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: Summary & Conclusion 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 & HNEI

December 2014

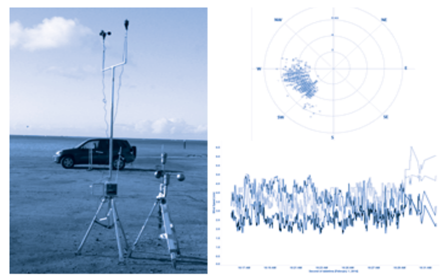
HNEI
Hawaiʻi Natural Energy Institute
University of Hawaiʻi at Mānoa

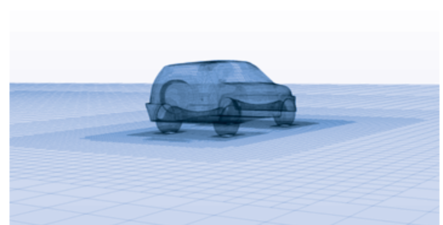
Project Phase 1- 7.A

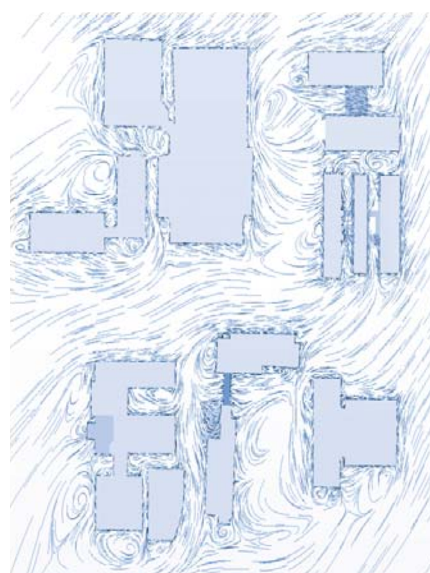
SUMMARY & CONCLUSION OF PROJECT PHASE 1 - EXTERNAL CFD

5

Dec 5, 2014







Prepared by:
Manfred J. Zapka, PhD, PE (Editor)
Tuan Tran, D.Arch
Eileen Peppard, M. Sc.
A. James Maskrey, MEP, MBA, Project Manager
Stephen Meder, D.Arch, Director

ERDL HNEI

Contract # N000-14-13-1-0463

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

Project Phase 1 – 7.A –

Task 7.a.4: Project summary Report and Presentation for Phase 7.a

Project Deliverable No. 5:

Summary and Conclusion of Project Phase 1 – External CFD

Prepared for Hawaii Natural Energy Institute

in support of

Contract #N000-14-13-1-0463

December 5, 2014

Prepared by:

Manfred J. Zapka, PhD, PE (Editor) [(1)]

Contributors:

Tuan Tran, D.Arch [(2)]

Eileen Peppard, M. Sc. [(3)]

James Maskrey, MEP, MBA, Project Manager [(4)]

Stephen Meder, D.Arch, Director [(2)]

(1) Sustainable Design & Consulting LLC, Honolulu, Hawaii
(2) Environmental Research and Design Laboratory (ERDL), School of Architecture, University of Hawaii
(3) University of Hawaii Sea Grant College Program
(4) Hawaii Natural Energy Institute Honolulu, Hawaii

ACKNOWLEDGEMENTS

The authors would like to thank the staff of the Environmental Design & Research Laboratory (ERDL) for their assistance in carrying out parts of this research study.

The authors especially acknowledge the dedicated work and valuable input by research assistants Christian Damo and Reed Shinsato, and as well as ERDL Post-doctoral fellow Aarthi Padmanabhan, D.Arch.

The valuable contribution of former ERDL Post-doctoral fellow Sanphawat Jatupatwarangkul, D.Arch, is especially acknowledged. Sanphawat was on the CFD team through the completion of the first two report of Project Phase 1-7.A. The ERDL team wishes Sanphawat all the best for his future academic and professional endeavors.

ACRONYMS & UNITS

ACRONYMS

3D	Three Dimensional
3D-Cad	Three Dimensional Computer Aided Design
ABL	Atmospheric Boundary Layer
BPG	Best Practice Guidelines
CAD	Computer Aided Design
Cp	Pressure Coefficient
DNS	Direct Numerical Simulations
CFD	Computational Fluid Dynamics
ERDL	Environmental Research and Design Laboratory
FDM	Finite Difference Method
FEM	Finite Element Method
FVM	Finite Volume Method
GUI	Graphical User Interface
HNEI	Hawaii Natural Energy Institute
HVAC	Heating, Ventilating, and Air Conditioning
ID	Identification Index of Descriptor
LES	Large Eddy Simulation
PDE	Partial Differential Equation
RANS	Reynolds-average Navier-Stokes
V	Velocity
WDR	Wind-driven Rain
WS	Weather Station

UNITS:

DC	Direct Current
ft	Foot or feet
hz	Hertz
m	Meter
m/s	Meter per second
mph	Miles per Hour
Pa	Pascal
sqft	Square Feet

TABLE OF CONTENTS

EXECUTIVE SUMMARY

The research work presented in this report is part of the ongoing HNEI sponsored research program about the use of Computational Fluid Dynamics (CFD) in building analysis and design, with an emphasis on naturally ventilated buildings. The research program has three parts:

Part 1. Use of **external CFD** simulations to model wind movements around buildings and how they affect natural ventilation,

Part 2. Use of **internal CFD** simulations to study air movement through a building and identify measures to improve naturally ventilation performance and

Part 3. Use of CFD simulations to study the effectiveness of measures that increase comfort (**CFD comfort**) in naturally ventilated spaces. This report presents the last external CFD investigation of Part 1 of the HNEI sponsored research program.

<u>This report is a summary of the project work carried out for **Part 1** of this research program.</u> The content of this report summarizes the main findings presented in the deliverables of Part 1.

The project work of Part 1 “External CFD Applications in Building Design” was submitted in four deliverables (six draft and final reports):

<u>Deliverable 1</u>:	Task 7.a.1 - Literature review for external CFD
<u>Deliverable 2.1 and 2.2</u>:	Task 7.a.2 Draft and Final report to establish an external CFD work process
<u>Deliverable 3</u>:	Task 7.a.3: Report to develop and calibrate a data verification process for external CFD simulations
<u>Deliverable 4.1 and 4.2</u>:	Task 7.a.4: Draft and Final report on external CFD simulation & field verification for selected building

The main objective of Part 1 were:

- Evaluate the existing application literature of external CFD and identify what topics relate to the intended application of external CFD in Hawaii.
- Develop application skills at the Environmental Research and Design Laboratory (ERDL) for advanced Computational Fluid Dynamics (CFD) analysis and related applied research in ventilation technology for buildings.
- Develop application skills at ERDL for the measurement of wind direction, velocity and pressures around buildings.

- Validation of external CFD simulation results by performing field measurements and comparing theoretical CFD predictions with quantified wind induced phenomena.
- Determine skill sets that can be used by the local building industry to assess wind movement around buildings in order to derive improved building designs and performance.

The CFD project team carried out the work presented in the summary report within a period of 13 months, starting in August 2013 and completing the project work of Phase 1 in September 2014.

The present summary report describes the applied research work that has been carried out for Part 1 of this research program. The project work included modeling of air movement around buildings with Computational Fluid Dynamics (CFD) and validating the theoretical CFD results through full-scale measurements in the field.

SECTION 1 - OVERVIEW AND RESEARCH APPROACH

The overreaching objective of part 1 of the research program was the development of skills and procedures at the Environmental Research and Design Laboratory (ERDL) to engage in advanced CFD simulations of wind movements around building and their effects on natural ventilation performance of buildings.

The project work of Part 1 of the research program was carried out in four project tasks 1.1 through 1.4, illustrated in Figure 1. These project tasks are briefly described below and are summarized in Sections 2 through 5 of this report.

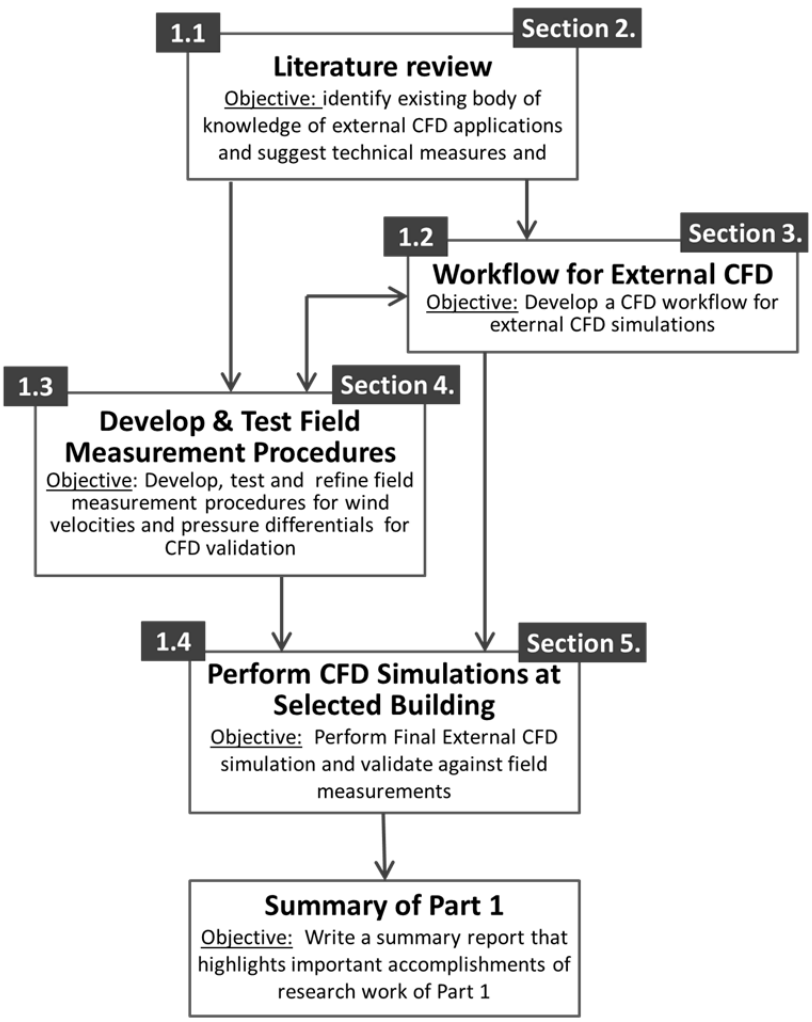


Figure 1: Flow chart of the project work for Part 1 of the research program

Project Task Group 1.1 - Literature Review: The literature review provided an overview of an identified body literature about external CFD applications. The literature reviewed comprised approximately 80 technical and scientific publications, as well as numerous software documentation furnished by CFD vendors. The focus of the literature review was CFD applications of wind movements around buildings. These wind patterns create driving forces for natural ventilation and determine pedestrian comfort. Typically, CFD applications in wind engineering are concerned with high wind events that can negatively affect structures and pedestrian safety. The study of wind movement around building relative to natural ventilation is a relative small field and there is a rather small international group of researchers and CFD practitioners from whom to obtain published input. The literature review is summarized in Section 2 of this report.

Project Task Group 1.2 – Workflow for External CFD: The project specific workflow for external CFD simulations was developed after completion of the literature review. Pertinent CFD techniques, settings and best practices were used to develop an external CFD workflow that was unique for this research project. Work on the development of the external CFD workflow occurred in parallel with the development and testing of field measurement procedures (Project Task 1.3). The CFD simulations procedures developed were validated against the "shake-down" tests of Project Task 1.3. Results from the validation tests were used to successively optimize the external CFD workflow.

Project Task Group 1.3 – Develop and Test Field Measurements Procedures: Field measurement procedures for wind velocity distribution around structures and pressure differentials were developed following guidelines and suggestions about techniques and instrumentation identified in the literature review. However, most of the instrumentation and field test procedures for wind engineering is focused on higher wind speeds and higher wind forces, such as assessing high wind effects on structures and wind measurements in wind tunnels. The test procedure for this project had to consider instrumentation constraints and new approaches to account for relatively small pressure differentials. Project Task 1.3 provided test results for early validation of the CFD workflow for external wind movement.

Project Task Group 1.4 – CFD Simulations & Validation at Selected Building: After completing and validating the workflow for external CFD, simulations were conducted at the selected building on the University of Hawaii Manoa campus. The CFD simulation results were validated against measurements of wind velocities around and pressure differentials across the selected building. This Project Task 1.4 completed the work on Part 1 of the research project.

The project work of Part 1 "External CFD Applications" was performed in four project task groups 1.1 through 1.4.; project task groups 1.2 and 1.3 were performed concurrently

SECTION 2 - LITERATURE REVIEW

This review of scientific and technical literature was 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 buildings
- Wind induced natural ventilation performance for buildings
- Convective heat transfer processes from exterior building envelope

The literature review of scientific and technical literature was conducted in accordance with three identified areas of concern, which are as follows:

1. <u>Fundamentals of Wind Movement around Buildings and Urban Wind Comfort</u>: Review of basic processes of wind movements around buildings, including but not limited to topics addressing atmospheric boundary layers, mechanism of wind induced pressures, effects of building geometry on wind regimes, basics of urban wind comfort.

2. <u>Assessment Methods of Wind Movements around Buildings:</u> Review of assessment methods of wind movement around building, including an assessment of the effectiveness of CFD compared with other prediction methods

3. <u>External CFD – Numerical Assessment of Air Movement around Buildings:</u> Review of the state of the art and trends in external CFD applications pertaining to the built environment, including but not limited to basic considerations of CFD calculation processes , verification and validation, tasks required for CFD pre-processing, solver and post processing.

Three areas of interest for literature review:

1. Fundamentals of Wind Movement around Buildings and Urban Wind Comfort
2. Assessment Methods of Air Movements around Buildings
3. External CFD – Numerical Assessment of Air Movement around Buildings

2.1 Fundamentals of Wind Movement around Buildings and Urban Wind Comfort

The study of wind movement phenomena around buildings that relates to natural ventilation and urban wind occupant comfort represents a relatively small segment of wind engineering.

Flow patterns around buildings: The external wind movement generates site-specific pressure and wind velocity patterns around the building envelope. The pressure and velocity distribution is dependent on the wind speed, wind approach direction and the shape of the building. Flow patterns around simple geometries can be predicted using existing design guidelines. For more complicated building patterns and especially with upwind and downwind flow obstruction present, wind velocity and pressure distribution around the building requires analytical methods and/or wind tunnel test.

Figure 2.1 illustrates a typical pattern of wind movements around a simple rectangular building. 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. On the upwind wall, surface flow patterns are largely influenced by approach wind characteristics. Figure 2.1 shows that the mean speed of wind approaching the building increases with height H above the ground (e.g. ground effect affecting the wind speed through friction and turbulence). Higher wind speeds 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).

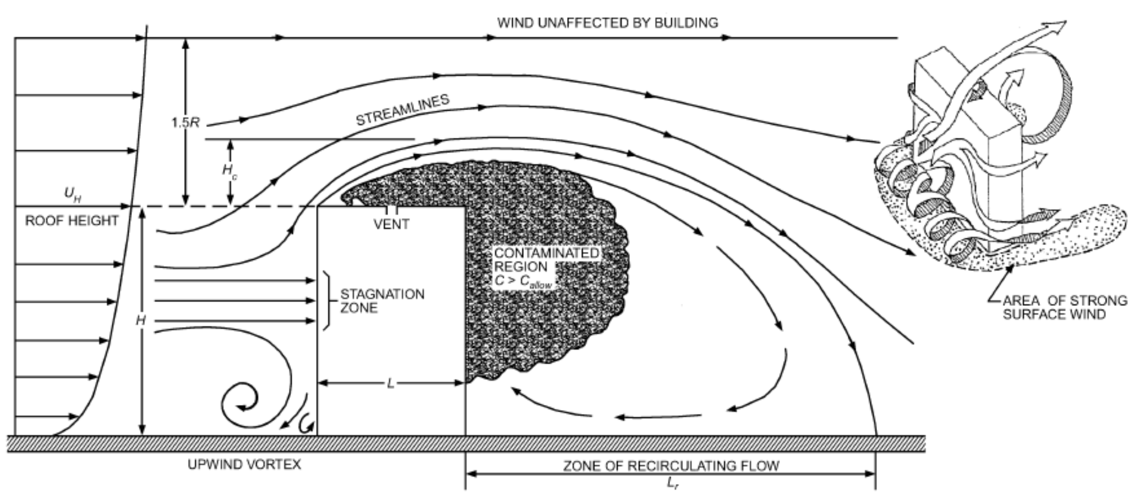


Figure 2.1: Typical pattern of wind movements around a rectangular building - vertical section (Figure presented in Project Report 1, Literature Review of External CFD)

The wind movement around buildings can cause discomfort or even safety concerns through high wind velocities and/or concentrated wind patterns. Some examples of wind flow patterns are presented in the following:

Downwash: 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 is referred to as downwash. Figure 2.2 illustrates downwash conditions. These wind conditions can have a significant adverse effect on pedestrian comfort around the buildings.

Effect of canopy and arcades: Figure 2.3 (a) illustrates the effect of a large canopy at the windward side of the building which interrupts the downwash. This flow interruption may protect the entrances and sidewalk area by deflecting the downwash at a higher story level. Figure 2.3 (b) illustrates the typical wind patterns associated with arcades or thoroughfare openings from one side of the building to the other. These openings connect positive pressure regions on the windward side with negative pressure regions on the leeward side. Strong wind flow can be generated through these building openings.

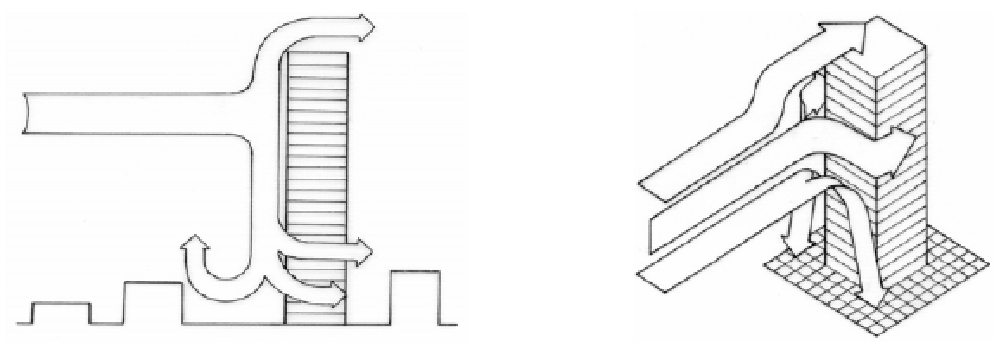
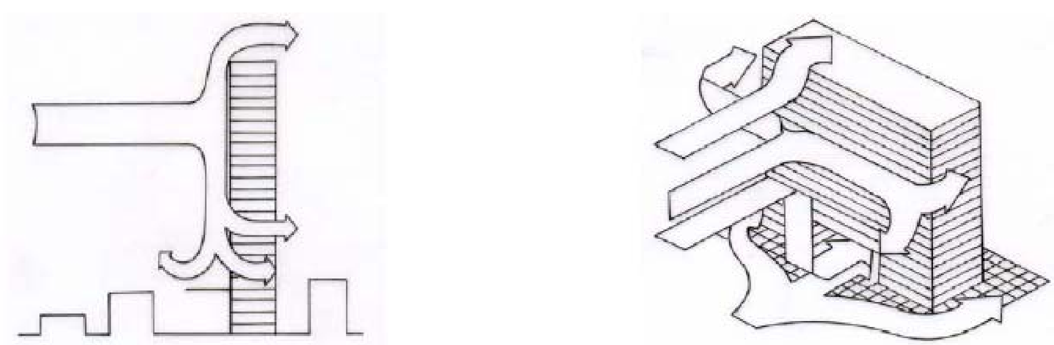


Figure 2.2: Flow patterns around buildings, typical downwash conditions (Figure presented in Project Report 1, Literature of External CFD)



(a) effect of canopy to deflect downwash

(b) wind flow through arcades

Figure 2.3: Flow patterns around buildings, effects of canopies and arcades (Figure presented in Project Report 1, Literature Review of External CFD)

Wind patterns around buildings can have a significant effect on the comfort of occupants and might even have negative effects on the safety of occupants.

Wind Induced Pressure Differentials on the Building Envelope

Pressures created on the envelope of a building by high winds can cause damages or even failures of the building structure. Much of the design recommendation of wind engineering is concerned with high wind events. The present literature study considers only pressure distributions around a building that derive in natural ventilation. This means that low and medium wind events are considered that create pressure differentials around a building and therefore create the driving forces for natural ventilation. The type of architecture significantly affects pressure distribution around a building and therefore affects the building's natural ventilation performance. There are two fundamentally different types of natural driving forces for natural ventilation available, thermal buoyancy and wind impinging on the building envelope:

Thermal buoyancy driven ventilation (also referred to as **stack ventilation**) occurs when there is a density difference between the internal and external air, which in turn is caused by temperature differences between the inside and outside. Thermal buoyancy driven ventilation 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. Figure 2.4 shows the basic premise of stack ventilation.

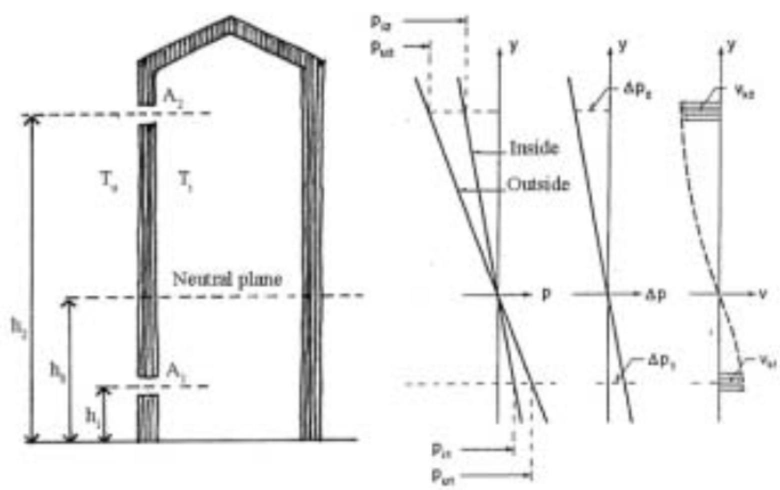
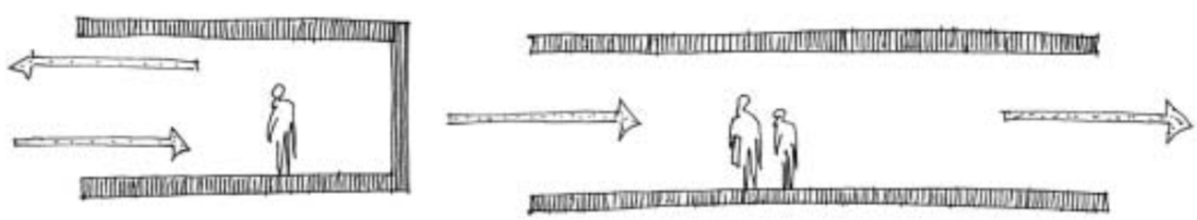


Figure 2.4: Thermal buoyancy in a space with two openings. (Figure presented in Project Report 1, Literature of External CFD)

Wind driven ventilation occurs as a result of pressures gradients created on the building envelope by the wind. These pressure differences that act at locations of openings in the building envelop provide the force to drive air into and out of the building through the openings in the building. There are two basic types of wind driven ventilation, single sided ventilation and cross ventilation. Figure 2.5 illustrates these two basic ventilation patterns.

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.

Cross-ventilation occurs when air flows between two sides of a building 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 locations of higher pressures to locations of lower pressures, which means from the windward side to the leeward side.



(a) single sided ventilation (b) cross ventilation

Figure 2.5: Two basic types of wind induced natural ventilation (Figure presented in Project Report 1, Literature Review of External CFD)

Thermal buoyancy and wind driving forces 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 when there are large temperature differentials, e.g. a day with low outside temperatures with practically no wind, whereas pressure differentials created by wind will typically be the dominating driving force on a windy hot day. For Hawaii this implies that pressure differentials created by wind impinging on the building typically represent the dominating driving forces for natural ventilation.

For Hawaii climate pressure differentials created by wind force on the building represent the dominating driving forces for natural ventilation.

Tabulated pressure distribution around a building: The literature provides predictions of the pressure distribution on the building envelope resulting from of external wind movement values around simple building geometries. For buildings of more complex geometry and including wind pattern disturbances from adjacent flow obstructions, reliable predictions of pressure distribution on the building envelope can only be determined by wind tunnel experiments or with the help of numerical simulation, e.g. Computational Fluid Dynamics (CFD).

A readily available assessment tool for the pressure distribution on a building envelope is the so-called wind pressure coefficient. The wind pressure coefficient is a proportionality factor which is applied to the local outdoor atmospheric pressure at the same elevation to obtain the locally active pressure on the building envelope. This means that a wind pressure coefficient Cp smaller than one results in a pressure lower than the local outdoor atmospheric pressure and vice versa, a Cp larger than one results in a higher pressure. Figures 2.6 and 2.7 illustrate the use of the coefficient in the determination of local pressure distribution around a tall and a low-rise building, respectively.

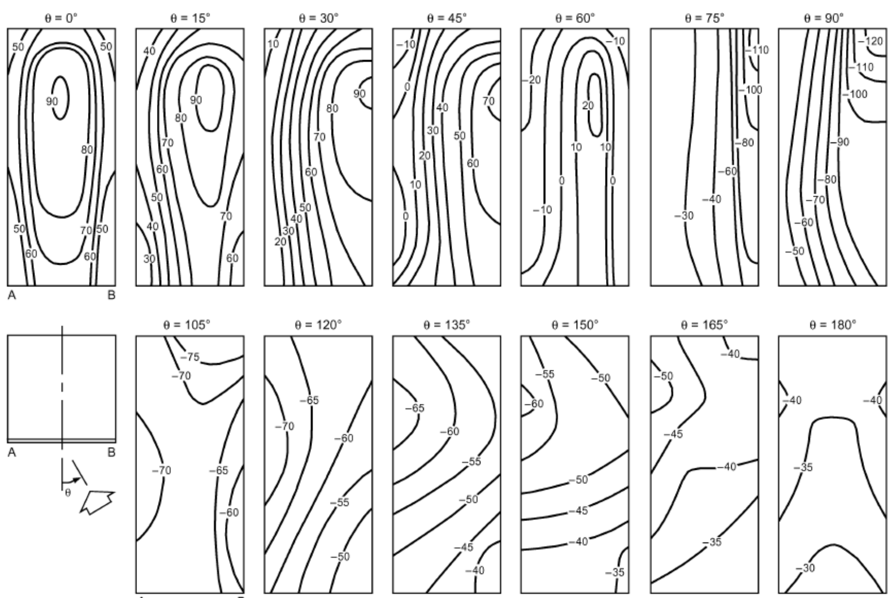


Figure 2.6: Local Pressure Coefficients (Cp × 100) for Tall Building with Varying Wind (Figure presented in Project Report 1, Literature Review of External CFD)

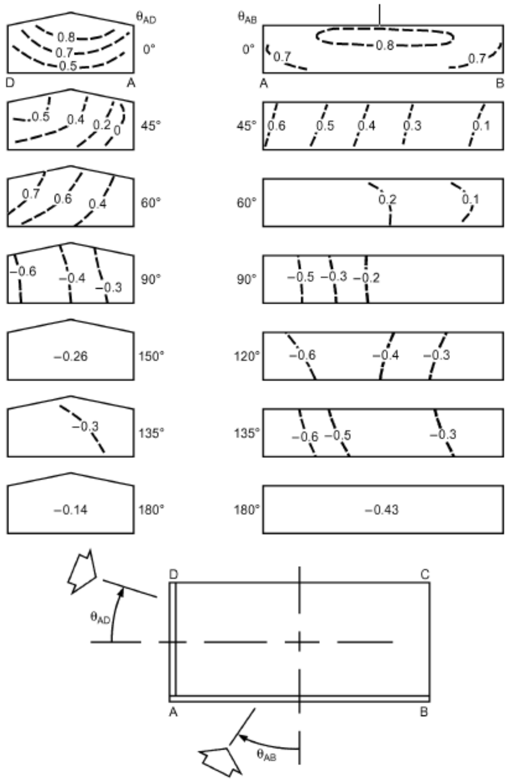


Figure 2.7: Local Pressure Coefficients Cp for Low-rise buildings with Varying Wind (Figure presented in Project Report 1, Literature Review of External CFD)

Tabulated values for pressure distribution around buildings can reasonably predict wind induced pressures around simple building geometries.

Boundary layer adjacent to the ground: Free flow wind velocities are affected by friction and other resisting forces near the ground. As a consequence the vertical wind velocity profile shows a gradual increase of velocities with higher elevations. The shape of the vertical distribution function is dependent on the terrain, which means the amount of friction the ground exerts on the wind stream. Figure 2.8 shows the mean wind velocity profiles for different terrains. It can be seen that over flat and unobstructed ground the wind velocity increases more rapidly than for ground with significant obstructions, such as buildings.

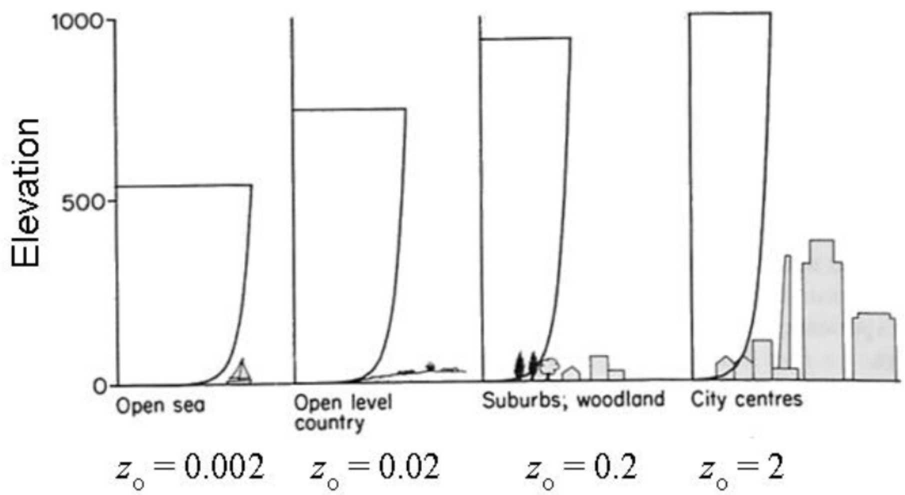


Figure 2.8: Mean wind profiles for different terrains Wind (Figure presented in Project Report 1, Literature Review of External CFD)

The elevated wind speeds at higher elevations have a significant effect on the wind patterns around the building. They control the effectiveness of natural ventilation at different heights of the building and vertical movement around tall buildings.

2.2 Assessment Methods of Air Movements Around Buildings

Assessment and prediction methods for wind pressure coefficient Cp around buildings are categorized primary and secondary sources of Cp. Primary sources include full scale measurements, reduced-scale measurements in wind tunnels and Computational Fluid Dynamics (CFD) simulations. Secondary sources include assessment based on analytical models. Predicted values for Cp of different source methods, and even for observations of the same source methods can vary significantly. The uncertainty of predicted Cp at different locations on the building envelope represents a design challenge for naturally ventilated buildings since knowledge of the pressure differentials between openings in the building envelope is essential for optimized natural ventilation performance. Two important parameters which affect predictions of Cp are the position on the façade and the degree of exposure/sheltering of the building.

As mentioned in the preceding section tabulated values of Cp are available for relatively simple building geometries and ranges of uniform wind approach directions. These secondary sources of Cp should only be used if more accurate primary sources are not available. Primary sources are considered to be the most reliable Cp data sources. The three major primary sources are as follows:

Full-scale measurements: The literature suggests that on-site full-scale measurements at real building facades provide the most reliable source of Cp predictions. For full-scale measurements there is no need to reproduce boundary conditions, uncertainties stemming from scaling down not exist and no physical models are required. However, full-scale measurements are complex and expensive, as well as site specific, and they 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. 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. However, quantifying uncertainty of a measurement represents challenges.

Wind-tunnel measurements: 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. However, wind-tunnel experiments have their own specific challenges. These challenges include, but are not limited to, calibration of the wind tunnel test scenarios, quality assurance procedures, and the know-how of the personnel involved in the test set-up and execution. Figure 2.9 show a comparison of Cp values obtained through wind tunnel tests and measurement at the full-scale prototype.

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. In recent years, the application of CFD has increased significantly due to improvements in computer performance and price reduction in soft- and hardware. There are concerns 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. Areas for improvement of CFD applications in the future are summarized 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.

CFD simulations have become a preferred and cost effective design tool to predict wind movement around buildings. CFD simulations do not substitute wind tunnel tests but rather complement them.

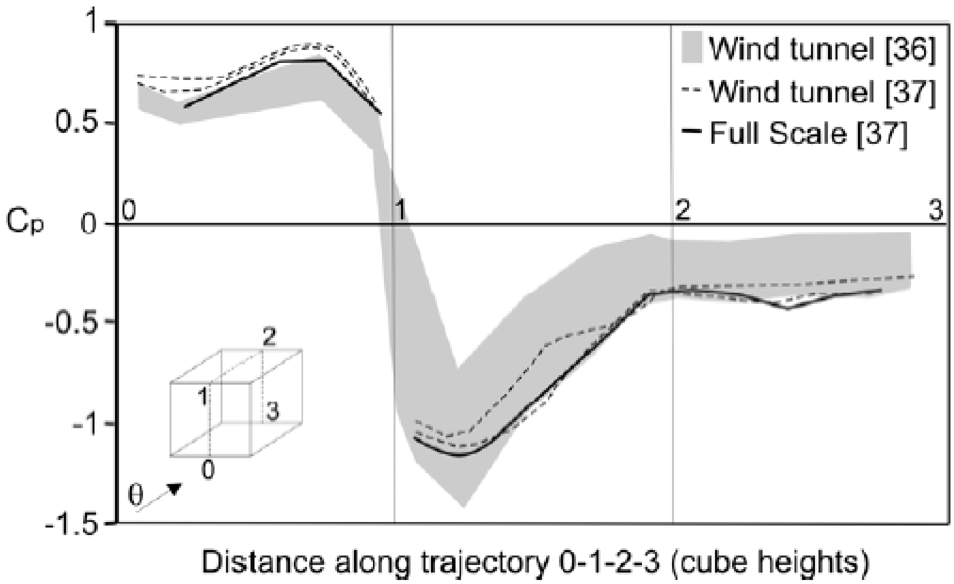


Figure 2.9: Comparison of different wind tunnel experiments with full scale measurements (Figure presented in Project Report 1, Literature Review of External CFD)

2.3 External CFD – Numerical Assessment of Air Movement around Buildings

Solutions Methods of Partial Differential Equations: Computational Fluid Dynamics (CFD) has been increasingly used in building design to determine wind patterns around buildings. The underlying principle of CFD is the use of mathematical models described by partial different equations (PEDs). There are three classical methods to solve to PEDs: the finite difference method (FDM), the finite volume method (FVM) and the finite element method (FEM). Each method has its advantages and disadvantages. Table 2.1 shows the advantages and disadvantages of the three methods.

<u>The finite difference method (FDM)</u> uses the 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, but more terms in the

series cause the complexity and number of discrete points or nodes of the solution to increase significantly. The method can usually only be applied to a regular building shape.

The finite volume method (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). The flux is calculated at the mid-point between discrete nodes in the domain. Hence, the flux between all neighboring nodes in the domain must be assessed to obtain a solution. In a topologically regular mesh (same number of divisions in any one direction), the flux calculation is effective, but in an irregular mesh (as in an automatically generated tetrahedral mesh), this calculation can demand a significant amount of computing.

The finite element method (FEM) is based on the integral approach, where the governing partial differential equations are integrated over an element or volume after having been multiplied by a weight function. The main advantage as well as the main disadvantage of finite element is that it is a mathematical approach where it is difficult to ascertain physical significance on the terms in the algebraic equations. The finite volume method always is dealing with fluxes - not so with finite elements.

Table 2.1: Advantages and disadvantages comparison between FDM, FVM and FEM

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)

Common Applications of External CFD: The literature review suggested four major areas of wind patterns investigations for which CFD is used in the built environment:

(1) Pedestrian wind affected environment around buildings,

(2) Wind-driven rain on building facades,

(3) Convective heat transfer coefficients at exterior building surfaces,

(4) Air pollutant dispersion around buildings.

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. There are , however, 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. Notwithstanding such shortcomings the use of CFD in the built environment is increasing.

The investigation of the external wind movement around buildings with the goal to assess performance level of natural ventilation represents a relatively narrow application of external CFD. However, for Part 1 of this research program the external wind patterns and the resulting pressure distribution represent the main focus of the applied research.

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 quality assurance of the CFD analysis. Best Practice Guidelines (BPG) are effective tools to improve quality and precision of the analysis of wind induced phenomena 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 literature reviewed suggests different sources of errors and uncertainties that are known to occur in numerical simulation results. 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 literature suggests that most general distinction 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

<u>Numerical errors and uncertainties</u> result from numerical solution of the mathematical model:

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

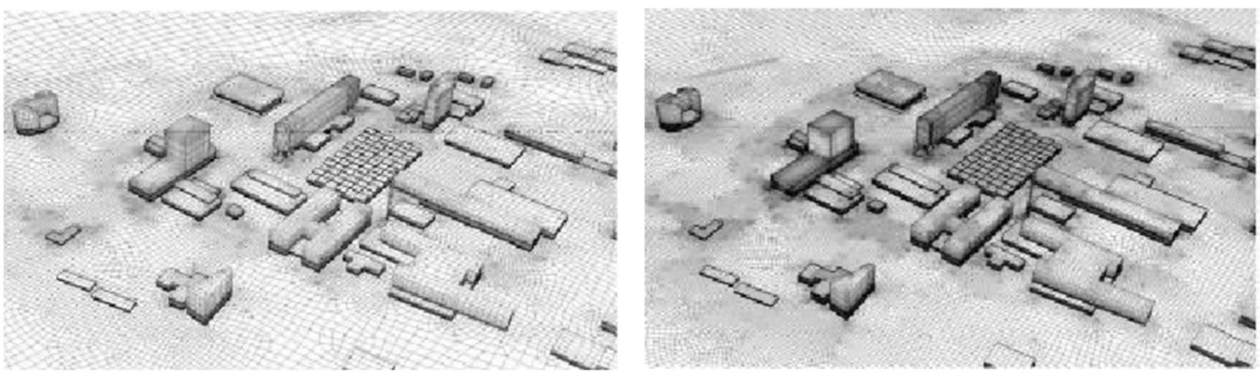
Some major aspects of the best practice guidelines are summarized below:

- The computational geometry should contain all buildings and obstacles that have a significant effect on the wind flow at the location of interest. These objects should be modeled with their main representative shape.
- The computational domain should be large enough to avoid artificial acceleration of the flow caused by the confinement of the boundaries. The size of the computational domain can be based on the height of the tallest building in the urban configuration and/or on the blockage ratio. 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, and sand). The central area of interest should be reproduced with as much detail that is feasible. 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 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 with three different grid resolutions preferred. The main criteria for the selection of the computational gird are the discretization method that is used for solving the basic equations. The computational results depend on the grid that is used to discretize the computational domain. The grid has to be selected 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. (refer to Figure 2.10 so compare a CFD model with different grid resolutions)
- The boundary conditions (for inflow, ground roughness and top and lateral) should be consistent. Iterative convergence should be monitored and a simulation termination criterion should be established. 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.

- While most of the BPGs are consistent with these main elements, some BPCs do not give recommendations on the turbulence model.

Best Practices Guidelines for CFD Applications of wind related phenomena in and around buildings provide good and sound guidance to plan, prepare, perform and interpret CFD simulations.



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

Figure 2.10: Example of grid resolution for a coarse and a fine grid (Figure presented in Project Report 1, Literature Review of External CFD)

Coupled Versus De-coupled Modeling of External Wind Flow

The “coupled” versus “uncoupled” CFD modeling refers to the approach of accounting for the computational domains inside and outside of the building. The literature reviewed used different terms to describe the same CFD domain modeling method in regard to account for the internal and external domain. The term “full-domain” is synonymous with “coupled flow”. The terms “decoupled domain”, “domain decomposition” or “decomposed domain” is synonymous with “un-coupled flow.

Furthermore, the literature at times 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 analysis provide only a snapshot in time of a steady state ventilation scenario, the multi-zone ventilation assessment typically extends over a longer time period, such as an annual prediction of ventilation. In the following discussion “coupled CFD” only refers to the method of accounting for combined internal and external computational domains.

In the coupled approach, there is a single computational geometry and computational domain, which includes both the external and internal air volume of a building (refer to Figure 2.11). 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 assessed in detail, using the appropriate governing equations.

Alternatively, in the decoupled approach, there are two different computational geometries and two different computational domains: one for the external air volume and one for internal air volume of the building. In the decoupled approach, the wind flow simulation in the external CFD is performed 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 this pressure information are subsequently used as boundary conditions for the CFD simulation of the indoor air flow.

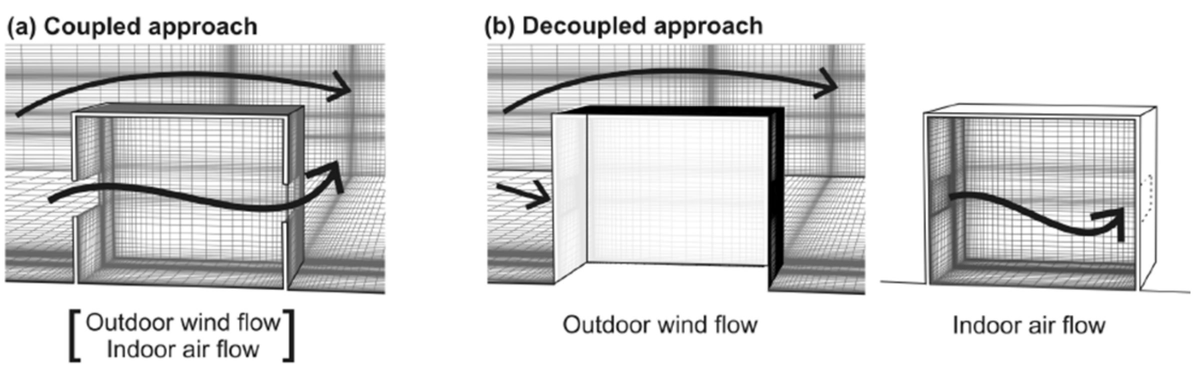


Figure 2.11: (a) Coupled and (b) decoupled approach for analysis of wind-induced cross-ventilation of buildings (Figure presented in Project Report 1, Literature Review of External CFD)

Figure 2.12 shows a comparison of results obtained through coupled and uncoupled domain CFD simulations for the situation depicted in Figure 2.11. Figure 2.12 also shows an actual measurement used for validation of CFD simulations. Figure 2.12 refers to the “Full-domain CFD” and “Domain decomposition CFD” as the “coupled” and “uncoupled” domain method, respectively. As can be seen in the figure the decoupled domain method only defines conditions inside the space depicted in Figure 2.12..

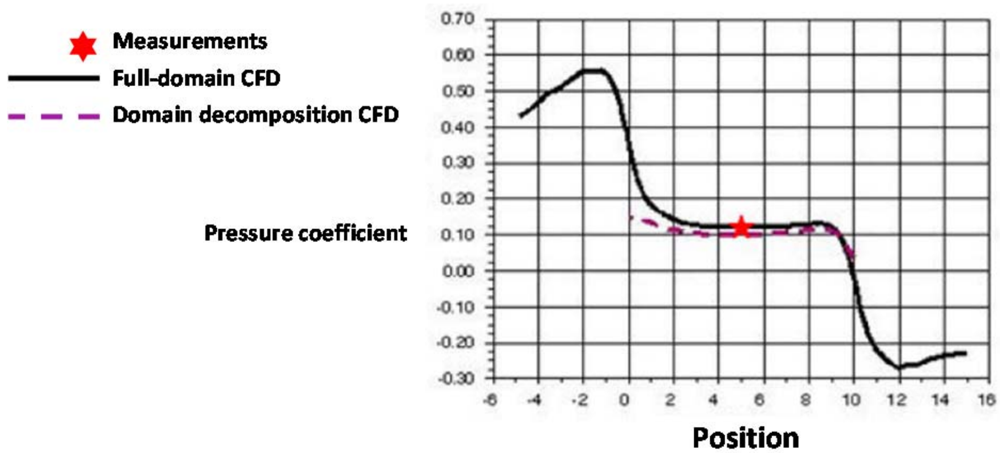


Figure 2.12: Static pressure coefficient, Cp, along line joining inlet and outlet openings for Case C (Figure presented in Project Report 1, Literature Review of External CFD)

While decoupled CFD simulations can provide accurate CFD representations of external wind movement around buildings, coupled CFD simulations can provide increased scope and accuracy of the solution by including air flow through the building. Historically, the main objective of decoupled CFD simulations was the reduced needs for computational resources, as compared to coupled simulations. With increased computational resources coupled domain CFD simulations might be preferable.

2.4 CFD Pre-Processing

A reliable CFD simulation begins with a good preparation of the model and computational domain. The following aspects of CFD pre-processing need to be considered:

Representation of the surroundings: The required levels of detail in CFD modeling of buildings or physical features dependent on their distant to the central area of interest. This means that the central area of interest should be modeled with high details while the buildings or structures away from the central area of interest can be represented as simple blocks.

Computational domain size: The overall size of the computational domain depends on the area represented by the model 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. The reviewed literature suggested typical dimensions of the computation domain should be selected in accordance with the following criteria:

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. The maximum blockage ratio should not be larger than CFD (3%).

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

c. Extension of the domain in flow direction:

There should be enough space between the inlet and outlet to minimize boundary effects. The literature suggests required upwind and downwind distances relative to the central area of interest and the inlet and outlet. These distances are also dependent on the blockage ratio.

Figure 2.13 shows typical computational domain dimensions which are based on guidelines for the vertical and lateral extension as well as the extension in flow direction.

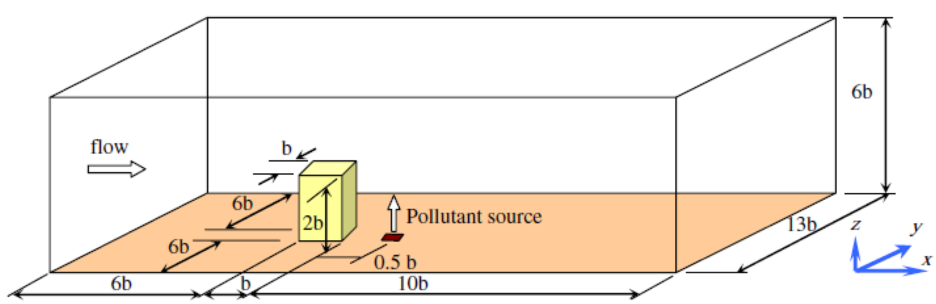


Figure 2.13: Example of a Computational domain (Figure presented in Project Report 1, Literature Review of External CFD)

Selection of Type of Mesh and Cells: The quality of the mesh plays a significant role in the accuracy and stability of the numerical solution. The main attributes associated with mesh quality are node point distribution, smoothness, and skewness. According best practices, the grid should to be constructed to minimize errors, have adequate resolution and capture important physic phenomena.

Domain Inflow and Outflow Air Flow Modeling: The reviewed literature emphasized the importance of the upwind velocity wind profiles for CFD simulations. The selection of an appropriate vertical wind velocity profile is dependent on the type and roughness of the ground. The urban wind profile at the inlet boundary is the most important issue, as it determines the whole domain turbulent flow. The velocity profile can be expressed by the power law (exponent decided by terrain types) or logarithmic law (friction velocity and roughness length). An alternative approach is to directly simulate the roughness elements, similar to wind tunnel tests.

Considering flow characteristics associated with the atmospheric boundary layer (ABL) in the computational domain is important to accurate and reliable predictions. These wind velocity profiles are representative of the terrain outside the computational domain, e.g. terrain upstream of the inlet plane. The reviewed literature suggested that it is important to distinguish between three regions in the computational domain (See Figure 2.14), such as inlet flow, approach flow and incident flow. Wall functions can be applied to the bottom of the domain to characterize the appropriate terrain roughness.

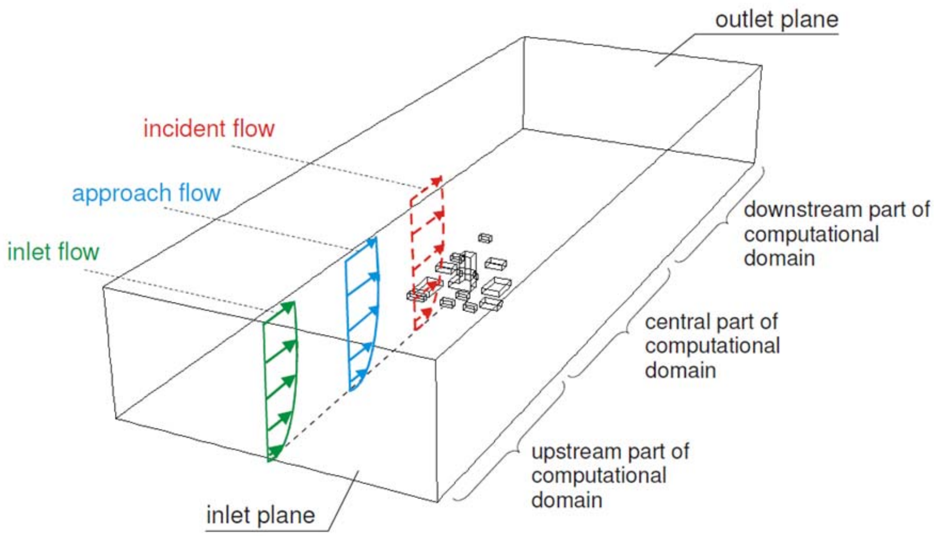


Figure 2.14: Computational domain with building models for CFD simulation of ABL flow – definition of inlet flow, approach flow and incident flow and indication of different parts in the domain for roughness modeling (Figure presented in Project Report 1, Literature Review of External CFD)

2.5 CFD Solver

With the CFD simulation prepared and all pre-processing parameters defined, the actual CFD simulation is controlled by a number of solver criteria. These criteria are defined hereafter:

Turbulence Modeling: The selection of the turbulence model has a significant effect on result of the CFD simulation, since virtually all air flow applications are turbulent and hence require a turbulence model. Turbulence causes the appearance of eddies in a wide range of length and time that interact in a dynamically complex way. The following three categories of numerical methods can capture the effect of turbulence modeling. These are:

- Turbulence models for Reynolds-average Navier- Stokes (RANS) equation:
- Turbulence models for Large Eddy Simulations equation (LES):
- Direct Numerical Simulations (DES):

For the present CFD project only RANS turbulence models were considered. 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 and mean pressures. There are general and specialized turbulence models. The typical CFD software application provides several more general turbulence models. The most commonly used turbulence models are briefly listed in the following, in the order of significance to the present CFD project.

<u>The k-ε turbulence model:</u> 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 generation and destruction of turbulence energy. Advantages of the k-ε model include that it is the simplest turbulence model for which only initial and/or boundary conditions are required. The k-ε model shows an excellent performance in many industrial CFD applications and a wide range of CFD applications.

<u>Menter SST k-ω model</u>: This turbulence model has advantages over the k-ε model in regard 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 SST k-ω model provides a good resolution of pressure gradient near boundary layers.

<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 boundary k and ω have to be specified.

Boundary Conditions for External Flow: The mathematical model of the CFD simulation requires specifying conditions on the domain boundaries. Conditions related to start time are called Initial Conditions. Conditions related to space are called boundary conditions. Boundaries are surfaces (or lines in a two-dimensional case) that completely surround and define a region. Basic types of boundary conditions, which are significant elements for the CFD external flow simulation, include inlet boundaries, outlet boundaries, no-slip wall, and free stream boundary. Figure 2.15 illustrates these types of boundaries conditions.

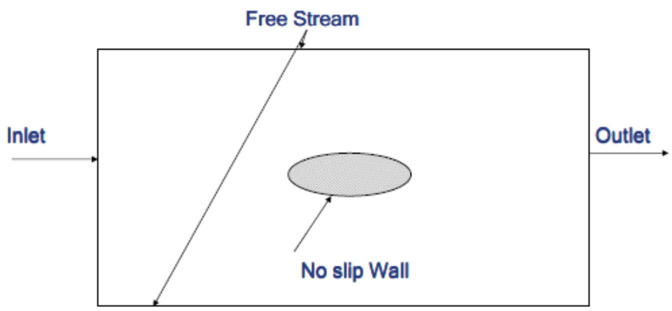


Figure 2.15: Types of boundary conditions (Figure presented in Project Report 1, Literature Review of External CFD)

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. A mass flow inlet is defined when an 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.

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 one or multiple ducts. For multiple flow split outlets a fraction of the total mass flow through each of the outlet boundaries is defined, with the sum being unit (=1). A pressure outlet boundary is a flow outlet boundary at which the pressure is specified.

No-Slip Wall / Symmetry plane requires prescription of velocity at the wall. A symmetry plane boundary represents an imaginary plane of symmetry in the simulation.

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.

Convergence: Convergence is the tendency of limiting behavior in the solution of the governing equations. Convergence is typically represented by diminishing residuals of important parameters of the numerical solution. The fact that a solution converges is not necessarily a proof the converged solution is not necessarily an accurate one. The following present several basic assumption of convergence:

<u>General approach:</u> The iterative calculation 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.
  - Solutions are converged when the flow field and scalar fields are no longer changing.
  - It is most common to monitor the residual.

<u>Monitor residuals:</u> Figure 2.16 illustrates monitoring residuals:

- 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 cannot not infer the accuracy of 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 does not suggest that first order solutions usually produce more accurate results.

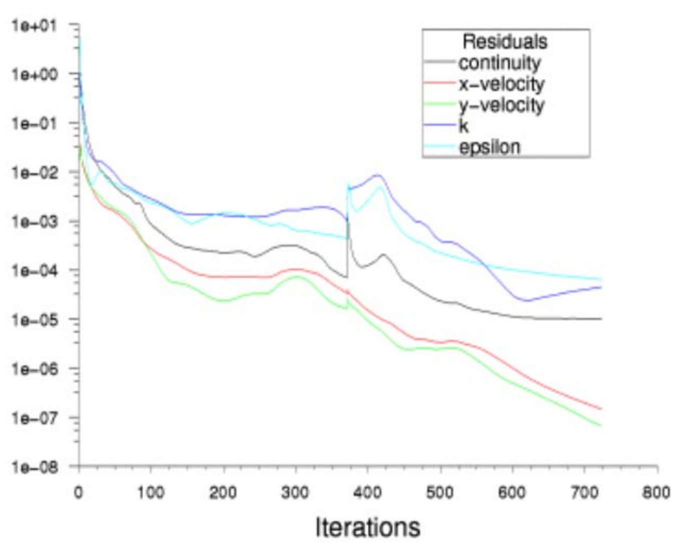


Figure 2.16: Residuals monitoring (Figure presented in Project Report 1, Literature Review of External CFD)

Stopping criteria: 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.

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.

The selection of the appropriate turbulence model is of high importance for a successful CFD simulation outcome. The CFD team needs to make conservative choices to select a robust turbulence model that Is consistent with the physical description of the simulation. Solution convergence or non-convergence can have multiple reasons and needs to be evaluated.

2.6 CFD Post Processing

Post-processing is the method of visualization of the result of solution of the CFD simulation. The complete solution is represented by a very large number of discrete numbers, each associated with a cell inside the computational domain.

The most widely used post-processing methods include the following:

- 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
- Tables of numerical data

Figures 2.17 through 2.19 show various visualizations of the CFD solution. With all the different types of visualizations the CFD user is able to produce the kind of presentation of data, which is most suitable for the application and the intended audience. The more intuitive the CFD results are depicted, the better is the physical characteristics of the CFD simulation presented to the target audience. While typically colored contour map presentation is intuitive to comprehend the distribution of solution parameters in the computational domain, at times it is equally important to extract specific numeric values of parameters at specific locations. This is especially important for CFD validation.

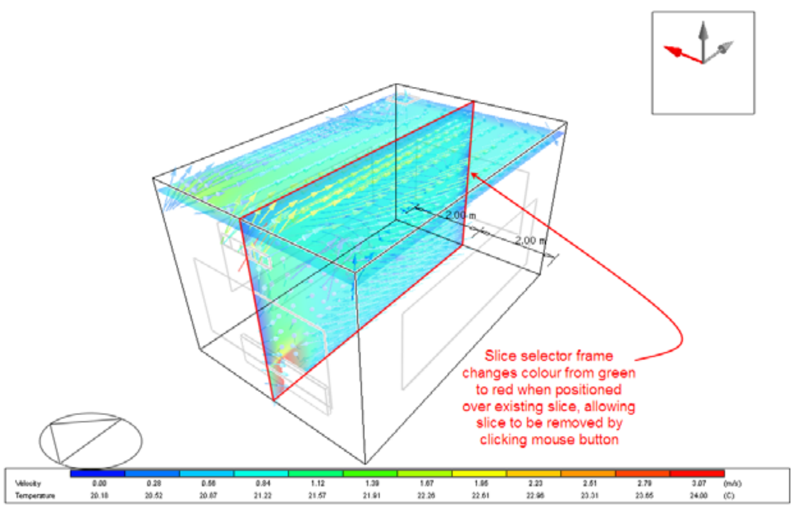


Figure 2.17: Example of 2d data visualization (Figure presented in Project Report 1, Literature Review of External CFD).

The figure shows an example of visualization by means of 2D-slices. These slices are 2D-planes defined inside the computation domain on which values of parameters are projected to show the solution. Most common are colored contour maps where the parameters are depicted in color palette that defines value ranges of solution parameters.

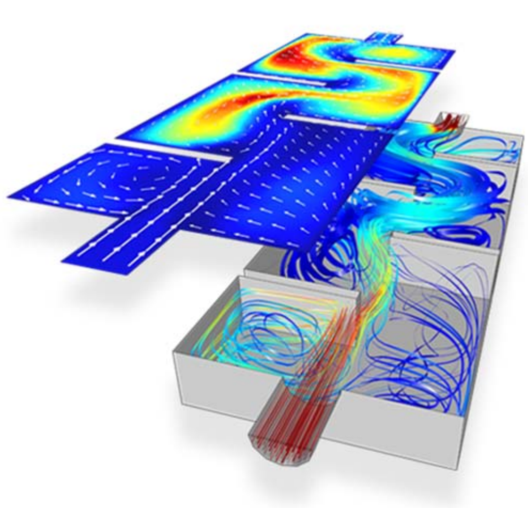


Figure 2.18: Example of 2d data visualization (Figure presented in Project Report 1, Literature Review of External CFD).

The figure shows a combination of colored contour maps (scalar values of velocities) and streamlines (vector representation)

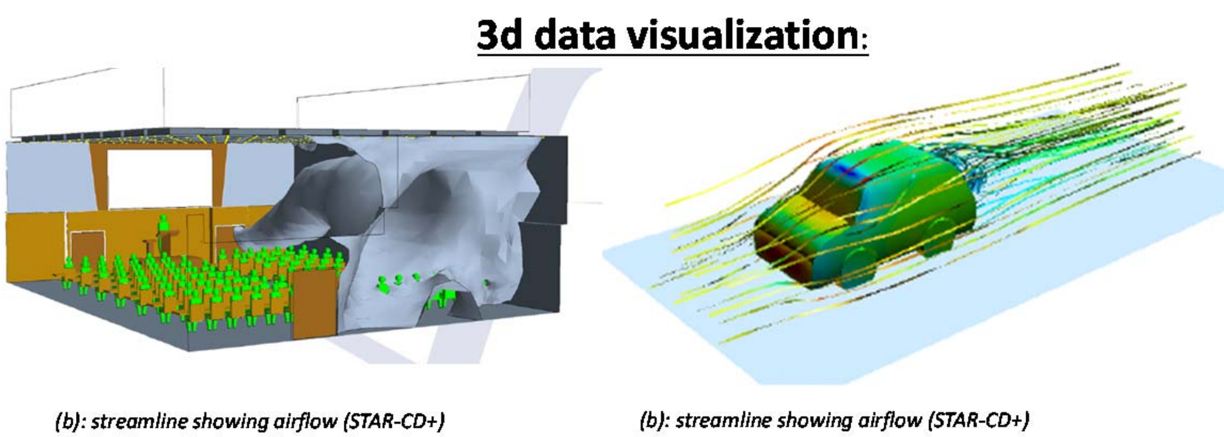


Figure 2.19: Example of 2d data visualization (Figure presented in Project Report 1, Literature Review of External CFD)

CFD presentations in form of colored contour maps and streamline diagrams are pleasing and powerful ways to interpret the CFD calculations and convey the physical relevance of the CFD computations.

SECTION 3 - ESTABLISHING EXTERNAL CFD WORKFLOW FOR ERDL-HNEI

This section summarizes the project work done on developing an external CFD workflow to be used by the project team for investigating wind movement phenomena around buildings.

The first of the two main objectives of the project work was to describe a generic external CFD workflow process to be used for simulating air movement and pressure distribution adjacent to buildings, and as a subset of the generic workflow processes, consider specific requirements for Hawaii's climate. A summary of generic external CFD workflow is presented Section 3.1.

The second main objective was the selection of one or more CFD software products that would enable the project team to carry out the specific analysis efforts for external CFD. For the selection of the preferred CFD software products out of an initial group of seven candidate CFD software products, the ERDL-HNEI CFD research team used a quantitative comparison by means of a two-tiered ranking procedure. While the original project scope intended to compare only three CFD products on select one, the project team made the decision to compare seven candidate CFD software products and select the most promising. The related project work is presented in Section 3.2

Other objectives of this project phase were:

- Review fundamentals of CFD calculations and modeling as it applies to buildings in Hawaii
- Review different approaches to build and adjust CFD meshing, evaluate different mesh geometries and determine the best suitable geometries (including façade appurtenances) for air flow and wind induced pressure phenomena around buildings.
- Evaluate required capabilities of the candidate CFD software products as they apply to CFD applications in Hawaii climate
- Evaluate various ways to post-process (visualize) CFD data, both basic as well as more advanced CFD visualization functions

3.1 Description of an External CFD Generic Workflow

Assigning Project Phases and Project Team Responsibilities

The proposed workflow for external CFD applications is illustrated in Figure 3.1, which depicts elements of the generic external CFD workflow. Figure 3.1 suggests two categories of process steps in the CFD workflow:

- Process steps (A) and (B) are decision steps in the work flow, which require the setting of goals and objectives of the CFD calculation process and determining how close these goals have been achieved.
- Process steps (1) through (4) are four basic sequential CFD steps in the workflow. This workflow elaborates the computational solution of the CFD problem. The process steps (1) through (4) are basically the "tool-set" of the CFD calculation workflow. The validity of the process steps by themselves is not dependent on a suitably defined objective of the CFD project. This means while the calculation procedure might be correct, a wrong input typically creates inaccurate results.

There are several different team functions for different process steps in the generic external CFD workflow. The responsibilities of the team members are described in this section.

Process step (A): The first step (A) of the CFD workflow is setting objectives for the CFD investigation; for example, the main objective might be determining the wind speed around buildings, the dispersion of exhaust from a building stack or the pressure distribution on the building envelope. Each objective requires different physical settings and calculation requirements of the CFD analysis. At the conclusion of the process step (A) all relevant calculation parameters and requirements have determined.

<u>Responsibilities of project team members for process step (A)</u> are shared between the principal investigator or project manager. The decision about the objectives has to be elaborated in close cooperation with the CFD project team. The CFD investigation has to serve the design or compliances objective at hand. The principal investigator needs to have a good understanding about the overall CFD work process to translate the project objectives into a viable CFD project approach.

Process step (B): After the completion of a CFD simulation the obtained results have to be examined regarding their consistency with the objectives, and design conclusions have to be drawn. The assessment of results (B) can determine whether the results appear to be valid or whether CFD results favorably compare to previous validation results. In case of a favorable review, the CFD analysis is accepted and the results allow conclusions to satisfy the objectives and goals. If the results appear to be questionable or a sensitivity analysis of certain parameter is deemed preferable, a new CFD calculation iteration might be needed. If the assessment of CFD results suggests that the objectives have not been answered in a consistent way, the objectives might have to be revisited. In the more likely event a new CFD calculation iteration will be required along with changing parameters for the 3D-modelling of the numeric domain, the meshing of the domain or the particular solver settings.

Responsibilities of project team members for process step (B) are shared between the principal investigator and project manager. The results of the CFD calculation have to be adequate to allow meaningful answers and results to the project objectives. The results of the CFD calculation need an appropriate degree of validation and verification.

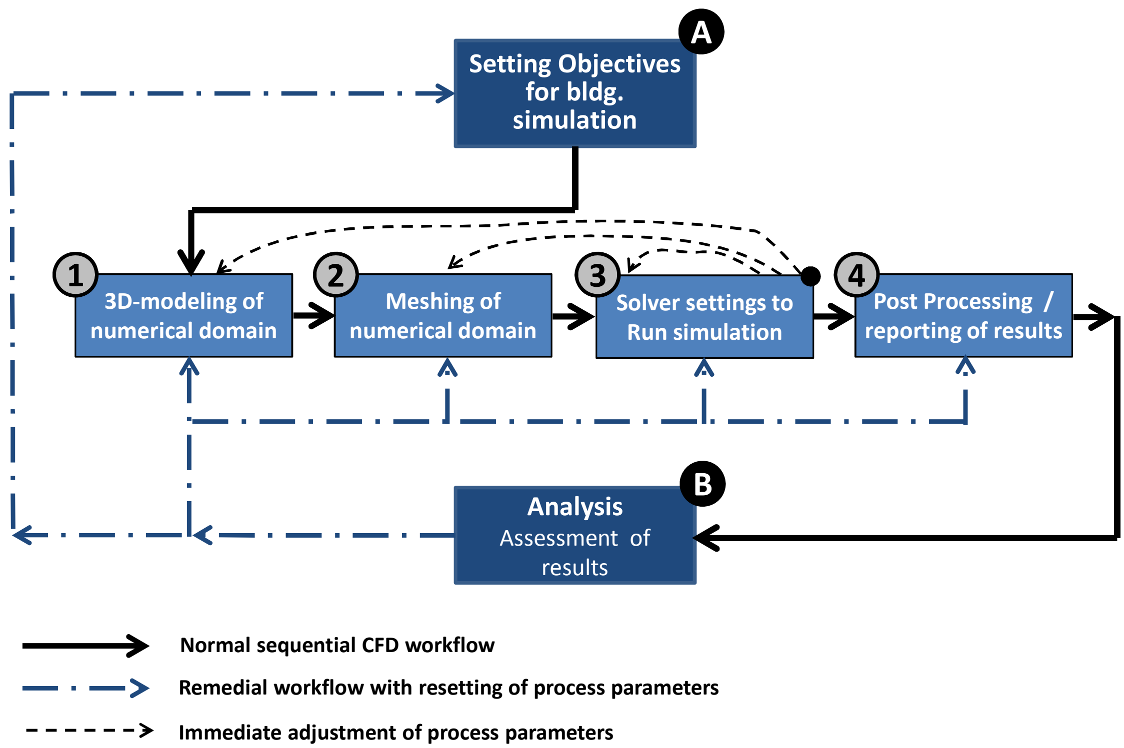


Figure 3.1: Generic workflow for external CFD showing process steps involved (Figure presented in Project Report 2, Establishing External CFD Workflow)

Process step (1)- 3D-Modelling: The initial step of the operative CFD workflow, e.g. the part of the workflow when the team is working with the software, is the creation of a suitable geometry of the computational domain. The computational domain represents the volume of fluid which is subjected to the selected physics conditions. The fluid body is confined by the outer domain boundary (e.g. walls of the virtual wind tunnel) as well as inner domain boundaries (e.g. solid structures inside the confinement of the outer domain boundaries). Typically the solid structures represent objects around which the fluid movement or effects of the fluid movement has to be determined. The geometry of the solid structures can either be created inside the CFD software or imported from a third party 3D-CAD program. The advantage of using a third party 3D-CAD program is that such programs typically allow the creation of a wide range of solid objects, which are composed of either straight or curved outer surfaces. Importing the 3D-geometries into the CFD program requires CAD interfaces that translate the CAD geometries into the format that can be used in the CFD program.

<u>Responsibilities of project team members for process step (1)</u> are shared between the principal investigator, the CAD operator and the CFD software operator. The CAD operator is responsible to create an accurate 3D-solid CAD model of the structure (e.g. buildings or other structures) around which wind movement will be investigate with the CFD calculation. The principal investigator has to ascertain that the 3D-geometry is sufficient in detail but not too detailed to ensure an effective use of computation resources. Both the CFD software operator and the principal investigator need to ensure that the 3D-geometry is being correctly imported into the CFD software.

Process step (2) - Meshing: The second step of the CFD calculation procedure is the creation a suitable grid, or mesh, which represents the digitization of the numerical solution. For most of the high performance CFD software codes, the meshing occurs in two phases. The first phase is the preparation of an “impermeable” surface mesh where the surface has to be a continuous mesh that surrounds the solid object. The mesh must not “leak”; in other words, the entire mesh has to be a continuous assembly of individual 3D-faces or other 3D-solid geometry definitions. A mesh that is not consistent with these requirements would negatively affect the accuracy or convergence of the solution. The surface mesh basically represents the outside surface of the objects around which the fluid motion and other flow induced effects are analyzed. The second phase of meshing is the creation of the volume mesh which encompasses the extent of the fluid that is analyzed in the computational domain. The volume of the fluid is discretized into a large number of cells for which the governing equations are solved.

<u>Responsibilities of project team members for process step (2)</u> are shared between the principal investigator and the CFD software operator. The CFD software operator is responsible of placing the 3D-geometry of the buildings and structures into the viral wind tunnel, which is the outer extent of the computational domain.

Process step (3) – Solver & Simulation: The third step of the CFD calculation procedure is the solver setting to initiate and run the CFD simulations. It is important that appropriate parameters are used for the simulation; otherwise the solution might not be precise and stable or the solution might not converge. Important parameters to define are boundary conditions including surface roughness and the type and detailed parameters of the turbulence model. During the third step of the CFD workflow, the sensitivity of certain parameters should be evaluated. If the solution does not show robust convergence, some changes of the geometry, the mesh or the solver settings must be performed. Changes to the geometry of the numeric domain could also include changes of the blocking ratio of the buildings in the virtual wind tunnel. Changes to the mesh might include raising the mesh resolution (e.g. the number of the cells in the computational domain). Changes to the solver might include using a different type of turbulence model or varying coefficients of the turbulence equations being used.

<u>Responsibilities of project team members for process step (3)</u> are shared between the principal investigator and the CFD software operator. The CFD software operator is responsible for entering all input variables which the principal investigator describes so that the physics of the simulations along with other required software parameters represent appropriate settings. An important solver setting is the choice of the turbulence model. The CFD operator and principal investigator monitor the convergence of the solution. In case the results appear non-conclusive, which could mean that the solution does not readily converge, the principal investigator and the CFD operator will attempt to improve the stability of the simulation by adjustments of the grid, a change in meshing or changing the solver settings. This means that the process steps (1), (2) and (3) might be revisited, with changed input data.

Process step (4) – Post Processing:

The fourth and final step of the CFD calculation procedure is the post processing of the calculation results in a way that can be readily interpreted if the objectives of the CFD calculation are met. The significant advantage of using CFD simulations to describe a fluid flow field and related physical effects is that for basically all points in the computational domain there is quantitative representation. A typical way of CFD post-processing involves defining surfaces on which the physical properties such as wind velocity, pressure, temperature or derived properties such as predicted occupant comfort level are visualized by colored contours maps. The post processing can also include other effective visualization means such as velocity vector fields or static or animated particles streamlines. Other options are 3D-contours of properties and integrating properties over a volume or area. While the results of CFD simulations typically represent many thousands or even millions of data points, a convenient visual representation of the data facilitates interpreting the results of the CFD simulation.

<u>Responsibilities of project team members for process step (4)</u> are shared between the principal investigator and the CFD software operator. The CFD software operator is responsible for performing the required software function to visualize the CFD results so that the principle investigator can determine the outcome of the simulation. The principal investigator usually asks the CFD operator to create a variety of contoured surfaces, 3D-grid data or vector graphics. The principal investigator might also ask for tables or other statistical representation of the CFD solution.

3.2 Other Aspects of an Efficient CFD Workflow

Required Proficiency of CFD Team Members

An effective CFD team includes functions and roles that require certain proficiency for each team member. The different functions and roles could potentially be carried out by the same person, especially in the case of smaller CFD investigations, where the principal investigator might also perform the software technical aspects of the simulation. For larger projects, however, the different functions will most likely be carried out by different team members. Table 3-1 shows function, responsibility and required proficiency of members of the CFD project team.

Table 3.1: Required Proficiency of CFD Team Members

Role in project team	Required Proficiency	Training needs
Project manager	Needs a high proficiency in overseeing projects, setting project objectives and identifying how CFD can be used to satisfy project objectives	Has to be highly trained in managing complex projects that require considerable technology (e.g. projects that are in need of CFD simulations usually represent interdisciplinary solutions and technology integrations)
Principal investigator	Needs a high proficiency in building physics and a comprehensive and detailed understanding of all aspects of the CFD workflow. Needs proficiency to supervise the CFD team to produce reliable CFD results and advise the project manager in realizing the project objectives.	Has to be highly trained in the scientific, technical and software application details of CFD simulations. Has to be trained in building physics and how CFD results can be used to serve specific problems in architecture and urban design.
CAD operator	Needs high technical proficiency in manipulating CAD geometry on the basis of plans and specifications. Needs to understand what level of detail should be included. Needs	Has to be trained in operating 3D-CAD systems and create 3D drawings from plans.

Table 3.1: Required Proficiency of CFD Team Members

Role in project team	Required Proficiency	Training needs
	good application knowledge of the CAD application used.	
CFD operator - meshing	Needs proficiency to understand the importance of geometric representation of structures in the numeric domain. Needs proficiency to create effective meshes and select most appropriate cell geometries for the mesh. Needs good application knowledge of the CFD software.	Has to be trained to validate 3D-geometries or to import 3D-geometries from external CAD applications. Has to be trained in selecting and creating effective surface and volume meshes for a variety of CFD applications. Has to be trained in the CFD software product.
CFD operator - solver	Needs high proficiency to understand the multiple settings and physical properties that have to be selected to run effective CFD calculations. Needs good application knowledge of the CFD software.	Has to be trained in the solver settings of the CFD simulation. Has to be trained to understand the physical properties and settings and other required setting to run the simulation. Has to be trained in the CFD software product.
CFD operator – post processing	Needs proficiency in creating informative graphical representations of the CFD results. Needs flexible and good communication skills to serve the project teams with a variety of ways to show the CFD results.	Has to be trained in operating the post processor of the CFD software. Should be trained in other graphics software products that can integrate CFD results into high quality visualization.

The functions and roles of the CFD team have to be clearly defined and the CFD team members need the technical and scientific proficiency to carry out the specific tasks of the work flow.

CFD Best Practice Guidelines

Several versions of CFD Best Practice Guidelines (BPG) are available to support the CFD project team to carry out effective and valid external CFD simulations. The project manager in conjunction with the principal investigator should review what BPGs are most beneficial for the project. BPGs are continuously updated and it is best for project manager and principal investigator to consult BPGs and select the one best suitable for the specific external CFD application. The recurring topics presented in the BPGs reviewed in the literature search, which are important to a project specific external CFD work process, are presented below:

Best Practice Guidelines Recommendations to Reduce Errors and Uncertainties:

Selection of target parameters: The first step in a CFD analysis should be the definition of the target parameters, such as pressure and velocity, which are representative of the goals of the simulation.

Selection of approximate equations describing the physics of the wind movement: The selection of the basic equations has significant impact on the simulation errors and uncertainties. The turbulent flow within urban or industrial environments is generally modeled by Navier-Stokes equations.

Selection of the geometrical representation of buildings and structures: Buildings and structures are obstacles in the wind flow that impact flow patterns and related physical phenomena. Wind flow obstacles created by vegetation and other surface characteristics (e.g. roads, grass, sand) have a lesser importance, though foliage can have significant effects on the wind regime around buildings. The level of detail required for individual buildings and structures is dependent on their distance from the central area of interest. The central area of interest should be reproduced with as much detail as possible. Buildings further away may normally be represented as simple blocks.

Selection of the computational domain dimensions: The size of the computational domain in the vertical, lateral and flow directions depends on the area of interests to be represented and on the boundary conditions that will be used. 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. The BPGs provide recommended extensions for the vertical, lateral and longitudinal extension of the domain. Generally, ample “room” has to be provided between the central area of interest and the external boundaries.

Selection of boundary conditions: The boundary conditions represent the influence of the surrounding environment that has been cut off by the computational domain. The proper choice of boundary conditions is very important to obtain accurate solutions. The following boundary conditions have to be identified:

- Inflow boundary conditions, where the velocity and turbulence profile have to be defined.
- Wall boundary conditions, depending on the shear stress conditions either no-slip, viscous sublayer or wall functions have to be defined.
- Top boundary conditions, usually defined through constant shear stress.
- Lateral boundary conditions, is usually defined as a symmetry plane with lateral distance from the built area of interest large enough to minimize effect of the boundary on the flow field.
- Outflow boundary conditions, are either outflow or constant static pressure boundary conditions

Selection of the computational grid: The grid (or mesh) is created 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. A high resolution grid does not always ensure a high quality solution or better convergence; but a high resolution ALWAYS requires more run time

Selection of iterative convergence criteria: The iterative calculation process starts with an initial guess of the flow variables and then recalculates these variables in each of the iterations until the equations are solved up to a user-specified error. The termination criterion is usually based on residuals of the corresponding equations. These residuals should tend towards zero.

3.3 Special Considerations for External CFD applications for Hawaii Climate

In order to create a more direct value of CFD for the building industry in Hawaii, wind pattern predictions around buildings and through neighborhoods require consideration of the specific climatic conditions. High quality CFD simulations can be used for design or compliance decisions.

The main physical phenomena which can affect CFD solutions significantly enough to warrant considerations are thermally induced secondary wind motions. These secondary thermally induced wind motions are be caused by buoyancy induced driving forces that are strong enough to alter the wind pattern and to cause sizable variations in important wind induced phenomena, such as pressures on the external building envelope. When non-neutral atmospheric stratification, small wind speeds and large temperature gradients are present, thermal and possibly also humidity equations might have to be part of the solution.

Several aspects have been identified that have the potential of affecting pedestrian comfort in neighborhoods and around buildings as well as wind induced phenomena on the buildings themselves:

- Using the correct air density and viscosity: In many cases of air movements default values of the software are used by CFD teams. The default values for air density and dynamic viscosity are that of standard conditions, which are typically not present in Hawaii in its hot and humid climate. Air density in Hawaii can differ several percentage points from the default values.
- Assumptions of the atmospheric boundary layer (ABL) conditions and approaches to model its effects are important consideration for the CFD calculations. The atmospheric boundary layer comprises the lower layers of the atmosphere, within which wind flows are affected by considerable shear stresses and induced eddies. High temperature gradients resulting from high solar incidents can produce localized plumes of hot air rising under buoyancy forces. Hot air rising from the heated ground influences the wind velocity profile and therefore might have to be included in the boundary conditions. These kinds of thermally induced wind profiles are typically not being modeled in wind tunnel tests since they include complicated physical phenomena and equally sophisticated experimental challenges. There are certain CFD simulation options that can use thermal equations to enhance the solution in the presence of thermal occurrences. While some CFD software products can model thermally induced fluid motion for external CFD applications, other CFD products restrict their simulation capabilities to isothermal numerical approaches.
- Influence of thermal plumes generated by hot air rising around buildings can affect wind movement and the resulting pressure distribution around buildings. Thermally induced external plumes around buildings are not uniformly distributed around the building, but change with the solar gain at the envelope and the adjacent ground. The CFD software needs capabilities to create boundary conditions that can appropriately model localized buoyancy induced air movement adjacent to the exterior building envelope. In addition to generic wind movement around buildings that can be affected by thermally induced phenomena, wind induced pressures on the buildings can also be affected. Since pressure differentials across the building envelope are important driving forces for natural ventilation, even small pressure fluctuations as a result of thermally induced air plumes might appreciably influence the ventilation effectiveness.
- Humidity is an important aspect in the effectiveness of building occupant comfort. For external comfort considerations, the effect of humidity might be less relevant than for internal building applications. An important issue could nevertheless be the adsorption and desorption of humidity at external building surfaces, as well as at internal surfaces or in building openings that serve natural ventilation systems.

3.4 Candidate CFD Software Products

Part of the development of an external CFD workflow for the present CFD project was the identification of candidate CFD software products and the selection of one or more CFD software products.

In an initial scoping phase the project team had identified the following six candidate software products:

1. **ANSYS Fluent**, provides powerful capabilities of modeling a wide range of dynamic fluid problems from incompressible and compressible, laminar and turbulent fluid flows, as well as related transport phenomena (heat transfer and chemical reactions) in industrial equipment and processes. ANSYS Fluent's featured capabilities include both steady-state and transient CFD analyses, a wide range of turbulence models, complex geometry meshing tool as well as the integration into ANSYS Workbench platform for productive workflow.
2. **CdAdapco Star CCM+,** provides the world's most comprehensive engineering physics simulation inside a single integrated package. Much more than just a CFD solver, STAR-CCM+ is an entire engineering process for solving problems involving flow (of fluids or solids), heat transfer, and stress. It provides a suite of integrated components that combine to produce a powerful package that can address a wide variety of modeling needs
3. **CRADLE scTETRA**, SC/Tetra is a general purpose unstructured mesh thermal-fluid analysis system. The CFD software is usability-focused software with the specific goal of easily enabling calculation of complex geometries. The software product provides both automatic mesh generation and a Wizard based interface that guides the user through the set-up process step by step
4. **CRADLE scSTREAM** uses a structured mesh to provide extraordinary performance in both meshing and computation speed. Its performance is maximized for applications that do not require precise reproduction of fine geometrical curvature to predict accurate flow structures.
5. **Autodesk Simulation CFD**, (formerly CFD-design) provides accurate, flexible fluid flow and thermal simulation tools to help predict and validate product behavior and optimizes designs before manufacturing.
6. **DesignBuilder (CFD Module)** is an easy to use Graphical User Interface to EnergyPlus. The software also includes a CFD module that provides an easy-to-use external CFD function for wind flow around buildings.

3.5 Ranking Framework used in the Selection of Software Product

The six candidate CFD software products were assessed with a two-tier quantitative ranking process. In the first step of the ranking process overall ranking weights are assigned to a range of criteria, expressed as percentage of the sum of all overall weights for all criteria, which is obviously 100%. The overall weights indicate how important the specific ranking criterion is compared to the other criteria on an overall scale. In the second ranking step each criteria for every candidate software is given a specific weight, expressed in percentage. The product of software products specific weight per criterion and the generic overall weight for that specific criterion results in the ranking percentage for the specific criterion and for the specific software. All resulting weights for the ranking criteria for the software are then added and the resulting percentage represents the overall result for that software, expressed as a percentage of the potentially achievable score.

The procedure of ranking the candidate software products and assessing the quantitative ranking for the software for the particular criteria is explained in the Figure 3.2. The figure only shows an abbreviated view of the ranking table, with only one criteria group shown to illustrate the basic steps of the two-tiered ranking process.

Figure 3.2: Layout of Example for Assessing the Quantitative Ranking for Criteria

Ranking matrix for CFD software products for external CFD

A			B	C	D	E	F	G	H	I
No.				overall weights levels				**CFD Software Product x**		
Number first	Number second	Number third	Description	weight first	weight second	weight third	weight overall	ranking	result. contribution	result. Points
3			**Workflow**	50%						
	1		**Ease of use / GUI**		**15%**					
		1	GUI is easy to use for trained operator			60%	4.5%	3	50%	2.3%
		2	User guide and online help is available for each command			40%	3.0%	4	80%	2.4%
			sum third level >>>			100%				
			sum of second level >>>		X					
			sum of first level >>>	X				sum overall points >>		X
								CFD Software Product x		

The ranking criteria is shown below:

ranking criteria

1	5.0%	This criteria is not achieved or is only achieved to a very limited extent'
2	30.0%	This criteria is somewhat achieved, but more than a little achievement
3	50.0%	The criteria is achieved on an average (50%) level
4	80.0%	This criteria is achieved to an above average level
5	95.0%	This criteria is totally achieved or to a very high extent

Figure 3.2 shows a yellow row at the top with column numbers. The significance of the columns in Figure 3.2 is described below:

Column group A: This indicates the criteria number. Here the criteria 3.1.1. and 3.1.2 are described.

Column B: Here a description of the criteria groups as well as the individual criterion is given.

<u>Column group C through F:</u> This column group defines the generic overall weight, which is applied to all software products:

Column C: This is the first level overall weight. In our example the work flow group has 50% of the entire overall weight of all ranking criteria.

Column D: This column represents the second level overall weight which is equal to a relative weight within the first level criteria group. In our example the second level weight is 15%. It follows that all sub-criteria under the second level group (e.g. criteria 3.1.1 and 3.1.2) have an overall weight of 50% * 15% = 7.5%.

Column E: This column represents the relative weight within the second level criteria group. The third level overall weight designates the relative weight of all criteria that fall under the second level criteria group and assigns a percentage weight relative to the second level criteria group.

Column F: This column represents the overall weight of the particular criteria. The percentage is the product of columns C, D and E.

In our example, the criterion 3.1.1. has the following overall weight:
*50% (Column C) * 15% (Column D) * 60% (Column F) = 4.5 % in overall weight.*

<u>Column group G, H and I:</u> This group of three columns determines how well the specific software performs relative to the value proposition statement of the specific criterion.

Column G: This column represents the degree to which the particular software complies with the definition, or value proposition, of the criteria. A discrete ranking scale of 1 through 5 is used. The discrete rank “1” means that the criterion is not at all fulfilled or only to a rather limited extent. On the other side of the range a “5” means that the criterion is fully fulfilled or at least to a high degree. The designations between 1 and 5 are subjective and are based on a clear definition of value propositions as described in Table 2-2.

Column H: This column reflects the percentage value which is associated with the discrete ranks 1 through 5. The mappings of the discrete numbers 1 through 5 against percentage of compliance with the criterion description or criterion value proposition is shown at the lower portion of Table 2-1. By entering “2” into column G, for example, 30% is automatically selected in column H. This percentage indicates to what degree the particular software is consistent with the criterion value proposition.

Column I: This column indicates the actual contribution of the software to this criterion, expressed in overall weight. Column I shows the product of the generic overall weight of this criterion and the software specific contribution. In our example, the overall weight of criterion 3.1.1 is 4.5% (column F) and the result of the ranking assigns 3 (column G) or 50% (column H). The result of 4.5% * 50% is 2.25%, or 2.5% rounded.

3.6 Description of Ranking Criteria

Table 2.2 describes all ranking criteria with their value proposition. These ranking are used for the ranking of the seven software products.

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
1	**Value for the CFD software product to the architecture and building market**	
1.1	Benefit for external CFD applications to architecture	The benefit of the software is reflected in its direct usefulness in external CFD applications. A high ranking would imply that the software product has the required complexity and numerical capability to provide a high level of confidence in predicting external wind phenomena.
1.2	Previous applications in architecture / urban design	The benefit of the software product is reflected in its current spread of use in the building industry. With a high ranking, the software has a track record in successful applications in not only academic institutions but also in the commercial architecture and building industry.
1.3	Previous mentioning of software product in scientific or applied professional articles	The benefit of the software is reflected by its mentioning in the literature regarding application of the CFD software in the built environment. A high ranking would imply that the software product is being mentioned to a larger extent in scientific papers or professional articles.
1.4	Using graphics output for architectural presentations	The value of the software is established by the fact that results of the CFD are readily usable for illustration purposes. A high ranking would imply that the software product can be easily used to integrate airflow patterns with 3D-renderings of the building (and appurtenances).

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
2	**Support by vendor / expert network**	
2.1	Technical support available	The value of the software product is reflected in a high quality level of support by the software company. A high ranking of the technical support refers to the responsiveness of the technical support to assist the user.
2.2	Online tutorials available on demand	The value of the software product is reflected in the wide selection of online tutorials that can help the user to learn the software workflow and understand a certain depth of application knowledge. A high ranking would suggest that the software package offers a significant cope and number of CFD videos online.
2.3	Online classes available (low entry barriers)	The value of the software product is reflected in the availability of online courses for general product knowledge or more specific (applied) product knowledge. A high ranking would imply that a large number of on-demand videos are available for viewing.
2.4	Network of CFD professionals in architecture field Ease of working with academic support	The value of the software is reflected in the availability of an existing network of users in the building industry as well as in academia who are actively engaging in the sharing of application knowledge. A high ranking implies that the software company supports its own network of professional CFD practitioners or educators so they can share knowledge and engage in discussion and forums.
2.5	Ease of working with academic support	The value of the software product is reflected in the level of support afforded to the academic community and/or user. A high ranking would imply that the software company actively supports academic institutions, including faculty as well as students, to acquire the means of using the software for teaching and basic research.

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
2.6	User guide provides the efficient information	The value of the software is reflected in the clarity and comprehensiveness of the user guide and other product documentation in regard to efficient information transfer. A high ranking would imply that the user guide offers a wealth of application information (i.e. how to use software as well as how to interpret the results of the software calculations).
3	**Workflow (the work flow criteria group includes several sub-criteria groups)**	
3.1	**Ease of use / GUI**	
3.1.1	GUI is easy to use for trained operator	Graphic user interface (GUI) benefits the user with an intuitive and easy-to-use user interface. A high ranking would imply that the software product offers an efficiently organized GUI with ready access to all key elements for the CFD workflow.
3.1.2	User guide and online help is available for each command	There are many technical, physics, and definition for CFD simulation, therefore user guides and online help are the major significant support functions for the user. A high ranking would imply a user guide that is easily available online and has hyper-link accessibility.
3.2	**Modeling / Import geometry**	
3.2.1	Create geometry inside CFD application	Internal or built-in CAD modeling tool inside the CFD software is helpful to allow user to create, edit and modify proper geometry before running the simulation. A high ranking implies that the software product as a built-in CFD geometry modeling tool.
3.2.2	Import geometry / repair surface before meshing	Users may prefer to use the third party 3D-modeling application for building and structures. The high ranking measures the availability of importing different model or geometry different formats into CFD applications.

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
3.2.3	Support Revit Architecture	The majority of architecture students and professionals are using Revit for building design and architectural documentation. A high ranking means that Revit files can easily be imported.
3.2.4	Support CAD format and Inventor	Inventor is one of the popular applications in the industrial design and mechanical model study. A high ranking of the CFD software indicates that the CFD software can readily import CAD format or Inventor file formats (.ipt, .iam) or geometry.
3.2.5	Support Rhino	Rhino is a growing model application for the architecture and design industry. A high ranking of the CFD software would imply that the CFD application can readily import Rhino file format (.stl, .iges, .iges, and etc) geometry.
3.3	**Meshing**	
3.3.1	Surface meshers	An effective surface meshing tool is important to prepare a high quality surface mesh before a volume mesh can be generated. A high ranking would imply that the CFD software has a surface-mesher tool.
3.3.2	Maximum cells and 2D-3D cell volume	A high ranking would imply that the maximum number of mesh cells can be well above 10 million cells.
3.3.3	Boundary types	Specific boundaries types are required in CFD physics models to define the primary variables of the simulation (such as pressure, temperature, and velocity). A high ranking would imply that the CFD software product provides various boundary types for physics model purposes.
3.3.4	Structured and Unstructured meshing	This meshing function is related to the availability of a wide range of meshing techniques. A high ranking would imply that the CFD software provides advanced meshing capabilities.
3.3.5	Adaptive meshing capability	This criterion refers to the ability of the CFD software to modify the existing mesh to accurately capture the fluid flow

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
		features.
3.4	**Physics Models**	
3.4.1	Solvers – Coupled / decoupled	A high ranking would imply that the CFD software can support coupled domain modeling or not only decoupled domains.
3.4.2	Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)	This criterion is used to identify which numerical method for discretization approach is employed in the CFD software. Although FEM is flexible and capable to handle complex geometries, FVM has volume controlled elements that establishes flux properties in their numerical equations. A high ranking would imply that the CFD software uses FVM.
3.4.3	Steady state/Transient state	This criterion is used to check whether the given CFD software can be used for steady state or transient state simulations or both. A high ranking would imply that the CFD software product is capable of transient state simulation.
3.4.4	Turbulence models	This criterion is used to assess capacity of the CFD software in terms of range of available turbulence models. A high rank for the CFD software indicates the availability of a large number of turbulence models.
3.4.5	Materials	This criterion is used to assess the availability of built-in materials of a given CFD applications. The high rank implies that a given CFD application offers a wide range of the above features.
3.4.6	Customizable / special physics models	This criterion is used to assess the availability of customizable / special physics models of the CFD software product. A high rank would imply that a given CFD application offers a wide range of special models and provides the opportunity to customize models.
3.5	**Post-Processing**	
3.5.1	Visualization	This criterion is used to assess multiple visualization features offered by a given CFD application for post-processing. A high

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
		ranking indicates a wide range of visualization features offered by the CFD software, as well as the quality of those outputs and the ease to the user to produce them.
3.5.2	Quantitative analysis capabilities of components	This criterion is used to assess the availability of the CFD software product in offering the quantitative analysis capabilities of certain nodes, line or grid within the computational domain. The high ranking indicates a wide range of features offered by the CFD software, the quality of those outputs as well as the ease for the user to produce them.
3.5.3	Comparative analysis (multi-scenarios)	The benefit of this feature for post-processing is that it allows for effectively comparative analyses of multi-scenarios or testing different turbulence models in CFD simulation.
3.6	**Customization / special functions**	
3.6.1	Pedestrian comfort	Pedestrian comfort standards associate comfort rating with wind velocity on the pedestrian and ground level. A high ranking would imply that the CFD software is able to determine atmosphere boundary layer or velocity profile due to terrain roughness (height z_0).
3.6.2	Natural Ventilation	Natural ventilation is significant passive building design technology. A high ranking of the CFD software would imply that a workflow specifically for natural ventilation is available.
3.6.3	Wind-driven rain (WDR)	Wind-driven rain (WDR) or driving rain is rain that is given a horizontal velocity component by the wind and that falls obliquely. A high ranking would imply that the CFD software is capable in defining and simulating a multiphase physics model, moisture, and humidity.
3.6.4	Humidity	Humidity is a major physical property that contributes to the definition of comfort condition. A high ranking would imply that the CFD software can simulate condensation of dew or

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
		the propagation of humidity in porous material.
3.6.5	Wind load towards to building structure	Typically wind load from natural force and global wind force are predicted by wind pressure coefficient. A high ranking would imply that CFD software is able to demonstrate and visualize wind pressure distribution on building surface, and also provide tools of detecting solution parameter values at node point, line, and grid on building façade.
3.6.6	Pressure distribution along streamlines/pressure losses	CFD derived pressure losses are a significant investigation on flow behavior in different obstruction or constrained geometry. A high ranking would imply that the CFD software can visualize pressure changes along streamlines.
3.6.7	Conjugate heat transfer	Define the solid material conduction and the fluid convection is analyzed simultaneously. The type of fluid convection (natural, forced or mixed) determines the analysis parameters. A high ranking would imply that the CFD software can calculate simultaneous heat transfer in both a solid and a fluid.
3.7	**Support for validation**	
3.7.1	Validation document / example	The benefit of this criterion is that it allows users to check the accuracy levels of a given CFD application as well as the experimental or real-world test set-up that the CFD software used for the data validation
4	**Cost of software**	
4.1	Commercial pricing model	The benefit of the software product is a good benefit to cost ratio, with a high ranking suggesting a good benefit to cost ratio.
4.2	Academic pricing	The value of the CFD software is reflected in the availability of an academic pricing (discount) opportunity. A high ranking implies that the cost of the academic version is significantly

Table 3.2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix (weights)	Description of Ranking Criteria or Value Proposition
		lower than the commercial product pricing.
4.3	Software includes all modules (no extra costs)	The value of the software is reflected in the extent at which all functionality is included in the base software price. A high ranking would imply that the software product has a wide array of all capabilities built-in the one software product that is purchased.
4.4	Option to acquire additional seats at no extra or small costs	The value of the software product is the flexibility to acquire additional seat at no or small extra costs. A high value would imply that the CFD software (or vendor) offers academic student licenses at very favorable conditions.

The ranking of software products carried out by the CFD project team only reflects their preference based on a set of project specific selection and ranking criteria. The outcome of the selection only reflects that the software product is suitable for the specific application in conjunction with the present CFD research work.

3.7 Benchmarking of Six Candidate CFD Products for Workflow Performance

CFD research team performed benchmarking workflow tests to obtain first-hand experience of the specific CFD software products. The CFD team had received temporary access to the six CFD software products and conducted the benchmarking tests within a period of six weeks. The conclusions obtained in the benchmarking tests were used to compile the final overall ranking of the candidate CFD software products.

For the benchmarking tests a ranking framework was used to allow direct comparison of the workflow and a qualitative assessment of the results of simulations using a prototype case of external wind movement around a set of buildings. The framework included the following steps in the workflow process:

Pre-processing: One building with adjacent structures was created for which wind induced phenomena (velocity and pressure) was tested. The geometry was created in an external CAD program and imported into the different CFD software products. All software products used the same geometry. The computational domain was established in such a way that the wind approach direction could be changed around 90 degrees, without the need to recreate the virtual wind tunnel (e.g. the computational domain with inlet, outlet, lateral, top and ground boundaries). The surface and volume mesh was created in accordance to the specific meshing tools of the tested software.

Solver: All solver settings were selected to be similar for all six software CFD products. Since not all CFD software products provide the same sophistication in selecting certain physics parameters, the physics settings, especially the turbulence model, were selected to accommodate the least sophisticated CFD software products. In essence this means that the test with the more sophisticated software products, such as Star-CCM+ and Fluent, used only basic settings and the quality of benchmarking simulation results did not reflect the overall potential of these two advanced CFD software products. Therefore, the results of the benchmarking test should be viewed only in regard to the effectiveness of the workflow and not in terms of direct quantitative comparison of the results for velocity and pressure.

Post-processing: The post processing was used in order to depict the results in accordance of the testing framework. The post processing capabilities of all software products varied, but all of the tested CFD software products were capable of basic slice visualization, which was mainly used for the visualization of the test results.

The project team developed a list of eleven criteria / statements, 1 through 11, to perform a qualitative comparison between candidate software products. These eleven criteria / statements were identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process. The eleven criteria/statements used were as follows:

1. When operating the CFD program what was the overall impression of the GUI (graphic user interface)
2. Preprocessing: How effective was the conversion of the external 3D geometry data into the CFD software? Was it possible to read the 3D-geomerty into the CFD program easily; or did you use the CFD own 3D-geometry builder?
3. How effective was the surface and volume mesh preparation? Where the functions of the CFD program effective to create a mesh resolution that is sufficiently high in the vicinity of the structures of the interest? If yes, could you easily define the extent of the computational domain in which the mesh resolution needed to be higher? Did the mesh generation proceed smoothly or were frequent errors detected?
4. What function do you like or dislike for the mesh generation?

5. Solution – Simulation: How effective were the software functions in setting the required boundary conditions and other physical settings of the computational domain?
6. How was the convergence of the simulations? Where there any issues that impeded the progress of the simulation?
7. Were you satisfied with the performance of the simulations? How was the time for the simulation to converge?
8. Were there any special issues that came up and whose solving was easy or complicated using the software internal functions?
9. Post processing: Were the post processing procedures effective and straight forward, and did they produce appropriate quality images? Is there anything that hampered the effective production of post processing images? What function did you recognize as specifically helpful, or on the other hand as not helpful?
10. In general, what do you judge are very effective functions of the software? Please describe and state how it will assist our work flow of external CFD?
11. Was there sufficient assistance given by the user manual and/or help function? Did you contact the software support for assistance, and if so was the support helpful?

3.8 Representative Results of the Benchmarking

In order to illustrate the process of benchmarking of all software, the resulting conclusions and results of one software product, software CdAdapco STAR-CCM+ (Star-CCM+), are presented. The results of benchmarking for all software products are provided in detail in the report:

The answers to the eleven questions about the external CFD Workflow of Cd Adapco STAR-CCM+ are provided in the following:

1. *When operating the CFD program what was the overall impression of the GUI (graphic user interface)*

   The GUI of Star-CCM+ (Figure 3.3) is a java script interface which can load fast, use little computation resources and provide high resolution. The GUI is designed as a drop down browser menu that has a clean and simple interface and there are no unnecessary and distracting pop-up menus or windows. The drawback of the GUI, a significant shortcoming for new users, is that fact that the interface does not offer clear work flow patterns (e.g. a scripted and sequential ways to set up the simulations and run it). This means that the lacks an indication of the successive steps in the process. Without the guidance of this graphic pattern, the user needs to know where to start and end in the workflow, and also where to input all valuables. The CFD team found it a bit difficult at the beginning to understand the advanced CFD workflow of Star-CCM+. Different from the other CFD software products Star-CCM+ requires the user to have an advanced understanding of CFD, which means the user is required to study the user guide documents for better understanding of the workflow.

2. *Preprocessing: How effective was the conversion of the external 3D geometry data into the CFD software? Was it possible to read the 3D-geomerty into the CFD program easily; or did you use the CFD own 3D-geometry builder?*

   Star CCM+ can use multiple 3D-CAD model file formats; the preferred file formats being Inventor and Revit (but need to export as SAT). While the software can accept multiple file formats, the quality of the surface generated from the imported geometry can be insufficient to create a suitable surface mesh. Starr-CCM+ offers a powerful tool, the Surface Mesher, to repair defective surfaces and create surface and volume mesh. The surface mesher is effective even for complex surfaces imported from a CAD program. There is a simple CAD function inside Star-CCM+ with which simple surfaces can be constructed. However, the generated geometry represents only basic and simple 3D geometry in the Star-CCM+ and the use of the internal geometry builder is more reserved for benchmarking and testing purposes rather than for the analysis of real world problems.

3. *How effective was the mesh preparation? Where the functions of the CFD program effective to create a mesh resolution that is sufficiently high in the vicinity of the structures of the interest? If yes, could you easily define the extent of the computational domain in which the mesh resolution needed to be higher? Did the mesh generation proceed smoothly or were frequent errors detected?*

   Star-CCM+ has an outstanding mesh generation tool. Meshes created in Star-CCM+ can be adjusted in regard to resolution control. The software can adjust grid resolution in the meshing process and generate an effective meshing structure in areas of interest. Setting up the mesh refinement is done by selecting a scaling parameter between 1 and 2, although there are some other parameters that need to be specified, such as target area or ratio (Figure 3.4).

4. *What function do you like or dislike for the mesh generation?*

   Since the Star-CCM+ user interface and the way of navigating to any input factors and parameters are non-sequential within the menu browser (Figure 3.6), the user needs to know all necessary set-up details to avoid missing parameters and/or scripting steps. The mesh generation process is straightforward by first creating the surface and then the volume mesh. Successful meshing requires a closed surface. If the surface is defective, e.g. there are “holes” in the surface, effective repair surface functions can be used, such as the “Surface Wrapper”. For complex model details, high mesh resolution, and targets with high details, the mesh set-up may require longer time to generate a quality surface mesh. There are different mesh structures and cell geometries which require a proper surface mesher that depends on the geometry type, but the most advanced is the polyhedral mesh (Figure 3.4).

5. *Solution – Simulation: How effective were the software functions in setting the required boundary conditions and other physical settings of the computational domain?*

   Star-CCM+ uses all standard boundary condition layer types (Figure 3.5). The boundary conditions and domain size needed inside the software are significantly affected by the surface mesh, therefore surface extraction from the imported CAD geometry and possible repair of the surface are key features in STAR-CCM+. The software contains a complex set-up of physics variables which needs to be defined in the simulation set-up. The user needs a good understanding of the type of physical properties of investigated in the CFD analysis. The complexity of possible input parameters is both an advantage and a disadvantage of the software. An advantage is created since the software can simulate a broad array of fluid and thermal system phenomena. A challenge is created for the user since there is no "script" available in the software to guide the user through (industry) specific CFD application scenarios; such as external wind movement around buildings.

   As an advantage to using CFD for external wind movements around buildings, the software offers a broad range of turbulence models (Figure 3.7 and Figure 3.8).

6. *How was the convergence of the simulations? Where there any issues that impeded the progress of the simulation?*

   Convergence performance of simulations was good throughout the benchmarking tests. Figure 3.9 shows a typical distribution of residuals over first 1000 iterations of the simulation. The fact that good convergence has been attained might be due to the somewhat simple configuration of the building model. Time required for the simulation could be an issue in the case of high grid resolution and small aspect ratio (small blockage ratio). Simulations were re-run, which required clearing prior simulation result after any modifications.

7. *Were you satisfied were you with the performance of the simulations? How was the time for the simulation to converge?*

   The simulation performance was good. Most simulations took between three and four hours to accomplish the first 1000 iterations.

8. *Were there any special issues that came up and whose solving was easy or complicated using the software internal functions?*

   As stated before, with sufficient experience of the user, the solver set up is straight forward and effective. In the benchmarking the only issues that arose was defective surfaces which would point to an errors or incomplete (“open”) surface geometry.

9. *Post processing: Was the post processing effective and did it produce appropriate quality images? Is there anything that hampers the effective production of post processing images? What function did you recognize as specifically helpful, or on the other hand as not helpful?*

   STAR-CCM+ provides very good post processing tools both in regard to visualization and quantification of parameter values . The simulation visualization is able to produce a range of visualization options with and without contour lines (Figure 3.10 through Figure 3.13). The highest resolution of output image is 4000 x 3000 pixels. Alpha-numeric data of simulation results (such as velocity and pressure) can be obtained for a specific point, for lines and areas. This data can be exported to spreadsheets or as ACS characters.

   When using the streamline visualization option “line integral convolution” in high resolution (Figure 3.14) runtime problems and in some cases system crashing occurred. The CFD team attributes this to hardware limitations and the team anticipates that these problems can be avoided when the software is installed on the newly acquired computer in the CFD lab. Overall the most useful tool for the CFD team has been as pressure distribution on building surface, pressure and velocity probe slice plane, velocity streamline, and velocity vector slice plane. These visualization techniques are very useful to depict distribution of wind occurrences and pressures on and around the building envelope (Figure 3.15).

10. *In general: What do you judge are very effective functions of the software? Please describe and state how it will assist our work flow of external CFD.*

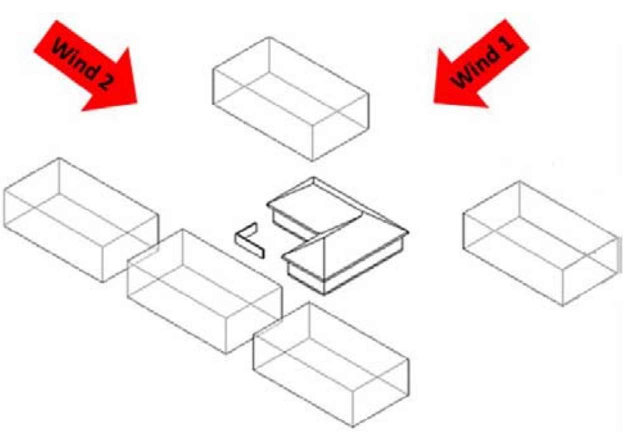
    Over all, STAR-CCM+ is an advanced CFD software tool that provides a wide range of very specific CFD simulation capabilities as well as visualization and data quantitation options. Moreover, there are multiple meshing structure styles for proper tasks and models. Working interface and GUI are simple and fast, at least for the experienced user

    The modeling capabilities inside the software are not yet as comfort and intuitive as creating models with external CAD programs, such as Inventor. If the imported surface has to be repaired the STAR-CCM+ provides an effective surface repair option, although the tool requires special application experiences. Having a clean 3D geometry inside STAR-CCM+ is an absolute requirement in order to create a quality surface and volume mesh. Most of the function settings in the GUI are not explained sufficiently and therefore users need to consult the voluminous user guide documents.

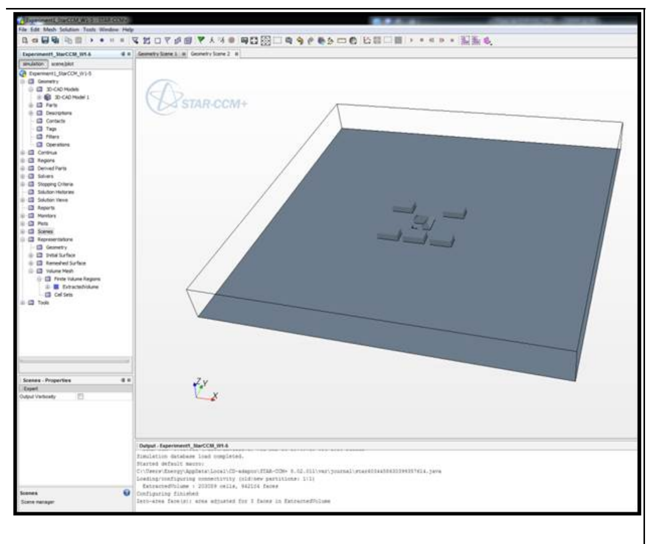
11. *Was there sufficient assistance given by the user manual and/or help function? Did you contact the software support for assistance, and if so was the support helpful?*

    The complexity of the software and the lack of guided sequences in setting up the simulation and entering all required parameters without being prompted, require a comprehensive understanding of user guides and specific descriptions for modeling, surface repair, and other set-up functions for the simulation. Star-CCM+ provides all required documents which are very helpful and the software provides the user with sufficient guidance to set-up, run and post-process building simulations.

Figure 3.3 through Figure 3.15 are presented below:



This is the generic building geometry that was used in the benchmarking for all CFD software products



Star-CCM+ GUI. (the image on the right depicts the building and adjacent structure of the benchmarking test)

Figure 3.3: Example screen-shot of the graphic user interface (GUI) of Star-CCM+. (Figure presented in Project Report 2, Establishing External CFD Workflow)

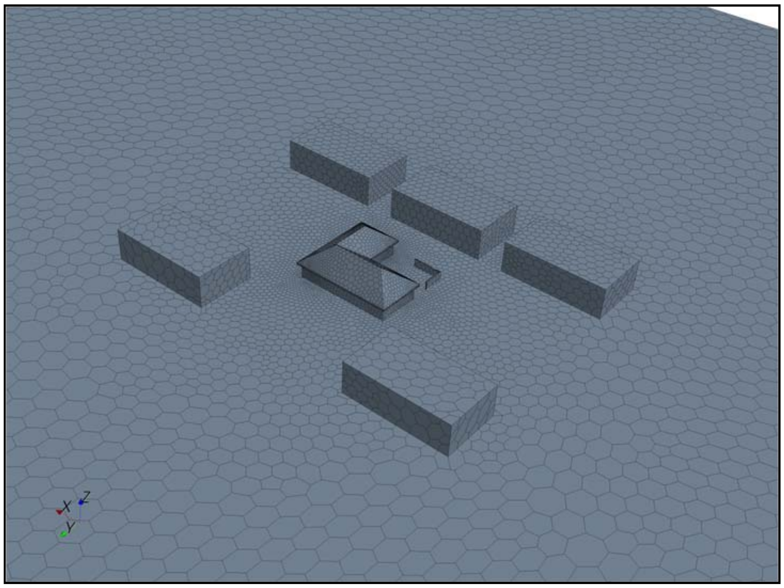


Figure 3.4: Mesh structure in Star-CCM+ shows target area and refinement level around the area of interest. (Figure presented in Project Report 2, Establishing External CFD Workflow)

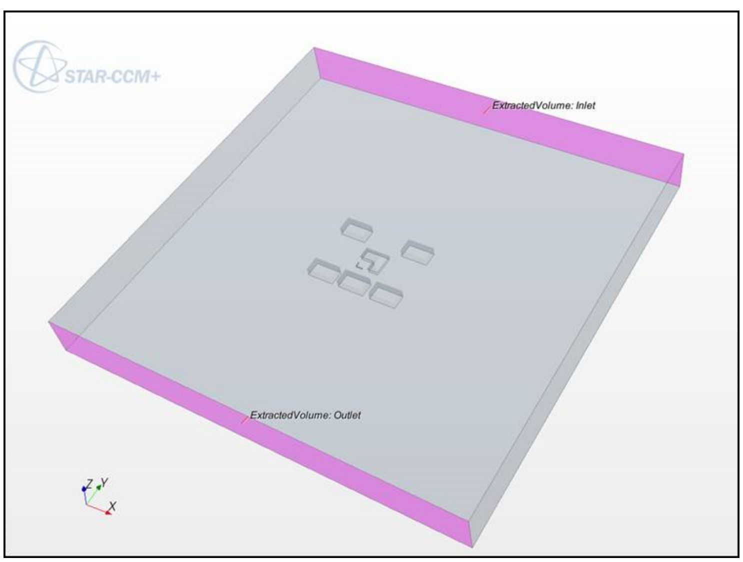


Figure 3.5: Assigned boundary conditions and layer types (Figure presented in Project Report 2, Establishing

External CFD Workflow)

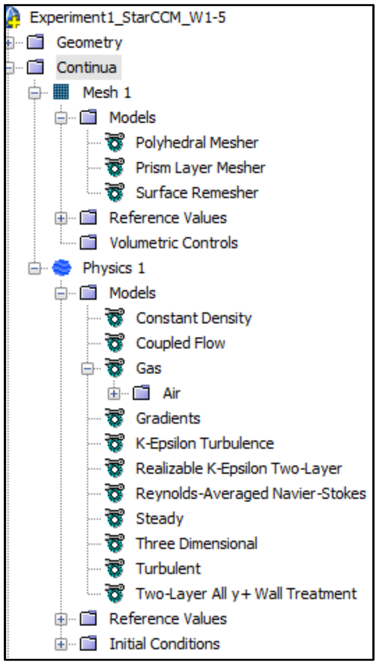


Figure 3.6: Star-CCM+ shows a continua functions for mesh model and physics drop down menu set up

(Figure presented in Project Report 2, Establishing External CFD Workflow)

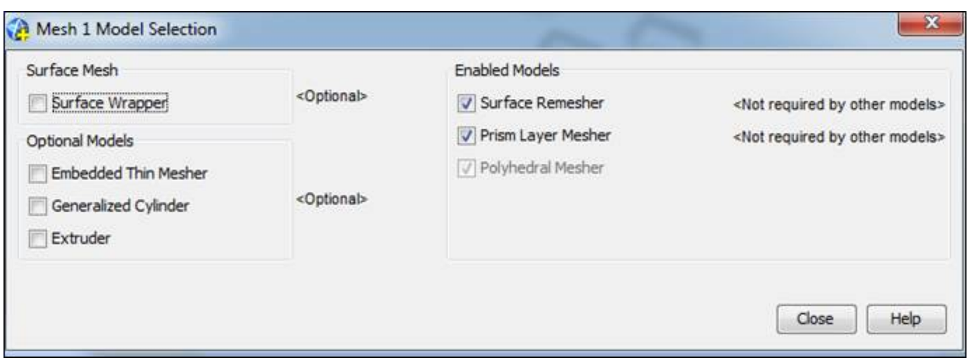


Figure 3.7: Star-CCM+ shows mesh model set up and selection (Figure presented in Project Report 2, Establishing External CFD Workflow)

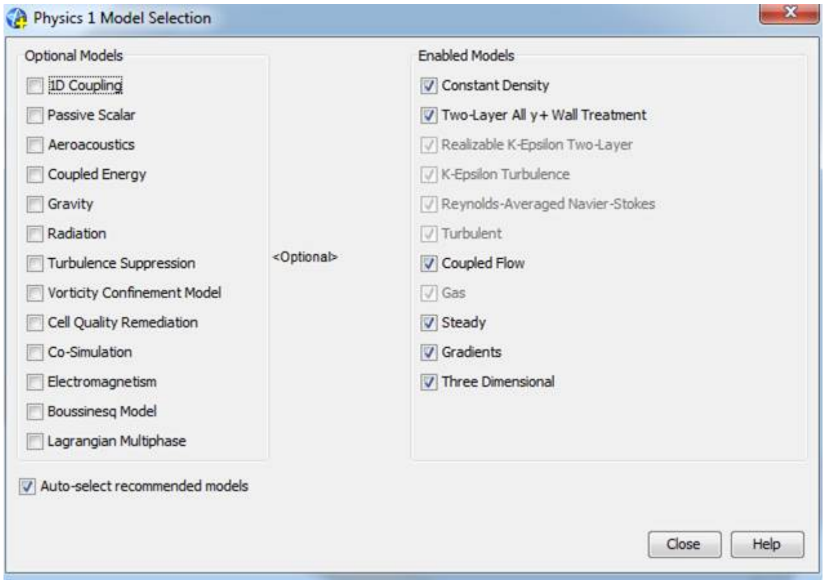


Figure 3.8: Star-CCM+ shows physics set up and selection (Figure presented in Project Report 2, Establishing External CFD Workflow)

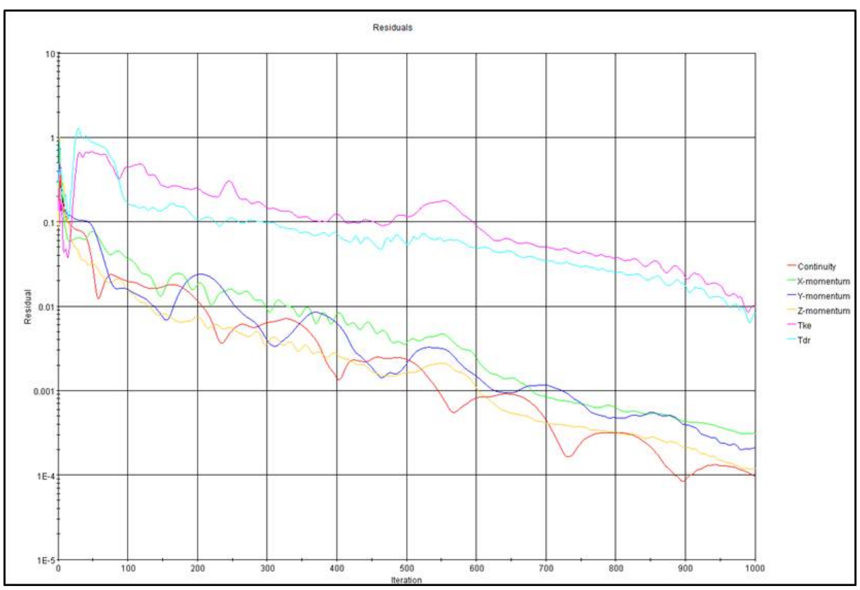


Figure 3.9: Star-CCM+ residuals report depicts the convergence in the tests

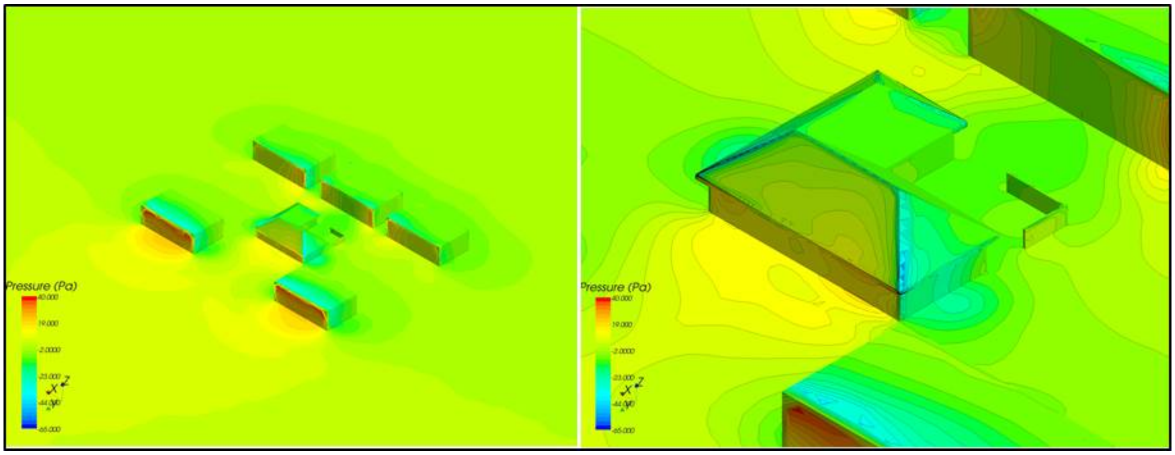


Figure 3.10: Star-CCM+ shows the external CFD simulation result of pressure (Pa) distribution on building faces (Figure presented in Project Report 2, Establishing External CFD Workflow)

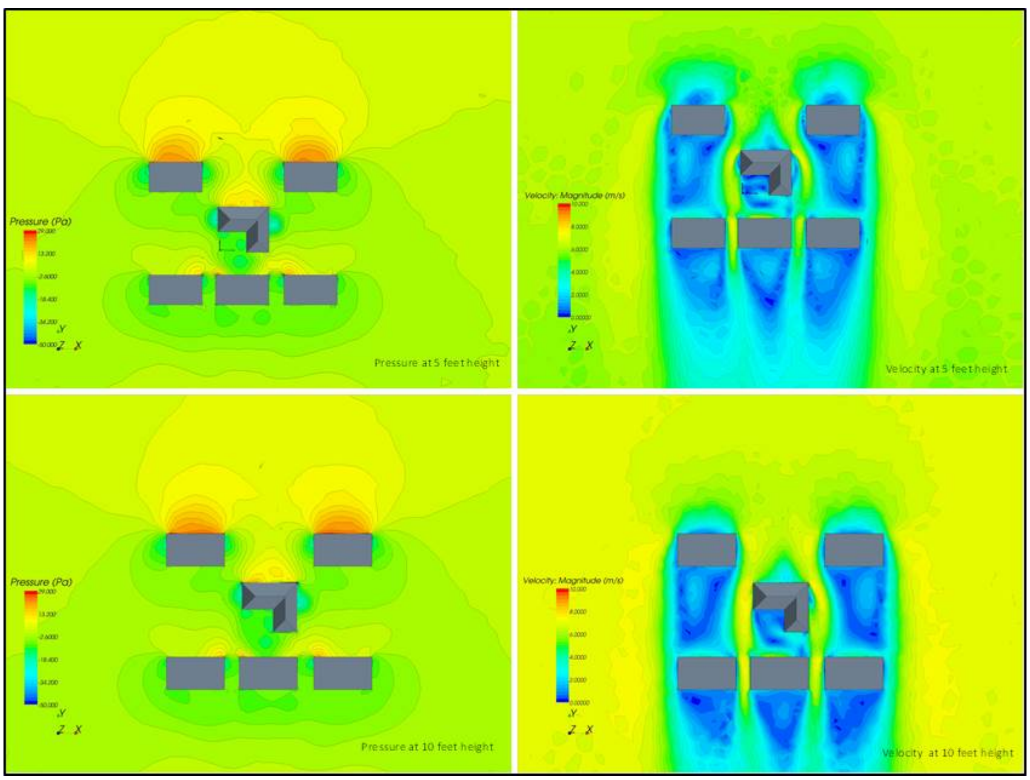


Figure 3.11: Star-CCM+ shows the external CFD simulation result of pressure (Pa) and velocity (m/s) at the different slice plane height levels (Figure presented in Project Report 2, Establishing External CFD Workflow)

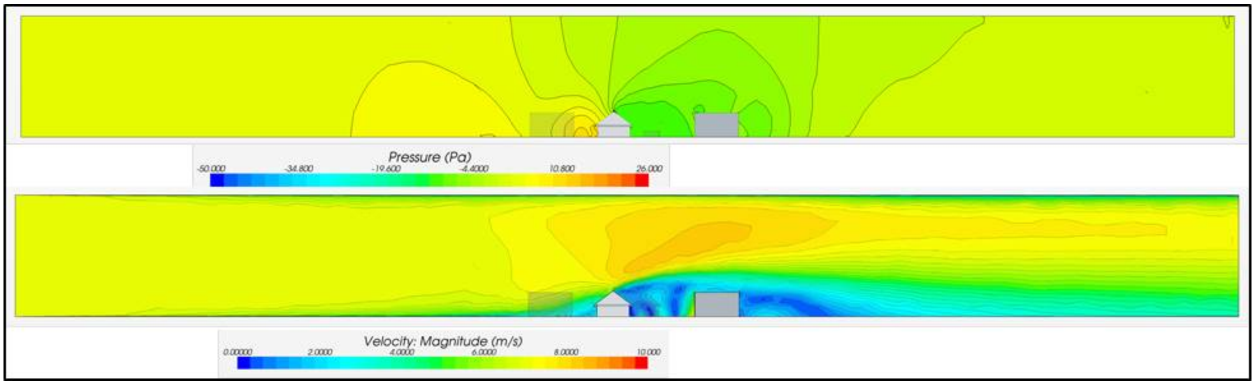


Figure 3.12: Star-CCM+ shows the external CFD simulation result of pressure (Pa) and velocity (m/s) at the cross section plane. (Figure presented in Project Report 2, Establishing External CFD Workflow)

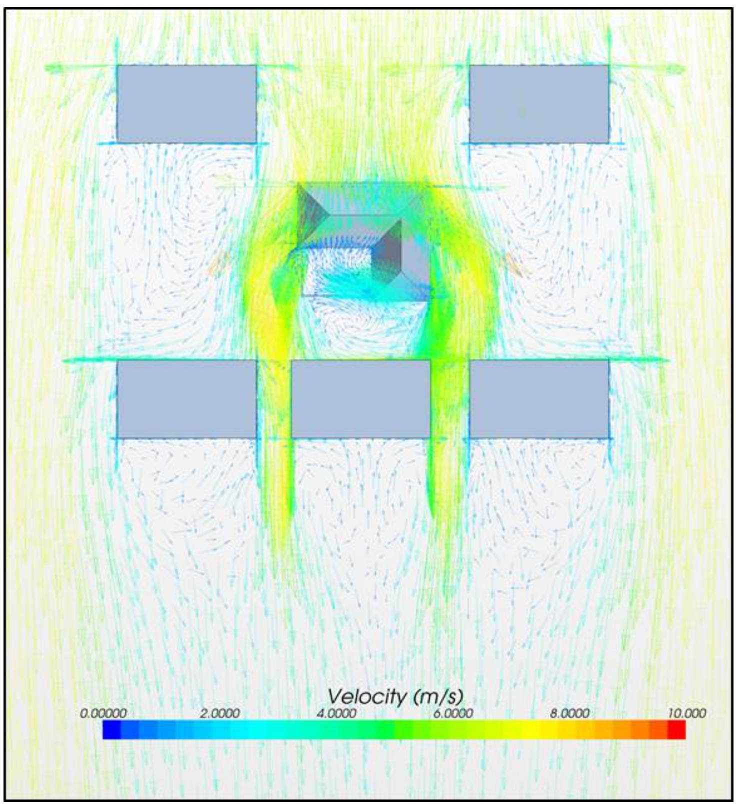


Figure 3.13: Star-CCM+ shows the results of external CFD simulation with wind vector streamline at 10 feet height

(Figure presented in Project Report 2, Establishing External CFD Workflow)

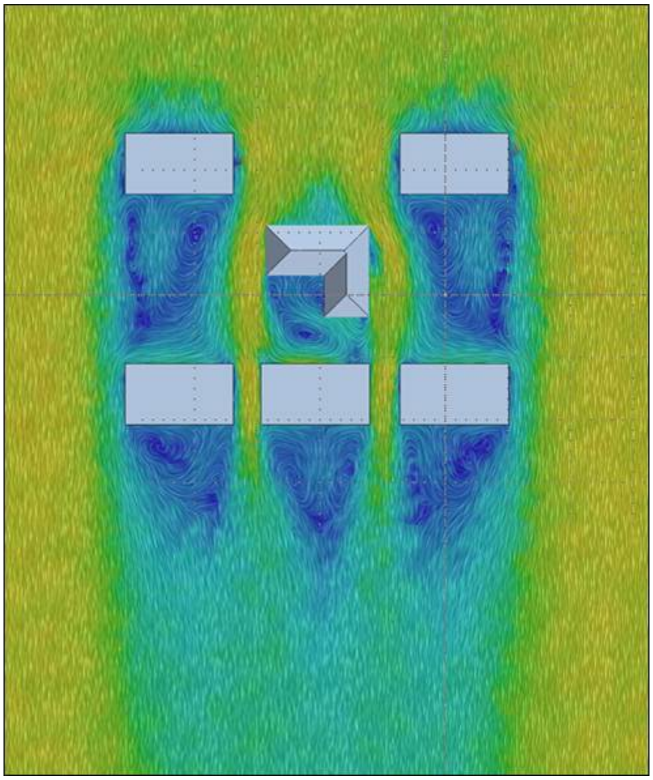


Figure 3.14: Star-CCM+ shows the external CFD simulation of streamline options in high resolution of line integral convolution.

(Figure presented in Project Report 2, Establishing External CFD Workflow)

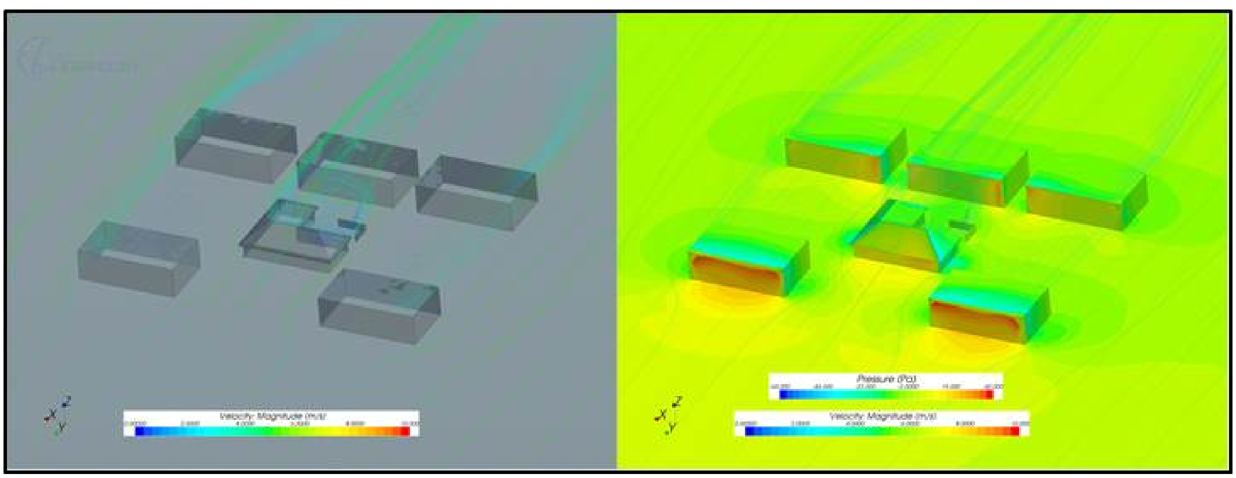


Figure 3.15: Star-CCM+ shows the external CFD simulation of velocity streamline (m's) and pressure distribution on building faced (Ps) (Figure presented in Project Report 2, Establishing External CFD Workflow)

3.9 Results of Final Ranking of CFD Software Products

The results of the final ranking provided a quantitative assessment tool to the ERDL/HNEI CFD research group for the selection of the CFD software products which be used during the remainder of this CFD research project. The final ranking included conclusions drawn from the review of vendor documentation, active (or lack of active) interactions with software support staff and active engagement with the particular CFD software user interface (benchmarking).

It must be noted that the final ranking was performed solely on the basis of workflow issues for the ERDL-HNEI workflow process. The derived final ranking uses a specific two-tier ranking approach with a unique generic framework that considers the needs and objectives of the ERDL-HNEI CFD research team. The final ranking should not be applied to general CFD workflow conditions and represents in no way an endorsement of certain CFD software products by the ERDL staff.

The summary of the final ranking results is provided in Table 3.2:

Table 3.2: Summary of the final ranking results for six candidate CFD software products

Criteria Group	Description of criteria group	possible	ANSYS - Fluent		STAR-CCM+		ACAD CFD		Cradle SD-Stream		Cradle SD-Tetra		Design Builder	
		overall weights	overall weights	achieved of total	overall weights	achieved of total	overall weights	achieved of total	overall weights	achieved of total	overall weights	achieved of total	overall weights	achieved of total
1	**Value for the CFD software product to the architecture and building market**	**15.0%**	13.7%	91%	10.5%	70%	12.1%	81%	13.0%	87%	8.1%	54%	12.3%	82%
2	**Support by vendor / expert network**	**15.0%**	7.1%	48%	14.3%	95%	10.9%	73%	12.5%	83%	12.5%	83%	10.0%	67%
3	**Workflow - summary**	**52.0%**	39.7%	76%	45.5%	88%	33.4%	64%	41.6%	80%	41.9%	81%	20.8%	40%
3.1	Ease of use / GUI	7.8%	5.3%	68%	6.7%	86%	7.4%	95%	6.9%	89%	6.9%	89%	7.4%	95%
3.2	Modeling / Import geometry	7.8%	4.7%	60%	6.9%	88%	5.3%	68%	4.7%	61%	3.8%	49%	4.3%	55%
3.3	Meshing	10.4%	7.1%	68%	9.9%	95%	6.1%	59%	8.5%	82%	7.1%	68%	2.5%	24%
3.4	Physics Models	18.2%	16.5%	91%	17.3%	95%	11.8%	65%	14.6%	80%	17.3%	95%	4.8%	27%
3.5	Post processing	2.6%	2.1%	80%	1.9%	73%	1.3%	50%	2.4%	91%	2.4%	91%	0.7%	29%
3.6	Customization / special functions	2.6%	2.0%	77%	2.1%	82%	1.3%	52%	2.0%	78%	2.0%	75%	1.0%	37%
3.7	Support for validation	2.6%	2.1%	80%	0.8%	30%	0.1%	5%	2.5%	95%	2.5%	95%	0.1%	5%
4	**Cost of Software**	**18.0%**	9.0%	50%	14.8%	82%	11.5%	64%	9.7%	54%	9.7%	54%	12.2%	68%
	Sum criteria 1 through 4 >>>	**100.0%**	**69.5%**		**85.1%**		**67.9%**		**76.8%**		**72.2%**		**55.3%**	

3.10 Selection of CFD Software Products for the EDL- HNEI CFD Research Project

On the basis of conclusions drawn from the review of the software product documentation, the hands-on experience with the CFD software during benchmarking tests as well as the results of the final ranking the following software products are selected for the subsequent analysis work of the ERDL–HNEI CFD project:

1. Star-CCM+ by Cd-Adapco
2. ScStream by Cradle USA Inc.
3. SC/TETRA by Cradle USA Inc.

To 1.: The CFD software Star-CCM+ by the company CdAdapco has been selected as a powerful and versatile CFD software tool. The decision to select Star-CCM+ over the ANSYS Fluent is based on the very pro-active cooperation with the academic program manager and account manager of CdAdapco. The software company has been very forthcoming with supporting all aspects of our CFD project work and has offered the CFD team with very favorable licensing conditions.

To 2 and 3.: The CFD software products scStream and SC/Tetra have both been selected. The CFD team could negotiate a favorable price for the lease of both software products. The CFD research team will be using both CFD software products in the upcoming external as well as internal CFD analysis work. Based upon experience the CFD team believes that both software products will favorably complement each other.

SECTION 4 - DEVELOP AND TEST FIELD MEASUREMENTS PROCEDURES

This section summarizes the work of developing and calibrating filed measurement procedures that were used in the validation for external CFD simulations - **Project Deliverable No. 3**. The objective of the project work summarized hereafter was to develop proficiency in validating CFD results with actual measurements in the field. So-called "shakedown" tests were conducted during which all procedures and phases of the overall verification process of CFD simulation results could be developed, tested and fine-tuned. The project work presented in this section included the following phases:

Phase 1- Selection of the test site and the test structure: The CFD research team identified two candidate test sites, where the wind movement and pressure distribution around a simple test structure could be measured. The first candidate site (Candidate test site A) was located next to the ERDL laboratory facilities on the University of Hawaii Manoa campus. This candidate test site offered the significant advantage of direct and short access from the ERDL facilities and therefore easy logistics. Scoping wind measurements of test site A, however, revealed significantly varying wind directions and velocities and therefore highly unsteady wind conditions. For the shake down tests CFD research team was seeking a site with relative steady wind direction and little wind flow obstruction. A second site candidate (Candidate test site B) was located close to the ocean at a distance of about nine miles from the ERDL facilities. The second test site featured the advantage of easy access and relatively few upwind obstructions, resulting in more steady wind directions and higher constant wind velocities. After considering alternatives of fixed temporary structures the research team decided to use a vehicle (e.g. a Toyota RAV 4 – SUV) as the test structure that creates obstructions to the wind. The vehicle had the advantage of easy and no-cost deployment.

Phase 2- Performing initial CFD scoping simulations: Initial CFD simulations were performed using most probable wind direction and speeds at the test site. The initial CFD runs used a simplified 3D-geometry of the test structure (e.g. RAV-SUV) and a coarser CFD computational domain and generic physical setting. The main objective of the initial CFD runs was to identify suitable locations of wind speed sensors and pressure tubing terminal units. The preferred locations of instrumentation around the SUV were areas of higher velocity and pressure gradients. The initial CFD results were also used to determine the predicted differential pressures around the test structure, so that the pressure transducers ranges could be selected correctly.

Phase 3- Selection of the instrumentation: Three properties were measured in the validation process: wind direction, wind speed and differential pressure. The team had ready access to six high sensitivity anemometers (wind speed sensors) and one weather station. The anemometers were used to measure wind velocity but not wind direction around the test structure (RAV-SUV). The weather station was used to measure the wind direction and velocity of the wind approaching the test structure. Records of wind direction and speed during tests collected by the weather station were used to determine the wind conditions for the finals CFD runs.

The research team had to select and procure suitable differential pressure transducers which offered the high sensitivity (e.g. low pressure ranges) required to detect the small pressure differentials around structures at the wind conditions anticipated for the finals test runs. The selection and procurement of suitable pressure transducers proved to be a considerable challenge, since the small pressure differentials on the building envelope under normal wind conditions require a very high resolution of pressures (e.g. low differential pressure ranges). It was found that there are few vendors who offer such low differential pressure transducers. After identifying about ten candidate differential pressure transducers the team selected two vendors, Setra (US company) and Halstrup-Walcher (German company), based on a favorable cost-benefit ratio and online customer reviews. The team selected a signal multiplexer by National Instruments (NI) as the data acquisition system. The data acquisition hardware was complemented by proprietary NI signal conditioning and analysis software.

<u>Phase 4- Preparation of the instrumentation in the laboratory:</u> The anemometers, differential pressure transducers and the data acquisition system were prepared and tested under laboratory conditions before being deployed in the field for the shakedown tests. The wind sensors and pressure transducers required correct signal excitation and the measured signals required calibration and signal condition. A simple ad-hoc wind tunnel was used to establish qualitative wind speeds and pressure differentials. The preparation and fine tuning of the instrumentation operation was finished in two stages, each following preceding field test days. The lessons learned from the first and second test days were used to identify shortcoming in instrumentation deployment, signal management and data analysis. The final instrumentation set-ups were used in the third and final day of field tests.

<u>Phase 5- Deployment of the instrumentation in the field and three test runs:</u> Tests were conducted at the site on three days, spread over a period of six weeks, between December 2013 and February 2014. On each of the three test days, important lessons were learned by the CFD research team. The team developed proficiency in test planning, test execution, instrumentation set-up and retrieval and data acquisition. The team developed a test check-list that will be of significant help to plan and conduct the remaining field test of the project and also serve to document field test procedures for future projects at ERDL. The field test concluded with a successful third day of testing when a comprehensive data set for two wind scenarios (e.g. different positioning of the test car relative to the approaching wind) was collected.

<u>Phase 6- Final field data analysis:</u> A comprehensive data analysis process was applied to the results of the third and final day of field testing. A variety of unsteady wind speed and differential pressure signals required the development and application of specific filtering procedures in order to produce statistically robust data sets to be used in the comparison of the field data and final CFD simulation results.

Phase 7- Final CFD simulations: The final CFD simulations depicted a more precise assessment of wind movement around the test structure (e.g. RAV-SUV) and of the pressure distribution on the test structure envelope. The final CFD simulations used the prevalent wind direction and speed recorded at the site during the field tests. These representative wind conditions were used as input variables, e.g. CFD boundary conditions. The final CFD also used a more refined mesh and 3D-model of the RAV-SUV. The final CFD simulations provided the data for the comparisons of the theoretical (calculated) wind velocity and pressure data with the actual field measurements of these properties. In addition to contoured property slices the final CFD simulations used small data grid to extract averaged CFD wind velocity and pressure data for those locations in the computational domains that corresponded with the actual locations of the sensor and pressure tubing terminals in the field tests.

Phase 8- Comparison of CFD predictions and result of field measurements: The results of CFD simulations and actual field measurements of wind velocity and differential pressures around the test structure were compared. The team found good correlations between CFD predicted wind velocities and the actual field measurements. The comparison of CFD predicted and actually measured differential pressures resulted in lower correlations between theoretical and actual data than detected for wind movement around the test car. The team identified some of the reasons for the diverging results between theoretical CFD predictions and actual filed test data. These lessons learned were documented and served as guidance for improved test procedures for the final part of the research program.

The following sections provide some illustrative results of the eight phases.

4.1 Phase 1- Selection of the test site and the test structure:

An initial scoping test suggested two candidate test sites. Test site one (Shakedown Candidate Test Site A) was located on a grassy area next to the School of Architecture building on the University of Hawaii Manoa campus. This site offered an easy deployment of the wind and pressure sensors. A second candidate test site (Shakedown Candidate Test Site B) was located on a vacant lot close to the Pacific Ocean, some nine miles away from the School of Architecture where the CFD laboratory is located. Figure 4.1 shows the two candidate test sites.

An initial wind velocity scoping measurement test at the test site A suggested that winds vary considerably in direction and strength, so it was deemed that the wind conditions at this site were not sufficient. The wind speeds coupled with the significant shifts in wind direction at this candidate test site were not considered appropriate for test purposes. The wind conditions at test site 2, Shakedown Candidate Test Site B, were found to be much more steady and predictable in terms of wind direction and speed. The disadvantage of site B was that the entire test set up had to be transported to a remote location and the site had to be secured from other users. Since the Site was a popular parking site for surfers this proved to be challenging, although at no time were surfers or other beach users unwilling to let the site be used for the test purposes. Consequently test site two (Shakedown Candidate Test Site B) was selected for the shake down tests.

A test structure was needed to provide obstructions to the wind flow and therefore create velocity and pressure gradients around the structure. It was found that a vehicle would serve as a mobile test structure and also would provide enough obstruction to the wind flow, while all sides of the structure could be easily accessible. The small SUV (RAV4) of a member of the CFD research team served as the test structure of the shake down tests. Figure 4.2 (a) shows the vehicle (SUV) in reality, with several wind sensors arranged around the vehicle, and (b) as the CAD model for the CFD analysis.

The selected test site, which was located close to the ocean, was considered and "cost effective substitute" of a wind tunnel. The test site with its relatively unobstructed wind approach made it possible to perform shake-down testing and fine-tuning of the CFD workflow and field measurements.

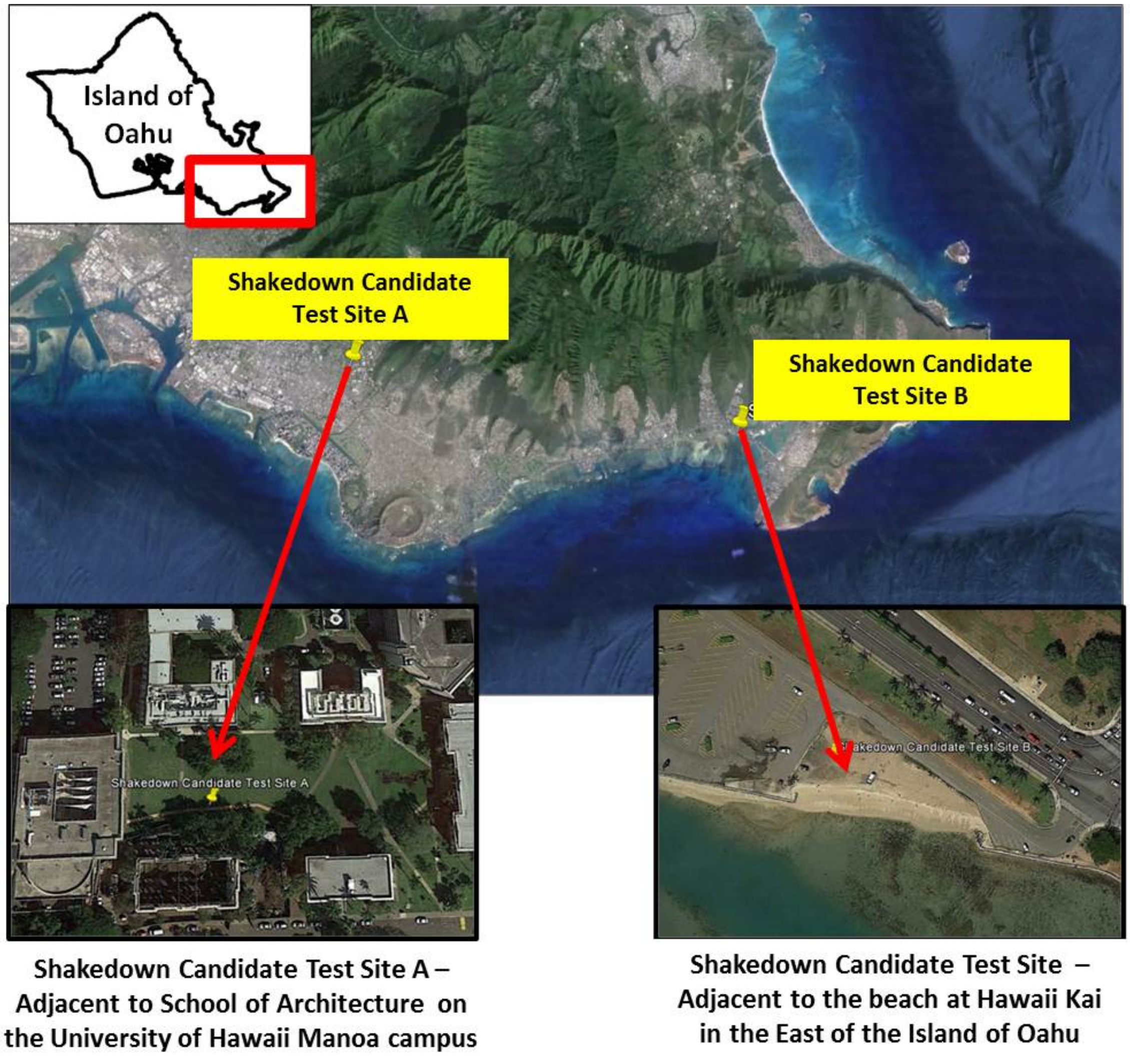


Figure 4.1: Two Shakedown Candidate Test Sites A and B (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.2 Phase 2- Performing initial CFD scoping Simulations

The initial CFD analysis allowed for estimating the airflow movement around the test car and the resulting the pressure distribution on the car surface. Figure 4.3 shows the initial CFD test runs to determine approximate pressure gradient. Having initial data about the pressure differentials allowed the acquisition of appropriate pressure differential transducers. A main objective of the initial CFD tests was to determine where to locate the anemometers and differential pressure tube terminals. Based on the initial results, locations close to the edges of the car, where the air accelerates as it passes the obstruction Figure 4.4 shows an example of the result of the initial CFD runs that used expected wind approach direction and speed based on a review of historical weather data at the site. The locations of instrumentation during the tests were based on such images, whereby different images were created for the number of chosen scenarios.

"Test structure" used for the shakedown testing. A RAV (SUV) served as an effective mobile structure around which to carry out the wind experiments

The "Test structure" shown as the CAD model for the CFD analysis

Figure 4.2: The test structure (= RAV-SUV) used for the shakedown testing B (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

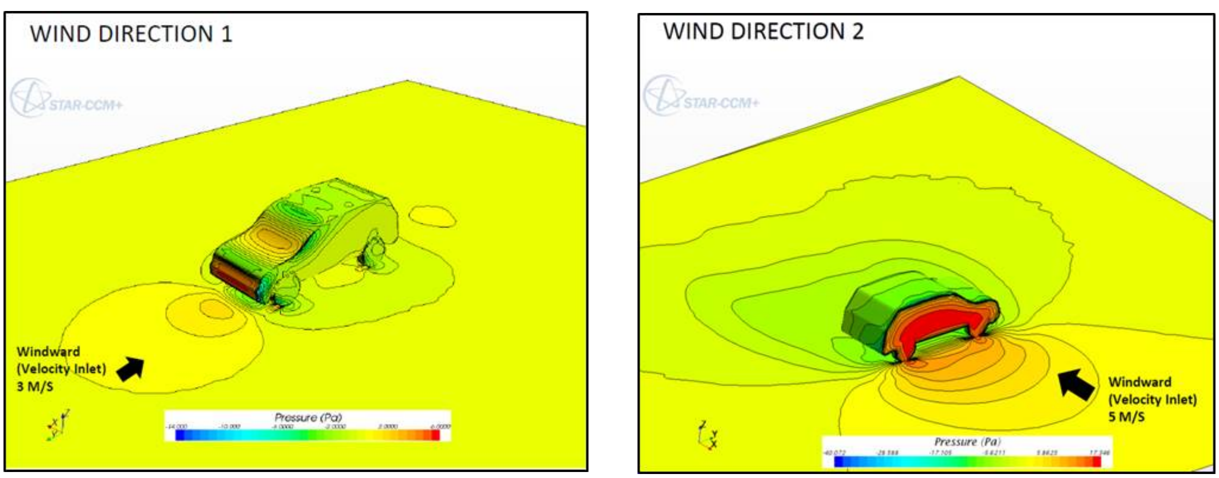


Figure 4.3: Initial CFD analysis results showing pressure distribution on the car surface and the ground (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Initial CFD simulations proved to be very helpful in preparing the shake-down testing and validating the CFD simulation with measurements around a simple, yet full scale test structure.

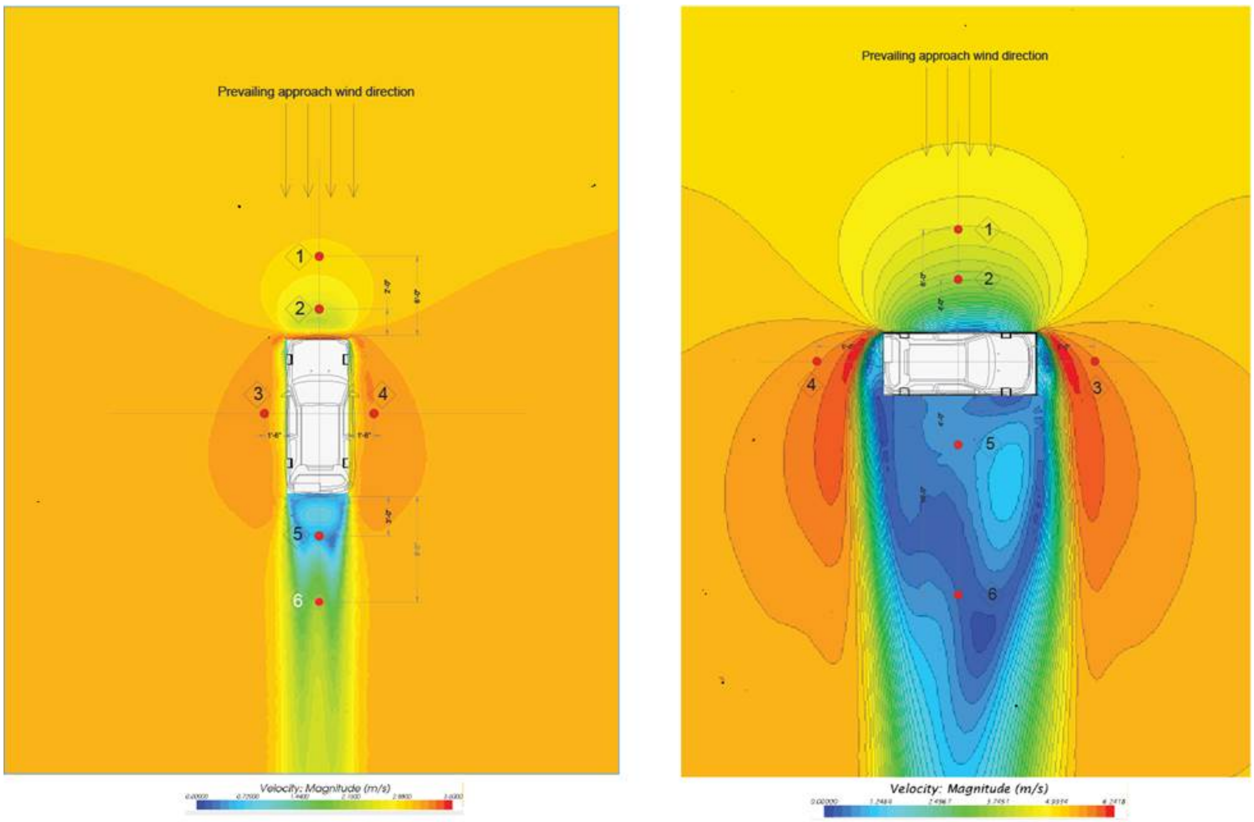


Figure 4.4: The example of sensors locations mapping on the initial CFD analysis for measuring test scenarios. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.3 Phase 3- Selection of the Instrumentation

The following instrumentation was used in the field measurements.

- Anemometers: The anemometers used in the measurements were Degree Controls Accusense F900-0-5-1-9-2 with the XS blade, which have a range of zero to 5 m/s air speed and an accuracy of 0.5 % of reading or 1% of full scale. Figure 4.5 shows the anemometer sensor with the data conditioning unit (the cylindrical unit in the figure). The cylindrical unit connects to the external

port on the Onset U12 Data Logger (bottom on Figure 4.5 and labeled “CFD-05”) via the grey cable. The anemometer has the following measurement parameters:
- Velocity range: 0.15-5m/s
- Accuracy: ± 5% reading or ±0.05m/s (10fpm)

- Differential pressure transducers: After interviews with vendors as well as performance and cost comparisons to other products the CFD research team selected two transducers. Figure 4.6 shows the two differential pressure transducers:
  - Setra Model 264 Differential Pressure Transducer: range of zero to 0.1 inches of water column (zero to 25 Pascals) and 0.5% accuracy of full scale (FS)
  - Halstrup-Walcher P26 Pressure Differential Transducer: range of 0 to 0.055 inches of water column (zero to 10 Pascals) and 0.5% accuracy of full scale.
    Other equipment used in the pressure measurements:
    Pressure tubing terminals: The high and low pressure tube connectors of the pressure transducer were connected to tubing terminals by means of 3/16 inch ID PVC tubing. Figure 4.7 shows one pressure tubing terminal (1 ½ inch ID PVC Schedule 40 pipe, with numerous 3/16 ID holes drilled into the pipe walls)
- Weather station (for the measurements of the background wind and weather conditions): Figure 4.8 shows the weather station, an Onset HOBO U30 device, powered by its 4-volt, 10-AHr on-board battery and 1.2W solar panel. Data recording was U30 internal and the recorded data was subsequently downloaded with USB cable to a laptop computer, which used Hoboware software. The U30 weather station consists of two sensors: wind speed sensor S-WSA-M003 and wind direction sensor S-WDA-M003. The elevation of the wind direction and speed sensors above ground was 9 feet.

Data logging and acquisition:

- Data logger for anemometers: Six Hobo U12 data loggers were used to record wind speed. The data loggers were powered by a 12V customized battery packages.
- Data acquisition for differential pressure transducers: A National Instruments USB-6341 (Figure 4.9) multiplexer system was used. Excitation of sensors was not provided by the multiplexer; so sensors were excited by AC to DC power adapters, which drew their electric power from a small gas-powered generator at the test site.

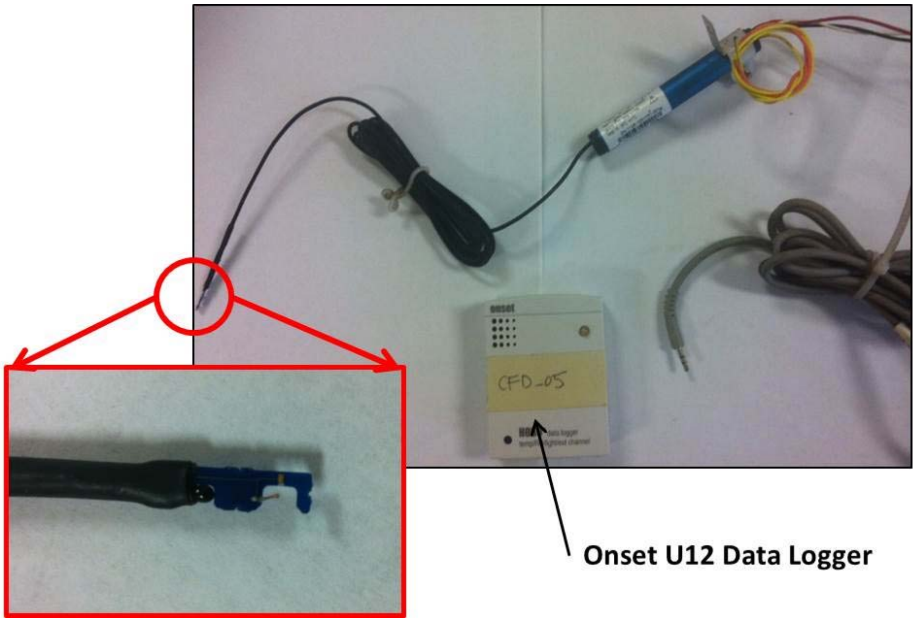


Figure 4.5: DegreeC F900 Anemometer. (shown with the cylindrical signal conditioning unit and the U12 data logger) (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Setra Differential Pressure Transducer Model 264

P26 Halstrup-Walcher Pressure Differential Transducer

Figure 4.6: Two types of differential pressure transducers used in the research work (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Figure 4.7: Pressure tubing terminal;

Two pressure tubing terminals were connected via clear 3/16" ID pressure tubing to the high and low pressure outlets of the differential pressure transducer. The purpose of the pressure tubing termials was to equalize pressures detected over a the length of about one foot by compensating for small localized eddies. For longer period installations the tubing terminals also avoided objects, rain and insects from entering the pressure tubing. B (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

The selected test instrumentation performed well. The "hot-wire" anemometers were available to the CFD project team through previous experiments and the sensors were operating well. In future other anemometers might be used for external wind measurements around buildings, which will also provide information about wind direction. The differential pressure transducers performed well and the low pressure difference could be detected.

Figure 4.8: Mobile weather station; used during the shake down testing at the baeach side test side. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Figure 4.9: Data acquisition for differentail pressure transducers

(Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.4 Phase 4- Testing the Instrumentation in the Laboratory before Field Deployment

The instrumentation was thoroughly tested in the laboratory before being used in the field. The instrumentation and data acquisition tests in the laboratory included the following:

- Testing the data connection between sensors and data acquisition, including sensor excitation and reading data.
- Testing of the differential pressure transducers in a small laboratory wind tunnel
- Testing the anemometers in a small laboratory wind tunnel
- Checking the data consistency after downloading the data for subsequent processing.

Figure 4.10 shows the differential pressure transducers being prepared and connected to data acquisition multiplexer.

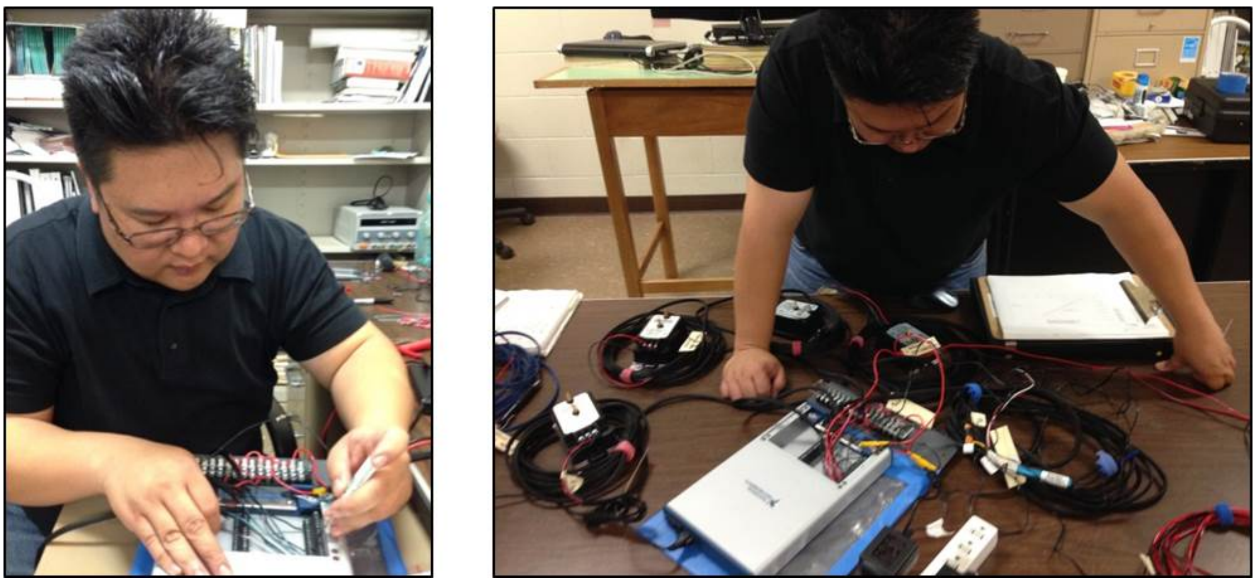


Figure 4.10: Instrumentation and dat acquisition equipment prepared for field test deployment (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.5 Phase 5- Deployment of the Instrumentation in the Field and three Test Runs

Test runs were conducted at three days over a six week period. During every field tests new measurement scenarios were selected before the tests in order to allow a wide range of instrument settings to be tested. The field tests were an essential part of the shake down tests since instrumentation issues that arose by operating in a “non-lab” environment and therefore not ideal environment could be identified and corrected before the subsequent tests. The team could gain important logistical experience in running, preparing and conducting tests. The logistic issues included creating check lists for test equipment, securing the test site for the test equipment and conducting expeditious testing.

The test runs included the following:

- Securing of the test site in the morning of the tests and coning / taping off the test site so that no other vehicvle could be parked at the site. The site was an approximately 1 acre gravel area diretcly adjacent to the Pacific Ocean. The site had no adjacent strcutures and only a few coconut trees on the land side of the site, therefore the wind approach was basically unobstrcuted within several hundred meters.
- Detecting the wind approach direction and orienting the test car (RAV 4 – SUV) to the desired orientation relative to the wind approach direction. (see Figure 4.11 which shows the placement of the test car (RAV4-SUV) on the testsite.
- Placing the instrumentation around the test SUV in accordance with several test scenarios for which initial CFD simuations were conducted before the field tests in order to detemine preferred locations of the anemometers and pressure transducers. (see Figure 4.12)
- Conducting the measurements of specific scenarios over a period of time, usually 20 to 30 minutes. (see Figure 4.13)
- Breaking down the test instrumentation and returning the access to test site for public use.

Figure 4.14 shows the CFD team which participated in the field tests, CFD simulations and data acquisition and analysis.

The three full days of shake-down test were conducted within a six week period. In between the test days, instrumentation set-up and data acquisition procedures were fine-tuned. The resulting instrumentation deployment and use, tested during the three days of shake-down tests , proved to be very helpful for the preparation and conduction of the final external wind measurements at the selected building on the University of Hawaii Manoa Campus

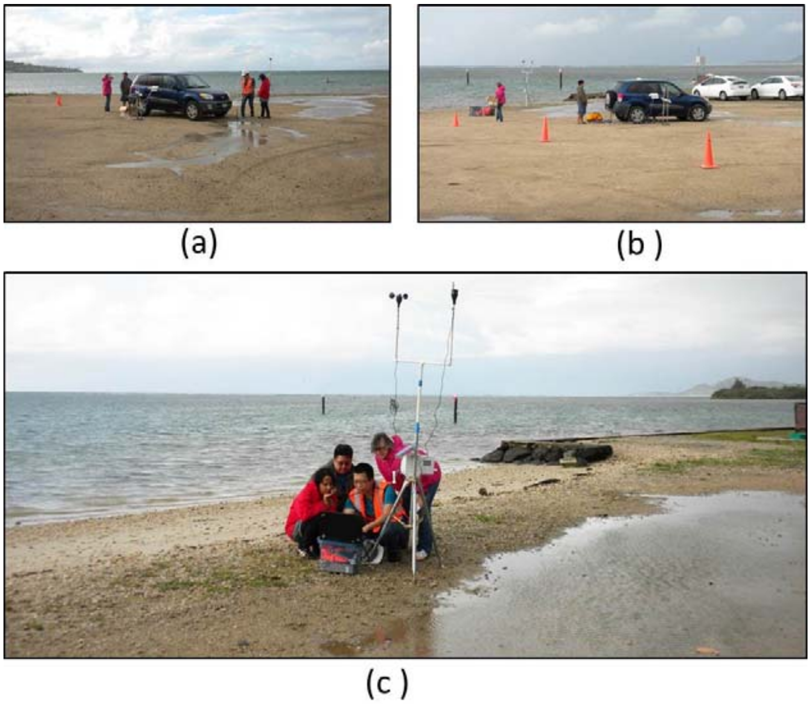


Figure 4.11: Field tests at the site adjacent to the Pacific Ocean; (a) and (b) show placements of the test object (RAV SUV) according to the prevailing wind approch direction; (c) placement of the weatherstation to measure the wind direction and speed. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

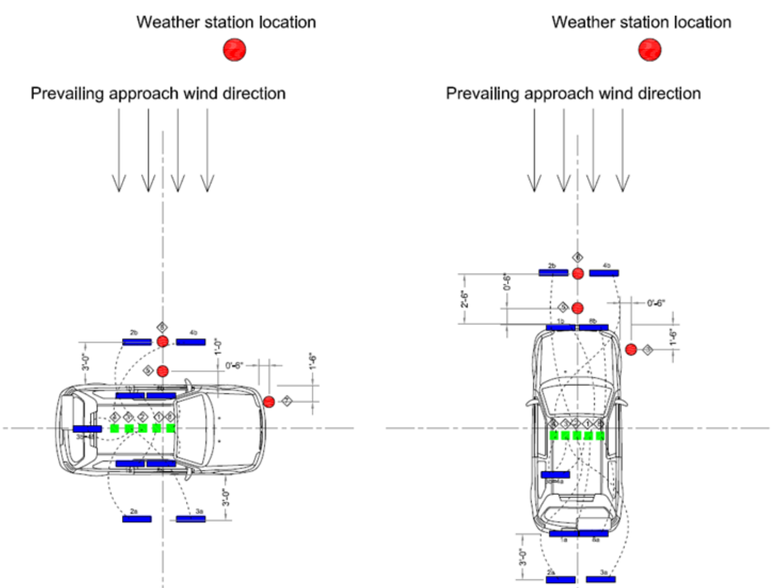
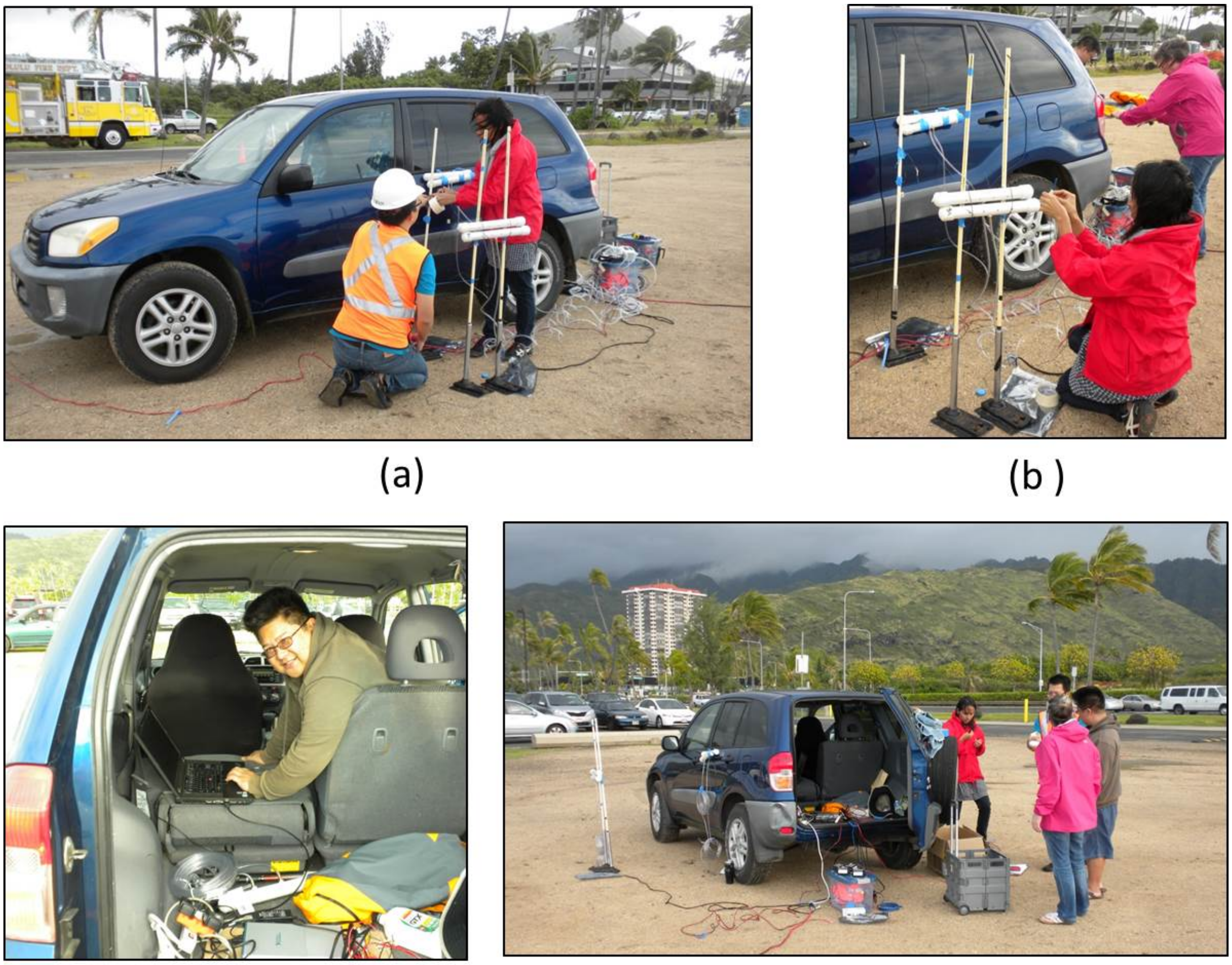


Figure 4.12: Example of placement instrcutions for the test scenarios; the preferred locatiosn of the test sensors around the test car (RAV4-SUV) was determined in initial CFD simulatiosn before the field tests (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)



(a) (b) (c) (d)

Figure 4.13: Field tests at the site adjacent to the Pacific Ocean; (a) and (b) show the installetion of the anemometer s and the pressure tubing terminals around the test car (RAV4-SUV), (c) shows the data acquisition equipment which was placed inside the car, (d) shos the test car with instrumentaion placed around it, the AC generator for test equipment is outside the picture and was about 30 feet away from the test car. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Figure 4.14: The CFD team members which contributed to the tests; In photo, from left to right: Sanphawat Jatupatwarangkul, D.Arch, (active in research team through December 2013, now faculty in his native Thailand); Charles Siu, D. Arch candidate; Aarthi Padmanabhan, D. Arch; Tuan Tran, D.Arch; Eileen Peppard, M. Sc.; Christian Damo, B.S. Electrical Engineering candidate; Manfred J. Zapka, PhD, PE; Phyllis Horner, PhD. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.6 Phase 6- Recorded Field Data Analysis

The data recorded in the field tests was downloaded for subsequent data analysis. Data acquired by the NI USB-6341 multiplexer and processed on SignalExpress software was exported as a tab separated text file. Data needed processing before being uploaded to a PostgreSql database. Several data filters were applied to select which data point was to be used for correlations between the approaching wind direction and speed and the resulting air speed and pressure around the test car. This means that data filters were used to only consider air movement and pressures around the car resulting from wind approach directions of, as an example, 225 degrees (+/- 5 degrees) and wind approach speeds of > 2 meters per second.

Figure 4.15 shows a typical data recorded from the weather station, showing the wind speed over the time of data recording, the wind approach direction and speed in a wind rose diagram and a wind direction histogram. Figure 4.16 shows an example of data correlation between incident wind speeds and differential pressures generated around the test car.

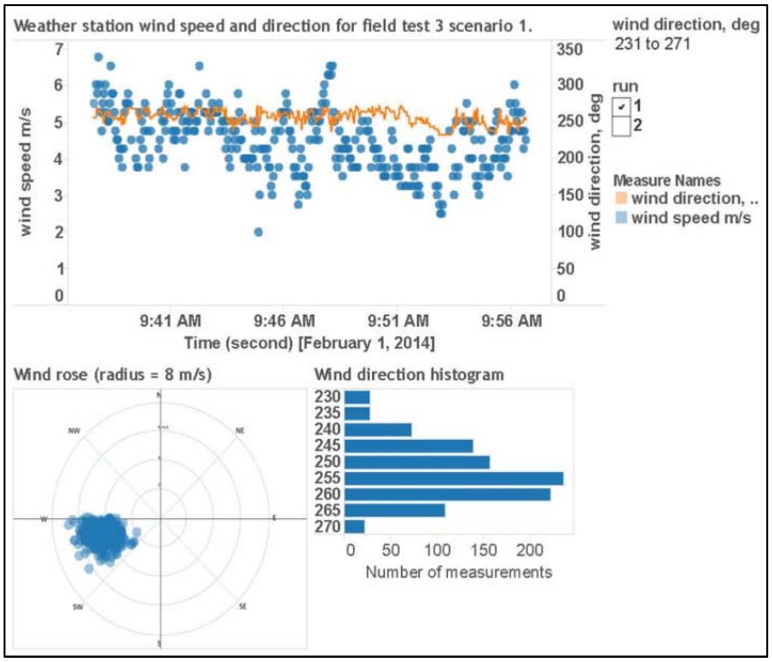


Figure 4.15: Data collected for approach wind direction and speed for third test day – (Note that the wind direction is from the South-East, which is referred to as "Kona" wind conditions in Hawaii)

(Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

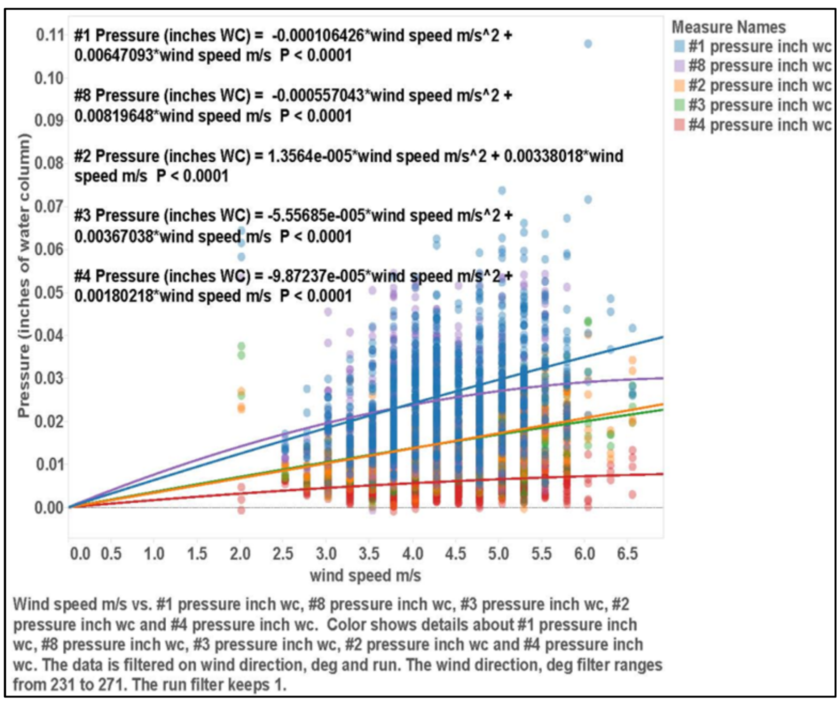


Figure 4.16: Data analysis of filtered data points

The depicted data analysis shows the correlation between incident wind and the resulting differential pressures

(Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.7 Phase 7- Final CFD Simulations

The final CFD simulations were performed with a detailed 3D-model of the test car (RAV-SUV). Figure 4.17 shows the rendered 3D-model (a) and the surface mesh of the test car (b).

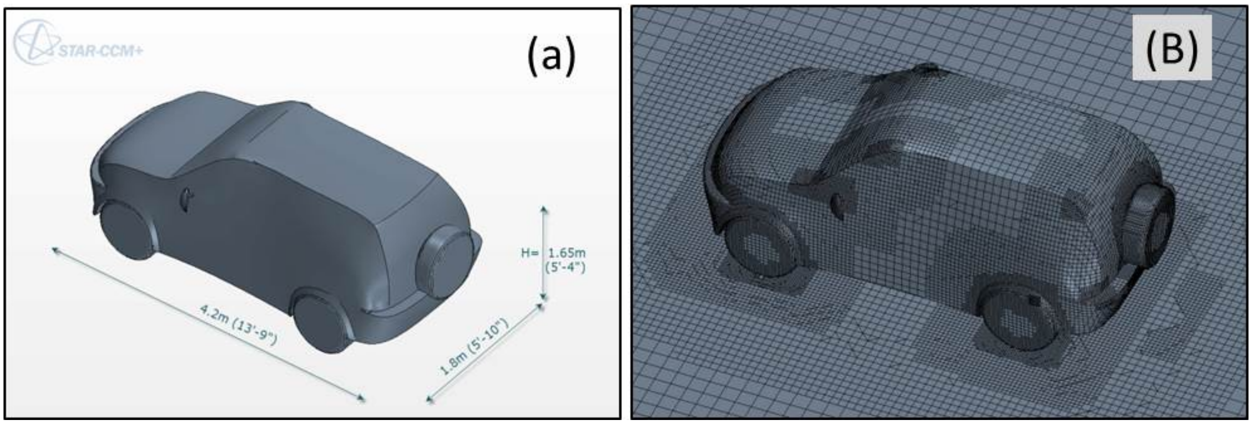


Figure 4.17: 3D-model of test car (RAV-SUV) for final CFD simulation; (a) rendered 3D-CAD model, (b) surface mesh of 3D-model (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

The 3D-model was placed into a virtual wind tunnel (e.g. computational domain), which is depicted in Figure 4.18. Also depicted in Figure 4.18 is the mesh created of the domain. The mesh had a higher resolution in the vicinity of the central area of interest, where the test car was located.

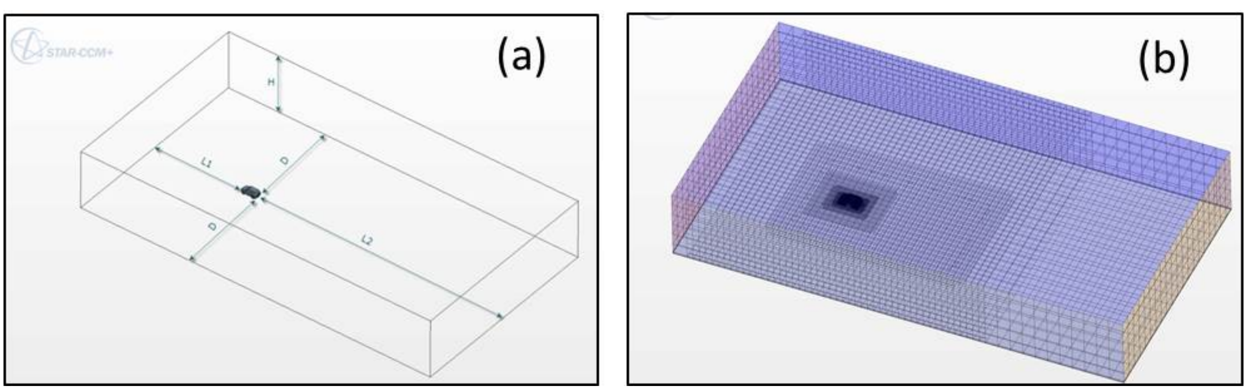


Figure 4.18: Illustration of the virtual wind tunnel (e.g. computational domain) for the final CFD simulations of then test car (RAV-SUV); (a) dimensions of the virtual wind tunnel, (b) mesh used showing high resolution in the vicinity of then central area of interest (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

The results of the final CFD simulations are illustrated showing resulting air speed patterns around the test car for wind approaching, for wind approach from the side (Figure 4.19) and from the front (Figure 4.20). Figures 4.19 and 4.20 show color contoured maps for air speed distribution around the test car.

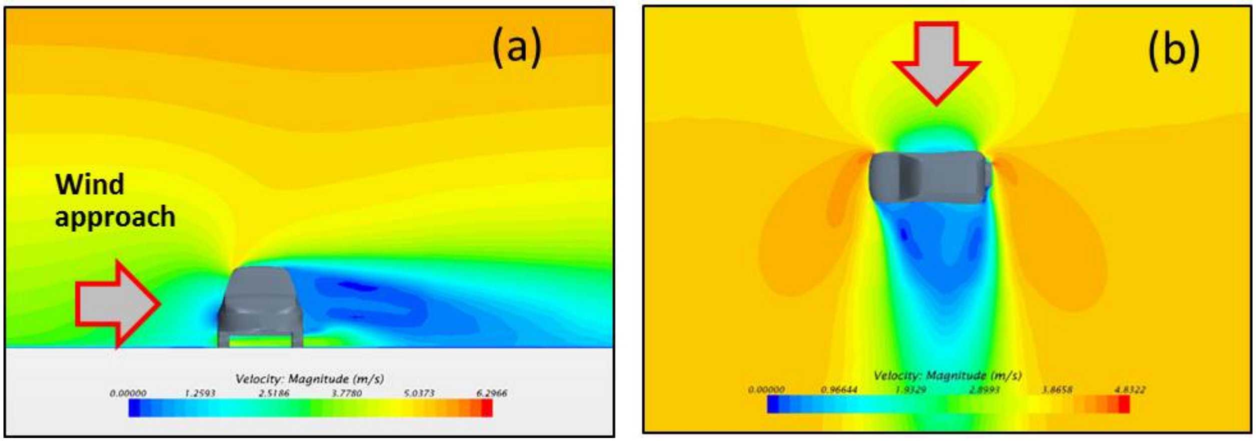


Figure 4.19: Air speed distribution around the test car due to wind from the side, (a) side view and (b) view from the top. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

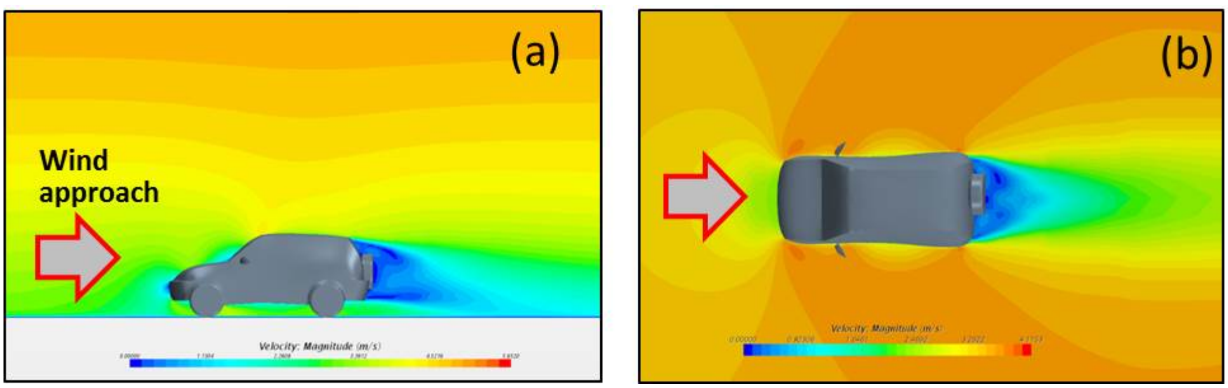


Figure 4.20: Air speed distribution around the test car due to wind from the front, (a) side view and (b) view from the top.

For a detailed assessment of differential pressures between two distinct points adjacent to the test car a small-grid probe (dimension of 1 x 1 feet) was used to extract CFD results for that specific location. The use of the small-grid probe is illustrated in Figure 4.21.

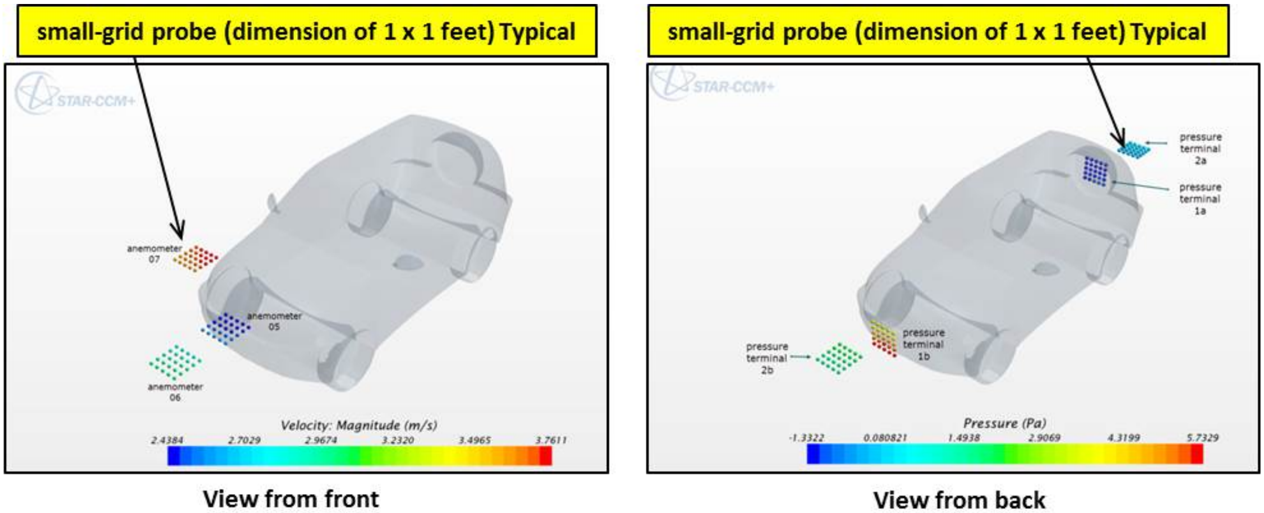


Figure 4.21: Small-grid probe (dimension of 1 x 1 feet) to determine exact pressure readings from the CFD results. (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

4.8 Phase 8- Comparison of CFD Predictions and Result of Field Measurements

The wind velocities and wind-induced differential pressures obtained from the field tests and final CFD simulations results were compared for validation of the CFD runs. Wind velocity data suggested a good correlation between field measurement and CFD prediction. For differential pressure data, however, the correlations were not as good as for wind data. Table 3.1 shows the data correlation for one of the test runs as an example.

The discrepancy between differential pressure results obtained in field measurement and CFD simulations suggested the following actions might be considered for the final CFD simulations at the selected building on the University of Hawaii Manoa campus:

- Apply appropriate filtering of data for given wind direction so that turbulence-related approach wind phenomena can be eliminated.
- Review of the settings of the CFD models such as roughness length, the wall-function coefficients, and the configuration of the pressure probes for extracting data.

The predicted CFD wind movement pattern showed good to very good consistency with actual measurements of wind speed and patterns around the test car. The CFD predictions for differential pressures around the test car did show smaller data consistency with actual pressure measurements.

Table 3.1: Field measurement and CFD result comparison of run 1 (approach wind perpendicular to longitudinal axis of the car). (Figure presented in Project Report 3, Develop and Test Field Measurement Procedures)

Sensor ID	Unit	Measurement	CFD	Difference (%)
Anemometer 5	m/s	1.30	1.31	**1.3%**
Anemometer 6	m/s	2.73	2.77	**1.6%**
Anemometer 7	m/s	3.76	4.02	**6.9%**
Differential Pressure (1b vs.1a)	Pa	6.57	5.78	**-12.0%**
	inch w.c.	0.026	0.023	
Differential Pressure (2b vs.2a)	Pa	3.77	2.13	**-43.4%**
	inch w.c.	0.015	0.009	

SECTION 5 - EXTERNAL CFD SIMULATION & FIELD VALIDATION FOR SELECTED BUILDING

Section 5 describes the CFD simulations of wind movement around a selected building on the University of Hawaii Manoa campus. Section 5 summarizes important findings which has been presented in project report of the Deliverable 4.1 (Draft and 4.2 (Final). The Keller Hall building was the selected building for the external CFD simulation work. CFD theoretical predictions of wind and pressures around were validated through full scale measurement at and around the building. The work on this study was carried out over a period of five months from May to September 2014. Keller Hall is a partially ventilated building and after the completion of the external CFD investigation the building will also be used for the following internal CFD investigations. Figure 5.1 shows Keller Hall building:

The project work to simulated and validate external wind movement around Keller Hall, presented hereafter in Section 5, was carried out in the following phases:

<u>Phase 1: Initial CFD Simulations:</u>

- Carrying out initial CFD simulations to determine preferred locations of instrumentation (Section 5.1)

<u>Phase 2: Field measurements:</u>

- Installing a weather station on Keller Hall's roof to obtain long-term information of weather and wind conditions at the site (Section 5.2)
- Conducting measurement of wind speed distribution around Keller Hall building on two test days (Section 5.3)
- Conducting measurements of external pressure differential across Keller Hall building over three month of recording (Section 5.4)

<u>Phase 3: Final CFD simulations:</u>

- Developing a detailed 3D-CAD model of Keller Hall and adjacent buildings; the extent of the 3D-model and the meshing around the central area of interest (e.g. Keller) was obtained through benchmarking test (Section 5.5)
- Carrying out CFD simulations using averaged data of measured wind conditions at the site as input (Section 5.6)
- Post Processing of CFD data (Section 5.7)

<u>Phase 4: Discussion of Results and Conclusions:</u>

- Results of the field measurements (Section 5.8)
- Comparison of CFD simulation results and field measurements (Section 5.9)
- Conclusions and recommendations (Section 5.10)

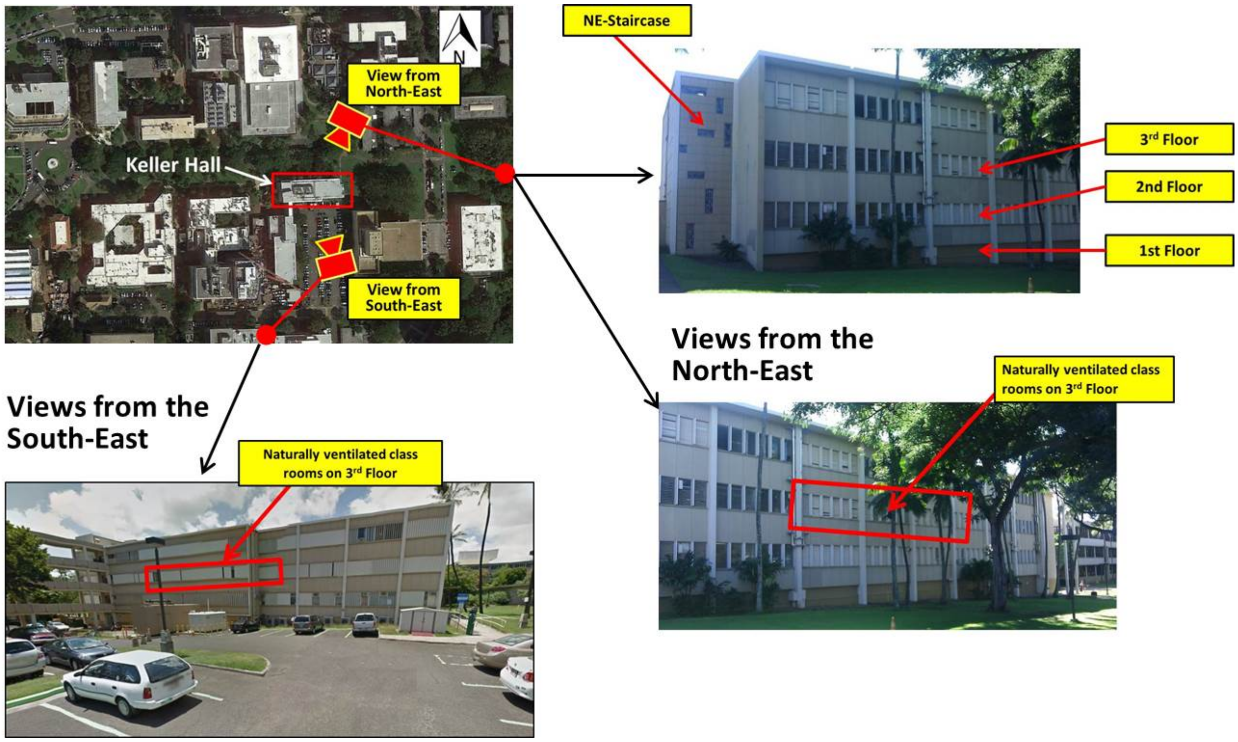


Figure 5.1: Selected building for external CFD investigation – Keller Hall, on the University of Hawaii Manoa campus. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.1 Phase 1 Initial CFD Simulations

The initial CFD simulations provided an approximate description of the wind regime around the Keller Hall building. The results of the initial CFD simulations were used to identify preferred location to place anemometers around the building for wind speed measurements. The 3D-model for the initial CFD simulations was an approximation of Keller Hall building. This approximate geometry was sufficient in order to obtain the initial wind patterns around the building. 3D models of Keller Hall and neighboring buildings (Physical Science Building, Kennedy Theatre, Henke Hall, Bilger and Bilger Addition, Hawaii Institute of Geophysics, etc.) were created in the SketchUp software. The terrain surrounding the modeled buildings was extended horizontally to increase the size of the computational domain. Figure 5.2 shows the CFD model of Keller Hall and surrounding buildings, e.g. the central area of interest inside the domain.

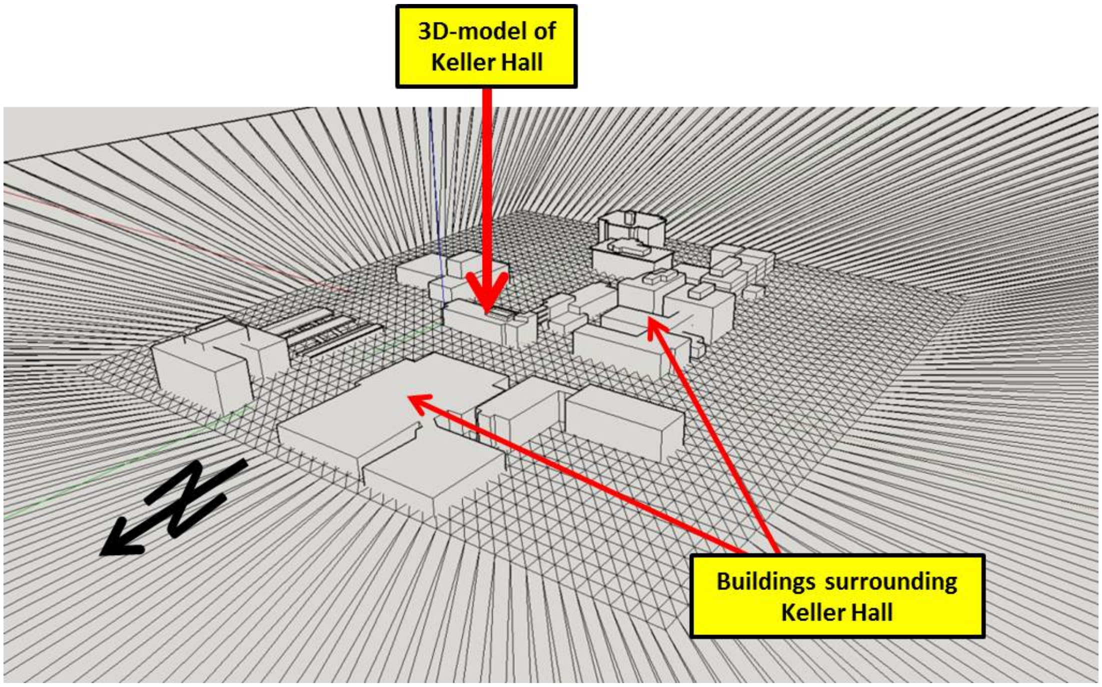


Figure 5.2: CFD model used for initial CFD runs. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The CFD settings used in the initial external air flow simulations included the following:

- Using previous weather data and selecting a northeast wind approach direction at 4.4 m/s (9.8mph) at the height of 25m (82 feet) above the ground.
- The size of the computational domain for the initial CFD run is 1500m x 1500m x 300 m, the ground was assumed as flat.
- The number of the cells in the computational domain was 2,424,267 cells
- Steady state RANS simulation with Realizable k-ε (Rk-ε) turbulence model.

Figure 5.2 shows the predicted wind velocity distribution around Keller hall and adjacent buildings obtained through the initial CFD simulations. The selected locations of wind speed sensors for the field measurements are depicted with red dots in Figure 5.3.

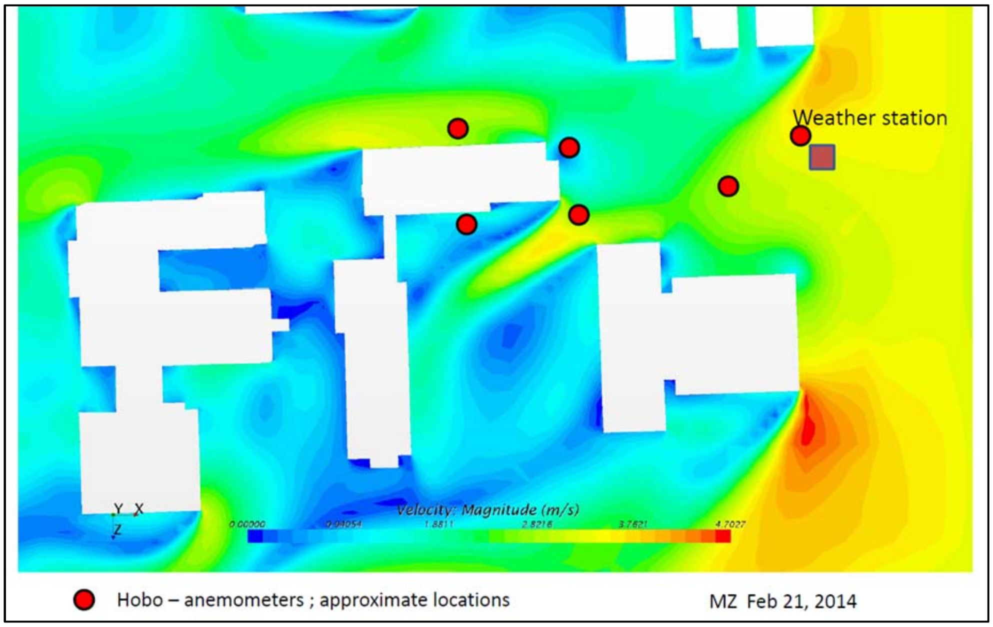


Figure 5.3: Results from initial CFD simulations and identified locations of sensors for wind velocity measurements. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.2 Phase 2 Field measurements - Installing a Weather Station on Keller Hall

A weather station was installed on the roof of Keller Hall and weather data, including wind direction and speed, was recorded over the period of five months. Figure 5.4 shows the Onset HOBO U30 weather station. The preferred location of the weather station was close to the edge of the roof in order sample wind conditions within a little disturbed wind flow regime. This preferred location of the weather station at the edge of the roof was not approved by the University facility representatives and therefore the weather station had to be located more towards the interior of the Keller Hall roof. A CFD analysis was conducted to determine the degree to which the wind flow was altered for the final location of the weather station. It was determined that the wind flow conditions at the final location of the weather station were acceptable. Figure 5.5 illustrates the CFD results of wind flow around the Keller Hall roof area.

Figure 5.4: Weather station installed on the roof of Keller hall for long term weather recording.

(Figure presented in Project Report 4, External CFD Simulation and Field Validation)

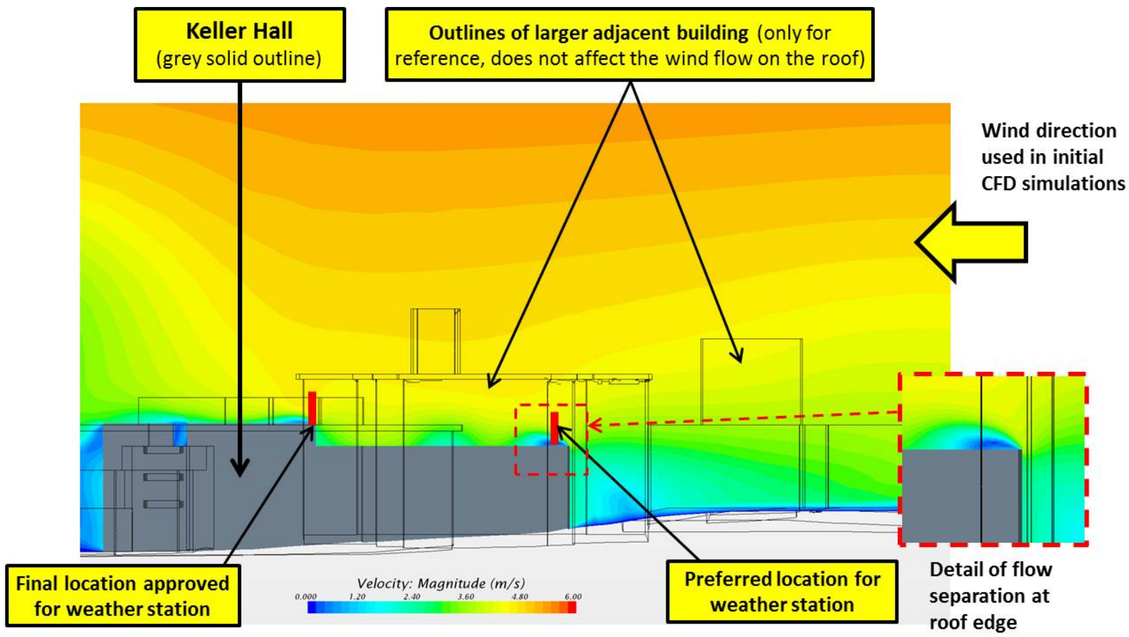


Figure 5.5: CFD simulation of wind flow patterns at or around the Keller Hall roof area. It was determined that the alternative location of the weather station yielded acceptable wind flow conditions to serve as the longer term wind measurement station. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.3 Phase 2 Field measurements - Conducting measurement of wind speed distribution around Keller Hall building

Wind speed measurements around Keller Hall were conducted on two days of data recording. On both of these days one weather station and six anemometers were placed around Keller Hall, according to the locations depicted in Figure 5.3. The locations of the anemometers around Keller Hall are illustrated in Figure 5.6.

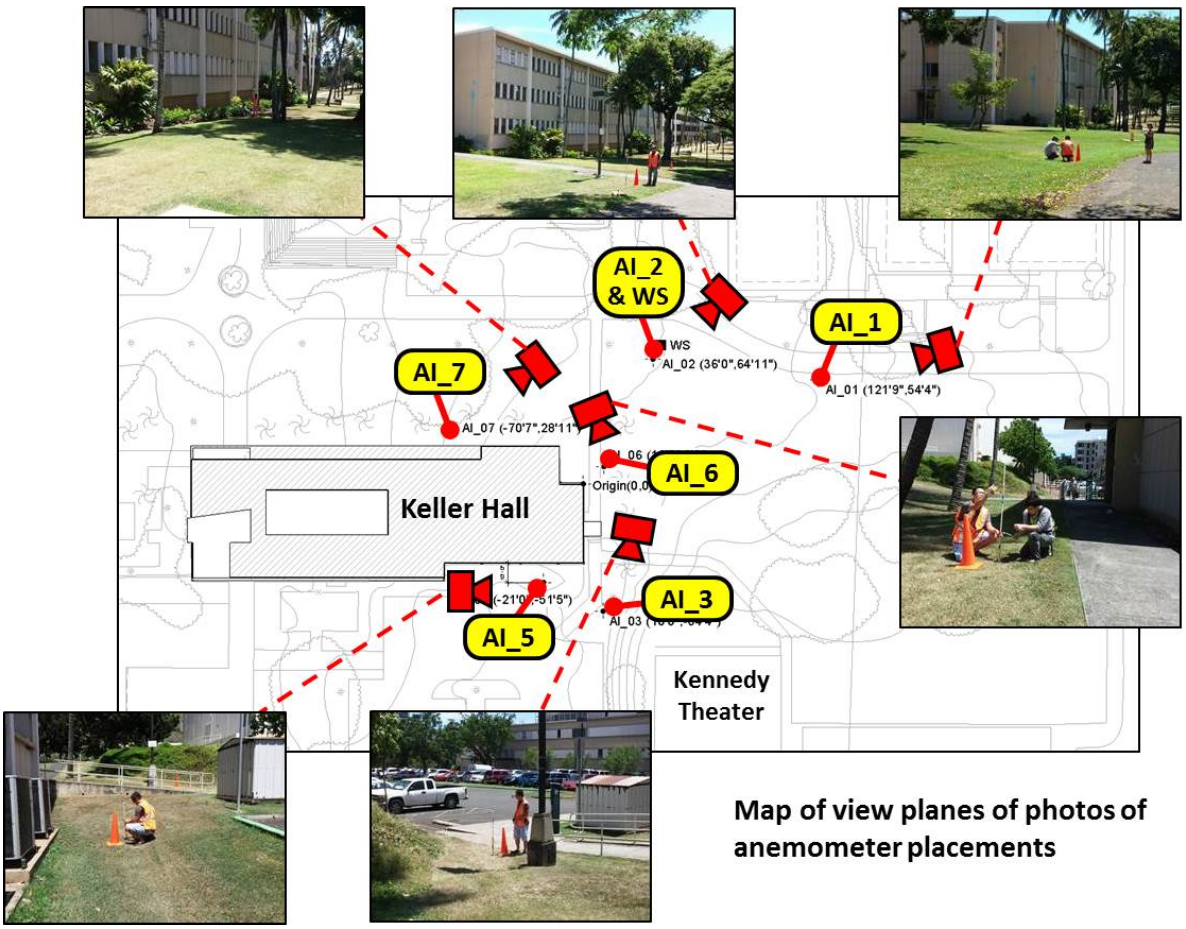


Figure 5.6: Locations of anemometer placements around Keller Hall (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

Six hotwire anemometers were used for the measurement of wind speed around the building. Airspeeds were measured using Degree Controls Accusense hotwire anemometers model F900-0-5-1-9-2 with the XS blade which has a range of 0 - 5 m/s air speed and an accuracy of 0.5 % of reading or 1% of full scale.

Instruments were setup on the North and East side of Keller Hall at locations that had been identified by the initial CFD simulations. Voltages from the sensors were measured by National Instruments USB-63341 data acquisition device (multiplexer) using National Instruments Signal Express software on a laptop computer. Anemometers were mounted on a wire extending from a vertical stand at a height of three feet above the ground.

5.4 Phase 2 Field measurements - Conducting external pressure differential measurements across Keller Hall building

Differential pressures between the North and South facades of Keller Hall were measure at four locations over a period of three months. The differential pressures measured could be correlated with the wind approach direction and speed measured by the weather station. Figure 5.7 shows a cross section of the third floor of Keller Hall with the naturally ventilated class rooms 302, 313 and 314 and the four locations of placement of pressure tubing terminals.

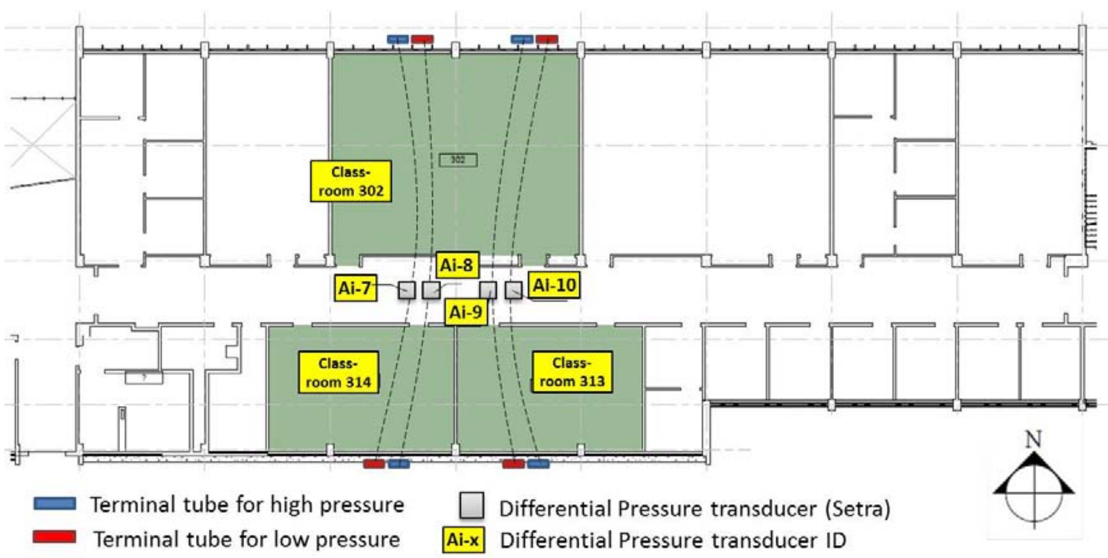


Figure 5.7: Locations of differential pressure measurements in Keller Hall (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The differential pressure transducers were connected to two pressure tubing, one for the higher pressure port of the transducer and one for the lower pressure port. The two pressure tubing sections were approximately 30 feet long each and the transducer was located in the middle between the tubing sections. Pressure tubing terminals were installed at the free ends of the pressure tubing. Figure 5.8

shows a typical instrument set to measure differential pressure, including differential pressure transducer, two pressure tubing sections and two tubing terminals.

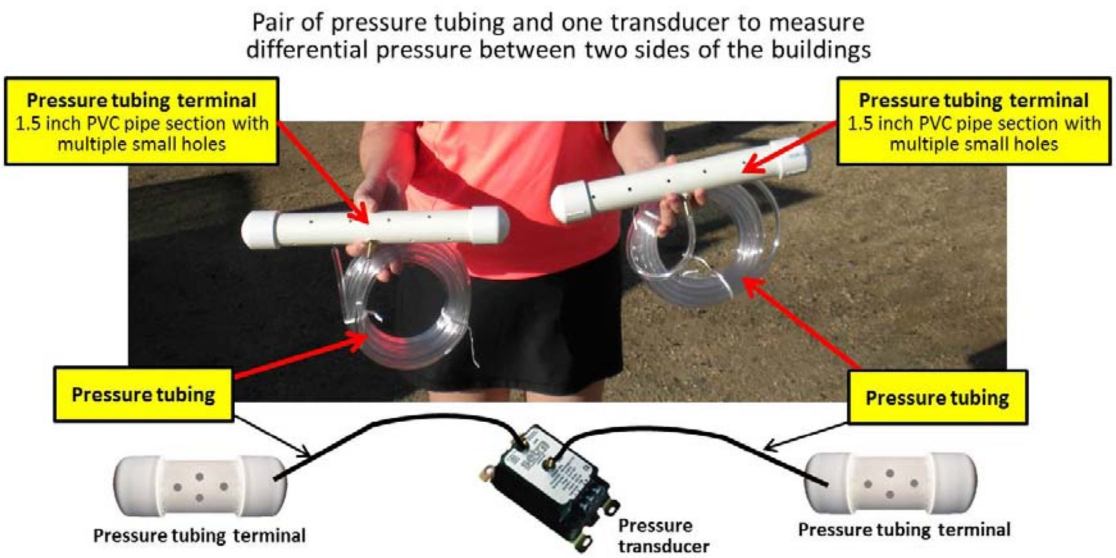


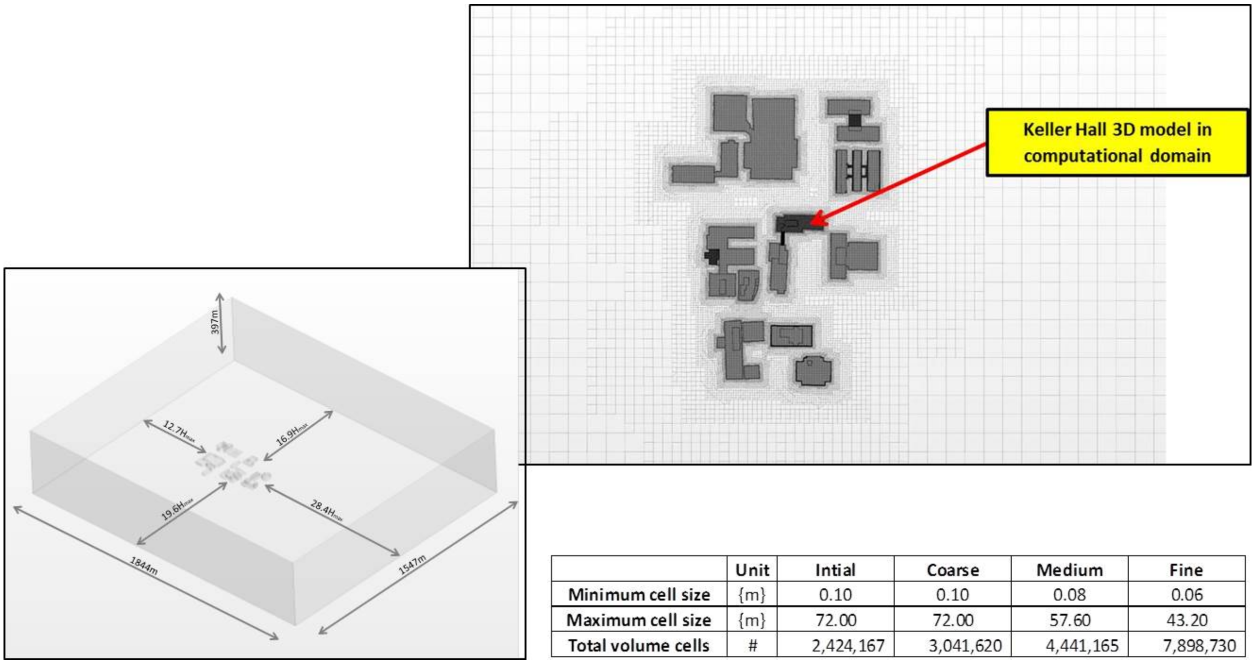
Figure 5.8: Instrumentation used for the differential pressure measurements. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.5 Phase 3 Final CFD simulations- Developing a detailed 3D-CAD model of Keller Hall and adjacent buildings

For the final CDF simulations the computational domain of the initial CFD runs was refined and expended. Figure 5.9 shows the computational domain for the final CFD simulations along with the surface mesh. As indicated in Figure 5.9 three sizes of mesh were used for CFD benchmarking in order to determine the effect of a higher resolution mesh on the convergence of the solution.

5.6 Phase 3 Final CFD simulations- CFD simulations using averaged data of measured wind conditions

The actual wind direction and speed conditions found at the site will be used for final CFD simulation. Since the wind direction and speed fluctuated due to gusts and other short term and localized wind occurrences, a statistically correct representative descriptor of wind direction and speed had to be determined. Due to the random nature of the wind fluctuation, a normal distribution was used in the analysis of the measured wind direction and speed at the test site. From the analysis of the data record time periods of the tests were identified when these three representative wind directions prevailed.



	Unit	Intial	Coarse	Medium	Fine
Minimum cell size	{m}	0.10	0.10	0.08	0.06
Maximum cell size	{m}	72.00	72.00	57.60	43.20
Total volume cells	#	2,424,167	3,041,620	4,441,165	7,898,730

Figure 5.9: Computational domain and the resulting mesh used in the final CFD simulations (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.7 Phase 3 Final CFD simulations- Post processing of the CFD Simulation Results

Contoured graphs and streamlined plots were used to illustrate pertinent parameters from the CFD simulation data sets. These types of post-processing are used for this report since it provides an intuitive understanding of the predicted wind induced fluid dynamic phenomena. Figure 5.10 through 5.17 show typical colors contour maps for wind speed, streamlines and pressures.

The validation of CFD simulations with data measured in the field required an exact determination of wind speed and wind induced pressures at exact given locations. Rather than interpolating numerical values from patterns shown on the colored contour maps, a procedure was adopted to extract simulated data from presentation grids for validation. Figure 5.18 shows how "representation grids" were used to determine the CFD generated wind speed values at the locations where the field measurements were carried out.

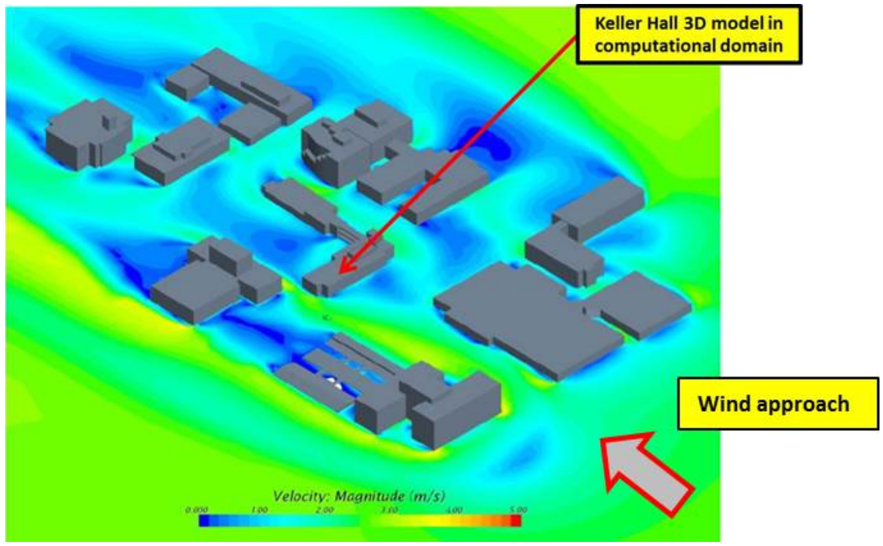


Figure 5.10: Overall Isometric View (from NE) of Wind Velocity Contour Map (Figure presented in Project Report 3, External CFD Simulation and Field Verification)

The colored contour map indicates the wind speed around the Keller Hall and surrounding buildings for a North wind approach direction. The horizontal slice (e.g. contour map) is referenced to the horizontal plane at 1m height above finishing level of the 1st floor of Keller Hall.

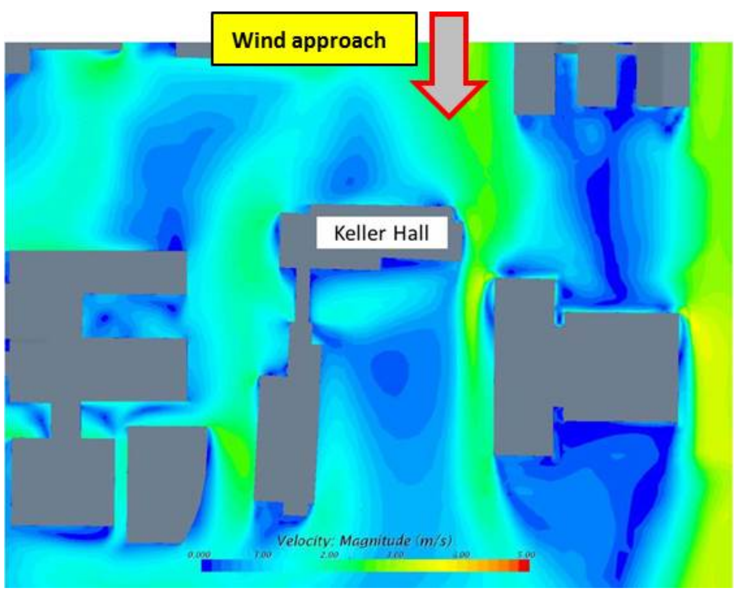


Figure 5.11: Plan View of Detailed Wind Velocity Contour Map around Keller Hall (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The colored contour map indicates the wind speed around the Keller Hall and surrounding buildings for a North wind approach direction. The horizontal slice (e.g. contour map) is referenced to the horizontal plane at 1m height above finishing level of the 1st floor of Keller Hall.

The streamline plots in Figures 5.12 and 5.13 combine color indicators of the wind velocity, and add directional information about the wind flow streamlines. The plots provide a good overview of the wind flow conditions around Keller Hall and the surrounding buildings. Wind approach is from the North.

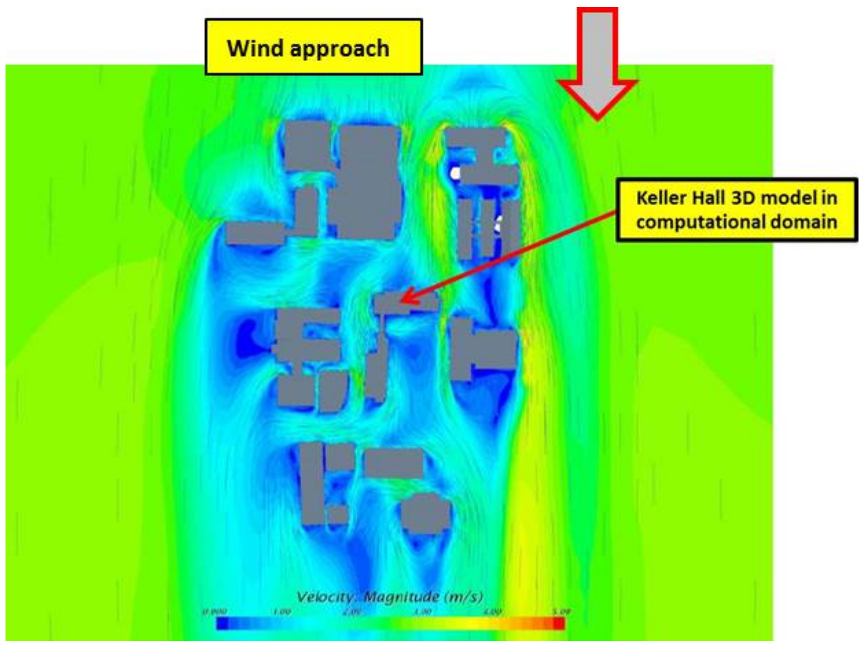


Figure 5.12: Plan View of Overall Wind Velocity Streamline Map (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

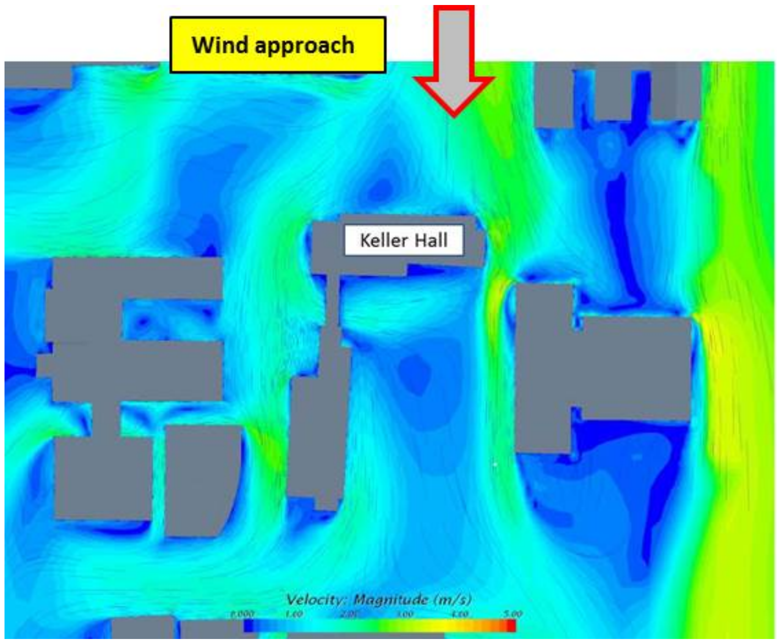


Figure 5.13: Plan View of Detailed Wind Velocity Stream Map around Keller Hall (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

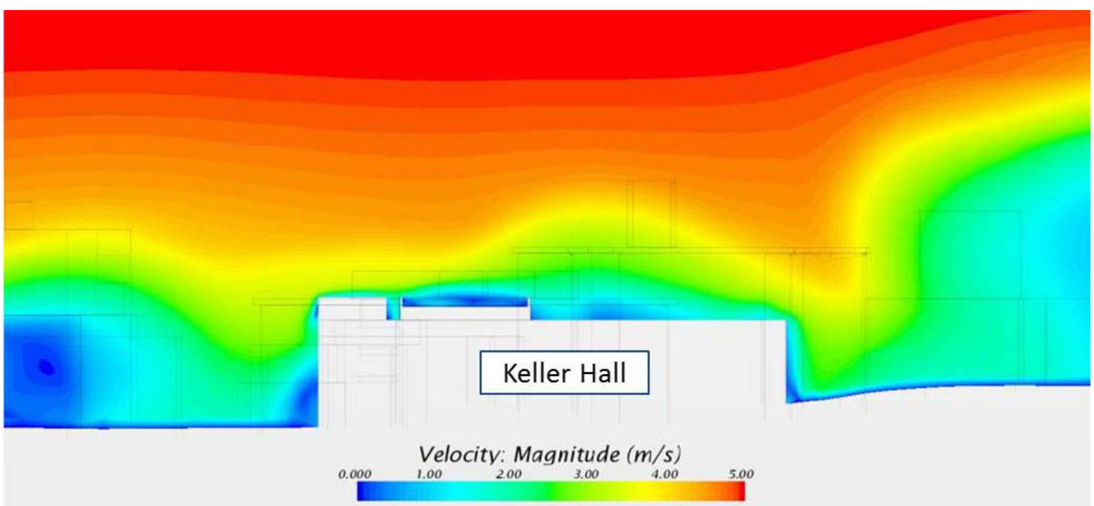


Figure 5.14: Detailed Wind Velocity Contour Map on Cross Section Along Long Axis of Keller Hall. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The colored contour map indicates the wind speed around the Keller Hall with North wind approach direction. The horizontal slice (e.g. contour map) is referenced to the East – West centerline of the Keller Hall building.

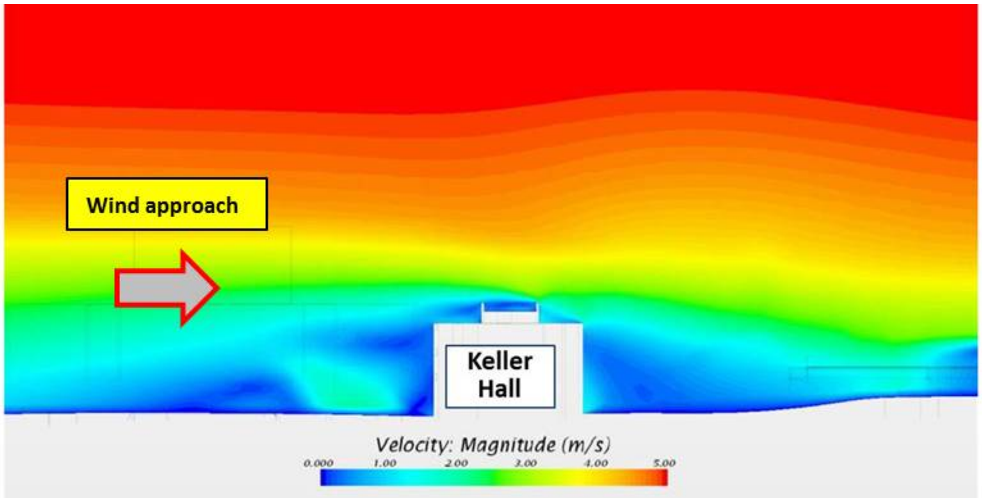


Figure 5.15: Detailed Wind Velocity Contour Map on Cross Section Along Short Axis of Keller Hall (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The colored contour map indicates the wind speed around the Keller Hall a North wind approach direction. The horizontal slice (e.g. contour map) is referenced to the North-South centerline of the Keller Hall building.

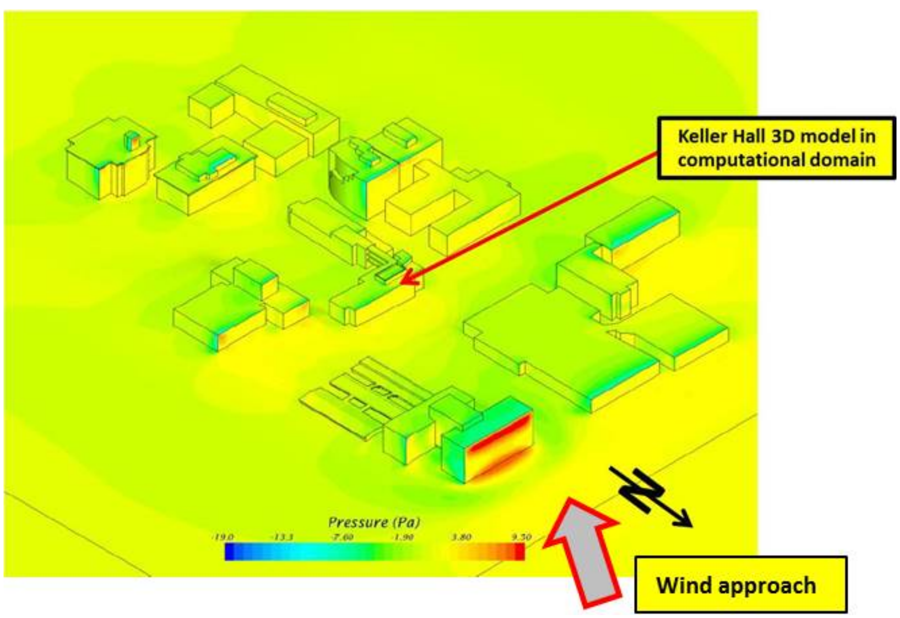


Figure 5.16: Overall Isometric View (from NE) of Wind-Induced Pressure Distribution on Building Facades (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The colored contour map provides an overview of absolute pressure distribution projected on the building envelope viewed from the Northeast on the Keller Hall buildings and surrounding buildings. Wind is from the North.

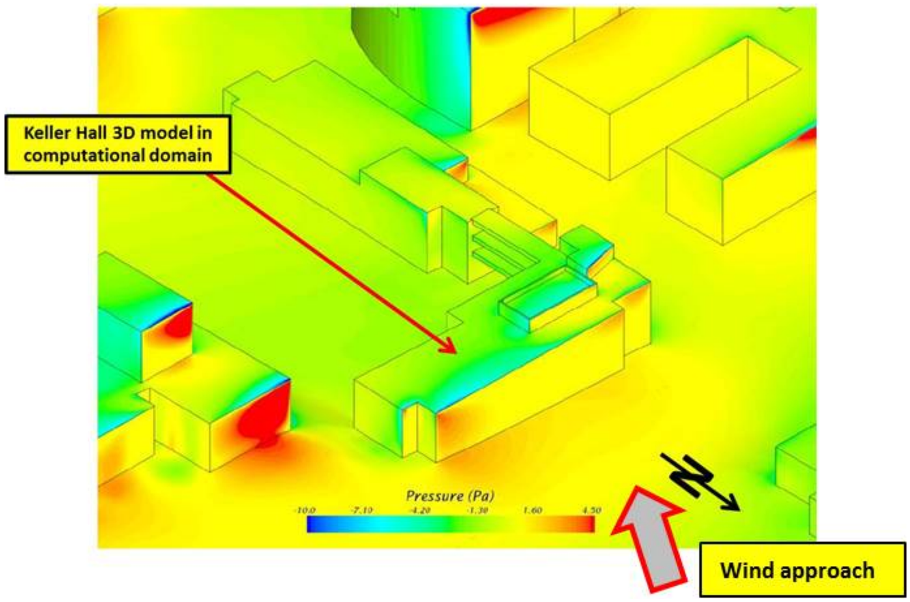


Figure 5.17: Detailed Isometric View (from NE) of Wind-Induced Pressure Distribution on Keller Hall's Facades (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The colored contour map provides a detailed view from the Northeast of absolute pressures on the Keller Hall buildings. Wind is from the North

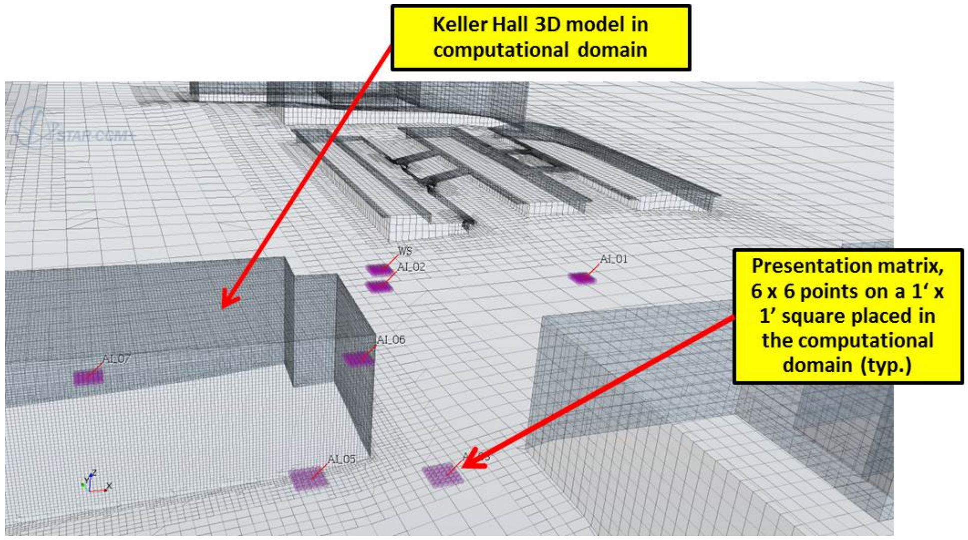


Figure 5.18: Depiction of all presentation grids for wind speed assessment at locations of anemometer in the field tests (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.8 Phase 4 Results and Conclusion- Results of Field Measurements

Data of the wind measurement around Keller Hall included three types of measurement:

Results from the weather station on roof of Keller Hall: The data obtained by the weather station included wind direction and speed. The data was recorded over a period of five months. Figure 5.19 show an example of data from the weather station wind measurements. The different images in Figure 5.19 show the resulting data sets when data filters for exceeded wind speed are applied. The date was used to determine the representative wind approach direction and speed for use in the final CFD runs in order to correlate the differential pressure data.

Results from weather station at ground level: This data was used to determine the representative wind direction and speeds for use in the final CFD runs in order to correlate theoretical CFD simulation results with wind measurement data.

Results from six hot-wire anemometers: The data sets from the six anemometers were filtered an averaged to obtain the representative wind speed for the selected sensor location.

Data of the differential pressure around Keller Hall: Differential pressures between several locations on the North and South façade of Keller Hall were recorded over a period of three months. Figure 5.20

shows an example time series of differential pressures measured by one transducer for a specific wind approach direction. As can be seen the data record shows that the differential pressure oscillates significantly around the neutral, or zero, axis. The differential pressures were recorded over a period of three month and a statistical analysis was done to correlate the differential pressures against the approach wind direction and speed.

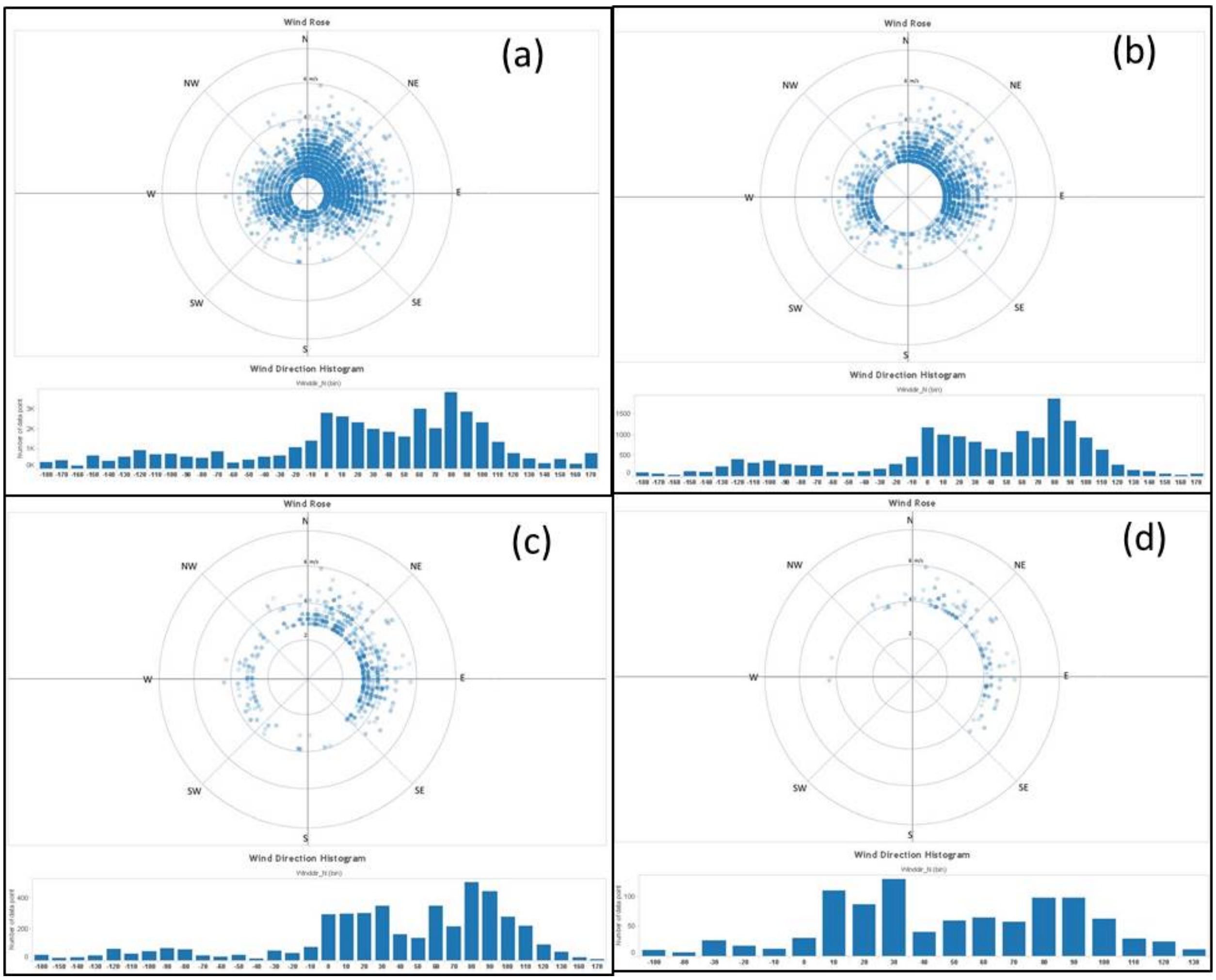


Figure 5.19: Data sets obtained for wind direction and speed from the weather station on the roof of Keller Hall, (a) entire date set, (b) only data for wind speed larger than 2 m/sec, (c) only data for wind speed larger than 3 m/sec, (d) only data for wind speed larger than 4 m/s (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

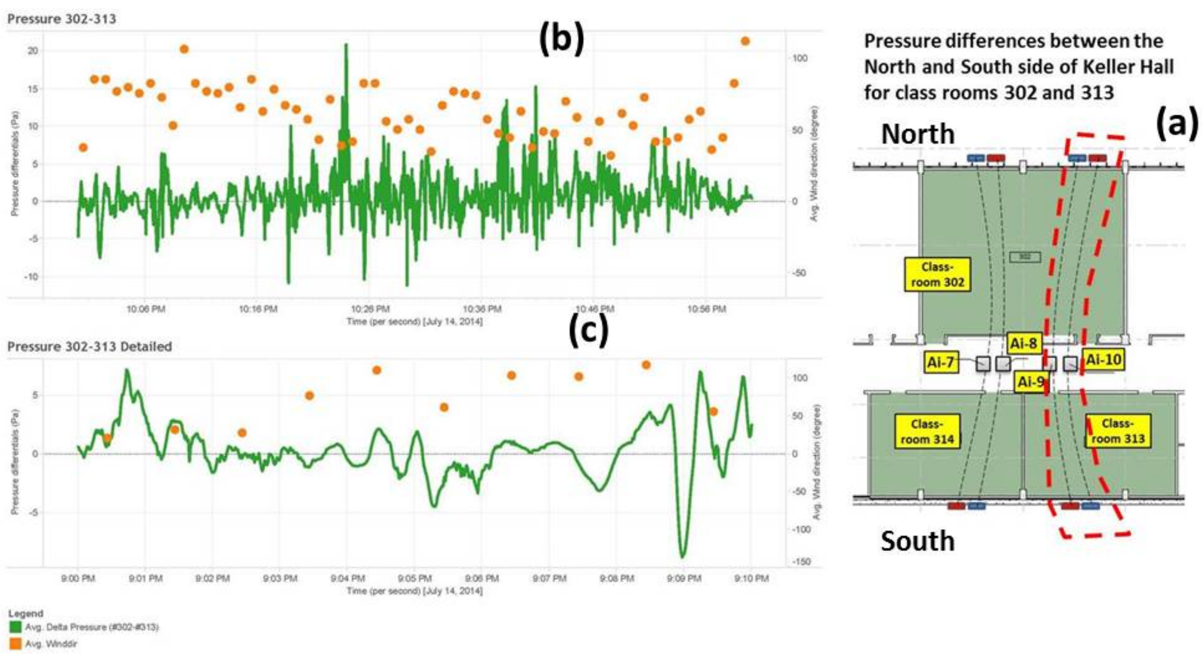


Figure 5.20: Example of a differential pressure data record; (a) shows the location of the two pressure tubing terminals in relation to the transducer, (b) a longer record showing the fluctuating differential pressures around the zero pressure mark, (c) a shorter record of the differential pressures between two locations on the North and South facades of Keller Hall. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.8 Phase 4 Results and Conclusion - Comparison of CFD Simulation and Field Measurements

Table 5.1 shows a summary of wind velocities obtained through data reduction of actual field measurements and CFD calculated values for all anemometer locations. The CFD values were obtained with a medium grid resolution.

Figures 5.21 through 5.23 show comparisons of measured and CFD predicted wind velocities for wind approach directions from the North, East and North-East, respectively. Figure 5.24 shows the divergence of measured and CFD simulated (medium grid resolution) values for the three wind directions. The comparison of representative values obtained by CFD simulation and field measurements depict the most consistent trend for wind approaching the building from the North.

The comparison of measured and averaged versus CFD calculated data suggests that values show different degrees of correlation for selected wind setting and anemometer locations. Figure 5.25 shows the probability density of the velocity divergence ratios V_d falling into certain ranges. The range of diverging values shown in Figure 5.25 indicate that about two thirds of the data points that correlate between measured and CD calculated values show less than 50% divergence. Compared with other studies presented in the literature these diverging values are not uncommon.

Table 5.1: Wind velocities obtained through field measurement and CFD simutaions, for all anemometer locations

Approaching wind direction		**AI_01**	**AI_02**	**AI_03**	**AI_05**	**AI_06**	**AI_07**
Measurement							
North	avg.	1.644	2.119	3.581	1.638	2.125	2.280
	stdev	*0.592*	*0.716*	*0.703*	*0.501*	*0.656*	*0.802*
East	avg.	1.678	1.861	3.650	1.564	2.352	2.583
	stdev	*0.584*	*0.691*	*0.644*	*0.319*	*0.823*	*0.823*
North East	avg.	1.709	1.762	1.841	3.242	1.827	2.301
	stdev	*0.613*	*0.633*	*0.804*	*0.679*	*0.584*	*1.027*
Simulation							
North	avg.	1.289	1.493	2.550	0.583	1.534	1.187
East	avg.	3.279	3.167	2.427	0.935	1.389	0.769
North East	avg.	1.895	2.263	3.523	0.712	1.093	1.761

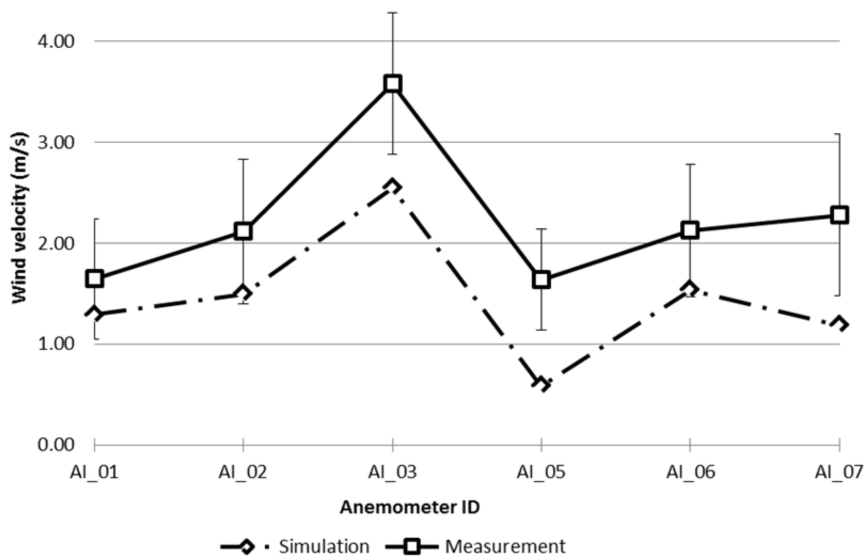


Figure 5.21: Comparison between averaged wind velocity for field measurements and simulation (medium grid resolution) wind is approaching from the North. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

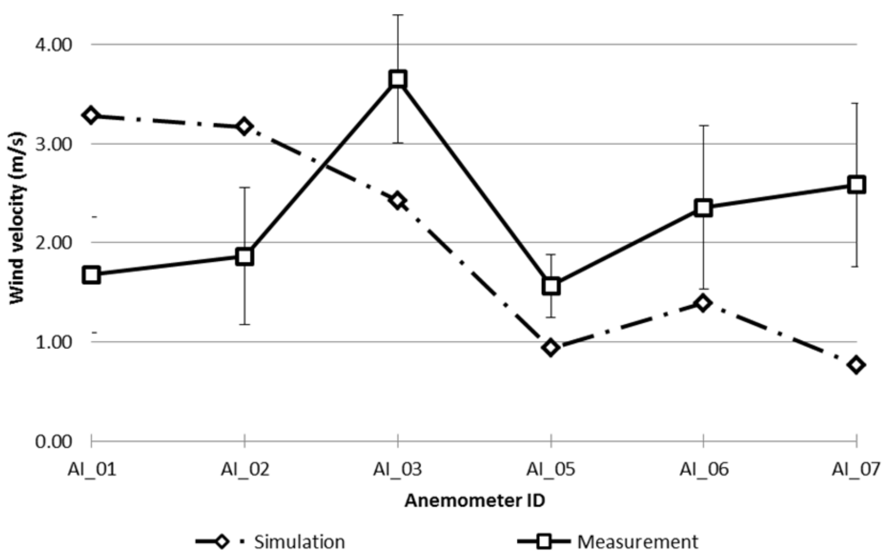


Figure 5.22: Comparison between averaged wind velocity for field measurements and simulation (medium grid resolution) wind is approaching from the East. (Figure presented in Project Report 3, External CFD Simulation and Field Verification)

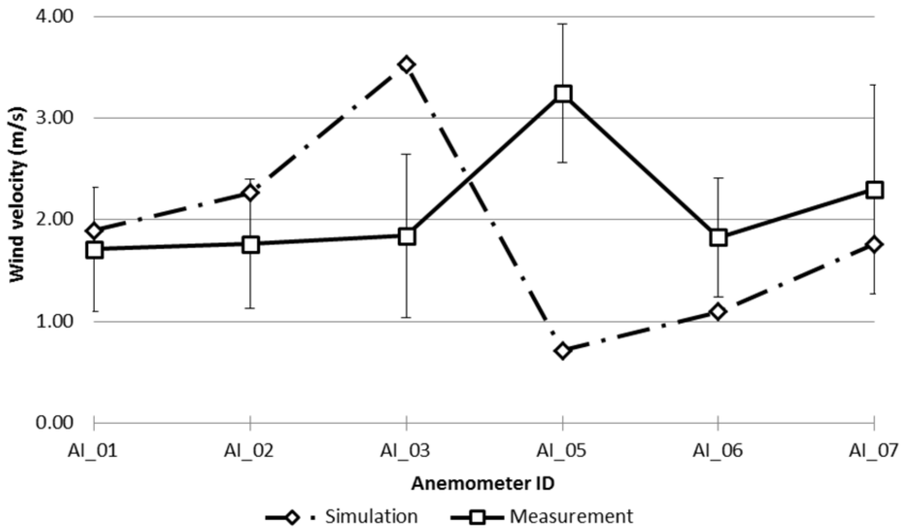


Figure 5.23: Comparison between averaged wind velocity for field measurements and simulation (medium grid resolution) wind is approaching from the North-East. (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

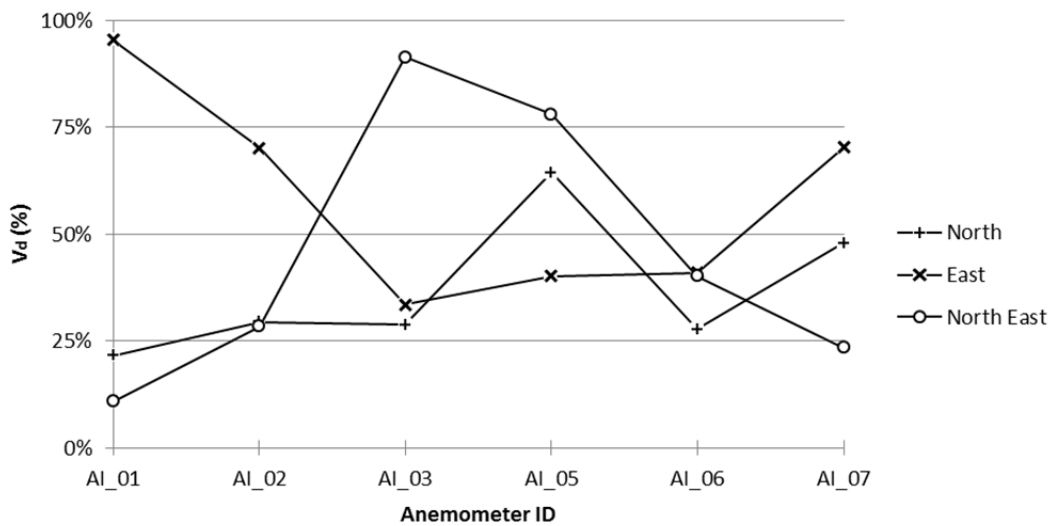


Figure 5.24: Wind velocity divergence ratios Vd (%) between measurement and simulation (medium grid resolution) at given approaching wind directions (V_d = ABS((V_s – V_m)/V_m)), where V_s is simulated wind velocity and V_m is the measured wind velocity). (Figure presented in Project Report 3, External CFD Simulation and Field Verification)

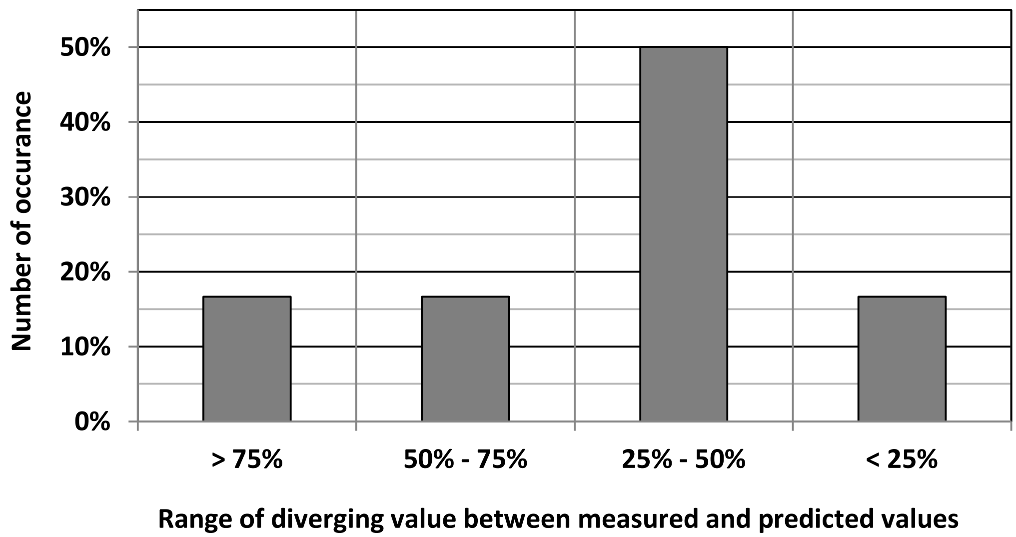


Figure 5.25: Probability density of velocity divergence ratios V_d (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

Figure 5.26 indicates differences of values for differential pressures obtained through measurements and CFD simulations. The results suggest that the CFD simulations over predict pressure differentials for wind approaching from the North and the North-East. Measured and calculated values for wind coming from the East suggest that a higher pressure is produced on the South façade. This pressure distribution results in negative values for East winds as shown in Figure 5.26, since a positive pressure difference is defined as higher pressures from the North. It should be noted, however, how small the pressure differentials are between the North and South façade. Reliable measurements of such small pressures are quite difficult to obtain, and one reason for the diverging measured and calculated values could be pressure transducer limitations at such small pressure differentials. Another point to consider is that the graphs representing the pressure differentials measurements represent averaged numbers of fluctuating pressures, which are by themselves highly unsteady occurrences. The CFD simulation results, however, represent steady state flow occurrences with unique pressure differentials.

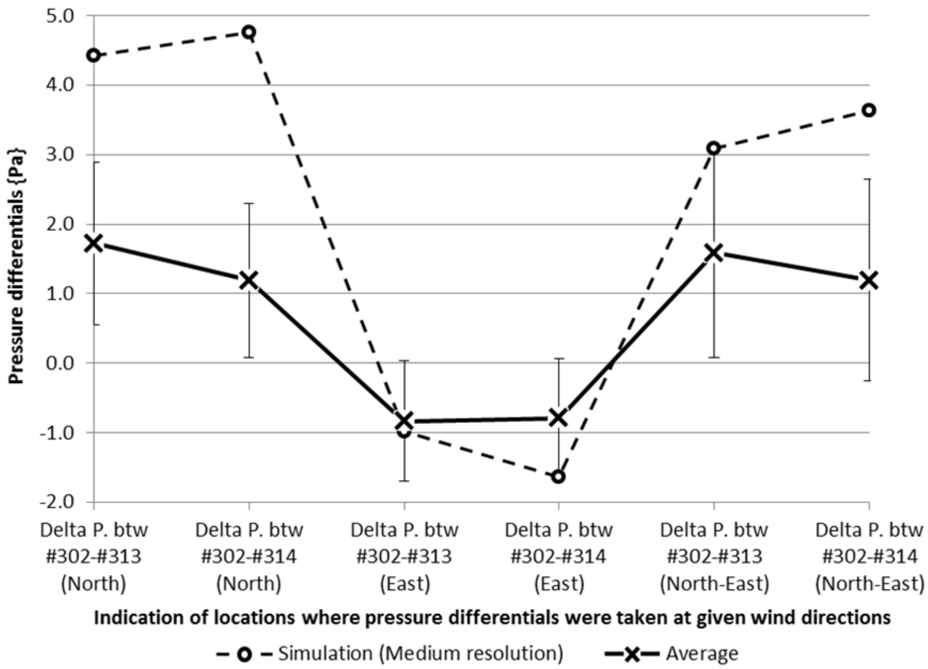


Figure 5.26: Comparison between averaged values of differential pressures across the North and South façade of Keller Hall obtained from measurements and CFD calculations (Higher pressure on the North façade indicate positive values) (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

The phenomenon of the fluctuation differential pressures presented in Figure 5.20 is depicted in more detail in Figure 5.27, which is a short-term data record of differential pressures measured during East winds. A positive pressure differential indicates higher pressures exerted by the wind on the North façade of Keller Hall. The record in Figure 5.27 suggests a time series of fluctuating differential pressures and observed instantaneous recorded wind directions. The data for wind direction indicates changes in the order of 100 degrees within 10 minutes of recorded time. The resulting pressure differentials indicate that wind pressures build up on opposite sides of the building with fluctuation periods between 10 seconds and one minute. This interpretation of the changing positive pressure differentials between the North and South façade was supported by observations of internal air movement inside the building. The research team encountered air flow reversal from northward to southward. The reasons for these air flow reversal observations are not conclusive at this point in time. Since fluctuations of external differential pressures and the somehow related air flow reversals observed inside buildings could negatively affect natural ventilation performance of buildings, further investigation is needed to study this phenomenon.

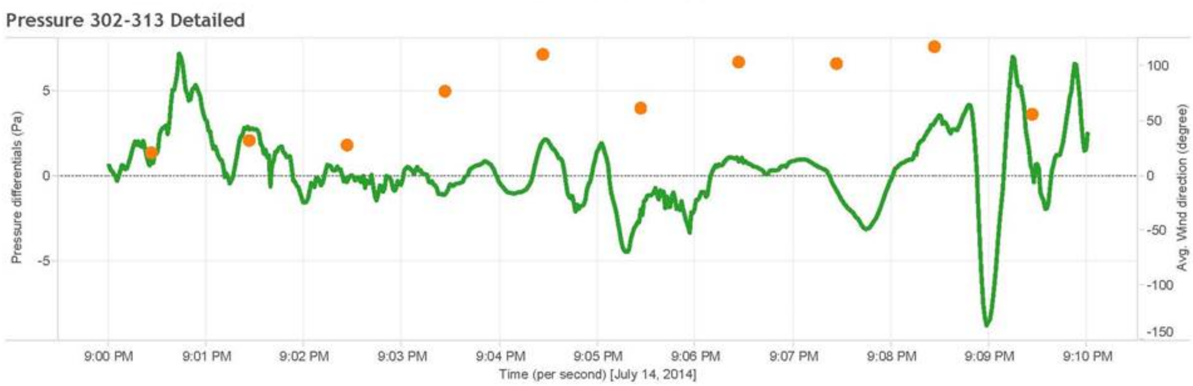


Figure 5.27: Fluctuations in measured differential pressures (positive differential pressures indicate pressure is higher on the North façade of Keller Hall building) (Figure presented in Project Report 4, External CFD Simulation and Field Validation)

5.9 Phase 4 Results and Conclusion – Conclusions and Recommendation

This section of the report presents main conclusions and recommendations of the investigation of the external wind movement around Keller Hall.

A. CFD Applications:

The CFD software STAR-CCM+ provided an efficient and powerful platform for advanced CFD analysis of wind induced air movement and pressure phenomena around the selected building as well as surrounding buildings. The technical support of the developer of STAR CCM+ provided helpful input for a couple of application issues.

A.1 For this study the research team applied a process of generating 3D-CAD geometries of flow obstructions inside and boundaries around the computation domain, which was different from previous processes. The newly developed process uses the CAD functions of Sketchup and Rhino to produce 3D-geometries and import these geometries into STAR-CCM+. The new process is more effective than the previous process of creating geometry with the CAD software AutoCAD Inventor and importing it into the STARR-CCM+ CFD application.

A.2 Benchmarking simulations were conducted early in the study to ascertain the required scope of surrounding buildings and type of topography to be included in the computational domain so that simulations would yield reliable results. Surrounding building and topography can significantly affect the air movement and pressure distribution around the target building, Keller Hall. The decision how much of the surrounding buildings and topography is included in the model has to balance two conflicting consideration of the simulation procedure. First, extending the reach of the surrounding geometry of buildings and topography can improve the quality of the simulation by including more interactions and second, increase air flow obstructions in the computational domain increases the required computational simulation running time. It basically comes down to a balancing act between high probability of simulation accuracy and length of time of simulation runs. The research team came up with a viable and effective choice of the extent and geometry of surrounding buildings and topography.

A.3 Foliage was not included in the geometry model of the domain. The literature reports on various approaches to inclusion of vegetation objects in the CFD simulations. The research team is aware that there are multiple trees in the vicinity of Keller Hall which will affect wind pattern to a certain extent. Foliage, however, was not included for this study since other influences to the wind pattern around the target buildings were considered more significant. For other building situations including foliage might be a more important consideration as for prevailing wind movement around Keller Hall.

A.4 The terrain surrounding Keller Hall used in the CFD simulations was taken as flat and horizontal. Benchmarking (see point 1.3 above) suggested that using more complicated topography for the case of Keller Hall would not increase accuracy of the flow prediction appreciatively to warrant the significantly higher demand on computational time and effort in modeling. The goal of this study is to identify and test CFD procedures that aid in the design process and the assessment of multiple performance criteria. Including complicated topography involves significant 3D-geometry data import and manipulation and is usually beyond the practical use of CFD to assess wind movement around buildings.

A.5 Four different mesh resolutions were used in this study. Initial CFD investigations used a rather coarse gird. This simplification for the initial CFD served the desired level of complexity since the initial CFD simulations also use simpler geometries and assumed (from historical weather files) speeds and direction of the approaching wind. The purpose of the initial CFD runs was to gain a basic understanding of the wind pattern around the target building and select preferred locations of wind speed measurements around the building. For future CFD investigations the same procedure of initial CFD runs will be used to determine preferred locations to place sensors for CFD validations of advanced CFD models.

A.6 Benchmarking with three different grid resolutions was conducted for a sensitivity analysis of the final CFD simulations. A coarse, medium and fine grid was used with a total number of cells of 3.0, 4.4 and 7.9 million cells, respectively. Using higher grid resolutions typically increases the accuracy of predicting relevant flow occurrence close to the target and at areas of higher flow property gradients. This means that in areas where an accurate prediction of flow phenomena is required, such as at areas with flow separation or flow stagnation, more and smaller cells in the computational grid are preferred. The disadvantage of higher grid resolution is the required computational recourses. From benchmarking conducted in this study it was observed that for identical setting and boundary conditions a course, medium and fine grid resolution required an average 6 hours, 12 hours and 36 hours simulation run times, respectively. Results suggested that a medium grid was sufficient and, apart from reasonable run times, also showed good convergence.

A.7 As a way of improving the accuracy of the CFD simulation solution, the literature proposes the process of "adaptive mesh refinement" (Blocken, 2007). The research team benchmarked this process against the normal procedure of setting the grid resolution through inbuilt CFD software functions. For external CFD analysis, the results suggested that adaptive mesh refinement did not result in significantly improved performance to warrant the extra time and effort. For the internal CFD analysis adaptive mesh refinement could, however, be a relevant procedure to increase performance of the simulation.

A.8 An important consideration for the cell dimensions of the mesh is the required number and size of prism layers. The process used in this study used sizing of prism layers in accordance of the selected surface roughness and recommended wall-function treatment procedures.

A.9 It was found that in addition to CFD simulation residuals, the behavior of absolute wind speeds detected for the (virtual) anemometer monitors proved an effective means to assess level of convergence of the simulations.

A.10 A typical CFD simulation run required up to 3,000 iterations for residuals to achieve the convergence criterion for properties such as momentum or turbulence. As pointed out under point 1.10 the absolute wind velocities for (virtual) anemometers converged to a final value much earlier.

A.11 The building façade was modeled as opaque or impermeable in the CFD model. This is consistent with the "decoupled" CFD simulation approach of the study. With assuming an opaque wall the wind impinges on the façade as a true stagnation and the resulting pressures are calculated accordingly. The pressure tubing terminals, however, were installed close to the open windows of the naturally ventilated spaces on the third floor of Keller Hall. This was done for logistical reasons since installation of terminals from the inside the class rooms through the louvered windows was the only practical way to access the building façade. Open windows, however, permit air movement to the inside which could change the air flow pattern in the vicinity of the windows. Air flow in the vicinity of the pressure tubing terminals can affect the absolute and time dependent values of the pressure measurements. The research team believes that this effect was not significant in magnitude, but this aspect might need further consideration for future CFD work of the team.

A.12 The presentation grid approach used to determine the absolute wind velocities in CFD post processing was effective in providing good representative values for wind velocities at locations of interest, e.g. the locations where the virtual anemometers were located. The use of presentation grids is not widely addressed in the literature that describes validation of CFD with field measurements. The good performance of presentation grids in this study, however, justifies the extra effort in post processing. In the future, the research team might use a 3D rather than a 2D presentation grid to make the determination of properties in the CFD domains even more effective.

B. Instrumentation and Data Handling:

B.1 The anemometers that were used for the measurements had some limitations since their sensitivity is a function of their face orientation towards the approach wind direction. The type of anemometers used for the wind measurements around the Keller Hall building have a preferred measurement sensitivity of +/- 30° of the direction towards which the face of the sensor is pointing. Any measurements of wind coming from a direction outside of this optimum range of about 60° would be affected. The research team did not find a reliable correlation how to adjust the data in accordance to the direction of the wind which was selected when the anemometers were deployed in their stands. The effect on the recording, therefore, cannot be ruled out and could also not be quantified. The preferred direction towards which the instruments were pointed was selected from the initial CFD runs. These initial CFD runs considered a wind approach from the North. Therefore it can be assumed that wind recording for wind approaching the building from North should be most consistent with the CFD simulation predictions. As a matter of fact, the results support this theory.

B.2 A further limitation of the anemometers used in the investigations is that they cannot identify the wind approach direction. The anemometers basically only measure property changes in conductance caused by a cooling effect of the passing wind. Hence, the anemometer can only determine the wind speed but not the direction. A good, albeit expensive, alternative to the type of anemometers used in the study are 3D ultra-sonic anemometers, which are highly accurate and can identify wind direction. An internal discussion with an European leading expert in CFD application in the urban environment suggested the use of two ultra-sonic anemometers, one for reference at a fixed location and one moveable that can be deployed in a patterns of sequential measurements at different predetermined locations.

B.3 The differential pressure ranges detected in the field measurements are quite small, with ranges well under 10 Pascal. The recorded pressure differentials compared well with those reported in the literature for other natural ventilation investigations. The differential pressure transducers had a full range of 10Pa and 25 Pa for the Halstrup-Walcher P-26 and Setra Model 264, respectively. In their performance the Setra transducers showed a very good cost benefit ratio. The Sentra transducer will be the differential pressure transducer of choice for future investigations.

B.4 The Tableau software for data analysis was very effective. The use of software allowed for expeditious data analysis, including display of data in accordance to filtering of parameters.

B.5 The data acquisition using a combination of web based instruments, with a wireless signal conditioning, and hard wired instruments proved to be very effective and resulted in stable and consistent recording. The time and effort, however that went into developing robust deployment procedures and data capturing and conditioning should not be underestimated. The time and effort was in fact considerable. The procedures and experience were recorded for future reference. With changing of the research assistants being a regular event at ERDL this is an important task and serves to capture important experiences for future research.

B.6 The procedure of filtering data in accordance with certain governing parameters such as approach wind direction or time was a powerful analysis process. The software Tableau proved to be an indispensable analysis tool.

C. Wind and Pressure Pattern:

Since wind and pressure measurements have been adequately addressed in the report, there are a few general observations that will have to be considered for future comparable applied research efforts at ERDL.

C.1 The use of wind speeds to validate CFD simulation results appears to yield better results than using pressures as indicators of wind induced air movement phenomena around buildings. The determination of pressures, however, is nevertheless important since the driving force for natural ventilation is the pressure differential between sides, or more precise between air intake and discharge of the building.

C.2 A surprising observation of the differential pressure measurements was the fluctuation of differential pressures around the building. These appear to be periodic in nature and significant periods in the range between 30 and 60 seconds have been identified. It is not surprising that eddies have a significant effect on the pressure distribution of the building. The fact, however, that such significant wind direction change occur in relatively quick succession and can be correlated with short period differential pressure reversals could have a significant effect on the natural ventilation performance of buildings. Further investigation in this phenomenon seems prudent.

SECTION 6 - CONCLUSIONS OF PART 1 - EXTERNAL CFD APPLICATIONS

The project work performed for Part 1 of this research program provided the research team with valuable opportunities to develop skills that are essential to perform advanced external CFD simulations of wind movement around buildings. The objectives of Part 1 of this research program were met and the overall outcome of the project can be viewed as a successful first step to develop advanced CFD investigations with validation in the field at ERDL.

The CFD project team has drawn the main following conclusions:

- The literature search concluded that CFD simulations of external wind movement around buildings are an increasingly important design tool for the build environment. The prediction of external wind movement is important to assure structure integrity, pedestrian safety and comfort and effectiveness of building location and form to support effective natural ventilation. The literature search suggested that CFD simulations of external wind movement around buildings represents challenges but existing best practice guidelines provide valuable input and guidance to CFD practitioners.

- Wind movement around buildings is significantly affected by the nature and strength of the approaching wind and structures which act as flow obstructions.

- Reliable external CFD simulations are dependent on a wide range of software input and operating tasks and interactions, such as:

  - The CFD team has to be proficient to plan, conduct and interpret CFD simulations. The CFD team has certain roles and functions that are delineated in the work flow report (Section 4 of this summary report).

  - One important function which the CFD has to perform is setting up the correct geometrical and physical model for the simulation. The CFD team has to be competent to make good judgment in regard to the level of detail that is used in the geometry and computation mesh and domain.

  - A powerful CFD software and code is required that carry out the calculations to the level of sophistication that is required for the external CFD application. The software needs a verified calculation process with the appropriate turbulence model and other functions that support the specific CFD application. The project team used the software Star CCM+ of CdAdapco which proved to be suitable and effective for the project work. There are more advanced CFD simulations methods, such as Large Eddy Simulations (LERS) and unsteady flow simulations, but at this time the CFD project team did not possess the computational resources to run these very demanding CFD code applications. A future continuation of advanced CFD work at ERDL might include LES and/or unsteady analysis.

   - The present CFD investigation only considered steady-state RANS simulations because of the constraints in software licenses and computational resources.

   - The CFD team has to be cognizant of the fact that any CFD simulation is a simplification of the actual conditions found at the building level. It is not possible, and in most cases not even advisable, to increase the sophistication and resolution of the CFD model to such a high degree, that the building model and adjacent structures resembles the actual geometry with all major details. A CFD model is NOT a CAD model, and it is good CFD simulation practice to make informed decisions on truncating the level of detail of the CFD model.

- A CFD workflow for the prediction of wind movement around buildings should be validated in the field through the measurements of local wind speed, wind directions as well as pressure distribution on the building envelope.

- Through the field measurements work, the CFD project team gas gained valuable proficiency in selecting the appropriate instrumentation, data acquisition system, and data analysis tools. The CFD team has found it valuable to first test all instrumentation in the lab and in so-called shakedown testing at a field test site before carrying out the final measurements which will be used for validation of the CFD simulation at the building prototype.

- The CFD project team experienced that data consistency of CFD predictions results and actual measurements of wind movement and resulting phenomena typically differ at various levels. The CFD project team has found both very good and medium data consistency. The CFD project team identified several reasons for data deviations between theoretical predictions and actual measurements. This knowledge is an important criterion in order to interpret CFD simulation results.

- The various processes that were developed to plan, prepare and conduct the CFD work as well as to obtain reliable field measurements for CFD validation and lessons learned in Part 1 “External CFD Investigations” have been well documented and will support ERDL to conserve the CFD skills for future students and professional staff.

REFERENCES

All references are made to the four projects reports which are summarized in this report. This report does not give any references other than was provided in the four reports.