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 Internal 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

April 2015

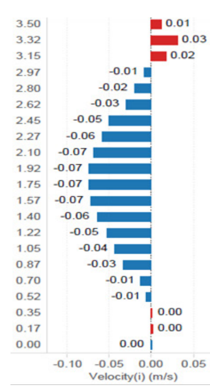
HNEI
Hawai'i Natural Energy Institute
University of Hawai'i at Mānoa

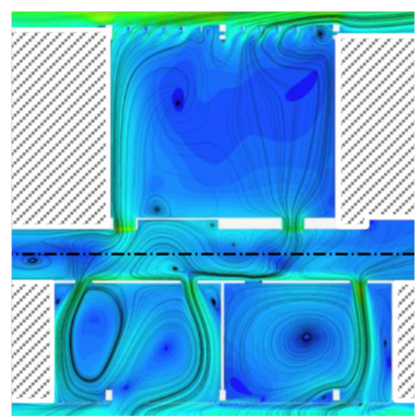
Project Phase 1- 7.B

SUMMARY AND CONCLUSION OF PROJECT PHASE 2 - INTERNAL CFD

10

April 2, 2015





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.B –

Task 7.b.4: Project Summary Report Phase 7.B (Phase 2)

Project Deliverable No. 10:

Summary and Conclusion of Project Phase 2 – Internal CFD

Prepared for Hawaii Natural Energy Institute

in support of

Contract #N000-14-13-1-0463

April 2, 2015

Prepared by:

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

[Type text]

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 contribution of postdoctoral fellow Aarthi Padmanabhan, D. Arch. The authors also acknowledge the dedicated work by research assistants Mike Poscablo, Christian Damo, Reed Shinsato and Steven Chen. Christian Damo has graduated in the meantime and we wish him all the best for his professional and personal future.

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.

This report is a summary of the project work carried out for **Phase 2** of this research program. The content of this report summarizes the main findings presented in the deliverables of Phase 2.

The project work of Part 2 “Internal CFD Applications in Building Design” was submitted in four deliverables (six draft and final reports):

Deliverable 6:	Task 7.a.1 - Literature review for internal l CFD
Deliverable 7.1 and 7.2:	Task 7.a.2 - Draft and Final report to establish an external CFD work process (Draft and Final report)
Deliverable 8:	Task 7.a.3: Report to develop and calibrate a data verification process for external CFD simulations
Deliverable 9.1 and 9.2:	Task 7.a.4: Draft and Final report on external CFD simulation & field verification for selected building (Draft and Final report)

The main objectives of Phase 2 - Internal CFD were to:

- Evaluate the existing application literature of internal CFD and identify what topics relate to the intended application of internal 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 air movement through buildings.
- Validation of internal 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 6 months, starting in June 2014 and completing the project work of Phase 2 in December 2014.

The present summary report describes the applied research work that was carried under Phase 2 of this research program. The project work included modeling of air movement inside buildings on the University of Hawaii Manoa Campus with Computational Fluid Dynamics (CFD) and validating the theoretical CFD results through full-scale measurements in internal spaces.

SECTION 1 - OVERVIEW AND RESEARCH APPROACH OF PHASE 2

The overreaching objective of Phase 2 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 air movement through buildings and their effects on natural ventilation performance of buildings. The project work of Part 2 of the research program was carried out in four project tasks 2.1 through 2.4, illustrated in Figure 1.1. These project tasks are briefly described below and are summarized in Sections 2 through 5 of this report.

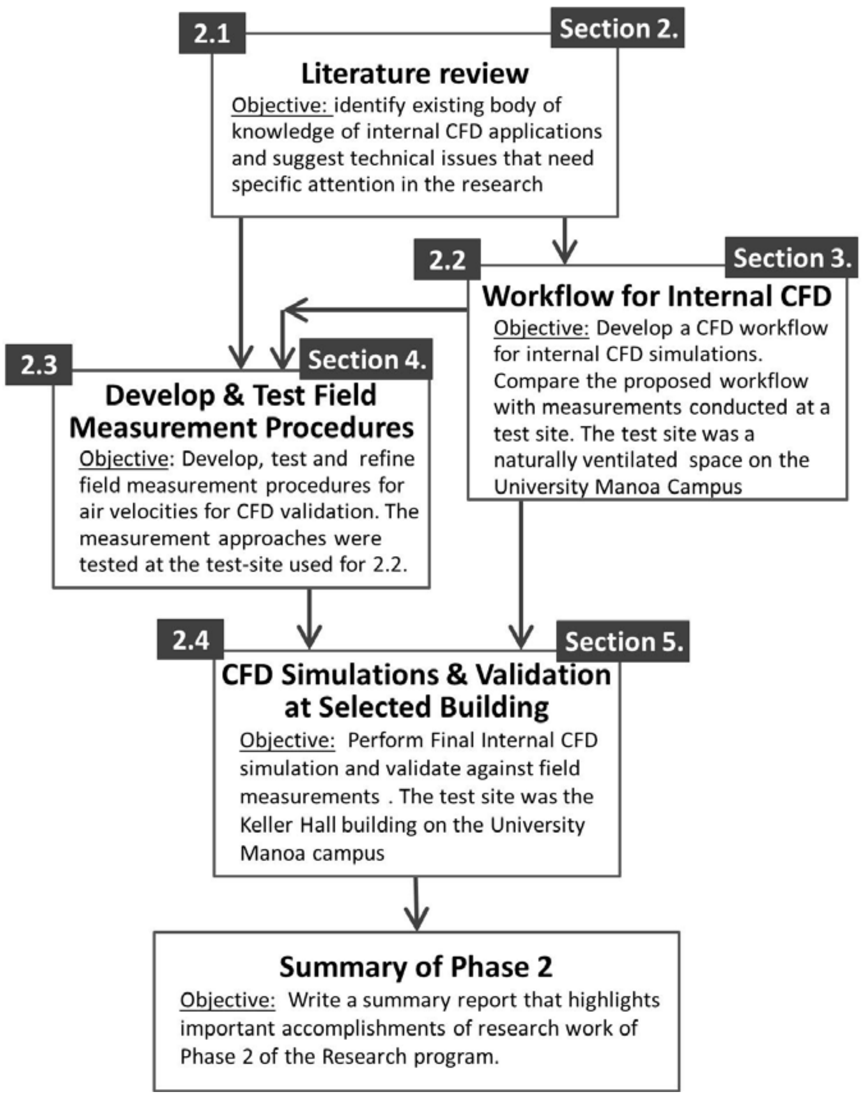


Figure 1.1: Flow chart of the project work for Phase 2 of the research program.

The sections indicated in the flow chart are sections of the present summary report.

Project Task Group 2.1 - Literature Review for Internal CFD: The literature review provided an overview of an identified body of literature about internal CFD applications and related fields of study. The literature reviewed comprised approximately 45 technical and scientific publications. The focus of the literature review was CFD applications of air movement inside buildings and physical processes that affect the air flow through buildings, which are mainly naturally ventilated but could have some form of mechanical assist ventilation. The topic of air movement inside buildings is a relative small field of study and the literature about this topic is therefore limited. The literature review is summarized in Section 2 of this report.

Project Task Group 2.2 – Workflow for Internal CFD: The project specific workflow for internal CFD simulation was developed after completion of the literature review. Pertinent CFD techniques, settings and best practices were used to develop an internal CFD workflow that was unique for this research project. Work on the development of the internal CFD workflow occurred in parallel with the development and testing of field measurement procedures (Project Task 2.3). The CFD simulations procedures developed under Project Task 2.2 were validated against the tests of Project Task 2.3.

Project Task Group 2.3 – Develop and Test Field Measurements Procedures: Field measurement procedures for air velocity inside spaces included guidelines and suggestions about techniques and instrumentation identified in the literature review. The test procedure for this project had to consider new measurement approaches to facilitate reliable measurements in the presence of relatively small anticipated air velocities inside the selected test site.

Project Task Group 2.4 – CFD Simulations & Validation at Selected Building: After completing and validating the workflow for internal CFD, simulations were conducted at the Keller Hall building on the University of Hawaii Manoa campus. The CFD simulation results were validated against measurements of air velocities inside Keller hall building. This Project Task 2.4 completed the work on Phase 2 of the research project.

The project work of Phase 2 “Internal CFD Applications” was performed in four project task groups 2.1 through 2.4. (as illustrated in Figure 1.1); project task 2.2 and 2.3 were performed concurrently

SECTION 2 - LITERATURE REVIEW

The review of scientific and technical literature was conducted to identify previous published research in areas that pertain to predicting air movement inside buildings by means of Computational Fluid Dynamics (CFD) investigations. The use of CFD programs to determine air movement inside buildings is a promising design and analysis tool for optimizing the performance of naturally ventilated spaces and/or for improving the performance of conventional or innovative HVAC technologies. CFD is also being used at present to understand and lowering risks for dispersion of gaseous matter or laden air inside buildings or spaces.

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

1. Basics of Natural Ventilation and Airflow Inside of Buildings: Review of basic processes of natural ventilation and air movement inside buildings, including but not limited to topics addressing wind driven and buoyancy driving forces for internal air movement, the performance of openings in the building envelope, pressure losses of air movement and mechanical assisted ventilation.

2. CFD as a Design Tool for Natural Ventilation Design: A review of previous CFD applications to assess internal air movement.

3. Internal CFD – Numerical Assessment of Air Movement Inside Buildings: Review of the state of the art and trends in internal 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. Basics of Natural Ventilation and Airflow Inside of Buildings
2. CFD as a Design Tool for Natural Ventilation Design
3. Internal CFD – Numerical Assessment of Air Movement Inside Buildings

2.1 Basics of Natural Ventilation and Airflow Inside of Buildings

This section of the literature review provided some important concepts of natural ventilation and related topics.

2.1.1 Basic Concepts of Natural Ventilation

Before the advent of widespread mechanical cooling and ventilation of buildings in Hawaii, as well as in other locations, buildings were designed and operated as naturally ventilated buildings. Although Hawaii has a tropical climate, the prevailing trade winds were sufficient in providing sufficient cooling and space ventilation. With higher standards of living and expectations of building performance which have generally evolved over the past decades, many originally naturally ventilated buildings have been modified by adding mechanically operated HVAC systems. Many new building designs have made mechanical cooling and ventilation a requirement, since natural ventilation could no longer provide the degree of thermal comfort and ventilation performance that came to be expected or that was required by code.

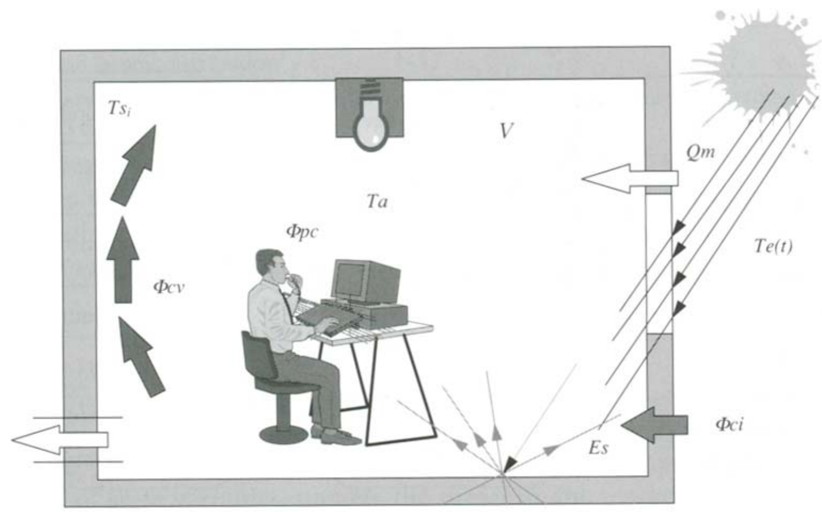
Mechanical cooling and ventilation has the significant advantage of safeguarding thermal comfort and sufficient ventilation at any time, irrespectively of the external environmental conditions. Since at least ventilation rates are required by local and national codes, the building operator needs to make sure that the building performs in accordance. The significant disadvantage of mechanical cooling and ventilation is the significant energy demand and related problems. On the other hand, natural ventilation can save significant amounts of energy if designed, built and operated correctly.

There are significant challenges to achieve a satisfactory building performance using natural ventilation in the hot and humid climate of Hawaii. The main aspects for achieving satisfactory natural ventilation performance are as follows:

Thermal balance of a room: In order to achieving satisfactory occupant thermal comfort levels in Hawaii's climate, heat that is either transferred from the outside to the interior, or heat that is generated inside the building needs to be expelled from the building. External heat gain is from radiant, convective and conductive solar heat sources. Internal heat gain is from heat sources such as electric loads or occupants. To achieve a steady state heat balance of spaces, the sum of heat inflow, outflow and generation within the system boundaries needs to be zero. Figure 2.1.1.1 shows a sketch of the thermal balance of a room. Figure 2.1.1.1 illustrates the links between the external and internal state variables as transfer phenomena.

Transfer equations that describe the flow of properties as heat or mass	**Balance equation** that define the state of properties
o Thermal diffusion (or conduction)	o Energy
o Radiation	o Pressure
o Convection	o Mass
o Mass transfer	

E_s represents short-wavelength radiation;
Φ_{cv} is the convective flux exchanged between the internal surfaces and the air;
Φ_{pc} is the total convective internal gains (occupants + equipment);
Φ_{ci} is the total conductive flux through the walls;
Q_m is the total mass flow rate of air at temperature T_e;
V is the volume of the room;
T_a is the air temperature;
T_{s_i} is the interior surface temperature.



$$VCp\frac{dT_a}{dt} = \Phi_{pc} + \sum_{i=1}^{nS} hc_i S_i\left(T_{s_i} - T_a\right) + Q_m Cp(T_e - T_a)$$

Figure 2.1.1.1: Thermal balance of a zone (Santamouris, et. a., 1998)

Driving forces for ventilation: Naturally ventilated buildings rely on pressure differences to move outside air through buildings. Pressure differences, or driving forces, can result from external wind actions impinging on the building envelope or from internal buoyancy effects caused by air density differences within the buildings and the outside environment. The same pressure driving force does not necessarily result in the same amount of air flowing through a building, since internal pathways create different energy losses. As an example, the size and type of openings in the building envelope and between internal spaces and other air resistances along the internal air pathways determine the quality and quantity of internal air movement.

<u>Wind driving forces – cross ventilation:</u> Wind impinging on a building creates a pressure distribution with respect to the background, will mean atmospheric, pressure. Wind actions on a building create a positive pressure on the windward side and a negative pressure on the leeward side of buildings. Due to the pressure differences, air will enter into windward openings and leave the building through leeward openings. The resulting pressure representing the driving forces is calculated by correcting the average dynamic pressure of the wind with a pressure coefficient C_p. This results in the following relationship:

$$Ps = C_p * P_v$$

With: $\mathbf{P_s}$ being the pressure at a specific location of the building envelope
$\mathbf{C_p}$ is the wind pressure coefficient
$\mathbf{P_v}$ is average dynamic pressure as calculated with $P_v = .5 * \rho * V_H^2$, where ρ is the outdoor air density (e.g. function of temperature and atmospheric pressure) and V_H is the mean wind velocity at upwind building height

The coefficient C_p is dependent on a range of variables, such as the approach direction of the wind, the building form, and the influence of nearby buildings. The building designer can get a basic understanding of the distribution of C_P on the building envelope by using generic tables or graphs. Figure 2.1.1.2 shows a generic distribution of C_P as a function of the building orientation, form and location of the pressure point on the building façade.

As with all tabulated design figures, there are inherent simplifications and generalizations to approximate the actual wind induced pressures on the building envelope. An external CFD analysis can calculate the pressure distribution on the external façade. The external pressure distribution is a function of the approach wind direction, the shape of the building as well as on the localized façade configuration.

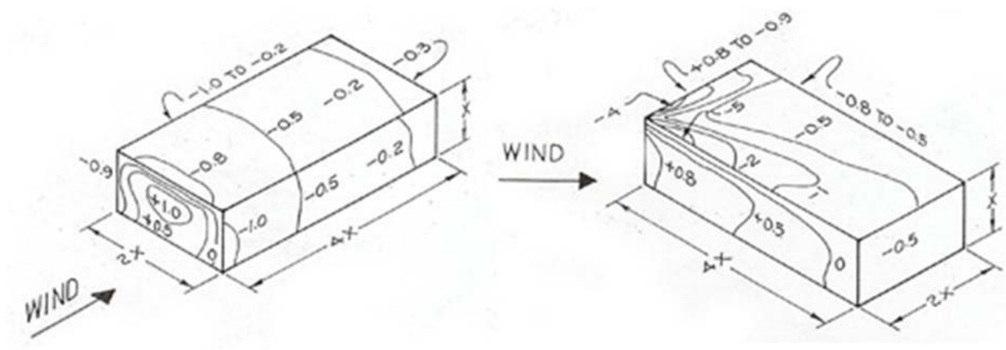


Figure 2.1.1.2: Pressure coefficients on walls and roof of rectangular buildings without parapets (Lstiburek, 2006)

The Stack Effect: The Stack effect is a physical phenomenon that affects movement of air into and out of buildings, as well as other structures. The wind movement caused by the stack effect is created by buoyancy due to air density differences between the outside and inside of the building. In the applications that are relevant to natural ventilation, air density is mainly dependent on temperature and moisture content of the air. The greater the thermal difference and the height of the structure, the greater the buoyancy force, thus creating the stack effect. For the test sites that were selected for these investigations, the stack effect was not an important driving force for ventilation; only cross ventilation was the governing driving force mechanics.

Overall Pressure Driving Force and Bernoulli Equation: The sum of wind and buoyancy induced pressures establishes the overall pressure driving force which is responsible for air movement inside under the natural ventilation process. The fundamental equation that is used to describe air flow in buildings is the so-called Bernoulli Equation. This equation is a simpler form of the Navier-Stokes equation and is applicable for steady, incompressible and non-viscous flow. This equation describes the transport effect upon the fluid along a streamline (e.g. air flow path inside the building) in terms of velocity effects, pressure gradient effects and gravity effects.

$$\frac{1}{2} * \rho * v^2 + P + \rho * g * z = \text{constant}$$

The combined velocity, pressure gradient and density effects are set constant as any energy losses are neglected. In more practical applications, energy losses occur along a streamline.

Humidity Effect in Naturally ventilated buildings: Natural ventilation, unlike fan-forced ventilation, uses the natural forces of wind and buoyancy to deliver fresh air into buildings. Fresh air is required in buildings to alleviate odors, to provide oxygen for respiration, and to increase thermal comfort. However, unlike air-conditioning, natural ventilation is ineffective at reducing the humidity of incoming air. This places a limit on the application of natural ventilation in humid climates.

Design Recommendations for Effective Naturally Ventilated Buildings: Several design recommendation for effective natural ventilation in building design are summarized from various sources. Some of these are presented hereafter:

<u>Site selection:</u> Wind-induced ventilation can be maximized by locating the longitudinal axis of the building perpendicular to the prevailing winds. There are various data sets that provide wind information in regard to wind direction and velocities over extended periods for standard locations such as from the National Oceanographic and Atmospheric Administration (NOAA). A more reliable source of wind data are site specific wind information, such as from local weather stations. The wind regime during the seasons can be quite different. Plants and specifically larger trees or higher hedges can significantly affect the local wind flow.

<u>Cross-ventilation:</u> Wind induced pressure differences on the building can best be used in so called cross ventilation. This means that the building should have effective pathways through the building with large supply and discharge openings.

<u>Narrow buildings:</u> Naturally ventilated buildings should have a relatively small width. This recommendation goes hand-in-hand with the need for good connection, which means unobstructed pathways, for wind to transfer the building. Wide building cause challenges to distribute outside to all zones in buildings using only natural ventilation.

<u>Provide effective air pathways:</u> Windows should be located across the room and offset from each other to maximize mixing within the room while minimizing the obstructions to airflow within the room. This arrangement avoids dead zones and results in smaller energy losses.

<u>Operable openings:</u> Window and other openings should be operable by the occupants. This allows an effective control of the amount and speed of air movement through the space. The occupant's involvement in natural ventilation of a building is one of the key elements that increase satisfaction with indoor environmental quality.

<u>Adequate internal airflow:</u> While the primary consideration is the airflow in and out of the building, airflow between the rooms of the building is important. As a measure to ensure privacy, ventilation between spaces can be through high louvers or transoms.

Ventilation through skylights and clerestories: Skylights and clerestories can be used to take advantage of wind induced pressure differences and through stack effect.

Ridge and attic / roof vents: Removing heat from the building through ridge and attic / roof vents can have significant benefits. A ridge vent is located at the highest location of the roof and provides a good outlet for both buoyancy and wind-induced ventilation.

Fan-assisted ventilation: In mainly cross ventilated buildings natural ventilation can obviously not provide sufficient building ventilation when wind movement provides less than the required pressure differentials to drive air through the building. During these periods of low wind movement, fans can contribute to the indoor comfort by removing heat and providing amounts of fresh air.

Open or closed building ventilation approach: A closed ventilation approach is referred to as using night flushing to cool the interior of the building and then close the building during the hot day hours. The best results of the closed ventilation approach are achieved for buildings with significant interior thermal mass and in climates of low night and hot day temperatures. An open-building approach is best applied in warm and humid areas, where the temperature changes are small from day to night. With open-ventilation daytime cross-ventilation is encouraged to maintain indoor temperatures close to outdoor temperatures. In periods of low external wind movement the daytime ventilation can be aided by mechanical assist ventilation.

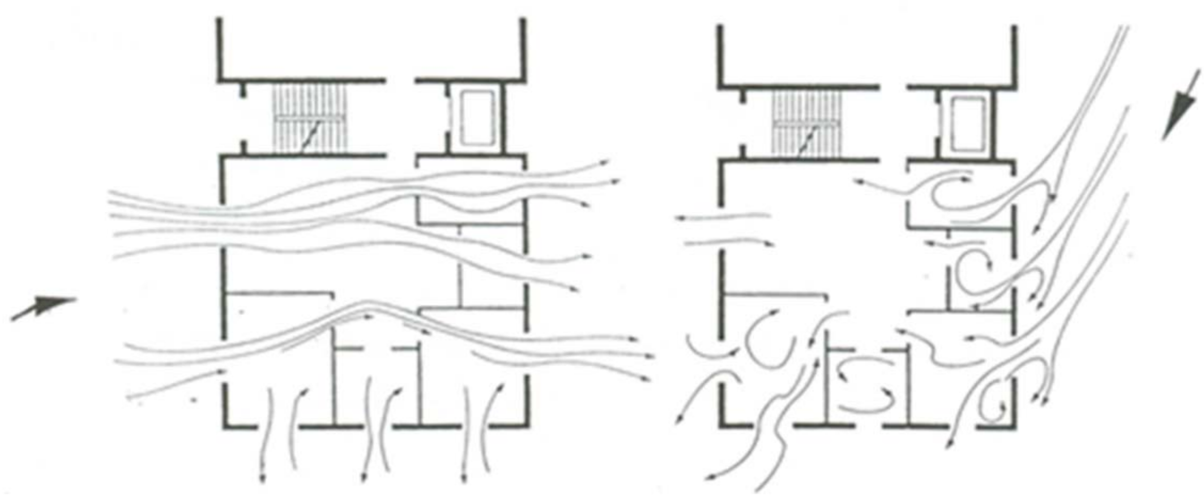
Summarizing Section 2.1.1 : Under favorable conditions, natural ventilation can maintain thermal balance of an internal space when the amount of heat from the room is equal to the heat that enters and is generated inside the space. Natural ventilation is dependent on driving forces that are either wind or buoyancy induced. The effectiveness of natural ventilation is invariably dependent on external environmental conditions, but can be positively affected by specific design practices.

2.1.2 Distribution of Air in Internal spaces

The proper distribution of air in the interior of the building is the most important criterion to gauge the effectiveness of natural ventilation. Optimizing the entire building system, exterior and interior, is more important than only concentrating of individual space components. An efficient natural ventilation performance begins with a suitable orientation and siting of the building itself, since external building conditions and its surrounding (siting) can significantly affect the effectiveness of internal air movement and therefore effectiveness of the natural ventilation.

The horizontal layout should be designed in such a way to maximize effectiveness of cross ventilation. Ventilation through a single opening or at the same side of the building is typically far less effective.

Figure 2.1.2.1 shows an example of a layout that has good cross ventilation for one particular wind approach direction, while under a different wind approach ventilation is less effective. This example illustrates that building layout needs to allow good ventilation performance for a range of wind directions. In areas where there is a predominant wind direction layout should be optimized for this wind approach while making provisions for secondary wind approach.



Good cross ventilation for incident wind direction

Poor cross ventilation for incident wind direction

Figure 2.1.2.1: Effectiveness of cross ventilation under different wind directions

The vertical layout is primarily affect by stack effect ventilation driving forces. There might be situations where ducting between floors is used to take advantage of cross ventilation pressure differentials but these cases will most likely be few. In order to enhance the stack effect through wind induced low pressures the opening of the stacks should be positioned to areas of low pressures.

Summarizing Section 2.1.2 : The internal layout of a building or space has significant effect on the effectiveness of natural ventilation. Horizontal layout is important for cross ventilation and vertical layouts is important for buoyancy forces.

2.1.3 Mechanical Assist Ventilation Systems

During periods when external wind conditions are insufficient to generate the driving forces for natural ventilation, (e.g. the required pressure differentials), mechanical assist ventilation can safeguard that adequate ventilation is provided. The reviewed literature suggests that the type of ventilation includes both mechanical and sustainable technologies. These technologies comprise factors that can be characterized as follows:

- Continuous supply and exhaust systems
- Intermittent supply and exhaust systems
- Combined exhaust and supply (Balanced)
- Infiltration with operable windows
- Passive Stack Ventilation
- Solar Chimney
- Hybrid Systems

Mechanical whole-house ventilation: A variety of mechanical whole-house ventilation approaches are in use, either individually or in combination. These include exhaust, supply and balanced systems which operate either continuously or intermittently. In conventional buildings, mechanical ventilation strategies provide more uniform ventilation rates than natural ventilation. Ineffective pathways for internal airflow can be solved by adding ducts and supplying the required pressure differentials with a driven fan. The disadvantage of mechanical ventilation is the additional energy required to operate the system and the need to properly maintain the system. The great challenge of combining natural and mechanical ventilation systems is that buildings are best designed to operate under either natural or mechanical ventilation modes. Mechanical ventilation systems are typically designed and operated with significantly more pressure differentials than natural ventilation.

Exhaust systems: Whole-house exhaust systems provide ventilation by using one or more fans to remove air from the building. As the air is extracted from the building supply air enters the building envelope through gaps (infiltration) or planned openings. When considering infiltration, supply air enters the building in an uncontrolled manner and may be pulled in from the outside air or from relatively undesirable areas such as garages or other areas with potential contaminants (i.e. dust). Figure 2.3.1 shows the working principle of an exhaust system.

Whole-house exhaust systems can be built in different configurations:

A single point exhaust system has the lowest construction and installation, basically only one fan and possibly some simple exhaust ducting to the outside. The disadvantage of single-point ventilation systems is the non-uniform distribution of fresh air especially to closed rooms.

Multi-point exhaust systems provide better distribution than single-port exhaust systems and therefore they improve room-to-room ventilation uniformity. The disadvantage of multi-point fan systems is the extra cost of installing ductwork between rooms and the central exhaust fan. Reduction of noise levels of the fan can be achieved by installing the fan remotely.

Often the intermittent exhaust system consists of one central fan to remove stale air from the building, but may also incorporate several fans in areas of high usage (i.e. bathrooms and kitchens). In this case, the fan(s) run only part of the time at a higher rate and are sized to provide the necessary ventilation. Since the exhaust fans do not operate all the time, the system has to be designed to handle a larger capacity than does the continuous system.

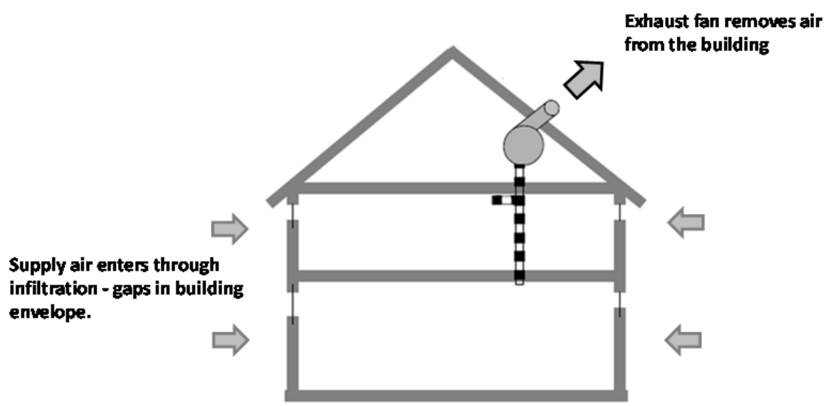


Figure 2.1.3.1: Basic mechanical exhaust system where supply air enters through infiltration

Supply Systems (positive pressure supply systems): Supply systems provide pressure to drive outside air into the building. Air is supplied by a central fan ducted to some or all of the rooms. The outside air supply forces air in the rooms out of the building through leaks in the envelope or by dedicated ducts. By regulating the ducts, supply systems allow the occupant to control the location of the supply air to maximize air quality. Different from exhaust systems, filtering the supply air is possible.

Balanced systems, suggested in the literature, represent a combination of exhaust and supply systems where two fans with separate ducting systems are used. One ducting system supplies outside air and the other removes stale air from the building. Balanced systems are well suited to install an energy recovery system, where heat energy is recovered. Figure 2.3.2 shows a typical flow scheme of a balance system with heat recovery.

Energy and Costs: Whenever mechanical ventilation is used, energy is required to move the air and, if applicable, condition the supply air. The energy for only mechanical ventilation, without mechanical cooling, can account for as much as one third to one half of the total space conditioning. Using mechanical ventilation without cooling can conserve energy while still providing healthy ventilation rates. These measures can include avoiding unnecessary air changes (due to leaky buildings), using effective control strategies (not opening windows during periods of heating and cooling), and optimizing fan and equipment efficiencies. Mechanical ventilation systems can save energy used to condition supply air if the building envelope is tight and infiltration is limited. Energy consumption can be reduced 9 to 21% by installing a mechanical ventilation system with heat recovery. A mechanical exhaust system is most likely the least expensive to operate. Retrofitting an existing house is more expensive than new construction and multi-point distribution systems are more expensive than a single point system

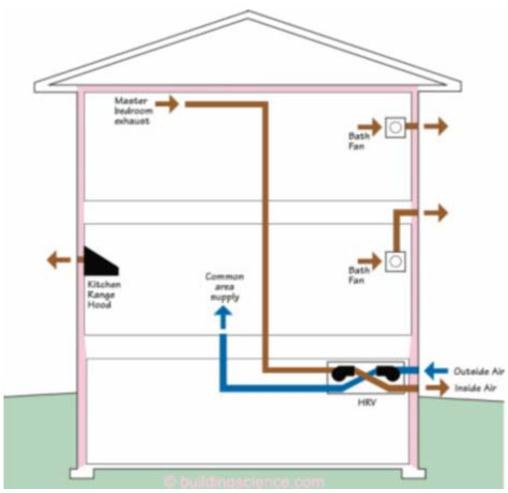


Figure 2.3.2: Balanced supply and exhaust system with heat recovery

<u>Summarizing Section 2.1.3:</u> Mechanical ventilation can safeguard the required ventilation rates at virtually any outside environmental conditions. Since natural ventilation effectiveness is inherently intermittent, mechanical ventilation can provide important redundancy. Mechanical ventilation systems work with establishing a differential pressure against the outside air to which inside air is discharged. Mechanical ventilation can be either supply or discharge systems, or a combination thereof.

2.1.4 Ventilation Opening

The reviewed literature provided information about four categories of building envelope openings that are of importance to natural ventilation:

- Windows: Different types of windows can be categorized according to:
  - Plane of placement (e.g. vertical, non-vertical, horizontal, tilted)
  - Position of window in the building envelope
  - Opening systems (swinging, pivoting)
- Screens: Can serve as ventilation as well as shading device.
- Doors: Provide interconnections between the exterior and interior and between interior spaces.
- Vents and ventilator: Function as means to enhance and direct air movement

Windows: Windows are important air flow control devices. Their effectiveness depends on the size and also on type, or the way they are opening. There are three basic types of windows, single opening, vertical-vane and horizontal vane windows. The air flow patterns generated by the different windows during their operation can significantly affect the effectiveness of natural ventilation.

- Simple openings typically do not affect the velocity and direction of the air flow except near the opening. (see Figure 2.4.1)
- Vertical vane windows exert a wide variety of influences on both pattern and velocity of the airflow. The most common type, the side-hinged casement window, has a great versatility with regard to airflow control. (see Figure 2.4.2)
- Horizontal vane windows typically affect the velocity and direction of the air flow mainly in the vertical direction. Projected sash and other horizontal-pivot windows direct the airflow upward. Jalousie windows have the possibility to direct the airflow in either direction according to the position of the sashes. (see Figure 2.4.1)

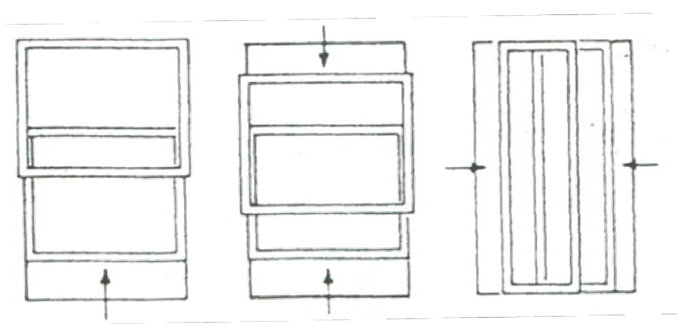
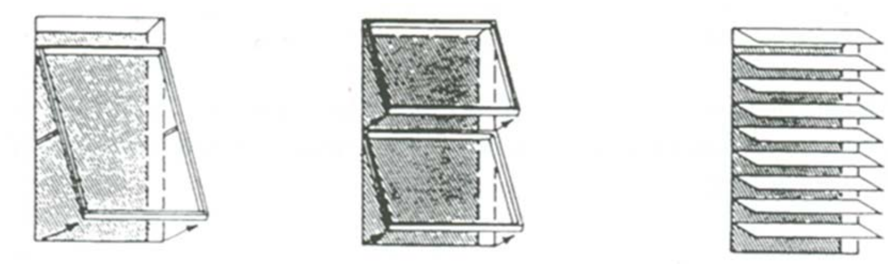


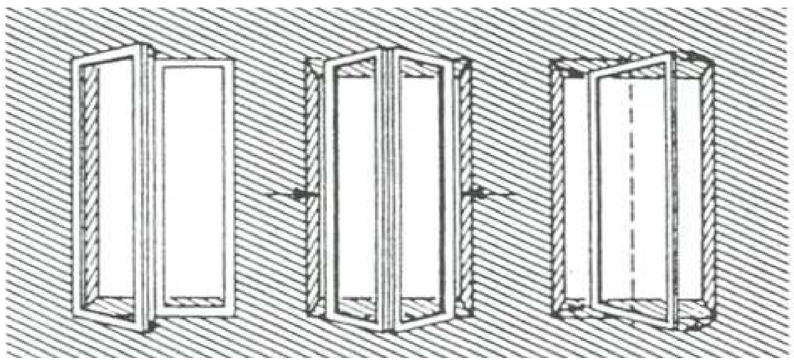
Figure 2.1.4.1: Simple opening windows

Single hung Double hung Horizontal hung



Sash window   Awning window   Jalousie window

Figure 2.1.4.2: Typical horizontal vane windows



Side-hinged casement   Folding casement   Vertical pivot

Figure 2.1.4.3: Typical vertical vane windows

<u>Screens:</u> Screens can be either fixed or operable screens. Fixed screens are usually externally fixed and function for shading, security or insect deterrents. Operable screens are either exterior or interior screens.

- <u>Fixed exterior screens</u> (i.e. insect screen, safety grilles, etc.) are reducing the approaching airflow in a uniform way over the entire surface, without changing the direction of the airflow.
- <u>Operable exterior screens</u> (i.e. rolling blinds, awning frame, sliding sash shutter, etc.) typically, slow the airflow as it passes through the screens and they also change the airflow direction. Awning-frame rolling blinds have the best performance as they integrate good shading with good airflow performance.
- <u>Operable interior screens</u> (i.e. vertically and horizontally operating curtains, louvers, etc.) have less airflow deflection and more wind energy absorption, owing to their relative lightness and softness.

Doors: Doors are normally not used as dedicated airflow control device in the same manner as windows are. The function of doors is significantly different than windows and normally doors remain closed, except perhaps doors of businesses which have a considerable number of customers passing through them. Exterior doors can be effectively used as ventilation openings if the door is equipped to let air through but not insects or intruders. Many exterior doors cannot be left open without automatic shut controls, because of code requirements such as fire-safety doors. If doors should be used as a ventilation opening an operational schedule has to be considered.

Vents: Vents can be distinguished in non-mechanical and mechanical openings.

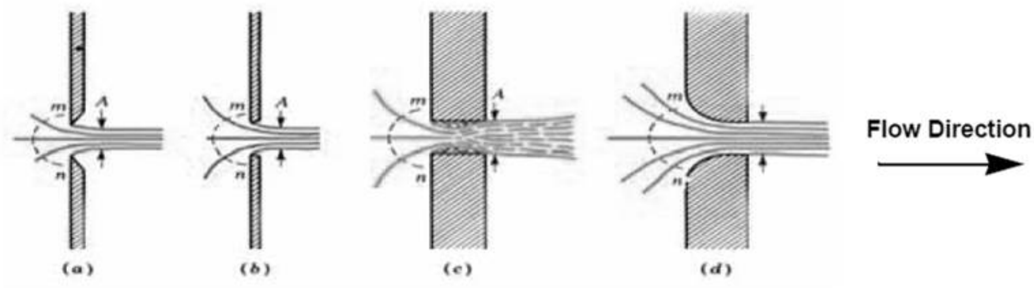
Summarizing Section 2.1.4 : Openings in the building envelope provide pathways for natural ventilation. Types of opening exert different amounts of resistance to the inflow and out flow of air.

2.1.5 Pressure Losses of Building Openings and Airflow Obstructions

The knowledge of pressure losses along the air passageways through the building is an important design factor for naturally ventilated buildings. Pressure losses occur at external openings and at locations inside the space, where the air pathways are subjected to flow obstructions and/or flow diversions. With the same exterior pressure differentials between sections of the envelope, the pathway with the least pressure losses, or flow resistance, will have higher airflows through the space.

Pressure losses are directly proportional to the changing discharge coefficients and a function of air velocity squared. This means that given the same discharge coefficient for an opening, twice the air velocity will result in four times the pressure losses. As an example, Figure 2.1.5.1 shows discharge coefficients for orifices (noted as C_D) with the same diameter but with different inlet geometries. The objective of creating effective internal pathways for natural ventilation is to lower pressure losses.

Tables are available that correlate so-called minor loss coefficients to absolute pressure losses. (see Figure 2.1.5.2). Typically these tables apply to air ducts in conjunction with HVAC systems. The literature provides minors losses for typical flow obstructions in air ducts, such as sharp and rounded 90° bends, as 0.5 and 0.25, respectively. The concept of minor losses and associated pressure losses can also be applied to internal air pathways for natural ventilation. Figure 2.1.5.3 shows the relationship of air flow through architectural louvers.



$C_d = 0.62$ $C_d = 0.62$ $C_d = 0.86$ $C_d = 0.98$

Figure 2.1.5.1: Discharge coefficients for orifices (noted as C_d) with the same diameter but with different inlet geometries

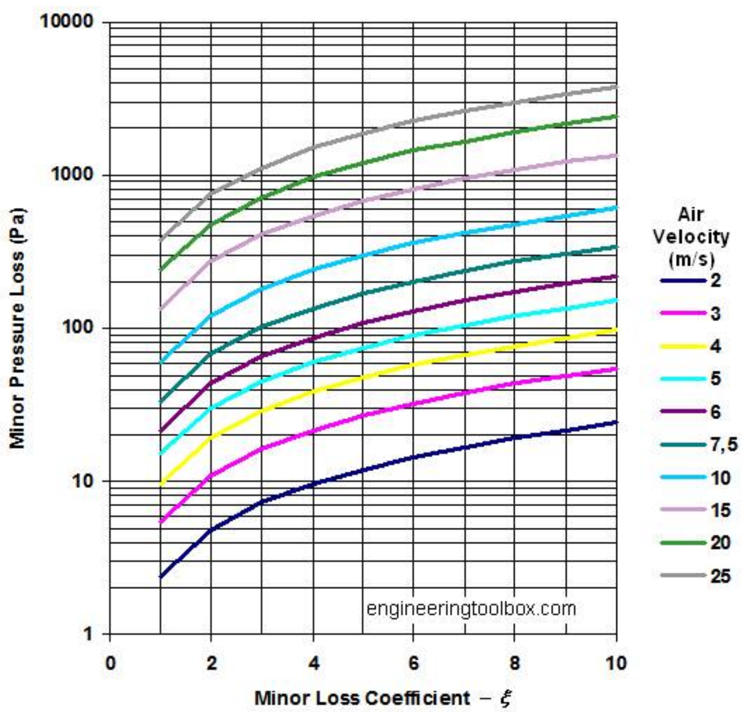


Figure 2.1.5.2: Conversion of minor losses to minor pressure losses

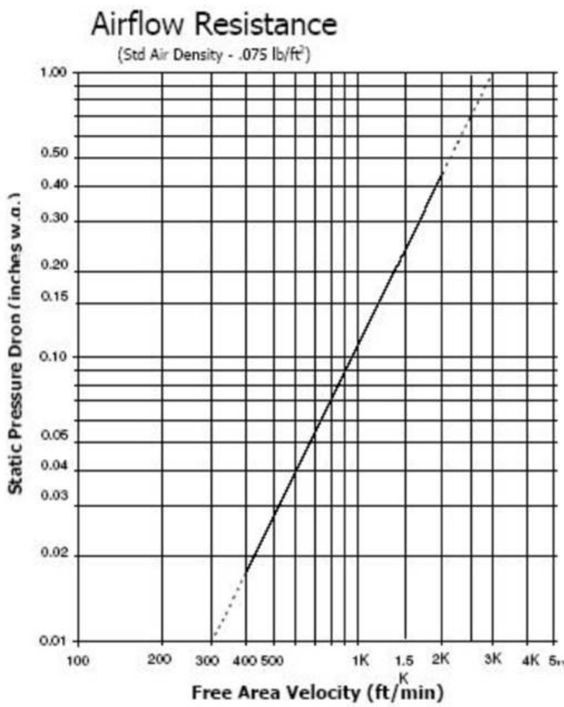


Figure 2.1.5.3: Airflow resistance of architectural louvers

Summarizing Section 2.1.4 : The pressure loss along a streamline can be evaluated by adopting a known functionality referred to as the Bernoulli Equation. Pressure losses caused by individual obstructions or changes in the air flow can be expressed by the product of a friction loss factor multiplied by an expression that is a function of velocity squared.

2.2 CFD as a Design Tool for Natural Ventilation Design

Over the past decades the role of CFD as a tool to assess effective solutions has been investigated by numerous building science practitioners. Most of the studies concluded that CFD is a promising design for naturally ventilated buildings, but several authors in the literature review caution that CFD, being a simplified, and sometimes oversimplified, virtual model of the actual conditions, should be regarded with some skepticism. The argument was made that a good CFD practitioner should have good analytical and physical skills to interpret the results. Almost all of the studies recognize the need to validate the results with either scaled wind tunnel testing or testing in the full scale prototypes.

The use of CFD is consistent with expanding traditional scientific methods with a newer paradigm, the use of computers and simulation. The traditional scientific method has two foundations; experimental and theoretical. The newer model includes the role for computing and simulation as an extension of the traditional processes. This newer framework established relationships between the processes of scientific model development, computational model verification, and simulation validation.

Each physical experiment, either in the wind tunnel or at the full scale prototype, or theoretical simulation, has its unique advantage and disadvantages.

A physical fluid modeling study employs “real fluids”, not models of fluids; hence, the fluid model is implicitly non-hydrostatic, turbulent, includes variable fluid properties, non-slip boundary conditions, and dissipation and even flow separation and recirculation. All conservation equations are automatically included in their correct form without truncation or differencing errors, and there are no missing terms or approximations. Model studies, while using the same “real material”, are scaled models with necessary simplifications, and are prone to certain inaccuracies due to similitude.

CFD modeling, on the other hand is not intrinsically limited by similitude or scale constraints, although CFD analysis has many limitations associated with grid resolution, choice of turbulence model, or assignment of boundary conditions. One tremendous potential of CFD is the ability of presenting “colorful and attractive” results and obtaining a solution at any point of the domain. Yet the accuracy of the CFD results is directly dependent on the choice of domain resolution, turbulence models, and boundary conditions. Caution has to be used about the tendency displayed in many CFD applications to believe implicitly in the realism of the beautiful graphical displays. Instead continued verification and validation at almost every level of CFD prediction should be applied.

Refinements in wind tunnel testing techniques such as boundary layer simulation and improvements in modelling wind characteristics can usually give reasonable estimates of ventilation rates. However, as the authors point out, factors such as complex topographic features, scaling errors, and influence of architectural features can restrict the accuracy of any estimate.

Summarizing Section 2.2 : CFD is a becoming an increasingly powerful tool to predict air movement around and inside buildings. It is important for the CFD practitioner to understand the need for verification and validation of results, so that results can be used for design decisions, experimental ventilation studies, either scale models or full scale prototypes, have their own limitation and also need interpretation. The significant advantage of CFD is the avoidance of possibly expensive model studies and the ability to determine air movement properties at any point in the domain.

2.3 Internal CFD – Numerical Assessment of Air Movement Inside Buildings

The literature review revealed that the process of using CFD for ventilation studies includes a wide range of software (solver) settings and procedures. In the following section, two important categories are presented. These categories guided the CFD team through the current investigations.

2.3.1 Best Practices and Quality Control

Best practices have been developed that provide important guidance to CFD practitioners to ensure quality control over the CFD process. The specific main steps are as follows.

Definition of the problem: The important first step is to define clearly the physical objectives of what has to be simulated.

- What do we want to obtain from the prediction?
- Which are the steps on the way to the final answer?
- How are these single problems defined in the simulation process?

Besides selecting the simulation approach, e.g. steady or transient, 2D or 3D, nature of the fluid, symmetry) a crucial issue is how we apply the question of degree of simplification of the model used for the CFD simulation. The CFD practitioner needs to determine to what a degree of the model can be simplified while it is still yielding representative values. Complicated model geometry might render some interesting details in the flow regime, but on the other side a complicated model typically requires significantly more computational resources than simple model geometry. The question how much simplification is required depends on the questions that should be answered in the investigation.

If the nature of the CFD analysis is air intrusion in the vicinity of a window, then simplification can reduce the amount of objects to be modelled inside the space. If an intricate airflow in the room, involving complex inlet and thermal plumes, is required much more interior objects should be included. Much care has to be taken that simplification matches the objectives and the computer resources involved. As an illustration Figure 2.3.1.1 (a) and (b) shows a room with all furniture, heat sources and flow obstructions and only with the radiator under the window modelled, respectively.

(a) Office situation with all details

(b) Simplified office situation

Figure 2.3.1.1: Comparison of situations to be modeled with different degrees of simplification

Meshing or computational grid: An important consideration is the computation grid (or mesh) that is being used for the CFD analysis. The issues that are important are:

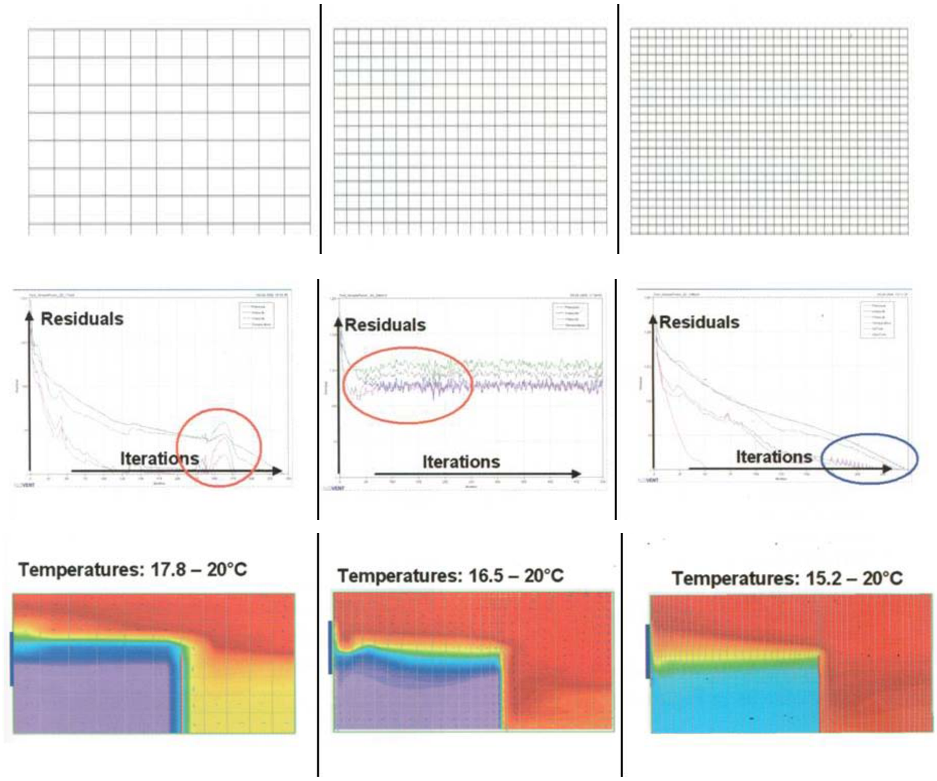
- Number of cells (resolution)
- Cell distribution
- Cell quality

Great care has to go into creating a quality grid. In our present CFD research work, a 3D-CAD model was created outside the CFD software and was imported to create the computational grid. Grids with different resolutions can have significantly different characteristics of convergence and representations of fluid flow and heat properties. Figure 2.3.1.2 illustrates three different grid resolutions and the convergence and fluid flow representations of their respective solutions.

Grid resolution >>>>

Convergence performance >>>>

Representation of fluid flow properties >>>>

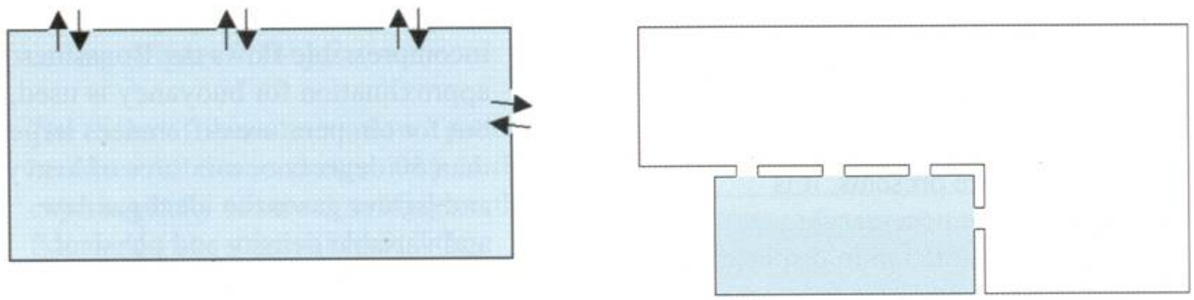


(a) Coarse grid (b) Medium grid (c) Fine grid

Figure 2.3.1.2: Three types of grids used in simulating flow in simple room geometry

Turbulence models: There are various turbulence models that can be chosen. Typically high-level turbulence models require more computational resources. The commonly used k-ϵ-model has limitations that can be overcome with the use of SST model. Disagreements with measurements can sometimes, but not always, attributed to the use of turbulence models .

Boundary conditions: The correct selection of boundary conditions is one of the most important sources for errors, since it can result in computational instability. It might be necessary to try several numerical possibilities and make an "informed" judgment which boundary condition is appropriate. In the case of modeling free flow opening the CFD research team considered the use boundary conditions which are depicted in Figure 2.3.1.3, where the outer domain is large enough so that the settings at its boundaries do not influence the flow through the connections to the room.



(a) Incorrect boundary conditions for free flow openings

(b) Correct boundary conditions by enlarging the computational domain (area in blue is the smaller domain shown in (a)

Figure 3.1.10: Examples of boundary conditions in free flow openings

Summarizing Section 2.3.1 : Best Practices guidelines can help CFD practitioners to prepare, conduct and post-process CFD simulations. Following proposed guidelines ensure that appropriate care is exercised for the CFD simulation and quality control is applied.

2.3.2 Domain Decomposition and Coupled CFD Analysis

There are two domain approaches to predict internal airflow phenomena through the use of CFD analysis. The first approach is called “domain decoupling” or “decoupled” approach, where the exterior of the building is first analyzed and the results in regard to pressure distribution and airflow. These results are then used as boundary condition for the CFD simulation of the interior space. The advantage of the decoupled approach is that it requires fewer computational resources than the coupled approach. The disadvantage is that internal airflow cannot be precisely predicted, especially close to free flow openings.

The second approach is the so-called “coupled” approach where the exterior and interior space is combined into one computational domain. In CFD simulations of cross-ventilation, involving large openings as in the case of our present investigations, a major issue of concern is the accurate modeling of the interaction between the outdoor wind flow around the buildings and the indoor air flow inside the buildings, which interact with each other at the ventilation openings. Figure 2.3.2.1 illustrates the difference between the coupled and uncoupled domains as used in CFD simulations.

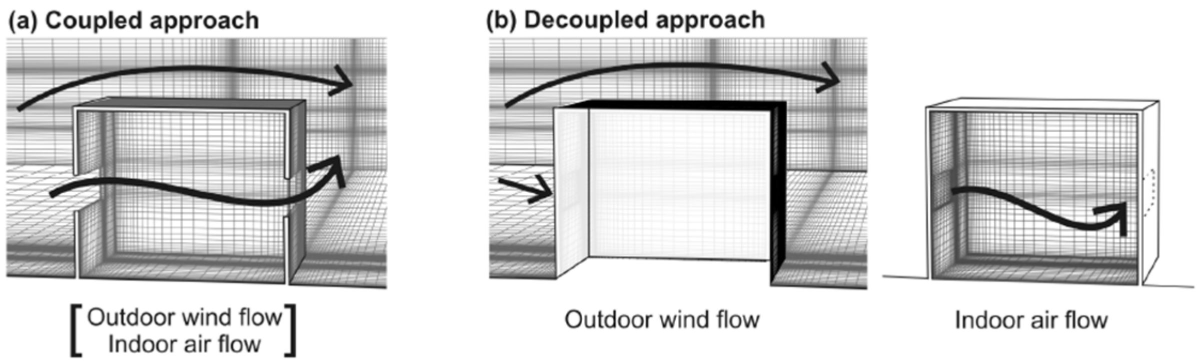


Figure 2.3.2.1: (a) Coupled and (b) decoupled approach for analysis of wind-induced cross-ventilation of buildings

The coupled domain requires significantly more computation resources and that can be a challenge. However, coupled domains have been used for very large models, such as depicted in Figure 2.3.2.2. Figure 2.3.2.3 compares the resulting air flow of two inlet and outlet conditions of a simple building structure.

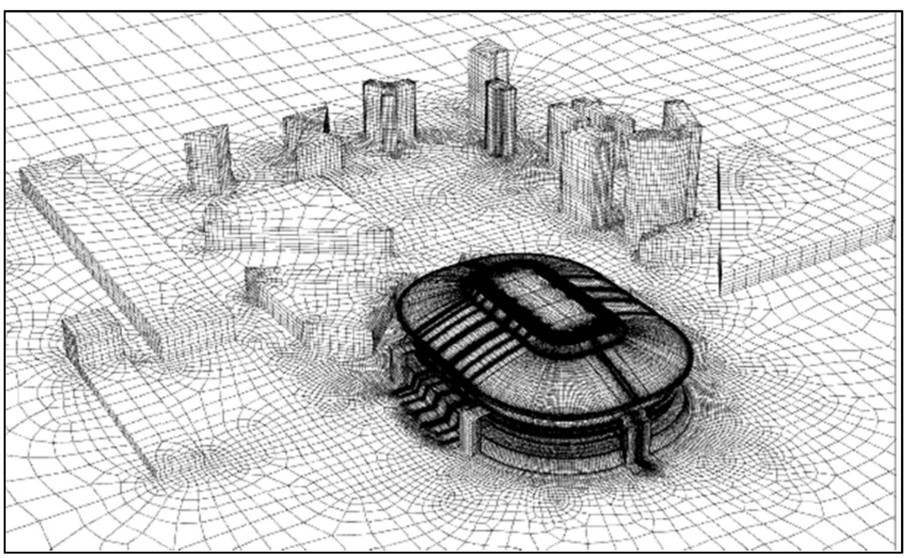


Figure 2.3.2.2: Computational grid (about 150 million cells) on the building surfaces and part of the ground surface. A high resolution is used in the proximity of the stadium, and a lower resolution at a larger distance from the stadium.

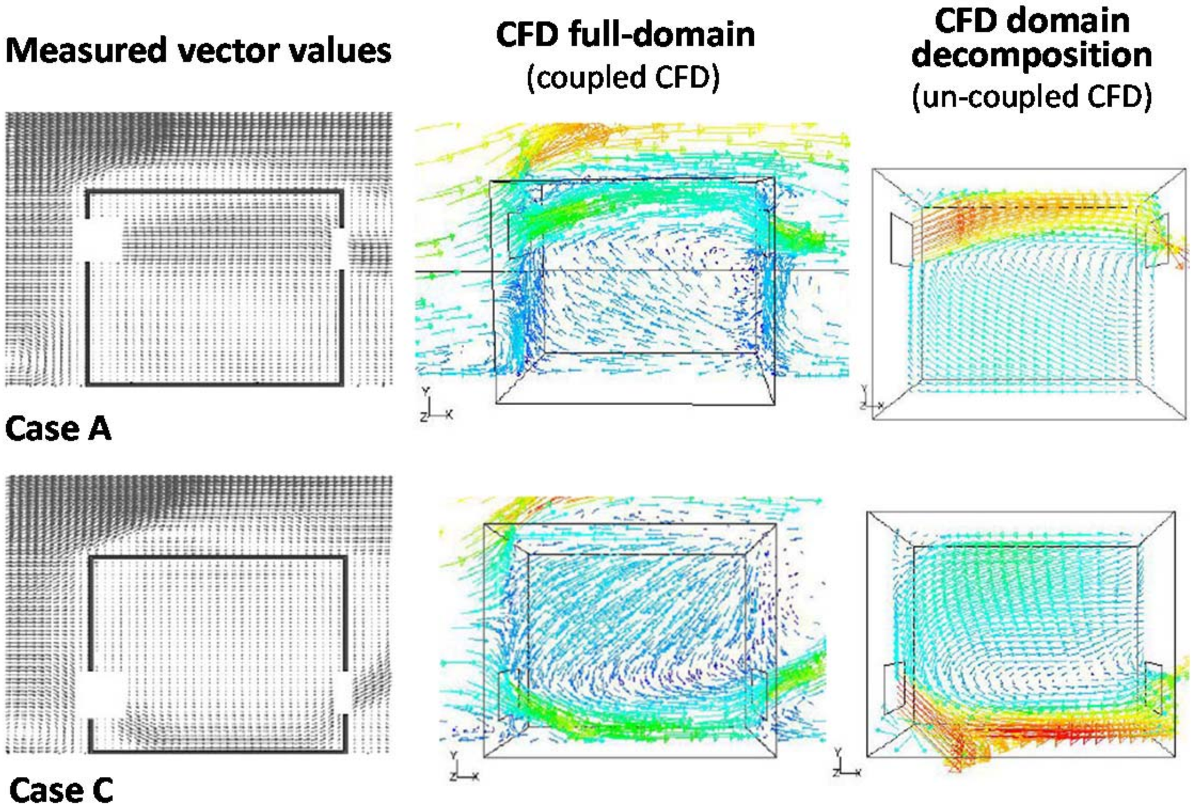


Figure 2.3.2.3: Velocity vectors on building vertical midplane for two case Cases, obtained from measured air flow and coupled and uncoupled flow domains

<u>Summarizing Section 2.3.2:</u> Both the coupled and de-coupled domain approach have merits in CFD applications. The decoupled domain requires less computation resources. The coupled domain approach can provide important flow description, especially for natural ventilation. The decision if coupled or de-coupled domains are used, rests with the CFD practitioner, and is a case by case decision.

SECTION 3 - DEVELOPMENT OF AN INTERNAL CFD WORK PROCESS

Before the internal CFD investigations of a selected naturally ventilated building on the University of Hawaii Manoa campus were conducted, the CFD research team develop and tested a workflow for internal CFD that was could be used for the final internal CFD simulations. The investigations in conjunction with the workflow development included as follows:

- Type of computational domain, coupled or decoupled
- With or without furniture (structural elements such as columns were modeled in every case)
- Type of turbulence model
- Type of boundary condition for inlet of the domain
- Type of boundary condition for outlet (exhaust fans) of the internal space
- Air density and viscosity; either standard conditions (= CFD defaults) or value calculated from measured data at the site

The objectives of this phase of project work were as follows:

- Evaluate CFD simulations of internal air movement and validate theoretical CFD predictions with actually measured air flow conditions in a controlled test environment.
- Utilize a large room on the University of Hawaii Manoa campus, Room 301 in Sinclair Library, to carry out full scale validation test of internal CFD simulations. Room 301 was equipped with a temporary exhaust fan systems. This enabled the ERDL project team to assess the contribution of mechanical assist ventilation to the otherwise naturally ventilated Room 301.
- Conduct CFD benchmark simulations with varying CFD setting parameters, such as domain configuration, mesh configuration, solver setting and turbulence models, to determine the effects of these parameters on the convergence and air flow description performance.
- Determine the effectiveness of different post processing procedures of CFD simulations to obtain qualitative and quantitative determination of the CFD solutions.
- Use the measurement procedures for air velocity and pressure distribution in internal air flow, developed under the project task “Task 7.b.3: "Development and calibration of a data verification process for internal CFD simulations"
- On the basis of experience of the CFD work performed for this project phase, develop preferred domain geometry, mesh generation and CFD solver setting that should be used in subsequent internal CFD analysis.

Summarizing Section 3.0 (Preamble): The objective of the work presented in the Section 2 was to develop a workflow and technical CFD procedures that would be applied at the final internal CFD investigations.

3.1 Technical Aspects of the CFD Workflow

The technical phases and aspects of the internal CFD workflow process are illustrated in Figure 3.1.1. The internal CFD workflow process includes the following main phases:

1. Creation of the geometry of the model and boundaries: The 3D-geometry of the model and the boundaries of the computational domain are created with an appropriate modeling program. The geometry is then imported into the CFD software application.
2. Preprocessing: Meshing and setting Physics: This phase of the generic workflow included surface preparation of the imported 3D-CAD geometry, creation of surface and volume grid and the setting of the physical parameters for fluid flow analysis
3. Solution / Solver: This phase included considering the turbulence model, relaxation and convergence criteria.
4. Post processing: This includes the appropriate representation of the CFD results as quantitative and qualitative plots, graphs and data.

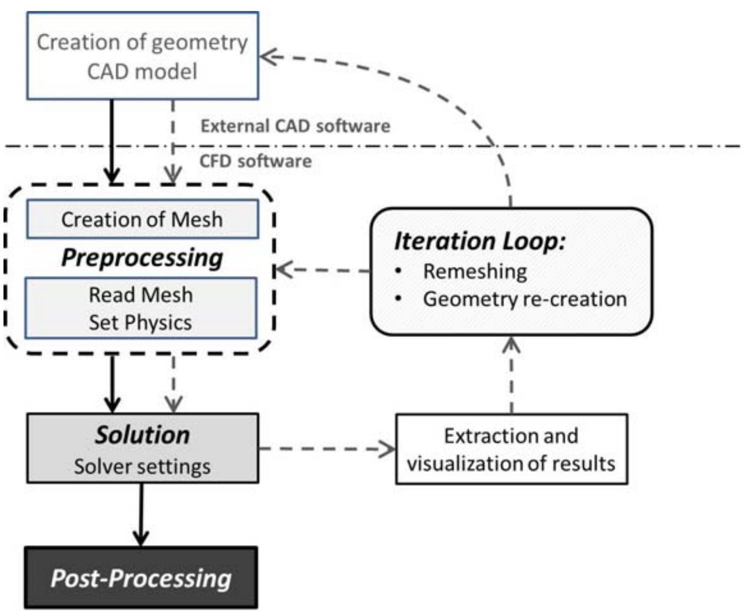


Figure 3.1.1: Technical phases and aspects of the internal CFD

3.2 Test Site

The workflow for internal CFD investigation was tested in a Room 301, a 5,000 sqft naturally ventilated space, inside the 123,000 sqft Sinclair Library, on the University of Hawaii Manoa campus. Figure 3.1.1 depict the test site and installed two temporary exhaust fans.

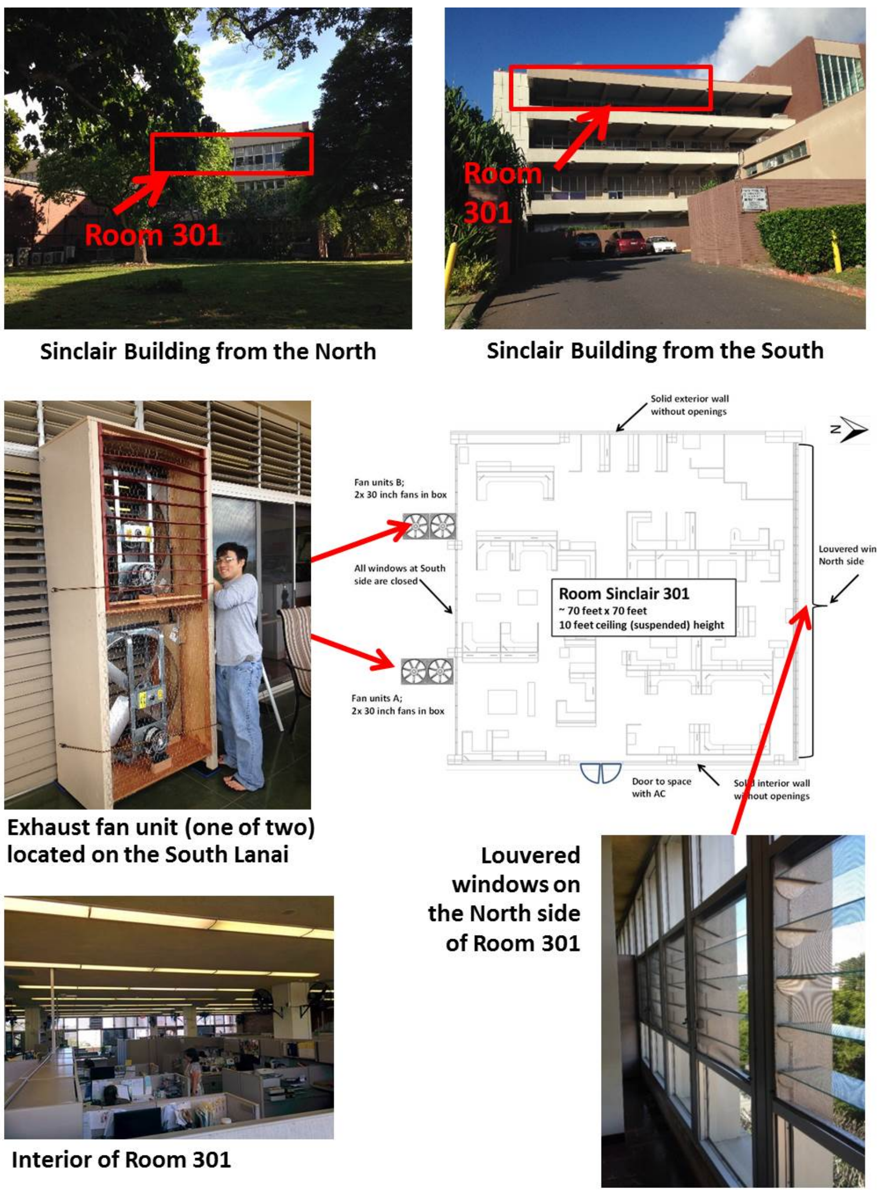


Figure 3.2.1:Test site for investigation to develop and test the workflow for the internal CFD

3.3 Creating the Model Geometry

The CFD investigation included testing a coupled and decoupled approach. Figures 3.3.1 and 3.3.2 show the 3D-geometries of the decoupled and coupled domains.

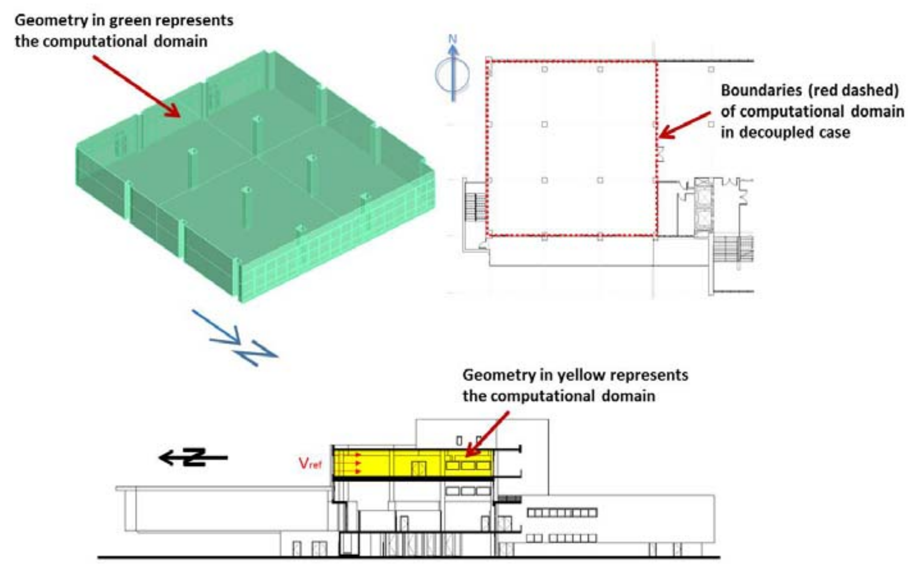


Figure 3.3.1: Geometry modeled for the decoupled domain case

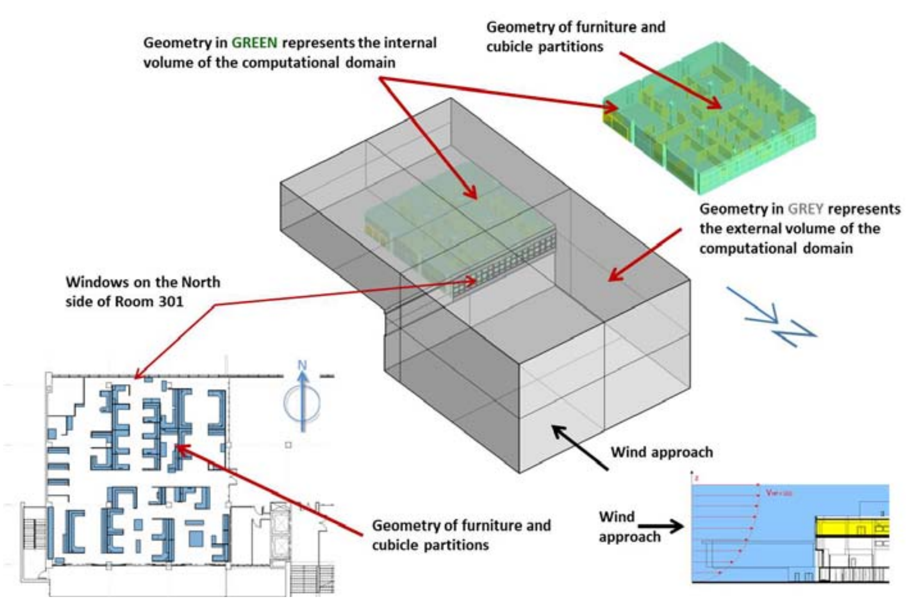
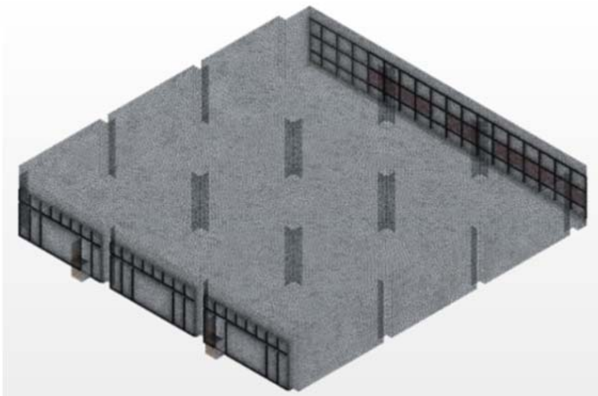


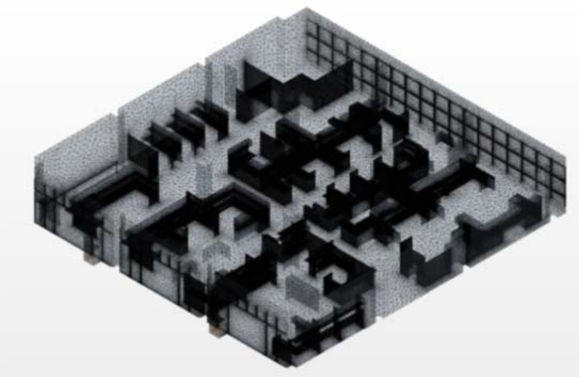
Figure 3.3.2: 3D CAD model illustration of the coupled domain CFD; case with furniture
Note: The coupled domain had furniture and cubicle walls included

3.4 Meshing

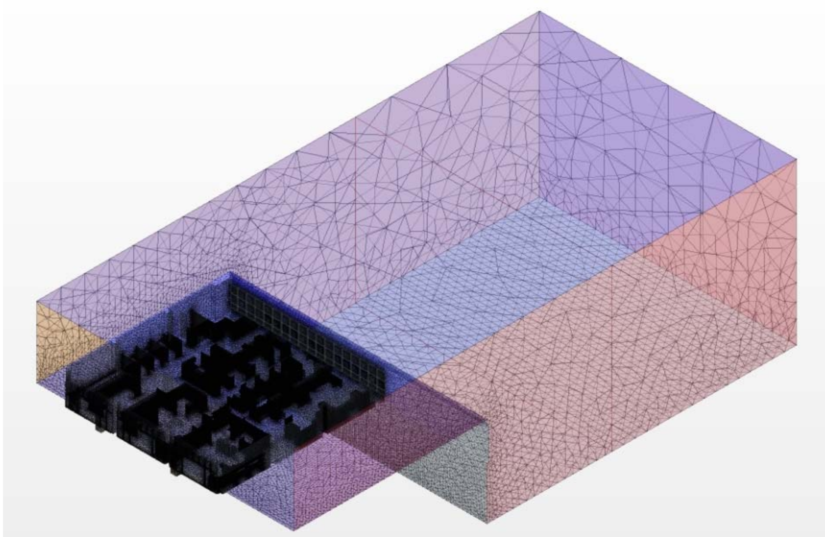
A range of meshes were created to benchmark grid resolution and related convergence performance. Figure 3.4.1 illustrates meshes used for the coupled and uncoupled domain approach used in the tests.



a) Surface mesh generated in the Decoupled Domain modeling approach; computational domain is limited to the internal space



b) Surface mesh generated in the Coupled Domain (with furniture) modeling approach; only internal space is shown as part of the computational domain.



c) Surface meshing of coupled domain modeling approach, internal AND external portions of the domain are shown.

Figure 3.4.1: Illustration of surface meshing of decoupled domain modeling approach versus coupled domain modeling approach

3.5 Scope and Structure of CFD Test Scenarios

Approximately 40 initial CFD runs were conducted to narrow the documented benchmarking process to 18 test cases. The parameter matrix for the 18 CFD simulation test cases is depicted in Figure 3.5.1.

Each simulation test case represents a combination of the following parameters:

1. Type of computational domain, coupled or decoupled
2. With or without furniture (structural elements such as columns were modeled in every case)
3. Type of turbulence model
4. Type of boundary condition for inlet of the domain
5. Type of boundary condition for outlet (exhaust fans) of the internal space
6. Air density and viscosity; either standard conditions (= CFD defaults) or value calculated from measured data at the site

Solution ID	Coupled / Decoupled	With furniture	Turbulence model	Inlet	Outlet	Air density and dynamic viscosity	Test Scenario
A	Decoupled	No	k-ε	Constant velocity	Pressure outlet (0 Pa)	default	N/A
A1	Decoupled	No	k-ω	Constant velocity	Pressure outlet (0 Pa)	default	N/A
A2	Decoupled	No	k-ε	Constant velocity	Constant massflow rate (4.359kg/s)	default	N/A
B	Coupled	No	k-ε	Wind log profile	Constant massflow rate (4.359kg/s)	default	N/A
B1	Coupled	No	k-ε	Wind log profile	Fan curve	default	N/A
B2	Coupled	No	k-ω	Wind log profile	Fan curve	default	N/A
B3	Coupled	No	k-ε	Wind log profile	Fan curve	Actual test condition	N/A
C	Coupled	Yes	k-ε	Wind log profile	Fan curve	default	N/A
C1	Coupled	Yes	k-ω	Wind log profile	Fan curve	default	N/A
C2	Coupled	Yes	k-ε	Wind log profile	Fan curve	Actual test condition	N/A
D1	Coupled	Yes	k-ε	Wind log profile	Fan curve	Actual test condition	Scenario 1
D2	Coupled	Yes	k-ε	Wind log profile	Fan curve	Actual test condition	Scenario 2
D3	Coupled	Yes	k-ε	Wind log profile	Fan curve	Actual test condition	Scenario 3
D4	Coupled	Yes	k-ε	Wind log profile	Fan curve	Actual test condition	Scenario 4
E1	Coupled	Yes	k-ω	Wind log profile	Fan curve	Actual test condition	Scenario 1
E2	Coupled	Yes	k-ω	Wind log profile	Fan curve	Actual test condition	Scenario 2
E3	Coupled	Yes	k-ω	Wind log profile	Fan curve	Actual test condition	Scenario 3
E4	Coupled	Yes	k-ω	Wind log profile	Fan curve	Actual test condition	Scenario 4

Figure 3.5.1: CFD study model matrix

The following presents a brief discussion about the six parameters used in the matrix in Figure 3.5.1.

Coupled / decoupled domain: A coupled domain approach was the preferred approach. The decoupled case was investigated as a benchmark comparison. The resulting meshes for the three domains and geometries are described in Table 3.5.1. As can be seen in the table the coupled domain with furniture represents a significantly larger mesh than the other two domain cases.

Table 3.5.1: Number of volume cells for meshes used in three different CFD domains

	Decoupled domain	Coupled domain without furniture	Coupled domain with furniture
Total number of cells	2,481,517	4,347,720	15,820,425

Turbulence models: Two turbulence models were used in the CFD simulations runs, the Realizable two layer k-ε model and the SST k-ω turbulence model. Various forms of the k-epsilon (k-ε) turbulence model have been in use for several decades, and it has become the most widely used model for industrial applications. One of the improved versions of the standard k-ε model is the Realizable two layer k-ε, used in this study. The k-ω model used in the study is the SST k-ω turbulence model. Using this model can avoid problems of sensitivity of the flow to inlet free-stream/inlet condition, which the standard k-ω model is prone to. In the SST k-ω model, the ε transport equation from the standard k-ε model is transformed into an omega transport equation by variable substitution and adding an additional non-conservative cross-diffusion term.

Inlet and outlet boundary conditions: The boundary conditions of the three computational domains included the atmosphere boundary layers (logarithmic profile wind for velocity inlet, constant pressure outlet), the ground surface, surfaces inside building (including cubicle walls) and pressure outlet with electric fan performance curve. The inlet boundary condition considered a logarithmic wind profile. The outlets in the South and West of the coupled domain are pressure outlets with zero pressure relative to the ambient pressure. The air outlet through the discharge fan units considers a fan curve. Figures 3.5.2 and 3.5.3 depict the inlet and outlet boundary conditions used in the simulations.

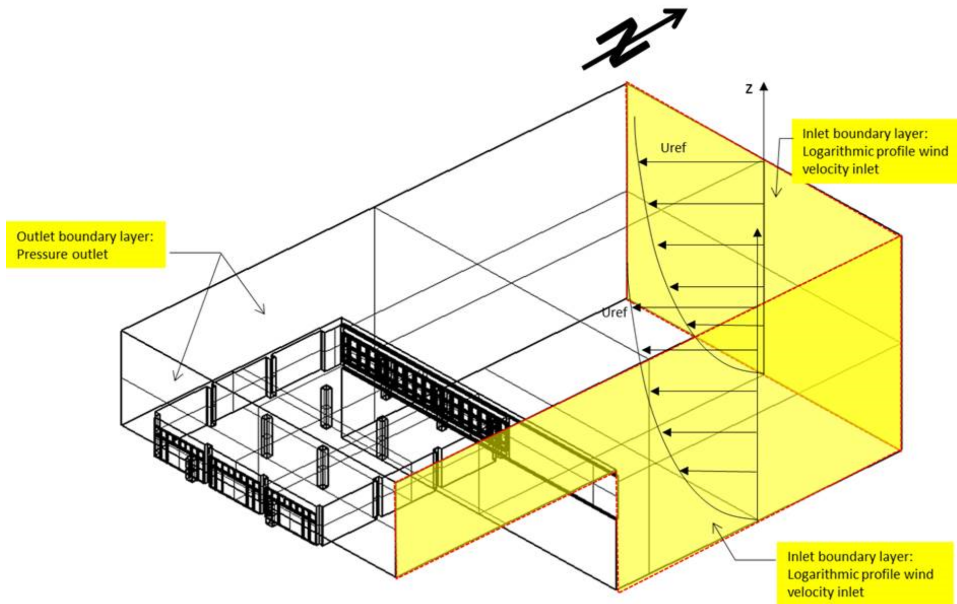


Figure 3.5.2: Inlet boundary condition for the North and East and Outlet boundary conditions for the West and South domain surfaces.

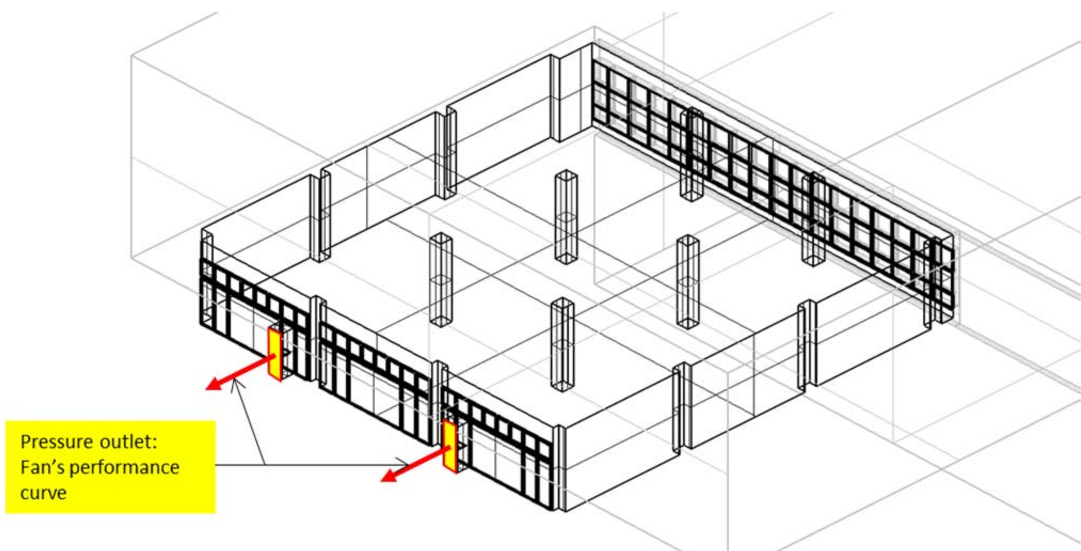


Figure 3.5.3: Pressure outlet boundary conditions for the exhaust fans located on the South lanai adjacent to Room 301.

Air Properties used in the CFD Simulations: Often CFD simulations are conducted with generic, "default" air properties at standard conditions, such as density and dynamic viscosity. For the present CFD simulation default air property values were used as well as air property values that reflect Hawaii conditions, including temperature and humidity.

Four test scenarios: Four test scenarios were investigated. The four test scenarios represented different combinations of windows open and closed on the North side of Room 301 and operating and non-operating exhaust fans on the South side. Figure 3.5.4 illustrates the definition of the four test scenarios.

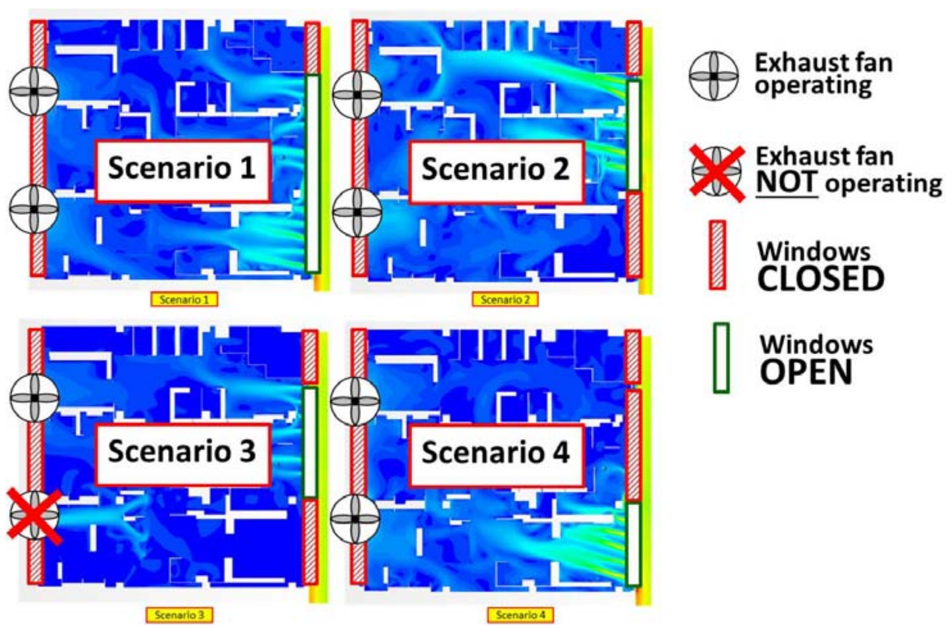


Figure 3.5.4: Definition of the four (4) test scenarios

3.6 Results

The first 10 of the 18 test cases (see Figure 3.5.1) , groups A through C in Figure 3.6.1 were used to benchmark the different parameters. The last eight test cases investigated the four test scenarios with two turbulence models. Test case groups D (D1 through D4) and E (E1 through E4) used Realizable k-ε turbulence model and the SST k-ω turbulence model, respectively.

Figure 3.6.2 and 3.6.3 compare the different convergence performances and the airflow patterns of the turbulence model of test scenarios D1 and E1. As can be seen the k-ε shows a more robust convergence performance while the SST k-ω turbulence model defines the air flow pattern in more detail. An interesting case when the CFD simulation could predict a complicated flow patterns is depicted in Figure 3.6.4 (Test scenarios 3). Here exhaust fan B was turned off. While the propeller fans were not moving and therefore not discharging air from Room 301, the opening through the fan housing was not sealed off. Therefore air could enter Room 301 from the South. This condition is clearly indicated by the scaled streamlines in Figure 3.6.4..

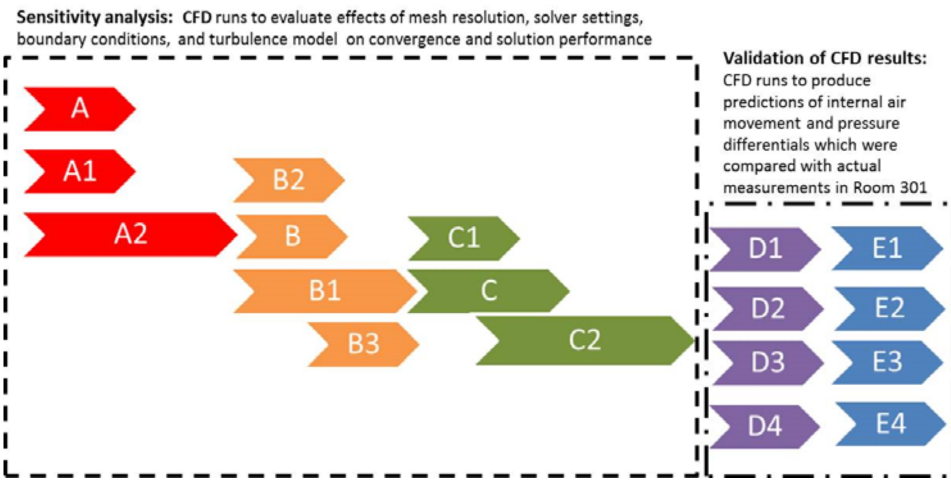
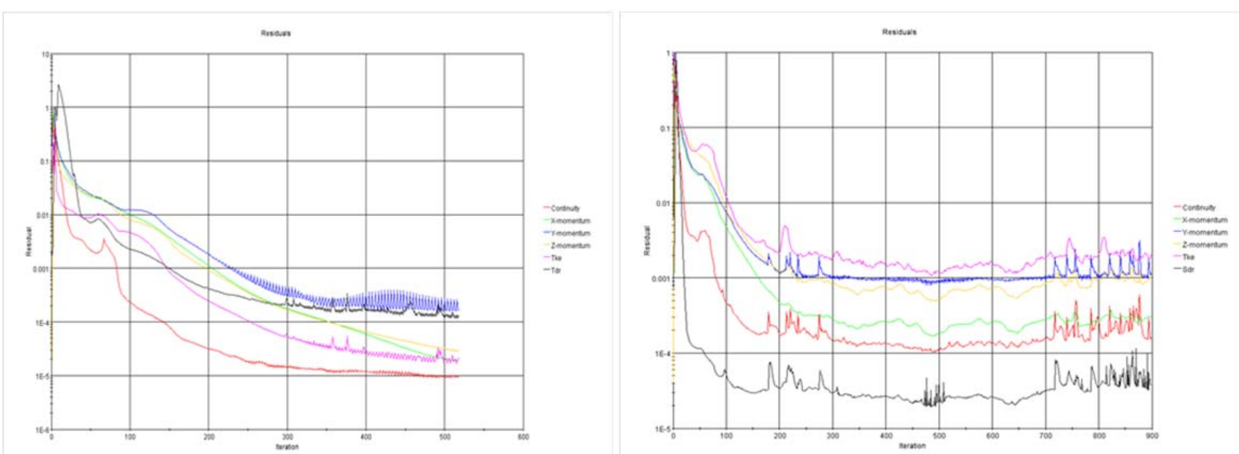
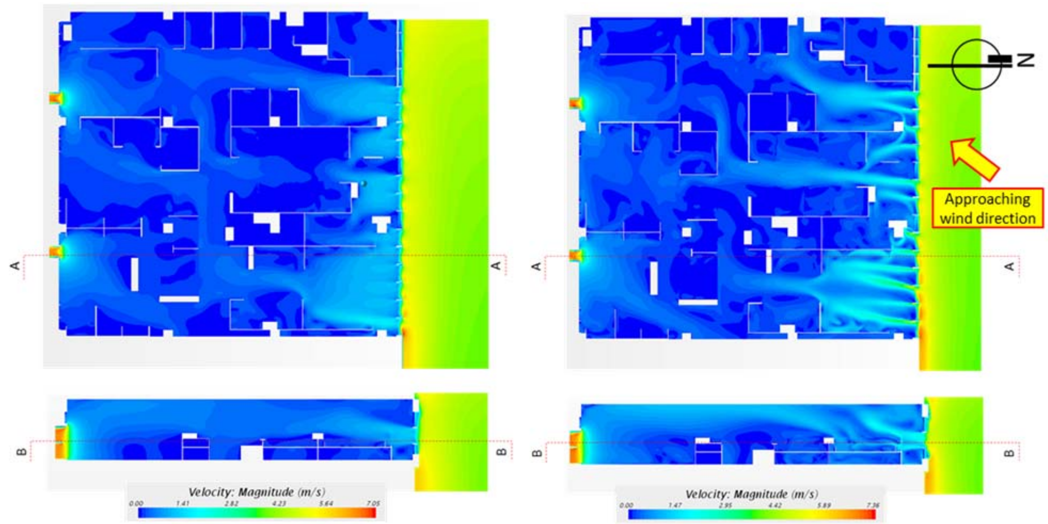


Figure 3.6.1: Structure of test cases under which CFD simulation runs were conducted



(a) Realizable k-ε turbulence model (b) SST k-ω turbulence model

Figure 3.6.2: Residuals plot of CFD solutions of two different turbulence models.



(a) Realizable k-ε turbulence model (b) SST k-ω turbulence model

Figure 3.6.3: An example of airflow pattern from simulation results of two different turbulence models from the same CFD scenario

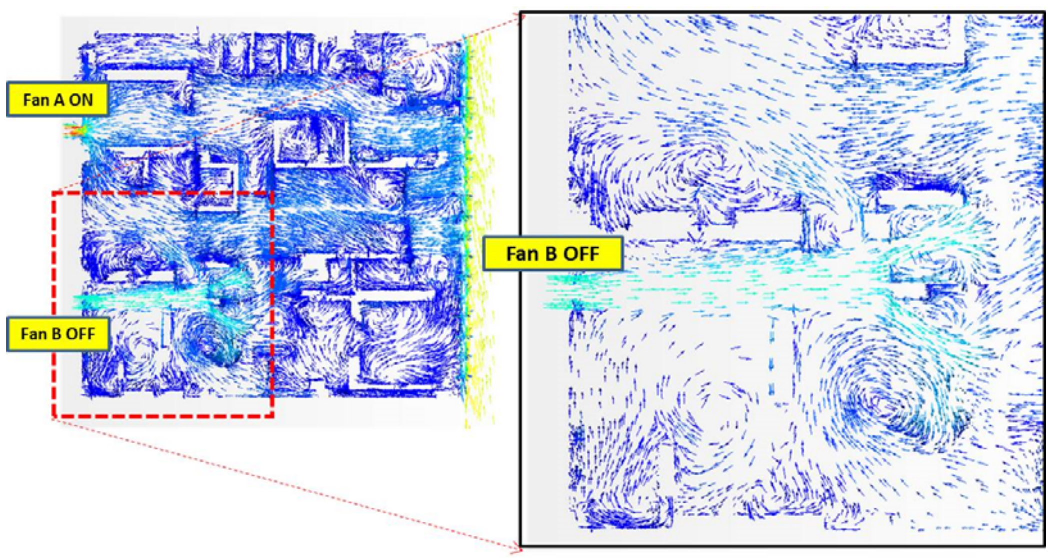


Figure 3.6.4: Airflow simulation from Scenario 3 (E3), under which the air flows into the interior through the non-operating exhausting fan B; the air flow vector map shows air flow direction and velocity on the section plane at 4 feet above the floor (a) and detailed view around the exhausting fan B (b).

3.7 Validation of the CFD Predictions with Test measurements at the Test Site

Room 301 was fitted with eight hot wire anemometers and five differential pressure transducers. The sensor locations and IDs of the instrumentation are depicted in Figure 3.7.1. The measurements obtained were compared with air velocity and pressure differential values that were obtained from the CFD for the corresponding locations.

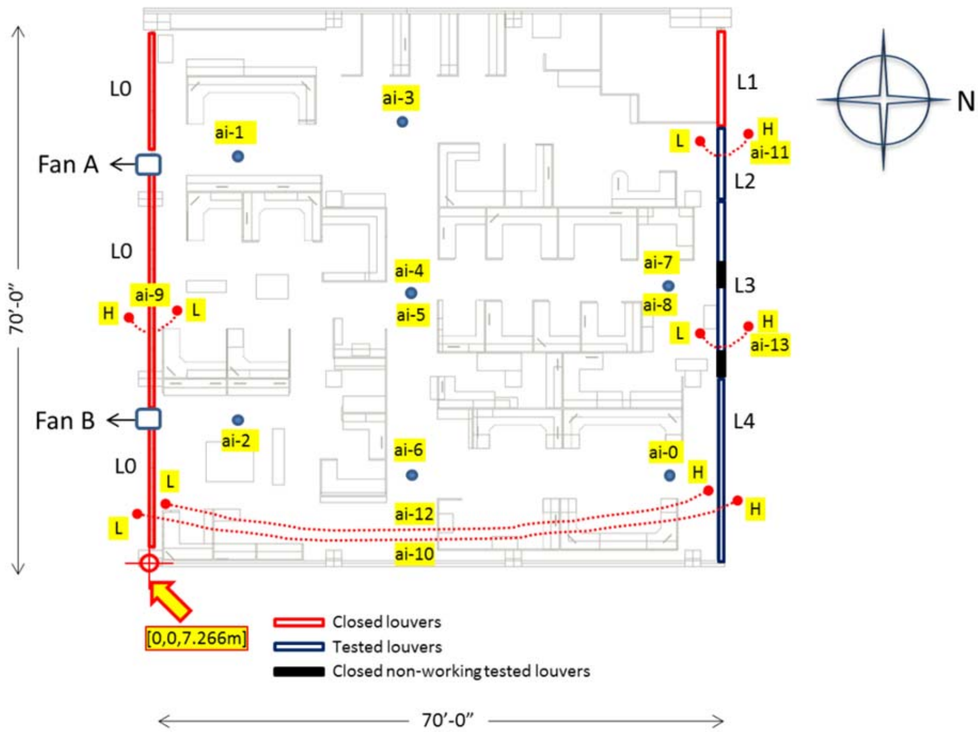
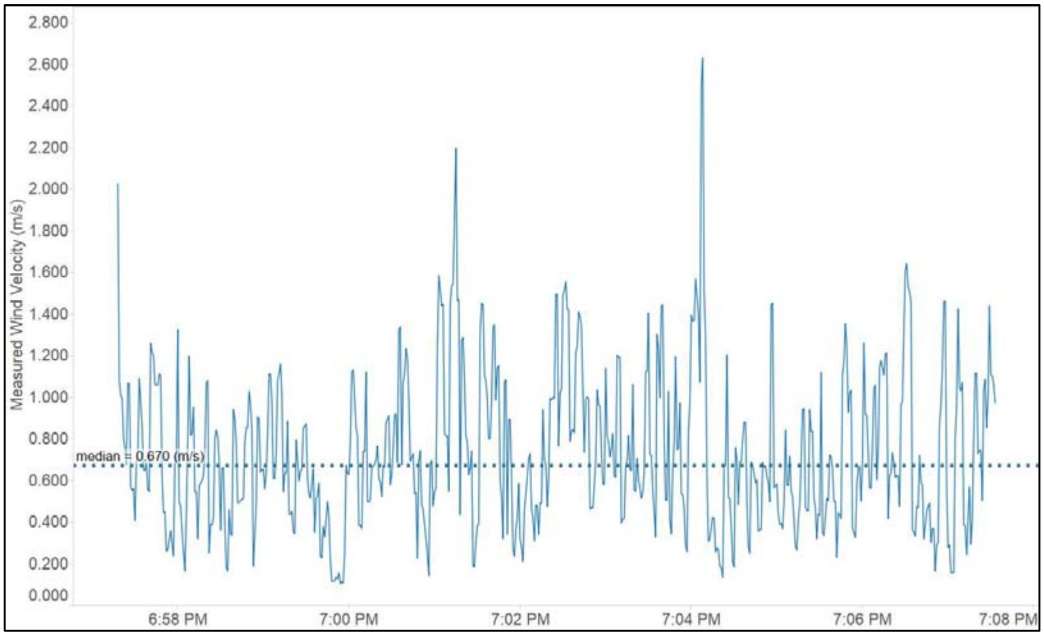


Figure 3.7.1: The sensor IDs and their indication of locations inside Room 301; these locations correspond with the locations of the CFD probes inside the computational domain

For validating the CFD results time averaged data of wind velocities and pressure differentials were derived from the test records. Typical data records for air velocity and differential pressure obtained in the measurements are depicted in Figure 3.7.2. The figures suggest that the air velocities and differential pressures showed significant variation around the average values. Figures 3.7.3 and 3.7.4 present examples of comparisons between measured and CFD predicted measured air velocities and differential pressures, respectively.

(a)



(b)

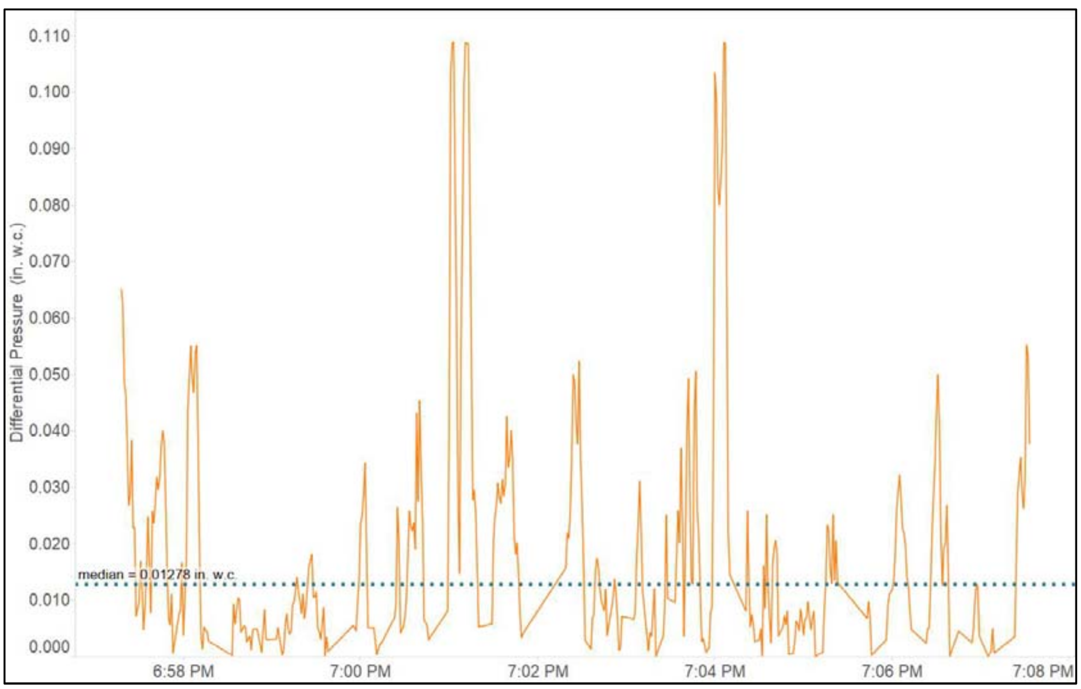
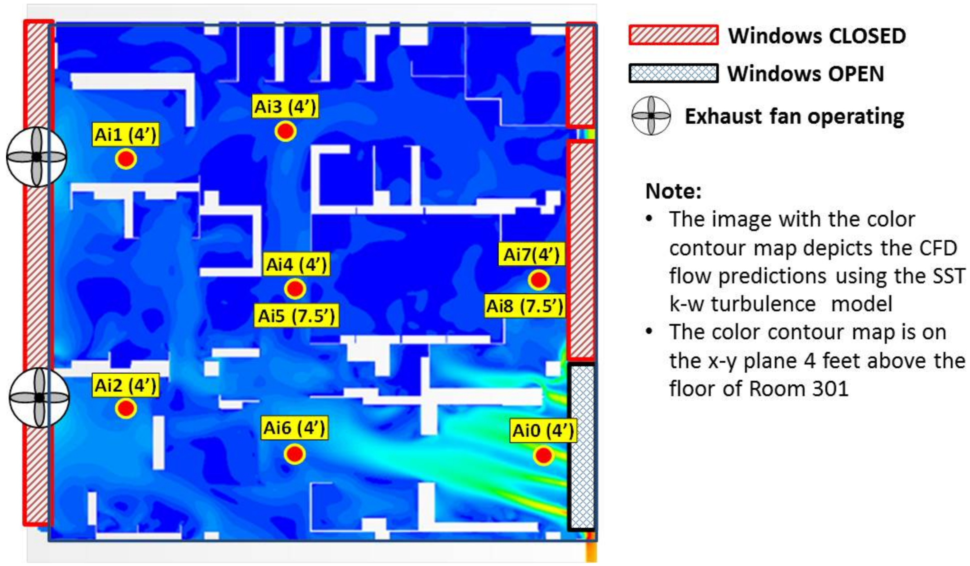


Figure 3.7.2: Typical data records obtained during tests in Room 301, (a) Data record and trend line for air velocity (b) Data record and trend line for pressure differential

Comparion of measured and predicted air velocities in Room 301–SCENARIO 4 (CFR runs E-4 and F-4)



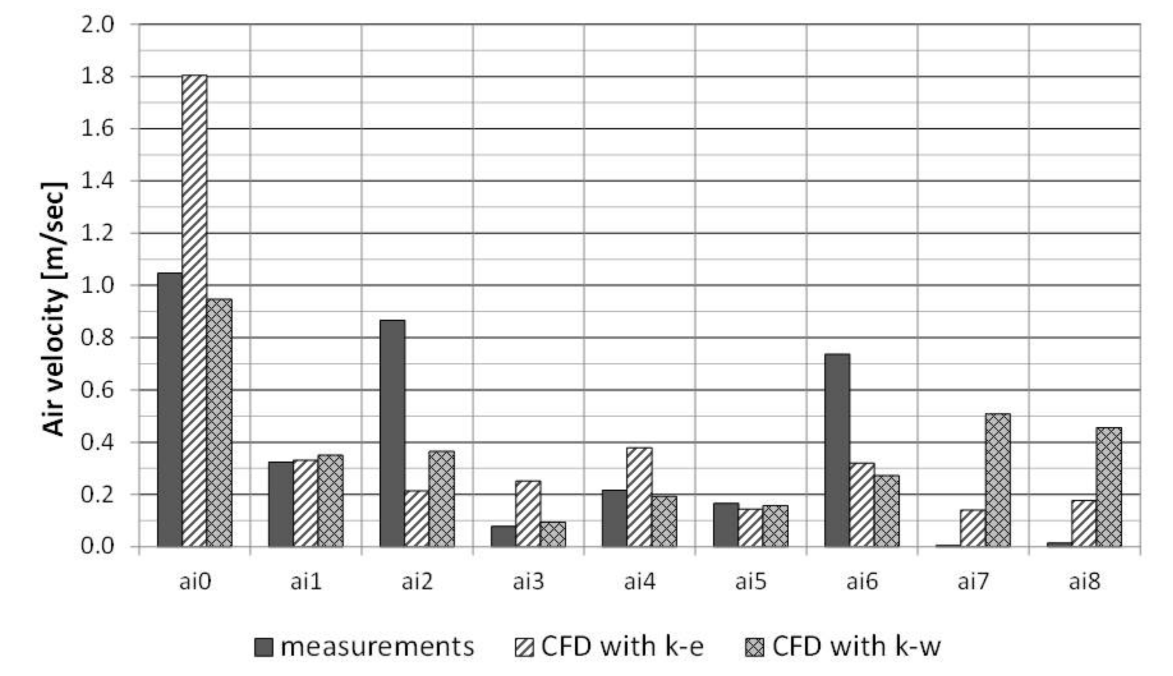


Figure 3.7.3: Comparison between measured air flow velocities and CFD predictions; **SCENARIO 4**

Comparion of measured and predicted pressure differences –SCENARIO 1 (CFR runs E-1 and F-1)

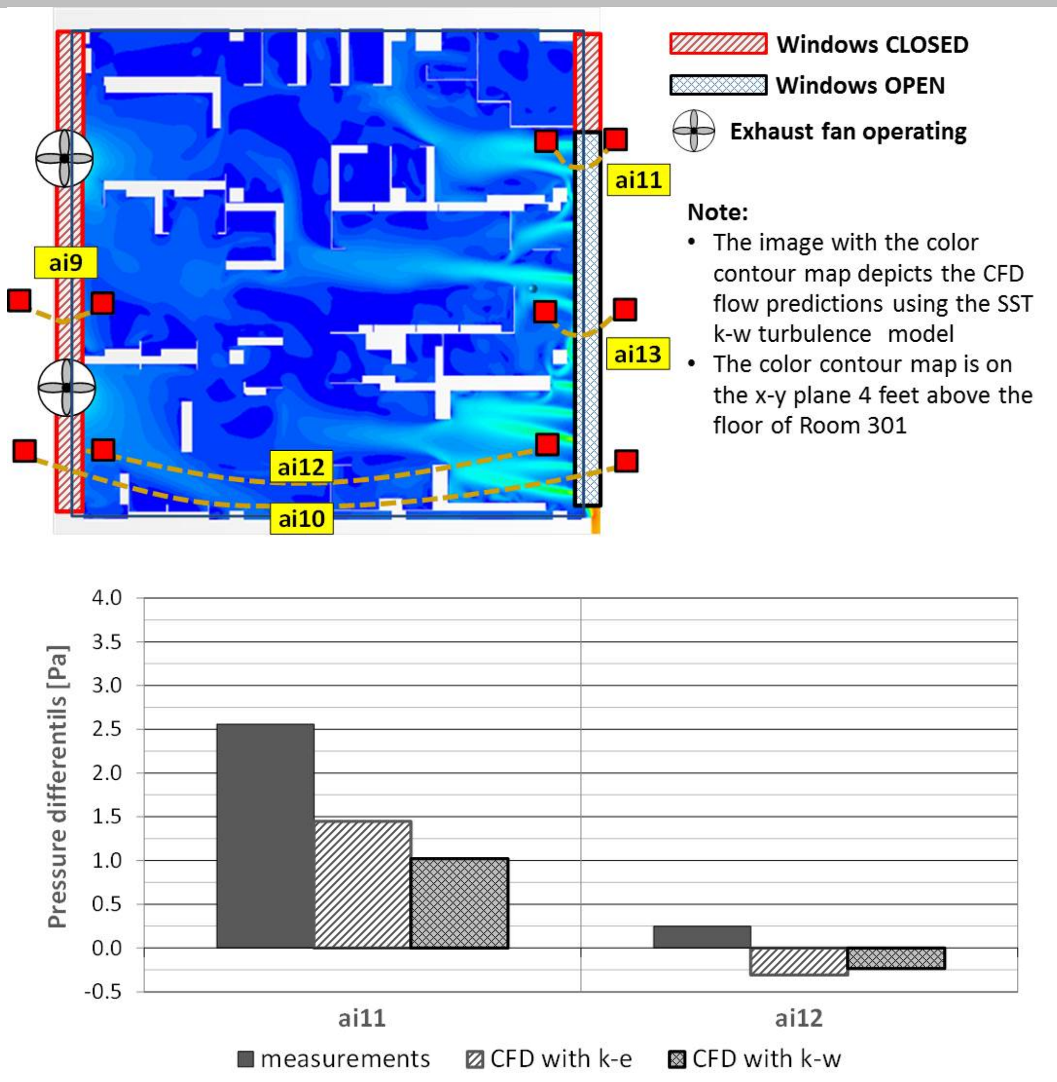


Figure 3.7.4: Comparison between measured pressure differences and CFD predictions; **SCENARIO 1**

3.8 General Observations and Conclusions

The results of the CFD study and validation of CFD results through actual measurements in Room 301 suggest the following general observations:

Room Conditions

- **Fans** - The presence of large exhaust fans in Room 301 provided the CFD research team with the opportunity to investigate complex internal air movement under inflow and outflow conditions that could be controlled by four large propeller type exhaust fans.
- **Internal Structures -** The geometry of internal structures of Room 301 was complex and the resulting effects of inlet and outlet conditions as well as numerous structural elements, cubicle walls and furniture on the internal flow patterns were perhaps too complicated for the type of benchmarking and parametric studies of the internal workflow development.
- **Irregular Flow -** The internal flow patterns showed significant interactions of flow separation, eddies and areas of small or no air movement in flow constrained areas.
- **Boundary Conditions -** The boundary conditions of Room 301 with respect to inflow and outflow conditions exhibited more uncertainties than was initially anticipated.

CFD Software Considerations

- **Physical property parameters** - The CFD work used virtual monitors and probes to detect and quantify relevant parameters at locations of interest.
- **Turbulence Model** -. The CFD team had applied two well-known turbulence models, the SST k- ω and the Realizable k-ε turbulence models, for the CFD investigations. Both turbulence models have advantages and disadvantages in describing internal air flow phenomena and related physical properties.
- **Velocity** - In general, air velocity was deemed to be a more reliant indicator of flow occurrences, and overall level of validation was more consistent when using air velocities than using differential pressures. Therefore, in order to measure actual flow performance and validate theoretical CFD results, air velocity should be the preferred validation parameter over differential pressure measurements.
- **Use of 3rd party Software -** The procedure used of creating the 3D geometry model with third party CAD software and importing the geometry model into the CFD software was effective.
- **Semi-coupled Domain -** the type of computational domain was a “semi-coupled” domain. By including a portion of the external building geometry in the domain, a “stagnation inlet” condition could be established instead of using constant mass flow inlet conditions used in the decoupled domain case. The “stagnation” approach allowed inclusion of momentum transfer in

addition to pressure gradient as the inlet conditions, thereby increasing the reliability and robustness of the free flow air inlet condition for the internal CFD analysis of Room 301.

- **Mesh Size -** The size of the mesh was found to be significant if all small detail flow obstructions were modeled with high resolutions. The CFD team had created large meshes of up to 30 million cells. The CFD team chose medium size meshes as preferable and according to benchmark simulations conducted for this CFD project, medium size meshes are not necessarily inferior to very large meshes.
- **Color Contour Maps -** The post processing procedure of using color contour maps was found effective and provided the CFD team with the ability to observe the entire flow field and to obtain qualitative estimates of the overall air flow pattern.
- **Customized Probe -** Additional post processing procedure included extracting specific values of air velocities and differential pressures at locations of interest within the computational domain.

Equipment and Environmental Factors

- The fan curves of the propeller type exhaust fans were not available from the vendor (e.g. the CFD team had to use a locally available residential whole house fan with a simple plane propellers). The CFD team assumed a generic propeller type fan curve as the input variable for the CFD outlet condition. The CFD team considered that this approximation was reasonably accurate.
- The density and dynamic viscosity of air under typical Hawaii temperature and humidity conditions varies from standard air conditions used as default setting in the CFD software to as much as 3%. This variation is considered significant enough to warrant specific Hawaii climate values for air in the CFD calculations.

Overall Results of Learning

- The CFD software used of the investigation performed well and the developed internal CFD workflow used a wide range of standard and customized features of the CFD software.
- The degree of data validation consistency between the CFD simulation results and actual measurements was generally limited and ranged from good to low. There were numerous likely causes that could have contributed to the deviation of theoretical results and actually measured.

Summarizing Section 3.7: The CFD investigation of the air flow patterns inside Room 301 revealed very complex air movements. The lessons learned from the CFD investigation provided a good basis to develop a workflow that was going to be used for internal CFD investigation at Keller Hall.

SECTION 4 - DEVELOPMENT OF DATA ACQUISITION PROCEDURES

The work for this project phase was conducted parallel to the development of a CFD work process, which is summarized in Section 3. The instrumentation and procedures developed in this phase were used to validate CFD simulation result.

4.1 Overall Approach

<u>The main objective of the project work presented in this report was as follows:</u>

> Gain proficiency in the research team in developing and executing a test procedure to select, test and deploy instrumentation for air velocity and differential pressure inside building spaces to obtain data that can be used for validation of internal CFD simulations.

<u>The following work task objectives were required to</u>

- Select instrumentation for internal air velocity and differential pressure measurements which were suitable for airflow patterns and differential pressure conditions inside a space.
- Determine the site-specific conditions for instrumentation deployment and actual filed measurements inside the test space
- Perform initial CFD scoping simulation to determine preferred locations inside test space where instruments should be deployed, which were best suited for validating results of internal CFD simulations inside the test space.
- Develop and test a data acquisition and analysis procedures for the data air velocity and differential pressure measurements.
- Test the instrumentation and analysis and reduction procedures in the lab of the team before deployment in the test space.
- Custom build and Install the exhaust fan units in the test space.
- Coordinate with the test space occupants to perform the tests at times that were outside regular working hours.
- Install the instrumentation and data acquisition system during three days of data acquisition at the test site.
- Perform the final data analysis and provide the results to the CFD team for validation of the final CFD simulations of air flow through the test space.

Figure 4.1.1 shows the systematic of work efforts carried out under this project phase.

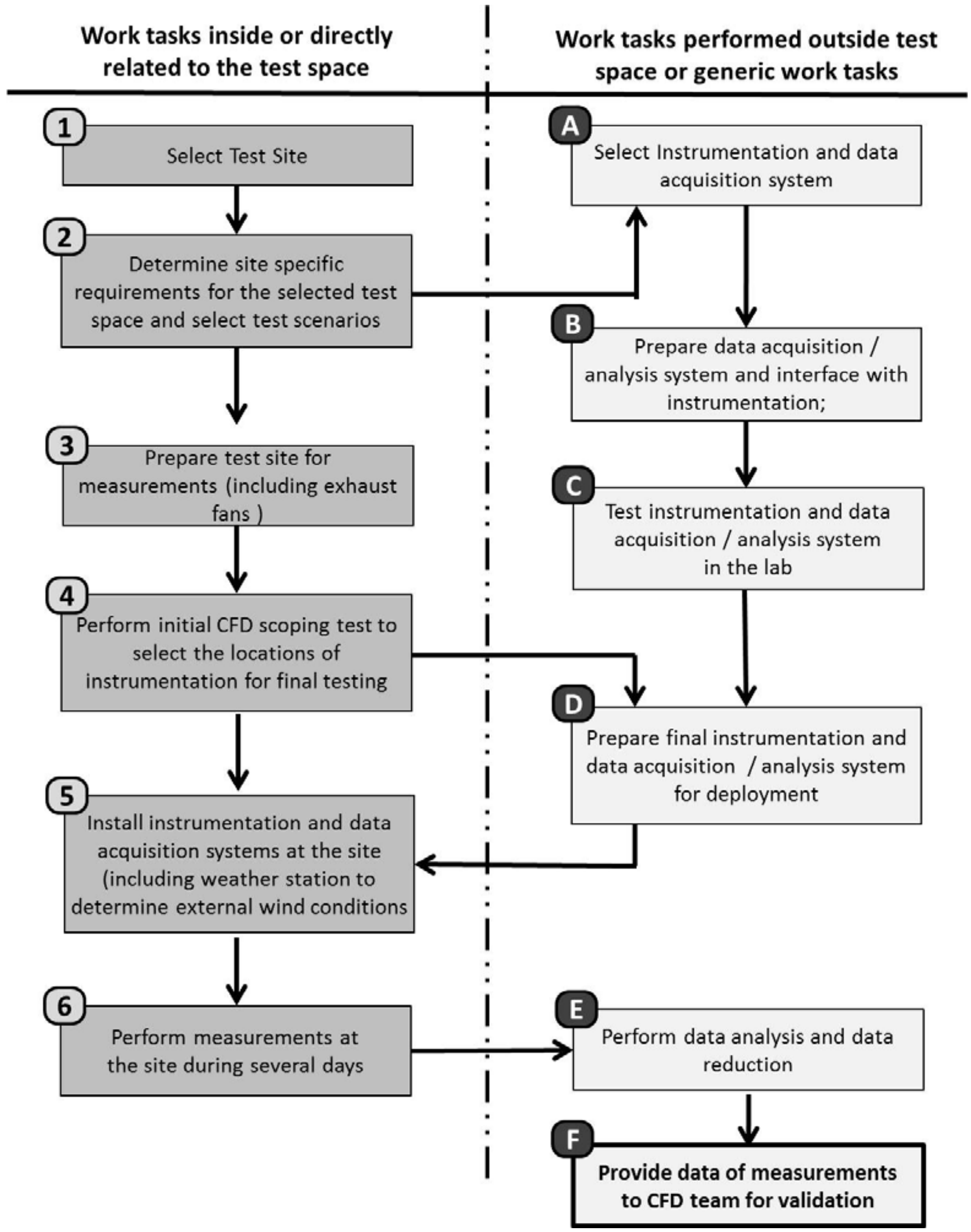


Figure 4.1.1: Diagram describing the interrelated work tasks performed under this project phase

4.2 Selected Instrumentation and Data Acquisition

Three types of measurement devices for air velocity and differential pressures were used in this study:

Weather station (temporarily installed during several days of measurements): Wind velocity and wind direction were measured and logged with an Onset HOBO U30 weather station installed on the west side of the roof of Sinclair, directly above Room 301 (see Figure 4. 2.1). The wind velocity and direction indicator sensors were 9.5 feet above the roof and 50 feet above the ground. Recorded data resolution was set at 5 Hz.

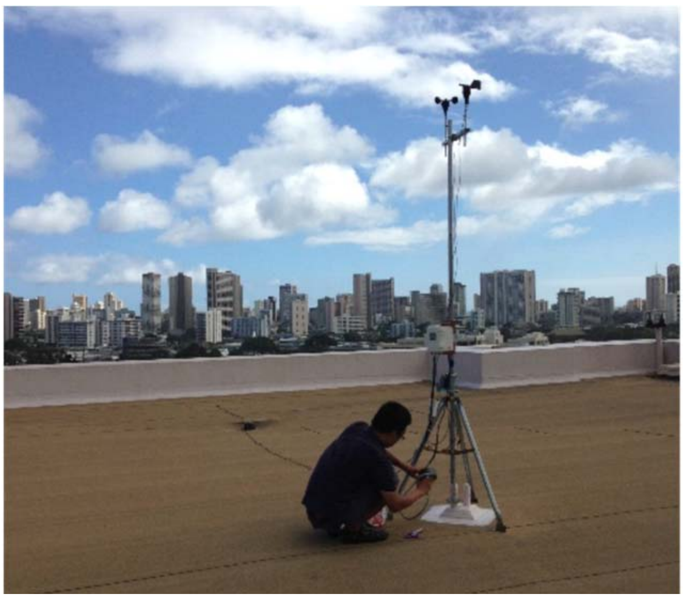


Figure 4.2.1:
The weather station (Onset Hobo U30 data logger) with wind velocity and direction sensors was installed on roof of Sinclair library during the duration of the measurements

Differential pressure transducers: Differential pressures were measured across selected points inside Room 301. Two types of differential pressure transducers were used. Four Setra differential pressure transducers, Model 264, were used, which had a range of 0 to 0.1 inches of water column (zero to 25 Pascal) and 0.5% accuracy of full scale. One Halstrup Walcher differential pressure transducer Model P26 was used, with a range of 0 - 0.055 inches of water column (0 - 10 Pascal) and 0.5% accuracy of full scale. Figure 4.2.2 shows the two type of differential pressure transducers used in the study.

Both types of differential pressure transducers were connected to pairs of flexible vinyl pressure tubing ($^{3}/_{16}$" internal diameter for the four Setra transduces and ¼" for the Halstrup Walcher). Each transducer had two pressure tabs, one high and one low pressure tab, which were connected to the pressure tubing. The two pressure tubing terminals connected the transducers to two points of pressure measurements, thereby detecting the pressure difference between these points. The pressure tubing extended to pressure tubing terminals, which were PVC pipe sections with holes drilled in and brass barbed fitting to connect the tubing. Tubing was color-coded to track which tubing was to be connected to the high or low side of the pressure transducer. Figure 42.3 shows one instrument unit to measure differential pressures, including transducer, pressure tubing and pressure tubing terminals.

(a) Setra differential pressure transducer Model 264

(b) Halstrup Walcher differential pressure transducer Model P26

Figure 4.2.2: Two types of differential pressure transducers used in the study

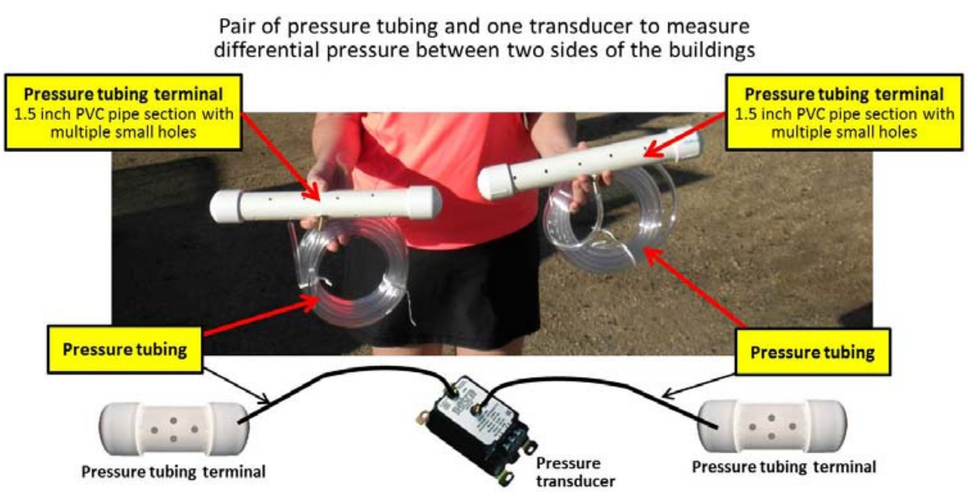


Figure 4.2.3 One unit to measure differential pressures, including differential pressure transducer, pressure tubing and pressure tubing terminals

Anemometers: Air velocity was measured with 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 velocity and an accuracy of 0.5 % of reading or 1% of full scale. Anemometers were mounted on stands at a height of 4 feet and 7.5 feet using a wire wrapped around the stand and holding the tip of the anemometer 3 inches away from the stand (see Figure 4.2.4).

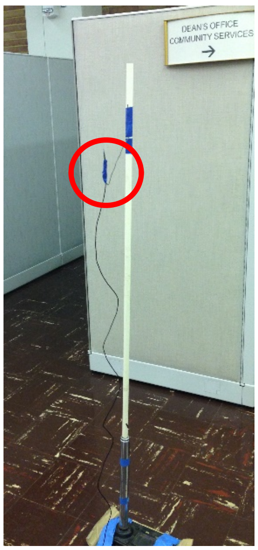


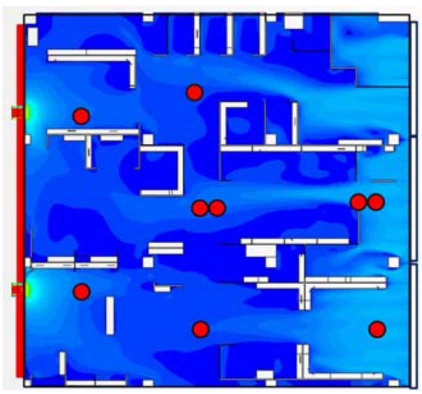
Figure 4.13:
Anemometer installed on stand with tip held 3" from stand at height of 4' above the floor

Data acquisition was done with a National Instruments USB-6341 device X series DAQ (referred to as the DAQ) and a laptop computer equipped with National Instruments Signal Express software and set to log at a 5-Hz resolution. The anemometers were wire-connected to the DAQ using standard AC extension cords. Although these cords are bulky and heavy, they are easy to piece together to extend for a large site when needed. Pressure transducers were wired to the DAQ using 24-gauge stranded 2-conductor audio cable (these had been previously wired for another study, so extension cords were not used).

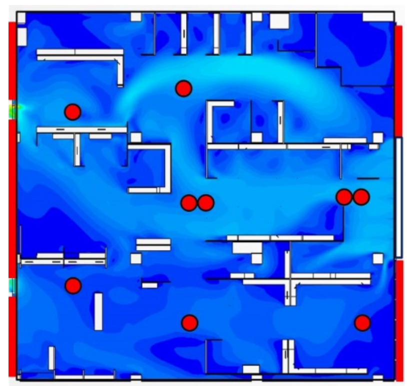
Additional instrumentation: Temperature (dry bulb and globe temperature) and relative humidity were logged using Onset Hobo ZW wireless sensors, a Hobo ZW receiver, and a laptop computer with Hoboware software. Data resolution was 1-minute (the maximum for that system). The globe thermometers were made with an Onset air/water/soil temperature probe with a standard table tennis (Ping-Pong) ball glued onto it using 5-minute 2-part epoxy, and then spray-painted with a black matt paint (instructions from George Loisos of Loisos + Ubbelohde Architects, personal communication, 2011).

4.3 Initial CFD Scoping Tests to Determine Sensor Location

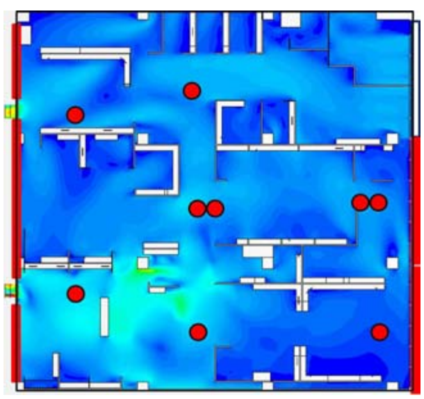
In order to determine locations at which sensors would be deployed, initial CFD scoping tests were carried out. These initial scoping tests used coarse grids. Figure 4.3.1 shows the scoping CFD runs carried out in Room 301 (see section 3) and the selected placement of the anemometers for the validation tests. The objective was to evaluate an approximate distribution of air flow patterns inside at the test site, to provide a basis for locations of air velocity to be determined.



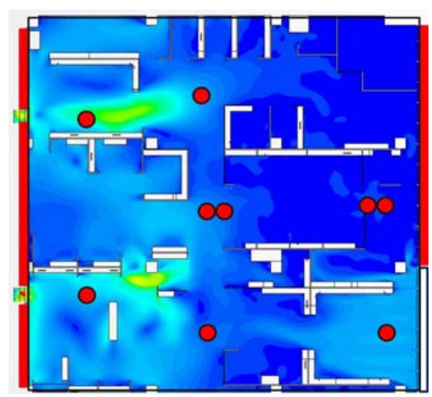
Scenario A used in the initial CFD scoping tests;
All widows on the North wall are open



Scenario B used in the initial CFD scoping tests;
Only section of widows on the North wall are open



Scenario C used in the initial CFD scoping tests;
Only section of widows on the North wall are open



Scenario D used in the initial CFD scoping tests;
Only section of widows on the North wall are open

Louver windows closed at all times during test

Louver windows that were adjusted (either open or closed) during the test

2 anemometers at same location, installed at different heights on the same stand

1 anemometer installed at location

Figure 4.3.1: Four scenarios A through D of air flow though Room 301 calculated with initial CFD scoping tests; Note: the locations of anemometers in Room 301 was kept the same for all four scenarios

4.4 Running Tests and Data Recording

The Signal Express software was set up ahead of time for these particular sensors and the project was saved in the software. For each of the four test scenarios (see Section 3), data was collected for calibration first. The average reading for this calibration period was used to adjust the measurements taken during the scenarios. The orientations of the anemometer blades were adjusted to be perpendicular to the assumed main direction of the air movement, which was determined by the scoping test (Section 4.3).

The data recording on the Signal Express software was started and stopped for each scenario to store data in separate files. Data was exported from Signal Express in a TDMS format. Data was processed with Python scripts to convert to CSV format, add timestamps, calibrate and scale values, and average the 5-Hz data to 1-second resolution. Post-processing allowed ample time to carefully review calibration data for outlier data points before calculating the average for calibration. Occasionally, an outlier data point was detected and removed. One instance of an outlier was attributed to a technician stepping on tubing connected to a pressure transducer. Processed data was uploaded to a PostgreSQL database on a server.

4.5 Analyzing the Weather Station Data

The process of analyzing the weather station data was carried out in conjunction with the four test scenarios presented in Section 3. Weather station data was filtered for wind velocity ≥ 0.05 m/s and direction 30-60 degrees. The median values for each scenario were used as the reference wind conditions for the CFD simulation. Figures 4.5.1 and 4.5.2 illustrate weather data after filtering as a wind rose and as a time series.

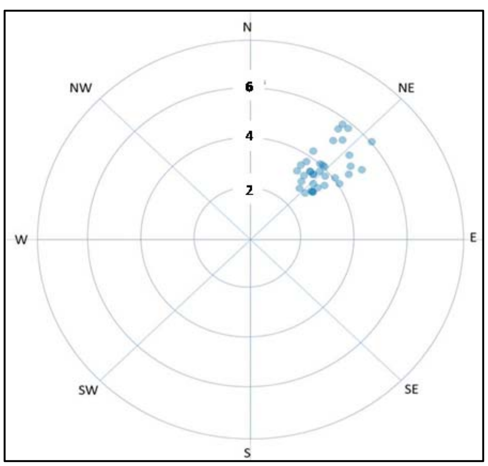


Figure 4.5.1: Wind rose showing filtered wind velocity

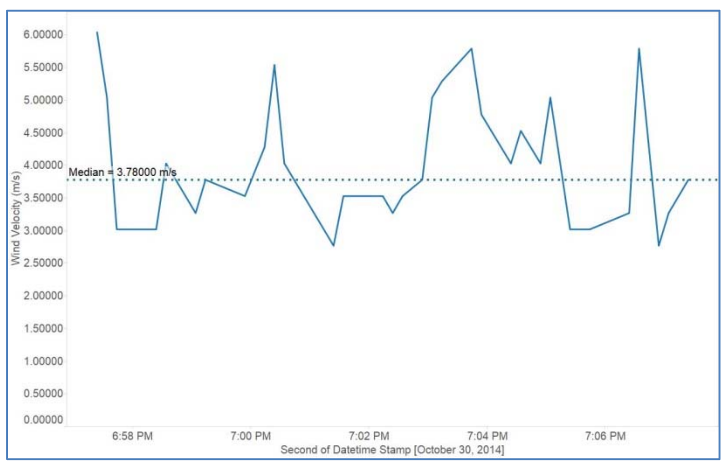


Figure 4.5.2: Time series Trend line for wind velocity after filtering velocity to > 0.05 m/s and wind direction to 30° – 60°

4.6 Analysis of Air Velocity and Differential Pressure Data

Median air velocities and differential pressures measured inside Room 301 were calculated from anemometer and differential pressure transducers data obtained during all four scenarios (see Section 3). Data was filtered to remove negative values, and filtered to keep only data from timestamps that correspond to filtered weather station data (wind velocity >0.05 m/s and direction 30-60 degrees). As an example, Figure 4.6.1 and 4.6.2 show a typical time record and a trend line for air velocity and differential pressure data, respectively.

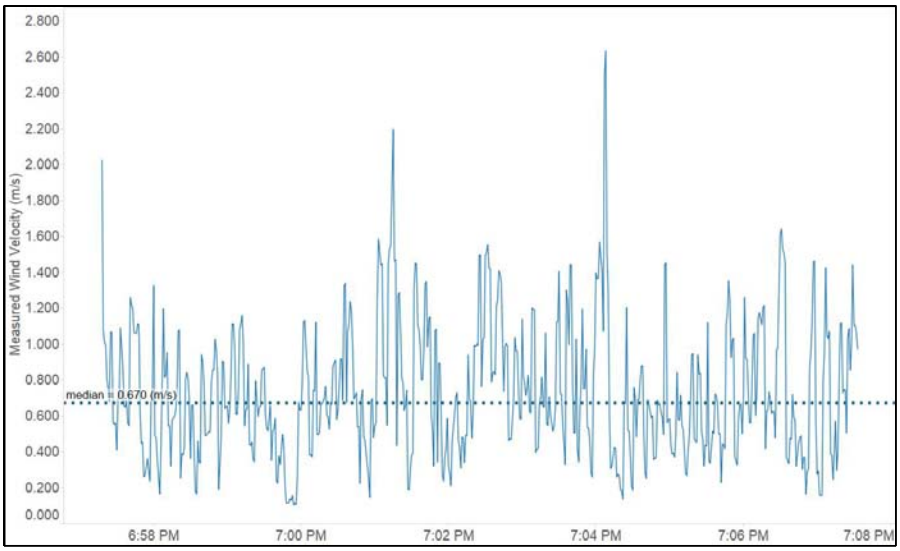


Figure 4.6.1: Typical time record and trend line (mean air velocity) for air velocity measurement inside Room 301.

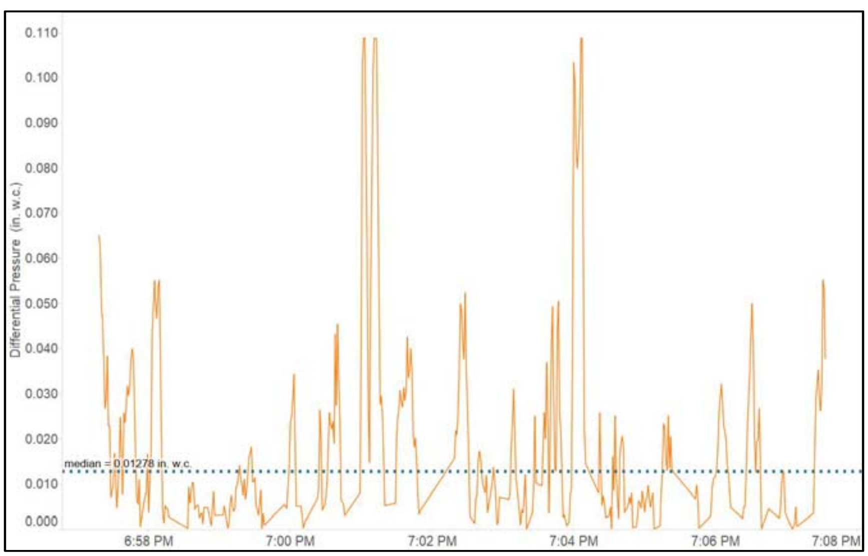


Figure 4.6.2: Typical time record and trend line (mean air velocity) for differential pressure measurement inside Room 301.

4.7 Conclusions and Lessons Learned

This section summarizes main conclusions and lessons learned from the project work of this study. These conclusions and lessons learned apply to test sites which have similar conditions as the current test site on the University of Hawaii Manoa campus.

Prepare the test site and inform occupants

- Plan trials for when the space is unoccupied and notify the occupants.
- Print out data sheet with diagram of sensor locations, and matrix of scenarios and put on clipboard. Do this for every trial and label with the date.
- Ensure tools are on hand at the test site: screw drivers, voltage meter, tape, and marker.
- At the test site, be sure there are no fans running.
- Advise test site occupants of the plan to start the trial.

Prepare instrumentation in the lab before start measurements at the test site

- Prepare all wiring, stands, tubing, and terminal ends for sensors. Number and color-code parts as needed.
- Have all instruction booklets for sensors and DAQ device on hand for trouble-shooting purposes.
- Prepare laptop computer:
  - Setup the Signal Express “project” ahead of time, configured to record all voltage and current inputs, and save with date of trial.
  - Have fully charged laptop, power cord, and optical mouse in a computer bag.

Deploy the instrumentations and data acquisition

- After connecting sensors to the DAQ device, check that the signals look as expected, if not:
  - Make sure sensors are checked in the Signal Express software to display the reading.
  - Check the sensor is getting power (use a multimeter).
    - Make sure the correct voltage from power supply is sent to sensor – check both the DAQ and the sensor sides.
    - Check the sensor is not damaged.
    - Re-tighten all wire connections.
- Always do a calibration run first.
- During data collection:
  - Don’t step on flexible tubing for pressure sensors
    - Don’t rush past sensors, creating air movement

- Record the start time for the scenario on the data sheet. Don't rely on memory for which trial is which in the event that you repeat a scenario and keep both sets of data. In lieu of that, assign proper and descriptive file names with the scenario number.
- When finished, reset the weather station data resolution back to its original setting.
- Photograph data sheets with a cell phone and email it as a backup.
- In the database, use the same naming conventions for columns of data
- For weather station data, be sure to export the data in consistent units from the Hoboware software. (e.g. m/s versus of mph).
- In future, tag the data with the trial and scenario number during the post-processing stage, using the Python script .

Summarizing Section 4: The test measurements in Room 301, which were carried out in conjunction with the CFD work flow development (Section 3), provided the team with important opportunity to fine tune the measurement procedures before the final internal CFD investigations were conducted.

SECTION 5 – CONDUCTING INTERNAL CFD INVESTIGATION AND VALIDATION

The main objectives of this project phase were;

- Develop applications skills at ERDL to perform CFD simulations of internal air movement and have an opportunity to validate theoretical CFD predictions with actually measured air flow phenomena in a naturally ventilated building.
- Determine a scope and approach for internal CFD simulation and validation that could be used to develop design tools for future internal air movement studies.
- Develop and test a methodology where the air movement in internal spaces can be predicted on a probability scale so that CFD simulations can be used as a design and optimization tool for naturally ventilated spaces to achieve high energy efficiency.

5.1 Methodology

The following methodologies were applied:

Selecting the Test Site:

- Natural ventilated spaces: The selected internal spaces were originally built as naturally ventilated space with cross ventilation, taking advantage of the predominant wind approach of trade winds.
- Selecting large rooms and not small offices: Classrooms were preferred as the test site over offices, since they contain a small amount of furniture and do not have any internal partitions.
- Ready access of the building and internal spaces: The investigation had to be carried out during times when the classrooms and the interconnecting causeways were not occupied.
- Controllable air flow through the interconnected spaces: The interconnected spaces had controllable internal openings to control the internal air flow pathways.
- Large intake and exhaust openings: Internal spaces that offer relatively large intake and exhaust opening in the wind ward and leeward side of the interconnected spaces (e.g. this could maximize driving forces and minimize air flow losses).

The CFD analysis:

- Coupled domain selected: A coupled domain approach was selected for the CFD investigation. The entire building geometry of Keller was modeled in full scale, with selected internal spaces being connected to the external air volume.

- Louvered intake and outlet openings: The intake and outlet of the selected classrooms were louvered windows without insect screens. For the tests the louvers were brought into a horizontal position where they offered a minimum air flow resistance.

- Outside appurtenances of building envelope: The 3D-CAD geometry included all main appearances on the building envelope, such as larger vertical fins on the North and South side of the building.

- Size of the domain grid (Meshing): As in previous investigation benchmarking with different mesh sizes were conducted to assess the convergence and computational performance of the simulations. Grid sensitivity analysis was part of this investigation and shows quality grids.

- Post processing: For the CFD analysis two types of post processing methods were used. As used extensively in previous phases of the CFD research color counter maps and streamlines illustrations were used to obtain qualitative descriptions of the air flow patterns. In addition, values of air velocities and mass flow rates were obtained at specific locations.

- Control volume for quantitative analysis: The three classrooms were interconnected by an internal hall way. The hallway was defined as a control volume with the internal openings to the class rooms and hallway cross sections serving as intake and exit for mass flows of the control volume, e.g. the control surfaces. For the control surfaces actual velocity and mass flow rates were determined.

- Defining test scenarios: The CFD investigation was carried out for five test scenarios. The test scenarios differed in their combination of open or closed doorways and internal louvers between the classrooms and the hall way as well as open external doors of the hall ways to the building exterior. The test scenarios were selected to allow CFD simulations which could be readily validated with the internal air flow measurements.

- CFD analysis to obtain percentage of exceedance: The CFD analysis obtained the number of air changes per hour (ACH) of the class rooms. The air change rates were correlated against external wind condition. The result of this data correlation was a probability of exceedance prediction of ACH for a space versus the approach wind velocity.

Measurements in the Field:

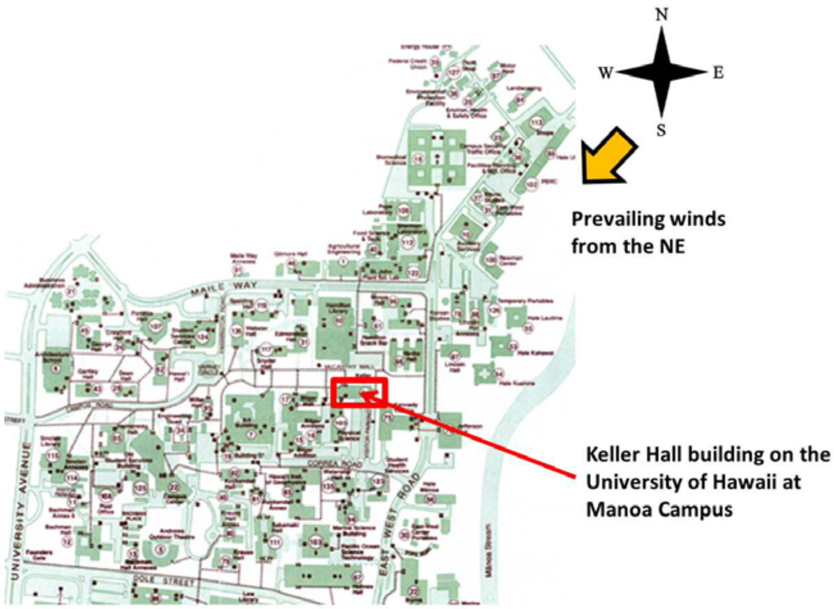
- Parameter used for testing: The measurement of air velocity was selected as the parameter for validating the CFD simulation results. Pressure differentials were not used to validate CFD simulations for internal air flow.
- Instruments used: For the measurements several hotwire anemometers were used. These instruments had already been used in previous field measurements.
- Selecting locations to measure: For the location of the air velocity measurements internal opening and internal air passageways with known cross sections were selected so that the mass flow rate could be readily verified.
- Placement of instrumentation: The anemometers were installed in the center of the respective opening.
- Data acquisition: Data acquisition was carried out with a data multiplexer in order to get good time dependent internal air velocity data for the deployed six anemometers.
- External wind conditions: The weather station which recorded the wind approach direction and wind speed on the roof of Keller Hall was synched with the internal air velocity measurements, in order to allow cross reference of internal air movement conditions to external wind conditions.

Validating the CFD Results:

- Validation methodology: Validating CFD simulation results with measured air velocities was carried by comparing the air velocity calculated by CFD simulation for the opening with the mean measured air velocity in the opening taken over the time series of the measurements for the different scenarios.
- Data filters: The measured wind velocity records for the different test scenarios were filtered in regard to wind approach direction so that only those internal air velocity measurements were considered, which corresponded with the approach wind direction used in the CFD simulations.

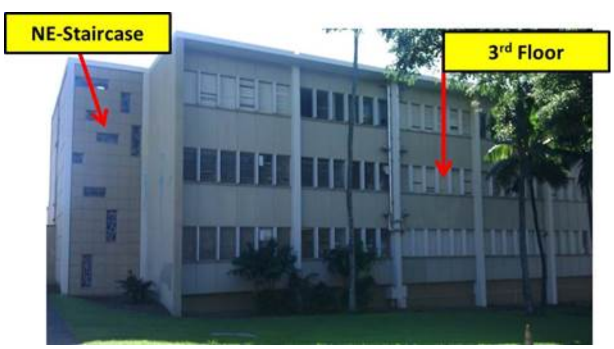
5.2 Description of the Test Site

The test site was the Keller Hall building, which is located on the University of Hawaii Manoa Campus. Figure 5.2.1 shows the location of Keller Hall building on the Manoa campus. The prevailing winds, e.g. trade winds, approach the building from the North-East. Figure 5.2.1 also shows the building from different viewpoints outside the building.

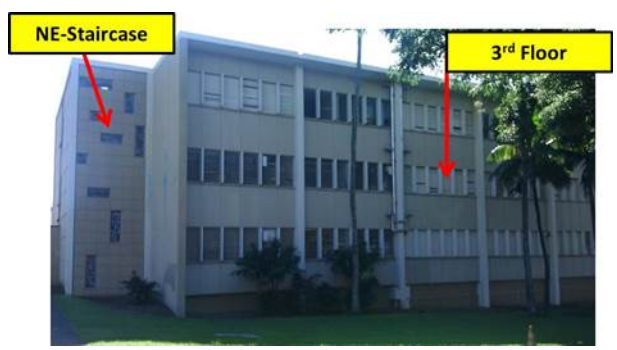


Location of test site on the University of Hawaii Manoa campus

Prevailing winds: The prevailing wind is coming from the North-East



Keller Hall building – View from the North-East



Keller Hall building – View from the North-East

Figure 5.2.1: Vicinity map and images of Keller Hall

The test site for the internal air flow investigation consisted of three interconnected classrooms and one internal hallway on the third floor of the Keller Hall building. Figure 5.2.2 shows the floor plan of the third floor and indicates the location of the three classrooms (e. g. the test site) on the third floor. Figure 5.2.2 indicates that the classroom 302 is located on the North side of the building and classrooms 313 and 314 are located on the South side. All three class rooms and the internal hallway were naturally ventilated. Figure 5.2.3 illustrates the layout of the classrooms and the interconnecting hallway. Figure 5.2.3 also shows the external and internal openings of class rooms and hallway.

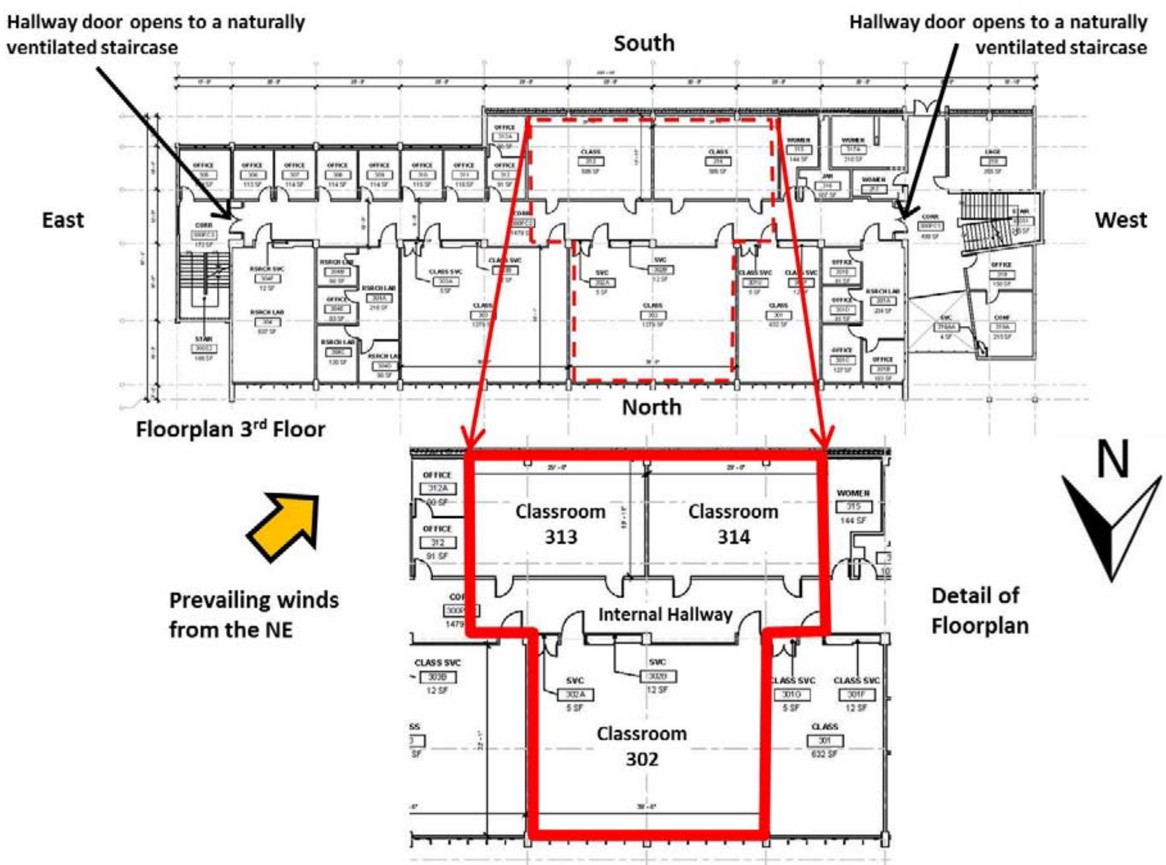
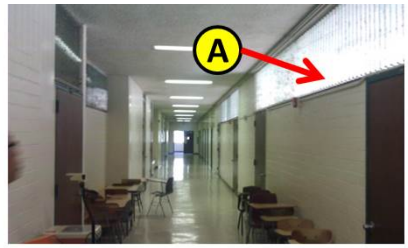


Figure 5.2.2: Floor plan of the 3rd floor of Keller Hall building and indication of the three classroom and interconnecting hallway which represent the test site

Typical classroom outfitting



Typical hallway configuration

Internal louvered opening between hallway and classroom, seen from the classroom

Internal louvered opening between hallway and classroom, seen from the hallway

A Internal louvered openings between hallway and classrooms.

Internal louvered opening between hallway and classroom, seen from the classroom

External louvered window in classroom 302, facing North

Figure 5.2.3: Internal configuration of the classrooms and hall way, which were the test site

5.3 Defining the Test Scenarios

Five test scenarios were selected for which CFD simulations and validation measurements were carried out. The test scenarios were defined by different combinations of open and closed doorways and open or closed internal louvered connections between the classrooms and the hallway. For the selection of test scenarios the internal doorways and louvered openings where identified with test IDs. The test IDs of the doorways and louvered openings are described in Figure 5.3.1. Tables 5.3.1 and 5.3.2 indicate what combination of open and closed doorways and louvered openings were used in the five different scenarios.

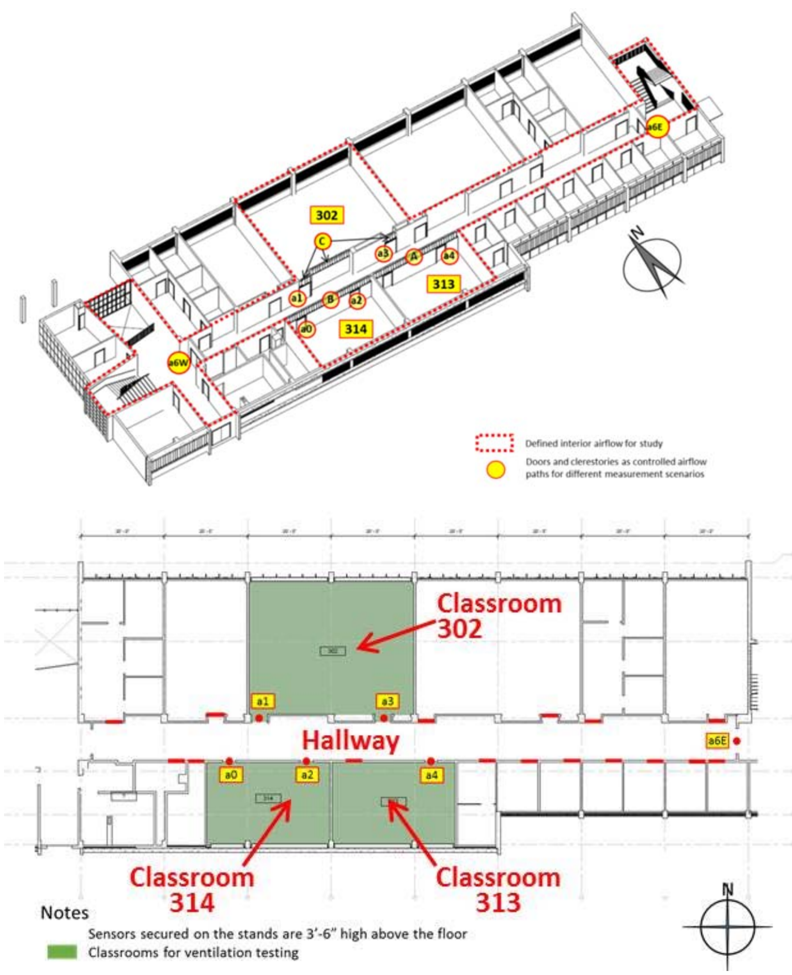


Figure 5.3.1: Definition and description of the IDs of internal doorways and louvered openings

The concept of a **control volume** is important in the understanding of the test scenarios. The control volume is a mathematical abstraction which allows balancing of properties across its boundaries. In the control volume, under steady state conditions, the change in mass is zero. In our case the control volume was used to balance the net mass flow rate for the air moving from room 302 to the classrooms 313 and

314. The inlets and outlets of the control volume, e.g. the control surfaces, were the internal doorways, louvered wall sections between classroom and hallway and sections of the hallway.

Label	Type	Room	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5
a0	Door	314	closed	open	open	open	open
a1	Door	302	closed	open	closed	open	open
a2	Door	314	closed	open	open	open	open
a3	Door	302	closed	open	open	open	open
a4	Door	313	closed	open	open	open	open
a6E	Door	Hall way	open	closed	closed	open	open
a6W	Door	Hall way	open	closed	closed	open	open
A	Clerestory	313	closed	closed	closed	closed	open
B	Clerestory	314	closed	closed	closed	closed	open
C	Clerestory	302	closed	closed	closed	closed	open

Table 5.3.1: Description of the five test scenarios as the combination of open and closed internal openings

ID used for the openings	Type of opening	Location between	Description	size of the openings	
				width (ft)	height (ft)
A0	doorway	Room 314 Hallway	Doorway between Classroom 314 and hallway	3	7
A1	doorway	Room 302 Hallway	Doorway between Classroom 302 and hallway	3	7
A2	doorway	Room 314 Hallway	Doorway between Classroom 314 and hallway	3	7
A3	doorway	Room 302 Hallway	Doorway between Classroom 302 and hallway	3	7
A4	doorway	Room 313 Hallway	Doorway between Classroom 313 and hallway	3	7
A7E	hallway section	Hallway Hallway	Virtual section inside the hallway on the Eastern side of the hallway	8	7
A7E	hallway section	Hallway Hallway	Virtual section inside the hallway on the Southern side of the hallway	8	7
C1	louvered section	Room 302 Hallway	Louvered opening section above the doorway A1	3	2.5
C2	louvered section	Room 302 Hallway	Louvered opening section above the internal wall at 6 feet height	10	2.5
C3	louvered section	Room 302 Hallway	Louvered opening section above the doorway A3	3	2.5
A	louvered section	Room 313 Hallway	Louvered opening section above the internal wall at 6 feet height	8	2.5
B	louvered section	Room 314 Hallway	ouvered opening section above the internal wall at 6 feet height	8	2.5

Table 5.3.2: Description of the internal doorways and louvered openings used in establishing the five test scenarios

5.4 Preparing the Site for Measurements of Test scenarios

This section describes and illustrates the preparation at the test site to carry out measurements of the air flow patterns between the interconnected class rooms and hallway. The five test scenarios represented combinations of doorways and internal louvered wall sections either open or closed. The louvered internal sections needed to be sealed by installing temporary tarps since the several louvers were too deteriorated to safeguard an appropriate seal. Figure 5.4.1 shows the preparation of the internal opening for the test measurements, e.g. the installation of the temporary tarps.

Installing the temporary tarp to seal sections of the louvered openings; seen from classroom 302

Installing the temporary tarp to seal sections of the louvered openings; seen in the hallway (looking East)

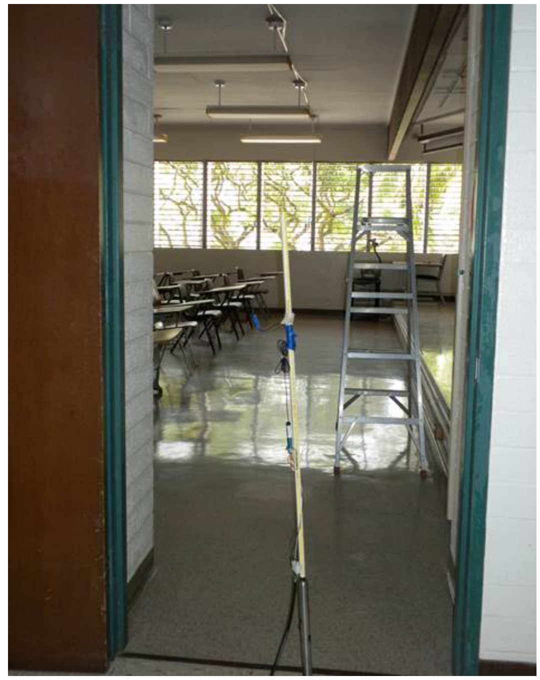
Completed sealing of the louvered section

Removing temporary cover on louvered sections

Figure 5.4.1: Images of preparing the test site for measurements of internal air flow – sealing of the internal louvered sections

Figure 5.4.2 shows the typical placement of the anemometers inside doorways and the hallway.

Deploying the anemometers at the test site; anemoters placement in doorway of room 313 (A4)



Deploying the anemometers at the test site; anemoters placement in doorway of room 302 (A3)

Deploying the anemometers at the test site; pleacement of all anemometers inside the hall way for measurements under scenario 1

Deploying the anemometers at the test site; data acquisition of anemometers with mutilexer; note the extension cords were used to supply the exciitation voltage as well as trasnmit the recorded data of the sensors.

Figure 5.4.2: Deploying the anemoeters at the test site

5.5 Analyzing Wind Data

Wind condition measured outside the Keller Building affect the airflow characteristics of the naturally ventilated spaces. Since the CFD simulations used a distinct approach wind direction a data filter had to be applied that selectively considered only those internal air velocities data points whose timestamp corresponded with a specific wind direction range used in the CFD simulations, e.g. the representative wind approach plus minus a certain degrees to establish the representative range for filtering. Figure 5.5.1 shows the wind conditions during the time when the tests measurements were carried out.

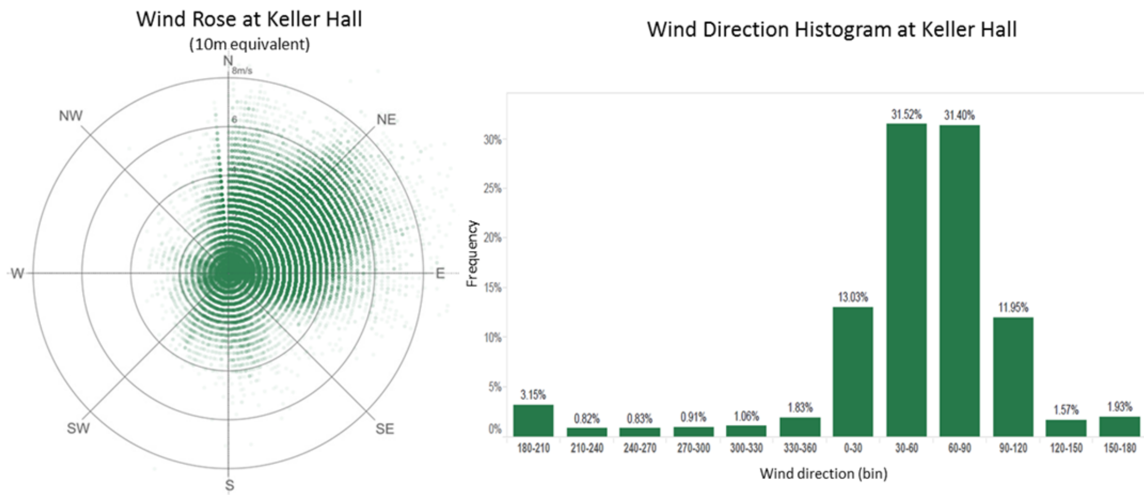
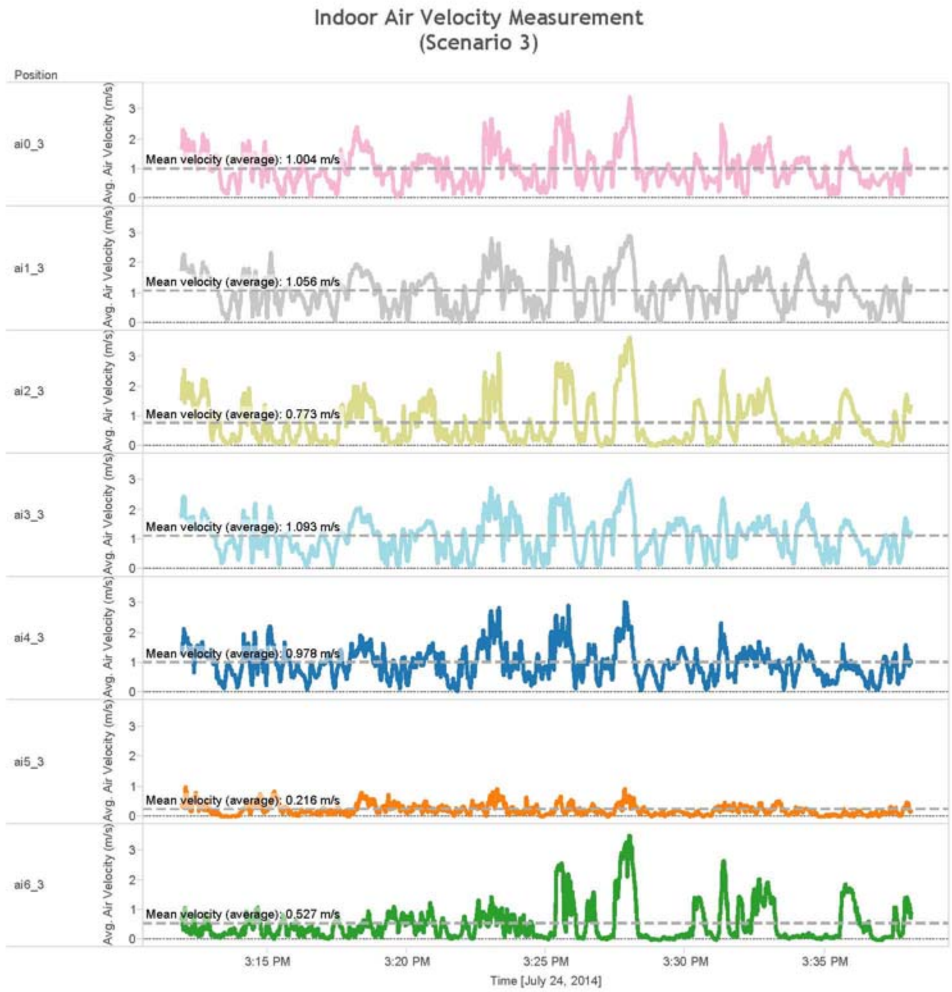


Figure 5.5.1: Sample wind observation for Keller Hall during predominate N/E winds.

5.6 Analysis of Air Velocities Measured in Keller Hall

Measurements of internal air velocities were carried out with hot wire anemometers for all five test scenarios at locations identified as A0 through A6. As a sample data set Figure 5.6.1 shows the recorded data sets for the anemometers for test scenario 3.

Mean air velocities were determined for all anemometers. These mean air velocities were used in the validation of the CFD predictions.



Indoor Air Velocity Measurement (Scenario 3) (Mean)

Position	July 24, 2014
ai0_3	1.004
ai1_3	1.056
ai2_3	0.773
ai3_3	1.093
ai4_3	0.978
ai5_3	0.216
ai6_3	0.527

Indoor Air Velocity Measurement (Scenario 3) (Stdev)

Position	July 24, 2014
ai0_3	0.583
ai1_3	0.607
ai2_3	0.752
ai3_3	0.611
ai4_3	0.529
ai5_3	0.178
ai6_3	0.616

Figure 5.6.1: Internal air velocities recorded for seven anemometers – Sample data set of test Scenario 3

5.7 CFD Investigations

A coupled domain approach was used. The meshing was done with varying sizes of cells. Figure 5.7.1 shows the extent of the computational grid. Figure 5.7.2 shows the mesh for the area of interest in the domain.

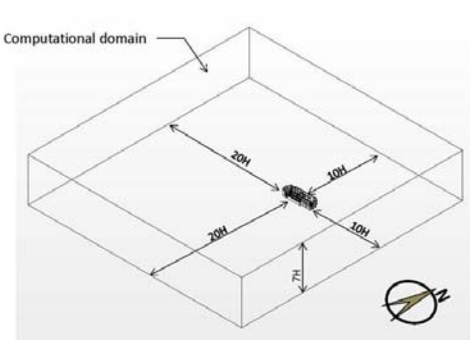
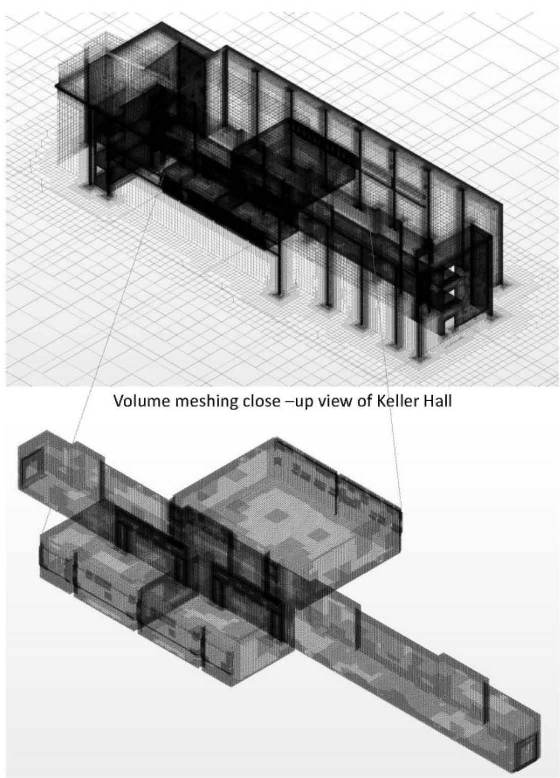


Figure 5.7.1: Extent of the computational domain used for the CFD investigations of the present study (H is the height of Heller Hall



Volume meshing close-up view of the area of interest

Figure 5.7.2: Volume mesh for the area close to Keller Hall

Note: a grid sensitivity analysis revealed a good quality grid

5.8 Categories of CFD Simulations

The methodologies of running the CFD simulations were grouped in three categories of CFD investigations, each with a specific objective. The three groups of CFD simulations carried out are described as follows.

Group 1 CFD runs: The objective was to conduct CFD runs to **determine the flow characteristics** under the five test scenarios using **qualitative post processing**.

- Based on the recorded wind conditions it was determined that a wind approach from North-East (e.g. 45°) was the representative wind direction for these CFD runs.
- Create the CFD boundary conditions that corresponded with the combination of closed and open internal doorways and louvered sections; e.g. for five test scenarios.

Group 2 CFD runs: The objective was to conduct CFD runs to determine predictions of air velocities and allow comparison of CFD air velocity predictions with actual measurement (e.g. **validation phase**)

- Use wind approach from North-East (e.g. 45°) as representative wind direction for these CFD runs.
- Use specific wind approach velocities for each test scenario that were measured during the investigations of the specific test scenario.
- Create the CFD boundary conditions that corresponded with the combination of closed and open internal doorways and louvered sections; e.g. for five test scenarios.
- Create a control volume and verify mass flow rates.

Group 3 CFD runs: The objective was to conduct CFD runs to develop a work **methodology of predicting anticipated air changes rates** for internal spaces based on site specific historical wind records.

- Analyze a longer wind direction record to determine the frequency distribution of wind directions.
- Determine three representative ranges of wind directions, e.g. three ranges of wind direction which have the highest frequency relative to the entire wind record
- For the three ranges of wind directions multiple air speeds were used in CFD simulations to determine the dependence of air changes rates (based on the CFD mass flow rates through these class rooms) on wind approach velocities.
- Establish a correlation function of air change rates versus approach wind velocities. For each range of wind directions one correlation function could be determined.

5.9 Results and Conclusions of Group 1 CFD Runs

The results of the CFD simulations provided descriptions of the air flow patterns through the three classrooms and the hallway. The air flow patterns were visualized with air velocity contour maps and streamlined images using several horizontal and vertical slices. The annotated Figures 5.9.1 through 5.9.4 illustrates the type of air flow visualizations obtained for Test Scenario 3.

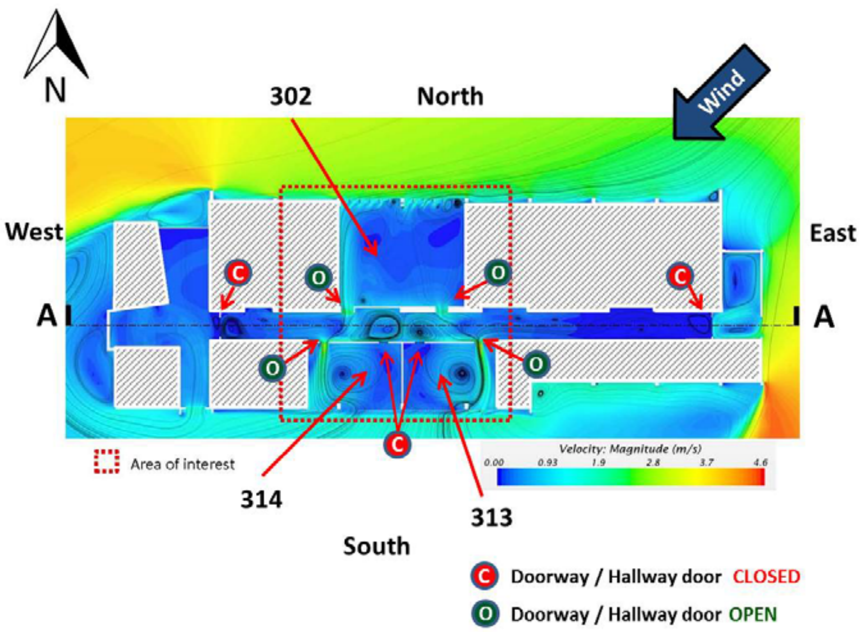


Figure 5.9.1: Visualization of Scenario 3 – Velocity Contour Map on the Horizontal Plane

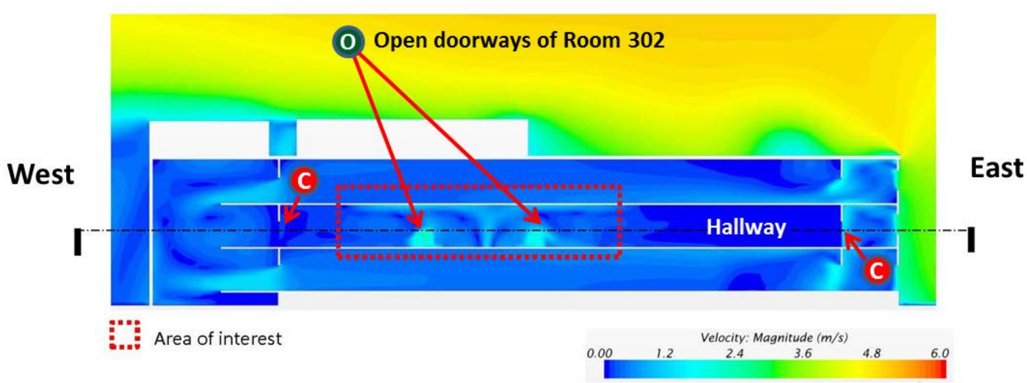


Figure 5.9.2: Visualization of Scenario 3 – Velocity Contour Map on the vertical Plane – Refer to Section A-A in Figure 5.8.1

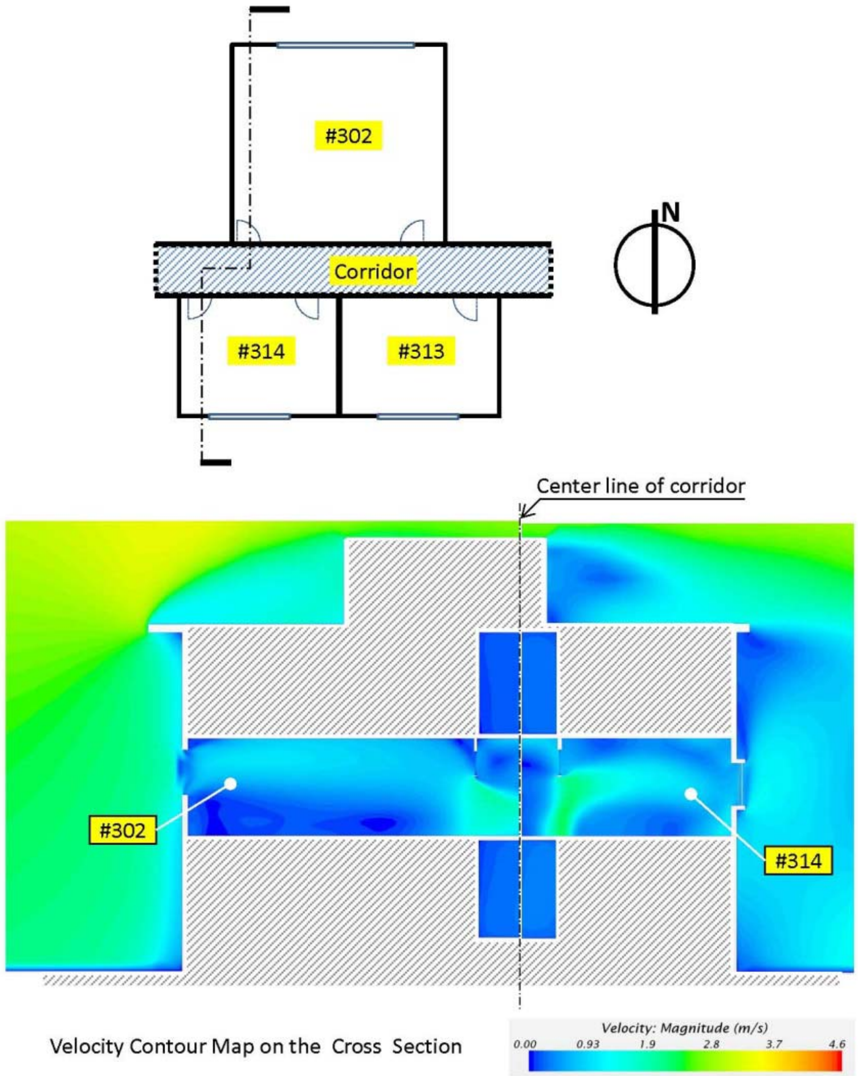


Figure 5.9.3: Visualization of Scenario 3 – Velocity Contour Map on the Cross Section

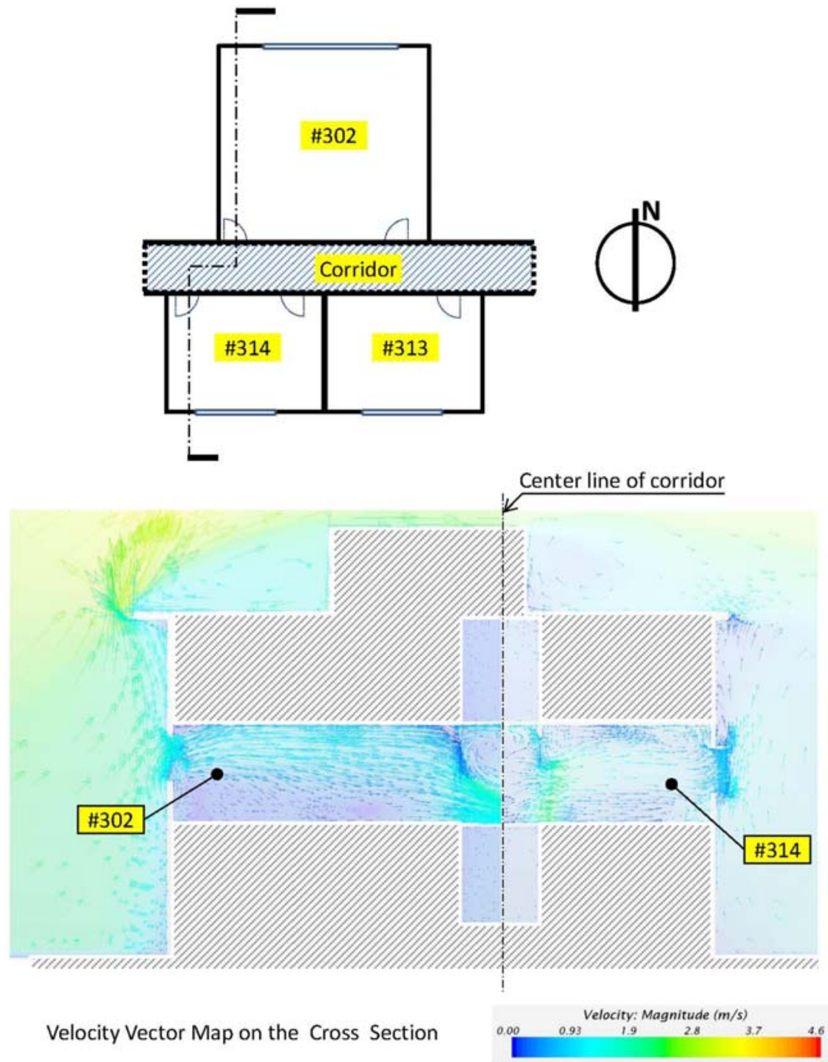


Figure 5.9.4: Visualization of Scenario 3 – Velocity Vector Map on the Cross Section

Summarizing Section 5.9: The visualization presented in Section 5.9 provides a good description of internal air flow patterns. These types of air flow visualizations can provide the designer with qualitative assessment of the internal air flow and how the internal airflow can be affected by internal openings. The type of post processing, color contour maps, streamline maps and presentation grids offer intuitive tools to determine internal flow patterns.

5.10 Results and Conclusions of Group 2 CFD Runs – Validation Test of CFD Results

Determining the representative wind velocity for test scenarios: The CFD simulations for all five test scenarios used the same approach wind direction of 45° but with varying wind velocities. The wind velocities for the CFD simulations of the CFD Group 2 were determined by analyzing the wind record collected during the test measurements and evaluating the specific wind approach for individual test scenarios. During each test scenario the wind velocities were measured by the weather station and one representative wind velocity for each of five test scenarios was defined. Figure 5.10.1 and 5.10.2 show the representative wind conditions from which the wind approach velocities were determined for the five test scenarios.

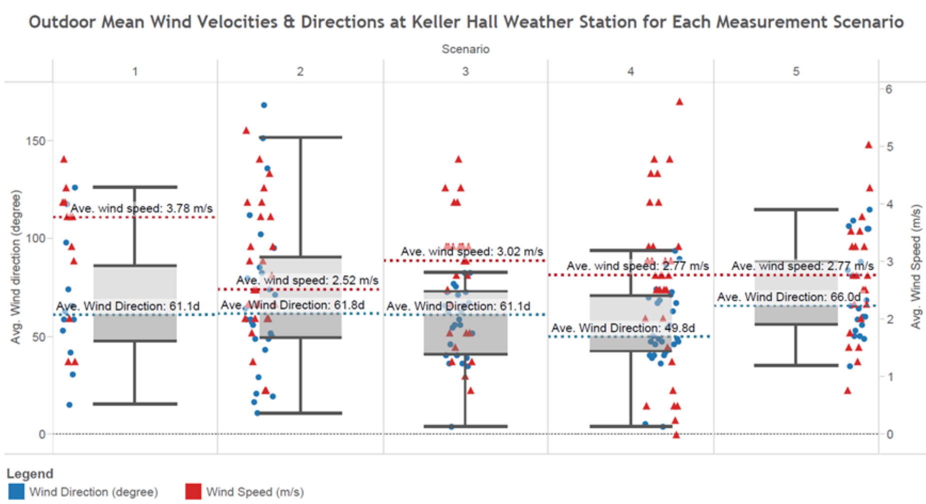


Figure 5.10.1: 10m equivalent height wind speed and wind direction at the Keller Hall weather station obtained at the same time as the indoor air velocity measurement for all test scenarios 1 through 5

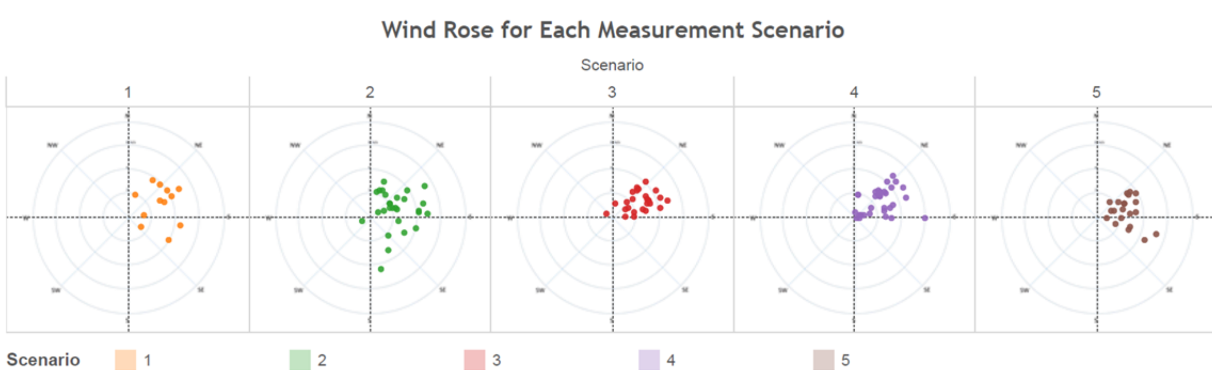


Figure 5.10.2: Wind direction at the Keller Hall weather station obtained at the same time as the indoor air velocity measurement for all test scenarios 1 through 5

Comparison between CFD Calculated and Measured Air Velocities: The results of the CFD simulations and the corresponding measurements are compared in Table 5.10.1 and Figures 5.10.3 through 5.10.7.

Table 5.9.1: Comparison between the CFD simulations and measurements for all five test scenarios.

Simulation							
Scenario	Uref {m/s}	a0 {m/s}	a1 {m/s}	a2 {m/s}	a3 {m/s}	a4 {m/s}	a6 {m/s}
1	3.78	0.32	0.32	0.31	0.31	0.30	0.63
2	2.52	0.86	1.22	0.88	1.10	1.08	#N/A
3	3.02	1.35	1.23	#N/A	1.06	1.46	#N/A
4	2.77	0.85	1.39	0.95	1.23	1.14	0.32
5	2.77	0.13	1.57	0.46	0.87	0.54	0.46

Measurement							
Scenario		a0 {m/s}	a1 {m/s}	a2 {m/s}	a3 {m/s}	a4 {m/s}	a6 {m/s}
1 note**	Mean	0.29	0.26	0.24	0.30	#N/A	0.83
	Stdev	*0.24*	*0.22*	*0.19*	*0.26*	*#N/A*	*0.58*
2	Mean	0.86	1.22	0.88	1.10	1.08	#N/A
	Stdev	*1.06*	*1.33*	*0.90*	*1.44*	*1.23*	*#N/A*
3	Mean	1.00	1.06	#N/A	1.09	0.98	#N/A
	Stdev	*0.58*	*0.61*	*#N/A*	*0.61*	*0.53*	*#N/A*
4	Mean	0.63	1.05	0.58	0.99	0.85	0.29
	Stdev	*0.57*	*0.73*	*0.42*	*0.67*	*0.61*	*0.24*
5	Mean	0.52	0.94	0.45	1.01	0.79	1.03
	Stdev	*0.37*	*0.57*	*0.28*	*0.61*	*0.57*	*0.82*

** Note: For Test Scenario 1 the anemometers a0, a1, a2, a3, a4, were installed along the axis of the hallway (offset 4'-6" away from its original locations) not at the doorways of the class

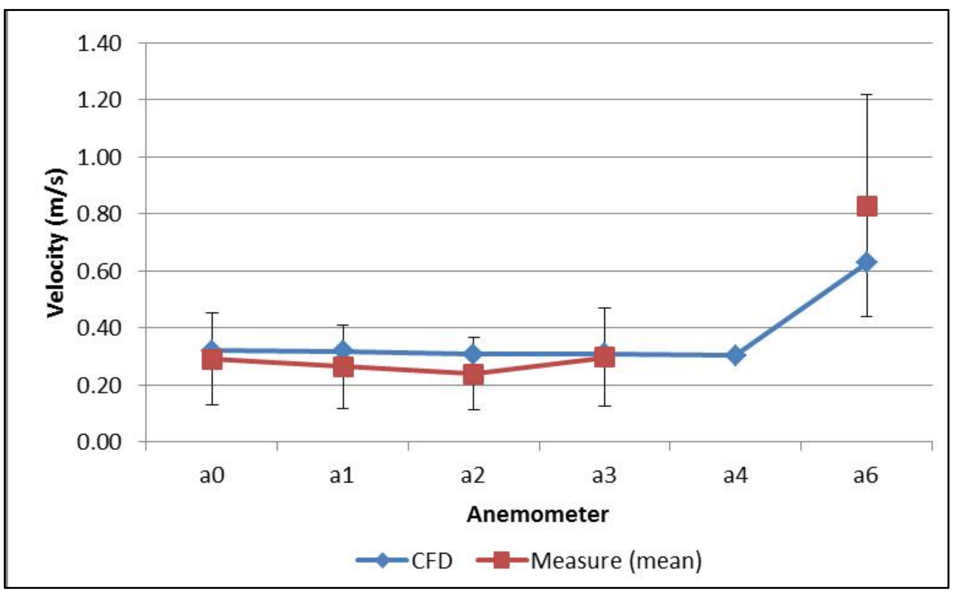


Figure 5.10.3: Comparison between the CFD simulations and measurements for test scenario 1.

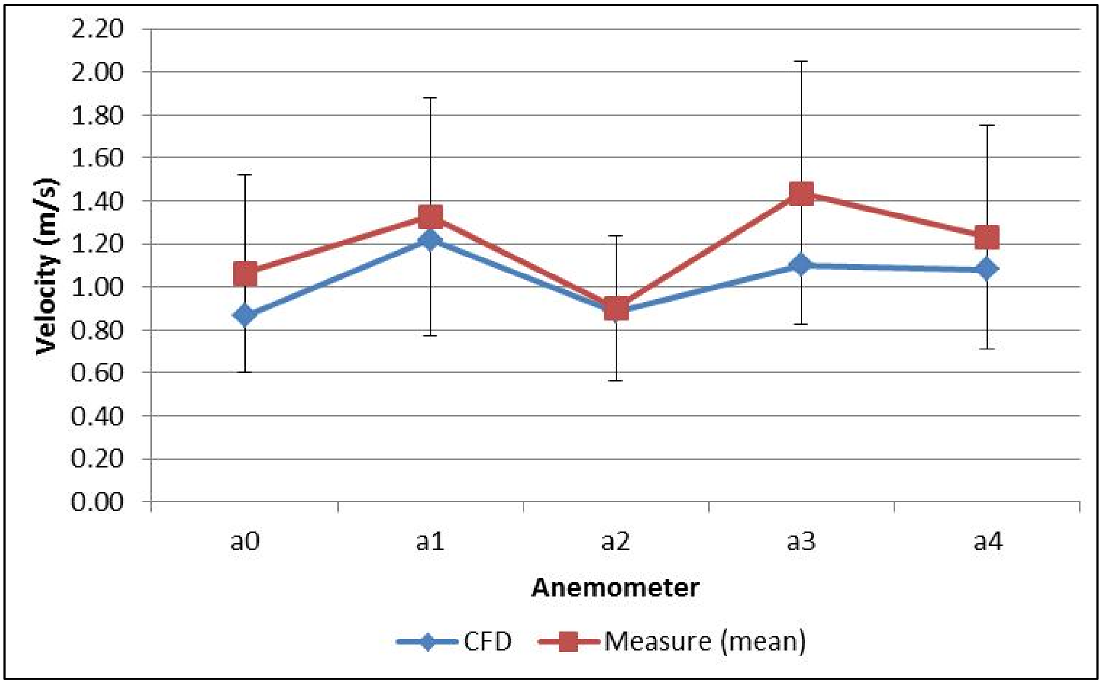


Figure 5.10.4: Comparison between the CFD simulations and measurements for test scenario 2.

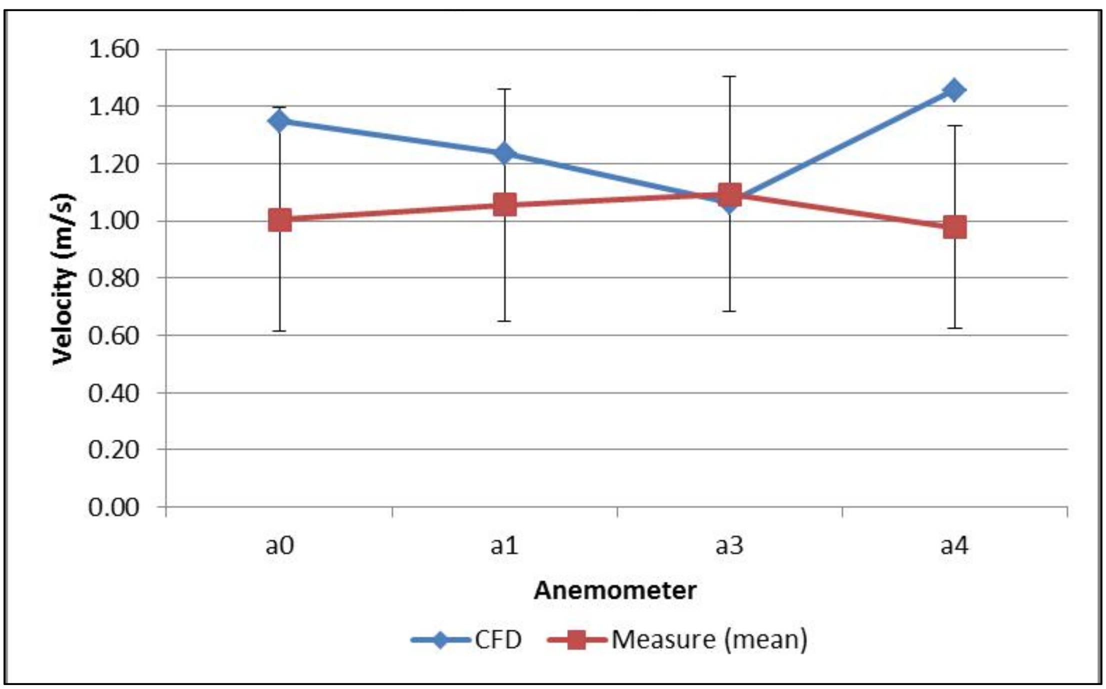


Figure 5.10.5: Comparison between the CFD simulations and measurements for test scenario 3

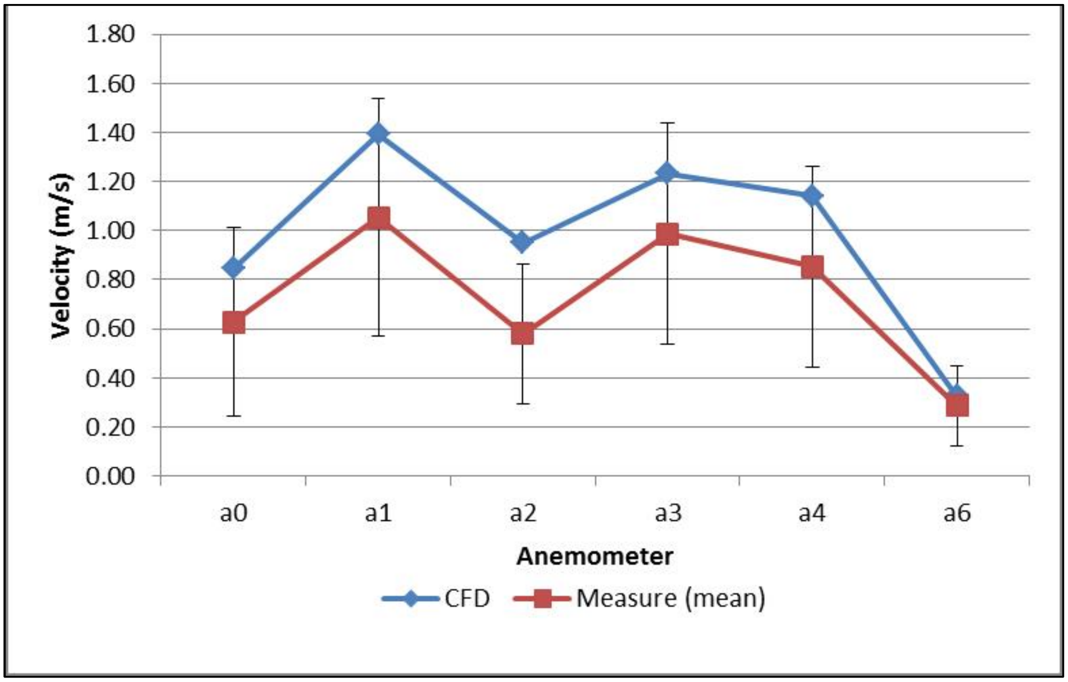


Figure 5.10.6: Comparison between the CFD simulations and measurements for test scenario 4

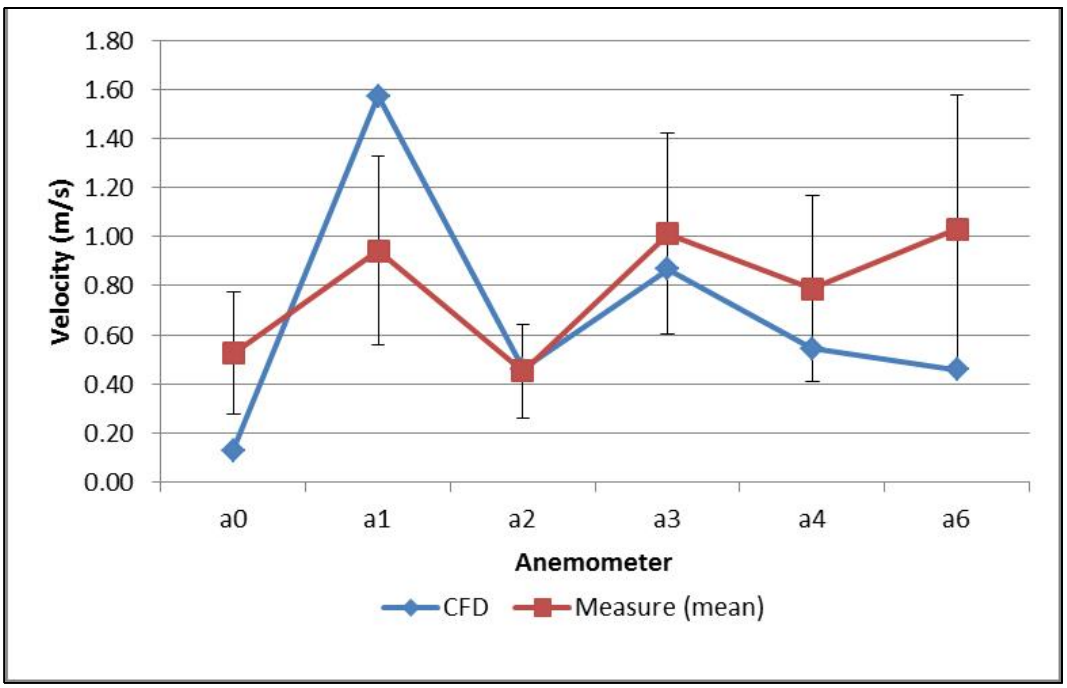


Figure 5.10.7: Comparison between the CFD simulations and measurements for test scenario 5

Mass Flow Verification for Control Volume: The results of the CFD analysis conducted for the validation of measured air velocities was verified by applying the principle of mass conservation to the control volume. For air flow conditions in the control volume under steady state the net sum of inflow and outflow must be zero. Table 5.10.2 presents the results of the mass balance of the control volume for the five test scenarios. As can be seen the net mass flow rate is zero four test scenarios and smaller or equal to 0.2% for one test scenario. Figure 5.10.8 defines the control volume and control surfaces

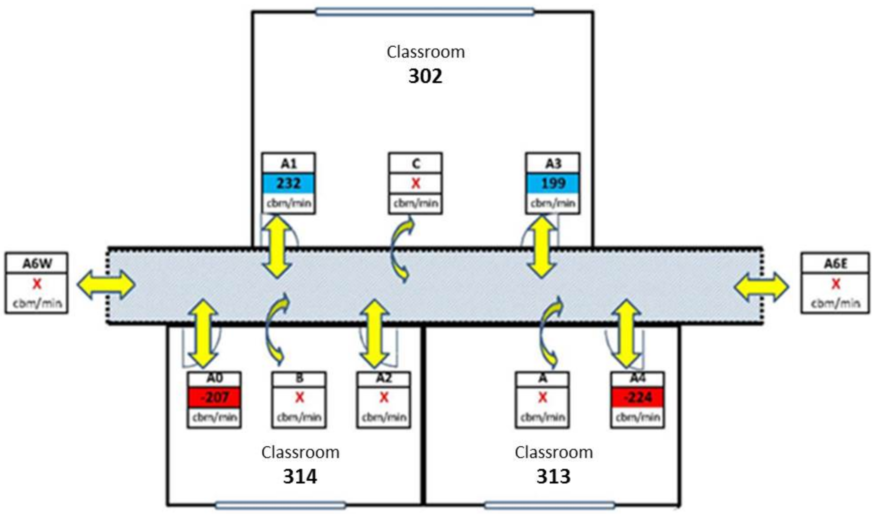


Figure 5.10.8: Verification of mass flow conditions for control volume for all test scenarios 1 through 5 – Definition of control surfaces.

Table 5.10.2: Verification of mass flows conditions for control volume for all test scenarios 1 through 5

	Mass flow rates of control surfaces [m^3/min]					
Control surface ID	**Test scenario 1**	**Test scenario 2**	**Test scenario 3**	**Test scenario 4**	**Test scenario 5**	**Descripion of control surface (see Fig. 5.2.4.1)**
A0		-157	-207	-142	-24	Doorway room 314
A1		270	232	283	320	Doorway room 302
A2		-159		-158	-73	Doorway room 314
A3		243	199	249	174	Doorway room 302
A4		-197	-224	-189	-91	Doorway room 313
A6E	147			107	95	Hallway section
A6W	-147			-149	-198	Hallway section
A					-357	Louvers room 313
B					-446	Louvers room 314
C					600	Louvers room 302
sums >>	0	0	0	1	0	

Mass balance of the control volume:

Inflow	147	513	431	639	1189
Outflow	-147	-513	-431	-638	-1189
Residual	**0**	**0**	**0**	**1**	**0**
Error	0.0%	0.0%	0.0%	0.2%	0.0%

Legend for color coding

	Inflow into control volume shown with Positive value
	Outflow out of control volumeshown with negative value
	Control surface was closed

Summarizing Section 5.10: The results of the CFD simulations and the measurements for the air velocities and the "virtual" and actual anemometers compare well. Differences between the two types of values, e.g. the theoretical CFD results and the actual measurements, were within the range that could be expected. Overall, the validation of CFD results with actual measurements can be regarded as **successful**. The fact that relatively high air velocities could be simulated and observed in the doorways and hallway sections gives confidence that the results depict actual conditions of air flow at the test site. The control volume approach proved that conservation of mass was achieved and the net mass flow rates on the control volume were zero. This verified that the CFD solution was a valid numerical solution.

5.11 Proposed Work Process to Predict Natural Ventilation Performance of Spaces

A method was developed that uses CFD prediction of air flow through buildings and links the resulting air flow characteristics with wind records at the site. The result is an expression of anticipated site-specific natural ventilation performance, expressed as probability of exceedance of air change rates. Figure 5.10.1 shows the flow chart of the proposed method:

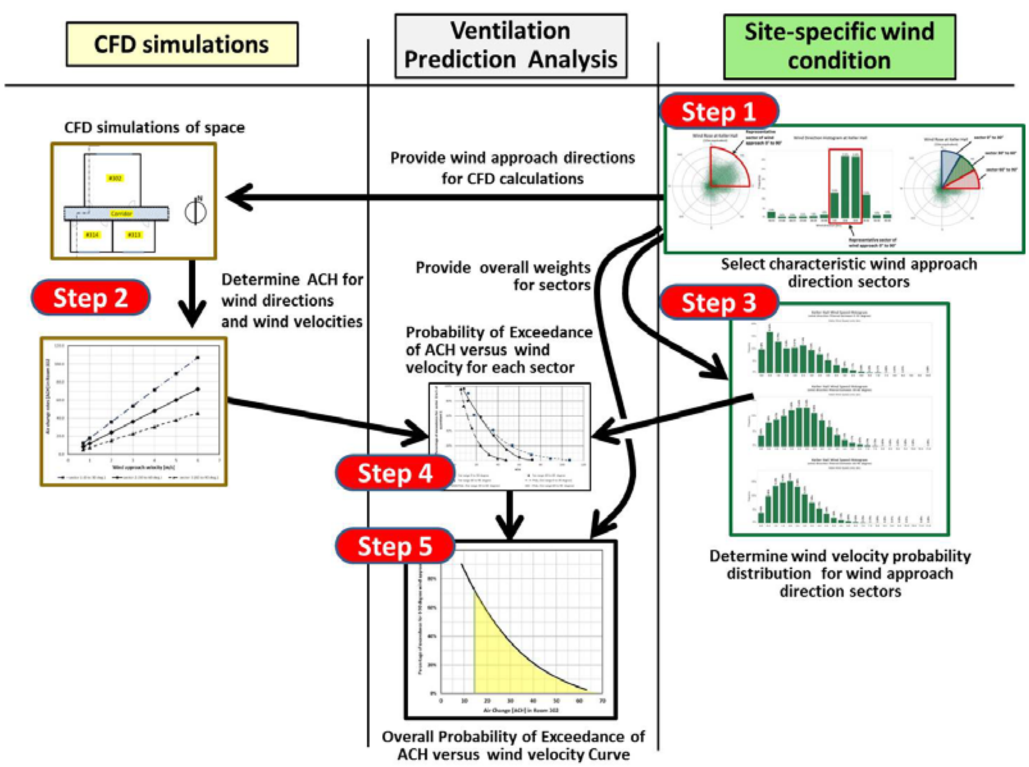


Figure 5.11.1: Flow chart for the proposed method of determining the predicted natural ventilation performance

Step 1 of proposed work process: In the first process step a site-specific wind record is obtained. The wind record is presented as a wind rose, which displays data for wind direction and wind velocities. The probability density of wind direction determines what wind direction are representative, which means what wind direction contribute most. In our case the first quadrant (e.g. 0° to 90°) of the wind rose contains some 75% of all wind direction observations. In order to get a more precise resolution the first quadrant is divided into three sectors, namely the sectors 0° to 30°, 30° to 60° and 60° to 90°. The representative wind directions for the three sectors are their bisectors, e.g. 15°, 45° and 75°, for sectors 1,2 and 3, respectively.

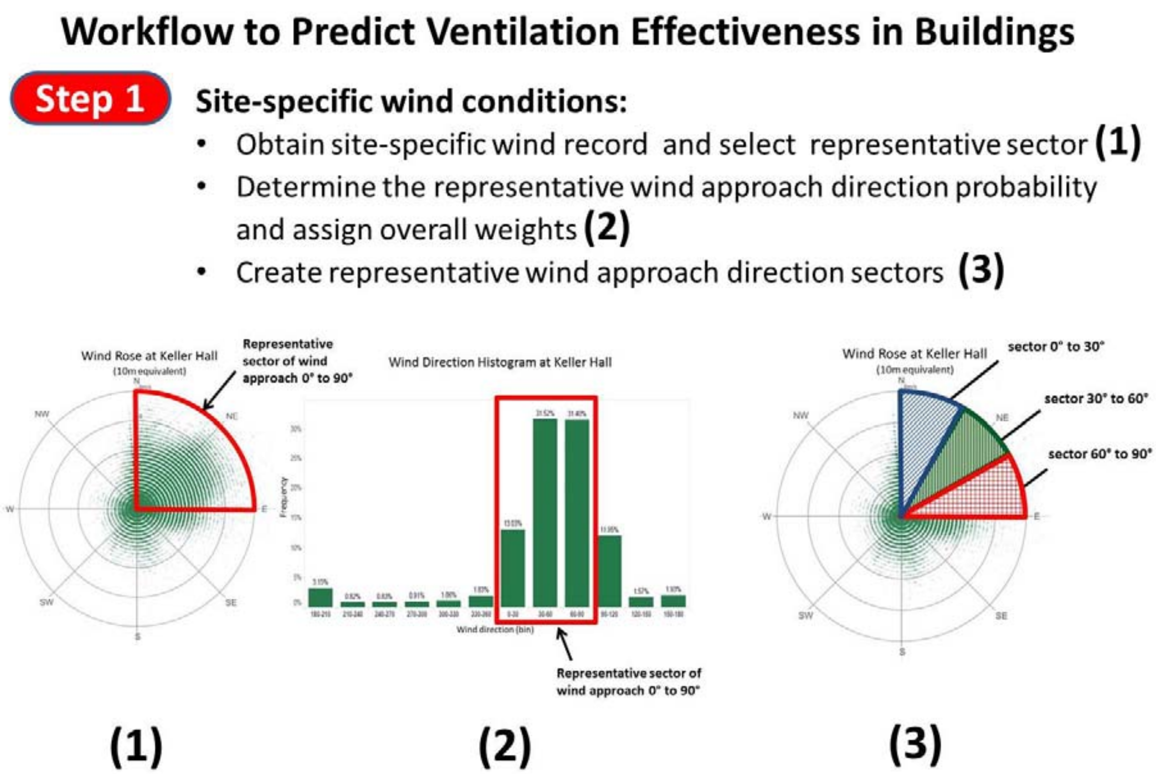


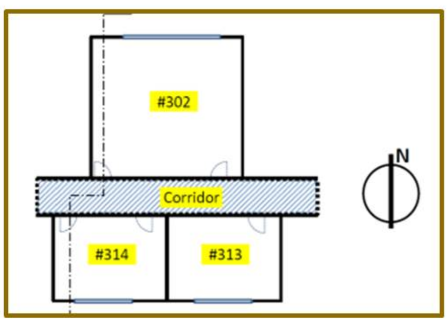
Figure 5.11.2: Step 1 of the Workflow to Predict Ventilation Effectiveness in Buildings; determining wind direction probabilities

Step 2 of proposed work process: In the second step of the work process CFD simulations are carried out for each sector, using the bisector of the sector as the representative wind direction. For each sector CFD simulations are carried out for unit wind velocities, such as 1.0, 2.0, 3.0 and 4.0 m/sec. The resulting air flow serves to determine the mass flow rate through the spaces that are investigated. The air change rate in ACH is determined and plotted against the wind velocity. The trend lines for the ACH versus wind velocities is determined in order to provide a means to predict the ACH for any wind velocities.

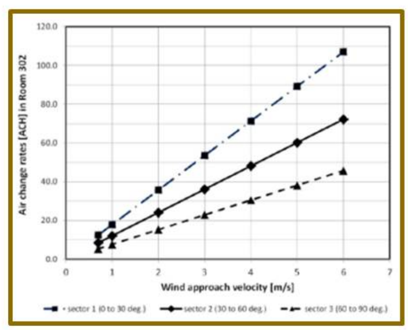
Workflow to Predict Ventilation Effectiveness in Buildings

Step 2 **CFD simulation:**

- For representative wind approach directions of sector determine air flow in space site-specific wind record **(1)**
- For unit different wind approach velocities (e.g. 1 , 2, 3 and 4 m/sec) determine air change per hour (ACH) for each sector
- Determine trend line of ACH versus wind velocity for each sector **(2)**



(1)



(2)

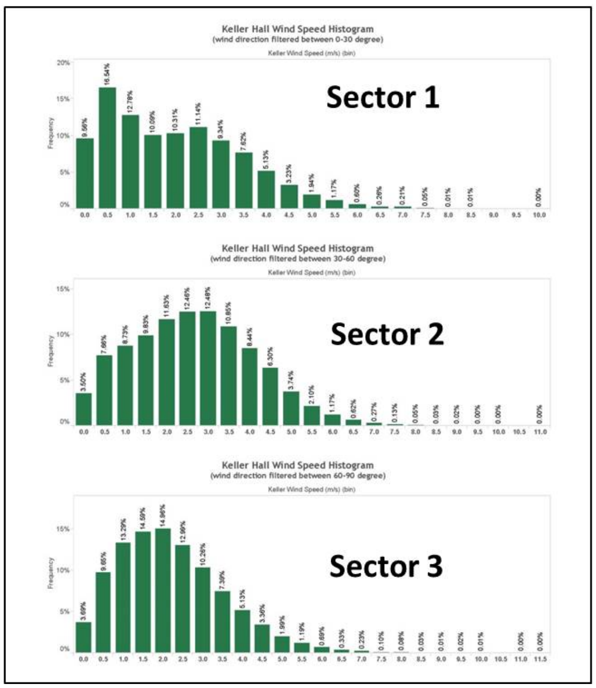
Figure 5.11.3: Step 2 of the Workflow to Predict Ventilation Effectiveness in Buildings; performing CFD simulations for representative wind direction and unit wind velocities

Step 3 of proposed work process: In the third step of the work process the probabilities of wind velocities for the sectors are determined, which represent the probability densities functions for wind velocities for every sector.

Workflow to Predict Ventilation Effectiveness in Buildings

Step 3 **Site-specific wind conditions:**

- Obtain the wind velocity probabilities for each sector **(1)**



(1)

Figure 5.11.4: Step 3 of the Workflow to Predict Ventilation Effectiveness in Buildings; obtaining the probability function of wind velocities for each sector

Step 4 of proposed work process: In the fourth step of the work process the trend lines of the ACH versus wind velocity relationships are applied to the probability densities for the wind velocities (Step 3). The correlation provides a probability of exceedance graph of air changes for the site. The probability of exceedance is relative to the wind record that is available, this mean the period covered by the wind record. Figure 5.11.5 shows three graphs which represent the three sectors, 0° to 30°, 30° to 60° and 60° to 90°.

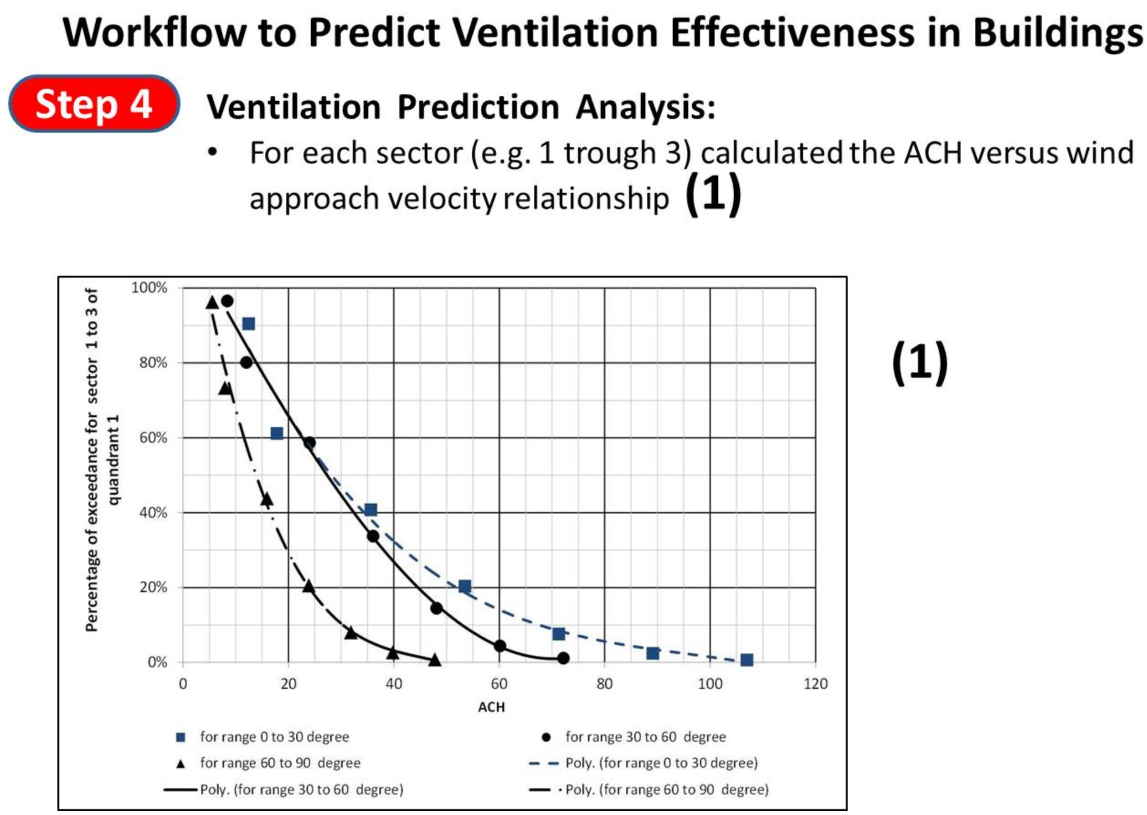


Figure 5.11.5: Step 4 of the Workflow to Predict Ventilation Effectiveness in Buildings; probability of exceedance of wind velocities at the site, for each sector

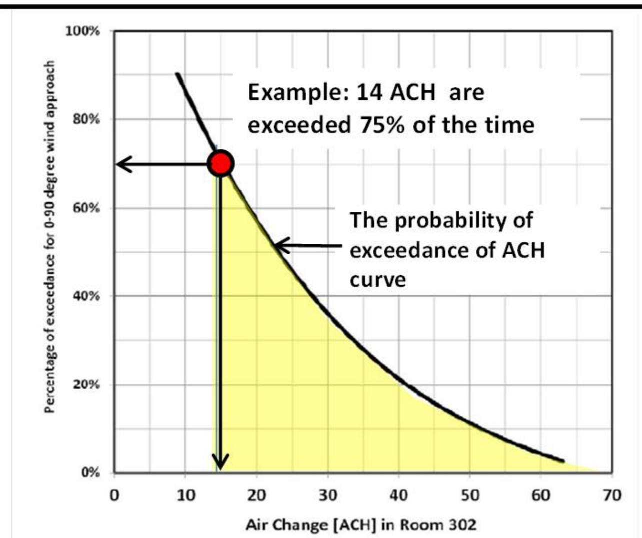
Step 5 of proposed work process: In the fifth and last step of the work process the overall probability of exceedance is determined by applying the overall weights of the sectors to the individual probability of exceedance graphs in Figure 5.11.5. The result of this is the overall graph of probability of exceedance of air changes per hour (ACH) for the building, or the space, at the site.

The graph can be a **valuable design tool** for the building designer since the natural ventilation rate can be determined in the early design phase. At this time only the concept massing of the building needs to be known in order to determine pathways through the building.

Workflow to Predict Ventilation Effectiveness in Buildings

Step 5 **Ventilation Prediction Analysis:**

- Apply overall weights for each sector to the ACH versus wind approach velocity relationship s to get overall ventilation performance prediction on the basis of probability of exceedance **(1)**



The probability of exceedance of ACH for room 302

(1)

Figure 5.11.6: Step 5 of the Workflow to Predict Ventilation Effectiveness in Buildings; overall probability of exceedance of wind velocities at the site

Summarizing Section 5.11: A workflow to predict exceedance rates of air changes per hour (ACH) for the site was developed during the studies and benchmarked using one class room and one test scenario. The procedure included the calculation of ACH as a function of wind approach directions from three different sub-sectors of the first quadrant of site-specific wind rose and varying wind velocities. These ACH results were then correlated with the long-term wind record at the test site. The resulting procedure provides an expression of expected exceedance of ACH for the length of the wind record, and therefore is an indicator of ventilation effectiveness based on probability of wind direction and velocities. This procedure, if more developed, can be an effective tool for the designer to assess the natural ventilation effectiveness of buildings.

Appendix

Presentation Given at the Star Global Conference 2015

STAR Global Conference 2015

March 16-18, 2015 in San Diego, CA

CFD Simulations of Air Movement Around & Inside Buildings in Hawaii

Manfred J. Zapka, PhD, PE
Tuan Tran, D-Arch
Environmental Design and Research Laboratory (ERDL)
University of Hawaii, School of Architecture

STAR Global Conference 2015

March 16-18, 2015 in San Diego, CA

CFD Simulations of Air Movement Around & Inside Buildings in Hawaii

Manfred J. Zapka, PhD, PE
Tuan Tran, D-Arch

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

Speakers

Manfred J. Zapka
Ph.D., P.E.,
Certif. ENV SP, LEED AP, CEM, CEA

Tuan Tran
D.Arch
Certif. LEED AP

Current Research:
- Sustainable systems design
- CFD in the built environment
- Advanced building modelling
- Natural ventilation and occupant comfort

- **Adjunct Professor for Sustainable System Design**
- **Architectural & Energy Engineering Advisor, ERDL**
  University of Hawaii, School of Architecture
  mzapka@hawaii.edu

- **Principal**
  Sustainable Design & Consulting LLC,
  sustainabledc@gmail.com

- **Postdoc Researcher at ERDL**
  University of Hawaii, School of Architecture

BACKGROUND

Hawaii = Gem of the Pacific

Sober (energy) reality in Hawaii

- Highest electricity rates in the Nation (Hawaii $0.36/kWh; US $0.12)
- About 80% of the electricity is generated with oil (US average about 1.5%)
- Buildings consume most of the electricity generated
- High energy demand from HVAC (cooling and ventilation)
- **University of Hawaii** requires **~ 2/3 of energy costs to cool and ventilate buildings**

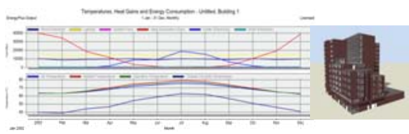
Solution (within our area of research)**:** Reduce high energy demand for cooling and ventilating buildings

RESEARCH AT ERDL

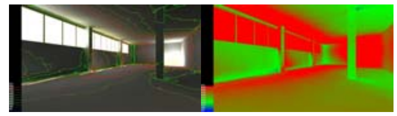
- **University of Hawaii** is very active in applied R&D in sustainable built environment and energy conservation.
- **ERDL** (Environmental Research and Design Laboratory) has been conducting research in high performance building modelling and optimization.
- **HNEI** (Hawaii Natural Energy Institute) is strategic partner of ERDL

BUILDING SIMULATION AT ERDL

Energy Modelling

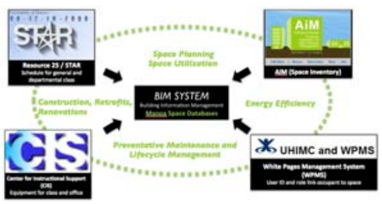


Daylighting Simulations

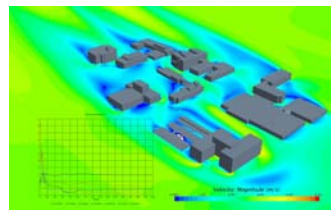


DAYLIGHTING SIMULATION

BIM



CFD



Current focus area
Natural Ventilation & Comfort

USE OF BUILDING SIMULATION

Opportunity

Challenge

Opportunity:

- Building simulations are **increasingly becoming an important tool** for the building designer;
- Mostly energy and lighting simulation;
- **CFD is something NEW** (as conventional building design tool) but is becoming more popular among architects and architects students for “normal projects”;

Challenge:

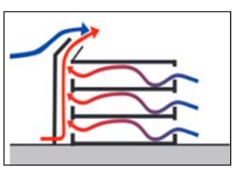
- CFD is COMPLICATED ; there are CFD “light” packages which offer “quick and easy” result. But good CFD analysis requires solid CFD application knowledge

RESEARCH PROGRAM AT ERDL

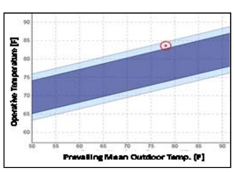
- **ERDL** is working on a two year effort develop skill sets in the use of CFD for architecture and engineering students and professionals in the building industry.
- HNEI is sponsor of current research.
- A **multidisciplinary team** combining CFD analysis and field measurements for validation.

OBJECTIVES OF CURRENT RESEARCH FOCUS

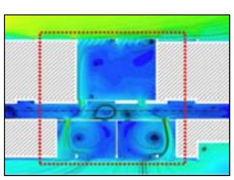
Strategies to Reduce energy demand of HVAC



Natural ventilation & cooling to replace or lower energy demand by mechanical ventilation and cooling



New occupant comfort paradigm of **"adaptive comfort"**

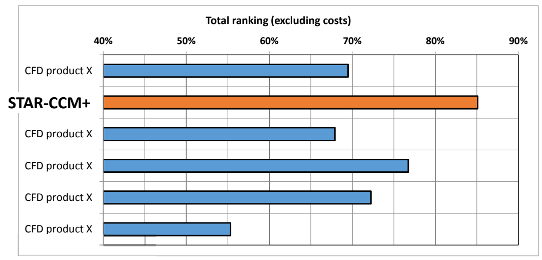


Sophisticated design tools = CFD for thermal and ventilation building performance

INCREASING CFD SOLUTIONS FOR THE BUILDING INDUSTRY

We performed a **ranking** (e.g. benchmarking) of **CFD software products** **Three level ranking ~ 50 criteria**

Criteria Group	Description of criteria group	possible overall weights
1	**Value for the CFD software product to the architecture and building market**	**15.0%**
2	**Support by vendor / expert network**	**15.0%**
3	**Workflow - summary**	**52.0%**
3.1	Ease of use / GUI	7.8%
3.2	Modeling / Import geometry	7.8%
3.3	Meshing	10.4%
3.4	Physics Models	18.2%
3.5	Post processing	2.6%
3.6	Customization / special functions	2.6%
3.7	Support for validation	2.6%
4	**Cost of Software**	**18.0%**



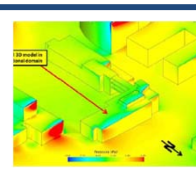
CD-adapco has been a great support to us - **Mahalo**

THREE PHASES OF OUR CURRENT RESEARCH

1 External CFD Applications

completed

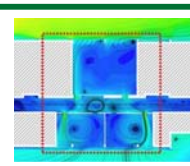
- CFD in urban environment
- Air flow around buildings
- External pressure distribution



2 Internal CFD Applications

completed

- CFD inside buildings
- Air flow inside buildings
- Optimize natural ventilation



3 Occupant Comfort

In progress

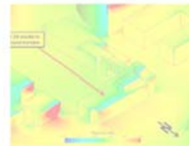
- CFD in comfort studies
- Providing good comfort with less energy demand for HVAC
- Comfort islands

DISCUSSION OF PROJECT PHASES

1 **External CFD Applications**

completed

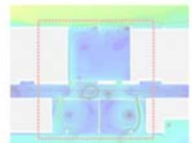
- CFD in urban environment
- Air flow around buildings
- External pressure distribution



2 **Internal CFD Applications**

completed

- CFD inside buildings
- Air flow inside buildings
- Optimize natural ventilation

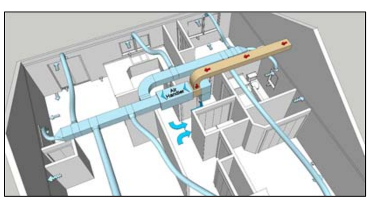


3 **Occupant Comfort**

In progress

- CFD in comfort studies
- Providing good comfort with less energy demand for HVAC
- Comfort islands

PROJECT PHASE 3: OCCUPANT COMFORT



Conventional thermal comfort paradigm:

Cool the entire room (building)

BAU (business-as-usual) in HVAC:

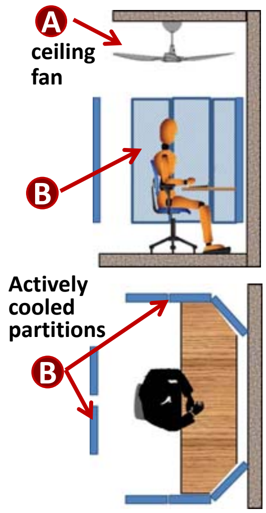
- avoid new HVAC approaches - “you are safe”
- BUT ... There is a high energy burden

We are going new ways ... **COMFORT ISLANDS** ..

Provide thermal comfort **where it counts** ... at the **occupant level** and avoid cooling the entire space

PROJECT PHASE 3: OCCUPANT COMFORT

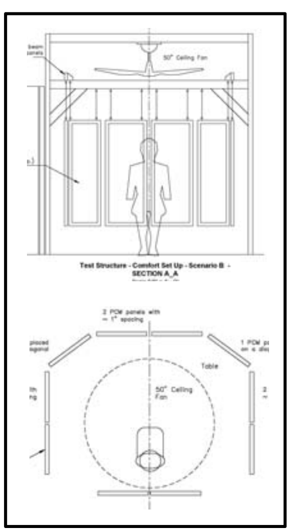
Two main technologies to increase thermal comfort in spaces - They are **NOT conventional** approaches in building design



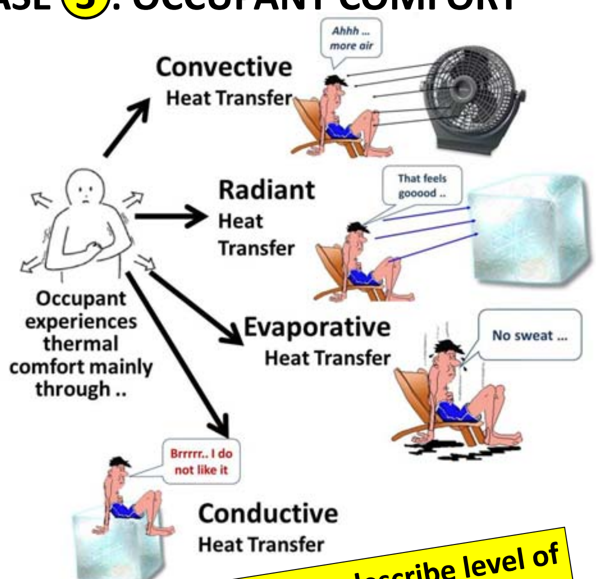
A **Increase air movement over body** = lowering the **perceived** temperature

B Provide **radiant cooling WITHOUT dehumidification** - New concept for tropical climate

PROJECT PHASE 3: OCCUPANT COMFORT



Our internal comfort test camber

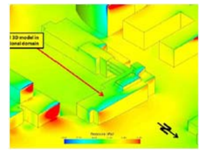


CFD to describe level of thermal comfort

THREE PHASES OF RESEARCH WORK

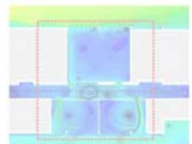
1 External CFD Applications — completed

- CFD in urban environment
- Air flow around buildings
- External pressure distribution



2 Internal CFD Applications — completed

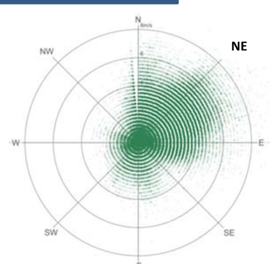
- CFD inside buildings
- Air flow inside buildings
- Optimize natural ventilation



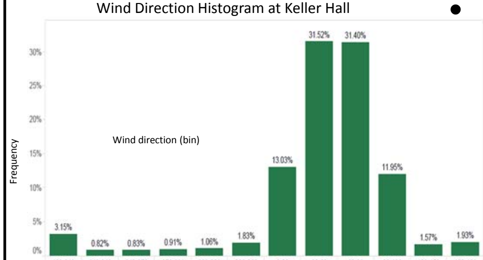
3 Occupant Comfort — In progress

- CFD in comfort studies
- Providing good comfort with less energy demand for HVAC
- Comfort islands

External CFD Applications



Wind Direction Histogram at Keller Hall



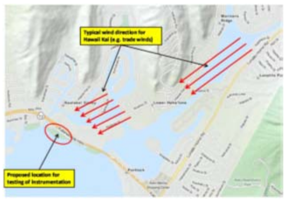
Wind Rose at Keller Hall
(10m equivalent)

OBJECTIVES

- Define consistent prevailing wind condition in Hawaii as a “wind tunnel” opportunity for selecting the full-scaled measurement site
- Measure air movement around buildings and validate with CFD predictions

External CFD Applications

SHAKE-DOWN TESTING WITH A CAR - "NATURAL WIND TUNNEL"



Site selection for wind consistence

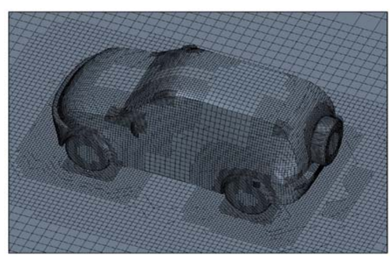
Installation of anemometers and pressure sensors

The test site at Hawaii Kai was chosen for quite consistent wind condition and few obstruction

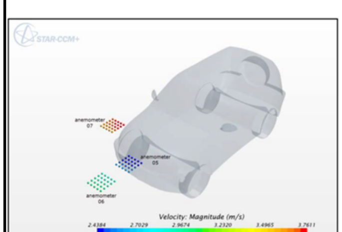
External CFD Applications

SMALL MODEL – A CAR

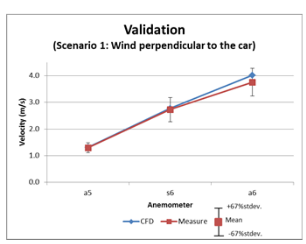
Installation of anemometers and pressure sensors

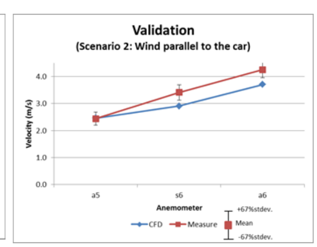


Volume cells of the car



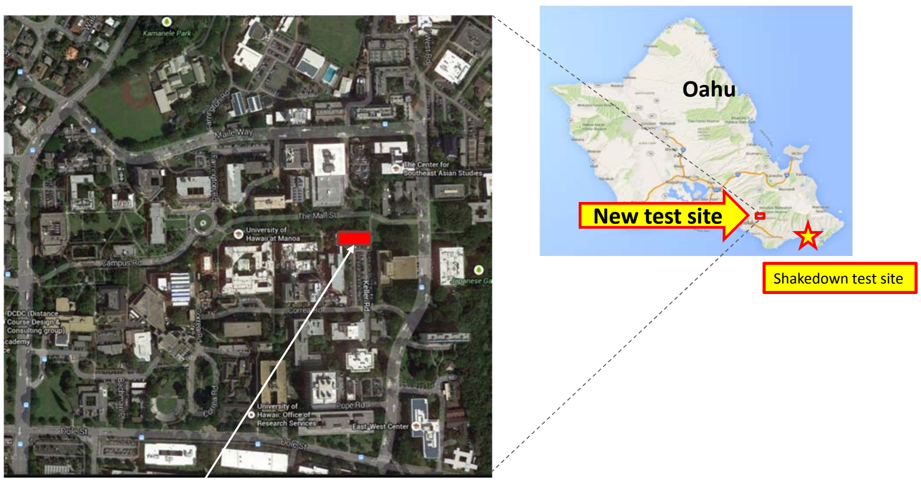
1'x1' presentation grid probes at locations of sensors to extract data for CFD validations (Scenario 2)





External CFD Applications

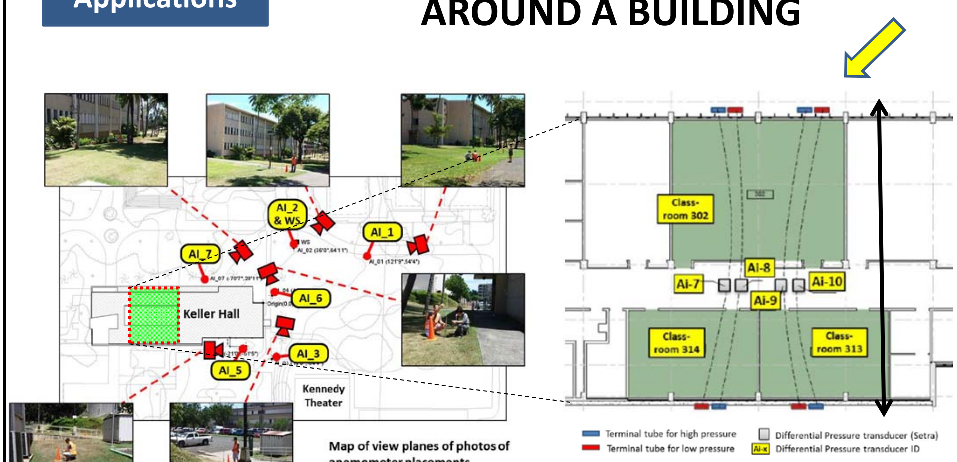
LARGE MODEL – A BUILDING



Keller Hall
University of Hawaii

External CFD Applications

FIELD MEASUREMENT OF AIRFLOW AROUND A BUILDING

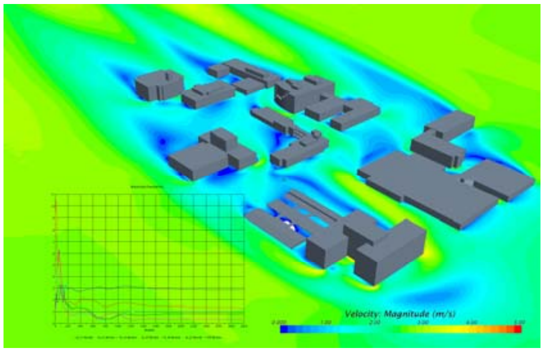


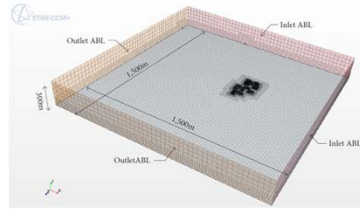
Installation of pressure tubing pairs and differential pressure transducers on the Keller Hall 3rd floor

Installation of anemometers around the Keller Hall

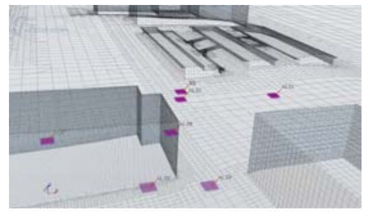
External CFD Applications

RESULTS

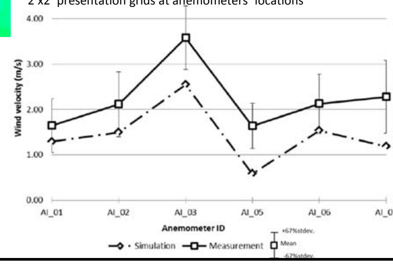




Volume cell meshing of the computational domain



2'x2' presentation grids at anemometers' locations



- Rhino3D (CAD) -> Mesher (Remesher/Wrapper)
- Trimmer
- Steady-state, isothermal model
- RANS Realizable k-epsilon turbulence model
- Two layer high y+ wall treatment with variable sizes for near-wall cells

External CFD Applications

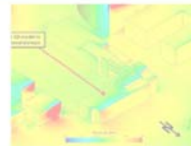
DISCUSSION & IMPORTANT CONCLUSIONS

- A lot of trial-and-error and **lessons learned** from deployment of experiment equipment and data acquisition.
- The **periodic fluctuations of differential pressures** over the building caused by eddies was observed, resulting in natural ventilation performance of the building.
- The use of **wind speed is preferred** to differential pressures to validate CFD simulation results

THREE PHASES OF RESEARCH WORK

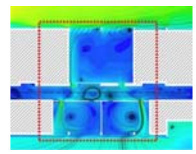
1 External CFD Applications
completed

- CFD in urban environment
- Air flow around buildings
- External pressure distribution



2 Internal CFD Applications
completed

- CFD inside buildings
- Air flow inside buildings
- Optimize natural ventilation



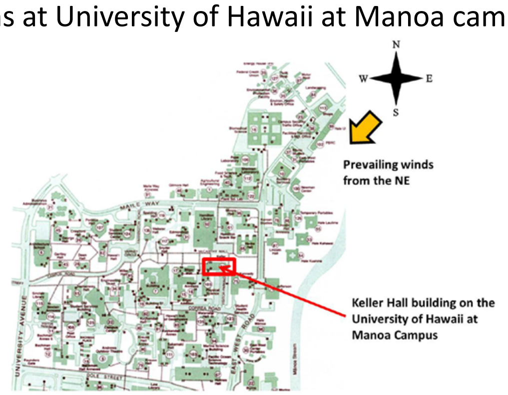
3 Occupant Comfort
In progress

- CFD in comfort studies
- Providing good comfort with less energy demand for HVAC
- Comfort islands

Internal CFD Applications

OBJECTIVES

- Conduct full-scaled measurement and validate CFD of internal air movement.
- Predict the potential of the natural ventilation for classrooms at University of Hawaii at Manoa campus.

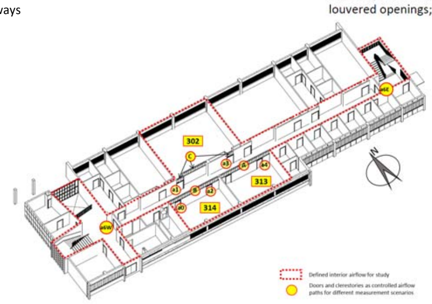


Internal CFD Applications

DEPLOYMENT OF INSTRUMENT AND TEST SCOPE

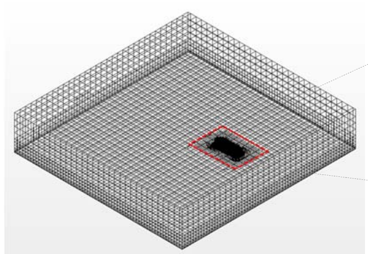
Installation of anemometers at the middle of the door ways

Installing the temporary tarp to seal sections of the louvered openings; seen from classroom 302

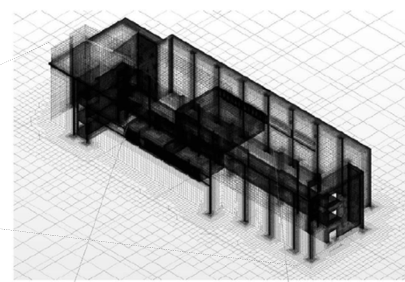


Internal CFD Applications

CARRYING OUT CFD SIMULATIONS

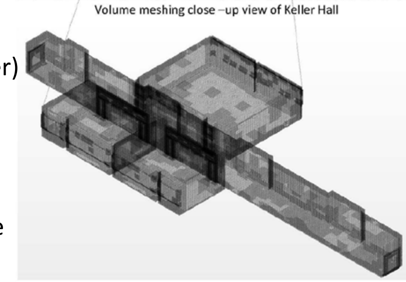


Volume cell meshing of the computational domain

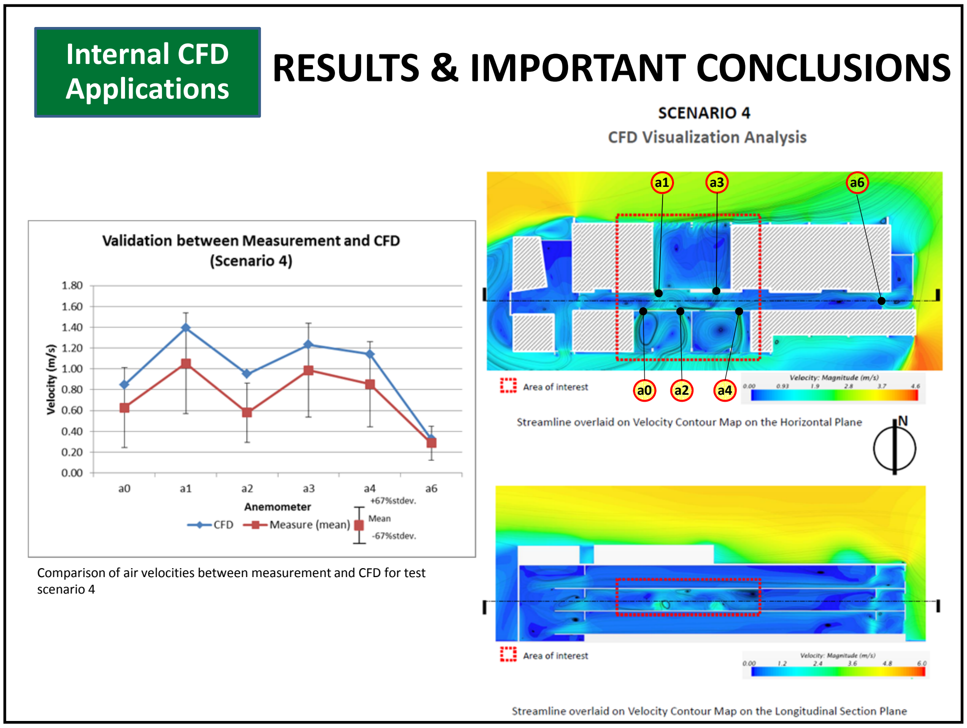


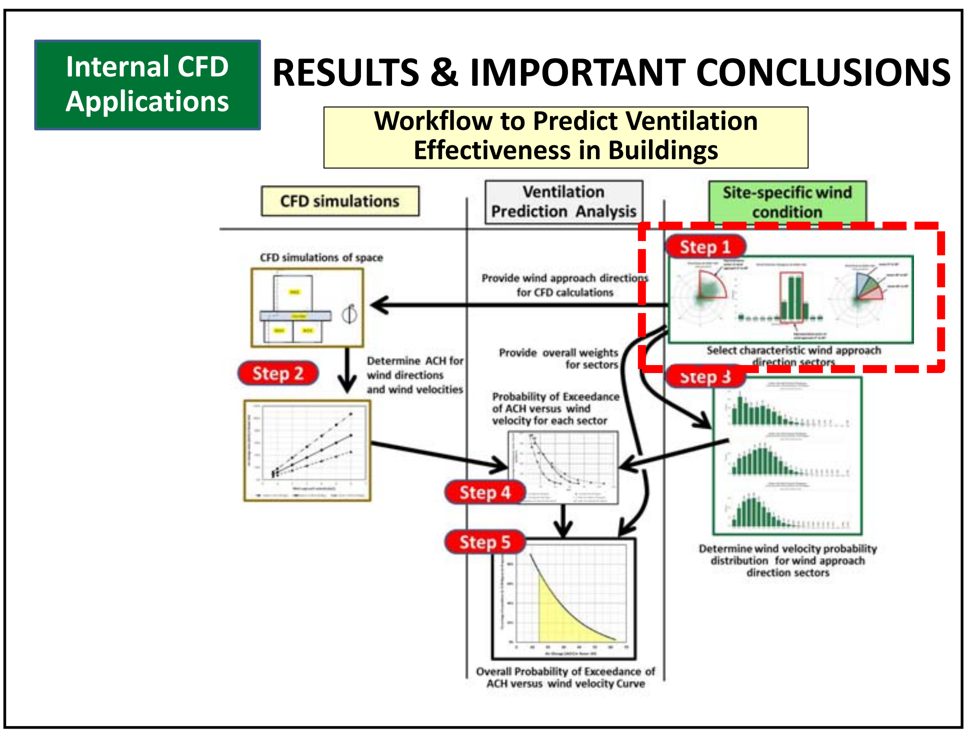
Volume meshing close –up view of Keller Hall

- Rhino3D (CAD) -> Mesher (Remesher/Wrapper)
- Coupled computational domain
- Trimmer (~20 million cells)
- Steady state
- RANS Realizable k-epsilon turbulence model
- Two layer high y+ wall treatment with variable sizes for near-wall cells



Volume meshing close-up view of the area of interest

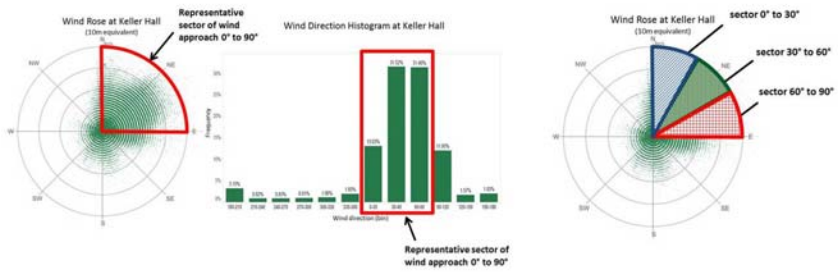

Internal CFD Applications
RESULTS & IMPORTANT CONCLUSIONS
SCENARIO 4
CFD Visualization Analysis
Validation between Measurement and CFD (Scenario 4)
Velocity (m/s)
Anemometer
CFD
Measure (mean)
+67%stdev.
Mean
-67%stdev.
Comparison of air velocities between measurement and CFD for test scenario 4
a0
a1
a2
a3
a4
a6
Area of interest
Streamline overlaid on Velocity Contour Map on the Horizontal Plane
N
Area of interest
Streamline overlaid on Velocity Contour Map on the Longitudinal Section Plane


Internal CFD Applications
RESULTS & IMPORTANT CONCLUSIONS
Workflow to Predict Ventilation Effectiveness in Buildings
CFD simulations
Ventilation Prediction Analysis
Site-specific wind condition
Step 1
CFD simulations of space
Provide wind approach directions for CFD calculations
Select characteristic wind approach direction sectors
Step 2
Determine ACH for wind directions and wind velocities
Provide overall weights for sectors
Step 3
Probability of Exceedance of ACH versus wind velocity for each sector
Step 4
Step 5
Determine wind velocity probability distribution for wind approach direction sectors
Overall Probability of Exceedance of ACH versus wind velocity Curve

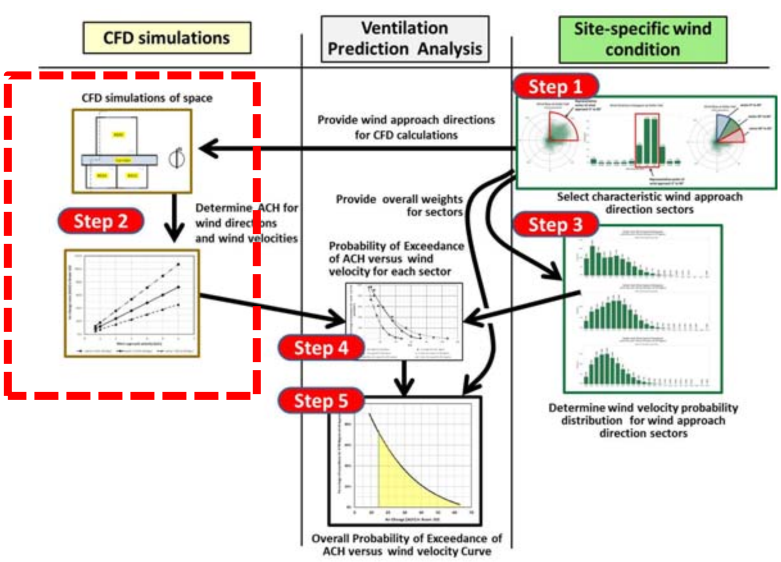
Workflow to Predict Ventilation Effectiveness in Buildings

Step 1 **Site-specific wind conditions:**

- Obtain site-specific wind record and select representative sector **(1)**
- Determine the representative wind approach direction probability and assign overall weights **(2)**
- Create representative wind approach direction sectors **(3)**



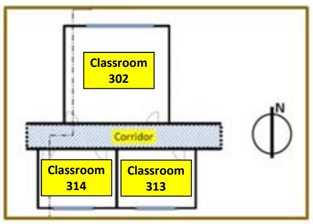
(1) **(2)** **(3)**

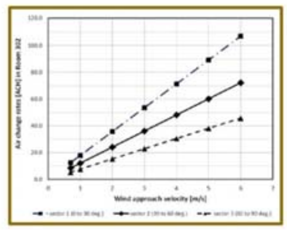


Workflow to Predict Ventilation Effectiveness in Buildings

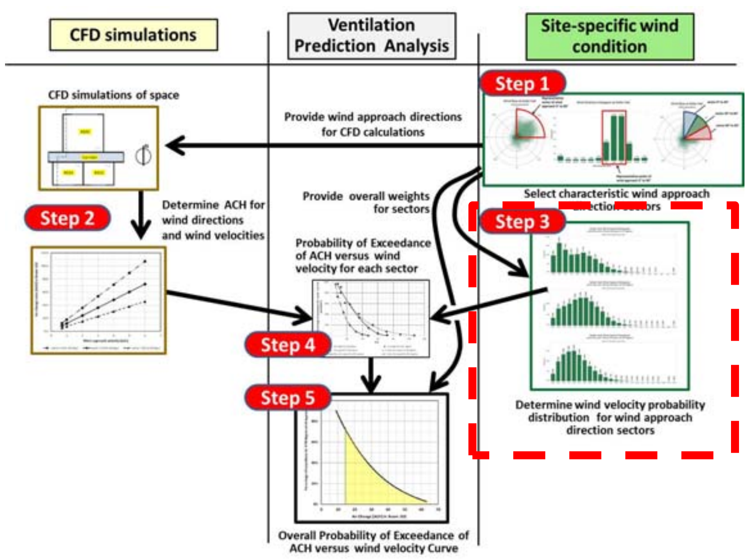
Step 2 **CFD simulation:**

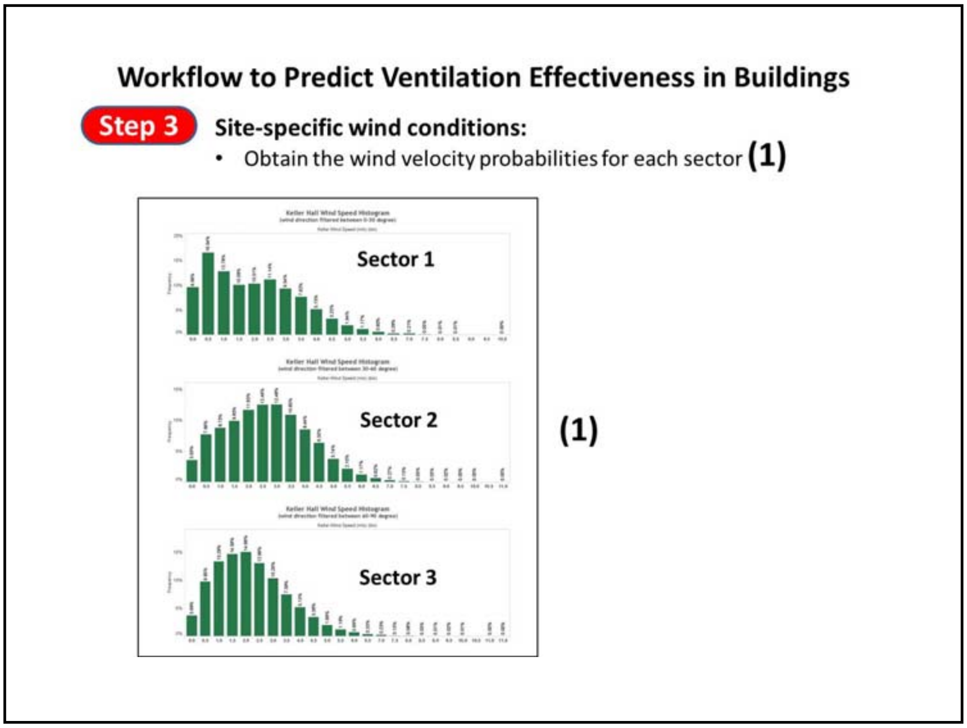
- For representative wind approach directions of sector determine air flow in space site-specific wind record **(1)**
- For unit different wind approach velocities (e.g. 1 , 2, 3 and 4 m/sec) determine air change per hour (ACH) for each sector
- Determine trend line of ACH versus wind velocity for each sector **(2)**

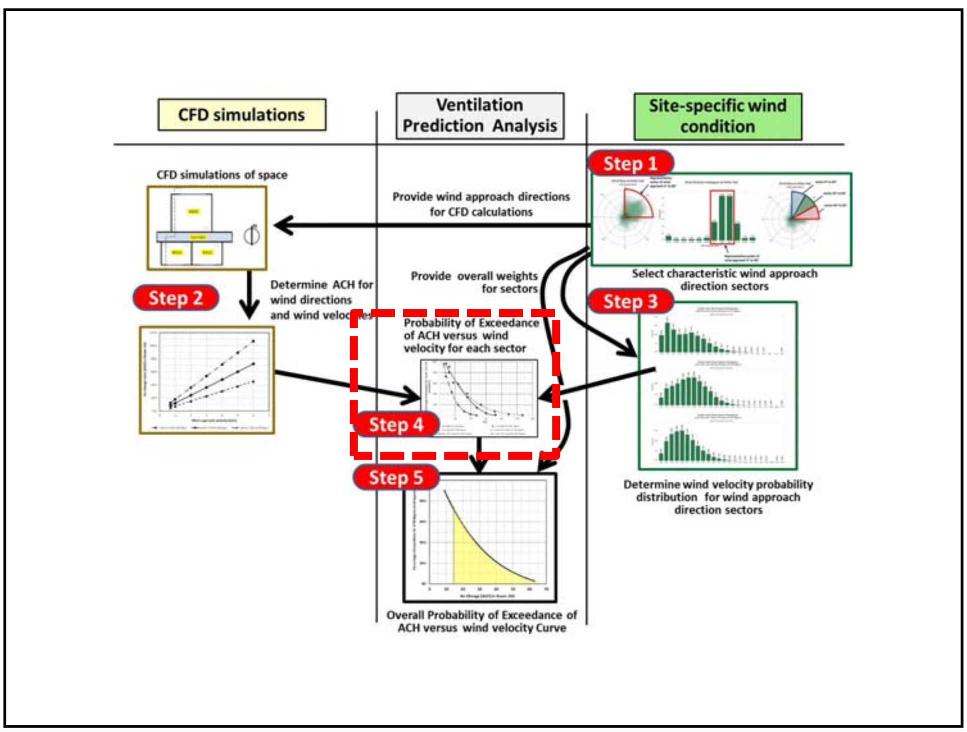




(1) **(2)**



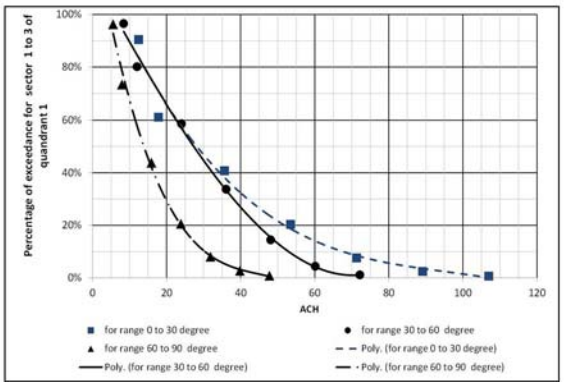

Workflow to Predict Ventilation Effectiveness in Buildings
Step 3
Site-specific wind conditions:
• Obtain the wind velocity probabilities for each sector (1)
Sector 1
Sector 2
Sector 3
(1)


CFD simulations
Ventilation Prediction Analysis
Site-specific wind condition
Step 1
CFD simulations of space
Provide wind approach directions for CFD calculations
Select characteristic wind approach direction sectors
Step 2
Determine ACH for wind directions and wind velocities
Provide overall weights for sectors
Step 3
Probability of Exceedance of ACH versus wind velocity for each sector
Step 4
Step 5
Determine wind velocity probability distribution for wind approach direction sectors
Overall Probability of Exceedance of ACH versus wind velocity Curve

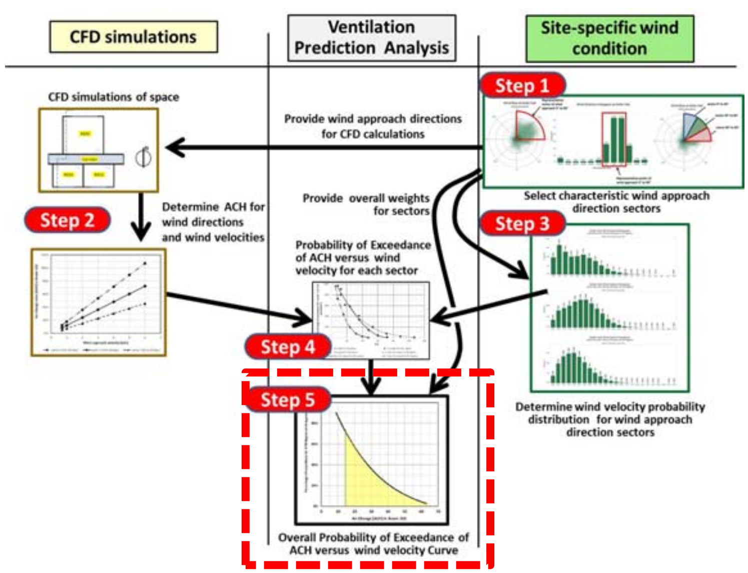
Workflow to Predict Ventilation Effectiveness in Buildings

Step 4 **Ventilation Prediction Analysis:**

- For each sector (e.g. 1 trough 3) calculated the ACH versus wind approach velocity relationship **(1)**



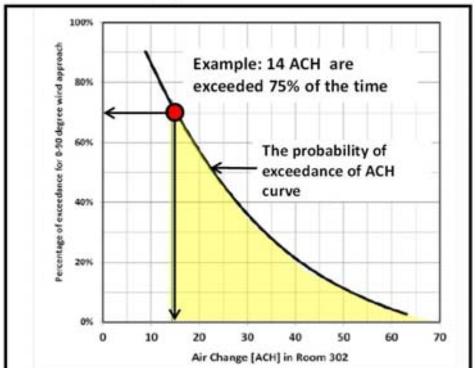
(1)



Workflow to Predict Ventilation Effectiveness in Buildings

Step 5 **Ventilation Prediction Analysis:**

- Apply overall weights for each sector to the ACH versus wind approach velocity relationship s to get overall ventilation performance prediction on the basis of probability of exceedance **(1)**



The probability of exceedance of ACH for room 302

(1)

THANK YOU

Q & A