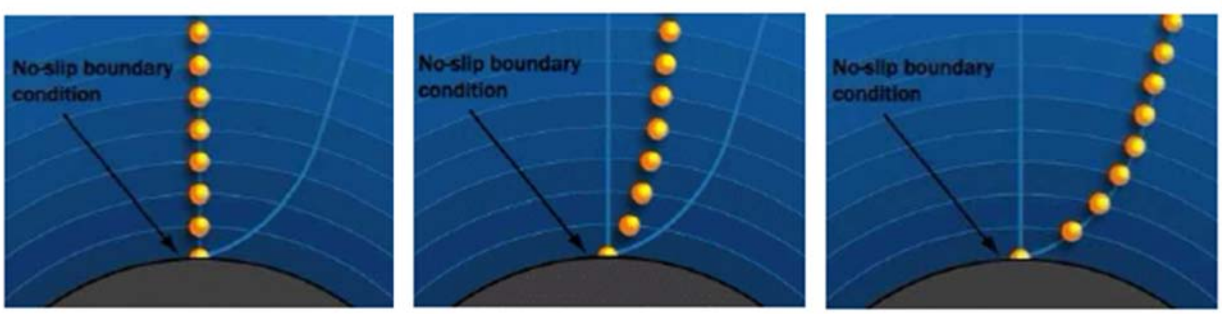


Figure 3.3.3D: No-slip wall (Bakker, 2006)

3.3.4 Grid Convergence

Bakker (2006)'s reports on the concept of grid convergence and iterative convergence, errors and residuals. The author provides a general concept and principle of iteration process, residuals, significant of error, and ideal of convergence gird approach.

General approach – convergence: The iterative process is repeated until the change in the variable from one iteration to the next becomes so small that the solution can be considered converged.

- At convergence:
  - All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance.
  - The solution no longer changes with additional iterations.
  - Mass, momentum, energy and scalar balances are obtained.
- Residuals measure imbalance (or error) in conservation equations.
- The absolute residual at point P is defined as:

$$R_P = \left| a_P \phi_P - \sum_{nb} a_{nb} \phi_{nb} - b \right|$$

- Residuals are usually scaled relative to the local value of the property f in order to obtain a relative error:

$$R_{P,scaled} = \frac{\left| a_P \phi_P - \sum_{nb} a_{nb} \phi_{nb} - b \right|}{\left| a_P \phi_P \right|}$$

- Residuals can also be normalized, by dividing them by the maximum residual that was found at any time during the iterative process.
- An overall measure of the residual in the domain is:

$$R^{\phi} = \frac{\sum_{all\ cells} \left| a_P \phi_P - \sum_{nb} a_{nb} \phi_{nb} - b \right|}{\sum_{all\ cells} \left| a_P \phi_P \right|}$$

- It is common to require the scaled residuals to be on the order of 1E-3 to 1E-4 or less for convergence.
- Proper convergence criteria should be defined before using a solution: solutions that do not converge can be misleading.
- Solutions are converged when the flow field and scalar fields are no longer changing.
- Determining when this is the case can be difficult.
- It is most common to monitor the residuals.

Monitor residuals: The author provides the following guidelines for monitoring residuals (see Figure 3.3.4. A for illustration of residual monitoring):

- If the residuals have met the specified convergence criterion but are still decreasing the solution may not yet be converged. In this case the convergence criteria should be revisited.
- If the residuals never meet the convergence criterion, but are no longer decreasing and other solution monitors do not change either, the solution is converged.
- Residuals should not infer the solution! Low residuals do not automatically mean a correct solution, and high residuals do not automatically mean a wrong solution
- Final residuals are often higher with higher order discretization schemes than with first order discretization. This fact cannot be interpreted that the first order solution produces more accurate results.

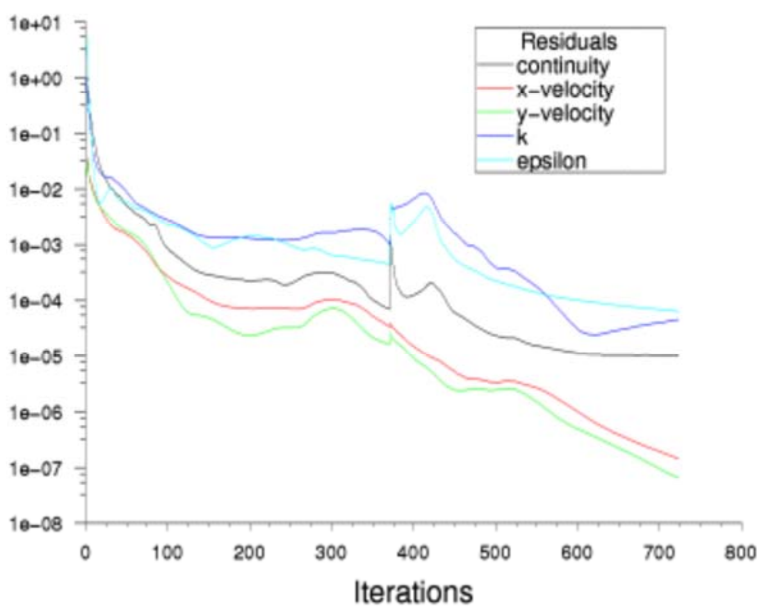


Figure 3.3.4A: Residuals monitoring (Bakker, 2006)

Other convergence monitors:

- For models whose purpose is to calculate a force on an object, the predicted force itself should be monitored for convergence. An example might be wind force on an object where one should monitor the predicted drag coefficient. Figure 3.3.4. illustrates the convergence of a predicted drag coefficient.
- Overall mass balance should be satisfied.
- When modeling rotating equipment such as turbofans or mixing impellers, the predicted torque should be monitored.
- For heat transfer problems, the temperature at important locations should be monitored.

- Flow field plots can automatically be generated at predefined iterations (example after 50 iterations) to visually review the flow field and ensure that it is no longer changing.

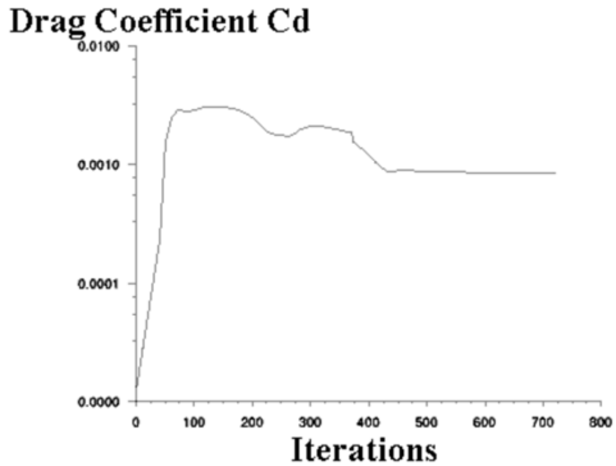


Figure 3.3.4B: Drag coefficient (Bakker, 2006)

Accuracy of numerical schemes: The author suggests methods to ascertain accuracy of the numerical solution (see Figure 3.3.4.C as an example):

- The first order upwind scheme only uses the constant and ignores the first derivative and consecutive terms. This scheme is therefore considered first order accurate.
- For high Peclet numbers the power law scheme reduces to the first order upwind scheme, so it is also considered first order accurate.
- The central differencing scheme and second order upwind scheme do include the first order derivative, but ignore the second order derivative. These schemes are therefore considered second order accurate. Quick does take the second order derivative into account, but ignores the third order derivative. This is then considered third order accurate.

Accuracy and false diffusion:

- False diffusion is numerically introduced diffusion and arises in convection dominated flows, for instance in high Peclet number flows.
- False diffusion occurs due to oblique flow direction and non-zero gradient of temperature in the direction normal to the flow.
- Grid refinement coupled with a higher-order interpolation scheme will minimize the false diffusion.

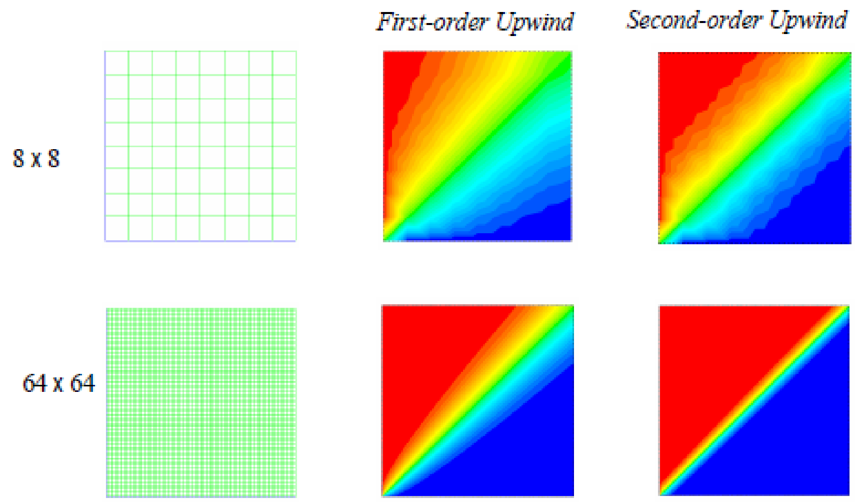


Figure 3.3.4C: Grid refinement coupled (Bakker, 2006)

Multi-grid solver:

- The algebraic equation can be solved by sweeping through the domain cell-by-cell in an iterative manner.
- This method reduces local errors quickly but can be slow in reducing long-wavelength errors.
- On large grids, it can take a long time to see the effect of distant grid points and boundaries.
- Multigrid acceleration is a method to speed up convergence for:
  - Large number of cells.
  - Large cell aspect ratios (e.g. Dx/Dy > 20).
  - Large differences in thermal conductivity such as in conjugate heat transfer problem.
- The multigrid solver uses a sequence of grids going from fine to coarse.
- The influence of boundaries and far-away points is more easily transmitted to the interior on coarse meshes than on fine meshes.

Fine meshes:

- Fine meshes give more accurate solutions.
- The solution on the coarser meshes is used as a starting point for solutions on the finer meshes.
- The coarse-mesh solution contains the influence of boundaries and far neighbors. These effects are felt more easily on coarse meshes.
- This accelerates convergence on the fine mesh.
- The final solution is obtained for the original (fine) mesh.
- Coarse mesh calculations only accelerate convergence and do not change the final answer.

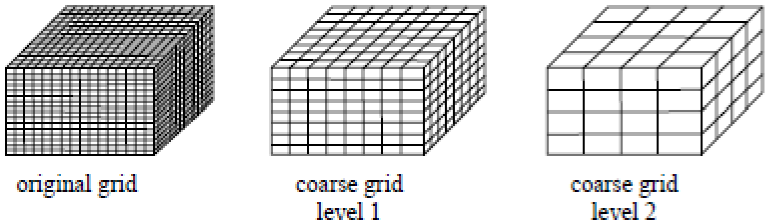


Figure 3.3.4D: Fine meshes (Bakker, 2006)

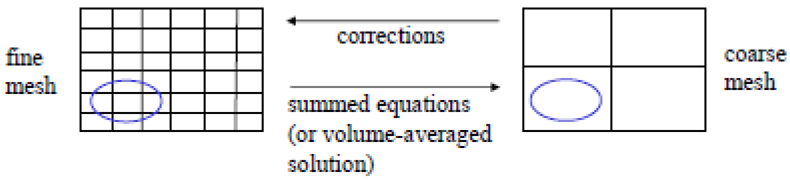


Figure 3.3.4E: Coarse mesh calculations (Bakker, 2006)

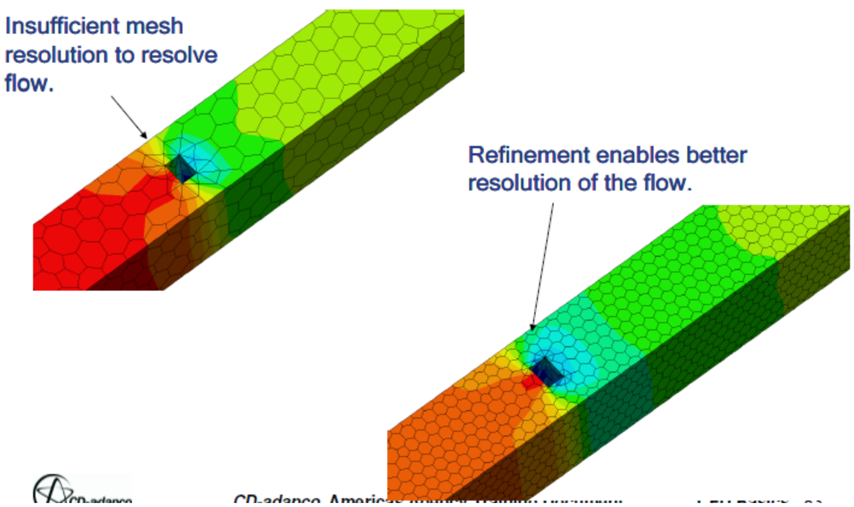
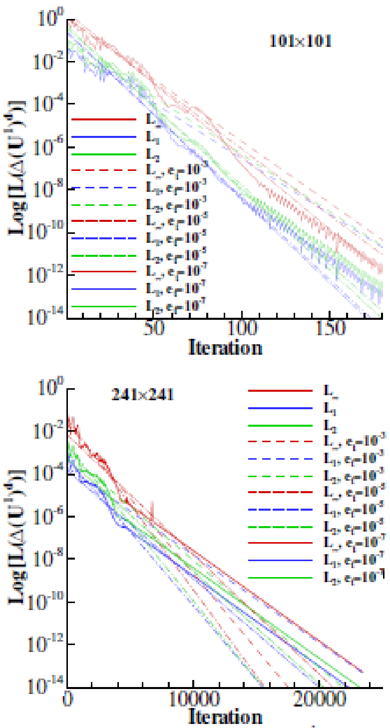


Figure 3.3.4F: Grid Generation Guidelines (CD-adapco, user manul)

Iterative error on the numerical uncertainty Eca (2006) describes the influence of the iterative error on the numerical uncertainty of the solution of the Reynolds-Averaged Navier-Stokes (RANS) equations. Two main topics are addressed: the estimation of the iterative error; and the influence of the iterative error on the estimation of the discretization error. Iterative error estimators based on the L_∞, L1 and L2 norms of the differences between iterations and on the normalized residuals are tested on three test cases: the 2-D turbulent flow over a hill, a 3-D flow over a finite plate and the flow around the *KVLCC2M tanker* at model scale Reynolds number. Two types of procedures are considered, one using the data of the last iteration performed and the other using an extrapolation based on a least squares fit to a geometric progression. In the latter case, the option of including the standard deviation of the fit in calculation of the error estimator is also tested. The results show that the most reliable estimates of the iterative error are obtained with the extrapolation technique including the effect of the standard deviation of the fit applied to the L∞, norm of the differences between successive solutions. To obtain realistic estimates of the iterative error the author suggests the use of the L2 and L1 norms and the use of the differences obtained in the last iteration. Figure 3.3.34.G illustrates an example of iterative error estimation.



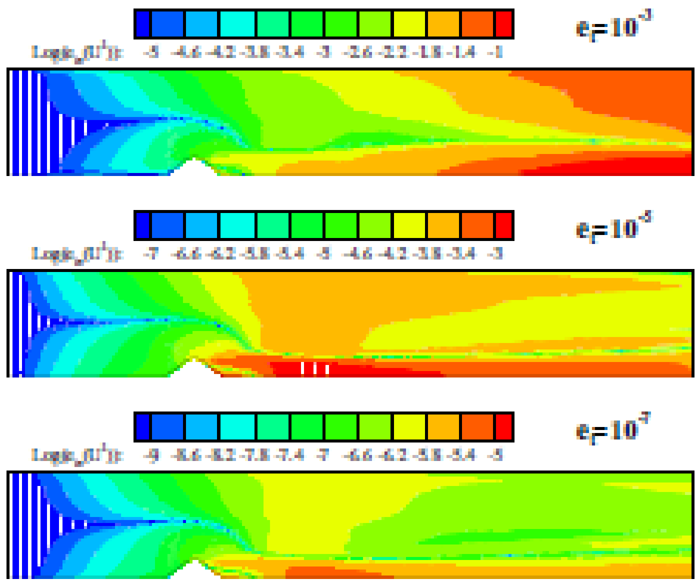


Figure 3.3.4G (Left): Convergence history of U1 for the flow around a two-dimensional hill. 101x101 and 241x241 grids. Rn = 60000, (Eca, 2006)

Figure 3.3.4H (Right): Estimated iterative error of U1, eie, for different levels of the tolerance criteria, et. Flow over a two-dimensional hill. 241_241 grid. Rn = 60000, (Eca, 2006)

Eca (2006) delineates the influence of the iterative error on the estimation of the numerical uncertainty of CFD predictions. Two types of problems have been investigated, the estimation of the iterative error and the influence of the iterative error on the estimated discretization error. The author suggests several variants for the estimation of the iterative error such as the differences between results of consecutive iterations or of the normalized residual. The simplest approach assumes that the iterative error is equal to the changes in the solution in the last iteration performed. The alternative procedure uses an extrapolation technique to an infinite number of iterations based on a geometric progression.

3.3.5 Numerical Stability and Selection of Time Steps

The process of solving a multiphase system is inherently difficult and can lead to some stability or convergence problems. This section references the 2009 ANSYS's Manual (ANSYS, 2009):

When solving a time-dependent problem, a proper initial field is required to avoid instabilities, which usually arise from poorly initiated fields. If the computational resources (e.g. CPU time) is a concern for transient problems, then the best option is to use Phase Coupled SIMPLE (which is available in most commercial CFD software products). When body forces are significant, or if the solution requires higher order numerical schemes, it is recommended to start with a small time step, which can be increased after performing a few time steps to get a better approximation of the pressure field. For a steady solution, it is recommended that the Multiphase Coupled solver is used. The iterative nature of this solver requires a good selection of the starting field.

If difficulties are encountered due to higher order schemes, or due to the complexities of the problem, it is advised to reduce the Courant number. A typical default Courant number is 200 but this number can be reduced to as low as 4 in order to enhance stability. The Courant number can later be increased if the iteration process runs smoothly. In addition, there are explicit under-relaxation factors for velocities and pressure. All other under-relaxation factors are implicit. Lower under-relaxation factors for the volume of fraction equation may delay the solution dramatically with the Coupled solver (any value 0.5 or above is adequate); on the contrary, Phase Coupled SIMPLE would normally need a low under-relaxation for the volume fraction equation.

In addition, ANSYS (2009) suggests that the software ANSYS FLUENT offers a Full Multiphase Coupled solver where all velocities, pressure correction and volume fraction correction are solved simultaneously. Furthermore, ANSYS FLUENT has an option to solve stratified immiscible fluids within the Eulerian multiphase formulation. This feature is similar to the single fluid VOF solution, but in the context of multiple velocities.

Initial conditions: Initial conditions in the continua specify the initial field data for the simulation. ANSYS's Manual (ANSYS, 2009) provide guidance which can be applied for other CFD codes as well. For steady-state simulations, the converged solution ought to be independent of the initial field. However, the path to convergence, and hence the computational effort that is required to reach convergence, is affected. Therefore, selection of the initial conditions and values carefully is important to achieve good solutions, particularly when the physics is complex.

Each model requires sufficient information for the primary variables that are associated with the model. In most cases, this is done directly but for some models, such as turbulence, the user may be offered the option of specifying the information in a more convenient form (for example, turbulence intensity and turbulent viscosity ratio instead of turbulent kinetic energy and turbulent dissipation rate).

Examples of initial conditions are:

- Pressure
- Temperature
- Velocity components
- Turbulence quantities

Stopping criteria: The cited stop criteria are extracted and summarized from the STAR-CCM+ Manual (CD-Adapco, 2013). The stop criteria can also be applied for other CFD applications as well. Stopping criteria specify how long the solution should run and under what conditions it should stop iterating and/or marching in time. Each enabled stopping criterion is evaluated at the completion of every simulation step and a logical rule is used to determine if the interaction of all the criteria should stop the solver. There are two methods for applying stopping criteria:

Automatically generated stopping criteria: For steady simulation an automatically generated stopping criteria is used (maximum steps, stop file). For unsteady simulations a maximum inner iterations is used (maximum physical time, maximum steps, stop file)

- Maximum steps stopping criterion allows for specifying the maximum number of interactions in a steady solver or the maximum number of time-steps in an unsteady solver.
- The stop file criterion allows for specifying the pathname of a file that, one in place, will cause the solver to stop. This can be useful for stopping a batch job.
- The maximum physical time stopping criterion is based on the simulation time that has elapsed in a transient analysis.
- The maximum inner iterations stopping criterion is based on the number of inner interactions executed by the solver for transient analyses.

Manually generated stopping criteria:

- Residuals do not always reflect changes after the first inner iteration in a transient analysis. The minimum inner iterations stopping criterion prevents a time-step from stopping until the solver finishes the specified number of inner iterations.
- Using monitor-based stopping criteria to judge convergence at specific nodes based upon some user-specific criteria.

Choice of time step size: To solve time-dependent solution in unsteady simulation, the time step size is important parameter for the accuracy of the result. Several methods for choosing the proper time step size was summarized by Franke et al. (2007): The first method is to estimate the time step size based on the highest frequency. The second method is to estimate the dominant advection based on the maximum wind velocity and the minimum grid spacing. Franke et al. (2007) suggest that a sensitivity assessment of the influence of time step on the accuracy of the result should be performed by using reducing or increasing the time step size.

Criteria for convergence: The calculation needs to be finished after sufficient convergence of the solution is achieved. For this purpose, it is important to confirm that the solution does not change by monitoring the residual variables on specified points or by overlapping the contours among calculation results at different calculation steps. According to Tominaga et. al (2008), the default values for convergence in most commercial codes are not strict because code vendors want to stress calculation efficiency. Therefore, stricter convergence criteria are required to check that there is no change in the solution. When the calculation diverges or convergence is slow, the points below should be examined:

- The aspect ratio and the stretching ratio of the grids may be too large.
- The relaxation coefficient of the matrix solver may be too small.
- Periodic fluctuations such as a vortex shedding may be occurring.

Judging convergence: Judging convergence of a solution can be done by monitoring the quantities of plots representing engineering quanties or physical properties (e.g. integrated forces, pressure changes, mass flow rates, residuals). While it is true that the residual quantity will tend toward a very small number when the solution is converged, the residual monitors cannot be relied on as the only measure of convergence (CD-adapco, 2013).

According to CD-adapco (2013), the choice of the engineering quantity, as well as the convergence criterion, is at the discretion of the user. Figure 3.3.5.A shows monitors of both lift and drag coefficients as well as the residuals from a large external aerodynamics solution. It is evident that not all quantities reach an asymptotic limit at the same time and judgment has to be used to determine which coefficient is the most critical in concluding convergence.

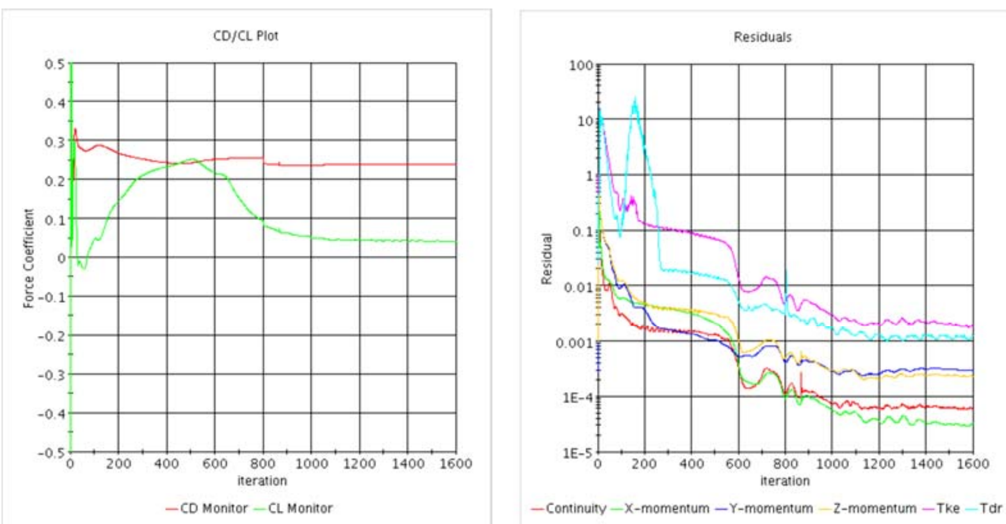


Figure 3.3.5.A: Monitor-based stopping criteria can be used to stop the simulation when an appropriate level of convergence has been reached, based on an appropriate combination of residual monitors or monitors of other physical quantities (CD-adapco, 2013).

3.3.6 Sensitivity Analysis – Guidelines and Evolving Standards

Results of CFD studies are dependent on a significant number of numerical parameters that comprise the input for the calculations. Sensitivity analyses are recommended to ascertain that suitable attention has been made by selecting the input parameters. There are contradictions among CFD predictions of cross-ventilation through prismatic buildings. Some reproduce experimentally observed inlet jet trajectories and others do not. Meroney (2009) states that it would be desirable to determine prediction sensitivity to approach flow conditions, turbulence levels, building porosity.

CFD calculations rely on sophisticated models that are used to define the numerical solutions. Jakeman et al (2006) describes some underlying issues related to identifying and mitigating uncertainties. He suggests that identification of model structure and parameters . The underlying aim is to balance sensitivity to system variables against complexity of representation. Uncertainty must be considered in developing any model, but is particularly important, and usually difficult to deal with, in large, integrated models. Uncertainty in models stems from incomplete system understanding , from imprecise, finite and often sparse data and measurements; and from uncertainty in the baseline inputs and conditions for model runs, including predicted inputs.

Jakeman et al (2006) indicates that practical convenience often dictates piecemeal identification of model components, and pre-existing models are often available for parts of the system. The authors suggests that it is wise to test the overall model to see whether simplification is possible for the purposes in mind. Sensitivity assessment plays a large role here. The results from extensive sensitivity testing can be difficult to interpret, because of the number and complexity of cause-effect relations tested. To minimize the difficulty, clear priorities are needed for which features of which variables to examine, and which uncertainties to cover. A good deal of trial and error may be required to fix these priorities.

According to Ramponi and Blocken (2012) the need for accuracy and reliability need detailed sensitivity studies. CFD simulation results can be very sensitive to the large number of computational parameters that have to be set by the user. Therefore, detailed and generic sensitivity analyses are important to provide guidance for the execution and evaluation of future CFD studies. The authors have carried out a comprehensive and exemplary sensitivity analysis , which was based on a reference case. The reference case was found to correlate well with previous results obtained in a wind tunnel. The systematic and detailed sensitivity analysis was conducted by systematically varying a single parameter compared to the reference case and evaluating the impact of this change on the simulation results. The parameters tested are the size of the computational domain, the resolution of the computational grid, the inlet turbulent kinetic energy of the atmospheric boundary layer, the turbulence model, the discretization schemes, and the convergence criteria. Table 3.3.6.A provides an overview of the computational parameters for the sensitivity analysis, with indication of the reference case.

Table3.3.6.A. Overview of computational parameters for sensitivity analysis with indication of the reference case (Ramponi and Blocken, 2012)

	Computational domain size (Section 4.1)	Computational grid resolution (Section 4.2)	Turbulent kinetic energy (Section 4.3)	Turbulence models (Section 4.4)	Discretization schemes (Section 4.5)	Level of iterative convergence (Section 4.6)
Ref. case	**$H_D = 5xH$**	**575,247**	**a = 1**	**SST k-ω**	**2^{nd} order**	**Conv.**
	$H_D = 4xH$	314,080	a = 0.5	Sk-ε	1^{st} order	10^{-4}
	$H_D = 3xH$	144,696	a = 1.5	Rk-ε		10^{-3}
	$H_D = 2xH$			RNG k-ε		10^{-2}
	$H_D = 1xH$			Sk-ω		
				RSM		

<u>Impact of size of computational domain</u>

For the assessment of the impact of computational gird only the cross-section (width and height) of the computational domain was varied. The upstream and downstream length of the domain remained unchanged at 3H and 15H, respectively, with H the building height. Figure 3.3.6.B indicates the variable parameters which included the height HD of the domain is HD = H + d and the while WD of the domain as WD = W + 2d, with W the width of the building. Several researches tend to refer to the blockage ratio (BR) which is the ratio between the frontal area of the model facade and the cross-section of the computational domain. Researchers suggest different yet similar minimum vales for the lateral and top clearance that describes the domain cross-section. For example Tominaga et al (2008) and Frank (2006) suggest a minimum distance (d) equal to 5 times the height (H) to avoid the interference of the domain size on the numerical simulation results. Thus, a distance d = 5H (B.R. = 2%) was applied in the reference case and then reduced to 4H (B.R. = 3%), 3H (B.R. = 4%), 2H (B.R. = 8%) and H (B.R. = 19%). Figure 3.3.6.B. indicates the influence of the cross-section of the computational domain on the indoor air speed. The result in Figure 3.3.6.B suggest that a considerable increase in indoor air speed occurs when the cross-section is strongly reduced (d=H).

<u>Impact of computational grid resolution</u>

The sensitivity of grid resolution was tested with three girds of different cell numbers. The grids were obtained by coarsening the reference (fine) grid with twice about a factor 2. The three grids are illustrated in Figure 3.3.6.C and the results on the three grids are shown in Figure 3.3.6.D The results show that grid sensitivity is most pronounced for the indoor area behind the inlet and behind the outlet. The sensitivity analysis shows that the reference (fine) grid is a suitable grid. The difference in the ventilation flow rates through the inlet openings is about 1.0% between the fine grid and medium grid, while the difference is 7.5% between fine grid and the coarse (fine) grid and grid A.

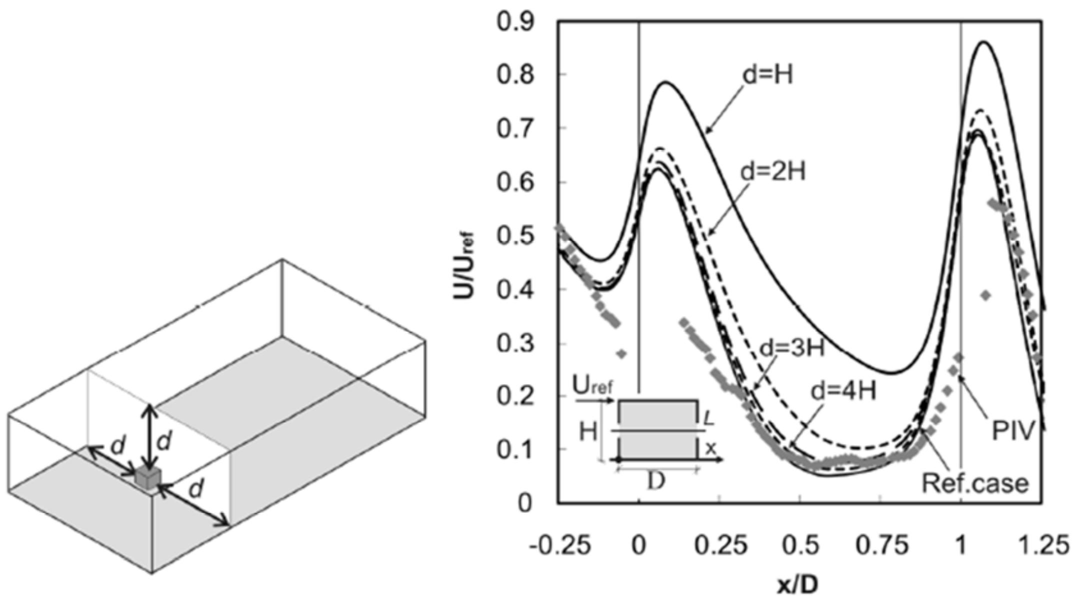


Figure 3.3.6.B: CFD simulation results for sensitivity analysis: impact of the size (cross-section) of the computational domain on the streamwise wind speed ratio along the centerline (Ramponi and Blocken, 2012).

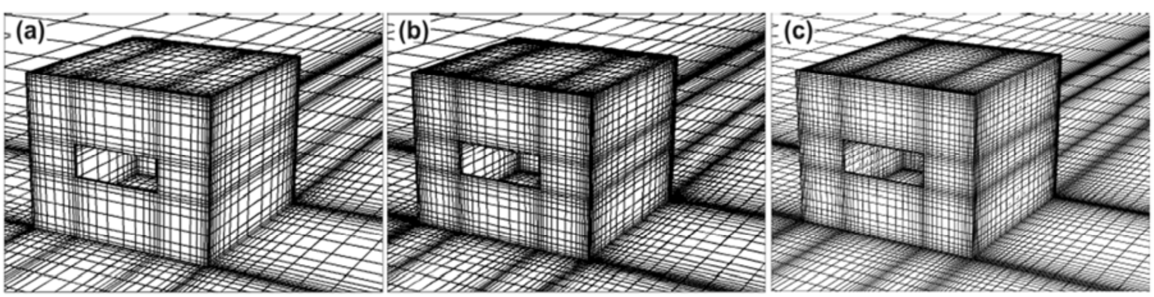


Figure 3.3.6.C: Perspective view of grids for grid-sensitivity analysis: (a) Coarse grid A with 144,696 cells; (b) Middle grid B with 314,080 cells; (c) Fine grid C with 575,247 cells (reference case). (Ramponi and Blocken, 2012).

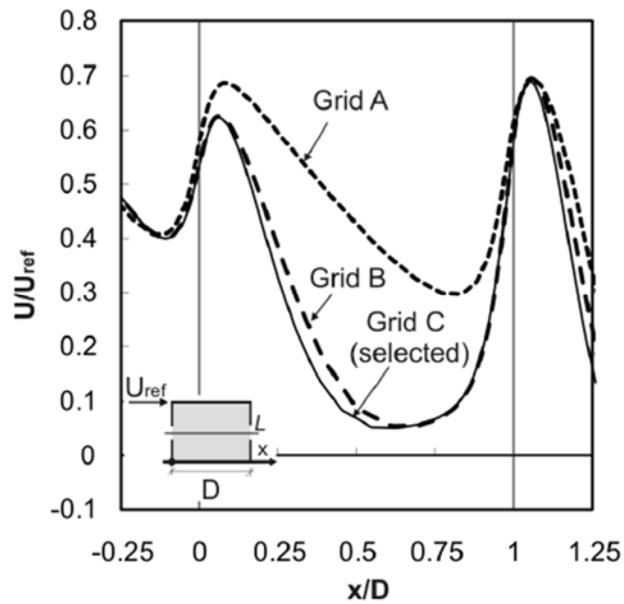


Figure 3.3.6.D: CFD simulation results for sensitivity analysis: impact of the grid resolution on the streamwise wind speed ratio along the centerline; Comparison of results from the three grids. (Ramponi and Blocken, 2012).

Impact of inlet turbulent kinetic energy

Impact of inlet turbulent kinetic energy profile at the inlet can be estimated from the measured wind velocity and turbulence intensity profiles. In the sensitivity study three values for "a", where a is linar coefficient that is used to calculate the turbulent kinetic energy. Different profiles of k and were defined by varying the parameter a. Results of two values of "a" were compared with the reference case (a = 1) for streamwise wind speed ratio along the centerline of the openings (Figure 3.3.6.E).The functions shown in Figure 3.3.6.F shows that varying the inlet turbulent kinetic energy with the parameter "a" has a very large impact on the wind speed ratio along the centerline. Other results indicae that the increased turbulent kinetic energy profiles at the inlet are not significantly affecting the horizontal homogeneity of the approaching flow. Only some small streamwise changes are noted for the turbulent kinetic energy profiles themselves.

Impact of order of discretization scheme

The authors suggest that CFD best practice guidelines consistently stress the importance of at least discretization schemes of second-order accuracy. The authors stress that first-order schemes provide necessary numerical diffusion to achieve convergence with unstructured computational grids that include tetrahedral and/or pyramid cells, rather than hexahedral and prismatic cells. However, the numerical diffusion by first-order schemes also decreases flow gradients, as shown in Figure 3.3.6.F , which indeed shows that for this situation accurate results cannot be obtained with first-order discretization schemes

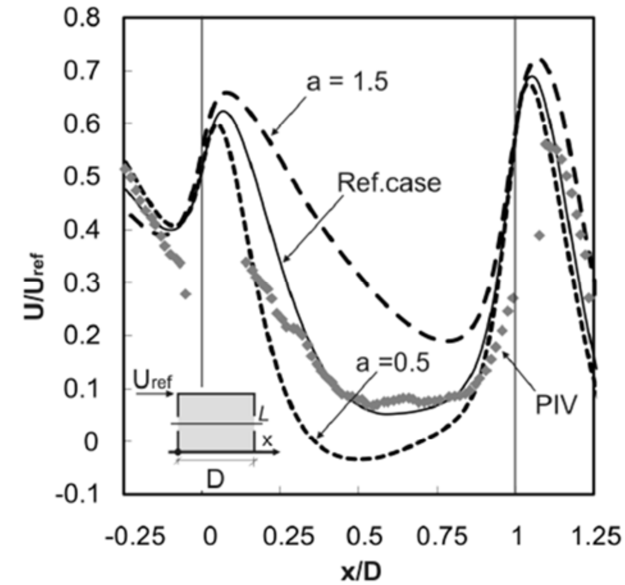


Figure 3.3.6.E: CFD simulation results for sensitivity analysis: impact of approach-flow turbulent kinetic energy profile parameter a on the streamwise wind speed ratio along the centerline (a = 1 for the reference case).. (Ramponi and Blocken, 2012).

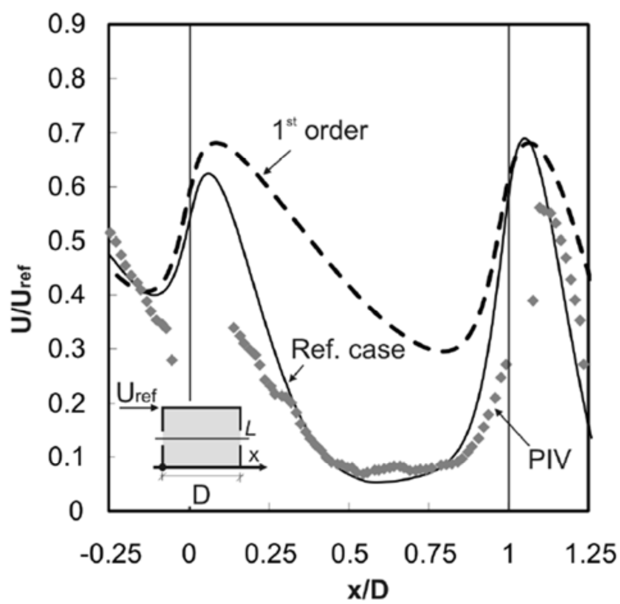


Figure 3.3.6.F: CFD simulation results for sensitivity analysis: impact of discretization scheme on the streamwise wind speed ratio along the centerline. For the reference case, a second-order discretization scheme is used. (Ramponi and Blocken, 2012).

Impact of turbulence model

The sensitivity analysis used a 3D steady RANS simulation with the following turbulence models:

- standard k-ε model (Sk-e)
- realizable k-ε model (Rk-e)
- Renormalization Group k-ε model (RNG k-e)
- standard k-ω model (Sk-w)
- shear-stress transport k-ω model (SST k-w)
- Reynolds Stress Model (RSM)

Figure 3.3.6.G shows the effects of the turbulence models on the indoor air flow are illustrated in terms of streamwise wind speed ratio along the centerline of the openings. Figure 3.3.6.H shows the contours of streamwise wind speed in the vertical centerplane. Figure 3.3.6.G indicates a comparison with the PIV measurements. The results suggest that the SST k-w model (the reference case) is clearly superior, followed by the RNG k-e model, which also provides a fairly good performance. The discrepancies by the other models are very large. In fact, Figure 3.3.6.G shows that the indoor streamwise wind speed obtained with the other k-e models are several times higher than the one obtained using the SST k-wmodel (Ref. case). Figure 3.3.G.F also shows that the RSM and the standard k-wmodels tend to over predict the experimental values by about a magnitude.

The authors point out that several previous studies show a superior performance of the RNG k-e model for indoor air flow modeling, especially compared to the standard k-e model. This is to a large degree confirmed by our study. However, in our study, the SST k-w model even outperforms the RNG k-e model. Figure 3.3.6.H shows the differences in flow characteristics obtained with different turbulence models. The figure suggests that the main discrepancies are related to the direction of the jet entering the building model.

Impact of level of iterative convergence

The authors suggest that there is no clear consensus in literature about the level of iterative convergence, apart from the statement that convergence of critical variables should be monitored and confirmed. Care should be taken to avoid convergence criteria that are not tight enough. Figure 3.3.6.I indicates the iterations beyond the of 103 are needed to achieve a converged solution that shows a good agreement with the experimental data.

The authors point out that the results of their sensitivity analysis substantiate recommendations found in various Best Practice Guidelines. However, they point out that the impact of the inlet turbulent kinetic energy profile was very large. This point is very important since, as the authors stress, this large impact has not been shown before. This is particularly important because there is no consensus in literature on how the inlet turbulent kinetic energy profile should be calculated from the profiles of mean wind speed and streamwise turbulence intensity. In terms of turbulence model, the best performance was shown by the SST k-w model, followed by the RNG k-e model. The other models were found insufficiently capable of reproducing the magnitude and position of the standing vortex upstream of the building facade, and of the resulting direction of the jet through the ventilation opening. The authors recommend the use of at least second-order accurate discretization schemes, as well as sufficiently stringent convergence criteria. It is stressed that the convergence criteria suggested by commercial CFD codes are often too lenient and not stringent enough for accurate simulation results.

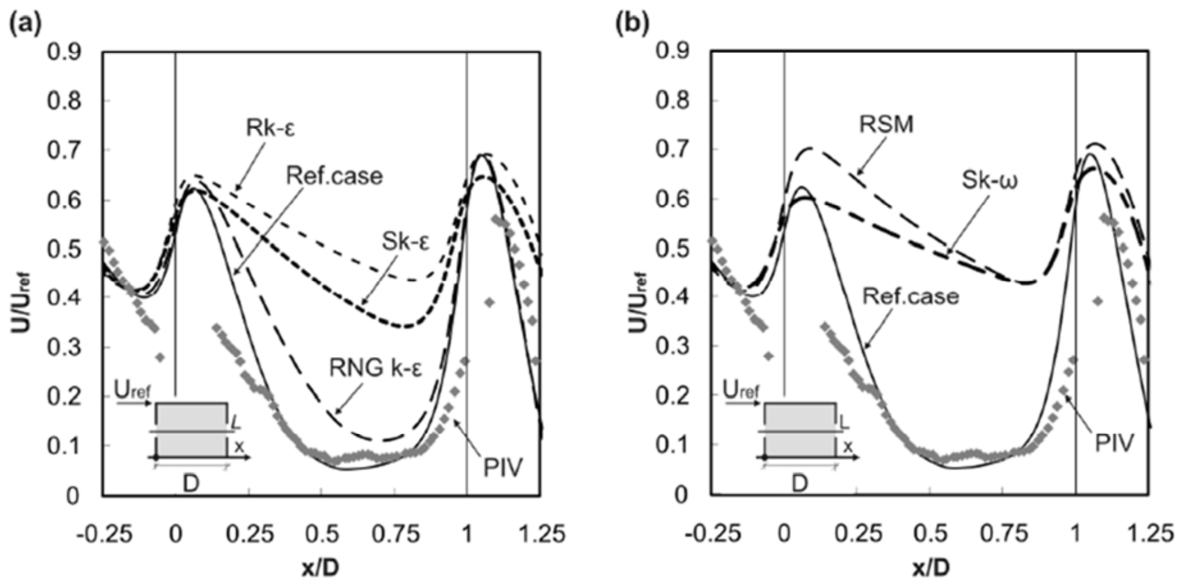


Figure 3.3.6.G: CFD simulation results for sensitivity analysis: impact of discretization scheme on the streamwise wind speed ratio along the centerline. For the reference case, a second-order discretization scheme is used. (Ramponi and Blocken, 2012).

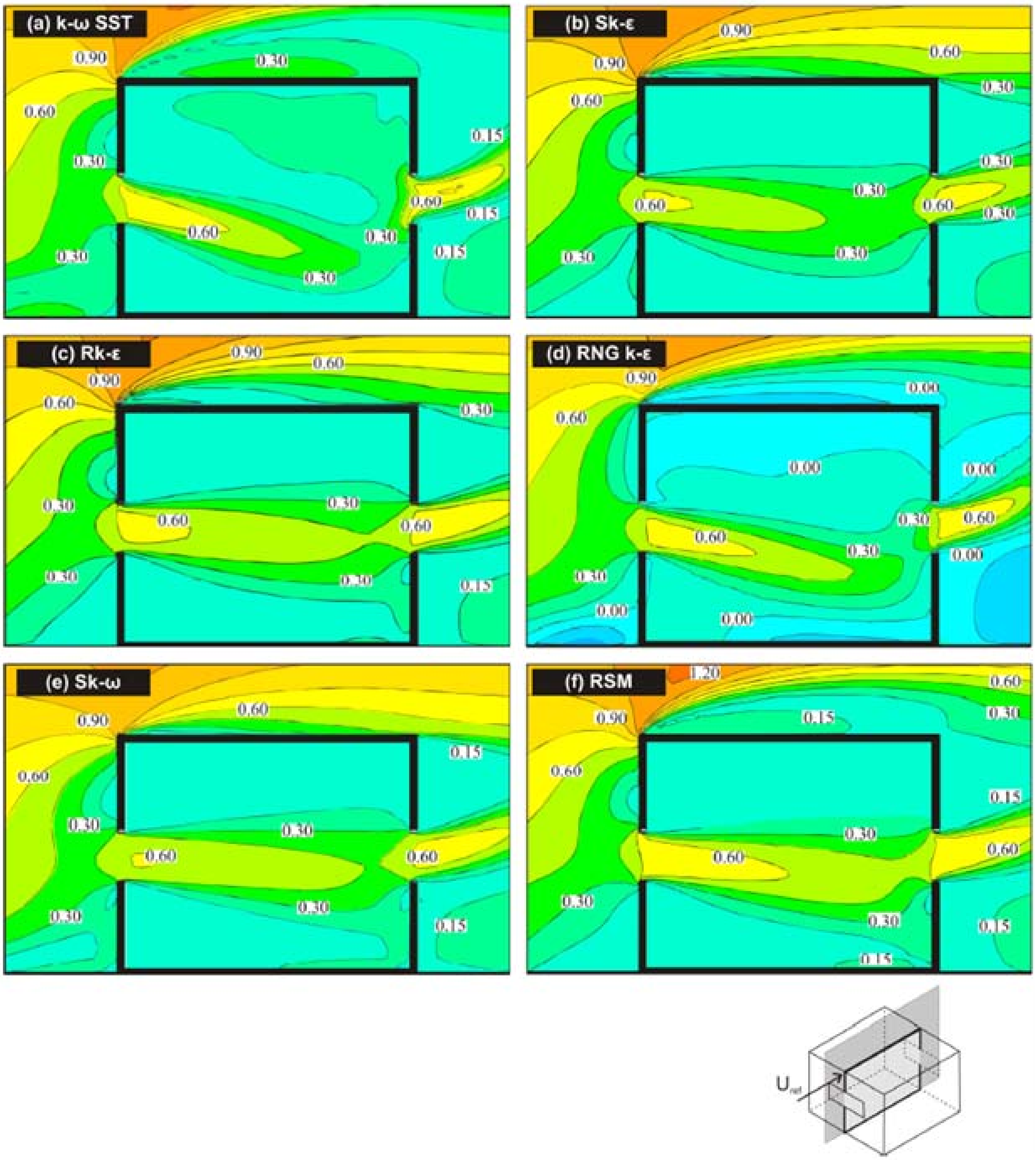


Figure 3.3.6.H: CFD simulation results for sensitivity analysis: impact of turbulence models on the wind speed ratio contours in the vertical centerplane. The SST k- model is the reference case in this study. (Ramponi and Blocken, 2012).

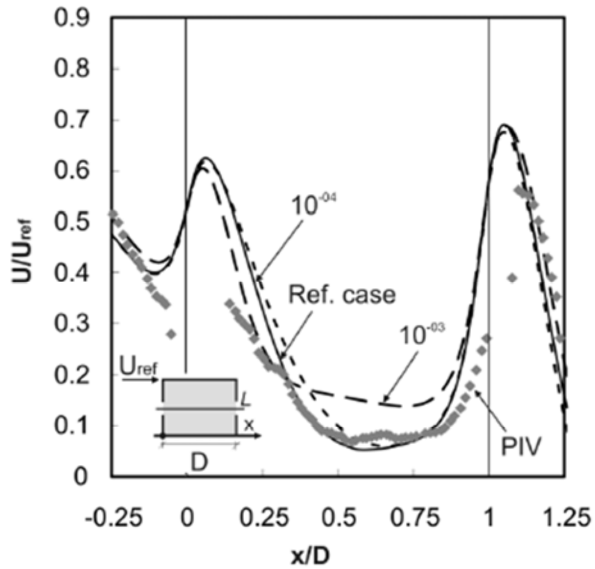


Figure 3.3.6.I: CFD simulation results for sensitivity analysis: impact of iterative convergence limit on thestreamwise wind speed ratio along the centerline. The convergence for the reference case is assumed when all the scaled residuals leveled off and reached a minimum of 10-6 for x, y and z momentum, 10-5 for k and 10-4 for ω and continuity. (Ramponi and Blocken, 2012).

3.4 CFD Post Processing

Some common post processing capacities of current CFD packages: Even from the comparison studying the capacities of CFD packages in 1998, Cook (1998) found that most CFD packages provide all important capabilities to visualize large amounts of data in their results files. At the present time most CFD packages can provide the following capabilities:

- 2d flow visualization: sections, contour plots / vector plots (line contours and shaded contours)
- 3d flow visualization: particle plot, streamlines, iso surfaces
- Animation
- Graphical plot of data

Some CFD packages also offer several post processing utilities such as user defined calculation (thermal comfort, moisture diffusion, etc.) or application-based specific calculation such as thermal comfort (PMV, PPD….) from DesignBuilderCFD, display multiple variables into a single scene, for examples, velocity vectors and temperature contours.

Some samples demonstrating post-processing related capabilities of several commercial CFD packages (ANSYS/FLUENT, STAR-CD+, DesignBuilder CFD, COMSOL)

2d data visualization: Most CFD packages allow the creating a snapshot of the selected solution data as well as provide user-friendly interfaces to locate section plans for visualizing the solution at specific locations and orientations defined by users. Figure 3.4.A illustrates slices showing the internal airflow within the horizontal plane and the vertical plane in DesignBuilder CFD application. Most CFD packages also allow visualization of results in contour/shaded format, particles, streamline format

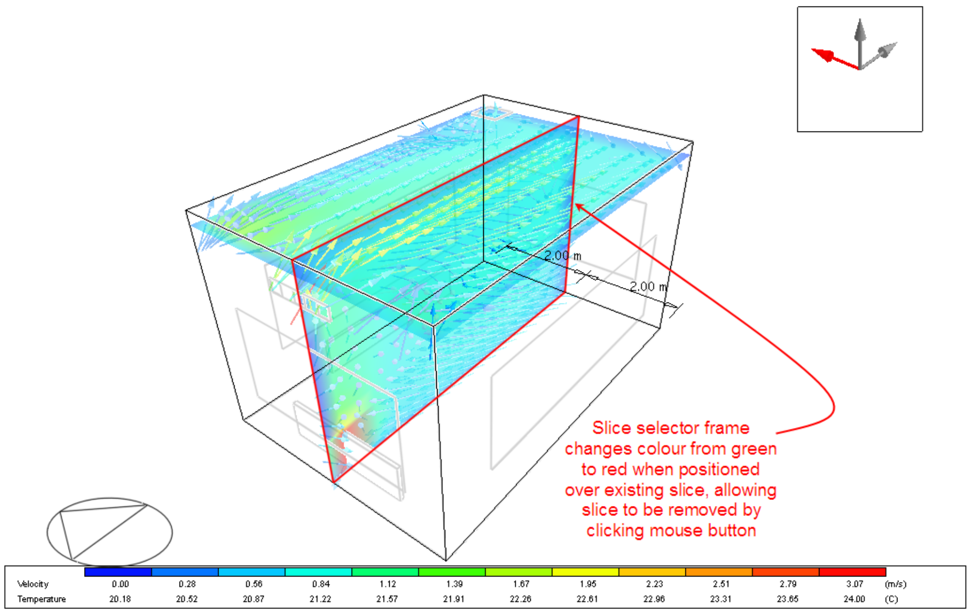
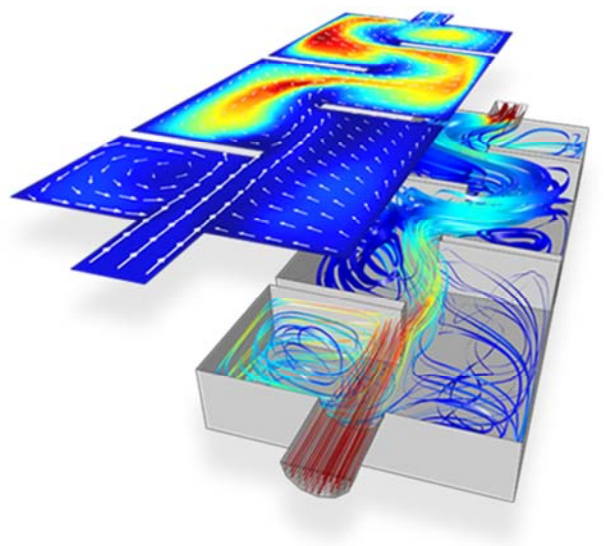


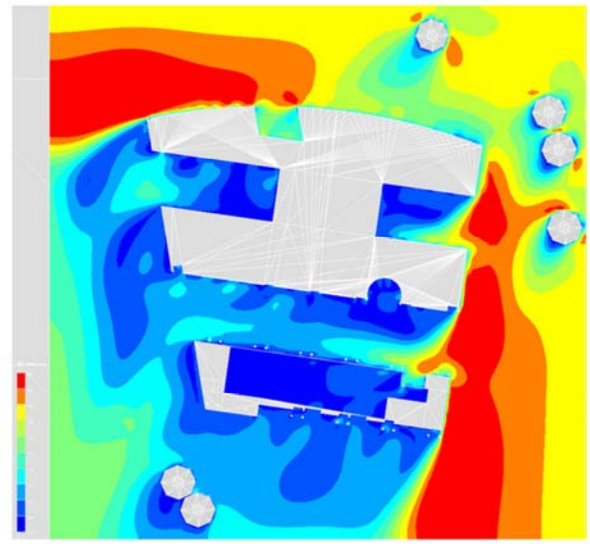
Figure 3.4.A: Setting up section plan for showing 2d data visualization in DesignBuilder CFD.

Some application also allows for creating different scenes/ design scenarios for comparing multiple steady state solutions in a single simulation file (Fig. 4.1.3) or displaying the result data at specific solution time (Fig. 4.1.4).

Graphic data plotting: Most current CFD packages allow integrated analysis and visualization tools (processing + post-processing) to providing live feedback on the progress of the simulation. This capability allows for stop the solution at any point, adjust parameters and resume the workflow as well as help to define required analysis before staring solution to reduce the number of runs (Fig. 3.4.B). Besides producing graphical information, many CFD packages also provide capacity to access to the results in ASCII format, which is very useful for comparing results with experimental data or other CFD codes.



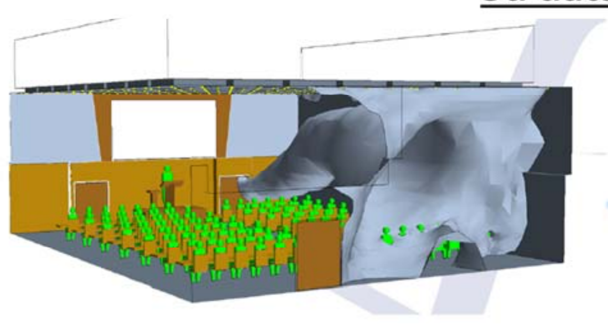
(a) Shaded contours, streamline format in COMSOL



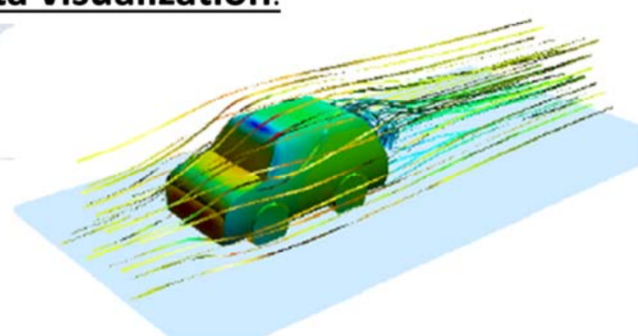
(b): Shaded contour (FlowDesignerCFD)

Figure 3.4.B: Different 2d data visualization formats

3d data visualization:



(b): streamline showing airflow (STAR-CD+)



(b): streamline showing airflow (STAR-CD+)

Figure 3.4.C:. Different 3d data visualization formats

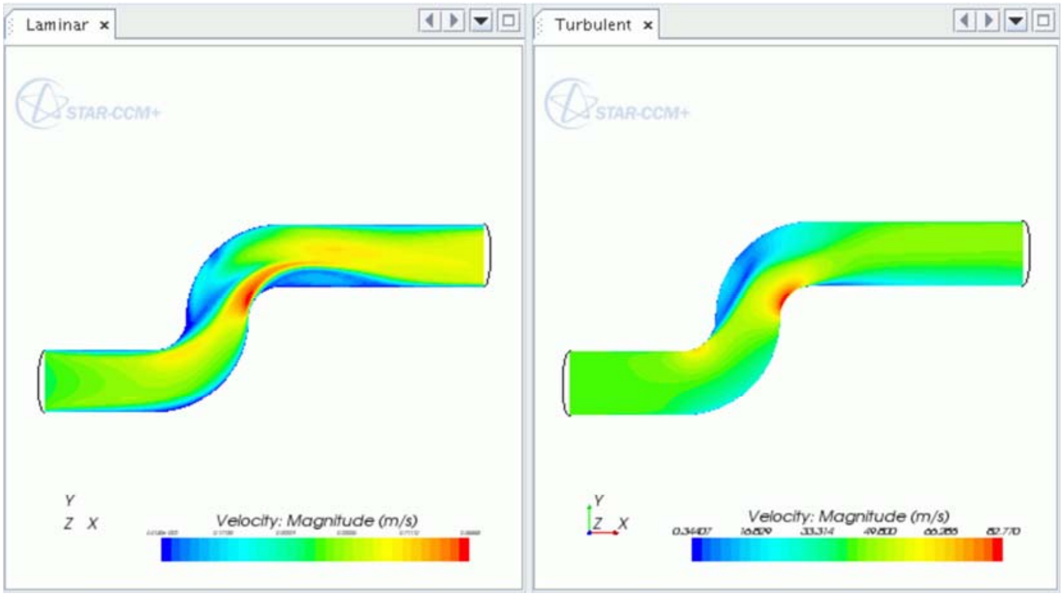


Figure 3.4.D. Multiple snapshot from different design scenarios for comparing the resulting data (STAR-CD+)

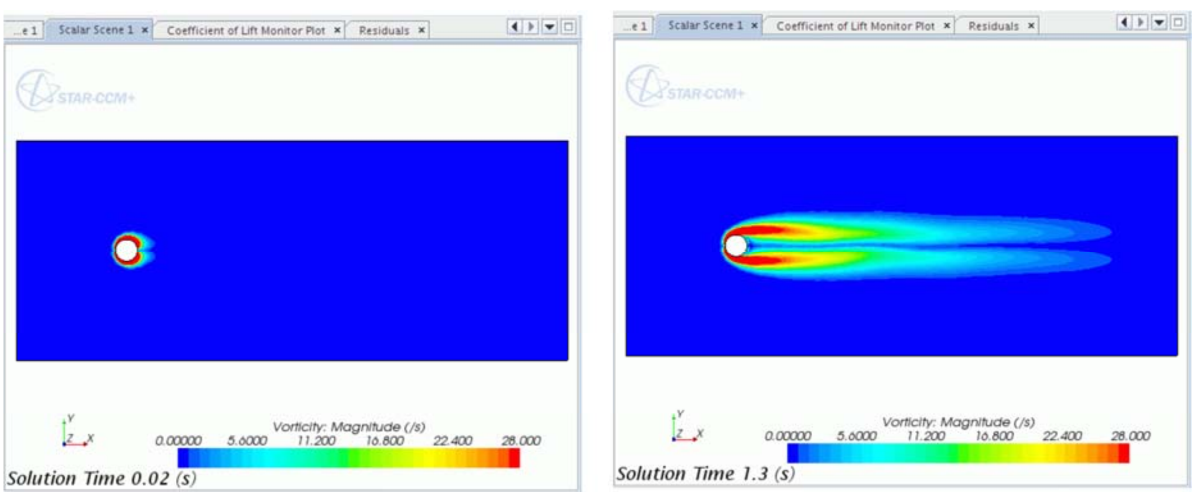


Figure 3.4.E: Displaying result data at specific solution time (STAR-CD+)

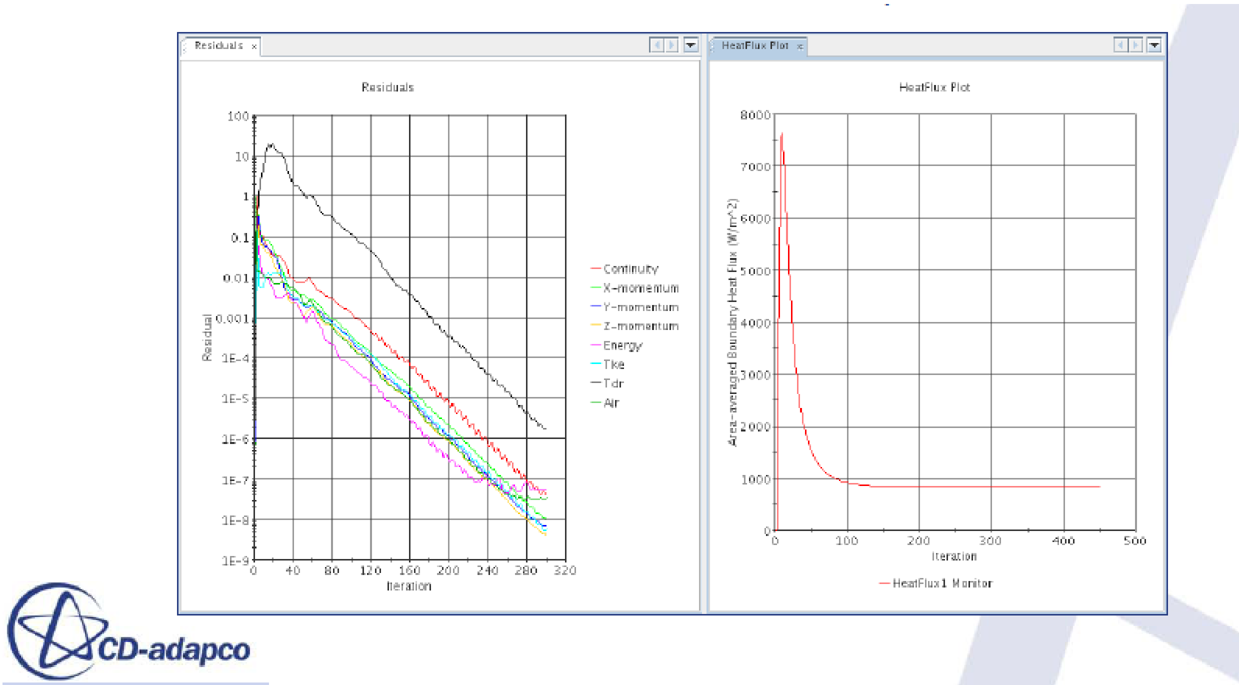


Figure 3.4.F: Data graphical plot showing the convergent criteria as well as result data

Part 2 – Listing of Primary Literature Reviewed

Abrishamchi, I. et al (2013) "A COMPARATIVE NUMERICAL INVESTIGATION OF FLOW THROUGH CHANNEL EXPANSION SYSTEM, USING FINITE VOLUME AND FINITE ELEMENT METHODS", Petroleum & Coal 55 (1) pages 20-25, 2013

Ahmad, S., Muzzammil, M. and Zaheer,I. (2011) "Numerical prediction of wind loads on low buildings", International Journal of Engineering, Science and Technology Vol. 3, No. 5, 2011, pp. 59-72

Ahuja, R., S. K. Dalui, et al. (2006). "Unpleasant Pedestrian Wind Conditions Around Buildings." Asian Journal Of Civil Engineering (Building And Housing) 7(2): 8.

ANSYS, (2009), “ANSYS FLUENT 12.0: User Guide”,Handbook, A. (2005). Airflow around buildings. Chapter 16, American Society of Heating Refrigerating and Air-Conditioning Engineers. 2005 ASHRAE Handbook—Fundamentals (SI)

Autodesk Simulation, (2013), “Finite Element vs. Finite Volume”, Wikihelp.autodesk.com

Biswas,R. and Strawn,R. (1998), “Tetrahedral and hexahedral mesh adaptation for CFD problem”, “Elsevier”, Applied Numerical Mathematics, November 1998

Blocken, B. and J. Carmeliet (2004). "A review of wind-driven rain research in building science." Journal of Wind Engineering and Industrial Aerodynamics **92**(13): 55.

Blocken,B.,Carmeliet, J. and Stathopoulos,T. (2007) "CFD evaluation of wind speed conditions in passages between parallel buildings – effect of wall-function roughness modifications for the atmospheric boundary layer flow", Journal of Wind Engineering and Industrial Aerodynamics 95(9-11): pages 941-962

Blocken B, Stathopoulos T, Carmeliet J. 2007. CFD simulation of the atmospheric boundary layer:– wall function problems. Atmospheric Environment 41(2): 238-252

Blocken etal (2011) "Application of CFD in building performance simulation for the outdoor environment: an overview", Journal of Building Performance Simulation, Vol. 4, No. 2, June 2011, pages 157–184

Blocken B, Janssen WD, van Hooff T. (2012) "CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus", Environmental Modelling & Software 30: 15-34.

Cabezon,D., Sanz,J. and Van Beek,J. (2010) "Sensitivitiy analysis on turbulence models for the ABL in complex terrain", CENER, National Renewbale Energy center, Wind Energy Department, Madrid (Spain)

Casey and Wintergerste (2000) "ERCOFTAC Special Interest Group on "Quality and Trust in Industrial CFD; Best Practice Guidelines", Fluid Dynamics Laboratory Sulzer Innotec; Published by European Research Community on Flow, Turbulence and Combustion; report 94 pages

Cóstola, D., B. Blocken, et al. (2009). "Overview of pressure coefficient data in building energy simulation and airflow network programs." Building and Environment.

Crastto, G. (2007). Numerical Simulations off the Atmospheric Boundary Layer Facoltà di Ingegneria, Dottorato di Ricerca in Ingegneria Industriale. Cagliari, Febbraio, Università degli Studi di Cagliari 195.

Franke, J. (2006) "Recommendations of the COST action C14 on the use in predicting pedestrian wind environmen", Fourth International Symposium on Computational Wind Engineering (CWE2006), Yokohama, 2006

Franke,J. et al (2007), "Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment", "COST", May 2007

Franke,J, Hellsten, A., Schluezen, H. and Carissimo,B. (2010) "Best Practice Guideline for the CFD simulation of flows in the urban environment : an outcome of COST 732", The Fifth International Symposium on Computational Wind Engineering

Franke,J, et al (2007) "Best Practices for the CFD simulation of flows in the urban environment", COST Action 732 QUALITY ASSURANCE AND IMPROVEMENT OF MICROSCALE METEOROLOGICAL MODELS, University of Hamburg, Meteorological Institute, Centre for Marine and Atmospheric Sciences

Glover,N., Guillas,S. and Malki-Epshtein,L. (2011) "Statistical calibration f CFD modeling for street canyon flows", Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney

Hefny, M. and Ooka,R. (2009) , "CFD analysis of pollutant dispersion around buildings: Effect of cell geometry", "Elsevier", Building and Environment, Volume 44, Issue 8, August 2009, Pages 1699-1706, ISSN 0360-1323

Heijmans,N. and Wouters,P. (2002) "Technical report: Impact of the uncertainties on wind pressures on the prediction of thermal comfort performances", International Energy Agency Energy Conservation in Buildings and Community Systems, IEA ECBCS Annex 35 (report 80 pages)

Denise Hertwig, GeorgeC.Efthimiou, JohnG.Bartzis, BerndLeitl, (2012), "CFD-RANS model validation of turbulent flow in a semi-idealized urban canopy", Elsevier, Journal of Wind Engineering and Industrial Aerodynamics, Volume 111, Page 61-72

Jakeman,A., Letcher, R., and Norton, J. (2006) "Ten iterative steps in development and evaluation of environmental", Environmental Modelling & Software 21 (2006) pages 602-614

Kleiven, T. (2003). Natural Ventilation in Buildings. Architectural concepts, consequences and possibilities. Department of Architectural Design, History and Technology, Norwegian University of Science and Technology. Doktor Ingeniør.

Kotani, H. and T. Yamanaka (2007). Wind pressure coefficient and wind velocity along building wall of apartment building with balcony. The 6th International Conference on Indoor Air Quality, Ventilation & Energy Conservation in Buildings IAQVEC 2007. Sendai, Japan, Airbase Database**:** 6.

Li, Jianqiang and Ward, Ian C. (2007) " Developing Computational Fluid Dynamics Conditions for Natural Ventilation Study ", Proceedings: Building Simulation 2007, Pages 1090 to 1096

McBride, D. et al (2008), "A coupled finite volume method for the computational modeling of mould filling in very complex geometries", "Elsevier", Computers & Fluids, Volume 37, Issue 2, February 2008, Pages 170-180, ISSN 0045-7930

Mendis,P., Samali,B. ad Cheun,J. (2007) "Wind Loading on Tall Buildings", EJSE Special Issue: Loading on Structures

Moeseke, G. v., et al. (2005). "Wind pressure distribution influence on natural ventilation for different incidences and environment densities." Energy and Buildings 37: 12.
File: Molina-Azi et. al – 2010 - Comparison of finite element and finite volume methods for simulation of natural ventilation in greenhouses

F.D. Molina-Aiz, H. Fatnassi, T. Boulard, J.C. Roy, D.L. Valera, (2010) "Comparison of finite element and finite volume methods for simulation of natural ventilation in greenhouses", Elsevier , Computers and Electronics in Agriculture, Volume 72, Issue 2, July 2010, Pages 69-86, ISSN 0168-1699

Montazeri,H. and Blocken, B. (2013) "CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: Validation and sensitivity analysis", Elsevier, Building and Environment, Volume 60, Issue 2, July 2010, Pages 137-149

Ngo and Gramoll, (1998), "Multimedia Engineering Fluid Mechanics", http://www.ecourses.ou.edu/cgi-bin/ebook.cgi?topic=fl

Nore,K., Blocken,B. and Thue,J, (2010) "On CFD simulation of wind-induced airflow in narrow ventilated facade cavities: coupled and decoupled simulations and modeling limitations", Building and Environment, February 15, 2010

Norris,S.. and Richards,P. (2010) "Appropriate boundary conditions for computational wind engineering models revisited, The Fifth International Symposium on Computational Wind Engineering (CWE2010) Chapel Hill, North Carolina, USA May 23-27, 2010

Ramponi R, Blocken B. 2012. CFD simulation of cross-ventilation flow for different isolated building configurations: validation with wind tunnel measurements and analysis of physical and numerical diffusion effects. Journal of Wind Engineering and Industrial Aerodynamics 104-106: 408-418.

Ramponi R, Blocken B. 2012. CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. Building and Environment 53: pages 34-48

Tominaga,Y. , Akashi Mochida, Ryuichiro Yoshie, Hiroto Kataoka, Tsuyoshi Nozu, Masaru Yoshikawa,Taichi Shirasawa, (2008), "Cross Comparisons of CFD Prediction for Wind Environment at Pedestrian Level around Buildings", The Sixth Asia-Pacific Conference on Wind Engineering (APCWE-VI), 2005

Tominaga,Y. et al (2008), "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings", Elsevier, Journal of Wind Engineering and Industrial Aerodynamics, Volume 96, Page 1749-1761

VanHooff,T. and Blocken,B. (2010).Coupled urban wind flow and indoor natural ventilation modeling on a high-resolution grid: A case study for the Amsterdam Arena stadium. Environmental Modeling & Software 25(1): 51-65.

Versteeg,H.K and Malalasekera,W. (2007) "Introduction to Computational Fluid Dynamics; The Finite Difference method", Second Edition, Pearson Education Limited,

Part 3 - Summary for each Primary Literature Reviewed

In Part 3 the reviewed primary literature is summarized by means of bullets indicating topics addressed in the reviewed literature which are useful in the external CFD research of the present project. In addition, important concepts of the reviewed literature are summarized and noted.

File: Abrishamchi-2013-comparison channel flow FVM and FEM.pdf

Summary of reference used in external CFD literature review:

Abrishamchi, I. et al (2013) "A COMPARATIVE NUMERICAL INVESTIGATION OF FLOW THROUGH CHANNEL EXPANSION SYSTEM, USING FINITE VOLUME AND FINITE ELEMENT METHODS", Petroleum & Coal 55 (1) pages 20-25, 2013

Topics that are useful for external CFD study:

- Comsol Multiphysics
- Fluent
- Comparison of FVM and FEM
- Influence of turbulent flow on heat transfer
- k-e turbulence model
- High Reynolds number and turbulence in equilibrium in boundary layer
- Experimental properties
- Swirl components

Important concepts covered in the report:

- Paper describes the turbulent flow in a pipe.
- Comparing Finite element and finite volume methods as numerical methods for solving differential equations
- The importance in using accurate methods
- The k-ε model is one of the most used turbulence models for industrial applications
- k-ε model is very suitable for completely disturbed flow
- Important assumptions of the k-e turbulence model, high Reynolds number and turbulence in equilibrium in boundary layer
- The result reveals that there is better agreement between experimental data and the numerical results by using finite element method rather than using finite volume method.

File: Ahmad etal -2011- wind loads on low buildings

Summary of reference used in external CFD literature review:

Ahmad, S., Muzzammil, M. and Zaheer,I. (2011) "Numerical prediction of wind loads on low buildings", International Journal of Engineering, Science and Technology Vol. 3, No. 5, 2011, pp. 59-72

Topics that are useful for external CFD study:

- Full-scale Reynolds number, boundary layer and turbulence properties
- Wind load effects (time averaged)
- Pressure coefficients
- Building with pitched roof and TTU (Texas Tech University)
- Codes for wind loads to predict pressure distribution
- Comparison between CFD and wind tunnel
- Pressure distribution and effects on natural ventilation.
- Prediction of time-dependent flow field with 3D Large Eddy Simulation
- Wind load predictions with k−εEddy Viscosity Model (k−εEVM), Algebraic Stress Model (ASM) and Large Eddy Simulation (LES).
- Steady velocity-pressure fields around an oscillating square prism by means of LES. The results of LES corresponded very well with the measured data.
- Process for computing with CFD products FLUENT
- Useful summary of parameters used in CFD simulations; i.e. power law, roughness, turbulence intensity

File: Ahuja-2006 -Unpleasant Pedestrian Wind Conditions around Buildings

Summary of reference used in external CFD literature review:

Ahuja, R., S. K. Dalui, et al. (2006). "Unpleasant Pedestrian Wind Conditions Around Buildings." Asian Journal Of Civil Engineering (Building And Housing) 7(2): 8.

Topics that are useful for external CFD study:

- Concept and fundamental of pedestrian level wind and pedestrian comfort
- Wind comfort criteria for tall building and surrounding
- Wind tunnel studies; Illustration details of wind flow and discomfort from high rise building configuration to pedestrians.
- Wind comfort recommendation to building design

Important concepts covered in the article:

- Wind velocity and velocity effects to pedestrian comfort.
- Wind comfort criteria for tall building and surrounding
- Wind load design and guide line
- Wind tunnel studies; Illustration details of wind flow and discomfort from high rise building configuration to pedestrians.

File: 3.2.2 ANSYS - 2009 -ANSYS CFD User Guide

Summary of reference used in external CFD literature review:

ANSYS, (2009), "ANSYS FLUENT 12.0: User's Guide", http://www.sharcnet.ca/Software/Fluent12/index.htm

Topics that are useful for external CFD study:

- Mesh quality (skewness, aspect ratio…)

Important concepts covered in the article:

- Turbulence models
- Selection proper turbulence model.

File: ASHRAE-2005-Chapter16-Air around buildings

Summary of reference used in external CFD literature review:

Handbook, A. (2005). Airflow around buildings. Chapter 16, American Society of Heating Refrigerating and Air-Conditioning Engineers. 2005 ASHRAE Handbook—Fundamentals (SI).

Topics that are useful for external CFD study:

- Concept of air flow around building
- Fundamental of wind patterns to buildings
- Wind pressure on buildings
- Atmospheric boundary layer parameter
- Local pressure coefficient

Important concepts covered in the article:

- Concept of air flow around building
- Fundamental of wind patterns to buildings
- Fundamental and example of calculation wind pressure on buildings
- Chart and example of calculation Atmospheric boundary layer parameter
- Fundamental and example of calculation of local pressure coefficient on building surface
- Fundamental of roof pressures
- Understanding source of wind data
- Estimating wind at sites remote from recording stations
- Wind effects on system operation
- Fundamental of Natural and Mechanical ventilation
- Building pressure balance and internal flow control
- Physical and computational modeling

File: AutodeskWiki – 2013 - Finite Element vs. Finite Volume

Summary of reference used in external CFD literature review:

Autodesk Simulation, (2013), "Finite Element vs. Finite Volume", Wikihelp.autodesk.com

Topics that are useful for external CFD study:

- General Fluid Flow and Heat Transfer Equations
- Turbulent Flow
- The advantages and disadvantages of FDM, FVM and FEM which should be concerned in CFD study in general.
- Application best practices (AEC best practices)

Important concepts covered in the article:

- Discretization methods (FDM, FVM, FEM)
- The difference between FDM, FVM and FEM.
- The advantages and disadvantages of FDM, FVM and FEM.
- Workflow of Autodesk CFD
- Application best practices

File: 3.2.2 Biswas and Straw - 1998 - Tetrahedral and hexahedral mesh adaptation for CFD problems

Summary of reference used in external CFD literature review:

Rupak Biswas, Roger C. Strawn, (1998), “Tetrahedral and hexahedral mesh adaptation for CFD problem”, “Elsevier”, Applied Numerical Mathematics, November 1998

Topics that are useful for external CFD study:

- Choice of mesh generation
- The advantages and disadvantages of applying adaptive tetrahedral and the hexahedral mesh adaptation procedures.

Important concepts covered in the article:

- Comparison between hexahedral and tetrahedral mesh adaptation.

File: Blocken and Carmeliet-2004-Wind Driven Rain:

Summary of reference used in external CFD literature review:

Blocken, B. and J. Carmeliet (2004). "A review of wind-driven rain research in building science." Journal of Wind Engineering and Industrial Aerodynamics **92**(13): 55.

Topics that are useful for external CFD study:

- Concept of Wind movement and Wind-driven rain (WDR) pattern on exterior condition.
- Moisture source affecting the hygrothermal performance and durability of building facades.
- Rain intensity vector; wind-driven rain intensity and horizontal rainfall intensity
- Field experiment and measurement on rain gauges and free stand and wall-mounted measurement methods
- Quantitative research of semi-empirical and numerical methods
- Predicted wind-driven rain forced by using steady-state of CFD simulation and standard k-ε turbulence model

Important concepts covered in the article:

- Wind movement and Wind-driven rain (WDR) pattern on external environment.
- Quantification of WDR assessment on building façade
- Accurate field measurement and CFD simulation of Wind-driven rain in the computational domain.
- Using semi-empirical and numerical methods serve for model validation and model development
- Predication CFD WDR model and future challenge on validating of spatial and temporal distribution of WDR.

File: Blocken et al -2007a- CFD evaluation of the wind speed conditions in passages

Summary of reference used in external CFD literature review:

Blocken,B.,Carmeliet, J. and Stathopoulos,T. (2007) "CFD evaluation of wind speed conditions in passages between parallel buildings – effect of wall-function roughness modifications for the atmospheric boundary layer flow", Journal of Wind Engineering and Industrial Aerodynamics 95(9-11): pages 941-962

Topics that are useful for external CFD study:

- Accurate CFD simulation of a horizontally homogeneous atmospheric boundary layer (ABL) flow
- Wind flow around buildings
- Wall function
- Roughness modification
- Atmospheric boundary layer;
- Horizontal inhomogeneity;
- Flow stability;
- Pedestrian wind environment;
- Venturi effect
- Fundamental studies of air flow around buildings and applied studies
- Comparison between wind tunnel and simulation
- Wall functions modified for roughness based on empirical formulae for sand-grain roughness, usually found in commercial CFD codes
- Internal Boundary Layer (IBL)
- CFD amplification factor
- Using experimental sand-grain roughened surfaces may be useful for modeling flow in roughened pipes and channels but can be unsuitable for the simulation of flow in the atmospheric boundary layer (ABL).
- Advised to assess the effects of horizontal in-homogeneity by first performing a simulation in an empty computational domain before conducting simulations with the building models present

Important concepts covered in the report:

- Performing validation simulations it is mandatory to quantify and reduce the different errors and uncertainties originating from these sources
- Avoid or at least reduce what is known as user errors
- Best practice guidelines on CFD for wind engineering problems concentrates on evaluation and validation of these models for flow around buildings and obstacles
- Turbulent flow modeled by the Navier-Stokes equations using closures for the turbulence: RANS, URANS, LES and hybrid RANS-LES
- Choice of the geometrical representation of obstacles: fist influence factor is the distribution of buildings, second he urban area include vegetation, orography and surface characteristics (e.g.roads, grass, sand).
- Choice of the computational domain, vertical and lateral extension
- Choice of boundary conditions; Inflow, wall, top, lateral, outflow

- Choice of initial data; for steady, stationary RANS as well as unsteady URANS and LES
- Choice of the computational grid; FVM and FEM
- Choice of the time step size, for unsteady simulations

File: Blocken et al -2007b- CFD simulation of atmospheric boundary layer

Summary of reference used in external CFD literature review:

Blocken B, Stathopoulos T, Carmeliet J. 2007. CFD simulation of the atmospheric boundary layer:– wall function problems. Atmospheric Environment 41(2): 238-252

Topics that are useful for external CFD study:

- Model sensitivity to the empirical constants contained within the k-" turbulence model
- Bayesian statistical calibration
- Direct numerical simulation (DNS) [is prohibitively computationally expensive]
- Standard k-e model (default values for empirical constants in most commercial CFD software)
- 2.5 dimensional simulation
- modeling the empty domain (no structures present) to ensure the correct profiles for velocity and turbulence
- Inlet boundary conditions, especially Inlet Velocity Profile, Turbulent Kinetic Energy profile and Turbulent Dissipation profile:
- Outlet boundaries
- Results indicate the a set of special empirical constants can be identified for specific CFD applications, e.g. environmental settings (in this paper it is the street canyon)
- Turbulence kinetic energy (TKE)
- First study to quantify uncertainties directly relating to the k-e model constants

Important concepts covered in the report:

- The paper suggests that varying the empirical constants in the k-e is sufficient to obtain good results for the simulations at the building level.
- Test of model sensitivity to the empirical constants contained within the k-" turbulence model and examination of how a systematic variation of these values could produce improved prediction of the turbulent kinetic energy when compared against wind tunnel data.
- The use of CFD modeling in the built environment is widespread for indoor applications, but simulations of the outdoor environment are still often carried out in atmospheric boundary layer wind tunnels.
- Achievable reduction of amount of computing power needed by focusing on the mean properties of the flow.
- Default values for these constants in most commercial CFD software.

File: Blocken etal -2011- CFD in building simulation outdoor environment.pdf

Summary of reference used in external CFD literature review:

Blocken etal (2011) "Application of CFD in building performance simulation for the outdoor environment: an overview", Journal of Building Performance Simulation, Vol. 4, No. 2, June 2011, pages 157–184

Topics that are useful for external CFD study:

- Mentioning Building Energy Simulation (BES), Building Envelope Heat-Air-Moisture transfer models (BE-HAM) and Computational Fluid Dynamics (CFD) as core bldg.. simulation methods
- Indoor CFD applications versus exterior environmental simulation involving the ABL
- Effects of scaling
- Comparison between Reynolds-averaged Navier-Stokes (RANS), unsteady RANS (URANS), Large Eddy Simulation (LES) or hybrid URANS/LES
- CFD simulation of wind flow around an isolated building
- Emphasis on steady RANS; but recognize the limitations of in external CFD
- Reported superior performance of LES compared to RANS for external CFD
- Streamwise gradients or horizontal inhomogeneity
- Pedestrian wind environment around buildings
- Convective heat transfer coefficients (CHTC) at exterior building facades
- Very few CHTC CFD simulations because of extremely high computational requirements for these simulations for building applications; e.g high Reynolds number and thin boundary layer close to the surfaces
- Air pollutant dispersion around buildings
- Schmidt number
- Simple cubic building model

Important concepts covered in the report:

- CFD in engineering practice is quite well established for indoor environment applications, this is considerably less pronounced for outdoor environment applications.
- Foundations for the current best practice guidelines, are the importance of grid resolution, the influence of the boundary conditions on the numerical results the performance of various types of turbulence models in steady RANS simulations.
- The main limitation of steady RANS modelling is incapability to model inherently transient features of the flow field such as separation and recirculation downstream of windward edges and vortex shedding in the wake. These features can be explicitly resolved by LES.
- Streamwise gradients or horizontal inhomogeneity, caused by inconsistency between the inlet boundary conditions, the wall functions, the computational grid and the turbulence model
- Steady RANS is the commonly used method; LES is still considered out of reach for practical pedestrian-level wind studies. This is mainly attributed to the much larger calculation time.

- Prediction of pollutant concentration distributions on and near buildings is important for building design and

File: Blocken etal -2012- CFD simulation pedestrian wind comfort

Summary of reference used in external CFD literature review:

Blocken B, Janssen WD, van Hooff T. (2012) "CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus", Environmental Modelling & Software 30: 15-34.

Topics that are useful for external CFD study:

- Framework for CFD wind studies in urban areas
- Integration of framework with BPG for CFD
- Reynolds-averaged Navier-Stokes equations with the realizable k-e model on an extensive high-resolution grid based on grid-convergence analysis
- Terrain related contribution versus design related contribution in CFD studies
- Advantage of CFD over wind tunnel experiments
- main elements of these guidelines:
    - The computational geometry should contain all buildings and obstacles that have a significant effect on the flow at the location of interest
    - computational domain should be large enough to avoid artificial acceleration of the flow (e.g select min. blockage ratio.
    - computational grid should preferably consist of hexahedral or prismatic cells near solid boundaries, with cell faces perpendicular to the boundary
    - grid resolution should be based on a grid-sensitivity or
    - grid-convergence analysis on at least three different grids
    - No unintended streamwise gradients
    - First-order discretization no best practice advice for the choice of turbulence models
- The existing BPG have mostly been developed for steady RANS

Important concepts covered in the report:

- Computational Fluid Dynamics (CFD), which offers considerable advantages compared to wind tunnel testing.
- Uncomfortable wind conditions have proven detrimental to the success of new buildings
- The aerodynamic information usually consists of two parts: the terrain related contribution and the design related contribution.
- Terrain related contribution represents the change in wind statistics from the meteorological site to a reference location near the building site i.e. the transformation of U pot to U 0 . The design related contribution represents the change in wind statistics due to the local urban design, i.e. the transformation of U 0 to the local wind speed U.

File: Cabezon etal -2010- sensitivity analyses turbulence ABL

Summary of reference used in external CFD literature review:

Cabezon,D., Sanz,J. and Van Beek,J. (2010) "Sensitivitiy analysis on turbulence models for the ABL in complex terrain", CENER, National Renewbale Energy center, Wind Energy Department, Madrid (Spain)

Topics that are useful for external CFD study:

- Fluctuating nature of wind and rapid variation of some physical properties within the first few meters in height
- Standard K-e turbulence model
- Using k-e for the simulation of the ABL overestimates the turbulence viscosity at steep areas with detached flows
- Alternative turbulence models such as k-e RNG and k-e Realizable (can be better for simulating certain ABL settings) [Renormalization Group Techniques (RNG)]
- Using Unsteady RANS and more sophisticated turbulence modes require a lot of computing time
- Reynolds stress and Large Eddy Simulations (LES)
- Different constants for standard k-e
- Computational domain for certain ABL simulations
- Boundary conditions for certain ABL simulations
- Wall functions, have special importance in the inlet profile
- Steep topographies
- Sensitivity analysis to verify the inlet profile, changing both velocity and turbulence intensity profiles

Important concepts covered in the report:

- CFD codes have to be adapted to simulate the behavior of the ABL through different turbulence closures schemes associated with different turbulence models
- Evaluating turbulence models that are different from the standard k-e model for a variety of fluid applications
- In the first layer (of the grid) the flow experiences a transition from fully turbulent to laminar with zero velocity at the wall.
- K-e with special set of constants can be used for CFD simulations of the ABL

File: Casey and Wintergerste -2000- ERCOFTAC Best Practice Guidelines

Summary of reference used in external CFD literature review:

Casey and Wintergerste (2000) "ERCOFTAC Special Interest Group on "Quality and Trust in Industrial CFD; Best Practice Guidelines", Fluid Dynamics Laboratory Sulzer Innotec; Published by European Research Community on Flow, Turbulence and Combustion; report 94 pages

Topics that are useful for external CFD study:

- Sources of error and uncertainties in CFD
- Numerical errors, convergence and round-off errors
- Turbulence Modeling
- RANS turbulences models
- Classes of turbulence models
- Near wall modeling
- Inflow boundary conditions
- Unsteadily and steady flows
- Laminar and transitional flows
- Application uncertainties
- Initial conditions and initial guesses
- Physical properties
- Training requirements
- Case study - Natural convection flow in square cavity
- Case study – Turbulent flow in a model outlet plenum

Important concepts covered in the report:

- The reference provides a wealth of guidelines how to apply industrial CFD applications
- A typical workflow with important steps and activities is presented
- The difference between errors and uncertainties and recommended remedies
- The objective of the guidelines is to encourage a common best practice by virtue of which separate analyses of the same problem, using the same model physics, should produce consistent results.
- Input and advice comes from a wide cross-section of CFD specialists, academics, end-users and, leading commercial code vendors.
- The guidelines are intended to offer roughly those 20% of the most important general rules of advice that cover roughly 80% of the problems likely to be encountered

File: Costola and etal-2009 -Overview of pressure coefficient data in BES:

Summary of reference used in external CFD literature review:

Cóstola, D., B. Blocken, et al. (2009). "Overview of pressure coefficient data in building energy simulation and airflow network programs." Building and Environment.

Topics that are useful for external CFD study:

- Comparison of pressure coefficient validation methods such full-scale measurement (on-site measurement), wind-tunnel measurement, and CFD.
- Wind pressure measurement on building façade
- Concept and fundamental pressure coefficient
- Comparison of pressure coefficient data
- Introducing site measurement and pressure coefficient validation

Important concepts covered in the article:

- Concept and fundamental for full-scale measurement (on-site measurement), wind-tunnel measurement, and CFD.
- Advantages and disadvantages of full-scale measurement (on-site measurement), wind-tunnel measurement, and CFD.
- Recommendation on site measurement and pressure coefficient validation instruments
- Overview of pressure coefficient data in Building Energy Simulation (BES) and Airflow network (AFN)
- Comparison of pressure coefficient data validation; validation across the façade and sheltering effects,
- Analytical models

File: Crasto-2007-Numberical Simulation of Atmospheric Boundary Layer

Summary of reference used in external CFD literature review:

Crastto, G. (2007). Numerical Simulations off the Atmospheric Boundary Layer Facoltà di Ingegneria, Dottorato di Ricerca in Ingegneria Industriale. Cagliari, Febbraio, Università degli Studi di Cagliari 195.

Topics that are useful for external CFD study:

- Concept of Atmospheric Boundary Layer and velocity vertical wind profile
- Discussion on similarity theory which refers to vertical profile of velocity and Reynolds stresses
- CFD Modeling of the ABL and turbulence model

Important concepts covered in the article:

- Fundamental, equation, and calculation of Atmospheric Boundary Layer and velocity vertical wind profile
- Discussion on similarity theory which refers to vertical profile of velocity and Reynolds stresses
- CFD Modeling of the ABL and turbulence model
- RANS equation and Eddy viscosity models (one equation and two equations turbulence models)
- 2D and 3D RANS simulation
- Eddies solving methodologies
- Wall function and rough surfaces
- Simulations 3D DES/LES over flat terrains
- Forrest modeling with WindSim

File: Davidson -2013- solids-and-fluids_turbulent-flow_turbulence-modelling.pdf

Summary of reference used in external CFD literature review:

Davidson,L. (2013) "Fluid mechanics, turbulent flow and turbulence modeling", Division of Fluid Dynamics, Department of Applied Mechanics Chalmers University of Technology, Goeteborg, Sweden

Topics that are useful for external CFD study:

- Complete text on fluid dynamics and turbulence modeling
- Governing flow equation
- Turbulent mean flow
- Transport equations for Reynolds stresses
- Reynolds stress models and two-equation models
- The SST Model
- Overview of RANS models
- Large Eddy Simulations
- Unsteady RANS
- DES
- Inlet boundary conditions
- and other topics

Important concepts covered in the report:

- Text is being used as a reference for all fluid dynamics and turbulence issues that relate to external CFD

File: Franke -2006- Recommend BPG CFD pedestrian wind MZ highL.pdf

Summary of reference used in external CFD literature review:

Franke, J. (2006) "Recommendations of the COST action C14 on the use in predicting pedestrian wind environmen", Fourth International Symposium on Computational Wind Engineering (CWE2006), Yokohama, 2006

Topics that are useful for external CFD study:

- Recommendations for the proper use of CFD in wind in built environment
- Sstatistically steady turbulent flows modeled by the RANS equations
- Boundary conditions
- Constant air density for high speed wind velocities
- Modeling errors from turbulence models and physical boundary conditions
- Comfort criteria
- Computational domain, e.g. lateral and top distance as well as downstream
- Recommended amount of blockage.
- Number of recommended wind directions
- Symmetry conditions when approach flow is parallel symmetric planes
- Inlet plane considerations, either measured profiles or equilibrium boundary layers
- Outlet conditions
- Roughness length
- Wall functions for rough walls
- Distance between first node and roughness
- Hexahedral cell with smaller truncation errors and better convergence
- Mesh at the boundary use prismatic cells
- Grid stretching

Important concepts covered in the report:

- Recommendations are restricted to statistically steady turbulent flows modeled by the RANS equations boundary conditions.
- Concern about the quality of the CFD predictions that many parameters which affect the outcome of the simulation can be freely chosen by the user.
- Numerical parameters which influence the accuracy of the solutions are addressed.
- Recommendations for the proper use of CFD for the simulation of pedestrian-level winds in the built environment.
- Several initiatives in Europe to create guidelines for the use of CFD.
- Pedestrian comfort criteria only require mean velocities; therefore time averaging of the basic equations is feasible for pedestrian wind predictions, leading to the well known RANS types that act as sources of error in
- Basically two type of errors; ffirst there are modeling errors that arise from the turbulence models used and the physical boundary conditions applied. The other errors stem from the numerical modeling, like the grid design, the truncation error of the discretization scheme and the error from incomplete iterative convergence.

File: 3.2.1 Franke et al - 2007- Effect of flow unsteadiness on the mean wind flow pattern in an idealized urban environment

Summary of reference used in external CFD literature review:

Jörg Franke, Antti Hellsten, Heinke Schlünzen, Bertrand Carissimo, (2007), "Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment", "COST", May 2007

Topics that are useful for external CFD study:

- Best practices for representing the geometries of obstacles
- Best practices for choosing computational domains
- Best practices for choosing boundary conditions

Important concepts covered in the article:

- Best practices for representing the geometries of obstacles
- Best practices for choosing computational domains
- Best practices for choosing boundary conditions
- Best practices for initial data
- Best practices for the computational grid
- Best practices for choosing time step size
- Best practices for choosing iterative convergence criteria

File: Franke et al -2010- MZ ..BPM for CFD in urban environment

Summary of reference used in external CFD literature review:

Franke,J, Hellsten, A., Schluezen, H. and Carissimo,B. (2010) "Best Practice Guideline for the CFD simulation of flows in the urban environment : an outcome of COST 732", The Fifth International Symposium on Computational Wind Engineering

Topics that are useful for external CFD study:

- Best practice guideline document for quality assurance and improvement of micro-scale meteorological models
- Procedures for undertaking simulation that are use to evaluate CFD codes.
- sources of error and uncertainty in CFD simulations that can be controlled and quantified by the user
- Errors and uncertainties in modeling the physics:
- Simplification of physical complexity
- Usage of previous experimental data
- Geometric boundary conditions
- Physical boundary conditions
- Initialization
- Numerical errors and uncertainties
- Computer programming computer round-off
- Spatial discretization
- Temporal discretization
- Iterative convergence

Important concepts covered in the report:

- Performing validation simulations it is mandatory to quantify and reduce the different errors
- and uncertainties originating from these sources
- Avoid or at least reduce what is known as user errors
- Best practice guidelines on CFD for wind engineering problems concentrates on evaluation and validation of these models for flow around buildings and obstacles
- Turbulent flow modeled by the Navier-Stokes equations using closures for the turbulence: RANS, URANS, LES and hybrid RANS-LES
- Choice of the geometrical representation of obstacles: fist influence factor is the distribution of buildings, second he urban area include vegetation, orography and surface characteristics (e.g.roads, grass, sand).
- Choice of the computational domain, vertical and lateral extension
- Choice of boundary conditions; Inflow, wall, top, lateral, outflow
- Choice of initial data; for steady, stationary RANS as well as unsteady URANS and LES
- Choice of the computational grid; FVM and FEM
- Choice of the time step size, for unsteady simulations
- Choice of iterative convergence criteria

File: Franke -2006- Recommend BPG CFD pedestrian wind MZ highL.pdf

Summary of reference used in external CFD literature review:

Franke,J, et al (2007) "Best Practices for the CFD simulation of flows in the urban environment", COST Action 732 QUALITY ASSURANCE AND IMPROVEMENT OF MICROSCALE METEOROLOGICAL MODELS, University of Hamburg, Meteorological Institute, Centre for Marine and Atmospheric Sciences

Topics that are useful for external CFD study:

- Direct Numerical Simulation
- Coriolis force (however, of little relevance in microscale domains)
- Simplification of the geometrical complexity present in the experiment Need to reduce the computational costs;
- Geometric details are often omitted.
- The difference between the exact solution of the basic system of partial differential equations and the numerical solution obtained with finite discretization in space and time.
- Iterative convergence
- Best Practice Guideline (this BPG differs from Casy & Wintergerste, 2000)
  BPG Choices of:
    - target variables
    - approximate equations describing the physics of the flow
    - geometrical representation of the obstacles
    - computational domain
    - boundary conditions
    - initial conditions
    - computational grid
    - time step size
    - numerical approximations
    - iterative convergence criteria

Important concepts covered in the report:

- Micro scale obstacle-accommodating meteorological models and their application to the prediction of flow and transport processes in urban or industrial environments
- Rule: these BPG offer “roughly those 20% of the most important general rules of advice that cover roughly 80% of the problems likely to be encountered”.
- For the evaluation of CFD codes it is necessary that all the errors and uncertainties that cause the results of a simulation to deviate from the true or exact values are identified and treated separately if possible.
- Iterative convergence (non-linear algebraic system that results from the discretization of the basic system of partial differential equations is either entirely or partly solved with an iterative method or by time integration towards a steady state)
- Best practice guidelines given to reduce errors and uncertainties.

File: Glover etl -2011- statistical calibration CFD streetflows

Summary of reference used in external CFD literature review:

Glover,N., Guillas,S. and Malki-Epshtein,L. (2011) "Statistical calibration f CFD modeling for street canyon flows", Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney

Topics that are useful for external CFD study:

- Model sensitivity to the empirical constants contained within the k-" turbulence model
- Bayesian statistical calibration
- Direct numerical simulation (DNS) [is prohibitively computationally expensive]
- Standard k-e model (default values for empirical constants in most commercial CFD software)
- 2.5 dimensional simulation
- modeling the empty domain (no structures present) to ensure the correct profiles for velocity and turbulence
- Inlet boundary conditions, especially Inlet Velocity Profile, Turbulent Kinetic Energy profile and Turbulent Dissipation profile:
- Outlet boundaries
- Results indicate the a set of special empirical constants can be identified for specific CFD applications, e.g. environmental settings (in this paper it is the street canyon)
- Turbulence kinetic energy (TKE)
- First study to quantify uncertainties directly relating to the k-e model constants

Important concepts covered in the report:

- The paper suggest that varying the empirical constants in the k-e is sufficient to obtain good results for the simulations at the building level.
- Test of model sensitivity to the empirical constants contained within the k-" turbulence model and examination of how a systematic variation of these values could produce improved prediction of the turbulent kinetic energy when compared against wind tunnel data.
- The use of CFD modeling in the built environment is widespread for indoor applications, but simulations of the outdoor environment are still often carried out in atmospheric boundary layer wind tunnels.
- Achievable reduction of amount of computing power needed by focusing on the mean properties of the flow.
- Default values for these constants in most commercial CFD softwares.

File: Blocken eta (2007) CFD simulation of the atmospheric boundary layer:

Summary of reference used in external CFD literature review:

Blocken,B., Stathopoulos,T. and Carmeliet,J.(2007) "CFD simulation of the atmospheric boundary layer: wall function problems", Atmospheric Environment 41(2): pages 238-252.

Topics that are useful for external CFD study:

- Simulations of atmospheric boundary layer (ABL) flow
- Wall function roughness
- Sand grain roughness applied to bottom of computational domain
- Unintended streamwise gradients
- Vertical mean wind and turbulence profiles at inlet plane
- neutrally stratified, fully developed, horizontally homogeneous ABL over uniformly rough, flat terrain
- Distinguish between approach flow and incident flow
- Sustainable boundary layer;
- Equilibrium vertical profiles;
- Horizontal homogeneity
- Internal boundary layer for terrain with roughness changes

Important concepts covered in the article:

- Importance to obtain accurate simulation of ABL flow in the computational domain.
- Correctly describe the flow profiles for mean wind speed and turbulence quantities at the inlet plane of the computational domain
- Describe different region: central region where structures are molded for CFD analysis, upstream and downstream regions, where their effect on the flow can be modeled in terms of roughness
- Using wall functions to represent surface roughness for the different regions of the computational domain
- Achieving horizontal homogeneity, e.g. vertical mean wind speed and turbulence profiles are in equilibrium with the roughness characteristics of the ground surface and there is no downstream gradient.
- Distinguishing between approach flow profiles that travel towards the building models and the incident flow profile that is obtained in a similar but empty computational domain.
- If wall roughness is expressed by an equivalent sand-grain roughness in the wall functions, four requirements are described which should be simultaneously satisfied.

File: 3.2.2 Hefny and Ooka - 2009 - CFD analysis of pollutant dispersion around buildings - Effect of cell geometry

Summary of reference used in external CFD literature review:

Mohamed M. Hefny, Ryozo Ooka, (2009) , "CFD analysis of pollutant dispersion around buildings: Effect of cell geometry", "Elsevier", Building and Environment, Volume 44, Issue 8, August 2009, Pages 1699-1706, ISSN 0360-1323

Topics that are useful for external CFD study:

- Cell geometry: tetrahedral-based vs. hexahedral-based messing
- Grid convergence index as the quantitative method on computational convergence criterion.

Important concepts covered in the article:

- Cell geometry
- Unstructured mess
- Grid convergence index

File: Heijmans and Wouters -2000 - Impact of the uncertainties on wind pressures

Summary of reference used in external CFD literature review:

Heijmans,N. and Wouters,P. (2002) "Technical report: Impact of the uncertainties on wind pressures on the prediction of thermal comfort performances", International Energy Agency Energy Conservation in Buildings and Community Systems, IEA ECBCS Annex 35 (report 80 pages)

Topics that are useful for external CFD study:

- Hybrid ventilation
- Ventilation prediction
- Wind pressures around buildings
- Uncertainties of predicting the wind induced ventilation driving forces
- Wind driven forces versus stack effect
- Strategies to obtain wind pressure coefficients besides CFD; from tabulated values and wind tunnel test
- Monitoring wind pressures coefficient on buildings
- Wind regime atmosphere versus local wind conditions
- Wind speed and wind direction distributions
- Assessing wind profiles; measurement devices
- Wind pressure monitoring; measurement devices
- Simulation of cross ventilation and stack ventilation
- Thermal model of the PROBE building
- Building appurtenances that affect ventilation performances and thermal exchange; Walls, windows and solar shading devices
- Coupling thermal and ventilation models
- Simulation validation
- Impact of these uncertainties on the performances at building level

Important concepts covered in the report:

- In a naturally ventilated building, airflows between zones are driven by two forces: the stack-effect and the wind. However, it is not evident to correctly estimate the wind effect. This depends the local wind velocity and the so-called wind pressures coefficient.
- To determine wind pressures around buildings, two parameters have to be known: the wind data (speed and direction) and the wind pressure coefficient.

File: CFD-RANS model validation of turbulent flow in a semi-idealized urban canopy

Summary of reference used in external CFD literature review:

Denise Hertwig, GeorgeC.Efthimiou, JohnG.Bartzis, BerndLeitl, (2012), "CFD-RANS model validation of turbulent flow in a semi-idealized urban canopy", Elsevier, Journal of Wind Engineering and Industrial Aerodynamics, Volume 111, Page 61-72

Topics that are useful for external CFD study:

- Reduced-scale wind tunnel measurement (physical modeling and flow measurement mehod)
- Using steady CFD code (RANS) from ADREA and STAR-CD applications for modeling unsteady flow in the urban context

Important concepts covered in the article:

- CFD validation using reduced-scale wind tunnel measurement (qualitative comparison, validation metrics, point-by-point validation, convergence analysis).

File: Jakeman et al.-2006- Ten iterative steps environmental models

Summary of reference used in external CFD literature review:

Jakeman,A., Letcher, R., and Norton, J. (2006) "Ten iterative steps in development and evaluation of environmental", Environmental Modelling & Software 21 (2006) pages 602-614

Topics that are useful for external CFD study:

- Basics of good model-development practice; The paper discusses good practice in construction, testing and use of models,
- Best practices in working with models in conjunction with natural resource
- Dangers of not working along guidelines
- Invalid conclusions very likely
- Conceptualizing the system
- Embracing alternative model
- Ten basic steps of good, disciplined model practice(flow chart in article included):
  - Define model purposes
  - Specify model context
  - Conceptualization of the system,
  - Select model features
  - Determine how parameter values are found
  - Choose performance criteria
  - Identify mode structure and parameter values
  - Verification
  - Quantification of uncertainties
  - Model evaluation

Important concepts covered in the report:

- Good practice in modeling increases the credibility and impact of the information and insight that modeling aims to generate. It is crucial for model acceptance and is a necessity for longterm, systematic accrual of a good knowledge base for both science and decision-making.
- Good models required in managing complex situations
- The only way to mitigate these risks is to generate wider awareness of what the whole modeling process entails.
- Stakeholder participation is a key requirement of good model development

File: Kleiven-2003-SJ-Natural Ventilation in Buildings

Summary of reference used in external CFD literature review:

Kleiven, T. (2003). Natural Ventilation in Buildings. Architectural concepts, consequences and possibilities. Department of Architectural Design, History and Technology, Norwegian University of Science and Technology. Doktor Ingeniør.

Topics that are useful for external CFD study:

- Natural ventilation; Driving force; Thermal buoyancy driven ventilation ; Single-sided ventilation; Cross ventilation; Stack ventilation; Wind pressure; Pressure coefficient; Wind speed; Building envelope; High-rise building
- Concept of natural ventilation and wind driving force
- Compare different driven ventilation; thermal buoyancy and wind driven ventilation
- Three different natural ventilation principles; single-sided ventilation, cross ventilation, stack and ventilation
- Wind pressure and wind pressure coefficient

Important concepts covered in the article:

- Principles and elements of natural ventilation
- Combination of natural and mechanical ventilation
- Local and central supply and exhaust paths
- Case studies of natural ventilation on architectural aspects
- Natural ventilation in a high-rise building
- Natural ventilation in a medium-rise building
- Natural ventilation in a low-rise building
- Architectural possibilities of natural ventilation related to façade, rood, plan and section, and interior space

File: Kotani and Yamanaka-2007-Wind pressure coefficient and wind velocity along building wall of apartment building with balcony

Summary of reference used in external CFD literature review:

Kotani, H. and T. Yamanaka (2007). Wind pressure coefficient and wind velocity along building wall of apartment building with balcony. The 6th International Conference on Indoor Air Quality, Ventilation & Energy Conservation in Buildings IAQVEC 2007. Sendai, Japan, Airbase Database**:** 6.

Topics that are useful for external CFD study:

- Wind pressure coefficient and wind velocity measurement and assessment
- Significant of wind-induced ventilation
- Experiment base on wind tunnel methodology
- Wind validation and measurement

Important concepts covered in the article:

- Wind pressure coefficient and wind velocity measurement and assessment along the building facade
- Significant of wind-induced ventilation and effects of fence, parapet, and parapet with partition
- Standard and baseline of model building scale in wind tunnel
- Grid validation technique for wind pressure coefficient and velocity

File: Li and Ward (2007) developing computational fluid dynamics conditions for urban natural ventilation study

Summary of reference used in external CFD literature review:

Li, Jianqiang and Ward, Ian C. (2007) " Developing Computational Fluid Dynamics Conditions for Natural Ventilation Study ", Proceedings: Building Simulation 2007, Pages 1090 to 1096

Topics that are useful for external CFD study:

- Domain sizes selected in CFD
- Urban wind profile at the inlet boundary as important input variable for CFD
- Dependencies of pressure coefficients on the density of structures around the tested building
- Comparison between wind tunnel roughness and CFD roughness settings
- Recommended Distance between CFD structure tested and the inlet, outlet, lateral and top boundary
- Formation of tested buildings in a normal and staggered pattern and the results on pressure coefficients

Summary of article:

The objective of the article is to describe methods of developing the correct conditions for a CFD model capable to simulating the **urban boundary layer**. The correct settings for the urban boundary layer can then be used to predict a representative **pressure coefficient regime around the built form**. The model concentrates on two aspects: the boundary conditions and domain size. The main methodologies involve developing a CFD boundary layers model and **testing it against the results of physical modeling in a boundary layer wind tunnel**. The domain size is dependent on the building area, density and layout.

The article suggests that urban wind profile is modeled more appropriately using urban roughness elements than the common log law or power law profile. The article suggests that results obtained CFD are sensitive to settings about the surface roughness of the ground, the domain size and the inline and lateral fetch. The computational domain size consists of two parts, one turbulence development, and another for neighborhood scale.

The article suggests that when codes are used to obtain the pressure coefficients the results obtained have a certain level of uncertainty since usually standards and codes do not account for the effect of structures adjacent to the building analyzed.

The article provides recommended distances as a function of height of building tested. The pressure coefficients are dependent on the patterns of groups of structures are placed in the CFD domain.

File: McBride et al - 2008 - A coupled finite volume method for the computational modelling of mould filling in very complex geometries

Summary of reference used in external CFD literature review:

D. McBride, T.N. Croft, M. Cross, (2008), "A coupled finite volume method for the computational modelling of mould filling in very complex geometries", "Elsevier", Computers & Fluids, Volume 37, Issue 2, February 2008, Pages 170-180, ISSN 0045-7930

Topics that are useful for external CFD study:

- A coupled finite volume method could be used for dealing with complex object geometries.

Important concepts covered in the article:

- A coupling method to combine the conventional cell-centered finite volume methods with a vertex-based flow solver approach

File: Mendis etal-2007-Wind Loading on Tall Buildings.pdf

Summary of reference used in external CFD literature review:

Mendis,P., Samali,B. ad Cheun,J. (2007) "Wind Loading on Tall Buildings", EJSE Special Issue: Loading on Structures

Topics that are useful for external CFD study:

- Determining the wind pressures on tall structures
- Consequence of turbulence on pressure loading on the building envelope
- Useful to start with a reference wind speed based on statistical analysis of wind speed records obtained at meteorological stations
- Fluctuating pressures, consequence on loading of building and pressure distribution
- Wind pressures function of characteristics of approaching wind, geometry of structure under consideration, and the geometry and proximity of the structures upwind.
- Actual pressures on building product of the gust dynamic wind pressure and the mean pressure coefficients; e.g. pressures are dynamic
- The flow pattern generated around a building is complicated by the distortion of the mean flow, flow separation, and the development of the wake.
- Along-Wind Loading versus Cross-Wind Loading
- For Cross-Wind Loading; Dominant periodicity defined by Strouhal Number; building subjected to periodic cross pressure loading,
- Interference from adjacent buildings
- CFD techniques may be used for determination of wind effects

Important concepts covered in the report:

- Wind is a phenomenon of great complexity because of the many flow situations arising from the interaction of wind with structures. Wind is composed of a multitude of eddies of varying sizes and rotational characteristics carried along in a general stream of air moving relative to the earth's surface.
- A consequence of turbulence is that dynamic loading on a structure depends on the size of the eddies. Large eddies, whose dimensions are comparable with the structure, give rise to well correlated pressures as they envelop the structure. On the other hand, small eddies result in pressures on various parts of a structure that become practically uncorrelated with distance of separation.
- Wind pressures on a structure are a function of the characteristics of the approaching wind, the geometry of the structure under consideration, and the geometry and proximity of the structures upwind. The pressures are not steady, but highly fluctuating, partly as a result of the gustiness of the wind, but also because of local vortex shedding at the edges of the structures themselves.
- Buildings of similar size located in close proximity to the proposed building can cause large increases in cross-wind responses. The designer should not only consider the existing conditions but make allowance for future changes in the surrounding area during the design life of the structure.

File: Moeseke and etal-2005-SJ- Wind pressure distribution influence on natural ventilation for different incidences and environment densities:

Summary of reference used in external CFD literature review:

Moeseke, G. v., E. Gratia, et al. (2005). "Wind pressure distribution influence on natural ventilation for different incidences and environment densities." Energy and Buildings 37: 12.

Topics that are useful for external CFD study:

- Concept of natural ventilation and wind driven ventilation
- Wind pressure coefficient experiment and validation methodologies, such as pressure coefficient input data, real scale measurements, wind tunnel tests, CFD, parametrical models, M. Grosso parametrical pressure model description and calibration
- Pressure coefficient validation from Computational Fluid Dynamic (CFD)
- 2D CFD

Important concepts covered in the article:

- Urban wind driven ventilation potential and concept of natural ventilation and wind driven ventilation
- Vertical and horizontal pressure coefficient gradients
- Investigate how wind may induce natural ventilation, with focus on wind incidence and large scale environment density influences
- Wind influence on natural ventilation is obvious. By creating high and low pressures on buildings' different faces, wind creates air flows inside buildings
- Wind pressure coefficient experiment and validation methodologies, such as pressure coefficient input data, real scale measurements, wind tunnel tests, CFD, parametrical models, M. Grosso parametrical pressure model description and calibration

File: Molina-Azi et. al – 2010 - Comparison of finite element and finite volume methods for simulation of natural ventilation in greenhouses

Summary of reference used in external CFD literature review:

F.D. Molina-Aiz, H. Fatnassi, T. Boulard, J.C. Roy, D.L. Valera, (2010) "Comparison of finite element and finite volume methods for simulation of natural ventilation in greenhouses", Elsevier , Computers and Electronics in Agriculture, Volume 72, Issue 2, July 2010, Pages 69-86, ISSN 0168-1699

Topics that are useful for external CFD study:

- Compare the efficiency of two different discretization methods (FVM and FEM) used CFD solvers for simulating the natural ventilation in greenhouses.

Important concepts covered in the article:

- The comparison between FEM and FVM in discretization approaches in terms of availability, computational accuracy and expense as well as physical data storage and virtual computer-memory required for intensively computational process.

File: Montazer and Blocken - 2013 - CFD simulation of wind-induced pressure coefficients on buildings with and without balconies - Validation and sensitivity analysis.

Summary of reference used in external CFD literature review:

H. Montazeri, B. Blocken, (2013) "CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: Validation and sensitivity analysis", Elsevier, Building and Environment, Volume 60, Issue 2, July 2010, Pages 137-149

Topics that are useful for external CFD study:

- CFD validation with wind-tunnel measurement of reduced-scaled models
- The expansion of the computational domain in CFD
- Setting for boundary conditions (turbulent kinetic energy k and turbulence dissipation rate ε, the aerodynamic roughness length z_o)
- Pressure coefficient distribution measurement
- Reference static pressure
- Grid-sensitivity CFD analysis

Important concepts covered in the article:

- Comparison between turbulence models (the standard k-ε (Sk-ε), the realizable k-ε (Rk-ε), the renormalization group k-ε (RNG k-ε), the standard k-ω (Sk-ω) and the Reynolds Stress Model (RSM)) in terms of computational accuracy and expense.

File: Ngo and Gramoll, (1998), “Multimedia Engineering Fluid Mechanics”, http://www.ecourses.ou.edu/cgi-bin/ebook.cgi?topic=fl

Summary of reference used in external CFD literature review:

Ngo and Gramoll, (1998), “Multimedia Engineering Fluid Mechanics”, http://www.ecourses.ou.edu/cgi-bin/ebook.cgi?topic=fl

Topics that are useful for external CFD study:

- Steady flow vs. Unsteady flow (periodic flow, non-periodic flow and random flow)
- Fundamental laws implemented in fluid dynamics (conservation of mass, linear momentum equation, moment of momentum equation, conservation of energy)
- Fundamental of boundary layer characteristics.

Important concepts covered in the article:

- Fluid kinematics
- Fundamental Laws (Integral Analysis)
- Fundamental Laws (Differential Analysis)

File: Nore etal -2010-CFD coupled decoupled.pdf

Summary of reference used in external CFD literature review:

Nore,K., Blocken,B. and Thue,J, (2010) "On CFD simulation of wind-induced airflow in narrow ventilated facade cavities: coupled and decoupled simulations and modeling limitations", Building and Environment, February 15, 2010

Topics that are useful for external CFD study:

- Coupled and decoupled CFD simulations for ventilated facade cavities
- Wind driven and buoyancy driving forces for the cavity ventilation
- Numerical heat-air-moisture (HAM) transfer models for building facades with ventilated cavities
- Darcy-Weisbach equation and friction and minor losses
- Pressure coefficients on the building envelope
- In coupled simulations, the ABL wind-flow pattern around the building and the resulting air flow in the cavities are calculated simultaneously and within the same computational domain; this has challenges sine Re domain to different in the same computational domain.
- In the decoupled simulations, two separate CFD simulations are made: a simulation of the outdoor wind flow around the building (with closed cavities) to determine the surface pressures at the position of the cavity inlet and outlet openings, and a simulation of the cavity air flow, driven by these surface pressures.
- Very little information on local losses for facade cavities.
- No standard values for local loss coefficients could be found for cavity hydraulic resistance data sets. This motivates the coupled CFD
- A combination of the high-Re number realizable k-εmodel with the so-called “enhanced wall treatment”
- Definition of the computational domain with distances to the inlet, top and lateral sides, and outlet.
- Comparing the results from the coupled and decoupled simulations allowed to assess the local losses (entrance and exit losses) of the cavities.

Important concepts covered in the report:

- In the coupled simulations, the wind-flow pattern around the building and the resulting air flow in the cavities are calculated simultaneously and within the same computational domain. In the decoupled simulations, two separate CFD simulations are conducted: a simulation of the wind flow around the building (with closed cavities) to determine the surface pressures at the cavity inlet and outlet openings, and a simulation of the cavity airflow, driven by these surface pressures. CFD validation is performed for the external and internal (cavity) flows.
- Cavity airflow rates can also be assessed by theoretical equations such as the Darcy-Weisbach equation that relates pressure differences to airflow rates, and in which a distinction is made between friction losses and local losses, such as entrance and exit losses.

File: Norris and Richards -2010- appropriate boundary conditions in cw.PDF

Summary of reference used in external CFD literature review:

Norris,S.. and Richards,P. (2010) "Appropriate boundary conditions for computational wind engineering models revisited, The Fifth International Symposium on Computational Wind Engineering (CWE2010) Chapel Hill, North Carolina, USA May 23-27, 2010

Topics that are useful for external CFD study:

- Paper describes **uncertainties in stagnation pressure calculations** caused by certain turbulence models
- Horizontally-homogeneous turbulent boundary layer (HHTBL); provide profiles and boundary conditions for HHTBLs modeled using the k-ω turbulence model
- HHTBL can also be analyzed using the Quasi-Isotropic Reynolds stress transport model
- Comparison between inlet and outlet profiles to show that there are no streamline gradients
- Eddy viscosity Stagnation pressure anomaly
- k-ε and k-ω models depend on eddy viscosity; resulting in an excessively large eddy viscosity
- For a high Reynolds number flow the flow is expected to obey Bernoulli's law, and the total pressure should remain constant along a streamline (or possibly dropping through viscous effects).
- Eddy viscosity based models may over-predict wind loadings (producing a too high stagnating pressure) Graph)
- Reynolds-stress models would not be expected to exhibit this spurious behavior.

Important concepts covered in the report:

- The onset flow for a wind-engineering model can be idealized as a horizontally homogeneous turbulent boundary layer, with the flow being driven by a shear stress at the top boundary.
- Modeling the atmospheric surface layer as a horizontally-homogeneous turbulent boundary layer (HHTBL); e.g. which is one with constant properties in directions tangential to the ground and hence the only variation is along the vertical axis.
- Boundary condition profiles should be for a horizontally-homogeneous boundary layer. They can be tested by computing the flow through an empty domain. The analytic profiles are specified at the inlet, the shear stress and ε or ω flux condition imposed at the top boundary, and a rough wall specified at the bottom of the domain.
- Ideally the **outlet profiles should equal those at the inlet**, and the value of the velocity and the turbulence scalars **should remain constant** along planes of constant elevation.
- Eddy-viscosity turbulence model can cause over-prediction of pressure on the windward face for the flow around a chimney, bringing into question the validity of using such a model in wind engineering flows. A Reynolds-stress transport model does not exhibit this spurious behavior

File: Ramponi and Blocken -2012a- simulation of cross-ventilation flow

Summary of reference used in external CFD literature review:

Ramponi R, Blocken B. 2012. CFD simulation of cross-ventilation flow for different isolated building configurations: validation with wind tunnel measurements and analysis of physical and numerical diffusion effects. Journal of Wind Engineering and Industrial Aerodynamics 104-106: 408-418.

Topics that are useful for external CFD study:

- Paper describes **that diffusion** is an important transport mechanism in cross-ventilation of buildings, and that special care is needed to select the right amount of physical diffusion and to reduce the numerical diffusion, by using high-resolution grids and by using at least second-order accurate discretization schemes.
- The present study has focused on validation of CFD simulations of coupled outdoor wind flow and indoor airflow for four different configurations of simple, isolated, single-zone buildings, and on the analysis of the effects of physical and numerical diffusion on cross-ventilation flow.
- The study only focuses on cross-ventilation.
- 3D steady Reynolds-Averaged Navier-Stokes (RANS) approach with the SST k-w model to provide closure.
- This turbulence model was chosen because of its superior performance compared to other RANS models for cross-ventilation of a simple isolated building,.
- The CFD simulations were performed at model scale; comparison to previous wind tunnel experiments.
- The SIMPLE algorithm was used for pressure-velocity coupling.
- Knowing the effects of numerical diffusion is also very important, because computational grids for suburban and urban configurations are often highly unstructured grids, on which convergence can only be obtained by introducing sufficient numerical (artificial) diffusion by the use of first-order discretization schemes.

Important concepts covered in the report:

- CFD simulations can be very sensitive to the large number of computational parameters that have to be set by the user. Therefore, CFD verification and validation studies are imperative, as well as detailed sensitivity studies that can provide guidance in the selection of computational parameters for future CFD studies.
- The measured vertical profiles of mean wind speed U and streamwise turbulence intensity I u are used to define the inlet boundary conditions for the CFD simulations.
- Careful selection of the inlet profiles and the roughness parameters k S and C S according to the consistency equation (Eq. (9)) is important to reduce unintended streamwise gradients in the flow profiles in the simulation, e. unintended changes between the inlet profiles and the incident profiles. The incident profiles are defined as those that would occur at the building position, if the building would be absent.

File: Ramponi and Blocken-2012c-CFD simualtion impact of computational parameters

Summary of reference used in external CFD literature review:

Ramponi R, Blocken B. 2012. CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. Building and Environment 53: pages 34-48

Topics that are useful for external CFD study:

- Review of cross-ventilation studies
- Detailed sensitivity analysis for CFD simulations of a cross-ventilated building model.
- The SST k- turbulence model shows the best agreement with PIV measurements.
- The turbulent kinetic energy strongly influences the convergence and the results.
- The prediction of the outdoor standing vortex largely affects the indoor airflow.
- Lack of extensive generic sensitivity studies for CFD simulation of natural cross-ventilation.
- Coupled approach, a single computational geometry and computational domain; includes both outside and inside environment of the building; most CFD research on wind induced cross-ventilation has applied the coupled approach.
- Decoupled approach, there are two different computational geometries and two different computational domains: one for the outdoor environment and one for the indoor environment of the building.
- Detailed sensitivity studies; systematically varying a single parameter compared to the reference case and evaluating the impact of this change on the simulation results.
- Detailed PIV measurements of wind-induced cross-ventilation for generic isolated building models were conducted.
- Impact of turbulence model
    - 3D steady RANS simulations are made with different turbulence models:
    - standard k- model (Sk-e)
    - realizable k-model (Rk-e)
    - Renormalization Group k-model (RNG k-e)
    - standard k- model (Sk-w) [87];
    - shear-stress transport k- model (SST k-w)
    - Reynolds Stress Model (RSM).
- indicates that the SST k-w model (Ref. case) is clearly superior, followed by the RNG k-e model, which also provides a fairly good performance.

Important concepts covered in the report:

- Accurate CFD simulation of coupled outdoor wind flow and indoor air flow is essential for the design and evaluation of natural cross-ventilation strategies for buildings.
- In CFD simulations of cross-ventilation involving large openings, a major issue of concern is the accurate modeling of the interaction between the outdoor wind flow around the buildings and the indoor air flow inside the buildings, which interact with each other at the ventilation openings. A distinction can be made between a coupled and a decoupled approach.

- Best practice guidelines were developed for indoor air flow. While these documents provide very valuable general information, they were not specifically focused on wind-induced cross-ventilation and therefore do not provide specific guidelines for cross-ventilation.
- Simulations. The simulations reproduce the main features of the flow, such as the standing vortex upstream of the building, which will be shown later to be very important, the contraction and expansion of the indoor flow and the separation zone on the roof.

File: Tominaga et al -2005-Cross Comparisons of CFD Prediction for Wind Environment at Pedestrian Level around Buildings

Summary of reference used in external CFD literature review:

Yoshihide Tominaga, Akashi Mochida, Ryuichiro Yoshie, Hiroto Kataoka, Tsuyoshi Nozu, Masaru Yoshikawa,Taichi Shirasawa, (2008), “Cross Comparisons of CFD Prediction for Wind Environment at Pedestrian Level around Buildings”, The Sixth Asia-Pacific Conference on Wind Engineering (APCWE-VI), 2005

Topics that are useful for external CFD study:

- CFD validation with full-scale measurement and reduced-scale measurement
- Boundary condition
- The refinement of grid discretization on the accuracy of the CFD validation

Important concepts covered in the article:

- On-site indoor measurement and CFD validation
- Reduced-scale wind tunnel measurement and CFD validation

File: Tominaga et al -2008- AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings

Summary of reference used in external CFD literature review:

Yoshihide Tominaga et al (2008), "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings", Elsevier, Journal of Wind Engineering and Industrial Aerodynamics, Volume 96, Page 1749-1761

Topics that are useful for external CFD study:

- Full-scale on-site measurement
- Computational domain
- Boundary layer condition
- Grid discretization (grid resolution for actual building complex)
- Canopy models for modeling small obstacle objects
- Turbulence models

Important concepts covered in the article:

- Reduced-scale wind tunnel measurement and CFD validation
- Guidelines for practical application of CFD
- Pedestrian wind environment
- Grid discretization

Convergence criteria

File: VanHooff and Blocken -2010a- Coupled urban wind flow and indoor natural ventilation modeling

Summary of reference used in external CFD literature review:

VanHooff,T. and Blocken,B. (2010).Coupled urban wind flow and indoor natural ventilation modeling on a high-resolution grid: A case study for the Amsterdam Arena stadium. Environmental Modeling & Software 25(1): 51-65.

Topics that are useful for external CFD study:

- Coupled CFD modeling approach for urban wind flow and indoor natural ventilation.
- Significant increase In natural ventilation effectiveness through appropriate alternative geometries
- Methods of determining urban wind flow are, 1) on-site full-scale experiments; (2) reduced-scale wind tunnel measurements; and (3) numerical modeling with Computational Fluid Dynamics (CFD).
- Difference between coupled and de-coupled simulations
- Computational grid for coupled simulations
- Advantages of coupled approach for urban wind flow and indoor natural ventilation studies.
- Need of high resolution grid
- 3D steady Reynolds-averaged Navier-Stokes (RANS) and realizable k-e turbulence model, supplemented with the Boussinesq model for thermal effects
- Pressure-velocity coupling is taken care of by the SIMPLE algorithm,
- Davenport roughness classification
- Alternative geometries to improve natural ventilation performance
- Actual maximum blockage ratio versus recommended maximum
- Complex grid large difference between smallest and largest lengths
- a bodyfitted (BF) grid
- Special geometry grid generation in and around ventilation openings
- Grid-sensitivity analysis
- Difference between isothermal thermal CFD simulations
- Validation through field measurements

Important concepts covered in the report:

- Describe a coupled (internal and external) CFD simulation to investigate natural ventilation performance of a larger building
- The paper describes a detailed case study with assumptions made, equations selected and empirical constants chosen
- The method of validation is shown.
- Describing the benefits and challenges of coupled CFD simulations
- The challenge to build a suitable mesh while considering the very large differences of cells
- Indicating different settings for steady state thermal simulations, although the thermal simulations should consider transient occurrences.

- Results obtained from steady RANS simulations goof fit with outdoor wind velocities but some discrepancies can possibly be attributed to the inability of steady RANS to simulate inherently transient effects in urban aerodynamics such as the collapse of separation and recirculation regions and vortex shedding in the wake of bluff obstacles.

File: No File – This is a Textbook

Summary of reference used in external CFD literature review:

Versteeg,H.K and Malalasekera,W. (2007) "Introduction to Computational Fluid Dynamics; The Finite Difference method", Second Edition, Pearson Education Limited,

Topics that are useful for external CFD study:

- The text book provides a comprehensive introduction to all applicable aspects of the external CFD