Chapter Four

Wind Studies and Research Tools

4.1. Introduction

This chapter is concerned with tools used for measuring wind direction and velocity. It tends to compare these tools to choose the most suitable one in terms of easy usage, low cost, availability, reliability, precision and accuracy.

There are two main tools: wind tunnels and computational fluid dynamics (CFD). The first one is the wind tunnels which has been in use since the 18th century. The latter computational fluid dynamics (CFD) has been discovered in the 21st century.

As an important element of climate, wind has significant effects on urban planning. In hot, dry, and dessert regions, wind direction and velocity constitute the main factors that ensure natural ventilation in buildings. These factors help reduce temperature in internal spaces. Wind and air movement should be studied and examined to explore the dominant direction and the mean velocity during the year to make use of the findings in urban planning process.

4.2. The Wind Measuring Tools

There are different tools used for measuring wind velocity and direction. Some of these tools are used to measure velocity only, others are used to measure only direction, and some are used for both purposes. The following paragraphs talk about some of them. [Elmusheir, 1977]

4.3. Anemometer

There are many types of anemometer, which used for measuring wind. Below are some of these types. [www.google.com/ wiki, 2015]

4.3.1. Vortex Hand held Wind Meter

The hand held anemometer has a digital display (see figure 4.1) It registers wind speeds up to and over 100 mph. It's powered by a coin battery for 1-2 years of typical use. It use wind sensor for kite surfing, windsurfing, target shooting, model aviation, weather monitoring, and all kinds of wind-related activities. [www.google.com/ wiki, 2015]

Figure (4.1) shows hand held wind meter [www.google.com/ wiki, 2015].

4.3.2. Vortex Pole Mount Anemometer

Wind sensor with digital display. It registers wind speeds up to and over 100 mph. It's powered by a coin battery for several months of typical use. It comes with a 25' wire standard, up to 500' custom. This anemometer is useful for applications and continuously monitoring where AC power is not available. [www.google.com/ wiki, 2015]

Figure (4-2) Pole mount anemometer [www.google.com/ wiki, 2015].

4.4. Wind Tunnels

It is the earliest techniques used in wind studies and analysis. It has been discovered in the 18th century. Over the past 50 years, wind tunnels have been used in both industry and research applications. Some of the wind tunnels are large enough to test aircraft. The others are small, so generators are used for blow up wind over the tested objects. [Kim, 2013]

4.4.1. History of Wind Tunnels

The concept of wind tunnels began in 18th century, when the flying machines were built to fly. To fly, man had to understand the flow of air over aircraft surfaces. This means that he had to build instrumented laboratories in which wings, fuselages, and control surfaces could be tested under controlled conditions. [Donald, 1960]

Today, the wind tunnel is used to measure aircrafts, lift, stability, and controllability of the aircraft before flight. In the early days of aeronautics, there were two approaches for testing the wind effects on aircrafts. In first method, the test model is moved through the air at the required velocity, and in second one, the air is blown past a stationary model.

The first wind tunnel was employed in 1707-1751 by Benjamin Robins, a brilliant English mathematician. His first machine called (whirling arm) had an arm 4 feet long, spun by a falling weight acting on a pulley and spindle arrangement, the arm tip reached velocities of only a few feet per second. The Whirling arm also used to measure the drag and lift of various airfoils, by Sir George Cayley (1773 -1857). The whirling arm was 5 feet long and attained tip speeds between 10 and 20 feet per second. The whirling arm provided most of the systematic aerodynamic data gathered up to the end of the nineteenth century. Its flaws, however, did not go unnoticed. Test results were adversely influenced as the arm's eggbeater action set all the air in the vicinity in rotary motion. Aircraft models on the end of an arm in effect flew into their own wakes. With so much turbulence, experimenters could not determine the true

relative velocity between the model and air. Furthermore, it was extremely difficult to mount instruments and measure the small forces exerted on the model when it was spinning at high speeds. Something better was needed.

That something better was a "wind tunnel." This utterly simple device consists of an enclosed passage through which air is driven by a fan or any appropriate drive system. The heart of the wind tunnel is the test section, in which a scale model is supported in a carefully controlled airstream, which produces a flow of air about the model, duplicating that of the full-scale aircraft

The aerodynamic characteristics of the model and its flow field are directly measured by appropriate balances and test instrumentation. The wind tunnels tests were and still have performed with scale models because wind tunnels are capable of handling full-sized model are too expensive.

Osborne Reynolds (1842-1912) of the University of Manchester demonstrated that the airflow pattern over a scale model would be the same for the full scale in both cases. This factor, now known as the Reynolds number, is a basic parameter in the description of all fluid flow situations including the shapes of flow patterns, the ease of heat transfer, and the onset of turbulence.

4.4.2. Definitions of Wind Tunnels

Wind tunnel is a tool used to study the effect of solid objects on wind pattern around it, and air moving past it. [Donald, 1960]

1- A closed tubular passage with the object under test is placed in the middle.

2- A powerful fan system to moves the air and pushes it toward the test object, the fan have straightening vanes and blades to smooth the air flow.

3- A sensitive balance connected with the test object to measure the forces generated by airflow.

4- Smoke or any colored gas instead of a sensitive balance, can be used and injected to make the flow lines around the visible.

There are many disadvantages of wind tunnels:

1- Full- scale objects (for instance, aircraft or vehicles), are tested in large wind tunnels, however these facilities are expensive to operate and some of their functions have been taken over by computer modeling.

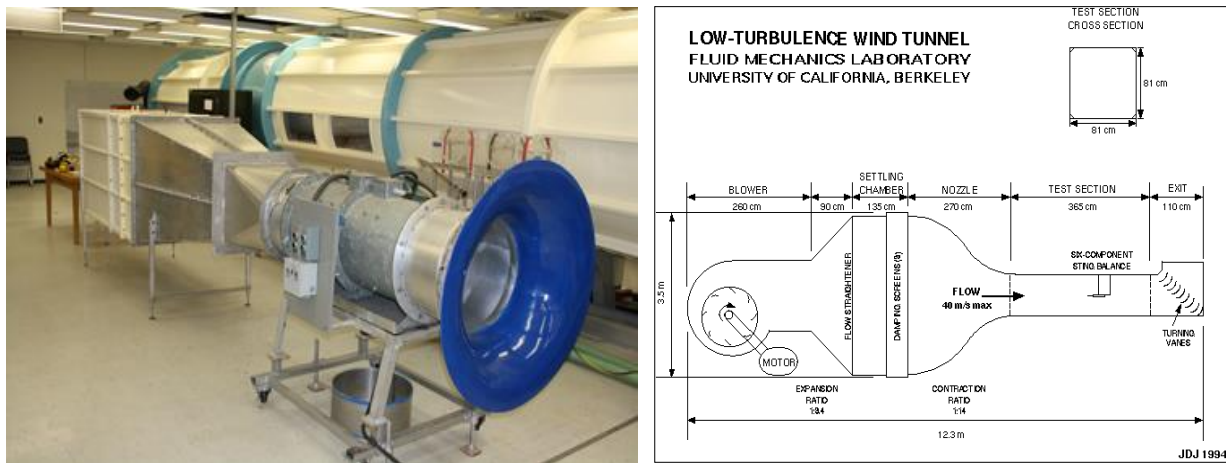2- It's difficult to use wind tunnels for full scale objects like bridges and office buildings.

Figure (4.3) shows medium wind tunnel [www.google.com/ wiki, 2015] Figure (4.4), Medium wind tunnel diagram. [www.google.com/ wiki, 2015]

4.5. Computational Fluid Dynamic (CFD)

4.5.1. Definition of CFD

CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. [Anderson, 1975]. It was employed to predict the flow fields in the early 1950s. The first CFD techniques were introduced when the digital computer was discovered. [Anderson, 1975]. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. [Anderson, 1975]. CFD means predicting physical fluid flows and heat transfer using computational methods [Anderson, 1975].

Ongoing research yields software capable of enhancing the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. [Anderson, 1975].

Validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests, building tests.

Fluid flows are found in virtually all areas of industry, especially during the manufacturing, operation of various machinery and components. For example, the automotive sector includes a whole world of different fluid and heat transfer mechanisms, such as cooling, combustion, ventilation and aerodynamics. Understanding how all these fluid and heat transfer mechanisms work is important for engineers and scientists to improve the operation of the mechanism and reduce its impact on the environment. Using CFD software, Engineers can build a virtual prototype of a product design they wish to analyze and get data and images that enable them to predict the performance of that design. [Anderson, 1975].

4.5.2. CFD Usages

CFD is used in a wide range of industries. Any industrial process that involves fluid flow and or heat transfer can benefit from CFD analysis. Below is a list of industrial and academic areas where CFD is commonly used. [google.com, wikipedia, 2015].

- Aerospace: Aerodynamics, wing design, missiles, passenger cabin

- Automotive: Internal combustion, underbody, passenger comfort

- Biology: Study of insect and bird flight

- Biomedical: Heart valves, blood flow, filters, inhalers

- Building: Clean rooms, ventilation, heating and cooling

- Civil Engineering: Design of bridges, building exteriors, large structures

- Chemical Process: Static mixing, separation, reactions

- Electrical: Equipment cooling

- Environmental: Pollutant and effluent control, fire management, shore protection

- Marine: Wind and wave loading, sloshing, propulsion

- Mechanical: Pumps, fans, heat exchangers

- Meteorology: Weather forecasting

- Oceanography: Flows in rivers, estuaries, oceans

- Power Generation: Boilers, combustors, furnaces, pressure vessels, nuclear

- Sports Equipment: Cycling helmets, swimming goggles, golf balls

- Turbo machinery: Turbines, blade cooling, compressors, torque convertors

- Heating ventilating and air conditioning design

Today, there are numerous commercial CFD software packages concerned specifically with building applications. [google.com, wikipedia, 2015]

4.5.3. How is CFD Used?

CFD software is used to calculate flow parameters for fluids, design and simulate fluidics. It is an important part for building and flow model analysis. Pre-processing is the first step in flow model construction. It involves the use of computer-aided design (CAD) software to design the model. The model is then overlaid with a mesh (grid work frame) so that statistical data about the fluid can be entered. Typically, this statistical data includes fluid viscosity, temperature and volume. CFD software is also used to simulate fluid mechanics. By entering the data required to build a model, mathematical computations are applied with programmed algorithms to arrive at the likely direction and movement of a fluid when acted upon by some outside force. [google.com, wikipedia, 2015].

CFD uses Navier-Stokes equations, differential notations that govern the motion of fluids. Finite element methods are also used for statistical modeling in CFD software. This methodology computes the movement through fluidic systems

using numerical methods called finite difference. Numerical computations are performed by using programming languages such as FORTRAN, C, and BASIC. Some types of CFD software use finite volume computation instead of finite differences. The finite volume method calculates fluid dynamics by solving differential equations using variables averaged across the volume of a fluid. Because of these averages, CFD software does not require modeling. Using a structured or constant mesh, and boundaries of the fluid do not need to be applied in an invasive fashion. This permits users of CDF software to focus on the analysis of the results rather than the tedious process of creating the model from the entry of many complex data sets. [google.com, wikipedia, 2015].

Suppliers of CFD software provide many different types of products. The most important factors to consider include cost, automatic mesh generation, fluid-flow regimes, CAD integration, and mathematical formulation.

Within the list of industries and applications listed above, CFD can include any of the following phenomena and flow regimes: [google.com, wikipedia, 2015].

- Laminar, turbulent flow

- Subsonic, Transonic, Supersonic, and Hypersonic flows

- Newtonian and Non-Newtonian fluid

- Multiple fluids, mixing and phase changes, and mass transfer

- Solid and fluid heat transfer, convection and thermal radiation

- Combustion of gas, liquids and solids

- Distributed resistances (porous media)

- Fluid-Structure interaction

- Aero acoustics and noise prediction

- Free surface flows, surface tension effects

- Time varying (transient) effects and moving boundaries

- Electromagnetic, electrostatic, electrochemical and other effects

- Casting, solidification and melting

4.5.4. The basics of CFD process. [google.com, wikipedia, 2015].

1. Pre-processing:

- Geometry, CAD, Solid model definition of domain

- Surface cleanup, preparation

- Volume mesh generation

- Definition of boundaries and conditions

- Physical property settings

- Numerical controls

2. Solving:

3. Perform computation using STAR-CCM+.

4. Post-processing:

- Analysis of CFD results

- Export results, Improve analysis

The fundamentals of CFD process will be discussed in details later in this chapter

4.5.5. CFD Benefits

- **Improved product quality**

Increasing product quality is a strategic objective of every company involved in product design or manufacture. Despite the fact that improvements in product quality are hard-won, increased product quality is the most frequently achieved benefit of using CD-adapco's CFD technology. [google.com, wikipedia, 2015].

- **Reduction in the number of physical prototypes**

The traditional product development process is built upon on an iterative "design-build-test" principle in which the influence of successive design changes is quantified by experimentation on a physical mock-up of the product. Increasingly, CFD is being used to replace some of these physical tests, reduce the number of physical prototypes required in the product development process and replace a number of 'design-build-test' iterations and runs with much quicker 'design-simulate' iterations. [google.com, wikipedia, 2015].

A faster-time-to-market is an obvious benefit of reducing the amount of physical prototyping required to bring a product to fruition, but also a direct benefit of the availability of CFD simulation data early in the design process. This allows designers to rapidly eliminate poor design variants, allowing them to focus their efforts on a smaller number of potentially more productive designs. [google.com, wikipedia, 2015].

- **Fewer field failures and avoided product recalls**

Although product recalls are rare, when they do occur, the cost can be enormous, in direct financial terms (the cost of executing the recall, performing repairs, providing replacements and compensating consumers), but more importantly in terms of lost reputation. CD-adapco's clients indicate that they have reported fewer product failures because of applying CFD. [google.com, wikipedia, 2015].

- **Increased satisfaction of external customers**

Customer satisfaction is the bigger picture. While the benefits of CFD listed above might help to increase margins and satisfy internal customers, the biggest benefit of any process improvement occurs when it makes a tangible difference to the end user of the product. [google.com, wikipedia, 2015].

-**Multiple Benefits**

Realized individually, any of the benefits described above is likely to yield significant bottom-line benefits for any organization that adopts CFD or CAE technology. However, the real benefit of CFD simulation is that even if you are seeking to realize a single specific benefit, the ancillary benefits of increased engineering insight will inevitably lead to a better overall product. [google.com, wikipedia, 2015].

4.5.6. CFD Philosophy

The approach of using supercomputers to solve aerodynamic problems began in the late 1970s. It was successful when it was applied in concepts of high maneuverability for the next generation of fighter planes. [Anderson, 1975].

CFD is cheaper and faster than wind tunnels. When the wing was redesigned, the cost of wind tunnel was $150,000 while the cost of the computer was estimated at $60,000. This appeared from the highly maneuverable aircraft technology of NASA in 1970. [Anderson, 1975].

Most of physical sciences and engineering are involved in pure theory on the one hand and pure experiment on the other hand. Recently as, say, 1960s, fluid dynamics drew only two approaches: theory and experiment. The appearance of supercomputers and the development of accurate numerical algorithms for solving physical problems have changed the way fluid dynamics is studied today. A new approach, the approach of computational fluid dynamics (CFD) has been introduced as the third fundamental one. See figure 4-5. [Anderson, 1975].

Today, the CFD is an equal partner with pure theory and pure experiment in the field of analysis and providing solutions for CFD problems. It complements the two approaches, but it doesn't replace either of them since both theories are indispensable. The future and advancement of fluid dynamics will depend on integration of all three approaches to help understand the results of theory and experiment, and vice versa. According to Anderson, CFD is commonplace enough [Anderson, 1975].
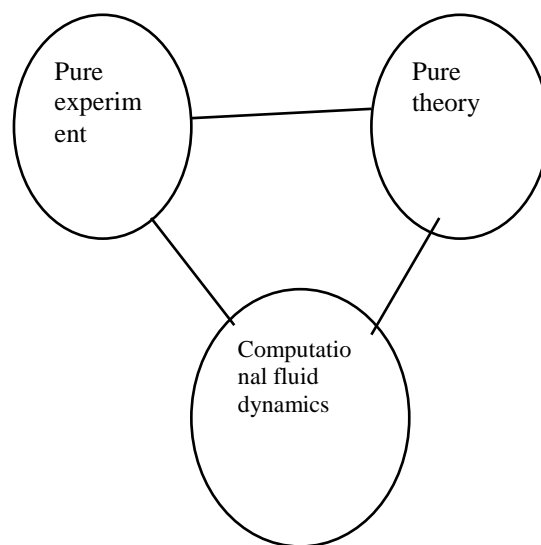

Figure (4-5) Three dimensions of fluid dynamics. [Anderson, 1975].

4.6. CFD as a Research Tool

CFD is transportable wind tunnel. Its results are directly analogous to wind tunnel results obtained in a laboratory. They both represent sets of data for given follow configurations at different Mach number, Reynolds numbers etc. However, the difference between wind tunnel and CFD is that wind tunnel is generally a heavy, unwieldy device. The CFD program (say in floppy disk, or memory card) is something you can carry around in your hand or better yet, a source program in the memory of a given computer that people (thousands of miles away) can access the terminals. [Anderson, 1975].

A computer program is a tool which can achieve numerical experiments. Figure 4-6 shows the solutions and output of CFD program. It solves complicated two dimensional Navier-Stokes equations for viscous flow by means of a finite difference numerical technique. [Anderson, 1975].
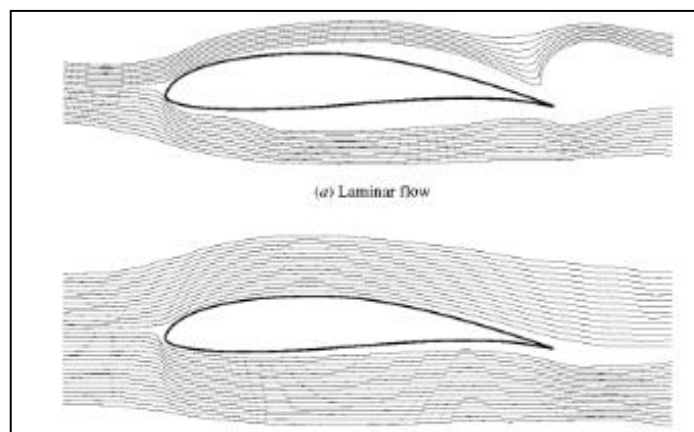
Figure (4-6) Example of CFD numerical experiment. [Anderson, 1975].

4.7. CFD as a Design Tool

CFD appeared in 1970 as analysis and design software but the types of computers and algorithms that existed at that time limited all practical solutions essentially to two-dimensional flows. CFD deals mainly in three- dimensional shapes analysis, but the storage capacity of computers and speed at that time were not appropriate for CFD. [Anderson, 1975]. However, since 1990 until today, a major development has occurred in the CFD in terms of ability to

86

analyze large three-dimensional shapes, fluid motion and wind. Since then the program has become available and easy to get. [Anderson, 1975].

Below are the five design phases introduced by AIA. [Kim, 2013].

1-Specify the project requirements by the design team and stakeholders. The scope of the design project should be outlined.

2- Create schematic design. In this stage the design concept should be conducted, the study drawings, documents, and any other media which illustrates the concepts of the design include spatial relationships horizontally and vertically, scale should be prepared and then revise by the owners.

3-Design development, in this stage, the initial design should be developed, the technical solutions are added, which, include, mechanical, electrical, plumbing, and structural elements, and the architectural documents are provided.

4- In this stage, construction documents after developing of the project are finalized and prepared.

5- Implementation of project by general contractor or builder.

CFD should be employed during the schematic phase to calculate airflows in and around the building, in order to modify and develop the project to achieve satisfactory indoor and outdoor environmental conditions for the building.

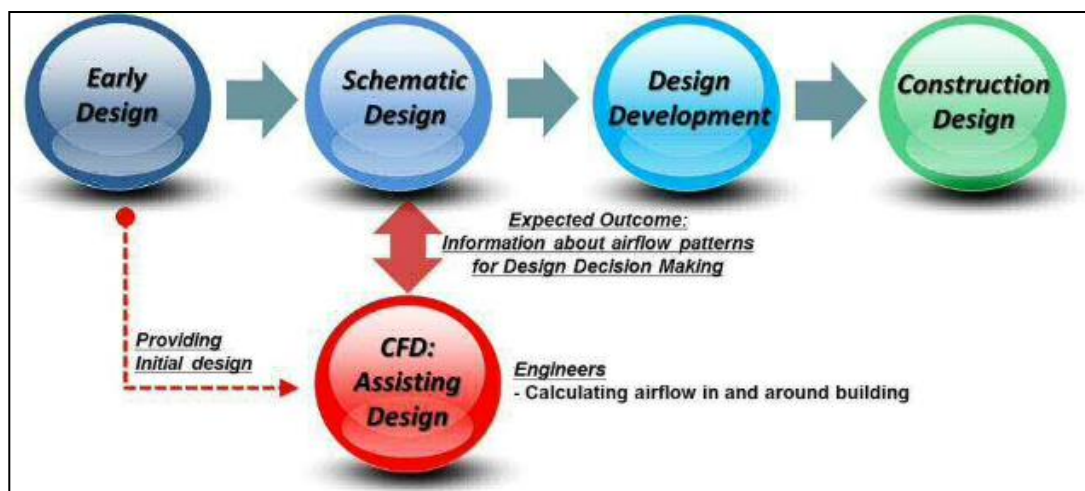Figure 4-7, shows the architectural design process when CFD simulation is used.

Figure (4-7) Architectural design process with CFD simulation. [Kim, 2013].

4.8. CFD simulation for building design &Urban Planning

CFD is successfully applied in building design and analysis and urban planning studies. Today, the software package is commercially available for building application purposes. CFD is important tool used for indoor environment evaluation of buildings and its interaction with building envelope as well as analyzing the outdoor environment surrounding the building, [Kimn, 2013]. CFD is also used for the prediction of external wind flow and the analysis of wind loading on building, bridges, and street canyons.

Here below are five examples showing different applications of CFD in buildings and urban planning:

1- Senates et al (2008) carried out study aiming to investigate the peak loads and stress at various locations on a roof, the effect of microburst and tornadic winds. The study was simulated by using CFD to quantify the resulting aerodynamic loading on building. Figure 4-8. [Kim, 2013].

2-CFD techniques were used to investigate pollutant concentrations in streets and on building surface surrounding the source. Figure 4-9. [Kim, 2013].

3-CFD has also been applied to test the natural ventilation, mixed-mode ventilation, and HVAC systems (head and ventilation air conditioning) in buildings. [Kim, 2013]. The test involves the prediction of air temperature, velocity, and relative humidity among other parameters. Figure 4-10. [Kim, 2013].

4- To protect building from biochemical and radiative agent and provide contamination control. CFD techniques could also be used to simulate the spread of fire and smoke through large volume spaces. Figure 4-11. [Kim, 2013].

5- The airflow around buildings was simulated by CFD in many urban areas in the word to study wind velocity, natural ventilation and the thermal comfort. Figure 4-12. [Kim, 2013].
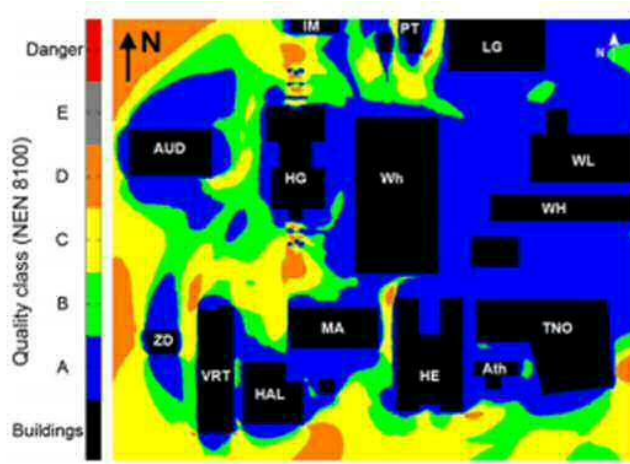
Figure 4-8   Assessment of pedestrian wind comfort [Janssen et al, 2013].
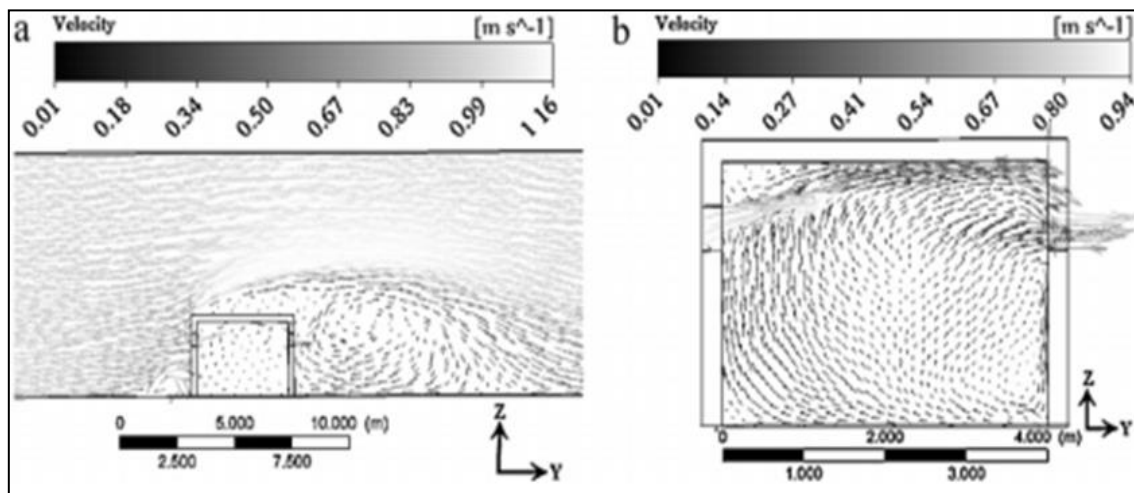
Figure 4-9 Prediction of natural ventilation [Bangalee et al., 2012]
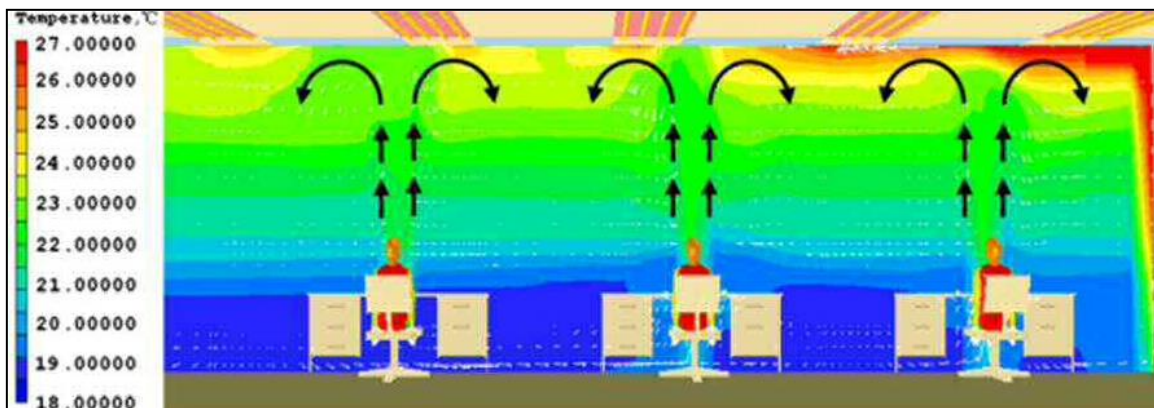
Figure 4-10 Prediction of natural ventilation [Bangalee et al., 2012]
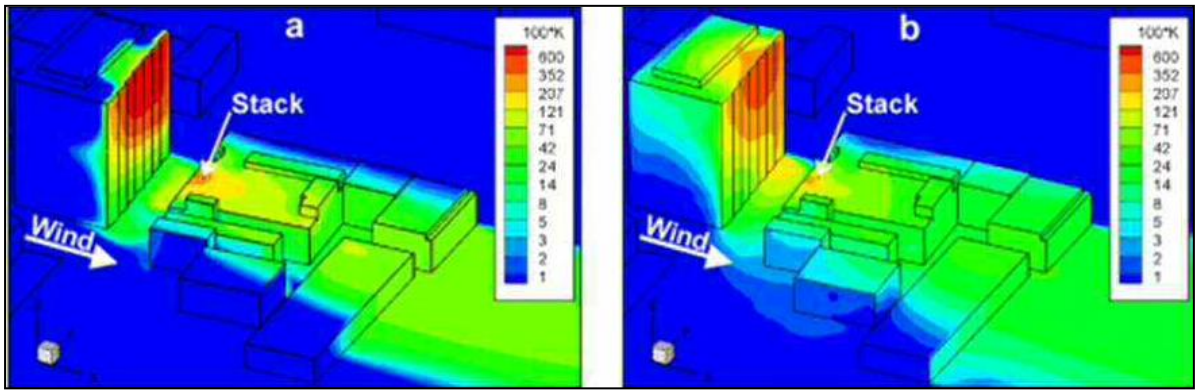
Figure 4-11 Prediction of natural ventilation [Bangalee et al., 2012]
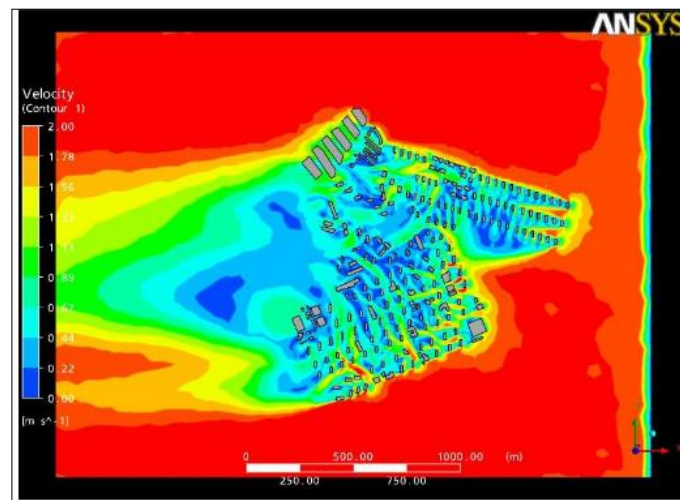
Figure 4-12 wind patterns around buildings [Renan, 2011]

4.9. Practice Guidelines for CFD

There are many important variables, which must be taken into consideration for efficient CFD performance. These variables are the approximate form of the governing equations, the turbulence model, and the level of detail in the geometrical representation of the buildings, the size of the computational domain, the type and resolution of the computational grid, the boundary conditions, the discretization schemes, and the iterative convergence criteria.

Since 1970s and 1980s, many investigations have been carried. The investigations focused on issues related to the effect of the size of the computational domain, the grid resolution, the boundary conditions and the turbulence model on the computational results. These investigations had a good contribution in practicing guidelines of CFD in urban aerodynamics. They

introduced the best practice guidelines by using RANS equations to achieve high-quality results by CFD. [Kim, 2013].

These guidelines aimed to help less experienced and experienced users move to a new application area, and ensure the accuracy of CFD in wind engineering applications. They introduced best guidelines and recommendations on the proper use of CFD simulations in the built environment. [Kim, 2013].

Moreover, a set of guidelines for CFD prediction about the pedestrian wind environment around buildings, and the numerical prediction of wind loads on buildings were introduced by Architectural institute of Japan. These guidelines based on steady RANS equations and use of large eddy simulation (LES). [Kim, 2013].

4.10. Recommendations for the Use of CFD for Wind around Buildings

The six recommendations mentioned below are derived from the best practice guidelines which aim to avoid or at least reduce user errors caused by the incorrect use of CFD and the lack of experience or resources. [Kim, 2013].

4.10.1. Geometrical Representation

Distribution of buildings and topography on wind flow, so the buildings and area of interest should verify the conditions mentioned below:

1-To avoid details of a small region around the region of interest to limit the consideration of details and thus avoid the need for increased computational resources. [Kim, 2013].

2- Buildings located away from the center region can be represented by simple blocks. [Kim, 2013].

3- Area of a radius 1-2 H from the building of interest should be reproduced as accurately as possible (H = height of building interest). [Kim, 2013].

At least one additional street block in each direction around the region of interest should be clearly reproduced. (Figure 4-12).
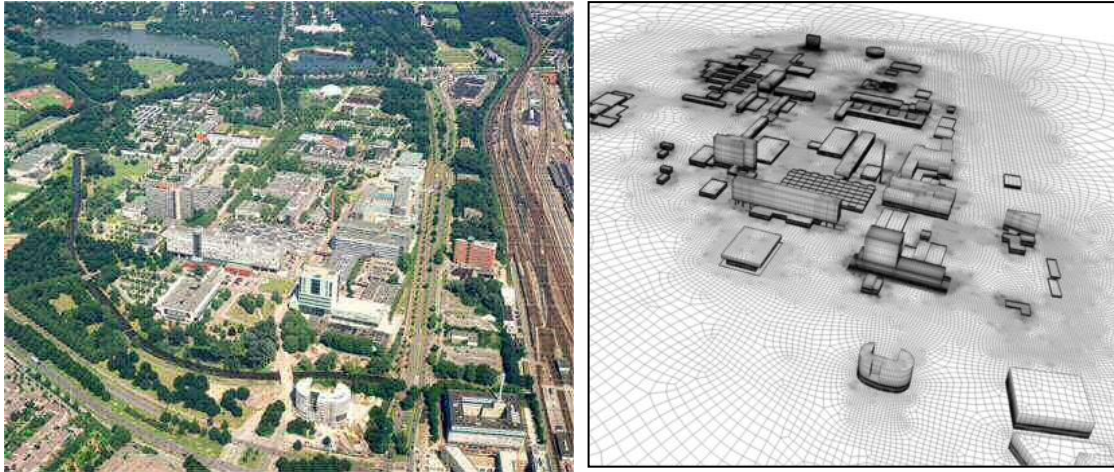
Figure 4-13 - An example of geometric resolution [Franke, et al, 2007]

4.10.2. The Size of the Computational Domain

Size of the entire computational domain depends on the conditions of targeted area and boundary. For a single building, the distance from the top of the building to the top of the computational domain should be at least 5H with a maximum blockage of 3%, where H is the building height. For the lateral boundary, 2.3H is required between the building's sidewalls and the edge of the computational domain. 5 H is also considered to be the minimum for the inflow boundary when the approach flow profiles are well known. If the approach flow profiles are not provided, a larger distance is required in order to establish a realistic flow profile. For the outflow boundary, at least 15H behind the building is suggested [Franke, et al, 2007].Also 5H for the lateral boundary and 10H for the outflow boundary is suggested.

The 5Hmax, for both the top boundary and the lateral boundary is recommended for urban areas with multiple buildings, where Hmax, is the height of the tallest building. Also the size of the computational domain in the case of the boundary layer wind tunnel. However, if the height of the wind tunnel is more than 6Hmax, a lower height of the computational domain can

be employed. For lateral boundary, if the distance of the lateral walls of the wind tunnel from the built area is much larger than 5Hmax, a smaller extent of the computational domain can be used. [Kimn, 2013].

Figure 4.13 shows the dimensions of the computational domain specified by Lateb et al. (2013), Here, the top and the lateral boundaries are 5H away from the building and the outflow boundary is 20H downwind from the building to allow adequate flow development. [Kim, 2013]

Therefore, 5H for the top and the lateral boundary is recommended in both the case of a single building and urban areas with multiple buildings. In addition, 5H and 15H can be used for the upstream and the downstream length respectively.
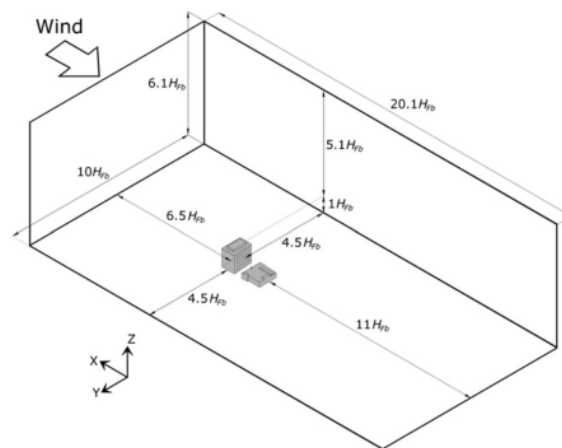
Figure 4.14 - Dimensions of the domain grid [Kim, 2013]

4.10.3. The Resolution of Computational Grid

Computational grids should be fine enough to provide an adequate resolution of the geometrical and expected flow features. Generally, the greater number of cells lead to the better CFD results, but as the number of cells increases, the calculation time also increases. The maximum number of cells that can be created for the solution depends on the computing resources available. [Kim, 2013].

Grid resolution must be at least 1/10 the building scale within the region that includes the evaluation points around the building of interest. [Kim, 2013] Also a minimum of 10 cells per building side and at least 10 cells per

93

cube root of building volume should be used since the initial grid resolution is recommended. [Kim, 2013]

Some recommendations for choosing grids and grid design: [Kim, 2013].

1) Choose a suitable global topology for the mesh to help satisfy the skewness, aspect and expansion ratios.

2) Computational domain should be chosen to capture all the relevant flow and geometrical features.

3) Use local grid refinement to capture fine geometrical details.

4) Avoid highly skewed cells, warped cells, and non-orthogonal cells near boundaries.

5) Observation is needed for any specific requirements on mesh expansion ratios.

6) Use a finer and more regular mesh in regions of interest with high flow gradients or with large changes.

7) Make use of a grid dependency study to analyze the suitability of the mesh in order to obtain an estimate of the numerical error in the simulation for each class of problem.

Grid stretching should be small in regions of high gradients and the truncation error should be small. A stretching method utilizing the standard stretching functions supplies one with a very simple means to cluster the nodes of the computational grid within the regions of steep gradients without an increase in the total number of grid nodes. The expansion ratio between two consecutive cells should be below 1.3 within regions and a value of 1.2 for the maximum expansion ratio. [Kim, 2013]

The shape of the grids can be categorized as either an unstructured or structured mesh. According to Franke et al (2007), hexahedral cells are preferable to tetrahedral cells because hexahedral cells produce smaller truncation errors and provide better iterative convergence. When using tetrahedral cells, prismatic cells must be used at the wall with tetrahedral

cells away from the wall since the grid lines are perpendicular to the wall. It is also important to arrange that prismatic cells parallel to walls and ground surfaces when using tetrahedral cells. [Kimn, 2013].

4.10.4. Boundary Conditions

The surroundings have a significant effect on the computational domain; therefore the boundary conditions should be determined in order to avoid negative effects to surface due to the surroundings. [Kim, 2013].

It is very important to choose proper boundary conditions because they decide to a large extent the solution in the computational domain. [Kim, 2013].

There are two types of boundary conditions: The Dirichlet condition defines the distribution of a physical quantity over the boundary at a given time step and the Neumann condition specifies the distribution of its first derivative. Casey and Wintergerte have proposed a guideline for selecting boundary conditions. [Kim, 2013].

1-Ensure that proper conditions are available for the case being considered.

2- Check whether the CFD code allows inflow for open boundary conditions.

The mean velocity profile and information about the turbulence quantities are needed when setting the inflow boundary conditions in order to create an equilibrium boundary layer. This means that the velocity profile can be achieved from either the logarithmic profile corresponding to the upwind terrain via the roughness length z_o or from the profiles of the wind tunnel simulations. In the case of steady RANS simulations, the mean velocity profile and information about the turbulence quantities is needed and suitable profiles can be achieved from the assumption of an equilibrium boundary layer. When wind tunnel data for turbulent kinetic energy is available, it should be employed to describe the profile. Franke et al. also suggest that an analysis should be conducted to ascertain whether the chosen grid and boundary conditions are consistent before simulating flow over obstacles. In this instance, an empty computational domain with the same grid and periodic

boundary conditions can be used for the analysis to obtain consistent profiles that match the velocity measurements at the meteorological station. [Kim, 2013].

For the top and lateral boundaries, the viscid wall can be used if these boundaries do not affect the calculated results around the target building [Kimn, 2013]. The symmetry conditions for these boundaries can also be used.

For the outflow boundary conditions, open boundary conditions are used at the boundary behind the obstacles and outflow or constant static pressure. The conditions are frequently used for the open boundary conditions in CFD codes. In addition, this outflow boundary condition has to be far enough away from the region of interest. [Kim, 2013]

for inlet, the mean velocity profile prescribed by logarithmic law and information about turbulence quantities is required. For the outlet boundary condition, constant pressure conditions are recommended. Symmetry conditions are used for the top and lateral boundaries. [Kim, 2013]

4.10.5. Turbulence Models

Turbulent flow in urban or industrial environments is modeled by the Navier-Stokes equations. It is important to decide whether the application needs a steady or an unsteady treatment. There is no universally valid general turbulence model for all classes of flows, so validation and calibration with experimental data is necessary for all applications. It is also necessary to examine the effect and sensitivity of results to the turbulence model which requires changing the models being used. A thorough review of the published literature regarding the known weakness of the model is needed in order to use a particular turbulence model. [Kim, 2013]

There are two approaches for reproducing wind flow around a building:

1- A steady- state RANS simulation which is used for many simulations and it provides an average flow field. In case of wind tunnel experiments, it can

96

represent the reality of wind tunnel, but it is difficult to reproduce the separation and reverse flow at the roof top of a building. [Franke, et al, 2007]

2- LES approach which is used to predict and represent wind loads on buildings can provide more information about the flow field than the RANS approach. [Franke, et al, 2007]

4.10.6. Convergence criteria

Generally, CFD codes employ iterative methods to calculate the algebraic system of equations and the termination criterion is subject to the residuals of the corresponding equations. [Franke, et al, 2007] The termination criterion must move from one to zero, the residuals are scaled iteratively and the termination criterion of 0.001 for industrial application is recommended. Another alternative approach suggests checking the solution directly using different convergence criteria such as the relaxation coefficient or periodic fluctuations than to simply use a yardstick of four orders of magnitude for the residuals. Thus, the scaled residuals should be terminated at 0.001. [Franke, et al, 2007]

4.11. Ten Iterative Steps Approach

The output of CFD simulations depend on computational parameters chosen by the users. The best practice guidelines have been proposed to reduce errors or uncertainties caused by the users. The best practice guidelines deal mainly with target variables, the extent of model, time intervals, the choice of turbulence models, boundary conditions, convergence criteria and initialization. However, these guidelines alone are not sufficient to ensure the credibility, acceptance and impact of CFD results.

The iterative steps approach, originally introduced in a position paper by [Jackeman et al 2006], has proposed constructing a comprehensive framework for CFD models. The approach develops purposeful, credible models from data and prior knowledge, working closely with the end-users and with every stage open to critical review and revision [Kim, 2013]. The paper was

intended to provide more general guidelines covering a wide range of model types, This includes also empirical, data-based, statistical models, specific theory-based or process-based models such as CFD, conceptual models based on assumed structural similarities to the system, agent-based, and rule-based models.

4.12. Characteristics of the Ten Iterative Steps Approach

Figure 4.15, shows the ten-iterative steps approach which is composed of a series of iterative steps including trial and error for pursuing good practice in model development and application. [Kim, 2013]

4.12.1. Definition of Modeling Purposes

In general, it is often difficult to be clear about the precise purpose of any modeling process. Different stakeholders will have different degrees of interest in the possible purposes of the end product. Thus, the modeler should determine the purposes and priorities of the model that is to be produced because these choices can have a profound effect on the model development, particularly during the later stages. [Jackeman, et al, 2006]

4.12.2. Specification of Modeling Context: Scope and Resources

The following issues should be identified in this step:

The specific questions, the interest groups, the outputs required, the forcing variables (drivers), the accuracy expected or hoped for, temporal and spatial scope, scale and resolution, the time frame to complete the model as fixed, the effort and resources available for modeling and operating the model, and the flexibility. [Jackeman, et al, 2006].

4.12.3. Conceptualization of the System, Specification of Data and other Prior Knowledge

Conceptualization is defined as the basic premises governing working of the system being modeled, it might be conducive to thinking such as an influence diagram, linguistic model, block diagram or bond graph, showing how model

98

drivers are linked to internal (state) variables and outputs (observed responses). The data, prior knowledge and assumptions about process are defined in this step. [Jackeman, et al, 2006]

4.12.4. Selection of Model Features and Families
Model feature are required in any modeling approach. These features are represented in types of variables covered, the nature of treatment, which family the model belongs to, and model structure. [Jackeman, et al, 2006]

4.12.5. Choice of how Model Structure and Parameter Values are to be found

This step discusses choice of methods for finding model structure and parameters. The relations between the variables in the model are based on prior science and theoretical knowledge. The simple structure of a model is adequate for a specific purpose. [Jackeman, et al, 2006]

4.12.6. Choice of Estimation Performance Criteria and Technique

This step, discuss the criteria and techniques of the modeling. Parameter estimation criteria reflect the desired properties of the estimates. The parameter estimation techniques are:

- Computationally as simple as possible to minimize the chance of coding error

- Robust in the face of outliers and deviations from assumptions (e.g. about noise distribution)

- As close to statistically efficient as feasible (as reflected by the amount of data required for the estimates to converge);

- Numerically well-conditioned and reliable in finding the optimum

- Able to quantify uncertainty in the results (not at all easy, as the underlying theory is likely to be dubious when the uncertainty is large); and

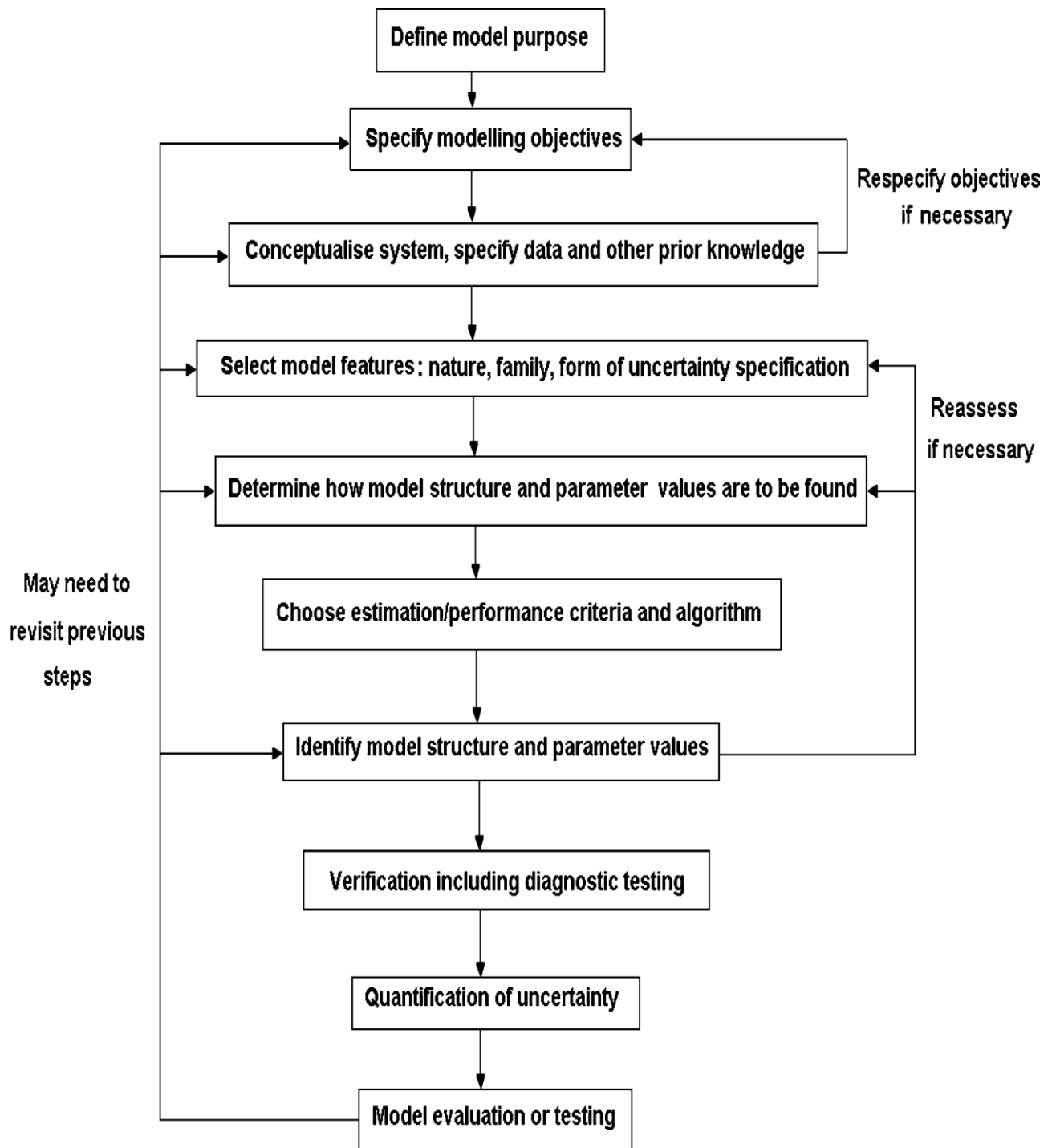- Accompanied by a test for over-parameterization. [Jackeman, et al, 2006]

```
                    ┌─────────────────────┐
                    │ Define model purpose │
                    └─────────────────────┘
                              │
                              ▼
             ┌────────────────────────────────┐
        ┌───▶│   Specify modelling objectives │◀───┐
        │    └────────────────────────────────┘    │  Respecify objectives
        │                    │                      │     if necessary
        │                    ▼                       │
        │   ┌──────────────────────────────────────────────┐
        │──▶│ Conceptualise system, specify data and other  │──┘
        │   │              prior knowledge                   │
        │   └──────────────────────────────────────────────┘
        │                    │
        │                    ▼
        │   ┌──────────────────────────────────────────────────┐
        │──▶│ Select model features: nature, family, form of     │◀──┐
        │   │           uncertainty specification                │   │
        │   └──────────────────────────────────────────────────┘   │  Reassess
        │                    │                                       │ if necessary
        │                    ▼                                       │
        │   ┌──────────────────────────────────────────────────┐   │
        │──▶│ Determine how model structure and parameter values │◀──┤
        │   │                 are to be found                    │   │
        │   └──────────────────────────────────────────────────┘   │
        │                    │                                       │
        │                    ▼                                       │
        │   ┌──────────────────────────────────────────────────┐   │
        │   │ Choose estimation/performance criteria and         │   │
        │   │                  algorithm                         │   │
        │   └──────────────────────────────────────────────────┘   │
        │                    │                                       │
        │                    ▼                                       │
        │   ┌──────────────────────────────────────────────────┐   │
        │──▶│   Identify model structure and parameter values    │───┘
        │   └──────────────────────────────────────────────────┘
        │                    │
        │                    ▼
        │   ┌──────────────────────────────────────────────────┐
        │   │   Verification including diagnostic testing        │
        │   └──────────────────────────────────────────────────┘
        │                    │
        │                    ▼
        │   ┌──────────────────────────────────────────────────┐
        │   │        Quantification of uncertainty               │
        │   └──────────────────────────────────────────────────┘
        │                    │
        │                    ▼
        │   ┌──────────────────────────────────────────────────┐
        └───│        Model evaluation or testing                 │
            └──────────────────────────────────────────────────┘
```

May need to revisit previous steps

Fig.4-15. Shows iterative relationship between model building steps [Jackeman, et al, 2006]

4.12.7. Identification of Model Structure and Parameters

This step aims to find a suitable model structure and parameter values. It is ideally involves hypothesis testing of alternative model structures. The

100

complexity of interactions proposed for the model may be increased or reduced according to the results of model testing. [Jackeman, et al, 2006]

4.12.8. Conditional Verification Including Diagnostic Checking

This step is very important because it checks the model's credibility, and builds the client's confidence in the model. Kim [2013] has explained the process involved as follows:

The model must be verified and tested to ensure its robust sufficiently. It is also necessary to verify that the interactions and outcomes given the objectives and the prior knowledge of the model are feasible and defensible.[Kim, 2013]

This step should include as wide a range of quantitative and qualitative criteria as circumstances allow. For quantitative verification this could encompass a wide range of criteria including goodness of fit and tests on residuals or errors, particularly for relatively simple empirical models. In the case of qualitative verification, this generally involves knowledgeable data suppliers or model users who are not modelers.[Kin, 2013]

4.12.9. Quantification of uncertainty

Uncertainty must be considered in developing any model, but it is particularly important, and usually difficult to deal with in large, integrated models (Kimn, 2013). Uncertainty should also be considered in the context of the purposes of the model. [Kin, 2013]

4.12.10. Model Evaluation or testing (other models, algorithms, comparisons with alternatives)

The following questions are important for evaluating the model, [Kim, 2013], whereas the model must be evaluated in the light of its specific:

- How well does the model reproduce an independent data set?

- How well does the model perform under unusual conditions?

- Is the complex model better than a simpler one?

- Can the model be used to improve understanding of underlying system function?

- Finally, and most importantly, does the model help answer questions about the system function and can it be used to make predictions about the future? .[Kim, 2013].

4.13. Conclusion

1- In this chapter, different methods and tools used for wind analysis has been discussed, starting from wind tunnels, the oldest tool used in the wind studies and analysis, and ending with CFD, the latest wind analysis tool. Wind tunnels tool is less used because it is highly costly compared with CFD which has become widely used in different scientific disciplines. 2- This research has adopted CFD as a tool to analyze the air movement pattern around the buildings because it is easy to use, less costly compared with the wind tunnels, available and has accurate analysis and reliable results.

3-Compared with the traditional wind tunnels, CFD has many advantages. CFD can generate full-scale simulations as opposed to scale models of many physical simulations, it is also provides more extensive data than can be measured in the lab and its results can be visualized clearly and in great details. [Kim, 2013]

4-There are six best guidelines for CFD practice. These guides have been developed to support the proper use of CFD simulations, and to reduce the errors or uncertainties. Now, many researchers use these guidelines. [Kim, 2013]. The best guidelines involve the size of computational domain, geometrical representation, and grid generation, the selection of boundaries and turbulence models, and convergence criteria. These guidelines can be summarized in the following:

- Details in the geometrical representation of buildings are required. These details involve reproducing the area of a radius 1-2H from the building of interest as accurately as possible, representing at least one additional street block around the central region of interest, and representing buildings located away from the region of interest by simple blocks. (Kimn, 2013)

102

- The proper size of computational domain should be 5H for the top and 10H for the lateral boundary in a single building and urban areas with multiple buildings, where H is the height of the tallest building. In addition, 5H and 15H can be used for the upstream and the downstream length respectively. [Kim, 2013].

- The appropriate type of computational grid and grid resolution should be hexahedral and tetrahedral mesh for CFD applications of wind flow around buildings. In addition At least 10 cells per cube root of building volume are recommended for the initial grid resolution.

- Regarding the appropriate boundary conditions, the mean velocity profile prescribed by logarithmic law as well as information about turbulence quantities are required for inlet boundary condition. Constant pressure conditions are recommended for outlet boundary condition; In addition, a symmetry conditions are used for the top and lateral boundaries. [Kim, 2013].

-For the proper turbulence model, both LES model and turbulence models based on the RANS approach can be used in order to reproduce wind flow around a building. (Kimn, 2013)

- For the proper range for the iterative convergence criteria, the scaled residuals are terminated at 0.001. [Kim, 2013].

5-CFD simulation is adopted in this study to analyse wind pattern around the buildings in urban residential areas and ensure accessibility of the air to all buildings. According to studies on wind analysis, it can be concluded that CFD is efficient and precise tool for air movement and wind analysis. Unlike wind tunnels, CFD helps arrive at accurate results. The CFD is well-known worldwide and reliable design tool; therefore it can be used in different scientific disciplines.