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

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

Task 7

Prepared For
Hawaii Natural Energy Institute

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

December 2013

HNEI
Hawaiʻi Natural Energy Institute
University of Hawaiʻi at Mānoa

Project Phase 1- 7.A

ESTABLISHING EXTERNAL CFD WORKFLOW

December 4, 2013

2.2

FINAL REPORT

Prepared by:

Manfred J. Zapka, PhD, PE (Editor)
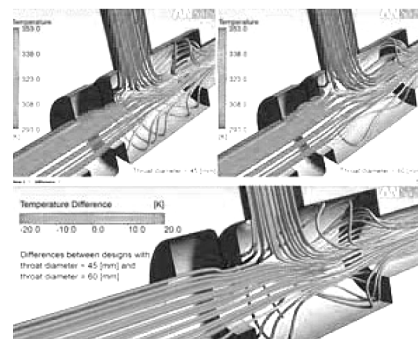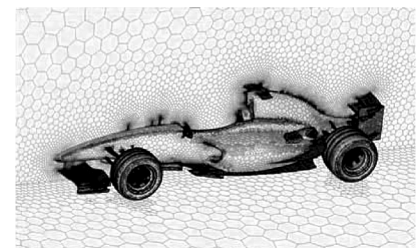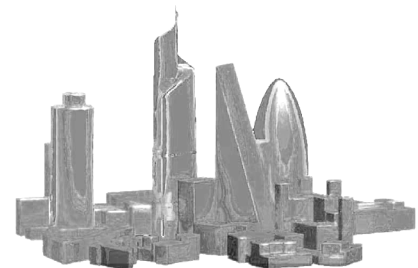
Sanphawat Jatupatwarangkul, D.Arch
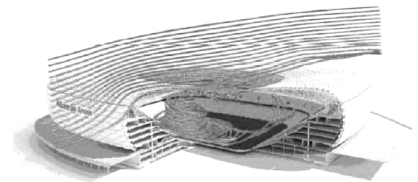
Tuan Tran, D.Arch

A. James Maskrey, MEP, MBA, Project Manager

Stephen Meder, D.Arch, Director

ERDL

HNEI

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

Project Phase 1 – 7.A

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

Project Deliverable No. 2.2: FINAL Report

Establishing External CFD Workflow for ERDL-HNEI

Prepared for Hawaii Natural Energy Institute

in support of

Contract #N000-14-13-1-0463

December 4, 2013

Prepared by:

Manfred J. Zapka, PhD, PE (Editor) (*)
Sanphawat Jatupatwarangkul, D.Arch (**)
Tuan Tran, D.Arch (**)
A. James Maskrey, MEP, MBA, Project Manager (***)
Stephen Meder, D.Arch, Director (**)

(*)   Sustainable Design & Consulting LLC
(**)   Environmental Research and Design Laboratory (ERDL), School of Architecture, University of Hawaii
(***)   Hawaii Natural Energy Institute

EXECUTIVE SUMMARY

This report is the FINAL report on establishing the external CFD workflow process for and on selecting the CFD software products to be used in the subsequent ERDL-HNEI CFD research project.

The first of the two main objectives of the report is to describe generic as well as software-specific external CFD processes that will be used to simulate air movement and pressure distribution adjacent to buildings in Hawaii's climate. The second main objective is the selection of one or more CFD software products that offer a favorable workflow process to the ERDL-HNEOI CFD research team.

Other objectives are:

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

For the selection of the preferred CFD software products out of an initial group of seven candidate CFD software products, the ERDL-HNEI CFD research team has used a quantitative comparison and ranking procedure. This quantitative evaluation uses a two-tiered ranking procedure. In the first step a generic ranking framework is established that consists of a range of criteria which represent important considerations and expectations of the CFD team towards a generic CFD workflow process.

In the generic framework each criterion has its own value proposition, with different individual overall weights. The sum of all individual overall weights is 100%. In the second step of the ranking framework the individual software products are assessed to what degree they are consistent with the value proposition. For the determination to what degree the software product is consistent with the value proposition a discrete rank of 1 through 5 is used.

While the original project scope intended to compare three CFD products, the CFD team made the decision to compare seven candidate CFD software products. This initial selection of seven, instead of three candidate CFD software products, was based on conclusions reached in the literature review (Project deliverable No.1), existing CFD software licenses available at ERDL and internal discussions of the CFD research team.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                      School of Architecture, University of Hawaii

December 4, 2013   Executive Summary - A

The work on this report was conducted in two phases. The first phase included a ranking of all seven original candidate CFD software products, which was solely based on the review of documentation available through the CFD software vendors. The result of the first phase was presented in the DRAFT report. During the second phase six of the original seven candidate CFD software products were actually tested in benchmarking to gain first-hand experience of specifics of the software workflow and the quality of support by the vendor. The experiences of the second phase resulted in a modification of the initial ranking which concluded in the selection of three CFD software products.

The selected CFD software products are:

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

The outcome of the final ranking is the basis for the selection of the three CFD software products. These three CFD software programs will be used in subsequent CFD analysis on this project. The final ranking reflects not only specific advantages in the software specific workflow and user interface features, but also good experience with the vendor, which includes favorable pricing terms.

The CFD team expects that the subsequent CFD research work will be effectively executed with the support of each of the selected software programs. Each offers specific features that will support the CFD team to simulate externals and internal air movement and thermal properties.

Disclaimer: It should be noted that the selection of the CFD software product only reflects the CFD software performance to very specific ranking criteria. The selection of the preferred CFD software products is task oriented and the selection of the three CFD software products is in no way intended to be a general software review and/or product endorsement by ERDL/HNEI.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                          School of Architecture, University of Hawaii

December 4, 2013   Executive Summary - B

TABLE OF CONTENTS:

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                         School of Architecture, University of Hawaii

December 4, 2013   Page i

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page ii

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
               School of Architecture, University of Hawaii

December 4, 2013   Page iii

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                              School of Architecture, University of Hawaii

December 4, 2013   Page iv

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page v

Part 1 – Description of an External Generic Workflow

This section describes important generic steps and phases of the external CFD work flow procedures that have been identified by the ERDL-HNEI CFD team.

1.1 Assigning Project Phases and Project Team Responsibilities

The proposed workflow for external CFD applications is illustrated in Figure A. The elements of the generic workflow are depicted in Figure A. Figure A suggests two categories of process steps in the CFD workflow:

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

There are several different team functions for different phases in the CFD workflow. The responsibilities of the team members are described in this section.

Process step (A): At the start of the CFD workflow the first step (A) is to set objectives for the CFD investigation, for example to determine the wind speed around building, the dispersion of exhaust from buildings or the pressure distribution on the building envelope. Each objective requires different physical settings and calculation requirements of the CFD analysis. The objectives also have to be supported by the software code used for the CFD analysis. At the conclusion of the process step (A) all relevant calculation parameters and requirements have determined.

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

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 1
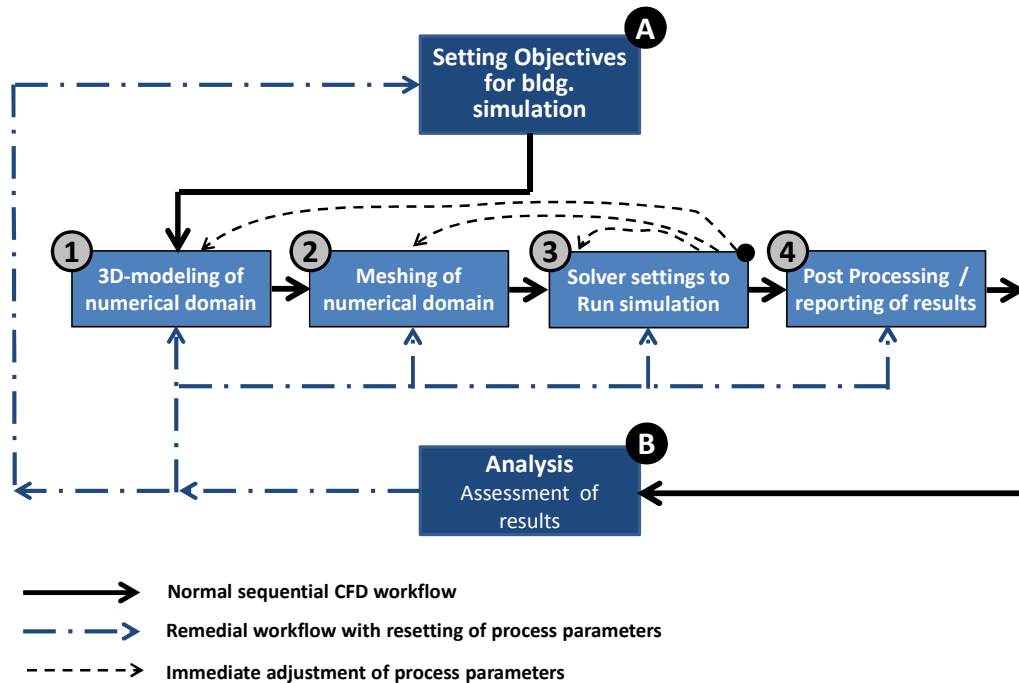
Figure 1.1:   Generic workflow for external CFD showing process steps involved

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

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

Process step (1)- 3D-Modelling: The initial step of the CFD workflow is the creation of a suitable geometry of the computational domain. The computational domain represents the volume of fluid

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                      School of Architecture, University of Hawaii

December 4, 2013   Page 2

which is subjected to the designated physics and boundary conditions. The fluid body is confined by the outer boundary of the virtual wind tunnel as well as inner domain boundaries (e.g. solid structures inside the confinement of the outer domain boundaries). Typically the solid structures represent objects around which the fluid movement or effects of the fluid movement has to be determined. The geometry of the solid structures can either be created inside the CFD software or imported from a third party 3D-CAD program. The advantage of using a third party 3D-CAD program is that such programs typically allow the creation of a wide range of solid objects, which are composed of either straight or curved outer surfaces. Importing the 3D-geometries into the CFD program requires CAD interfaces that translate the CAD geometries into the format that can be used in the CFD program.

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

Process step (2) - Meshing: The second step of the CFD calculation procedure is the creation a suitable grid, or mesh, which represents the digitization of the numerical solution. For most of the high performance CFD software codes, the meshing occurs in two phases. The first phase is the preparation of an "impermeable" surface mesh where the surface has to be a continuous mesh that surrounds the solid object. The mesh cannot "leak"; in other words, the entire mesh has to be a continuous assembly of individual 3D-faces or other 3D-solid geometry definitions, without discontinuities which would negatively affect the accuracy or convergence of the solution. The surface mesh basically represents the outside surface of the objects around which the fluid motion and other flow induced effects are analyzed. The second phase of meshing is the creation of the volume mesh which encompasses the extent of the fluid that is analyzed in the computational domain. The volume of the fluid is discretized into a large number of cells for which the governing equations are solved.

Responsibilities of project team members for process step (2) are shared between the principal investigator and the CFD software operator. The CFD software operator is responsible of placing the 3D-geometry of the buildings and structures into the viral wind tunnel, which is the outer extent of the computational domain. The CFD operator has to create the surface and the volume mesh required for the simulation. The principal investigator needs to review of software

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 3

analytical checks to make sure that the surface is completely wrapped and the volume mesh is established in accordance to the CFD solver requirements (e.g. use the right type of cell geometries).

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

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

 Process step (4) – Post Processing:

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

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                          School of Architecture, University of Hawaii

December 4, 2013   Page 4

typically represent many thousands or even millions of data points, a convenient visual representation of the data facilitates interpreting the results of the CFD simulation.

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

1.2 Required Proficiency of CFD Team Members

This section describes the required proficiency for members of the CFD team to carry out external CFD simulations. The different functions could potentially be carried out by the same person, such as for smaller CFD investigations, where the principle investigator might also perform the software technical aspects of the simulation. For larger projects, however, where CFD investigations are undertaken to provide input for solving design or compliance projects, the different team functions will most likely be carried out by different team members.

Table 1-1 shows function, responsibility and required proficiency of members of the CFD project team.

1.3 CFD Best Practice Guidelines to be Followed

Several versions of CFD Best Practice Guidelines (BPG) are available to support the CFD project team to carry out effective and valid external CFD simulations. The project manager in conjunction with the principal investigator should review what BPG is most beneficial for the project. The following lists four exemplary BPGs for external CFD analysis, which have been developed by different authors. While the main approaches and messages of each of the four listed BPGs are similar, the specific BPGs can give valuable advice for each application involving external CFD applications.

- Franke,J. et al (2007), "Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment", "COST", May 2007
- Franke,J, Hellsten, A., Schluezen, H. and Carissimo,B. (2010) "Best Practice Guideline for the CFD simulation of flows in the urban environment : an outcome of COST 732", The Fifth International Symposium on Computational Wind Engineering
- Blocken B, Janssen WD, van Hooff T. (2012) "CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus", Environmental Modelling & Software 30: 15-34.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                 School of Architecture, University of Hawaii

December 4, 2013   Page 5

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

BPGs are continuously updated and it is best for project manager and principal investigator to consult BPGs and select the one best suitable for the specific external CFD application.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 6

Table 1-1 Function, responsibility and required proficiency of members of the CFD project team

Role in project team	Responsibility in CFD workflow	Required Proficiency	Training needs
Project manager	Responsible to translate projects needs to the type of CFD simulations that can solve the problems at hand. Responsible to use the CFD results to arrive at decisions that affect the design or compliance project.	Needs a high proficiency in overseeing project, setting project objectives and identifying how CFD can be used to satisfy project objectives	Has to be highly trained in managing complex projects that require considerable technology (e.g. projects that are in need of CFD simulations usually represent interdisciplinary solutions and technology integrations)
Principal investigator	Responsible for the technical and scientific approach used in the CFD investigation. Supervises the CAD and CFD operators. Assists and consults the project manager in decisions.	Needs a high proficiency in building physics and a comprehensive and detailed understanding of all aspects of the CFD workflow. Needs proficiency to supervise the CFD team to produce reliable CFD results and advise the project manager in realizing the project objectives.	Has to be highly trained in the scientific, technical and software application details of CFD simulations. Has to be trained in building physics and how CFD results can be used to serve specific problems in architecture and urban design.
CAD operator	Responsible for creating the 3D-CAD geometry from plans or building specifications, either within the CFD software or in generic 3D-solid CAD	Needs high technical proficiency in manipulating CAD geometry on the basis of plans and specifications. Needs to understand what level of	Has to be trained in operating 3D-CAD systems and create 3D drawings from plans.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 7

Table 1-1 Function, responsibility and required proficiency of members of the CFD project team

Role in project team	Responsibility in CFD workflow	Required Proficiency	Training needs
	software and importing the geometry into the CFD software. Works under the supervision of the principal investigator.	detail should be included. Needs good application knowledge of the CAD application used.	
CFD operator - meshing	Responsible for validating 3D-geomerties for use in surface mesh and for creating volume mesh in accordance with mesh specifications. Works under the supervision of the principal investigator.	Needs proficiency to understand the importance of geometric representation of structures in the numeric domain. Needs proficiency to create effective meshes and select most appropriate cell geometries for the mesh. Needs good application knowledge of the CFD software.	Has to be trained to validate 3D-geometries or to import 3D-geometries from external CAD applications. Has to be trained in selecting and creating effective surface and volume meshes for a variety of CFD applications. Has to be trained in the CFD software product.
CFD operator - solver	Responsible to define all physical properties and settings for the fluid, boundary conditions and turbulence. Responsible to run the	Needs high proficiency to understand the multiple settings and physical properties that have to be selected to run effective CFD	Has to be trained in the solver settings of the CFD simulation. Has to be trained to understand the physical properties and settings and

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 8

Table 1-1 Function, responsibility and required proficiency of members of the CFD project team

Role in project team	Responsibility in CFD workflow	Required Proficiency	Training needs
	simulation and monitor convergence.	calculations. Needs good application knowledge of the CFD software.	other required setting to run the simulation. Has to be trained in the CFD software product.
CFD operator – post processing	Responsible to create the type of CFD visualization, which is requested and required by the principal investigator and the project manager.	Needs proficiency in creating informative graphical representations of the CFD results. Needs flexible and good communication skills to serve the project teams with a variety of ways to show the CFD results.	Has to be trained in operating the post processor of the CFD software. Should be trained in other graphics software products that can integrate CFD results into high quality visualization.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 9

The recurring topics presented in the cited BPGs, which are important to a generic external CFD work process, are presented below:

Identifying generic errors and uncertainties:

Simplification of physical complexity: CFD analysis should use simplification in the mathematical description of the problem. Some wind movement around buildings and in the urban environment are too complex, both in spatial and time domains, to be precisely modeled. And if the wind phenomenon can be expressed with very complicated math, the required computational resources would be prohibitively expensive for most technical CFD applications. These simplifications are used to lower the complexity to an affordable level while still obtaining useful CFD results.

Physical boundary conditions: The computational domain typically contains only a part of the urban area. Therefore the choice of the position of the boundaries of the computational domain influences the results, which adds to the uncertainty and even errors of the simulation results.

Geometric boundary conditions: The appropriately defined computational domain for the CFD analysis requires knowledge of the geometric details of the urban or industrial environment. This information is often not available with sufficient accuracy. The missing information therefore adds to the uncertainty of the simulation results. Furthermore, errors and uncertainties can result from the simplification of the geometrical complexity used in the CFD analysis experiment. To reduce the computational costs, geometric details are often omitted.

Identifying numerical errors and uncertainties:

Spatial and temporal discretization: The spatial and temporal discretization is described as probably the most critical sources of numerical error. These errors are a result of the difference between the exact solution of the basic system of partial differential equations and the numerical solution obtained with finite discretization in space and time. With regard to the space discretization, it is not only the degree of resolution that is important but also the distribution of the grid points. Therefore the mesh used to discretize the space is of great importance for the accuracy of the results.

Iterative convergence: A non-linear algebraic system that is formed by discretization of the partial differential equations is solved with an iterative method or by time integration towards a steady state. If the iteration or time integration is stopped too early, an iterative convergence error is introduced which is the difference between this intermediate solution and the exact solution of the algebraic system of equations. Assessment of the iterative convergence is based on the residuals.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 10

BPG Recommendations to reduce errors and uncertainties:

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

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

- Reynolds Averaged Navier-Stokes (RANS) equations are adequate representation of constant wind flow conditions. Interestingly, no best practice advice for the choice of the turbulence models is given here. Rather a validation strategy is proposed to evaluate the performance of the different turbulence models.
- Large Eddy Simulation (LES) uses partially filtered Navier-Stokes equations with the filtering being performed over a small volume.
- The Detached Eddy Simulation (DES) is a widely known hybrid modeling strategy.

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

Selection of the computational domain: The size of the entire computational domain in the vertical, lateral and flow directions depends on the area to be represented and on the boundary conditions that will be used. The extent of the built area (e.g. buildings, structures or topography) that is represented in the computational domain depends on the influence of the features on the region of interest. The BPGs provide recommended extensions for the vertical, lateral and longitudinal extension of the domain

Selection of boundary conditions: The boundary conditions represent the influence of the surroundings that have been cut off by the computational domain. Their proper choice is very important since the solution depends on the appropriate choices of the boundary conditions. The following boundary conditions have to be identified:

- Inflow boundary conditions, where the velocity and turbulence profile have to be defined.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 11

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

Selection of the computational grid: The grid (or mesh) is created in such a manner that it does not introduce errors that are too large. This means that the resolution of the grid should be fine enough to capture the important physical phenomena like shear layers and vortices with sufficient resolution. Also the quality of the grid should be high.

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

1.4 Special Considerations for External CFD applications for Hawaii Climate

In order to create a more direct value of CFD, wind pattern predictions around buildings and through neighborhoods in Hawaii are estimated, which then can be used for design or compliance decisions. The generic workflow might need adjustments to accommodate specific climatic conditions that prevail in Hawaii. This is specifically true when physical properties induced by wind movement around buildings are concerned, such as in the case of the building of wind induced pressures on buildings.

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

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

- Assumptions of the atmospheric boundary layer (ABL) conditions and approaches to model its effects are important consideration for the CFD calculations. The atmospheric boundary layer

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 12

comprises the lower layers of the atmosphere, within which wind flows are affected by considerable shear stresses and induced eddies. High temperature gradients resulting from high solar incidents can produce localized plumes of hot air rising under buoyancy forces. Hot air rising from the heated ground influences the wind velocity profile and therefore might have to be included in the boundary conditions. These kinds of thermally induced wind profiles are typically not being modeled in wind tunnel tests since they include complicated physical phenomena and equally sophisticated experimental challenges. There are certain CFD simulation options that can use thermal equations to enhance the solution in the presence of thermal occurrences. While some CFD software products can model thermally induced fluid motion for external CFD applications, other CFD products restrict their simulation capabilities to isothermal numerical approaches.

- Influence of <u>thermal plumes generated by hot air rising around buildings</u> can affect wind movement and the resulting pressure distribution around buildings. Thermally induced external plumes around buildings are not uniformly distributed around the building, but change with the solar gain at the envelope and the adjacent ground. The CFD software needs capabilities to create boundary conditions that can appropriately model localized buoyancy induced air movement adjacent to the exterior building envelope. In addition to generic wind movement around buildings that can be affected by thermally induced phenomena, wind induced pressures on the buildings can also be affected. Since pressure differentials across the building envelope are important driving forces for natural ventilation, even small pressure fluctuations as a result of thermally induced air plumes might appreciably influence the ventilation effectiveness.

- <u>Heat islands around the buildings affect pedestrian comfort</u>. Typically pedestrian comfort is negatively affected by higher wind velocities. It should be noted though that much of the wind comfort standards have been defined for moderate climates where effects of heat islands are not as prevalent as in Hawaii's climate. In tropical climates, wind around buildings can improve pedestrian comfort, since heat is expelled from the area and increased draft over the human body positively affects the sensation of comfort. In addition, radiant heat from heated surfaces on buildings or adjacent ground can increase the heat gain and level of discomfort for pedestrians.

- <u>Humidity</u> is an important aspect in the effectiveness of building occupant comfort. For external comfort considerations, the effect of humidity might be less relevant than for internal building applications. An important issue could nevertheless be the adsorption and desorption of humidity at external building surfaces, as well as at internal surfaces or in building openings that serve natural ventilation systems.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 13

Part 2 – Framework for Ranking Candidate CFD Software Products

This section presents a framework for a quantitative ranking of candidate CFD software products.

2.1 Description of Quantitative Assessment

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

The procedure of ranking the candidate software products and assessing the quantitative ranking for the software for the particular criteria is explained in the Table 2-1. Table 2-1 only shows one criteria group (Ease of use /GUI) to explain the basic steps of the two-tiered ranking process.

Table 2-1   Example for Assessing the Quantitative Ranking for Criteria

Ranking matrix for CFD software products for external CFD

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

The ranking criteria is shown below:

ranking criteria

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

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 14

Table 2-1 shows a highlighted (yellow) row with column numbers. The significance of the columns in Table 2-1 is described below.

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

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

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

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

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

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

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

In our example, the criterion 3.1.1. has the following overall weight:

50% (Column C) * 15% (Column D) * 60% (Column F) = 4.5 % in overall weight.

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

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

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                       School of Architecture, University of Hawaii

December 4, 2013   Page 15

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

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

2.2 Description of Ranking Criteria

Table 2.2 lists the value proposition for all ranking criteria which are used for the ranking of the seven software products.

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
1	**Value for the CFD software product to the architecture and building market**	
1.1	Benefit for external CFD applications to architecture	The benefit of the software is reflected in its direct usefulness in external CFD applications. This means that the software package is providing good results for external CFD predictions where the modeling of the atmospheric boundary layer is of importance. The benefit of the software product is that it can add immediate value to the prediction of external CFD simulation and therefore provide valuable design support for external wind phenomena. A high ranking would imply that the software product has the required complexity and numerical capability to provide a high level of confidence in predicting external wind phenomena, thus providing reliable predictions to improve design of buildings and neighborhoods. A low ranking would imply that the software product is more applicable to

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 16

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		internal CFD applications and lacks the sophistication to reliably predict external wind phenomena.
1.2	Previous applications in architecture / urban design	The benefit of the software product is reflected in its current use in the architecture or building industrial market. With a high ranking, the software has a track record in successful applications in not only academic institutions but also in the commercial architecture and building industry. There are already a significant number of professionals being trained or proficient in the use of the software. For companies, the benefit is while CFD might be new for the company, an existing workforce can readily be employed to use the CFD on a professional level. A high ranking would imply that the software is in use in a significant number of architecture and building companies. A low ranking would imply that the software is in use in only a very small number of companies or represents a very small niche market in architecture and wind engineering.
1.3	Previous mentioning of software product in scientific or applied professional articles	The benefit of the software is reflected in the existing body of knowledge in the open domain about application of the CFD software; either for specific practical applications or for specific components or parameter settings of the software. This implies that the software is in wider use for different sections of the building industry or in the academic field. A high ranking would imply that the software product is being mentioned to a larger extent in scientific papers or professional articles. A low ranking would imply that the software has not been mentioned to a significant extent.
1.4	Using graphics output for architectural presentations	The value of the software is established by the fact that results of the CFD are readily usable for illustration purposes. This means that, in the most advantageous application setting, the CFD software product would provide

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 17

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		the input for external graphics and ray tracing renderings. The ranking represents the ease of using the integrated workflow for illustrating the wind movement around building and/or through the buildings with advanced graphics. A high ranking would imply that the software product can be easily used to integrate airflow patterns with 3D-renderings of the building (and appurtenances). A low ranking would imply that the software product would require a complicated workflow to couple visual representations of airflow movements in combination with high quality external renderings of buildings.
2	**Support by vendor / expert network**	
2.1	Technical support available	The value of the software product is reflected in a high quality level of support by the software company. A high ranking of the technical support refers to the responsiveness of the technical support to assist the user. A high ranking would imply being able to speak to technical support and get immediate or at least timely responses and suggestions to the problems. A low ranking would imply that technical support can only be accessed through online means, e.g. through email and responses would be available after one or two days. Average technical support would be that the user can chat with the support team in a more or less real time talk mode.
2.2	Online tutorials available on demand	The value of the software product is reflected in the wide selection of online tutorials that can help the user to learn the software workflow and understand a certain depth of application knowledge. A high ranking would suggest that the software package offers a significant cope and number of CFD videos online (on demand) tutorials available through their web site. A high ranking would also refer to a

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 18

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		large number of YouTube videos available for certain applications. A low ranking would infer that the software product has only a few or no videos available online.
2.3	Online classes available (low entry barriers)	The value of the software product is reflected in the availability of online courses for general product knowledge or more specific (advanced) product knowledge. The value of the software would furthermore be represented by low entry barriers (i.e. low costs) to take the online course. A high ranking would imply that a large number of on-demand videos are available for viewing. A low ranking would imply that there are no or only very few online course available. A low ranking would further suggest that taking courses would represent high entry barriers (i.e. high costs) for the ERDL-HNEI team.
2.4	Network of CFD professionals in architecture field Ease of working with academic support	The value of the software is reflected in the availability of an existing network of users in the building industry as well as in academia who are actively engaging in the sharing of application knowledge. A high ranking implies that the software products support its own network of professional CFD practitioners or educators so they can share knowledge and engage in discussion and forums. A high ranking would also imply that the software product organizes official events, either in person or online where CFD practitioners who use the specific CFD software can exchange knowledge. A low ranking implies that the software product would not actively support a network or said events
2.5	Ease of working with academic support	The value of the software product is reflected in the level of support afforded to the academic community and/or user. A high ranking would imply that the software products actively support academic institutions, including faculty as well as students, to acquire the means of using the software

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 19

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		for teaching and basic research. A high ranking would also imply that the software company is actively engaged to lower entry barriers to using the software at academic institutions. A high ranking would also suggest that the software product provides students with the opportunity to freely or at a much discounted costs use the software for their academic endeavors. A low ranking would suggest that the software product offers no academic program including no incentives for academic community to use the software. An average ranking would imply that the software product has a basic academic program but does not actively support academic users or make no active effort to lower entry barriers for students and faculty.
2.6	User guide provides the efficient information	The value of the software is reflected in the clarity and comprehensiveness of the user guide and other product documentation in regard to efficient information transfer. The value is also based on whether the user manual mentions generic CFD knowledge that goes beyond the immediate need to operate the software interface, such as providing basic information about turbulence modeling. A high ranking would imply that the user guide offers a wealth of application information (i.e. how to use software as well as how to interpret the results of the software calculations). A high ranking would also imply that topics in the user guide are directly hyper-linked to specific task and GUI. A low ranking would imply that the user guide has insufficient clarity, lacks comprehensiveness and is cumbersome to access online.

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 20

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
3	**Workflow (the work flow criteria group includes several sub-criteria groups)**	
3.1	**Ease of use / GUI**	
3.1.1	GUI is easy to use for trained operator	Most of the advanced CFD application workflow is complex and unfamiliar in regard to process and key stroke manipulation. Graphic user interface (GUI) benefits the user with an intuitive and easy-to-use user interface. A well designed GUI is very important for the effective manipulation of both external and internal CFD analyses. A high ranking would imply that the selected application has efficiently organized all key elements for the CFD workflow. The well designed GUI shows the workflow process steps in obvious and sequential steps. An effective GUI is intuitive and can be learned quickly. A low ranking would imply that the software does not reliably guide users through the simulation workflow. A GUI that is lacking clarity and user-friendliness could possibly make it difficult to indicate what the next steps are.
3.1.2	User guide and online help is available for each command	There are many technical, physics, and definition for CFD simulation, therefore user guides and online help are the major significant support functions for the user. A high ranking implies a user guide that is easily available online and has hyper-link accessibility. Documentations and other resources are highly descriptive and informative and are also easy to find. A low ranking implies to the software only has cumbersome access to help and no or very limited online resources.
3.2	**Modeling / Import geometry**	
3.2.1	Create geometry inside CFD	Internal, or built-in CAD modeling tool inside the CFD software is important to allow user to create, edit and

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 21

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
	application	modify proper geometry before running the simulation. A high ranking implies how the durability and convenience of built-in CFD modeling tool. Especially for single building or multiple buildings, it is important that the creating and editing geometry tool is easy to modify the building shapes and forms. A low ranking implies the lack of efficient internal modeling capabilities for create, edit, and modify geometry.
3.2.2	Import geometry / repair surface before meshing	Users may prefer to use the third party 3D-modeling application for building and structures. Therefore, effective capabilities to import externally created 3D-CAD geometry into CFD applications lead to a high ranking. The high ranking also measures the availability of importing different model or geometry different formats into CFD applications. Imported CAD models should provide a well maintained geometric model without distortion and missing faces. Moreover, availability of repair surface, vertex, and edge functions are provided. A low ranking would imply that geometry from only very few CAD software products can be imported into the CFD software.
3.2.3	Support Revit Architecture	The majority of architecture students and professionals are using Revit for building design and architectural documentation. Therefore, the benefit of the CFD software is represented by its capability to easily allow import of Revit file formats (.rvt). A high ranking means that Revit files can easily be imported. Additional a high rank also indicates that the CFD software provides a seamless process to call the Revit program from within the CFD application. A low ranking would imply that lacking of support and file transition from the Revit model to the CFD software.
3.2.4	Support CADD format and	Inventor is one of the popular applications in the industrial

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 22

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
	Inventor	design and mechanical model study. A high ranking of the CFD software indicates that the CFD software can readily import CAD format or Inventor file formats (.ipt, .iam) or geometry. A low ranking would imply that lacking of support and file transition from and CAD format or Inventor to CFD application.
3.2.5	Support Rhino	Rhino is a growing model application for the architecture and design industry. A high ranking of the CFD software would imply that the CFD application can readily import Rhino file format (.stl, .iges, .iges, and etc) geometry. A low ranking would imply lack or absence of support for Rhino file format or any relevant to Rhino import/export file format to CFD application.
3.3	**Meshing**	
3.3.1	Surface meshers	An effective surface meshing tool is important to prepare a high quality surface mesh before a volume mesh can be generated. Depending on the initial quality of the geometry, there are several surface meshing tools available in CFD applications: - Surface Wrapper - Surface Remesher - Automatic Surface Repair The surface wrapper is used when the imported geometry is of poor quality. A surface remesher would be used to maintain the original features and improve surface meshes from the surface wrapper or improve the quality of imported tessellated geometry A high ranking would imply that the CFD software has a surface meshers tool. A low ranking would imply that the

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 23

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		CFD software lacks such effective surface meshing tools.
3.3.2	Maximum cells and 2D-3D cell volume	Cells associated with mesh construction represent an ordered collection of faces that define a closed volume in space. A high ranking would imply that the number of average mesh cells are 10 million cells or less. The recommendation for 3D cells is polyhedral shape, and 2D cell is polygon shape. A low ranking would imply that the CFD software has notable impairment sin calculation effectiveness with increasing cell number, already at a relative lower cell number level.
3.3.3	Boundary types	Specific boundaries types are required in CFD physics models to define the primary variables of the simulation (such as pressure, temperature, and velocity). Typical boundary types are the following: - Axis - Flow split outlet - Free-stream - Mass flow inlet - Pressure outlet - Stagnation inlet - Symmetry plane - Velocity inlet - Wall A high ranking would imply that CFD application provides various boundary types for physics model purposes. A low ranking implies only a limited number of boundary types
3.3.4	Structured and Unstructured meshing	This criterion assesses the availability of a wide range of meshing techniques from the given CFD application. The advanced levels of these techniques are based on meshing features such as their easiness to learn/use, time consumption for meshing process, their capability to handle

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 24

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		high levels of geometrical complexity and diversity, accuracy levels, and convergence time. These meshing techniques rank from less to more advanced as follows: 1. Structured meshing 2. Unstructured meshing (hexahedron) 3. Unstructured meshing (tetrahedron) 4. Unstructured meshing (polyhedron) 5. Hybrid meshing (structured + unstructured) The highest ranking level of this criterion will be applied for the CFD application that provide the most advanced meshing technique and vice versa.
3.3.5	Adaptive meshing capability	This criterion is used to assess the ability of the given CFD software to modify the existing mesh so as to accurately capture the fluid flow features. This can be achieved by improving the grid resolution of the flow features without having to increase the grid resolution for the entire computational domain. Other benefits are capabilities that allow for modeling complex and diverse geometries/buildings with high accuracy level without an excessive increase in computational effort.
3.4	**Physics Models**	
3.4.1	Solvers – Coupled / decoupled	This criterion is based on the capability of the given CFD software to support coupled domain modeling or only decouple domains. The coupled domain modeling is also called "full-domain" simulation which allows for solving the internal and external airflow simultaneously using appropriate governing equations. The benefit of coupled domain modeling lies in its representation of the continuum

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 25

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		of indoor and outdoor airflows in natural ventilation. A coupled domain can increase the accuracy of the ventilation prediction. Decoupled domain modeling is the conventional feature of most CFD applications, and therefore decoupled domains represent a low ranking.
3.4.2	Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)	This criterion is used to identify which numerical method for discretization approach is employed in the CFD software. Although FEM is flexible and capable to handle complex geometries, FVM has volume controlled elements that establishes flux properties in their numerical equations. Therefore FVM typically allows better simulation of incompressible flow (slow speed airflow). Furthermore, FVM is the center focus of improvement in many commercial CFD applications. Therefore, CFD software that uses FVM will have a higher ranking.
3.4.3	Steady state/Transient state	This criterion is used to check whether the given CFD software can be used for steady state or transient state simulations or both. The steady state simulation is used to simulate the airflow with constant properties while transient state simulation is used to simulate the airflow with varying properties throughout the simulation. The benefit of this criterion is that the CFD application capable of transient state simulation will allow for study of time-dependent airflows such as pollutant dispersion or humidity simulations.
3.4.4	Turbulence models	This criterion is used to assess capacity of the CFD software in terms of range of available turbulence models. The advanced turbulence models allow for modeling the effect of turbulence on airflow with higher level of accuracy, but commonly requires more computational resources. Turbulence models available in many CFD applications can

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 26

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		be simply categorized as below arranging from low to high for ranking: 1. Standard k-ε and variations 2. Standard k-ω and variations 3. DES (Detach Eddy Simulation) 4. LES (Large Eddy Simulation) The benefit of the software is represented in its range of turbulence models available for the CFD calculations. A high rank for the CFD software indicates not only the type of advanced turbulence models, but also the number of turbulence models that it can offer.
3.4.5	Materials	This criterion is used to assess the availability of built-in materials of a given CFD applications. The benefit of this criterion is that it makes the set-up for the CFD analysis easier and less time consuming if the software contains a list of ready material properties. The ranking is based on the combination of the following features: - Available built-in generic materials - Available AEC built-in materials - Customable materials - Distributed resistance materials The high rank implies that a given CFD application offers a wide range of the above features.
3.4.6	Customizable / special physics models	This criterion is used to assess the availability of customizable / special physics models of each CFD software. These models allow for more advanced/supplemental studying of airflow phenomenon in wind engineering field such as buoyance effect, pollution dispersion, humidity condensation, etc. The value of the software is represented

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 27

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		by the availability of a wide range of special models and therefore no need to require additional CFD code for specific simulations. These special models include CFD applications such as: - Heat transfer - Combustion and chemical reaction - Humidity absorption / dispersion - Customizable boundary condition profile (table, formulas…) A high rank implies that a given CFD application offers a wide range of special models and provides the opportunity to customize models.
3.5	**Post-Processing**	
3.5.1	Visualization	This criterion is used to assess multiple visualization features offered by a given CFD application for post-processing. These features consist of: - Surface distribution - Vector/streamline display - Implicit sections with gradient / contour display - Particle trace - Animation The high ranking suggests a wide range of visualization features offered by the CFD software, as well as the quality of those outputs and the ease to the user to produce them.
3.5.2	Quantitative analysis capabilities of components	This criterion is used to assess the availability of given CFD software in offering the quantitative analysis capabilities of certain nodes, line or grid within the computational domain. The benefit of this criterion is that it provides resultant data extracted for supplemental / customizable analyses. This

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 28

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		capability can be described as: - Resultant data at probes (points, line, grid) - Resultant data in table / chart form - Resultant data can be exportable into ASCII format The high ranking indicates a wide range of features offered by the CFD software, the quality of those outputs as well as the ease for the user to produce them.
3.5.3	Comparative analysis (multi-scenarios)	The benefit of this feature for post-processing is that it allows for effectively comparative analyses of multi-scenarios or testing different turbulence models in CFD simulation. This not only helps to reduce significant time for remodeling the whole model but also to allows for side-by-side comparison of the same variable at the same location of those computational domains.
3.6	**Customization / special functions**	
3.6.1	Pedestrian comfort	Pedestrian comfort standards associate comfort rating with wind velocity on the pedestrian and ground level. A high ranking would imply that the CFD software is able to determine atmosphere boundary layer or velocity profile due to terrain roughness (height z_0). A high ranking would imply that the CFD software has efficient meshing tools to define the wind movement in the lowest parts of the atmospheric boundary layer. Furthermore, a high ranking implies the ability of the CFD software define the velocity range for local wind comfort assessment and an determination if maximum of mean or gust equivalent mean (GEM). A low ranking would imply lacking the previously defined functions.
3.6.2	Natural Ventilation	Natural ventilation is significant passive building design

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 29

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		technology. A high ranking of the CFD software would imply that a workflow specifically for natural ventilation is available. This means that a high ranking infers a significant capability to predict and simulate natural wind flow from external environment through internal environment respectively. A low ranking would imply that the CFD software has no or few functions to simulate natural ventilation
3.6.3	Wind-driven rain (WDR)	Wind-driven rain (WDR) or driving rain is rain that is given a horizontal velocity component by the wind and that falls obliquely. WDR over unleveled ground such as hills or valleys results in a redistribution of raindrops by local wind flow deformations that can cause large precipitation variations. WDR is the most important moisture source affecting the hydrothermal performance and durability of building facades. A high ranking would imply that the CFD software is capable in defining and simulating a multiphase physics model, moisture, and humidity. A low ranking would imply that the CFD software is not capable to achieve above.
3.6.4	Humidity	Humidity is a major physical property that contributes to the definition of comfort condition. A high ranking would imply that the CFD software can simulate condensation of dew or the propagation of humidity in porous material. A high ranking implies that the CFD software is able to calculate the temporal and spatial distribution of relative humidity. A high ranking implies that the condensed liquid is calculated as a mixture fraction, i.e., the mass of the condensed liquid divided by the total mass of the liquid, vapor and carrier gas. A low ranking implies that the CFD software lacks capabilities to readily define humidity transportation processes.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 30

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
3.6.5	Wind load towards to building structure	Typically wind load from natural force and global wind force are predicted by wind pressure coefficient. Therefore, a high ranking would imply that CFD software is able to demonstrate and visualize wind pressure distribution on building surface, and also provide tools of measurement at node point, line, and grid on building façade. A low ranking would imply that the CFD software is not able to visualize and measure pressure coefficient.
3.6.6	Pressure distribution along streamlines/pressure losses	CFD derived pressure losses are a significant investigation on flow behavior in different obstruction or constrained geometry. A high ranking would imply that the CFD software can visualize pressure changes along streamlines. A low ranking would imply that the CFD software lacks of these built-in functions.
3.6.7	Conjugate heat transfer	Define the solid material conduction and the fluid convection is analyzed simultaneously. The type of fluid convection (natural, forced or mixed) determines the analysis parameters. A high ranking would imply that the CFD software can calculate simultaneous heat transfer in both a solid and a fluid. A low ranking would imply that CFD application is not capable to analyses and simulate heat transfer from solid to fluids in one integrated calculation process.
3.7	**Support for validation**	
3.7.1	Validation document / example	The benefit of this criterion is that it allows users to check the accuracy levels of a given CFD application as well as the experimental or real-world test set-up that the CFD software used for the data validation. Any recommendation from the validation process by the CFD software vendor is of significant help to the user. The ranking for this criterion is based on the availability of following validation

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 31

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		document/support: - Validation by CFD with reduced-scale model wind tunnel measurement regarding to architectural / wind engineering fields. - Validation by CFD with full-scaled onsite measurement regarding to architectural / wind engineering fields.
4	**Cost of software (Note: the ranking for cost will only be finalized in the FINAL report**	
4.1	Commercial pricing model	The benefit of the software product is a good benefit to cost ratio. The benefit to cost ration is established as the cost per value added. The ranking of these criteria is established as a continuous ranking scenario. The ranking value will be determined as a number between 1 and 5.
4.2	Academic pricing	The value of the CFD software is reflected in the availability of an academic pricing (discount) opportunity. A high ranking implies that the cost of the academic version is significantly lower than the commercial product pricing. The ranking of these criteria is established as a continuous ranking scenario. The ranking value will be determined as a number between 1 and 5.
4.3	Software includes all modules (no extra costs)	The value of the software is reflected in the extent at which all functionality is included in the base software price. This means that the software product should not require additional modules for different applications. As an example, the software should have a wide array of turbulence models included or the software product should be available to solve a wide array of CFD problems. A high ranking would imply that the software product has a wide array of all capabilities built-in the one software product that is purchased. A high ranking implies that the CFD

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 32

Table 2-2: Criteria used in Ranking of Candidate CFD Software Products

Criteria Number	Header Ranking Criteria Matrix	Description of Ranking Criteria or Value Proposition
		software contains all required modules at a "bundled price". A low ranking implies that the CFD software requires the purchase of additional modules to solve a variety of CFD problems; whereby each module would have to be purchased separately, thereby increasing the final price of the CFD software.
4.4	Option to acquire additional seats at no extra or small costs	The value of the software product is the flexibility to acquire additional seat at no or small extra costs. A high value would imply that the CFD software (or vendor) offers academic student licenses at very favorable conditions. When for example a CFD product is used in a class by students a high ranking implies that individual student licenses can be acquired a very favorable conditions; suggesting the academic pricing flexibility of the vendor. Therefore, a high ranking would imply a high flexibility and very favorable academic pricing for extra seats. A low ranking would imply a lack of these.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 33

Part 3 – Overall Ranking of CFD Software Based on Product Documentation Review

In this part of the report seven candidate CFD software products are analyzed and ranked in accordance to the ranking scheme and the definition of the criteria.

3.1 Introduction of Seven CFD Candidate Software Products

An initial research into CFD software products was conducted by the ERDL-HNEI CFD team. Seven candidate CFD software products were identified. The seven candidate CFD software products are briefly introduced in the following. The brief introductions of the software packages are based on product descriptions by the vendor. (The software packages are listed in alphabetic order)

ANSYS Fluent

ANSYS Fluent, a commercial CFD software package from ANSYS Inc., provides powerful capabilities of modeling a wide range of dynamic fluid problems from incompressible and compressible, laminar and turbulent fluid flows, as well as related transport phenomena (heat transfer and chemical reactions) in industrial equipment and processes. ANSYS Fluent's featured capabilities include both steady-state and transient analyses, robust turbulence models, complex geometry meshing tool as well as the integration into ANSYS Workbench platform for productive workflow.

ANSYS Inc., the owner of ANSYS Fluent founded in 1970, is the world's largest CFD company, and is the leading provider of CFD solutions to a wide range of industries, including aerospace, automotive, electronics, energy, materials and chemical processing, turbo machinery, academia, civil engineering, healthcare, sport, etc. Fluent was first developed by a group of researchers at Sheffield University in England in collaboration with Creare Inc. Fluent was first released in 1983 before being acquired by ANSYS, Inc in May 2006.

Source: Center for Computational Sciences, University of Kentucky

https://www.ccs.uky.edu/UserSupport/SoftwareResources/Fluent/; ANSYS Fluent 12.0 Theory Guide

Autodesk Simulation CFD

Autodesk Simulation CFD software (formerly CFdesign) provides accurate, flexible fluid flow and thermal simulation tools to help predict and validate product behavior and optimizes designs before manufacturing. Get additional fluid flow and thermal simulation functionality with Autodesk Simulation CFD Advanced. With Autodesk Simulation CFD Motion software, examine how mechanical devices components interact. The Autodesk Simulation family of products delivers a comprehensive set of

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 34

software tools allowing engineers to predict mechanical stress, flow, thermal, plastic injection molding, and more.

Autodesk Simulation CFD software provides comprehensive fluid flow simulation and thermal simulation, extending Digital Prototyping to applications ranging from electronics cooling to valve design, LED thermal management, thermal comfort, and product packaging. The company's Design Study Environment provides the tools needed to help make valid design decisions. One-Click Simulation Autodesk Simulation CFD tools simplify and expand simulation to every engineer on the team. You can set up automation to allow for just one-click to begin exploring flow and thermal design options. CAD-Entity Groups provide a convenient way to automate the assignment of setting. The Design Study Manager organizes and manages your CAD models for design studies directly inside your CAD tool. Design Study Rules are associations between CAD entities and Simulation CFD settings.

Source: http://www.uscad.com/other-products/autodesk-simulation-cfd/
and http://www.autodesk.com/products/autodesk-simulation-family/features/simulation-moldflows.

DesignBuilder with CFD Module

DesignBuilder is a powerful and easy to use Graphical User Interface to EnergyPlus. DesignBuilder is a complete 3-D graphical design modeling and energy use simulation program providing information on building energy consumption, CO2 emissions, occupant comfort, daylighting effects, ASHRAE 90.1 and LEED compliance, and much more.

DesignBuilder has an integrated CFD module that allows external as well as internal fluid flow analysis and visualization. The CFD module interfaces with the Energy Plus multi-zonal ventilation calculation process. It must be noted that the value of Design Builder is mostly in the ready integration of energy simulations and CFD calculations, using the same geometry and other pertinent data entries. Design Builder is not a stand-alone CFD software product.

DesignBuilder has been specifically developed around EnergyPlus allowing all of the EnergyPlus fabric and glazing data to be input. Databases of building materials, constructions, window panes, window gas, glazing units and blinds are provided.

Featuring an easy-to-use OpenGL solid modeler, which allows building models to be assembled by positioning 'blocks' in 3-D space and realistic 3-D elements providing visual feedback of actual element thickness and room areas and volumes. The user can import CAD data and there are no limitations on surface shape. Source> http://www.designbuilderusa.com/

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 35

CRADLE scTETRA

SC/Tetra is a general purpose unstructured mesh thermal-fluid analysis system. The CFD software is usability-focused software with the specific goal of easily enabling calculation of complex geometries. It provides both automatic mesh generation and a Wizard based interface that guides the user through the set-up process step by step. In addition to simulating standard fluid flow and heat transfer applications, SC/Tetra can calculate various phenomena such as Chemical Reactions, Arbitrary Lagrangian Eulerian for moving and/or rotating boundaries (ALE), Human Body Thermoregulation (JOS), Free Surfaces, and Acoustics

The SC/Tetra Pre-processor is used to prepare the geometry data for computation. It generates the computational mesh and defines boundary conditions. SC/Tetra's pre-processor is equipped with an intuitive graphical user interface (GUI) with Wizard based condition settings and CAD model repair functionality.

Mesh Generation: SC/Tetra uses an octant to specify the mesh resolution for a geometry and creates grids using the Advancing Front Method. It can create tetrahedron, pentahedron, and hexahedron elements. Prism layers can be inserted at walls to provide fine grid resolution in the boundary layer.

Adaptive Mesh Refinement: The Adaptive Mesh Refinement function can automatically refine the mesh resolution according to the calculation situation.

Condition Setting: SC/Tetra is general purpose CFD software that can simulate many different physical phenomena. Usually, CFD software having many functional options tends to be complicated and difficult to use.

CAD Data Repair Function: SC/Tetra can import 3D data in STL and DXF formats. If the 3D data has defects in it, a computational mesh cannot be created. The SC/Tetra pre-processor contains powerful geometry diagnostic and repair functionality that rival 3rd party grid generation software. Cradle's CADthru software can also be used to automatically repair 3D models.

Solver: The input data (computational mesh, boundary conditions, and material properties) prepared using the Pre-processor are read into the solver for computation of the thermal-fluid flow field. The user only needs to specify the name and location of the input file. The solver is equipped with a monitoring function that shows the status of the computation. In addition to solving basic fluid flow and heat transfer problems, SC/Tetra is equipped with many physical models capable of simulating a wide variety of industrial applications, such as:

- ALE (Moving and/or Rotating Boundaries) / Overset mesh / Dynamic ALE
- Free surface analysis (VOF method)
- Human Body Thermoregulatory Model (JOS)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 36

- Electric Current (Joule Heating) Analysis
- Cavitation
- Aeroacoustic Analysis

Post-Processor is a common tool used by many Cradle CFD programs and is used to visualize the data calculated by the Cradle solver. It is equipped with a multitude of functions such as contour, vector, and specifying values at specific points in the calculation domain which are important features for the design engineer. The Cradle Post-Processor also contains many other advanced visualization functions. Source: http://www.cradle-cfd.com/products/sctetra/

CRADLE scSTREAM

scSTREAM uses a structured mesh to provide extraordinary performance in both meshing and computation speed. Its performance is maximized for applications that do not require precise reproduction of fine geometrical curvature to predict accurate flow structures. The finite volume method is used for discretization which is common for fluid/thermal analysis. scSTREAM can solve a one million mesh model using less than 300 MB of RAM. Various useful functions such as a VB interface and a user function with tabular input are provided for customer user interfaces.

The scSTREAM Pre-processor is used to configure the data for calculation. It is used to create shapes, generate meshes, and define boundary conditions and material properties. scSTREAM's Pre-processor provides an easy-to-understand and use graphical user interface (GUI).

The scSTREAM Solver uses the input data defined in the Pre-processor to calculate the thermal-fluid flow field. Besides standard CFD functions for solving basic structured grid thermal-fluid problems scSTREAM provides a multitude of specialized functions for simulating highly complex thermal-fluid phenomena. Some of them are mentioned below:

- Free Surface Analysis
- Moving Object Function
- Panel Heat Conduction Radiation/Porous Solid
- Particle Analysis Function
- Dew Condensation / Evaporation Function
- Large Eddy Simulation (LES)
- Sky Solar Radiation

The scSTREAM Post-Processor is common for the suite of Cradle's CFD programs and is used to visualize the data calculated by the solver. The Post-processor is equipped with a multitude of display functions such as contour, vector and values at specific points. These basic functions along with other high-quality graphical output functions provide all the tools necessary to visualize and communicate CFD output. Source: http://www.cradle-cfd.com/products/stream/

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 37

OpenFOAM

OpenFOAM (Open source Field Operation and Manipulation) is a free and open source CFD software package as the library of applications written in C++ language. It consists of over 80 solvers and over 170 utilities to simulate specific problems in engineering mechanics covering a wide range of problems from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics as well as perform pre- and post-processing tasking (e.g. meshing, data visualization, etc.). By being under an open source license, OpenFOAM publishes its codes and allows users to customize and extend its existing functionalities. This is the important support to academic and scientific users who would like to write / modify codes for their mechanic problems, experiments or validations, which will be impossible to most commercial CFD packages.

OpenFOAM is maintained and developed by OpenFOAM Foundation, which is sponsored by ESI Group who is the owner of the trade mark to the name OpenFOAM. FOAM, the predecessor of OpenFOAM, was first developed in the last 1980s at Imperial College, London and then was sold to UK company Nabla Ltd before being released as open source in 2004. It was then bought by Silicon Graphics International (SGI) in 2011 before being transferred to ESI Group in September 12, 2012. Source: OpenFOAM Foundation http://www.openfoam.org/

CdAdapco Star CCM+

STAR-CCM+ provides the world's most comprehensive engineering physics simulation inside a single integrated package. Much more than just a CFD solver, STAR-CCM+ is an entire engineering process for solving problems involving flow (of fluids or solids), heat transfer, and stress. It provides a suite of integrated components that combine to produce a powerful package that can address a wide variety of modeling needs. These components include; 3D-CAD modeler, CAD embedding, Surface preparation tools, Automatic meshing technology, Physics modeling, Turbulence modeling, Post-processing, and CAE Integration.

STAR-CCM+ is based on object-oriented programming technology. It is designed to handle large models quickly and efficiently using a unique client–server architecture that seamlessly meshes and simultaneously solves and post-processes over multiple computing resources without requiring additional effort from the user. STAR-CCM+ recently became the first commercial CFD package to mesh and solve a problem with over one billion cells. The object-oriented nature of the code can be seen in the user interface. An object tree is provided for each live simulation, containing object representations of all the data associated with the simulation. The objects presented on the simulation tree reside on the server, which can run as either a serial or a parallel process. A client interface connects to a server, and displays the simulation objects available on that server. Source: Star-CCM+ user guide version 8.02

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 38

3.2 Ranking of ANSYS-Fluent

In this section the CFD software product ANSYS-Fluent is assessed. The software's level of consistency with the ranking criteria is determined. The result of ranking is presented in Section 3.9, along with the results of the ranking of other the other candidate software products.

The criteria referred to are the criteria listed in the ranking matrix.

3.2.1 Criteria Group: Value of CFD software product for architecture

Criterion 1.1:   Benefit for external CFD   **ranking score = 5**

ANSYS-Fluent (Fluent) offers many features capable to deal with most CFD applications, especially allow for modeling external computational domain in external CFD study. These features include wide range of robust turbulence models, unstructured meshing with wide range of mesh types to handle the complexity of the buildings and surroundings: triangle, quadrilateral, tetrahedron, hexahedron, pyramid, prism (wedge) and polyhedron, customizable capability to model atmosphere boundary layer.

Criterion 1.2:   Previous application in architecture / urban design   **ranking score = 5**

Fluent claims that its software code has been leading the CFD industry and Fluent and is presently being used by half of CFD global market. CFD Fluent has been used in architecture and urban design practice. For example, Fluent was used for studying the internal and external airflow pattern of the Allianz Stadium (ANSYS, 2009). There are several consultants using Fluent as CFD simulation tool for their service.

Criterion 1.3:   Previous mentioning of software product in scientific or applied professional articles
ranking score = 5

Fluent software is the common CFD codes found in scientific and applied professional articles. Particularly, among articles used as literature review for this project, the ERDL-HNEI CFD team found that about 80% of all articles reviewed used Fluent as CFD simulation tool. Fluent also has been established partnership with many academic institutions to apply CFD technology into research as well as develop tutorials and improvement. Those academic institutions include University of St. Thomas in Minnesota, University of Albert, Pennsylvania State University, Cornell University, Carnegie Mellon University, University of Texas Arlington, University of Wisconsin, University of Manitoba, Rochester Institute of Technology, University of Iowa, Dartmouth College (http://www.ansys.com/Industries/Academic/Tools/Curriculum+Resources)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                         School of Architecture, University of Hawaii

December 4, 2013   Page 39
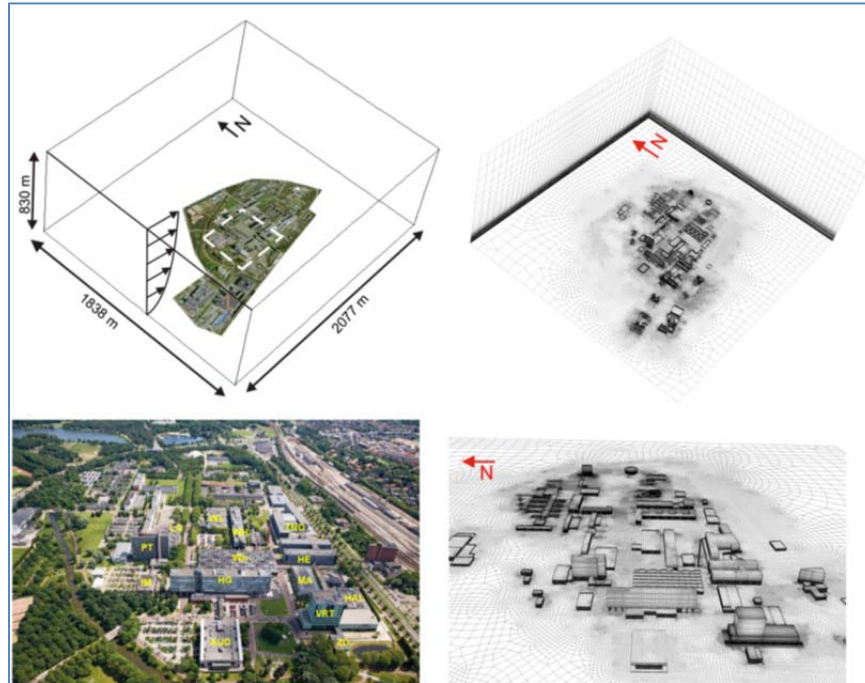
Figure 3.2.1A: An example of modeling atmosphere boundary layer and high-resolution computational domain grid of a campus using Fluent (Janssen et al., 2013)
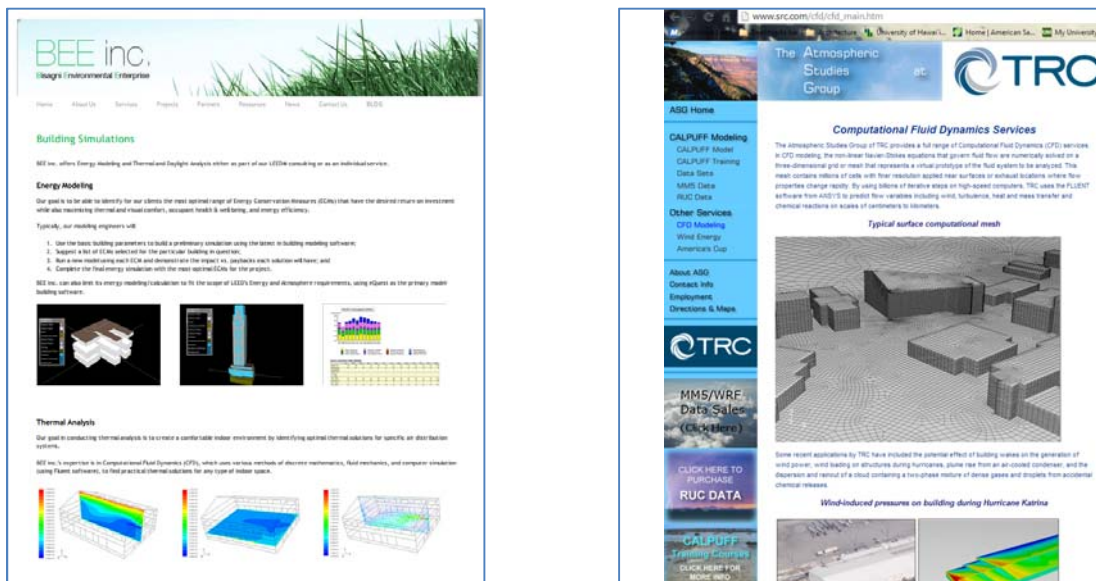

Figure 3.2.1B: Consulting companies use Fluent as CFD simulation tool for their service
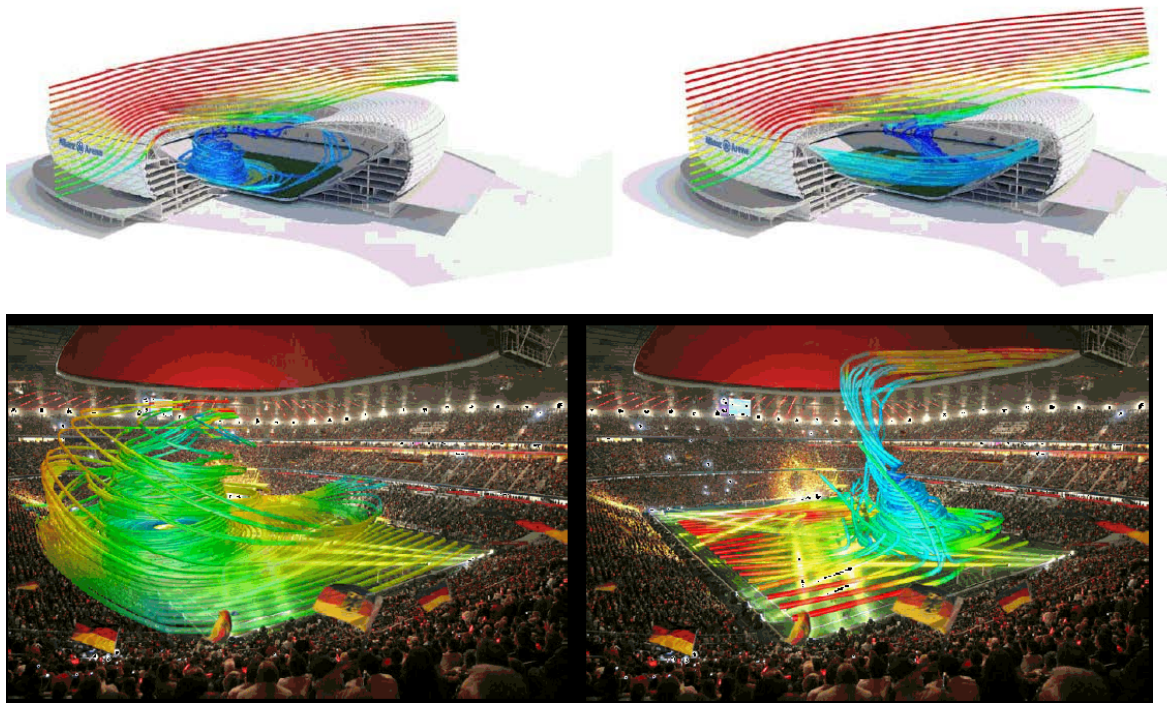
Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                          School of Architecture, University of Hawaii

December 4, 2013   Page 40

Figure 3.2.1C: Allianz Stadium simulated airflow pattern with closed and opened door scenarios (ANSYS, 2009)

Criterion 1.4:   Using graphics output for presentation   **ranking score = 4**

CFD Fluent provides built-in visualization capabilities to deliver different graphical output formats for presentation. Beside this, ANSYS also provides a CFD-Post application, which enables a variety of enhanced features for post-processing such as surface and volume rendering options, texturing and transparency, animations, etc. The quality of those graphics is appropriate for high-quality-demand architectural presentation.

3.2.2 Criteria Group Support by Vendor / Expert Network

Criterion 2.1:   Technical support available: **ranking score = 4**

Based on the information available on its website, ANSYS (Fluent) provides a variety of forms of technical support. For example, a customer portal is available where user-specific support and resource are presented to a wide range of clients' need; technical enhancements and customer support program is another customer service for ANSYS license maintenance, updated software and searching Technical

Support Knowledge Base. Moreover, ANSYS (Fluent), with its large number of staff, allows timely delivery of technical support inquires.
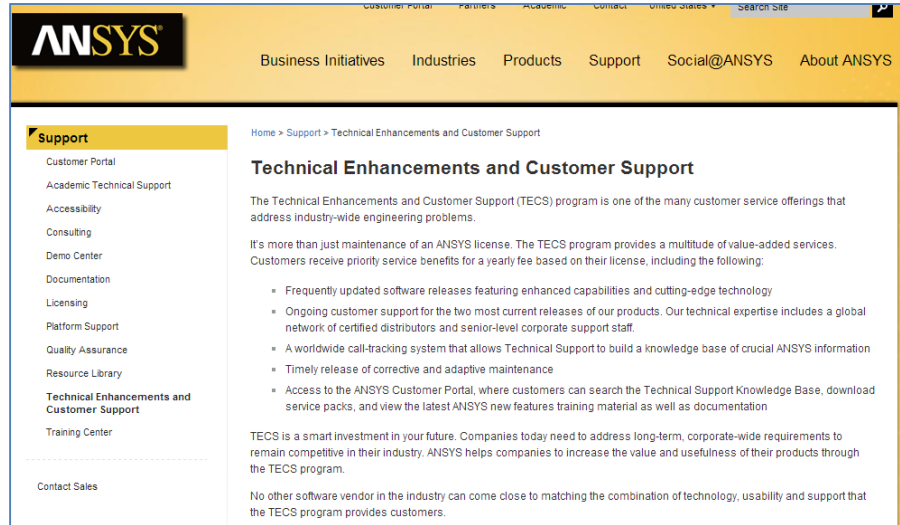
Figure 3.2.2A: Technical support options through the ANSYS website

Criterion 2.2:   Online tutorials available on demand   **ranking score = 3**

All of the company software's tutorials, manuals, best practices are available through customer portal. Therefore, obtaining the software license is necessary to access to this resource. Otherwise, many tutorials, CFD curriculum resources, which were developed by many academic institutions having partnership with ANSYS, are also available at those institutions' website without license requirement.

The quality (ease, intuition…) of these online tutorials needs a further assessment.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 42
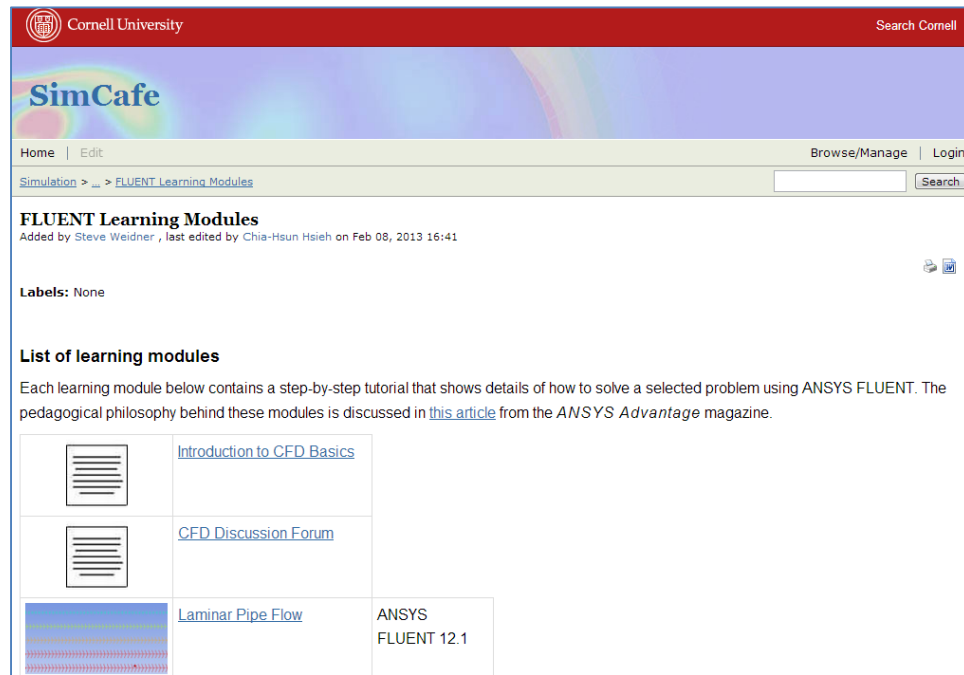
Figure 3.2.2B: Fluent learning modules at Cornell University

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 2**

ANSYS Fluent offers a variety of training through Customer Training Center, including on-site training, dedicated mentoring, jumpstart training, unlimited training and classroom training. Many of these training types can be carried out through online. These training classes, however, require fee.

Criterion 2.4:   Network of CFD professionals in architecture field: **ranking score = 2**

Despite ANSYS providing a variety of different social networks for the ANSYS user community (Social@ANSYS) such as YouTube, Facebook, Tweet, Blog, LinkedIn, XANSYS.org, ANSYSFORUM.com, etc. there is also a dedicated section in CFD-ONLINE for ANSYS Fluent user. However, Fluent does not support a user community network for CFD professionals in architecture field.

Figure 3.2.2C: An example of ANSYS user forum
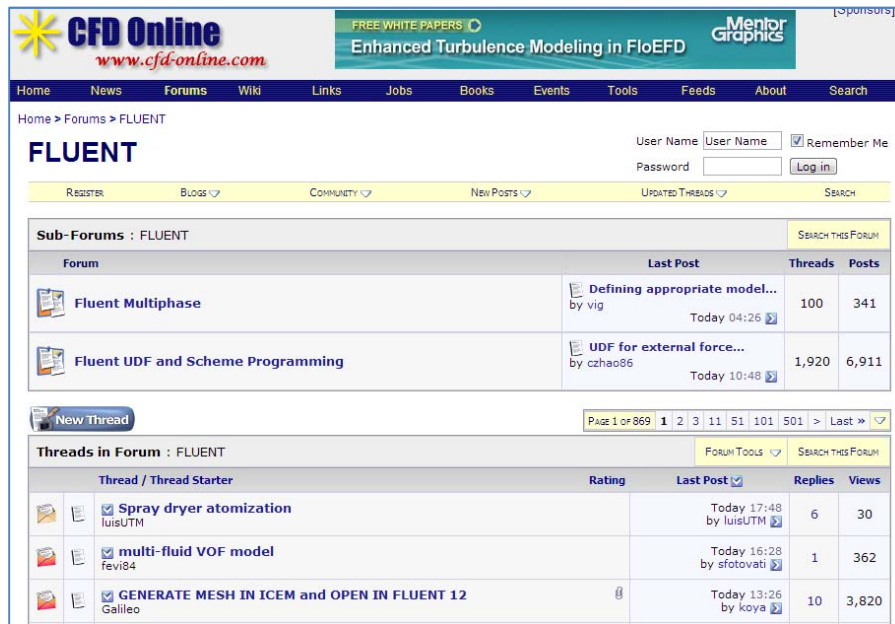
Figure 3.2.2D: ANSYS user forum at www.cfd-online.com

Criterion 2.5:   Ease of working with academic support: **ranking score = 4**

There is a dedicated academic support portal at ANSYS. Based on the information on its website, ANSYS has established a strong outreach to academic institutions. ANSYS has also engaged in significant collaboration with numerous universities and colleges worldwide in integrating ANSYS (Fluent) into their curriculum. Moreover, there is an academic server license for school and students can purchase

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                 School of Architecture, University of Hawaii

December 4, 2013   Page 44

affordable student client license. However, an actual assessment on this criterion should be further assessed when this study is going on.

Figure 3.2.2E: Academic support options through the ANSYS website

Criterion 2.6:   User guide provides efficient and comprehensive information: **ranking score = 5**

ANSYS (Fluent) provides many resources for users such as User guide with comprehensive learning resources linked to the CFD application. Static (PDFs) user guides are available to explain all technical solutions employed in the code. Those documentations are available at ANSYS.com at no cost.

3.2.3 Criteria Group: Workflow - Ease of use / GUI

Criterion 3.1.1: GUI is easy to use for trained operator   *ranking score = 4*

Fluent's GUI is organized accordingly to CFD workflow which allows for typical sequent procedures such as importing models from CAD or preparing model in Design Model, meshing, running solvers and post-processing.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 45

Geometry import from CAD -> Prepare model in Design Model -> Mesh geometry -> Inputs (boundary conditions, material properties…) -> Solution -> Post-processing

Figure 3.2.3: ANSYS (Fluent) workflow illustration

Criterion 3.1.2: User guide and online help is available for each command   **ranking score = 3**

Although user guide and manual guide documentation is available online as well as other online tutorials / wiki can be found in numerous CFD user forums, such as Fluent FAQ (www.cfd-online.com/Wiki?Fluent_FAQ), these resources however cannot be available at each command.

3.2.4 Criteria Group: Workflow – Modeling / Import Geometry

Criterion 3.2.1: Create geometry inside CFD application   **ranking score = 4**

ANSYS (Fluent) provides built-in geometry modeling tool called DesignModeler with flexible capabilities to allow the creation of geometry as well as meshing, automatically creating meshes or hand-crafting objects with high level of complexity.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                    School of Architecture, University of Hawaii

December 4, 2013   Page 46
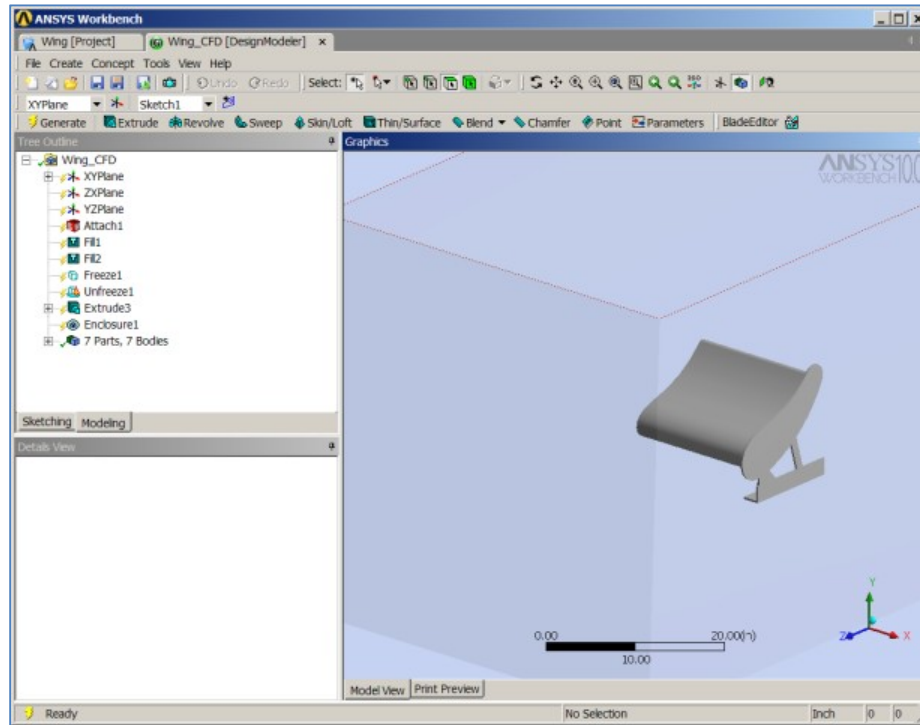
Figure 3.2.4A: DesignModeler as built-in geometry modeling tool in Fluent

Criterion 3.2.2: Import geometry / repair surface before meshing   **ranking score = 4**

With DesignModeler, ANSYS (Fluent) provides also offers advanced repair tools for imported geometry allowing partly or fully manual meshing.

Criterion 3.2.3: Support Revit Architecture   **ranking score = 3**

ANSYS (Fluent) does not directly support geometry from REVIT, it only accepts REVIT output (ACIS format).

Criterion 3.2.4:   Support CADD format and Inventor   **ranking score = 4**

ANSYS (Fluent) does support CADD format and Inventor. However, the quality of the transition has not been assessed in this study.

Figure 3.2.4B: ANSYS Geometry interfaces; source: http://www.ansys.com/staticassets/ANSYS/Conference/Confidence/Houston/Downloads/overview-145.pdf

Criterion 3.2.5: Support Rhino   *ranking score = 2*

ANSYS (Fluent) does not directly support Rhino3d. A transition to other formats which are compatible to ANSYS (Fluent) has not been investigated this study.

3.2.5 Criteria Group: Workflow - Meshing

Criterion 3.3.1: Surface meshers   **ranking score = 1**

ANSYS (Fluent) offers advanced surface meshers with automated and manual repairing tools for meshing. A full range of different mesh types are supported in ANSYS (Fluent) for generating highly complex geometries. However, other meshers' capabilities such as Surface wrapper and Surface Remesher could not be detected in Fluent and therefore this issue needs further investigation.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                           School of Architecture, University of Hawaii

December 4, 2013   Page 48

<u>Criterion 3.3.2:</u> Maximum cells and 2D-3D cell volume   **ranking score = 5**

The ERDL-HNEI knowledge, there is no documentation from ANSYS other resource mentioning this maximum allowable cells for Fluent application. However, many studies of external airflow using Fluent by Dr. Bert Blocken at Eindhoven University of Technology, the cell number can reach 6 million.

<u>Criterion 3.3.3:</u> Boundary types   **ranking score = 5**

ANSYS (Fluent) provides 10 types of boundary zone types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass flow inlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, and exhaust fan. These boundary condition types provide support for a wide range of physics models.

<u>Criterion 3.3.4:</u> Structured and Unstructured meshing   **ranking score = 5**

ANSYS (Fluent) provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include triangular, quadrilateral, tetrahedral, hexahedral, pyramid, prism (wedge) and polyhedral.
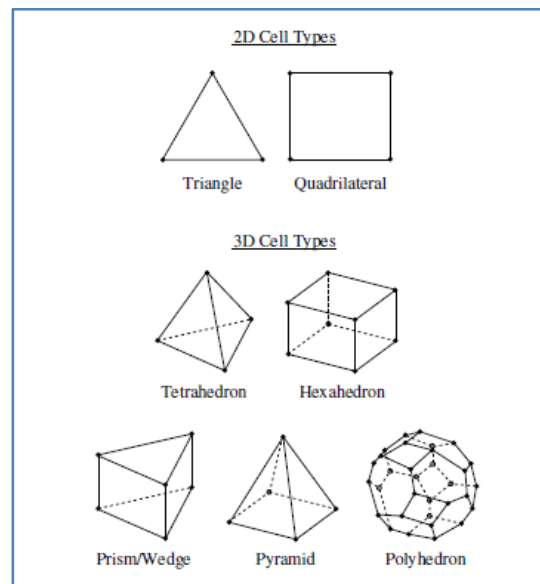
Figure 3.2.5A: 2D and 3D cell types supported by ANSYS Fluent

Figure 3.2.5B: Polyhedral meshing example from ANSYS Fluent

Criterion 3.3.5: Adaptive meshing capability   **ranking score = 5**

ANSYS Fluent allows dynamic refinement or coarsening of the mesh based on the flow solution. However, the ease of use as well as the amount of effort needed to achieve the accuracy and the high refinement level at the flow features should be further investigated.
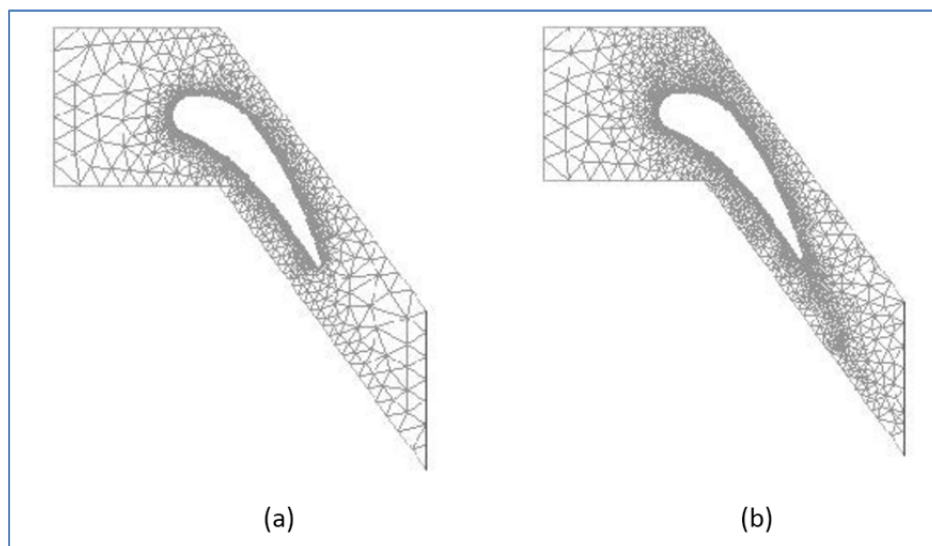

(a)   (b)

Figure 3.2.5C: Turbine cascade mesh before (a) and after (b) adaption. Source: ANSYS Fluent User Guide (2009)

3.2.6 Criteria Group: Workflow – Physics Models

Criterion 3.4.1: Solvers – Coupled / decoupled   **ranking score = 5**

With robust and flexible meshing tools, ANSYS (Fluent) provides capability to model "full-domain" simulation solving the internal and external airflow simultaneously. Dr. Bert Blocken at Eindhoven University of Technology has been using ANSYS (Fluent) for this approach.
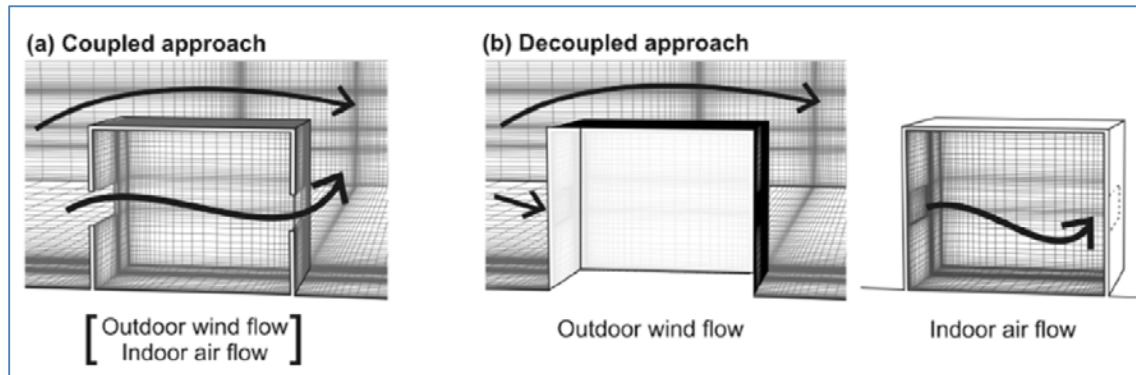
Figure 3.2.6A: Coupled and decoupled approach for analysis of wind-induced cross-ventilation buildings (Ramponi and Blocken, 2012)

Criterion 3.4.2: Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)   **ranking score = 5**

The ANSYS Fluent numerical method for discretization approach is based on Finite Volume Method (FVM), and therefore this application receives the highest ranking for this criterion.

Criterion 3.4.3: Steady state/Transient state   **ranking score = 5**

ANSYS (Fluent) offers not only steady state but also transient state solution which allows for simulating a wide variety of time-dependent external airflow phenomena such as vortex shedding and other time-periodic phenomena, transient chemical mixing and reaction in combustible dispersion simulation.

Criterion 3.4.4: Turbulence models   **ranking score = 5**

ANSYS (Fluent) delivers cutting-edge turbulence models, including several versions of the k-epsilon and k-omega models as well as the Reynolds stress model for highly anisotropic flows. Advanced scale-

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 51

resolving turbulence models are also available: large eddy simulation (LES), detached eddy simulation (DES) and scale adaptive simulation (SAS).

<u>Criterion 3.4.5:</u> Materials   **ranking score = 4**

ANSYS (Fluent) software provides a wide range of material modeling options. ANSYS (Fluent) comes with a comprehensive list of material properties for liquids, gases, and solids. For any other material or material property not already included in the material database, ANSYS (Fluent) users can take advantage of the flexibility of the user environment to easily define any number of new materials or dependencies of material properties on flow conditions such as pressure, temperature, shear-strain rate and more.
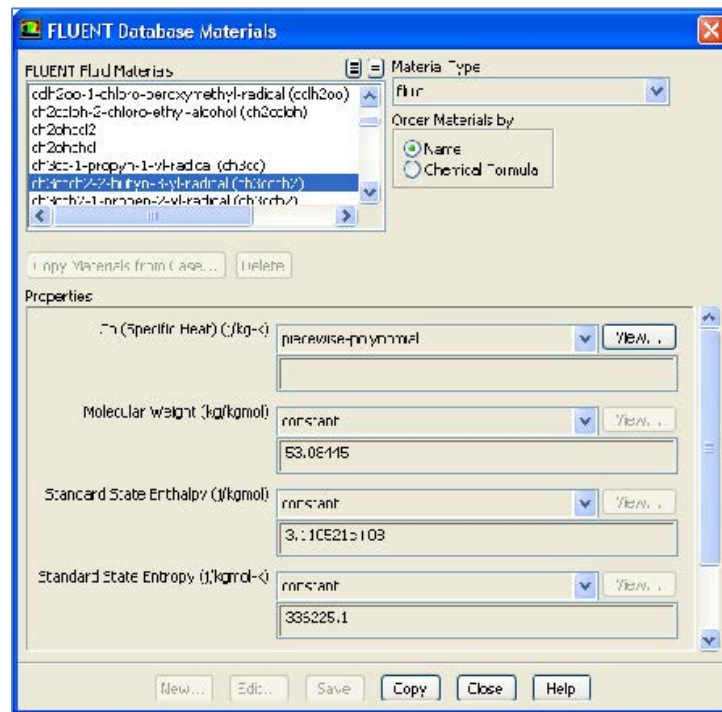
Figure 3.2.6B: ANSYS Fluent Database material dialog   Source: ANSYS Fluent User Guide (2009)

<u>Criterion 3.4.6:</u> Customizable / special physics models   **ranking score = 4**

ANSYS Fluent software features the latest technology for combining fluid dynamics solutions using conjugate heat transfer, combustion and chemical reaction in simulating combustion for assessing fire safety. In addition, users can enter any algebraic expressions for such custom models directly in the

ANSYS Fluent GUI using the simple syntax or user-defined functions. However, ANSYS Fluent does not support special models for humidity sorption in wall materials.
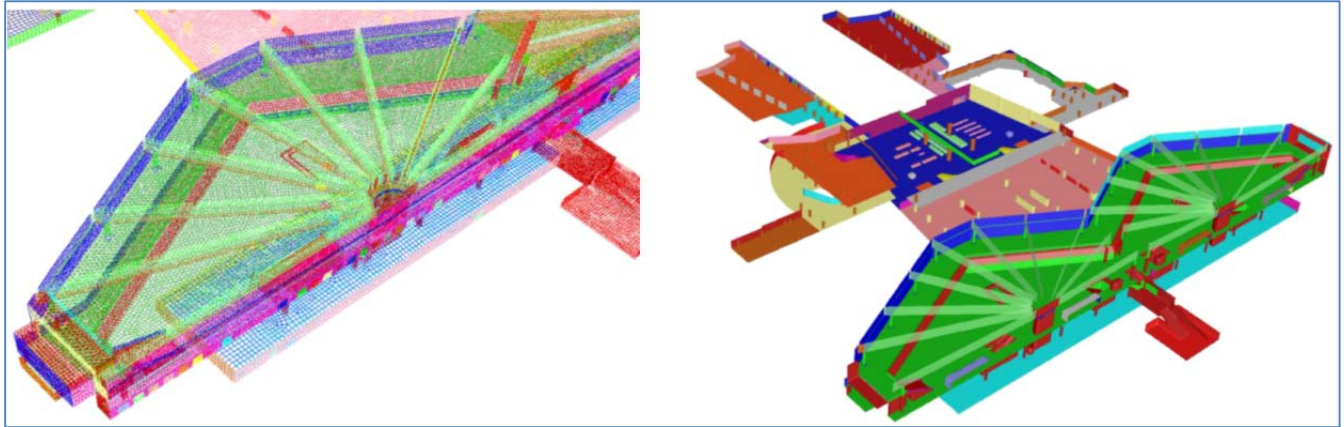

Figure 3.2.6C: Computational mesh and internal model for smoke study at Frankfurt Airport ANSYS 2009.

3.2.7 Criteria Group: Workflow – Post-processing

Criterion 3.5.1: Visualization   **ranking score = 4**

Post-processing tools for ANSYS (Fluent) can be used to generate a full range of graphical formats consisting of shaded and transparent surfaces, pathlines, vector plots, contour plots, particle trace and custom field variable definition as well as animations to convey fluid dynamics results.

Criterion 3.5.2: Quantitative analysis capabilities of components   **ranking score = 4**

ANSYS (Fluent) provides tools for computing and reporting integral quantities at surfaces and boundaries. ANSYS (Fluent) can directly produce the quantitative analysis capabilities of certain nodes, line or grid within the computational domain. However, solution data can be exported to ANSYS CFD-Post, third-party graphics packages or CAE packages for additional analysis.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                         School of Architecture, University of Hawaii

December 4, 2013   Page 53
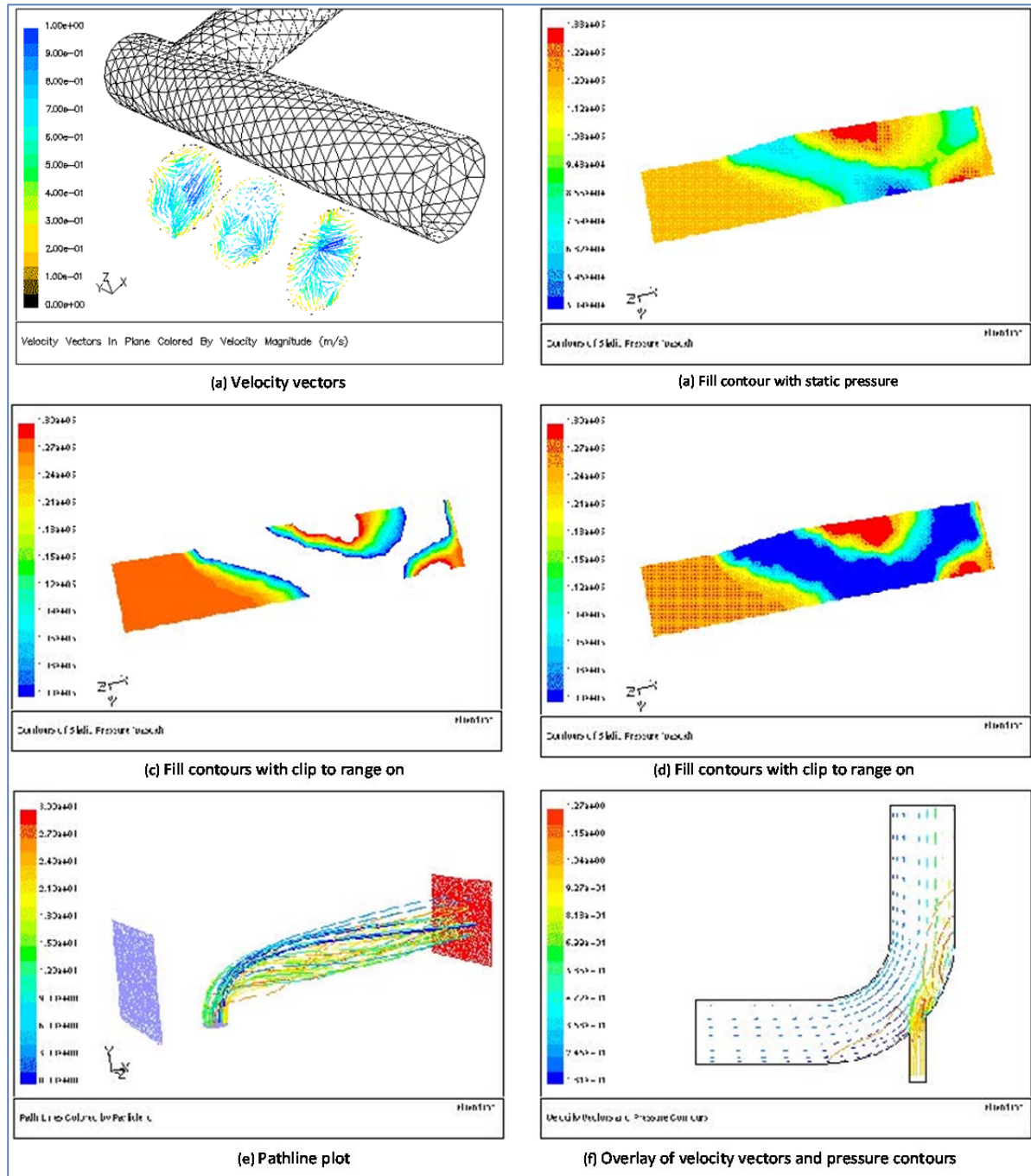
Figure 3.2.6C: Some examples of visualization outputs   Source: ANSYS Fluent User Guide (2009)

<u>Criterion 3.5.3:</u> Comparative analysis (multi-scenarios) **ranking score = 4**
ANSYS CFD-Post offers capabilities to visually and quantitatively comparing different solution scenarios of a design without re-meshing requirement.
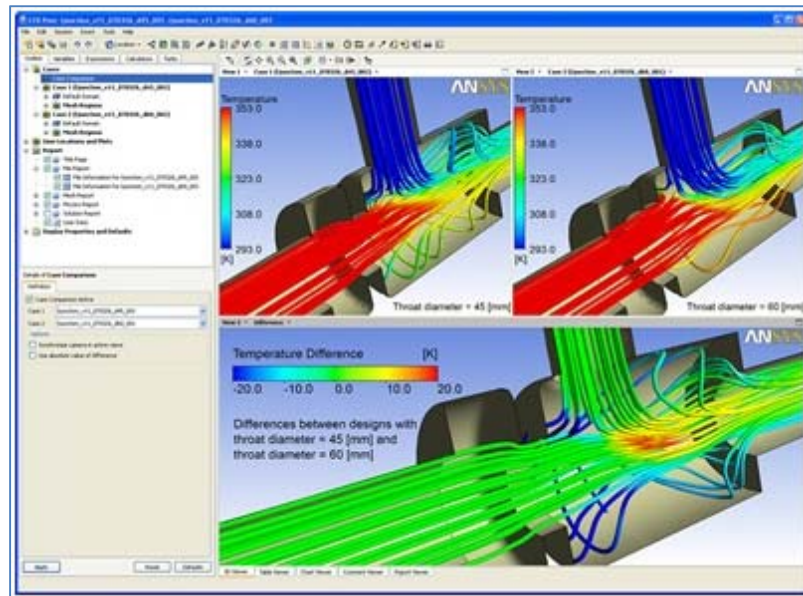

Figure 3.2.7: Different designs can be compared directly in ANSYS CFD-Post, both visually and quantitatively. Source: ANSYS ©

3.2.8 Criteria Group: Workflow – Customizable / Special Functions

<u>Criterion 3.6.1:</u> Pedestrian comfort **ranking score = 5**

With flexible and user-defined functions, ANSYS (Fluent) provides capabilities to model wind velocity related pedestrian comfort.

<u>Criterion 3.6.2:</u> Natural Ventilation **ranking score = 5**

Fluent provides robust and flexible meshing tools, capability to model coupled computational domain and heat transfer – conduction, convection (natural and forced), radiation (solar, thermal), conjugate heat transfer, indoor air quality (mean age of air, thermal comfort (PMV, PPD)). Fluent allows solving wind-driven and buoyance-driven natural ventilation.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 55

Criterion 3.6.3: Wind-driven rain (WDR)   **ranking score = 3**

With flexible and user-defined functions, ANSYS (Fluent) provides capabilities to model any wind driven airflow phenomena. However, since this criterion also requires multiphase physics models including moisture and humidity which is not available or dedicated in ANSYS (Fluent). The ranking for this criterion is at an average.

Criterion 3.6.4: Humidity   **ranking score = 1**

Humidity analysis is not available and there are no special models for humidity analysis in buildings in ANSYS (Fluent). Humidity modeling could be accomplished by using user define function (DUF).

Criterion 3.6.5: Wind load towards to building structure   **ranking score = 5**

ANSYS (Fluent) allows for external wind simulation as well as computed building surface pressure distribution used for wind loading analysis.
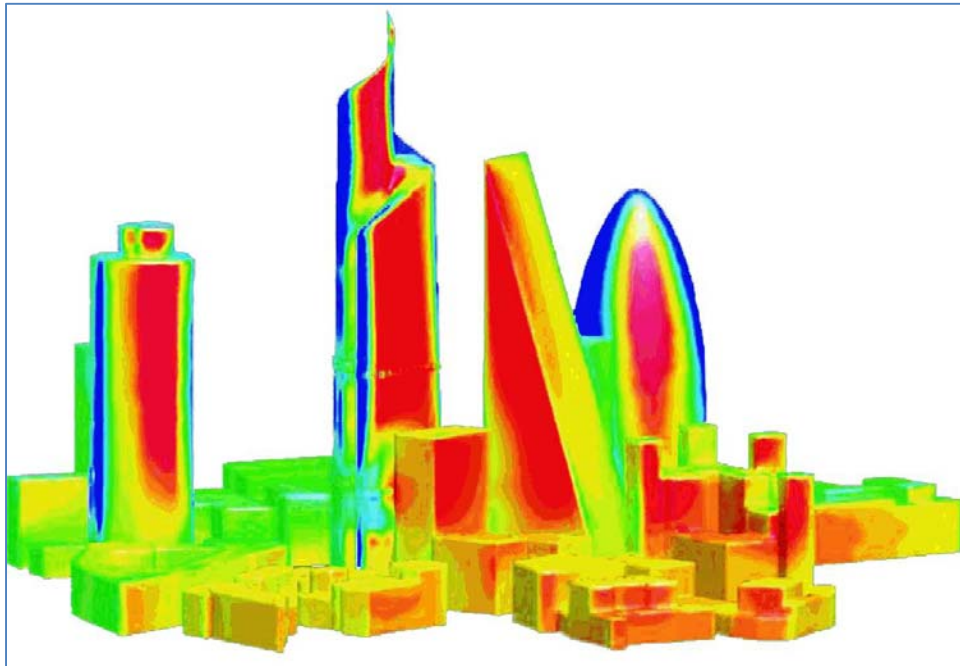
Figure 3.2.8A: Pressures on the surfaces of Bishops-gate Tower (London) and some surrounding buildings ANSYS (2009)

Criterion 3.6.6: Pressure distribution along streamlines/pressure losses   **ranking score = 5**

ANSYS (Fluent) allows for setting pressure loss coefficients of different obstruction or constrained geometries such as porous objects, radiators. In addition, Fluent has ready functionalities to visualize the pressure losses along the streamline.

Criterion 3.6.7: Conjugate heat transfer   **ranking score = 5**

ANSYS (Fluent) provides conjugate heat transfer which allows for simultaneously simulating solid material conduction and fluid convection which can be seen in buoyance-driven ventilation.
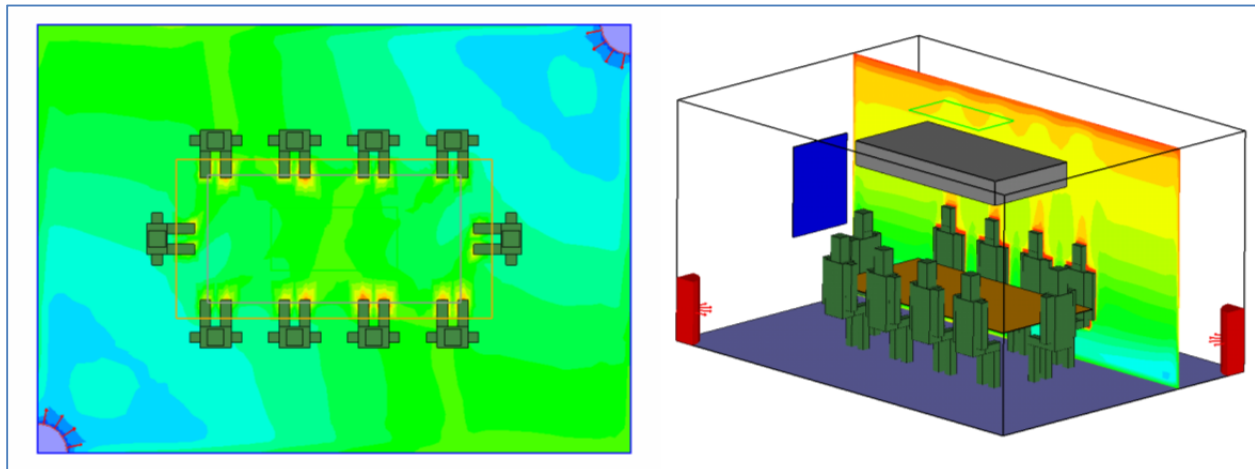
Figure 3.2.8B: Floor temperature and thermal stratification in a conference room with displacement ventilation by ANSYS (2009)

3.2.9 Criteria Group: Workflow – Support for validation

Criterion 3.7.1: Validation document / example   **ranking score = 4**

ANSYS (Fluent) has been validated against reduced-scale model wind tunnel measurement as well as full-scale onsite measurement. However, the availability of these validation documents from office ANSYS website could be verified at this time.

3.2.10 Criteria Group: Workflow – Cost of software

Criterion 4.1:   Commercial pricing model   **ranking score = 3**

Criterion 4.2:   Academic pricing   **ranking score = 4**

ANSYS offers very low academic pricing which cost only $25/year (at 2013) to U.S. engineering students who are associated with academic institutions which have purchased the ANSYS Academic Teaching or Research license.
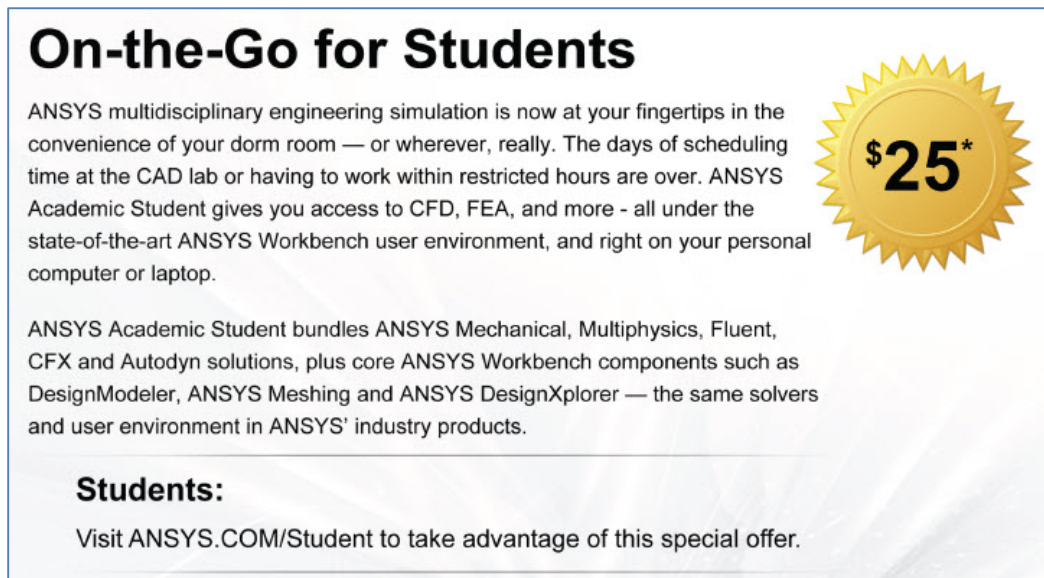
On-the-Go for Students

ANSYS multidisciplinary engineering simulation is now at your fingertips in the convenience of your dorm room — or wherever, really. The days of scheduling time at the CAD lab or having to work within restricted hours are over. ANSYS Academic Student gives you access to CFD, FEA, and more - all under the state-of-the-art ANSYS Workbench user environment, and right on your personal computer or laptop.

$25*

ANSYS Academic Student bundles ANSYS Mechanical, Multiphysics, Fluent, CFX and Autodyn solutions, plus core ANSYS Workbench components such as DesignModeler, ANSYS Meshing and ANSYS DesignXplorer — the same solvers and user environment in ANSYS' industry products.

Students:
Visit ANSYS.COM/Student to take advantage of this special offer.

Figure 3.2.10:   Academic license support at ANSYS website

Criterion 4.3:   Software includes all modules (no extra costs)   **ranking score = 5**

To the knowledge of the ERDL-HNEI CFD team, ANSYS Academic Student bundles include all modules from ANSYS Mechanical, Mutiphysics, Fluent, CFX , Autodyn solutions and ANSYS Workbench (Design Modeler, ANSYS Meshing and ANSYS DesignXplorer).

Criterion 4.4:   Option to acquire additional seats at no extra or small costs   **ranking score = 5**

Based on the requirement of this criterion, ANSYS meet the highest ranking for its supports on academic student licenses for extra seats.

3.3 Ranking of Star-CCM+ V8.02

This section provides a description of the capabilities of **Star-CCM+** software.

3.3.1 Criteria group: Value for the CFD Software Product for Architecture

Criterion 1.1:   Benefit for external CFD applications to architecture   **ranking score = 5**

STAR-CCM+ is a well validated CFD software to the study of airflow through a room, conference hall, or entire building, using real world ventilation grates and examining the exchange of the entire body of air within a certain time frame. STAR-CCM+ can model solar radiation within a building, including simulation of window glazing to reflect solar radiation. Up to 50% of the heat transfer in an environment can be attributed to solar radiation, making it a crucial design study. In addition, a well-designed ventilation system will greatly improve occupant comfort, a good bonus for any building. Of particular interest to the present CFD research project and also for the design of sustainable buildings in Hawaii, the is the examination of airflow patterns around a building, specifically examining the stress on a building caused by wind, and calculating the wind speed for pedestrians walking near the building. As more and more high buildings are built, and they are packed in more densely design neighborhoods, these structures begin to dramatically affect wind patterns around the buildings. This can create dangerous situations, such as high wind speeds on a building, causing significant stress on a building's primary load beams and high wind speeds for pedestrians walking in between them. STAR-CCM+ is well validated to perform all types of wind speed simulations.
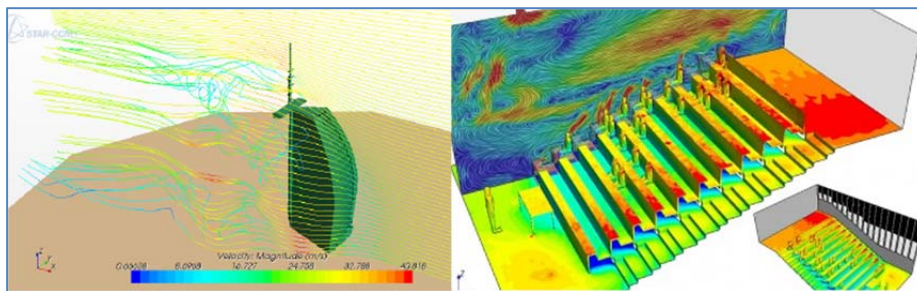(http://www.cd-adapco.com/industries/building-services)

Figure 3.3.1A: STAR-CCM+ for Building Services Analysis, 2011 and CD-Adapco, Building Service

Criterion 1.2:   Previous applications in architecture urban design**:**   **ranking score = 3**

CD-adapco is the world's largest independent CFD-focused provider of engineering simulation software. For more than 30 years the company has provided support and services for industrial strength

engineering simulation. The scope of activities extends well beyond software development to encompass a wide range of CAE engineering services in both CFD and FEA. There are over 7000 users of the software, working at 3000 different companies and institutions.

Criterion 1.3:   Software product in scientific or applied professional articles**:   ranking score = 2**

There are a few academic articles from Star CCM+ on CFD in building design and performance. The company is currently expanding its reach into the building industry. The company has extended considerable interest to the ERDL-HNEI CFD team to support the CFD work in building related work.

Criterion 1.4:   Using graphics output for architectural presentations   **ranking score = 1**
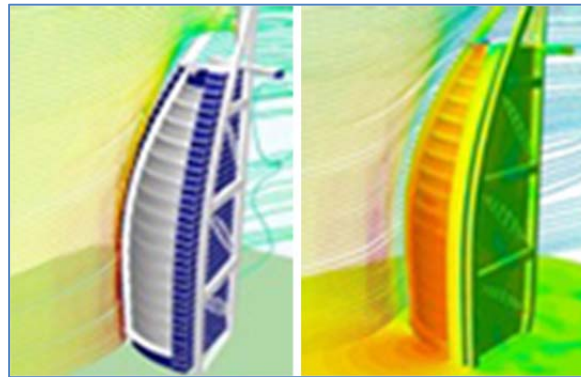
Figure 3.3.1B: CFD Simulation for the Building Environment, 2013

Star CCM+ lacks high quality graphics tools to showcase the result of CFD simulation, such as wind behavior on both pressure distribution on building façade and wind velocity. The visualization tools that StarCCM+ provides do not include architectural rendering images, such as ray-tracing renderings. The standard visualization that is provided by StarCCM+ shows basic shaded 3D geometry, without any advance lighting function. On the other hand, the software can provide graphics output that can be integrated into external advanced graphics tools.

3.3.2 Criteria Group Support by Vendor / Expert Network:

Criterion 2.1:   Technical support available: **ranking score = 5**

The technical support offers a full range of support services. The technical issue and problem to be answered by the support team is submitted through "The Steve Protocol". In addition the ERDL-HNEI has been using direct communication (e.g. calling the support desk) with support staff. The quality of the overall service is in high. The account executive for the ERDL-HNEI CFD team, Dr. Lisa Mesaros has always interacted and responded quickly. The company has been fully supporting the team in both academic and technical issues.

Figure 3.3.2A: Technical support option through the CD Adapco website

Criterion 2.2:   Online tutorials available on demand: **ranking score = 5**

CD Adapco offers full documentation online, which includes tutorials, manual guide, and best practice measures. Star CCM+ has shown a large number of YouTube videos available for certain applications. The quality of the online tutorial offerings is high.
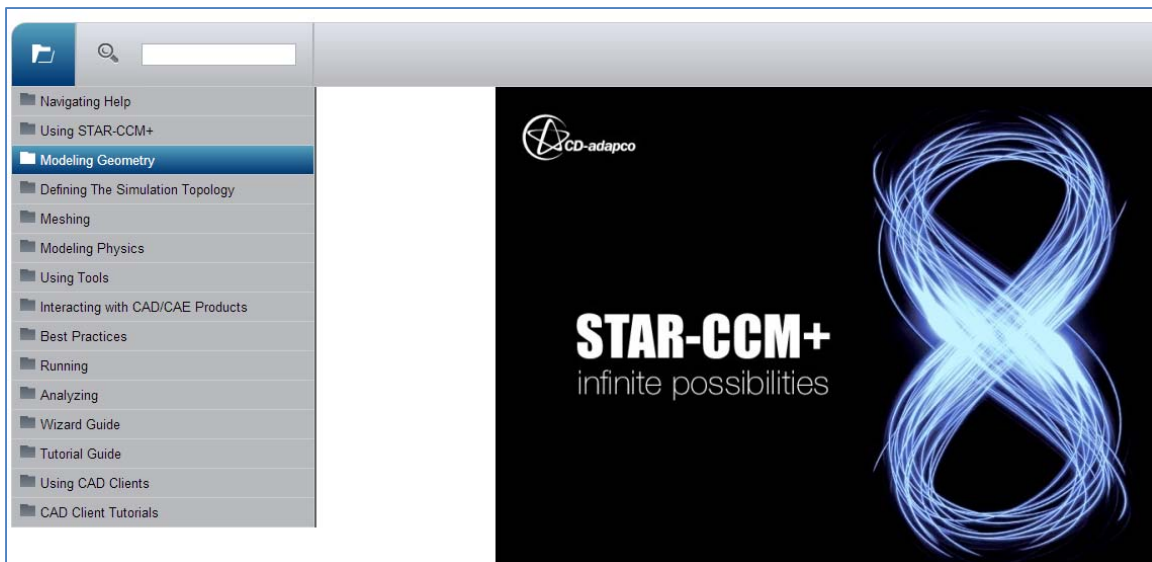

Figure 3.3.2B: Online user guide document of CD Adapco website

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 5**

On line classes are fully available and accessible from service representative. The company does offer free webinar and some discount for academic clients, and also there is no barrier to access any resources. Of particular benefit to the ERDL-HNEI CFD team, the company offers free attendance to all online training courses. This is unparalleled to any other CFD software products that the team has reviewed.

Training

FEATURED TRAINING NEAR YOU | MY UPCOMING COURSES | MY PAST COURSES

COURSE NAME	LOCATION	DATE/TIME
STAR-Tutor Interactive/ 1-hr Introduction to STAR-CCM+ - 9501	Other	Sep 09, 2013
Basic /STAR-CCM+ 3 Day Training - 9101	Detroit	Sep 09, 2013
Computational Engineering/ Aeroacoustics - 9312	Other	Sep 10, 2013
Computational Engineering/ Effective Heat Transfer Modeling - 9305	Seattle	Sep 12, 2013
Computational Engineering/ Process Automation through JAVA Scripting - 9306	Other	Sep 12, 2013

Figure 3.3.2C: Example of training schedule from CD Adapco website

Criterion 2.4:   Network of CFD professionals in architecture field: **ranking score = 5**

The ERDL-HNEI CFD team could identify a high-level supporting network of academics or professionals in architecture which uses Star CCM+ for external CFD applications. The STAR-CCM+ website offers an online forum where users address issues which are then answered by the technical support or are discussed by other users especially building service.

Criterion 2.5:   Ease of working with academic support: **ranking score = 5**

Star CCM+ is strongly supporting academic institutions, including faculty as well as students, to acquire the means of using the software for teaching and basic research. The company endeavors is actively

lower entry barrier of using the software at the academic institutions. Star CCM+ provides faculties and students with free or a much discounted access to the software for their academic endeavors.
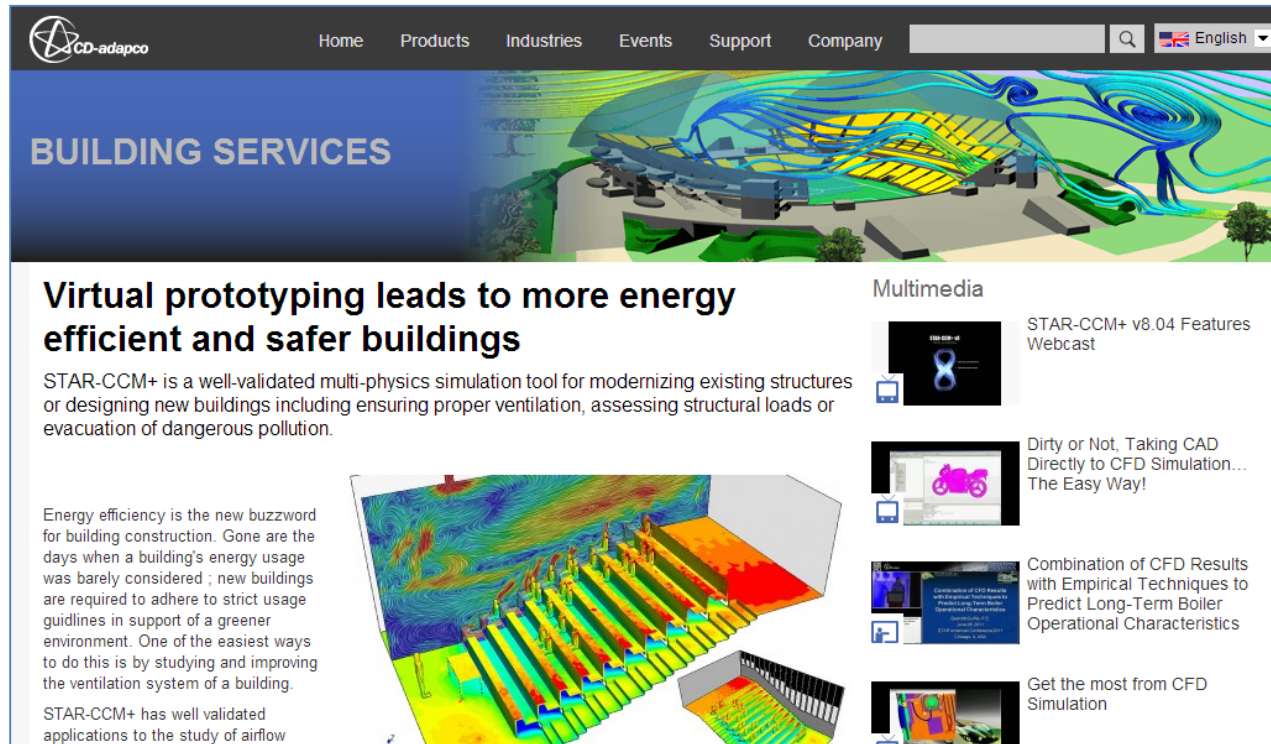

Figure 3.3.2D: Building service page from CD Adapco website

Criterion 2.6:   User guide provides efficient and comprehensive information: **ranking score = 5**

Star CCM+ user guide offers a wealth of application tutorial information and best practice guide in both how to use software and interpret the results of the software calculations. Any issues in the user guide are directly hyper-linked to specific task and GUI.

3.3.3 Criteria Group Workflow-Ease of Use / GUI:

Criterion 3.1.1: GUI is easy to use for trained operator: **ranking score = 4**

Star CC+ object tree or root tree provides a simply GUI of interface in Java script code. Star CCM+ avoids any complicated GUI element on the main tool bar or browser. However there are lots of background

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 63

data input and manipulation in the advance CFD application like Star CCM+, which doesn't demonstrate clearly on workflow. There is some level of difficulty for the less-than-expert user to find where to start and where to input numerical values for the hundreds of parameters, beyond the default set up.
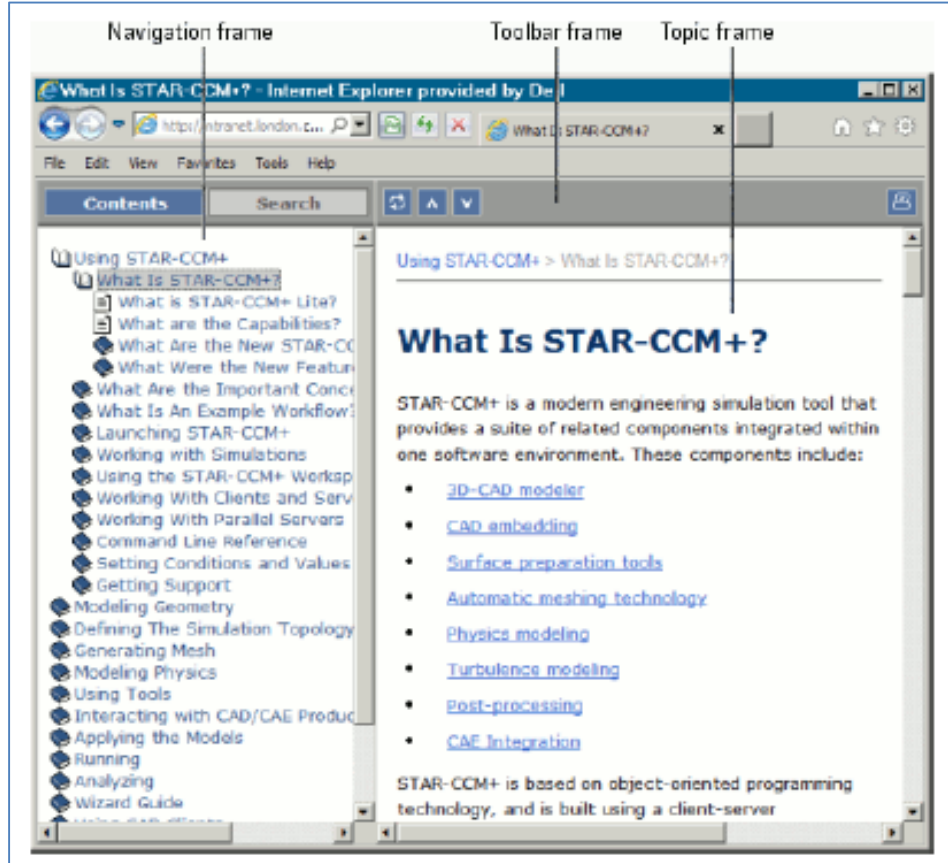

Figure 3.3.2E: Navigating Help from Star CCM+ user guide version 8.02
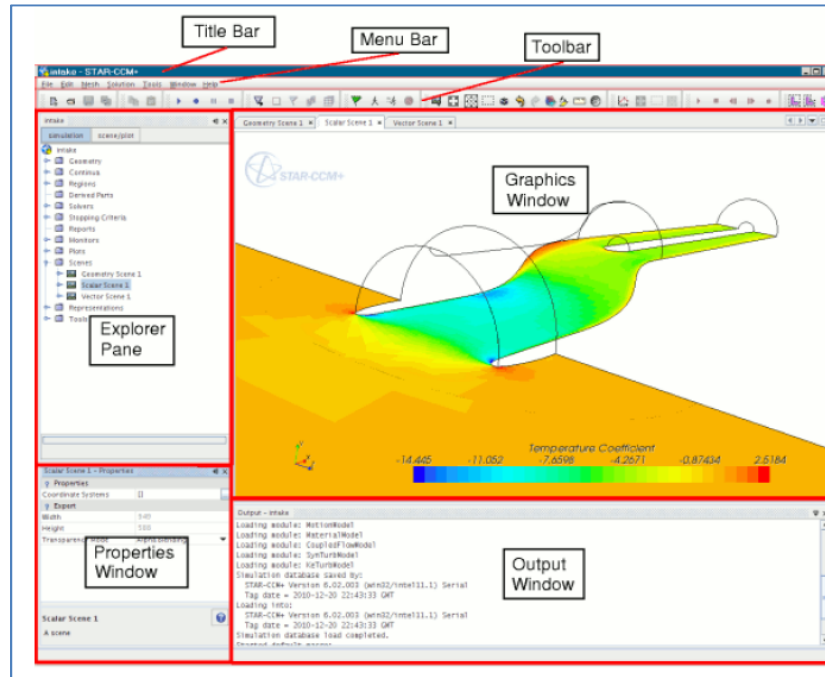
Figure 3.3.3A: Star CCM+ GUI from user guide version 8.02

<u>Criterion 3.1.2</u>: User guide and online help is available for each command: **ranking score = 5**

STAR-CCM+ launches an HTML browser at the title page of the online Help. User can also jump directly to other parts of the online Help using the same menu (Figure 4.3H). Users navigate directly to help topics that relate to specific objects in the simulation tree by pressing <F1> while the object is selected. For example, you could press <F1> on the Realizable K-Epsilon Two-Layer model node within the Models manager of a physics continuum and get the appropriate information.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 65
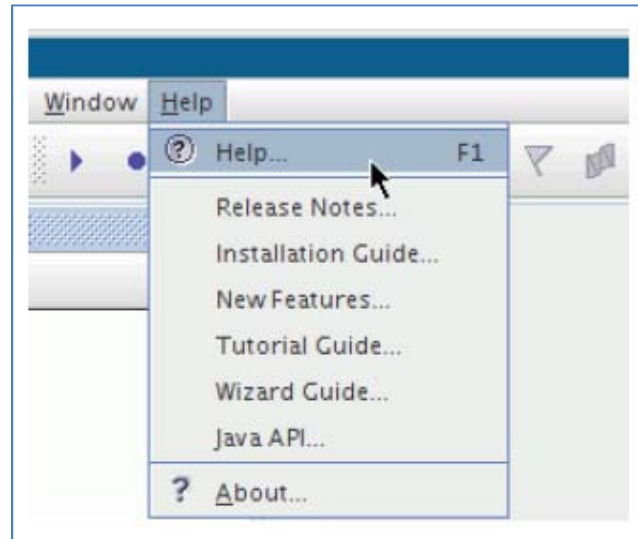
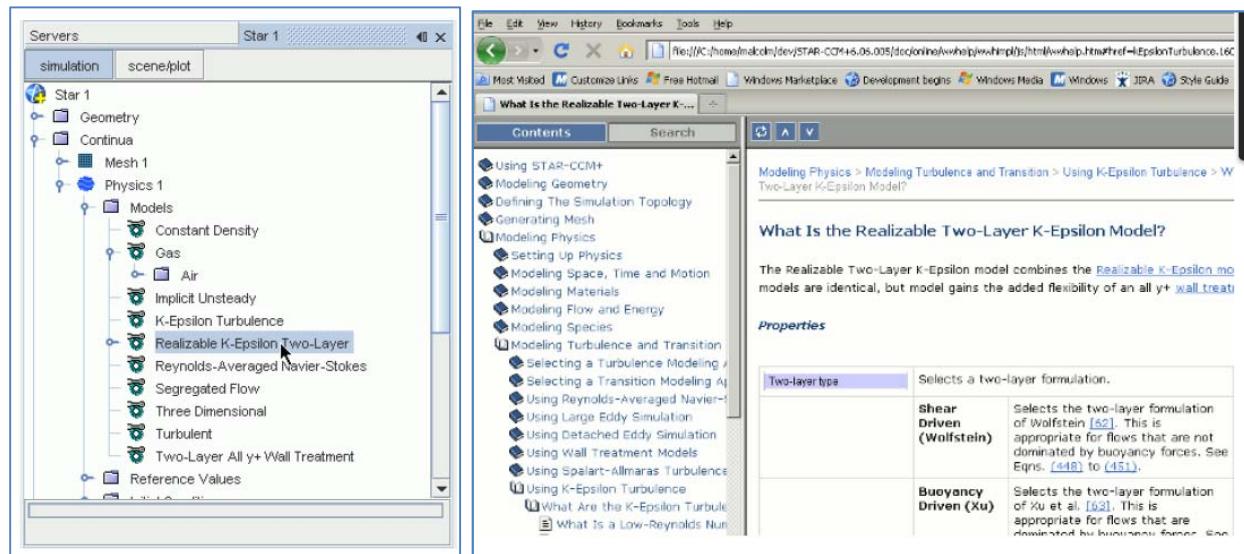Figure 3.3.3B: Star CCM+ browsing online help from user guide version 8.02

Figure 3.3.3C: Accessing context sensitive help from user guide version 8.02

3.3.4 Criteria Group Modeling / Import Geometry:

Criterion 3.2.1: Create geometry inside CFD application: **ranking score = 4**

The fully integrated CAD tool of StarCCM+ allows the user to build geometry from scratch within the STAR-CCM+ environment. 3D-CAD is tailored to suit the needs of a CFD analysis and allows the user to

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                   School of Architecture, University of Hawaii

December 4, 2013   Page 66

change the geometry quickly and easily, before rerunning the simulation. 3D-CAD is a feature-based parametric modeler within STAR-CCM+ that allows the user to create geometries from scratch. The geometries that the user creates with 3D-CAD are stored as 3D-CAD models, which can be converted to geometry parts for integration with the meshing and simulation process. However, there is some level of difficulty for the internal 3D-CAD which does not renders the geometry as the most comfortable to model building. Star CCM+ 3D-CAD modeling tool is using CAD modeling interface tool similar to Inventory and Solid work which the modeling interface are designed for object or machine. 3D-CAD navigator is not the most sophisticate tool. The CFD team found it a bit difficult to control view and rotate view., to name some examples of difficult CAD operations within Star-CCM+.
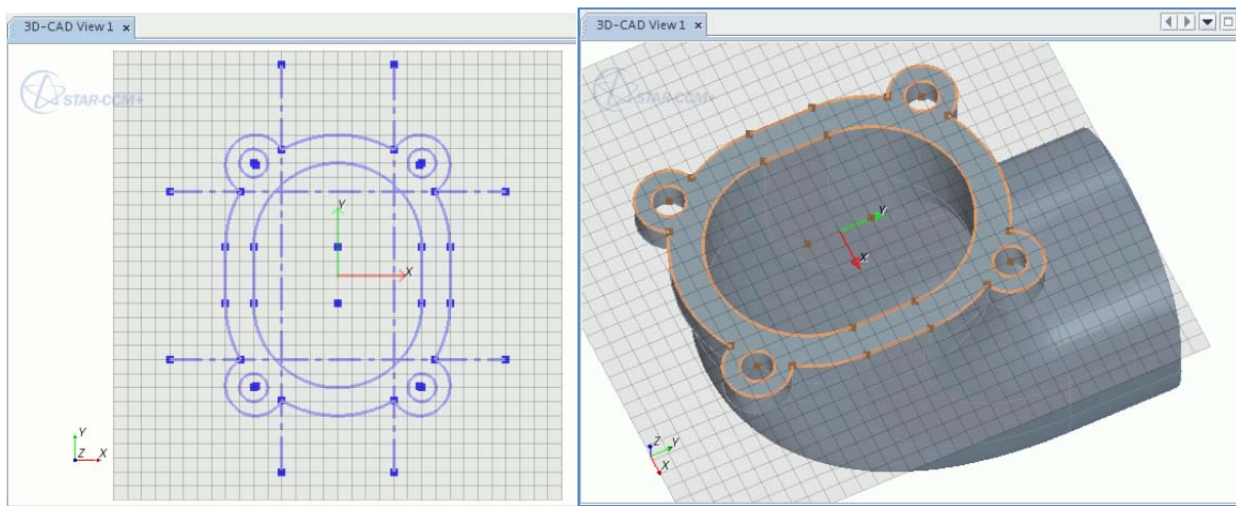
Figure 3.3.3D: 3D-CAD modeling tool from user guide version 8.02

Criterion 3.2.2: Import geometry / repair surface before meshing: **ranking score = 5**

Star CCM+ 3D-CAD allows user to import geometry from many different formats and modify it ahead of running a simulation. This ability facilitates the simulation process by allowing the user to define complicated geometries, extract an internal or external volume, and set up boundary names on the CAD model. At the heart of STAR-CCM+ is an automated process that links a powerful surface wrapper to CD-adapco's unique meshing technology. The surface wrapper significantly reduces the number of hours spent on surface cleanup and, for problems that involve large assemblies of complex geometry parts, reduces the entire meshing process to hours instead of days.
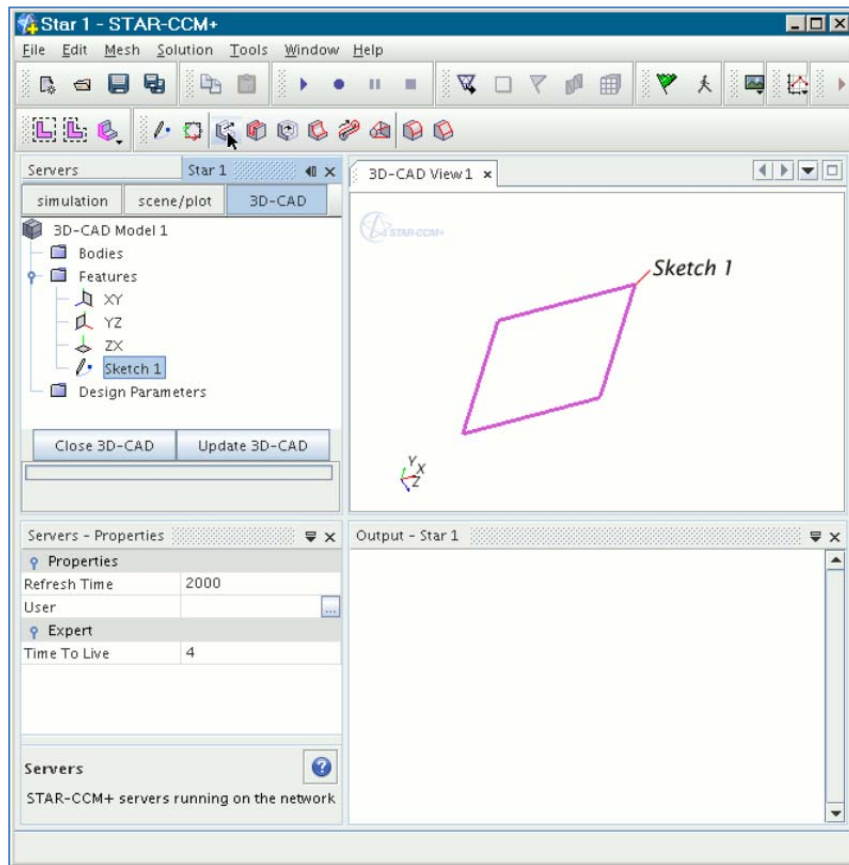
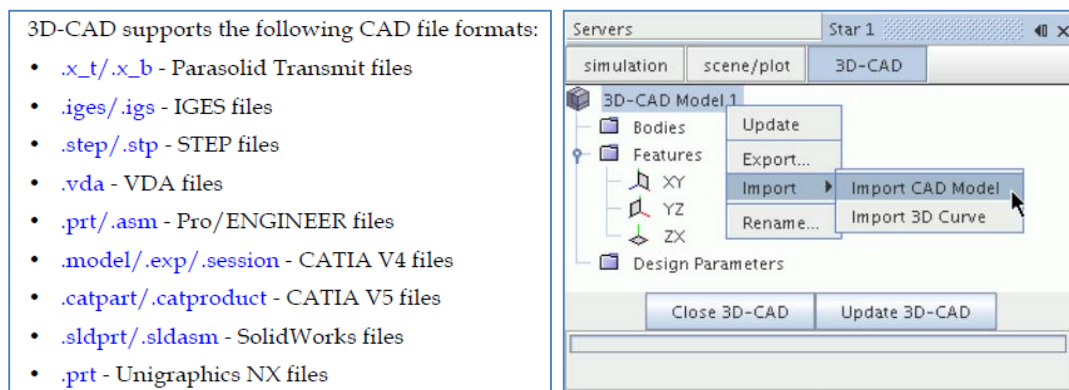Figure 3.3.3E: 3D-CAD modeling interface from user guide version 8.02

Figure 3.3.3F: Importing 3D-CAD file types from user guide version 8.02

Criterion 3.2.3: Support Revit Architecture: **ranking score = 1**

Star CCM+ doesn't provide direct Revit architecture import. It possibly needs to export Revit model to CAD Inventor file exchange format and then import the Inventor file into Star CCM+. However the quality of geometry surfaces still unclear.

Criterion 3.2.4: Support CADD format and Inventor: **ranking score = 5**

The type of native CAD file that can be readily imported into Star-CCM+ is dependent on the hardware that the user is using for running the STAR-CCM+ client. All the below listed CAD file formats support single or multiple region definitions in a single file. When importing multiple files as regions, the choice of generating either a single region per file or multiple regions is provided. None of the native CAD formats support feature curves (line cells) but these can be automatically created based on an angle or all feature edges during the import process if desired.

File Format	File Extensions	Versions Supported	Add-on Required	Type of Import
CATIA Graphics Representation	.cgr	R4 to R22	CAD Exchange	B-rep
CATIA V4	.model, .exp, .session	Up to 4.2.5	CAD Exchange	B-rep
CATIA V5	.CATPart, .CATProduct	R4 to R21	CAD Exchange	B-rep
3D XML	.3DXML	Up to 2012	CAD Exchange	Triangulated Data
SolidWorks	.sldprt, .sldasm	Up to 2012	CAD Exchange	B-rep
JT Open	.jt	Up to 9.5	JTOpen Reader & CAD Exchange	Triangulated Data & B-rep
NX	.prt	Up to NX 8.0	CAD Exchange	B-rep
Parasolid	.x_t, .xmt_txt, .x_b, .xmt_bin	Up to v25	None	B-rep
Pro/E	.asm, .prt	Up to Wildfire 5	CAD Exchange	B-rep
Inventor	.ipt, .iam	Up to 2013	CAD Exchange	Triangulated Data
IGES	.igs, .iges	5.1, 5.2, 5.3	None	B-rep
STEP	.stp, .step	AP 203 E1/E2, AP 214	None	B-rep
Stereo Lithography	.stl		None	Triangulated Data
VDA-FS	.vda	1.0 & 2.0	CAD Exchange	B-rep

Figure 3.3.3G: Supported file formats and corresponding version numbers from user guide version 8.02

STAR-Inventor provides the user with straightforward access to CD-adapco's STAR-CCM+ solver from within the Inventor environment. Standard physics models, including turbulence, ideal gas, heat transfer and gravity are supported for single body cases. Multiple body simulations involving conjugate heat transfer, rotating reference frames and porous media are also supported. Advanced physics models can be added to the simulation by either launching STAR-CCM+ and selecting them directly, or by implementing additional Java macro commands.
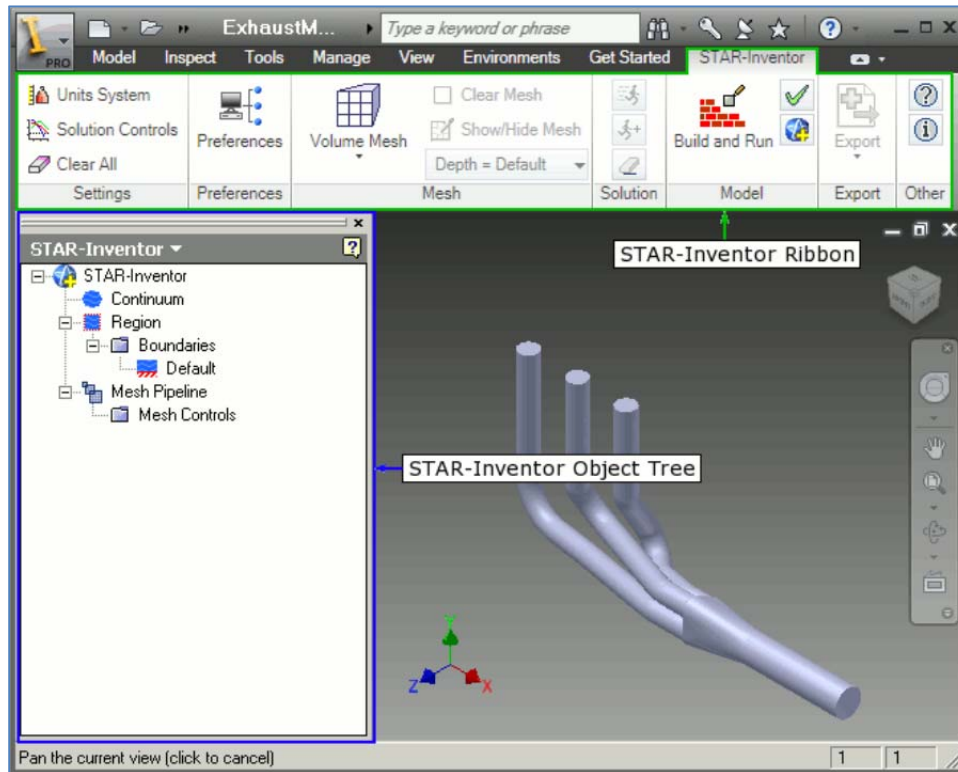

Figure 3.3.3H: STAR-Inventor interface from user guide version 8.02

Criterion 3.2.5: Support Rhino: **ranking score = 4**

Star CCM+ does not allow direct import of Rhinoceros models. There is possibility that export Rhinoceros to ".igs or .stl" file format then import into Star CCM+. However the quality of geometry surfaces still unclear.

3.3.5 Criteria Group Workflow- Meshing ison

<u>Criterion 3.3.1:</u> Surface meshers: **ranking score = 5**

Star CCM+ provides several surface mesher types (i.e. Surface Wrapper, Surface Remesher, and Automatic Surface Repair) To decide which surface meshing approach to use in preparation for volume meshing, weigh up the level of fidelity that is required against the acceptable level of manual effort. For example, in cases where it is important that there is little or no loss of surface detail, the surface remesher would be used to maintain the original features. In cases where a significant level of de-featuring is required, and there are many gaps and mismatches between surfaces and parts, the surface wrapper would be used.

The automatic surface repair tool provides an automatic procedure for correcting a range of geometric type problems that can exist in the remeshed surface once the surface remeshing process is complete. Up to three different metrics are used to determine if automatic fixes are required:
- Pierced Faces (surface intersections including opposing intersections)
- Surface Proximity
- Surface Quality

Figure 3.3.3I: Comparison of surface wrapper and surface meshers from user guide version 8.02

<u>Criterion 3.3.2:</u> Maximum cells and 2D- 3D cell volume: **ranking score = 5**

STAR-CCM+ is based on object-oriented programming technology. The CFD code is designed to handle large models quickly and efficiently using a unique client–server architecture that seamlessly meshes and simultaneously solves and post-processes over multiple computing resources without requiring additional effort from the user. STAR-CCM+ recently became the first commercial CFD package to mesh and solve a problem with ***over one billion cells.***
http://www.isc-events.com/cloud13/cfd-in-the-cloud-recent-activity-with-star-ccm.html

In STAR-CCM+, cells can have any number of faces as long as there are enough faces to create a closed cell volume. The faces of a cell cannot intersect each other, except where they touch along the common edges. In three dimensions, cells of arbitrary polyhedral shape are permitted. In two dimensions, cells of arbitrary polygons are permitted.

<u>Criterion 3.3.3:</u> Boundary layer types: **ranking score = 5**

Boundaries are surfaces (or lines in a two-dimensional case) that completely surround and define a region. The figure below shows a simple region where the lines surrounding the region represent the boundaries. In this example, the region could be surrounded by a single boundary or multiple boundaries. The choice depends on what conditions and values need be assigned. Star CCM+ demonstrates well on boundary types which is the significant important to assigned the surface function.
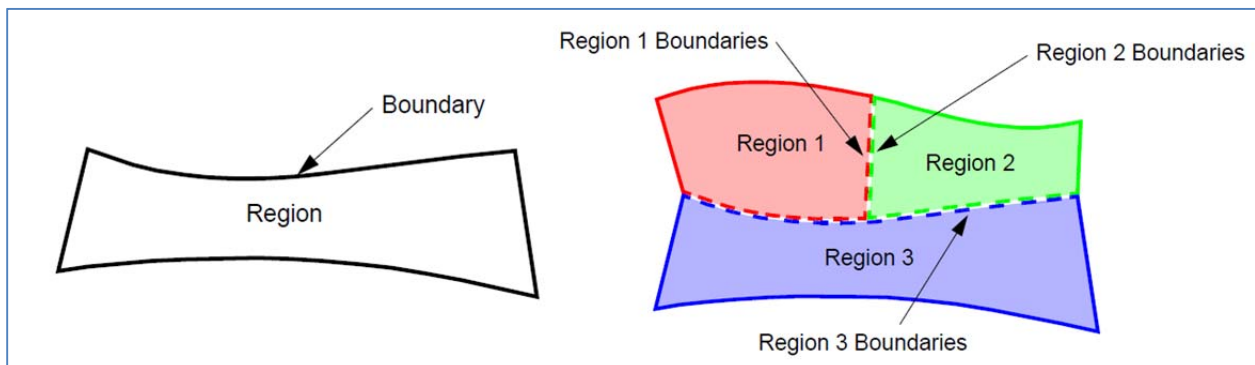

Figure 3.3.3J: Concept of boundaries from user guide version 8.02

|	
Axis ◀◀◀	The axis of an axisymmetric region. The two icons are for a fluid and solid region respectively.
Flow Split Outlet ▶»	A flow split outlet boundary.
Free-Stream ●	A free-stream boundary.
Mass Flow Inlet »»	A mass flow inlet.
Pressure Outlet ▦	A pressure outlet.
Stagnation Inlet ▶	A stagnation inlet.
Symmetry Plane ▨	A symmetry plane.
Velocity Inlet »»	A velocity inlet.
Wall ▨	A wall.
Displacement ▨	A solid stress displacement boundary
Traction ▨	A solid stress traction boundary
Fixed ▨	A solid stress fixed boundary
Free ▨	A solid stress free boundary
Pressure ▨	A solid stress pressure boundary
Normal Displacement ▨	A solid stress normal displacement boundary
Force ▨	A solid stress force boundary
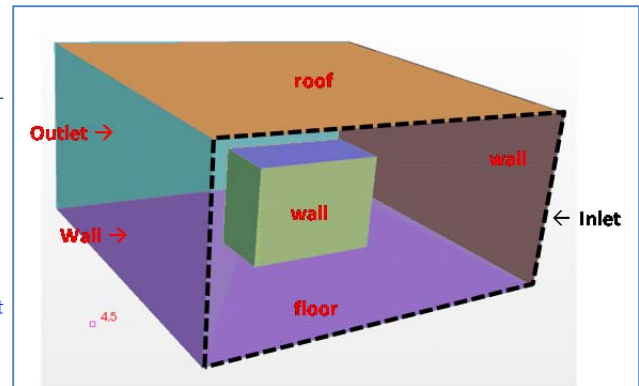
Figure 3.3.3K: Boundary types and sample of assigned boundaries from user guide version 8.02

Criterion 3.3.4: Structured and Unstructured meshing: **ranking score = 5**

A mesh is the discretized representation of the computational domain, which the physics solver uses to provide a numerical solution. STAR-CCM+ provides meshers and tools that can be uses to generate a quality mesh for various geometries and applications. STAR-CCM+ provides several meshing strategies that are suitable for different applications. STAR-CCM+ provides a high ranking in both structured and unstructured meshing strategies for user purpose.

Unstructured mesher
- **Trimmed Mesher:**
  Generates a volume mesh by cutting a hexahedral template mesh with the geometry surface.
- **Polyhedral Mesher:**
  Generates a volume mesh that is composed of polyhedral-shaped cells.
- **Tetrahedral Mesher:**
  Generates a volume mesh that is composed of tetrahedral-shaped cells.
- **Prism Layer Mesher:**
  Adds prismatic cell layers next to wall boundaries. The mesher projects the core mesh back to the wall boundaries to create prismatic cells.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 73

- **Advancing Layer Mesher:**
  Creates a volume mesh composed of prismatic cell layers next to wall boundaries and a polyhedral mesh elsewhere. The mesher creates a surface mesh on the wall and projects it to create the prismatic cell layers.
- **Extruder Mesher:**
  Generates an extruded mesh region from a boundary that one of the core volume meshers has meshed.
- **Thin Mesher:**
  Generates a prismatic layered volume mesh for thin geometries.
- **Generalized Cylinder Mesher:**
  Generates a volume mesh appropriate for elongated cylindrical regions.

Structured mesher
- Directed Meshing:
  Generates a mesh by creating a two-dimensional patch mesh on a surface and sweeping it through the volume along a specified guide surface.
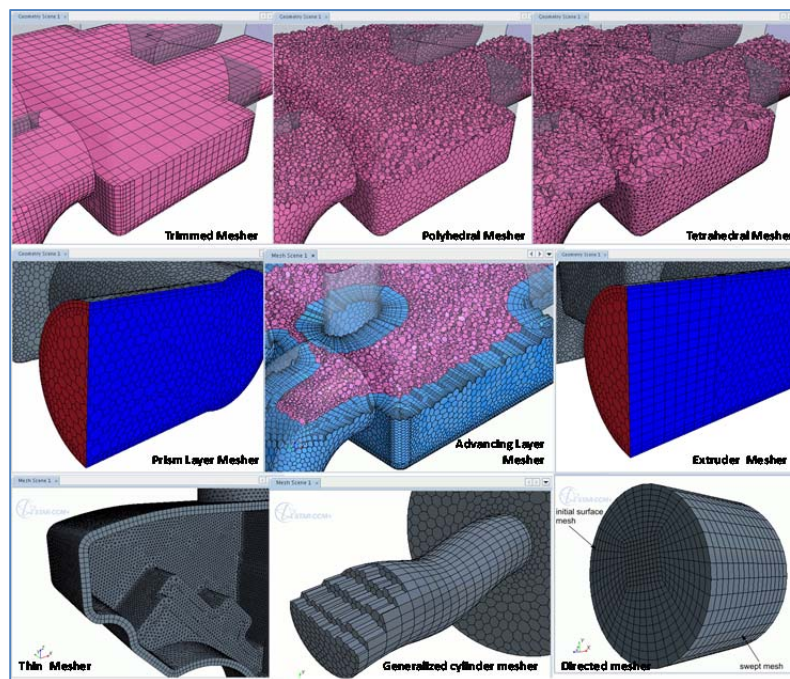

Figure 3.3.3L: Example of meshing strategies from user guide version 8.02

Criterion 3.3.5: Adaptive meshing capability: **ranking score = 5**

STAR-CCM+ does not provide a special function of "Adaptive meshing" but the functionality of Star-CCM+ Advancing Layer Mesher can provide the same functionality. The advancing layer mesher

generates layers of prismatic cells around the surfaces of regions and fills the remaining void with polyhedral cells. This mesher first generates a polygonal surface mesh and then advances this mesh into the region volume to form cell layers. Some advantages of this approach are a conformal match on either side of an interface, and the ability to generate thicker and more uniform cell layers.
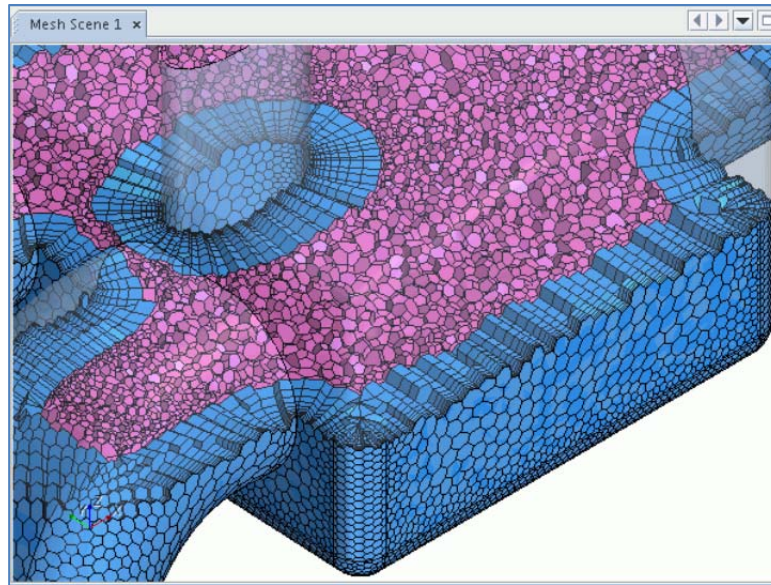

Figure 3.3.3M: Example of an advanced layer mesh shows the prismatic layers are colored blue and the core mesh is colored pink from user guide version 8.02.
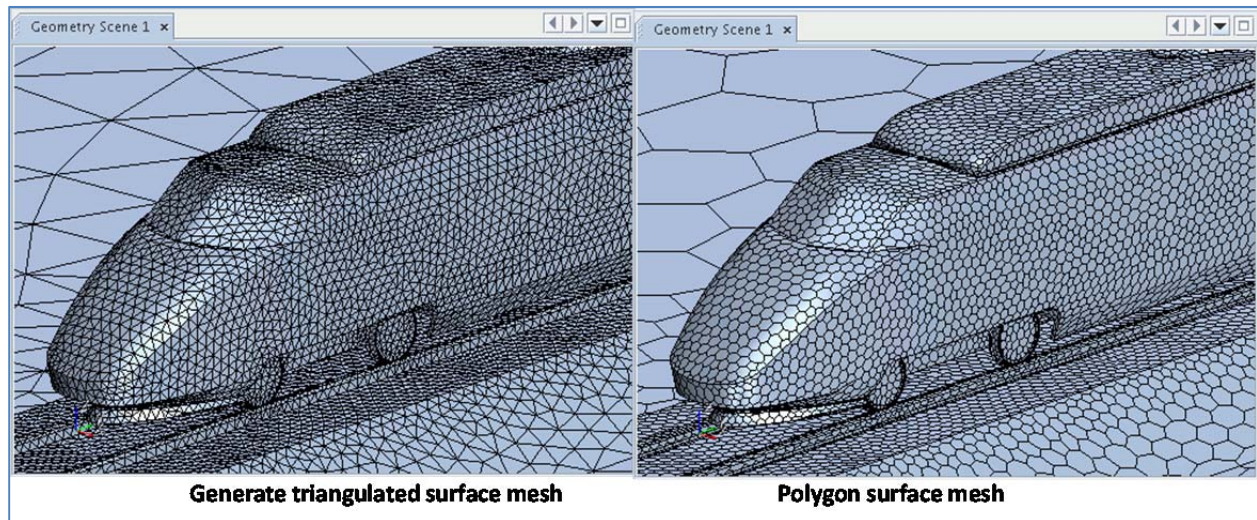

Generate triangulated surface mesh   Polygon surface mesh

Figure 3.3.3N: Example of an advancing layer mesh shows the prismatic layers generating process from user guide version 8.02.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 75

In the first step, the advancing layer mesher generates a surface mesh from the triangulated surface in Figure 4.3.3N-Left (provided by the Surface Remesher or Surface Wrapper). The advancing layer mesher takes this surface and generates the surface mesh using triangles, quadrilaterals, or polygons as specified in the surface element type property. An example of a polygon surface mesh is shown in figure 4.3.3N-Right. There is no instruction and description regarding a capability to maintain or handle complexity of model and geometry without excessive increase in computational effort.

However, STAR-CCM+ has described on the Parallel Meshing which is a Multiple processors can be used to speed up the volume mesh generation. As a prerequisite, the simulation in which the user wishes to use parallel meshing must be started in parallel mode. For an average mesh (10 million cells or less), the user can gain most advantage by using 4 - 8 processors. Using a high number of processors for a relatively small mesh reduces scalability and results in slower performance. As a general guideline, 1 - 2 million cells per core gives good performance for parallel meshing. Generally, the user might opt for a higher number of cores for running the simulation—a distribution of 250,000 cells per core is appropriate. This difference means that the user can restart the parallel server with a higher number of cores before running the simulation. "*Reduce geometry preparation and meshing time from months to hours*" http://www.cd-adapco.com/products/advanced-meshing

3.3.6 Criteria Group Workflow - Physics Models:

<u>Criterion 3.4.1:</u> Solvers – Coupled / decoupled: **ranking score = 5**

STAR-CCM+ has uses different terms to describe coupled and decoupled domains. The StarCCM+ refers to them as *coupled flow model, coupled energy model, and coupled Species model* in the solvers.

The Coupled Flow model solves the conservation equations for mass and momentum simultaneously using a time- (or pseudo-time-) marching approach. The preconditioned form of the governing equations that the Coupled Flow model uses makes it suitable for solving incompressible and isothermal flows. One advantage of this formulation is its robustness for solving flows with dominant source terms, such as rotation. Another advantage of the coupled solver is that CPU time scales linearly with cell count; in other words, the convergence rate does not deteriorate as the mesh is refined.

The Coupled Energy model is an extension of the Coupled Flow model. Together they solve the conservation equations for mass, momentum, and energy simultaneously using a time- (or pseudo-time-) marching approach. This formulation is robust for solving compressible flows and flows with dominant source terms, such as buoyancy. Due to the preconditioned form of the governing equations that the Coupled Flow and Coupled Energy models use, convergence rate is effectively independent of Mach number, ranging from incompressible through to supersonic regimes.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 76

The Coupled Species model is an extension of the Coupled Flow model. Together with the Coupled Energy model, it solves the conservation equations for mass, momentum, and energy of a multi-component fluid mixture simultaneously using a time- (or pseudo-time-) marching approach. For a mixture of N components, the Coupled Species model introduces N-1 transport equations into the system. Together with global mass continuity, these equations provide a means for updating the field of N mass fractions defining the mixture composition. Inter-equation coupling is fully maintained between all equations in the system, including species transport with respect to momentum and energy, and vice versa. For isothermal simulations of multi-component fluid mixtures, it is possible to omit the Coupled Energy model, in which case the Coupled Species model, together with the Coupled Flow model, solves the conservation equations for mass and momentum simultaneously.

Criterion 3.4.2: Solver-Finite Element Method (FEV) / Finite Volume Method (FVM): **ranking score = 5**

STAR-CCM+ uses the finite volume method, where the solution domain is subdivided into a finite number of small control volumes, corresponding to the cells of a computational grid. Discrete versions of the integral form of the continuum transport equations are applied to each control volume. The objective is to obtain a set of linear algebraic equations, with the total number of unknowns in each equation system corresponding to the number of cells in the grid. (If the equations are non-linear, iterative techniques that rely on suitable linearization strategies must be employed.) The resulting linear equations are then solved with an algebraic multigrid solver.

Criterion 3.4.3: Steady state/Transient state**:**   **ranking score = 5**

Modeling Capabilities		STAR-CCM+ Lite	STAR-CCM+
3D-CAD	Parametric geometry modeling	✓	✓
	Native CAD import (requires an appropriate add-on license)	✓	✓
Space	2D/Axisymmetric	✓	✓
	Cyclic/periodic symmetry	✓	✓
	3D	✓	✓
Time	Steady state	✓	✓
	Transient	✓	✓

Figure 3.3.4A: The relative capabilities of STAR-CCM+ and STAR-CCM+ Lite are given from user guide version 8.02.

STAR-CCM+ has solved steady-state as well as transient calculations. When the steady-state model is activated, the concept of a physical time-step is meaningless. Therefore, for those objects that offer a choice between Iteration or Time-Step for a trigger (such as Monitors or Scenes), the Time-Step option should not be activated since it will result in no-update occurring.

In most situations, the physics being simulated will clearly indicate the choice between the Steady and Unsteady (Implicit or Explicit) models. Simulations that are obviously time-dependent, and hence require the Unsteady model include:
- Time-varying boundary conditions
- Sliding or deforming mesh problems
- Free surface (VOF) problems
- Transient heat transfer

Transient analysis in STAR-CCM+ is used to solve time-accurate problems, including initial value problems. Two different time integration schemes are available for transient analysis:
- Implicit Unsteady
- Explicit Unsteady

Criterion 3.4.4: Turbulence models**:**   **ranking score = 5**

In addition to its provision for invisid and laminar flow, STAR-CCM+ has a comprehensive range of turbulence models:
- k-epsilon (Standard, V2F, Realizable, Two-layer)
- k-omega (Standard, SST and BSL)
- Reynolds Stress (RSM – linear and quadratic)
- Spalart-Allmaras Turbulence models
- Boundary-layer transition
- Large Eddy Simulation (LES)
- Detached Eddy Simulation (DES, in the new Delayed Detached Eddy Simulation or DDES formulation)

Criterion 3.4.5: Materials**:**   **ranking score = 5**

The material model is responsible for managing the material, that is, substance or substances, being simulated in the continuum. The material, in turn, is responsible for managing the various thermodynamic and transport properties relevant to that material and to the physical processes being modeled in the continuum. There are three general types of material models available in STAR-CCM+: single-component, multi-component, and multiphase mixture. These material models each manage one specific type of material: a pure substance, a multi-component mixture, or a multiphase mixture.
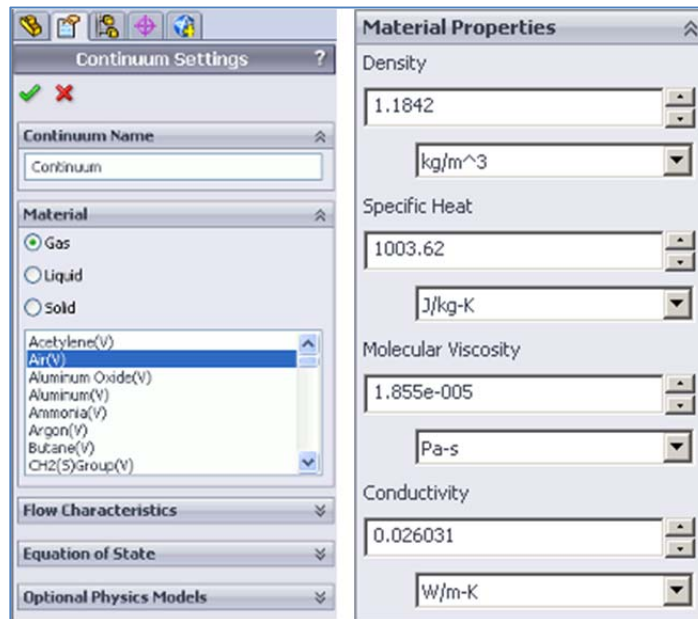
Figure 3.3.4B: STAR_CCM+ material properties from user guide version 8.02.

In STAR-Works each continuum in the model is assigned a material. This defines the material properties used by the CFD solver when the case is run. The choice of material will affect the options available for the model physics. For example, a material from the Gases or Liquids lists is selected, and the user is offered the opportunity to specify whether the flow is turbulent or laminar. By default the material type is set to Air, and this can be changed to any one of a number of predefined solids, liquids or gases that are available in STAR-Works. If necessary, it is possible to define an entirely different material by entering different material properties.

Criterion 3.4.6: Customizable / special physics models**:**   **ranking score = 4**

STAR-CCM+ offer some special physics modeling include:
Compressibility
- Ideal Gas
- Real Gas
Heat transfer
- Conjugate Heat Transfer (CHT)
- Multiband and gray thermal surface to surface radiation
- Solar radiation
- Discrete ordinates radiation (DOM) including participating media
Multiphase
- Lagrangian particle tracking
- VOF (incompressible and compressible)
- Cavitation & boiling
- Eulerian multiphase

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 79

- De-icing & De-fogging
- Melting & solidification

Moving Mesh

- Dynamic Fluid Body Interaction (DFBI or 6DOF)
- Rigid body motion
- Mesh morphing
- Multiple reference frames (MRF)

Combustion & chemical reaction

- Reaction kinetics
- Eddy break up (EBU)
- Presumed probability density function (PPDF)
- Complex chemistry
- Ignition
- NOx modeling

Distributed Resistance (Porous media)

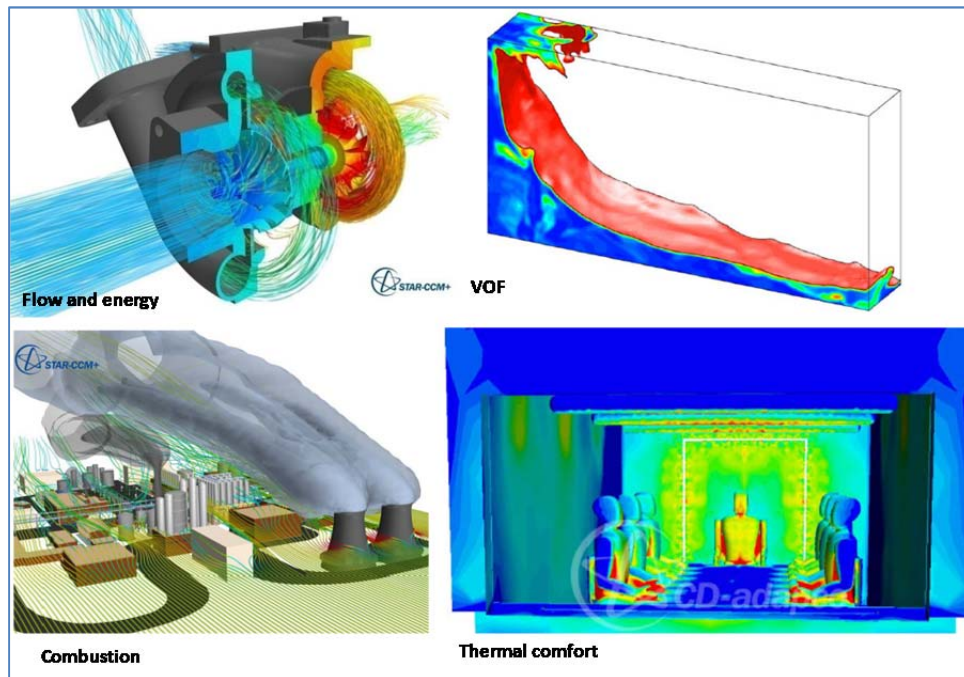- Anisotropic
- Orthotropic
- User defined
- Porous baffles

Figure 3.3.4C: Example of STAR_CCM+ special physics models from user guide version 8.02.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                              School of Architecture, University of Hawaii

December 4, 2013   Page 80

STAR-CCM+ does not offer a Humidity absorption / dispersion model; rather the user can opt to assess relative and absolute humidity in the advance calculation process for thermal comfort set up.

3.3.7 Criteria Group Workflow - Post-Processing:

Criterion 3.5.1: Visualization: **ranking score = 5**

Derived parts are used in STAR-CCM+ for the purposes of displaying, analyzing, and reporting simulation data. This section includes general information on working with derived parts, as well as specific information for the following derived parts:
- Implicit sections
- Scalar value-based parts
- Vector value-based parts
- Streamlines
- Vortex cores
- Separation/Attachment Parts
- Probes
- Topological parts
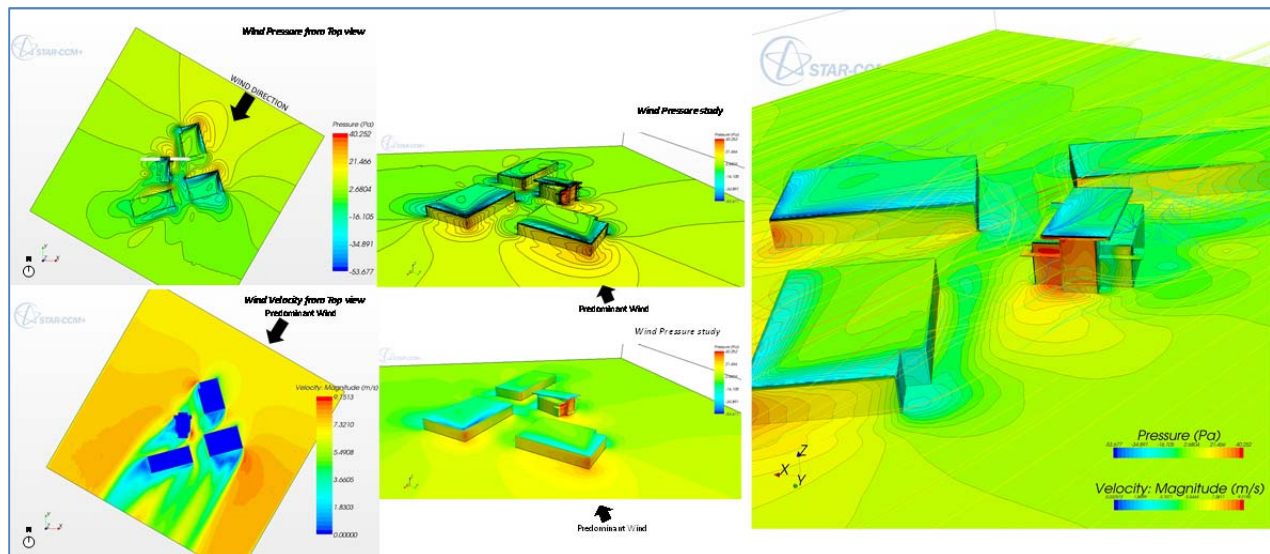
Figure 3.3.5A: Post processing / visualization on surface distribution and display streamline
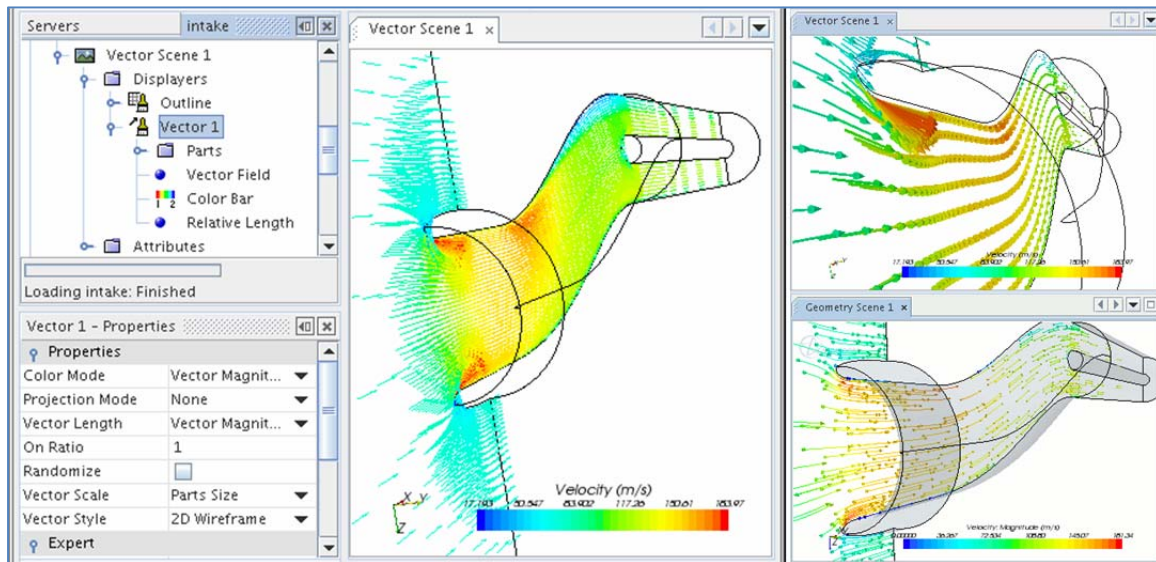
Figure 3.3.5B: Post processing / visualization on section display streamline from STAR-CCM+ user guide 8.02.

STAR-CCM+ has a comprehensive suite of post-processing tools designed to enable the user obtaining maximum value and understanding from the CFD simulation results. The visualization tools include scalar and vector scenes, streamlines, scene animation, numerical reporting, data plotting, import, and export of table data, and spectral analysis of acoustical data.
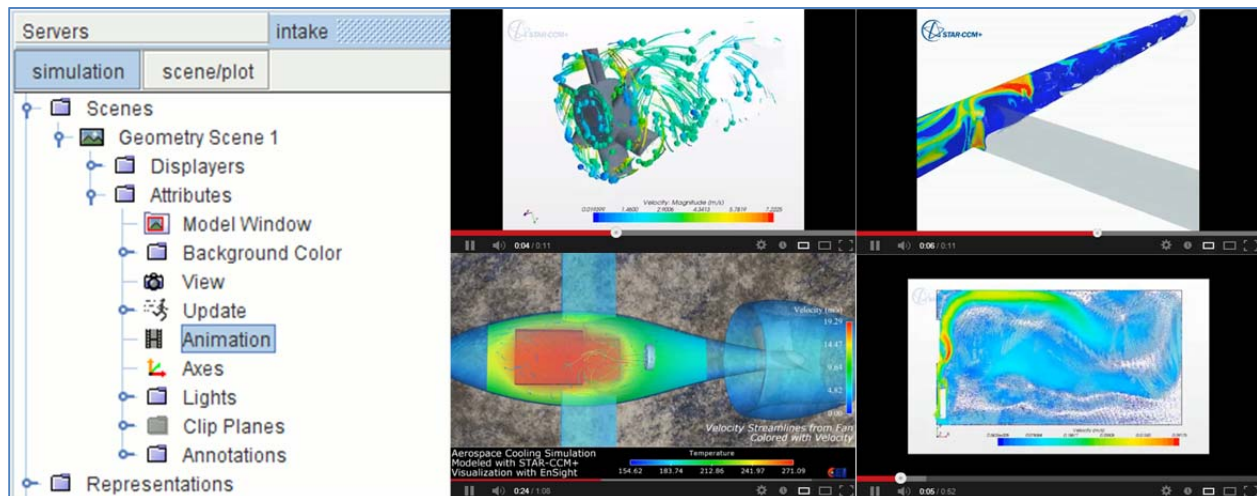

Figure 3.3.5C: Post processing / visualization on animation from youtube search engine

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 82

<u>Criterion 3.5.2:</u> Quantitative analysis capabilities of components**:**   **ranking score = 5**

STAR-CCM+ provides derived parts that take samples of data in a region based on coordinates or intervals that the user specifies. These parts are Point, Line-probe, Presentation grid, and Arbitrary probe. Users are able to create Plots which provide line graphs of information supplied from monitors, extracted simulation data, tables, or derived data sets. These can be customized with different line and point properties, axis scales, titles, and so on. The plotting features in STAR-CCM+ allow the user to create two kinds of two-dimensional plots:
- Monitor plots that display data from the simulation as it steps through the solution in two varieties:
  a. Monitor plots based on monitored quantities
  b. Residual plots (described elsewhere)
- XY plots that use solution data from the simulation and/or table data
- Histogram plots that display data, typically for particles or parcels

<u>Criterion 3.5.3:</u> Comparative analysis (multi-scenarios)**:**   **ranking score = 3**

STAR-CCM+ reports on consultants finding that STAR-CCM+ is unusually flexible. The user can easily make quick changes to the geometry, which enables the user to run multiple scenarios in a short period of time. If, as in the present CFD project, the user is studying labs and he has to make a change—say selecting a different diffuser—the process is very quick. StarCCM+ does not offer an actual interface to show multi-scenarios approach to visualize and compare different models result at the same time.
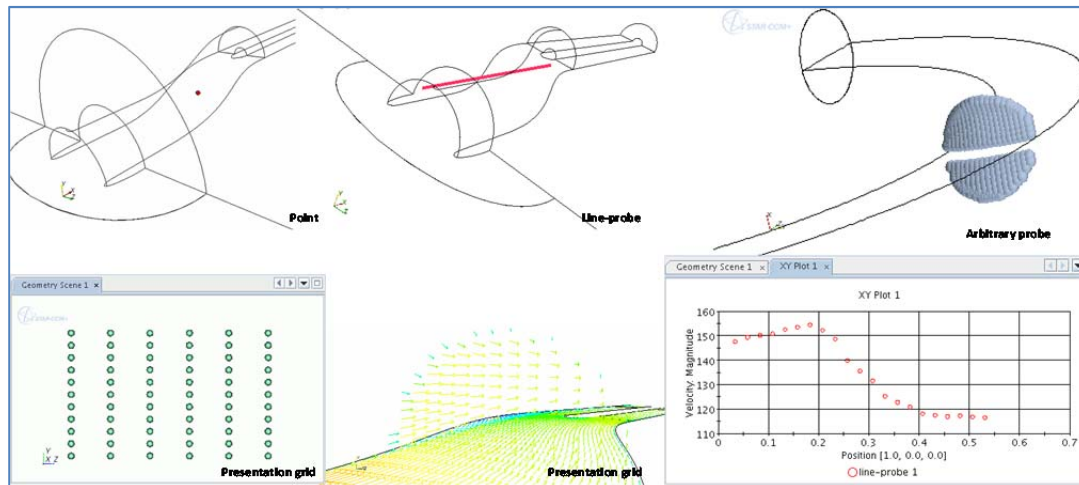

Figure 3.3.5D: Example of quantitative analysis capabilities from STAR-CCM+ user guide 8.02

3.3.8 Criteria Group Workflow - Customization / Special functions:

Criterion 3.6.1: Pedestrian comfort: **ranking score = 5**

STAR-CCM+ provides functionality regarding to the assessment of pedestrian comfort. The process basically emphasizes on wind velocity at ground level and defining a terrain factor.

Criterion 3.6.2: Natural ventilation: **ranking score = 5**

STAR-CCM+ is able to provide Building Services to examine the airflow patterns around a building, specifically examining the stress on a building caused by wind, and calculating the wind speed for pedestrians walking near the building. As more and more skyscrapers get built, and they are packed in more densely neighborhoods, these buildings begin to dramatically affect wind patterns through the neighborhood. The results of high winds in the urban environment can create dangerous situations, such as high wind speeds on a building, causing significant stress on a building's primary load beams and high windspeeds for pedestrians walking in between them. STAR-CCM+ is well validated to perform all types of wind speed simulations.
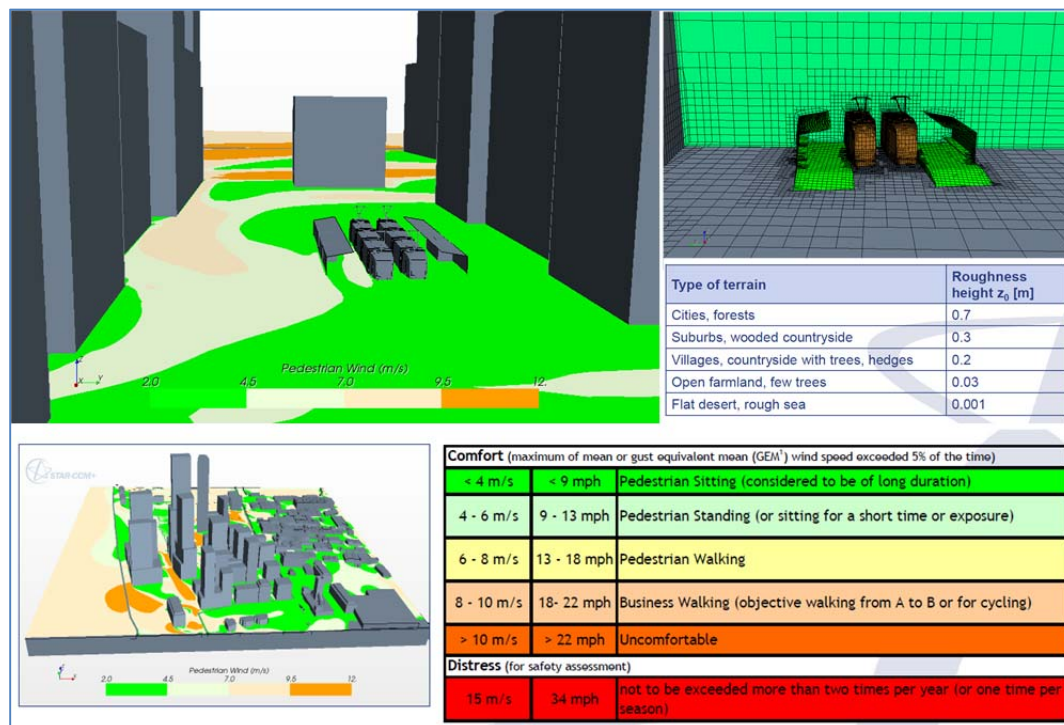

Figure 3.3.6A: Example of Pedestrian comfort from STAR-CCM+ from Building Services Analysis, 2011
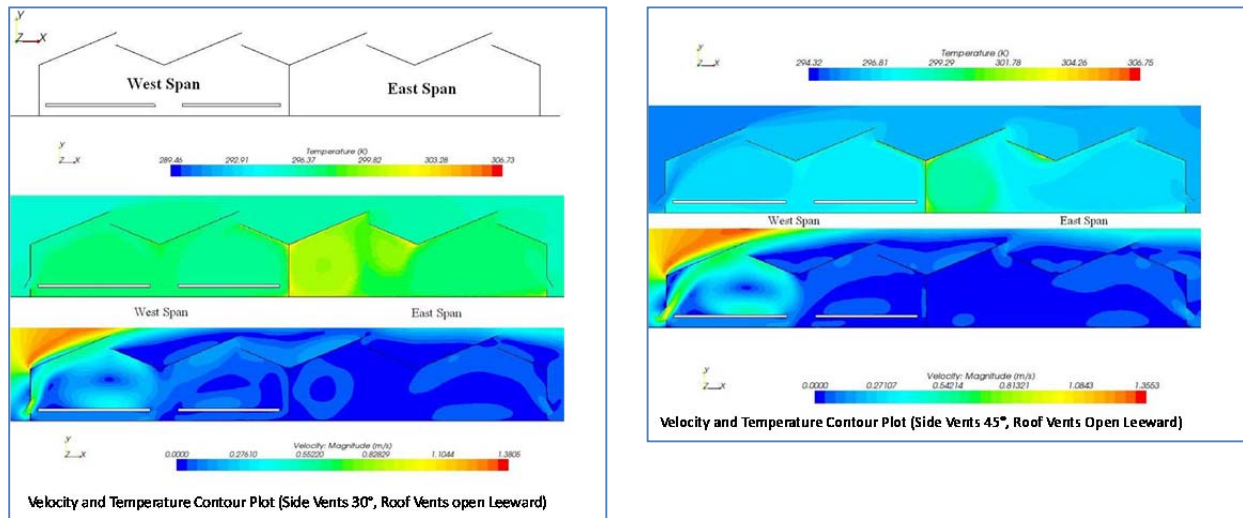
Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                       School of Architecture, University of Hawaii

December 4, 2013   Page 84

Figure 3.3.6B: Example of Natural ventilation of STAR-CCM+ CFD simulation (Kruger, 2006)

Criterion 3.6.3: Wind-driven rain (WDR): **ranking score = 5**
STAR-CCM+ involves a special function to model a substance such as air, water or aluminum, or several substances such as a combusting mixture of gases, or a sloshing mixture of air and water. Water is a liquid for, and the liquid material represents a single pure liquid substance. The Liquid model manages the material as outlined in the section describing single-component material models. The screenshot below shows the liquid model node and its "child" node H_2O which represents the pure liquid material (water) being modeled.
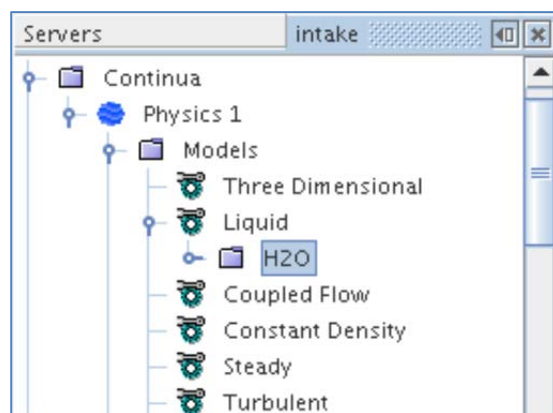

Figure 3.3.6C: Liquid model mode of STAR-CCM+ user guide version 8.02

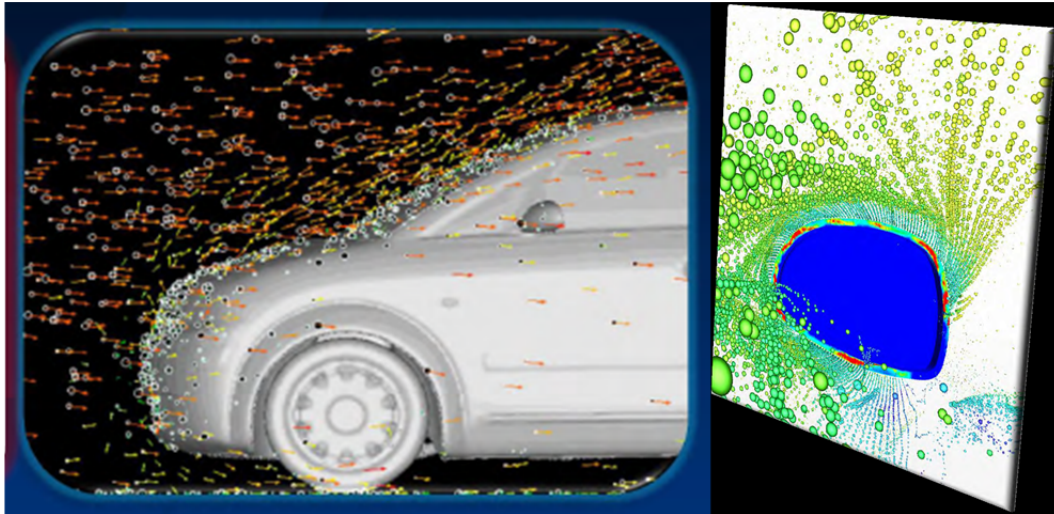Figure 3.3.6D: Modeling rain water / mist (CD-Adapco by Frederick J. Ross)

Criterion 3.6.4: Humidity: **ranking score = 3**

STAR-CCM+ doesn't offer the Humidity absorption / dispersion rather than take both in relative and absolute humidity in the advance calculation process for thermal comfort set up.
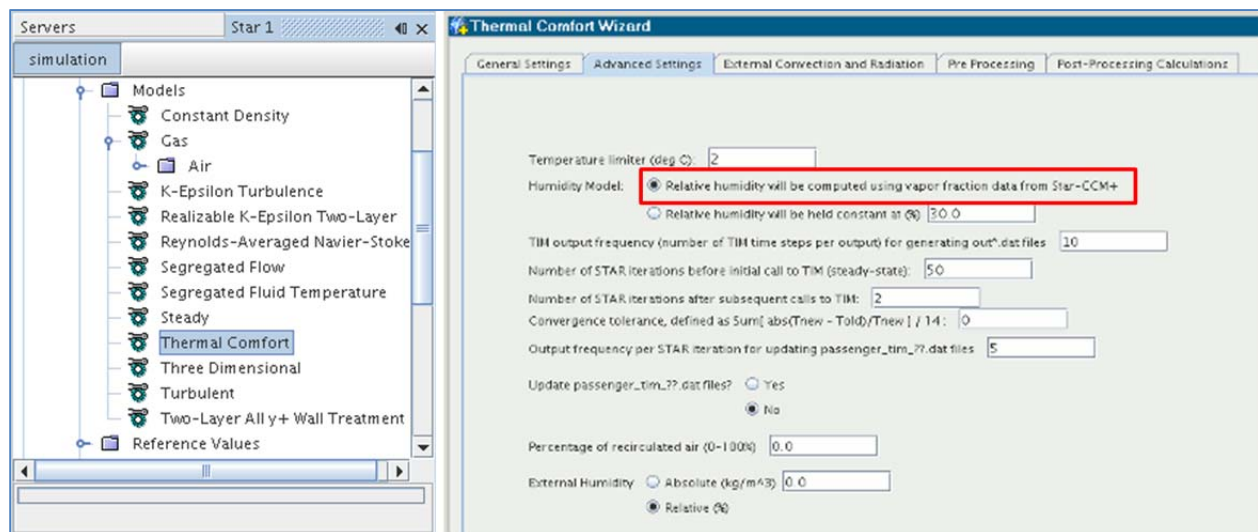

Figure 3.3.6E: Advance settings of STAR-CCM+ user guide version 8.02

<u>Criterion 3.6.5:</u> Wind load towards to building structure: **ranking score = 3**

STAR-CCM+ can determine and visualize wind velocity and wind pressure distribution on the building envelope. The software also has a tool to determine scalar measurement at node point, line, and grid on building façade. For example, Figure4.3.6F shows the velocity of air inside the gap, depending on the exterior wind around the building by considering a velocity of air exterior and perpendicular to façade. Figure 4.3.6G shows wind velocity and pressure distribution on building façade.
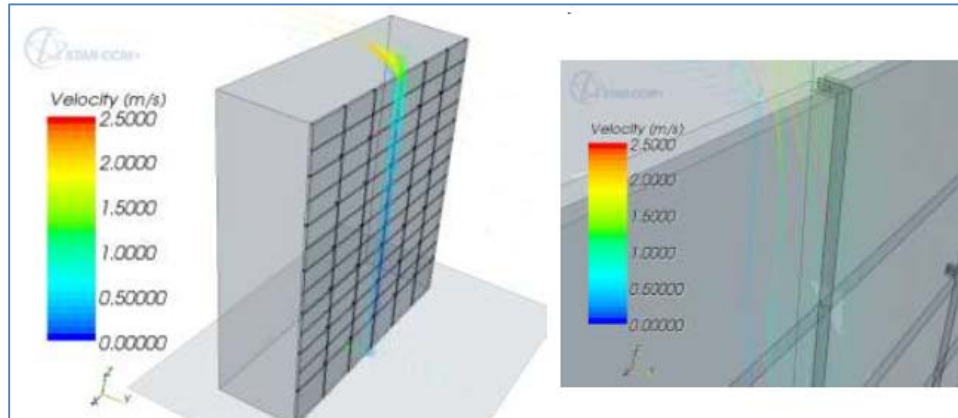

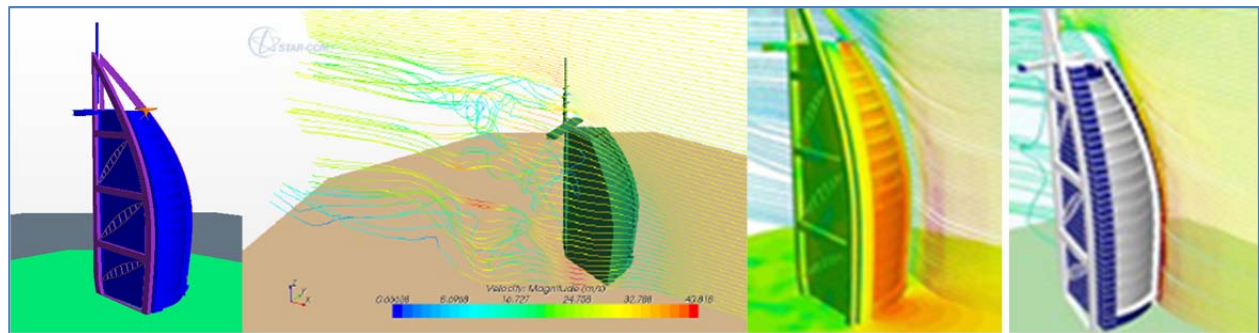Figure 3.3.6F: Velocity profile in medium line (López-Jiménez, 2010)

Figure 3.3.6G: Velocity and pressure distribution of STAR-CCM+ for Building Services Analysis, 2011

<u>Criterion 3.6.6:</u> Pressure distribution along streamlines/pressure losses: **ranking score = 2**

STAR-CCM+ doesn't provide the visualization of pressure losses distribution and streamlines. However there is the analysis manager on pressure loss.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 87
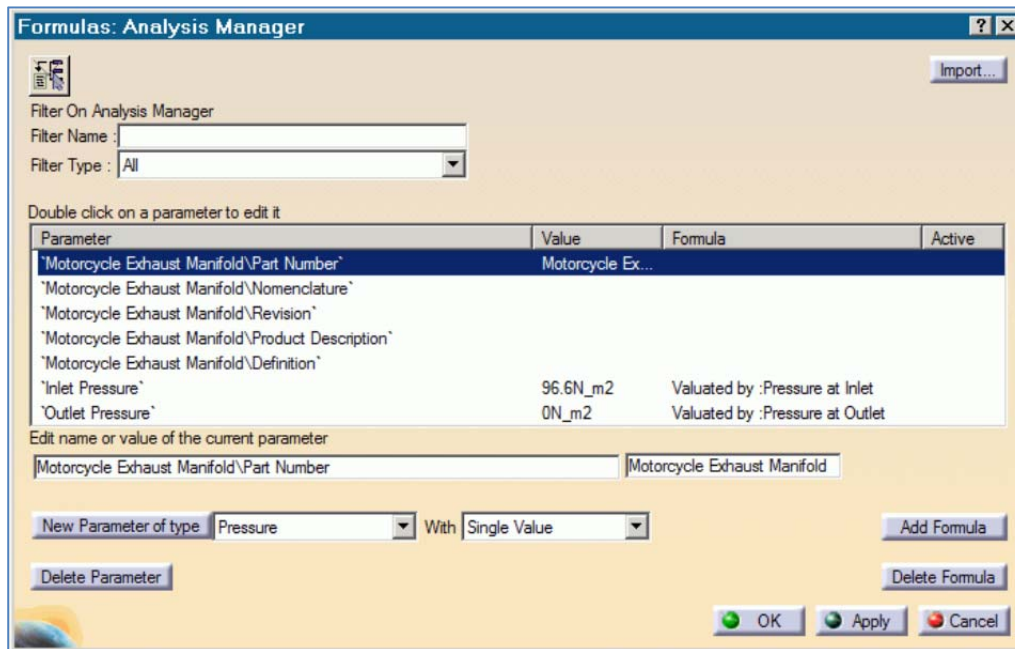
Figure 3.3.6H: Analysis manager on pressure loss of STAR-CCM+ user guide version 8.02

Criterion 3.6.7: Conjugate heat transfer: **ranking score = 5**

STAR-CCM+ can obtain solutions in both the solid and the fluid, or STAR-CCM+ can obtain the flow and thermal solution in the fluid only and pass thermal information to another application that solves for the thermal solution in the solid. An example of this arrangement is where STAR-CCM+ exchanges thermal data with Abaqus in a coupled simulation.

3.3.9 Criteria group: Workflow - Support for Validation:

Criterion 3.7.1: Validation document / example: **ranking score = 1**

It is difficult to find references and sources of the validation from CFD application, wind tunnel, and on site measurement on the StarCCM+ web-site. STAR-CCM+ may be not well known in the building and wind engineering yet.

3.3.10 Criteria group: Cost of Software:

Criterion 3.8.1: Commercial pricing mode   **ranking score = 3**

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                School of Architecture, University of Hawaii

December 4, 2013   Page 88

Criterion 3.8.2: Academic pricing: **ranking score = 5**

STAR-CCM+ has charged $1,450 for one academic / commercial (for University of Hawaii at Manoa) seat per year, which included the full webinars and some other online trainings.

Criterion 3.8.3: Software includes all modules (no extra cost):   **ranking score = 5**

STAR-CCM+ is a comprehensive and complex single CFD module which does not require any additional modules for advanced CFD analysis, including turbulence models.

Criterion 3.8.4: Option to acquire sufficient seats at no extra cost: **ranking score = 5**

STAR-CCM+ has fully supported academic research and is opening a new market in building science; therefore in additional seats will be at low or no cost.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                            School of Architecture, University of Hawaii

December 4, 2013   Page 89

3.4   Ranking of Autodesk Simulation CFD 2013-2014

This section provides a description of the capabilities of **Autodesk Simulation CFD** software, and specifically the external CFD module that is part of the overall software package.
http://www.youtube.com/watch?v=9-9DsWDQXSc

3.4.1 Criteria group: Value for the CFD Software Product for Architecture

Criterion 1.1:   Benefit for external CFD applications to architecture: **ranking score = 5**

Autodesk Simulation CFD provides a benefit direct to architectural engineering (AEC). AEC analyses can be classified into three major categories:
- Mechanical Ventilation
- External Flow or Wind Loading
- Natural Ventilation

The typical objectives of most AEC analyses include:
- Understand temperature distribution through an occupied zone to ensure interior comfort and improving efficiency
- Quantify and account for occupant-dissipated heat
- Reduce air changes per hour and operational costs by optimizing air distribution system efficiency
- Examine diffuser throws and flow patterns to evaluate effectiveness in custom HVAC installations
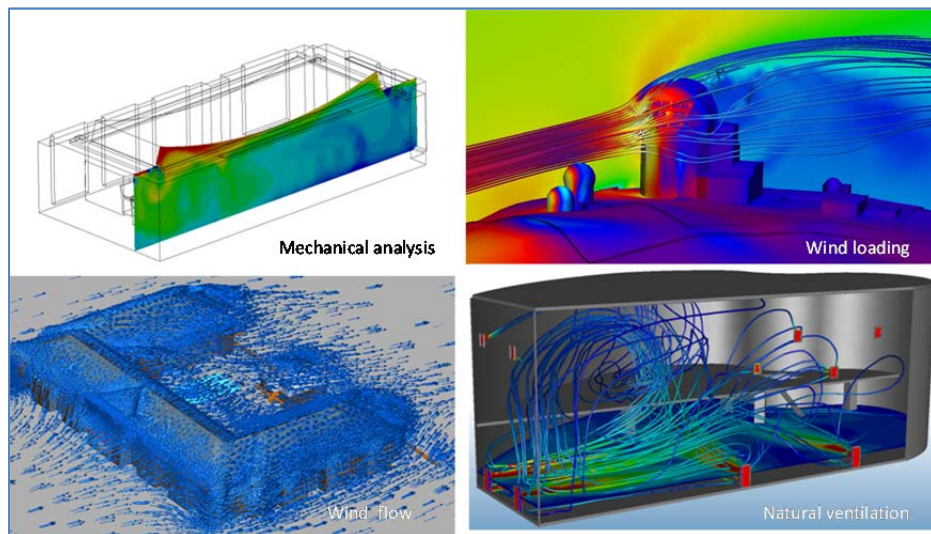

Figure 3.4.1A: Example of AEC analysis from Autodesk Simulation CFD user help

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 90

Criterion 1.2:   Previous applications in architecture urban design**:**   **ranking score = 1**

Autodesk Simulation CFD is a new application under Autodesk operating. There has been only limited documented professional contribution and collaboration.

Criterion 1.3:   Software product in scientific or applied professional articles**:**   **ranking score = 1**

Autodesk Simulation CFD is a new application under Autodesk operating. There has been only limited documented professional comment. Therefore, technical articles and reviews about using Autodesk Simulation are very limited, beside internal training and tutorial offered by Autodesk, for example;

http://sustainabilityworkshop.autodesk.com/buildings/simulation-cfd-detailed-fluid-and-heat-studies

http://www.autodesk.co.uk/adsk/servlet/pc/item?siteID=452932&id=19609897

Criterion 1.4:   Using graphics output for architectural presentations   **ranking score = 5**

Autodesk provides very powerful opportunity to integrate CFD applications with near-photorealistic graphics to produce images and videos of simulation results. These functions also allow visualizing animated depiction of results and the associative exchange of Building Information Modelling (BIM) data. Autodesk Showcase Interoperability for Photorealistic Visualization Interoperability between Autodesk Simulation CFD software and Autodesk® Showcase® 3D presentation and visualization software enhances design assessment through the examination of near-photorealistic renderings of digital prototypes and the related computational fluid dynamics simulation results. The ability to include simulation results, such as flow traces and cut planes, within photorealistic images and videos enables users to share simulation results in a new and powerful way. The Autodesk web-site offers several examples to show application of these visualization tools to architectural presentations.

3.4.2 Criteria Group Support by Vendor / Expert Network:

Criterion 2.1:   Technical support available: **ranking score = 1**

There is no direct technical service line that the ERDL-HNEI CFD team was able to call and there is no dedicated support service system that can contribute to solve problems for the user. However, there is a community or forum board that all users can post questions and comments.

Criterion 2.2:   Online tutorials available on demand: **ranking score = 5**

All Autodesk products provide wiki site base information, tutorial, training, and video for public and user. Online tutorials is fully available as long as they are up on the web and available to access.
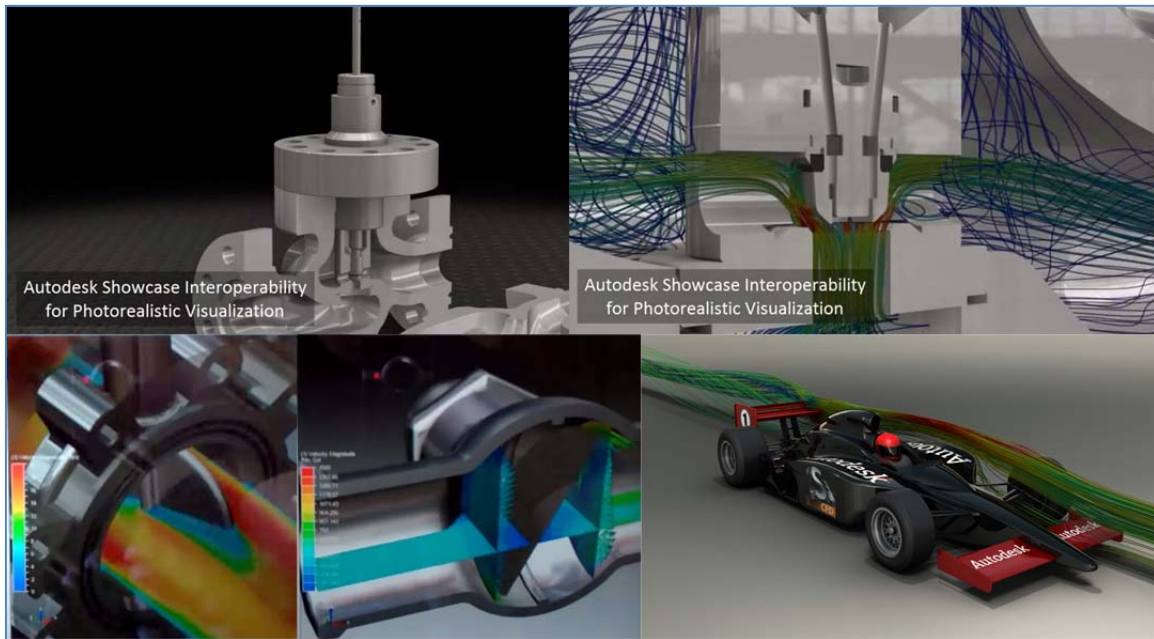
Figure 3.4.1B: Example of Revit and Autodesk Simulation CFD interoperability for photorealistic visualization. (http://www.autodesk.co.uk/adsk/servlet/pc/index?siteID=452932&id=17336394)
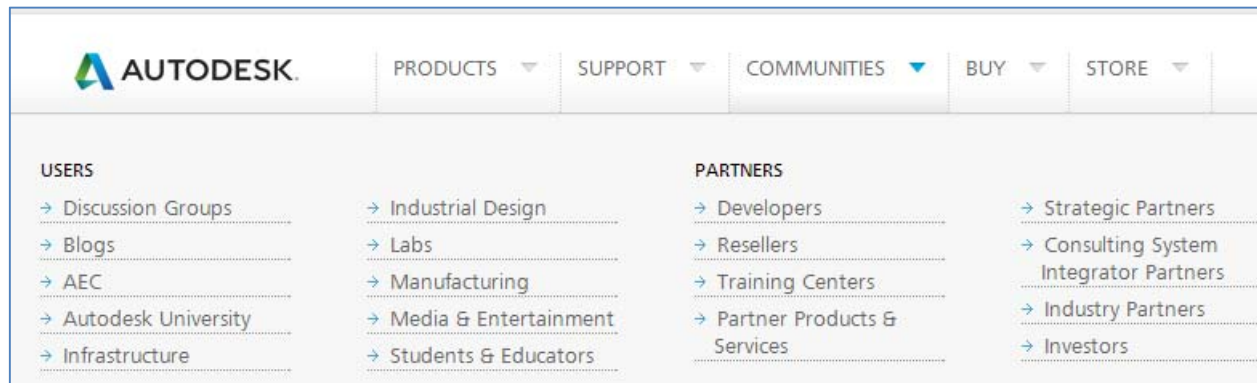

Figure 3.4.2A: Autodesk communities for user
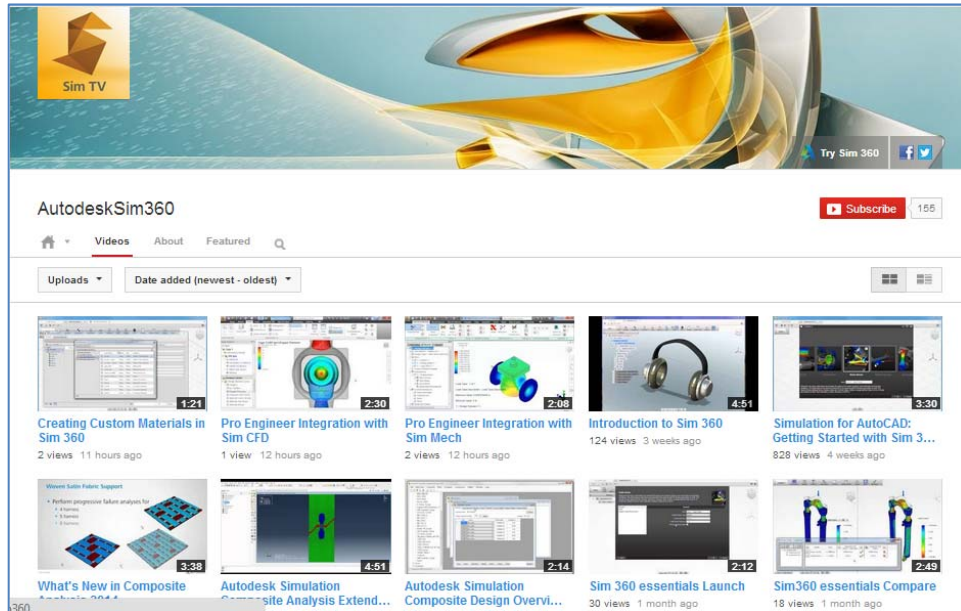(http://www.autodesk.com/products/autodesk-simulation-family/features/simulation-cfd)

Figure 3.4.2B: Autodesk WikiHelp web interface;   (http://wikihelp.autodesk.com/Simulation_CFD/enu/2014)

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 5**

Online tutorials are fully available as long as they are posted on the Autodesk wiki help site. There are also many tutorials as youtube video.
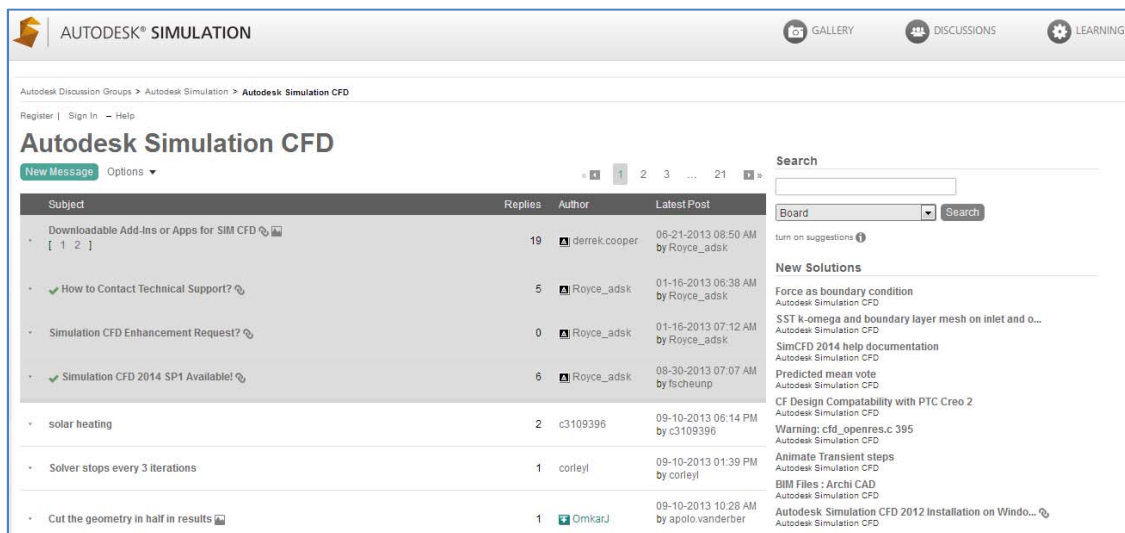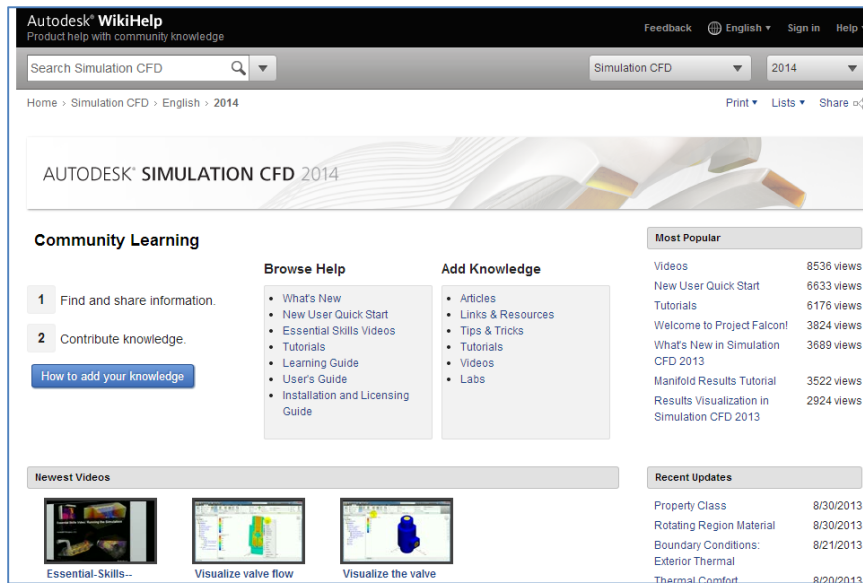
Figure 3.4.2B: Autodesk Simulation additional video tutorials in youtube;
                   http://www.youtube.com/user/AutodeskSim360/videos

Criterion 2.4:   Network of CFD professionals in architecture field: **ranking score = 1**

Autodesk Simulation CFD supports networking of professional CFD practitioners or educators can share knowledge and engage in discussion and forums therefore, users are able to exchange knowledge.
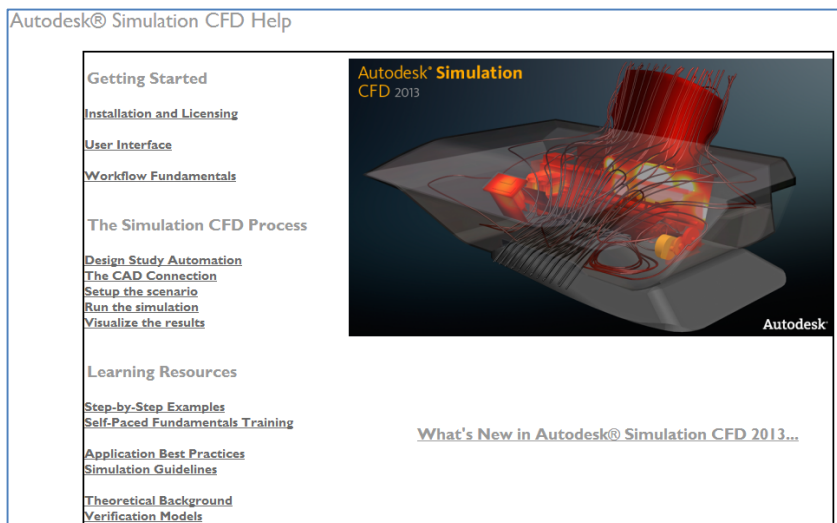

Figure 3.4.2C: Autodesk Simulation CFD group discussion (http://forums.autodesk.com/t5/Autodesk-Simulation-CFD/bd-p/330)

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 94

Criterion 2.5:   Ease of working with academic support   **ranking score = 5**

Autodesk Simulation CFD fully supports academic institutions, including faculty as well as students. There is Simulation CFD community and the Autodesk University to help and facilitate user, usually there is no charge to access the site for students and faculty.

Criterion 2.6:   User guide provides efficient and comprehensive information: **ranking score = 5**

Autodesk Simulation CFD provides comprehensive information for user guide and help.

3.4.3. Criteria Group Workflow-Ease of use / GUI

Criterion 3.1.1: GUI is easy to use for trained operator: **ranking score = 5**

Autodesk Simulation CFD has an effective GUI that significantly guides users to simulation workflow. Both Ribon (2) and Design study bar (10) are the helpful indicators that provide the sure with guidance on what is the next step.
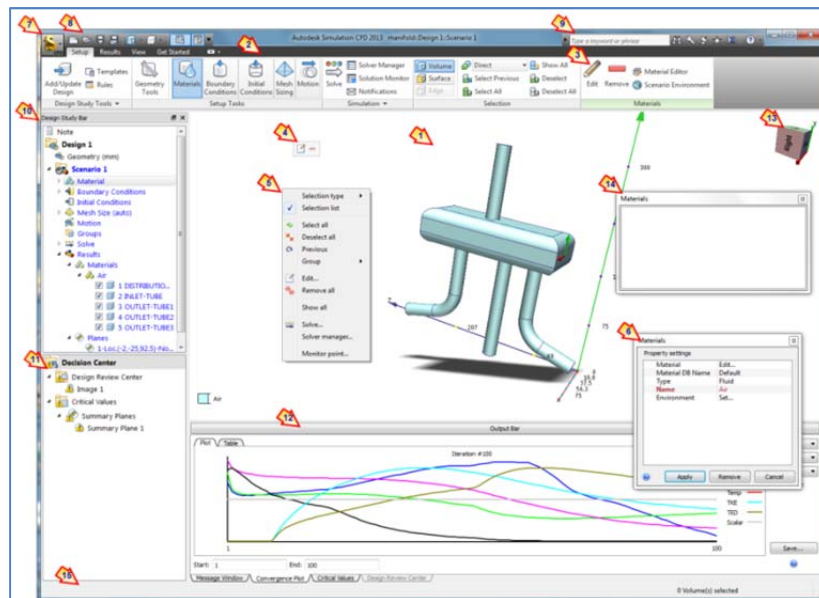

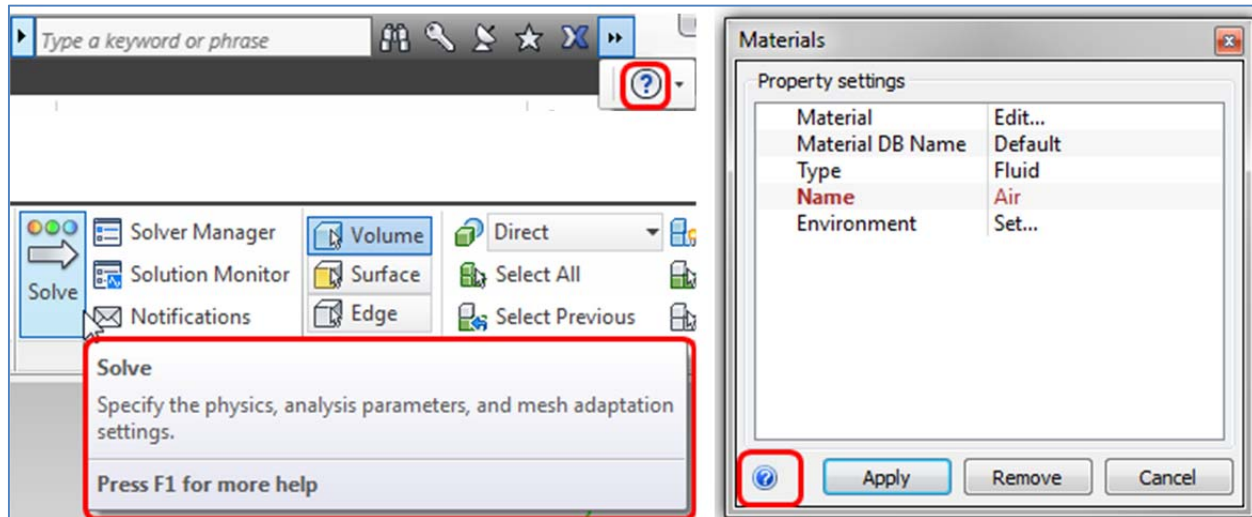Figure 3.4.2D: Autodesk Simulation CFD user help interface

Figure 3.4.3A: Autodesk Simulation CFD user interface

Criterion 3.1.2: User guide and online help is available for each command: **ranking score = 5**

Autodesk Simulation CFD provides an easy user guide and accessible help command in every part of component.
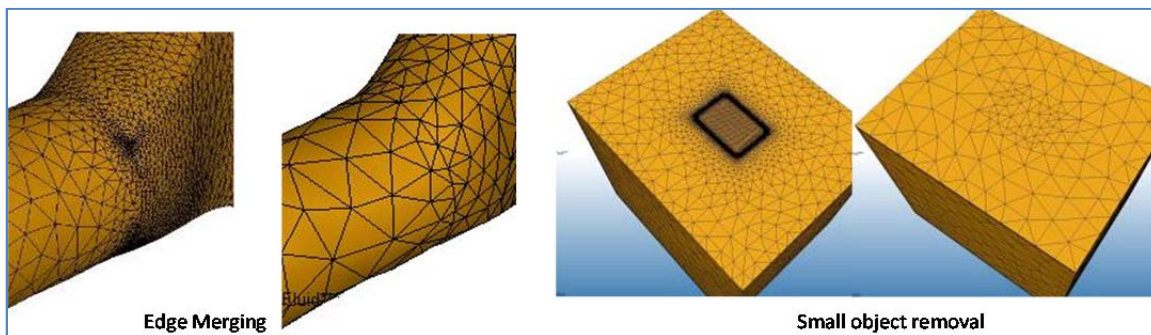

Figure 3.4.3B: Autodesk Simulation CFD help indicator and browser

3.4.4 Criteria Group Modeling / Import Geometry

Criterion 3.2.1: Create geometry inside CFD application: **ranking score = 1**

Autodesk Simulation CFD does not provide an internal geometry modeling capability. The software requires import of geometry from external CAD models or the software directly integrates with the Revit model.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                            School of Architecture, University of Hawaii

December 4, 2013   Page 96

<u>Criterion 3.2.2:</u> Import geometry / repair surface before meshing: **ranking score = 5**

Autodesk Simulation CFD requires that the CAD geometry be modified before the simulation can be run. In some cases, problematic entities such as sliver surfaces or extra edges can greatly complicate meshing. In other cases the simulation model requires additional parts to simulate the flow. These can be either inside a cavity or a flow volume that completely surrounds the original model. Geometry Tools provide a way to modify the CAD model for simulation.

- Edge Merging: Connect edges that share a vertex with an inflection less than a specified tolerance.
- Small Object Removal: Remove very small surfaces and edges that are typically too small to see, but can greatly affect meshing.
- Void Fill Creation: Fill an internal cavity with a meshable volume to simulate the flow.
- External Volume Creation: Immerse the model in a flow volume to simulate external flow.

<u>Criterion 3.2.3:</u> Support Revit Architecture: **ranking score = 5**

Enhanced Autodesk Revit Interoperability of Autodesk Simulation CFD 2013, an enhanced CAD connection with Autodesk Revit software, provides associative exchange of BIM data for computational fluid dynamics simulations. Users can now use the rule manager to assign settings automatically to frequently recurring CAD components and BIM data, providing faster and more accurate set-up of computational fluid dynamics simulations and helping building professionals to design, build and maintain higher-quality, more energy-efficient buildings.
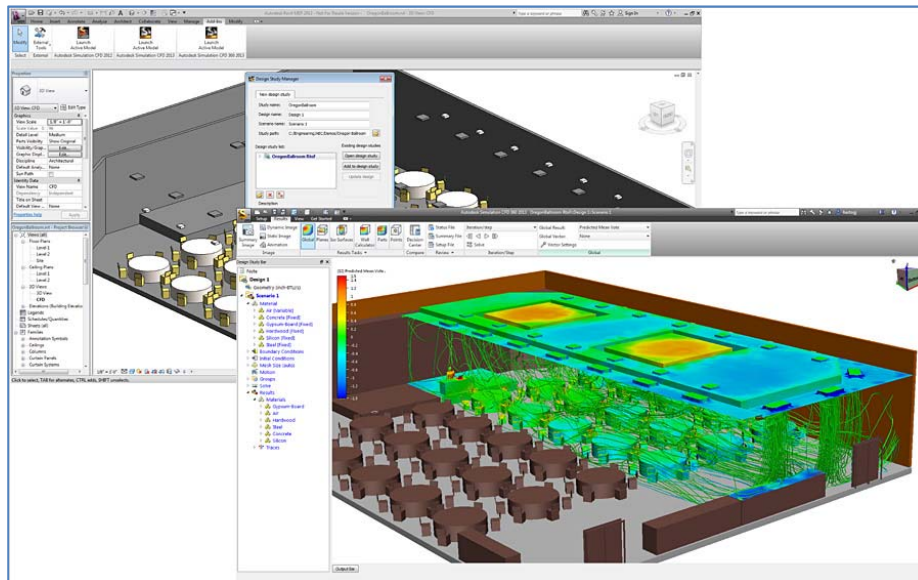

Figure 3.4.3C: Example of geometry repairs and modifies tool from Autodesk Simulation user guide
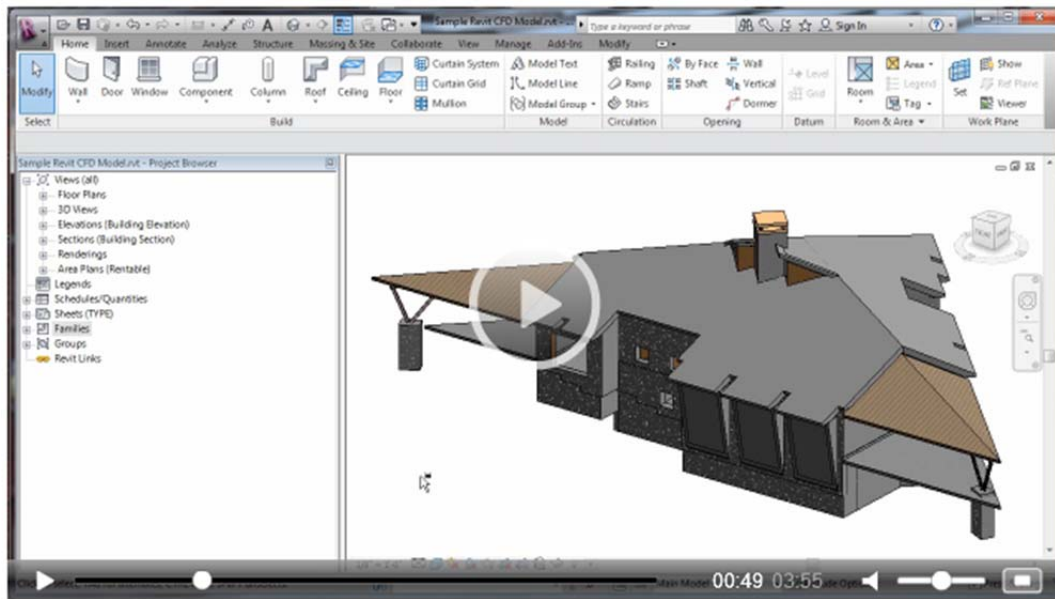
Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 97

Figure 3.4.3D: Example of Autodesk Revit with Autodesk Simulation CFD 2013
(http://wikihelp.autodesk.com/Simulation_CFD/enu/community/Videos/Autodesk_Simulation_CFD_connection_with_Revit)

Criterion 3.2.4: Support CADD format and Inventor: **ranking score = 5**

Autodesk Simulation CFD requires input from CAD system and model. After completing the model in CAD, the first step is to transfer the geometry into Autodesk Simulation CFD. There are several ways to do this, but the process originates in the CAD system. The following CAD systems use a CAD Connection that is based on an Autodesk® Simulation CFD menu or icon in their user interface:

- Autodesk® Inventor
- Autodesk® Inventor Fusion
- Autodesk® Revit
- Pro/Engineer Wildfire 4 and higher
- Solid Edge
- UGNX
- One Space Designer (CoCreate Modeling)
- SpaceClaim
- Solid Works

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 98

Other CAD systems, including the following, use a CAD Reader (Autodesk® Inventor Fusion) to transfer geometry:

- Pro/Engineer Wildfire 3 (and earlier)
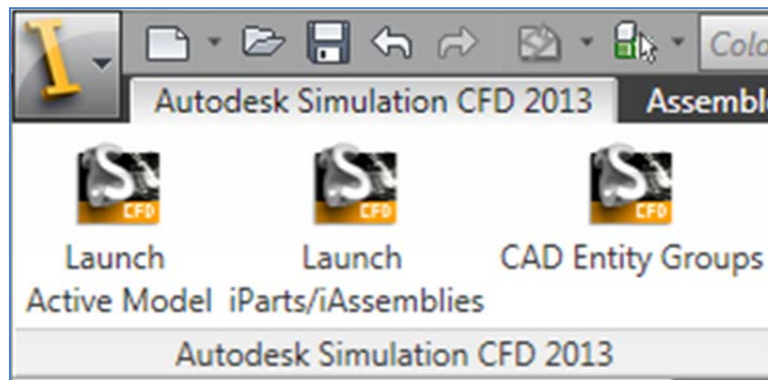- CATIA v5

Figure 3.4.3E: Launch CAD model to Autodesk Simulation CFD from Autodesk Inventor

Criterion 3.2.6: Support Rhino: **ranking score = 1**

Autodesk Simulation CFD has very strong ties to the leading CAD tools used in industry. By using the same geometry engines as the CAD systems, Autodesk Simulation CFD reads the native model ***without the need for STEP or IGES translations.*** Parametric changes to the geometry are read directly into Autodesk Simulation CFD (as Designs are updated within the Design Study), and model settings from similar analyses are automatically applied when Designs and Scenarios are cloned.

This functionality delivers two significant benefits:

- Multiple designs and scenarios are easily studied without having to go to great lengths to fix IGES or Step translations for each design alternative.
- Corporate PLM/PDM initiatives are supported because geometry manipulation occurs ONLY in the CAD system, where it can be tracked and archived. When geometry is manipulated in a third party system (such as many of the other CFD tools), changes often are lost, or have to be re-created back in the CAD system.

3.4.5 Criteria Group   Workflow-Meshing

<u>Criterion 3.3.1:</u> Surface meshers: **ranking score = 3**

Autodesk Simulation CFD does not provide a tool for complex surface meshing; therefore, it mainly concerns automatic size and refinement.

Meshing AEC Applications

A basic guideline for a high-quality analysis model is that the mesh distribution is sufficient to resolve the flow and temperature gradients efficiently. In regions where the flow circulates or experiences large gradients (such as in wakes, vortices, and separation regions), a finer mesh is required.

Automatic Sizing

For most models, use Autosizing to define the mesh distribution. It may be necessary to locally refine the mesh on geometric features that are highly detailed. In some cases, it may be necessary to adjust the Minimum Refinement Length to reduce their effect on the overall mesh count. To locally refine the mesh in high-gradient flow regions, adjust the mesh distribution on geometric volumes and surfaces.

Mesh Refinement

Create refinement regions in areas where large flow and thermal gradients are expected:
- Diffuser throw areas
- Heat dissipating items
- Wake regions on downstream side of flow obstructions
- Sources of smoke & chemical contaminants

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                       School of Architecture, University of Hawaii

December 4, 2013   Page 100
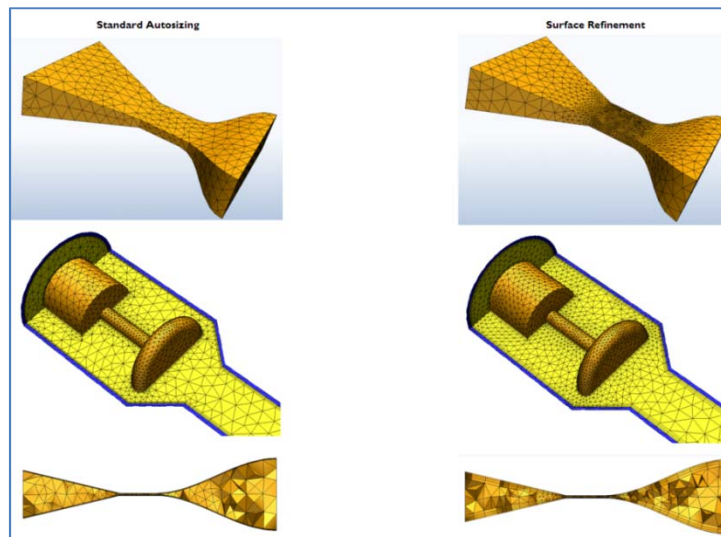
Figure 3.4.3F: Surface Refinement is only available for 3D Acis-, Parasolid-, and Granite (Pro/Engineer)-based models, Autodesk Simulation CFD user guide.

Criterion 3.3.2: Maximum cells and 2D- 3D cell volume: **ranking score = 3**

Autodesk Simulation CFD does not provide a maximum number of mesh cells. Autodesk Simulation CFD demonstrates that the geometry is broken up into small pieces called elements. The corner of each element is a node. The calculation is performed at the nodes. These elements and nodes make up the mesh. In three dimensional models, most elements are tetrahedral: a four sided, triangular-faced element. In two dimensional models, most elements are triangles:
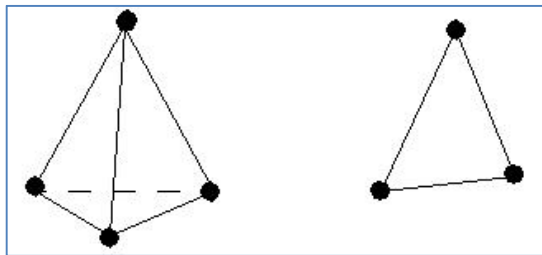

Figure 3.4.3G: 2D and 3D mesh cell model from Autodesk Simulation CFD user guide.

Criterion 3.3.3: Boundary layer types: **ranking score = 5**

Autodesk Simulation CFD has described the surface boundary conditions that typically represent a quantity or flux that enters or leaves the model (flow, temperature, or heat, for example). For 3D models, surface conditions are available when the selection type is Surface. For 2D models, Edge must be the selection type.

Boundary conditions define the inputs of the simulation model. Some conditions, like velocity and volumetric flow rate, define how a fluid enters or leaves the model. Other conditions, like film coefficient and heat flux, define the interchange of energy between the model and its surroundings.

Boundary conditions connect the simulation model with its surroundings. Without them, the simulation is not defined, and in most cases cannot proceed. Most boundary conditions can be defined as either steady-state or transient. Steady-state boundary conditions persist throughout the simulation. Transient boundary conditions vary with time, and are often used to simulate an event or a cyclical phenomenon.

Criterion 3.3.4: Structured and Unstructured meshing: **ranking score = 4**

Autodesk Simulation CFD provides unstructured meshing system. The software uses both automatic and refinement meshing models for the three dimensional model. Most elements are tetrahedral: a four sided, triangular-faced element. In two dimensional models, most elements are triangles:

Velocity	• Velocity is typically used as an inlet boundary condition. • Specify either normal to the selected surface or in Cartesian components. • A velocity can be applied to an outlet, if the direction is defined as out of the model. • To define a fully developed flow profile to a velocity condition, check **Fully Developed**.
Volume Flow Rate	• Apply to planar inlets (and sometimes outlets). • This is most useful if the density is constant. • To define a fully developed flow profile to a volume flow rate condition, check **Fully Developed**.
Mass Flow Rate	• Mostly applied to inlets (planar only). • A mass flow rate can be applied to an outlet, if the flow direction is out of the model.
Pressure	• The Pressure boundary condition is typically used as an outlet condition. • The recommended (and most convenient) outlet condition is a **static, gage** pressure with a value of **0**. • No other conditions are needed at an outlet. • If the pressure drop through a device is known, specify the pressure drop at the inlet (as a static gage pressure), and a value of 0 static gage at the outlet.
Slip/Symmetry	• The slip condition causes the fluid to flow along a wall instead of stopping at the wall, which typically occurs along a wall. • Fluid is prevented from flowing through the wall, however. • Slip walls are useful for defining symmetry planes. • The symmetry surface does not have to be parallel to a coordinate axis.

Figure 3.4.3H: Boundary condition types from Autodesk Simulation CFD user guide.
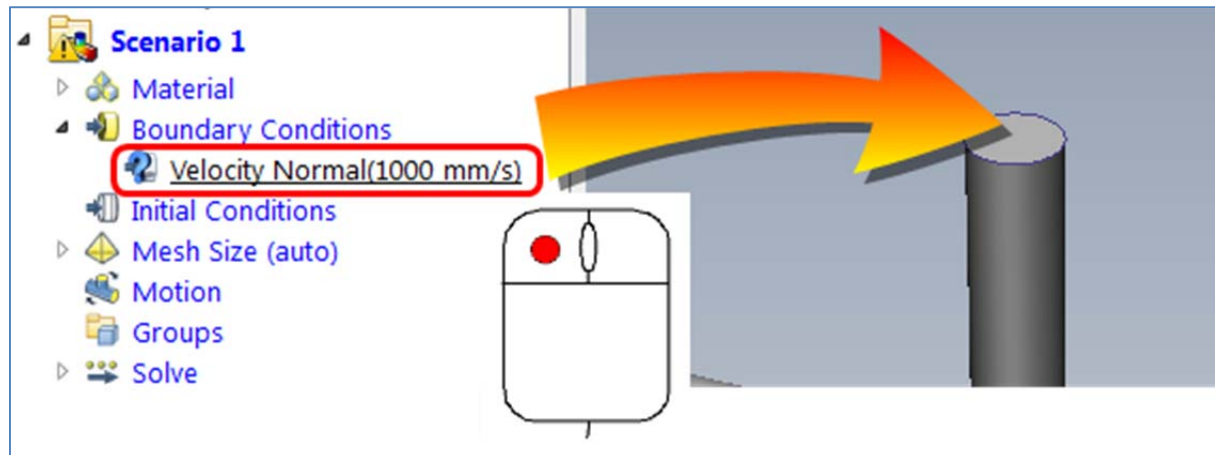

Figure 3.4.3I: Assigning the boundary condition types from Autodesk Simulation CFD user guide.
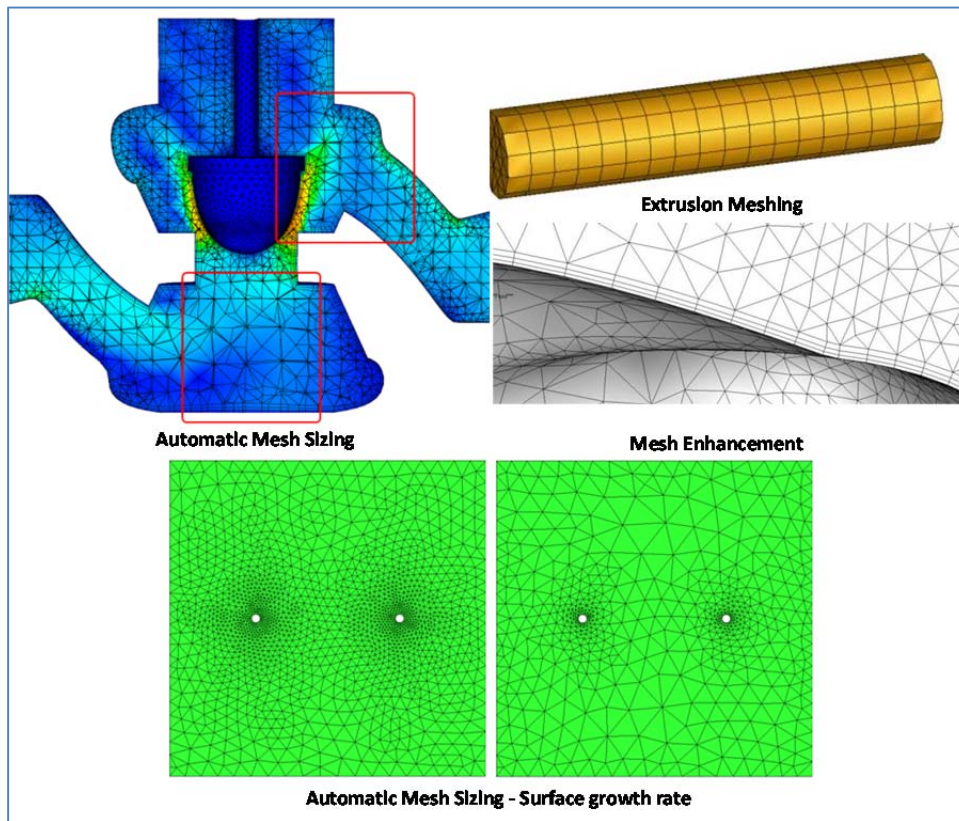
Figure 3.4.3J: Unstructured meshing - automatic and refinement meshing models from Autodesk Simulation CFD user guide.

Criterion 3.3.5: Adaptive meshing capability: **ranking score = 5**

Mesh Adaptation uses solution results to progressively improve the mesh definition. The simulation is run several times. Each time the results in the previous cycle are used to improve the mesh in the next cycle. The result is a mesh that is optimized for the particular simulation. The mesh is finer for high gradient regions and coarser elsewhere.

Limitations
Mesh Adaptation does not support the following simulation types or settings:
- Transient simulations (including Motion and Rotating)
- Quick Convection (including both Forced and Natural/Free)
- 2D models
- Surface parts
- Extruded meshes

- Models launched from Pro/Engineer with the Mechanica method.
- The following material types:
    - Distributed Resistance
    - Internal Fan
    - Centrifugal Pump /Blower
    - Check valve
    - Rotating Region
    - Compact Thermal Model (CTM)
    - Themoelectric device (TEC)
    - LED device

3.4.6 Criteria Group   Workflow Physic Models

Criterion 3.4.1: Solvers – Coupled / decoupled: **ranking score = 4**

Autodesk Simulation CFD provides solving solution in both coupled and decoupled.

Coupled: The partial differential equations governing fluid flow and heat transfer include the continuity equation, the Navier-Stokes equations and the energy equation. These equations are intimately coupled and non-linear making a general analytic solution impossible except for a limited number of special problems, where the equations can be reduced to yield analytic solutions.
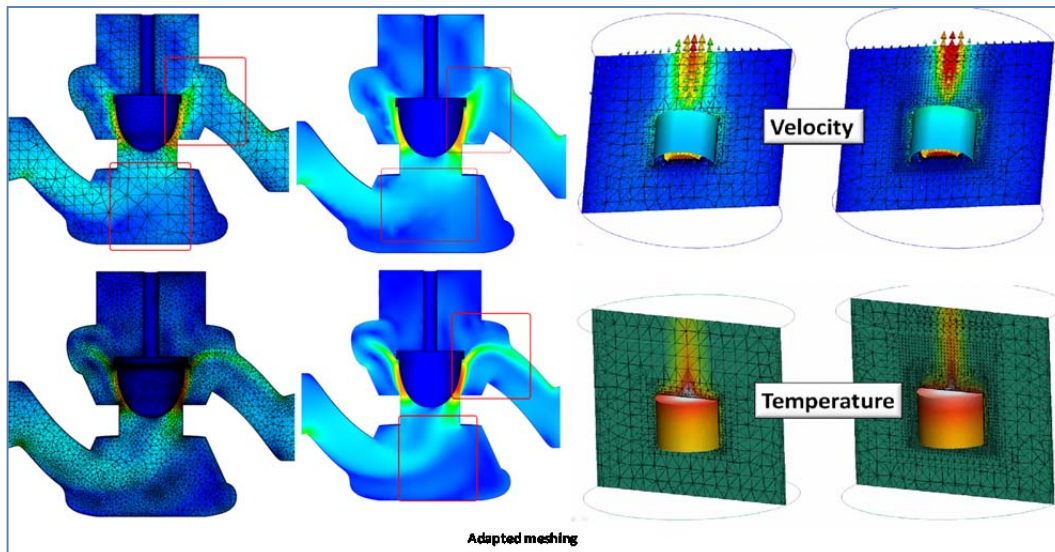

Figure 3.4.3K: Adapted Meshing from Autodesk Simulation CFD user guide.

Decoupled: In a forced convection analysis, the flow and heat transfer can be solved separately because the flow does not depend on the temperature distribution. An often used technique is to compute the flow solution prior to computing the thermal distribution. Unlike a buoyancy-driven solution, the flow and heat transfer solutions are decoupled from one another.

Criterion 3.4.2: Solver-Finite Element Method (FEV) / Finite Volume Method (FVM): **ranking score = 5**

Autodesk Simulation CFD has described that the partial differential equations shown earlier in this section must be discretized or transformed into a set of algebraic equations which can be solved digitally. There are numerous methods available to do this discretization. The three most popular (based on the number of commercial computational fluid dynamics (CFD) codes available) are:

Finite difference method, the partial derivatives are replaced with a series expansion representation, usually a Taylor series. The series is truncated usually after 1 or 2 terms. The more terms are included, the more accurate the solution. However, more terms in the expansion causes the complexity and number of discrete points or nodes of the solution to increase dramatically. Applying this method to a regularly shaped geometry is straightforward. However, for irregularly shaped geometries, the equations must be transformed before the Taylor series can be applied. This transformation introduces all sorts of problems in terms of additional cross-coupling of equations, mesh generation and general convergence.

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

Finite element method, Galerkin's method of weighted residuals is used. In this method, the governing partial differential equations are integrated over an element or volume after having been multiplied by a weight function. The dependent variables are represented on the element by a shape function, which is the same form as the weight function. The shape function may take any of several forms. Autodesk® Simulation CFD uses linear for 2D triangular elements, bi-linear for 2D quadrilateral elements, linear for 3D tetrahedral elements, tri-linear for 3D hexahedral elements and a mix for the 3D 5 and 6 sided elements. The main advantage as well as the main disadvantage of finite elements is that it is difficult to

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 105

put any physical significance on the terms in the algebraic equations. In the finite volume method, you are always dealing with fluxes - not so with finite elements. However, the application of finite elements on any geometric shape is the same. Also, the boundary conditions which must be added after the fact for finite volume methods are an integral part of the discretized equations.

Method	Advantages	Disadvantages
Finite Element	1. More mathematics involved 2. Natural boundary conditions (for fluxes) 3. Master element formulation 4. Any shaped geometry can be modeled with the same effort	1. More mathematics involved - less physical significance
Finite Volume and **Finite** Difference	1. Fluxes have more physical significance	1. Irregular geometries require far more effort

Figure 4.4.3M: Summarizes the advantages and disadvantages of the various methods from Autodesk Simulation CFD user guide.

Criterion 3.4.3: Steady state/Transient state**:**   **ranking score = 5**

When the Results Output Interval is set to the default value of 0, results are saved only when the analysis stops (either completing the specified number of iterations or because the Stop button was pressed). For complicated analyses, it is recommended to set a non-zero Results Output Interval. (Be careful that your Results Output Interval is not so small as to exceed your hard-drive capacity.) These saved results sets/time steps can be used for continuing the analysis from an earlier result set if there is a problem--in effect returning to an earlier saved state of the analysis without having to run it out again from the beginning. Results from saved result sets or time steps can also be animated. The intermediate summary information is available in the summary file (analysis-name.sum). Summary information from intermediate iterations is appended to the summary history file (analysis-name.smh). This information is useful for tracking the progress of an analysis.

Steady state: indicate the interval of Steps to be saved as a constant value or enter a table.

Transient state: For transient analyses, results can now be saved at either a specified interval of time steps or at a specified interval of seconds. The principal reason for this feature (besides greater flexibility) is that when Intelligent Solution Control varies the time step size (as described above) for transient analyses there is no way to ensure that results are saved at the desired times. This feature provides the ability to save transient results at exactly the desired times.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                          School of Architecture, University of Hawaii

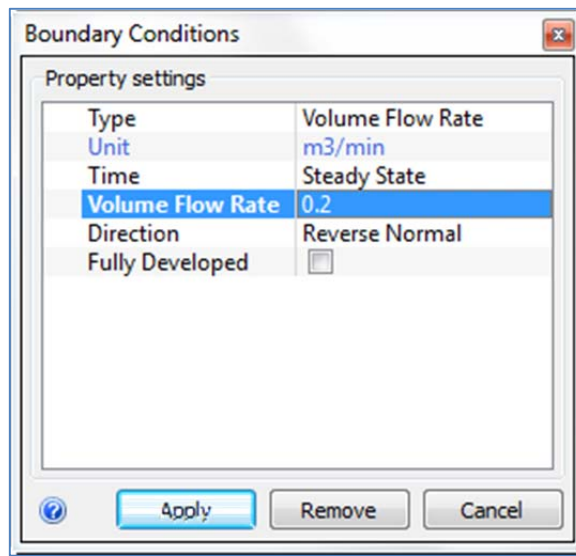December 4, 2013   Page 106

Figure 3.4.3N: Time step setting (Steady state / transient state) from Autodesk Simulation CFD user guide.

Criterion 3.4.4: Turbulence models**:**

ranking score = 5

K-epsilon: This is the default turbulence model. It is typically more accurate than the constant eddy viscosity model, but more computationally intensive and slightly less robust. It is not as resource intensive as the RNG model, but still gives good results. It is a general purpose model that performs well across a large number of applications.

Eddy Viscosity: The constant eddy viscosity model is slightly less rigorous than the k-epsilon model, but more numerically stable. This is a good choice for lower speed turbulent flows and some buoyancy flows. This model is useful if divergence occurs with one of the other models.

RNG: The RNG turbulence model is more computational intensive, but sometimes slightly more accurate than the k-epsilon model, particularly for separated flows. This model works best for predicting the reattachment point for separated flows, particularly for flow over a backward-facing step. When using the RNG model, it is often recommended to start with the k-epsilon model and after this model is fairly well converged, enable the RNG model.

Mixing Length: The Mixing Length turbulence model is primarily designed for internal natural convection analyses. Use of the mixing length model, in some cases, has been shown to reduce run times and provide better accuracy than the default turbulence model for internal buoyancy-driven flows. Note that the Mixing Length model is designed for gas flows (such as air), and will not produce good results when the working fluid is a liquid (such as water).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                 School of Architecture, University of Hawaii

December 4, 2013   Page 107

Low Re k-epsilon: This model is well suited for low speed, turbulent flows. The Reynolds number of such flow is typically between 1,500 and 5,000. Typical applications include pipe flows and external aerodynamic flow transitioning between laminar and turbulent, as well as flow situations that have both high speed and low speed areas.

Criterion 3.4.5: Materials**:**   **ranking score = 5**

Air is the primary physical material used in most AEC applications. Each application topic (Mechanical, Natural Ventilation, and External) describes the specific details about how Air should be used for specific application types. Solid materials are often frequently used to simulate building materials and other physical solids. Material devices such as Internal Fans and Distributed Resistances provide a convenient way to simulate complex devices (fans, filters, baffles, etc.) with simple geometry. This topic describes the details of using Solid materials, Internal Fan devices, and Distributed Resistance devices for AEC models.

Solid materials: The Default material database includes a wide variety of solid materials. The following are often used in AEC applications:
- Brick
- Gypsum Board
- Hardwood and Softwood
- Steel
- Glass

Custom materials: Several additional material types are also typically used in AEC applications, but due to the large number of varieties, are not included in the Default Material library. Examples of such solids include:
- Concrete
- Soil
- Insulation
- Composites

Resistance applications in AEC simulations: There are numerous occasions to use distributed resistances in AEC applications. They are extremely useful for applications involving HVAC systems, Datacenters, and Lab spaces. Some specific model elements include:
- Screens
- Filters
- Baffle assemblies
- Heating and cooling elements
- Crowds of people (seated or standing)
- Trusses
- Exterior landscaping

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 108
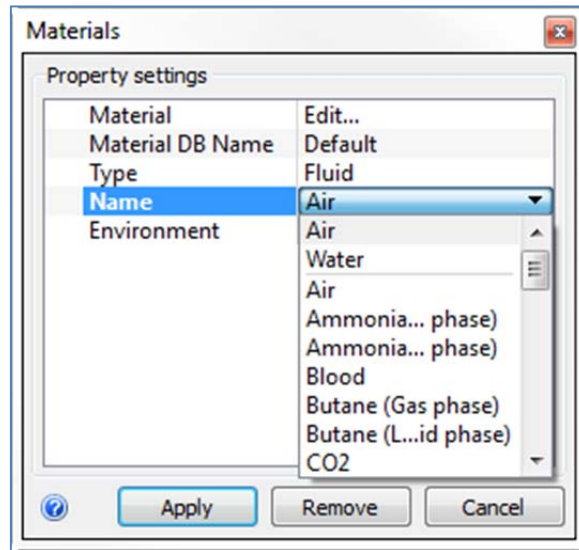
Figure 3.4.3N: Material settings from Autodesk Simulation CFD user guide.

Criterion 3.4.6: Customizable / special physics models**:**   **ranking score = 5**

Heat transfer:
There are several variations of heat transfer analyses that can be performed using Autodesk® Simulation CFD. Some of these can occur co-currently within the same analysis. For example, in most electronics analyses, heat is conducted through solid objects as well as convected by the flow. The following transfer modes can be simulated:
- Conduction
- Forced Convection
- Mixed Convection
- Conjugate Heat Transfer
- Natural Convection
- Radiation

The Smoke visibility:
Output quantity indicates how far away a sign is visible at every location in the model. The visibility distance near the burn source is typically very small because of heavy smoke. The visibility distance away from the burn source is usually significantly higher, indicating a reduced smoke concentration, and greater visibility.
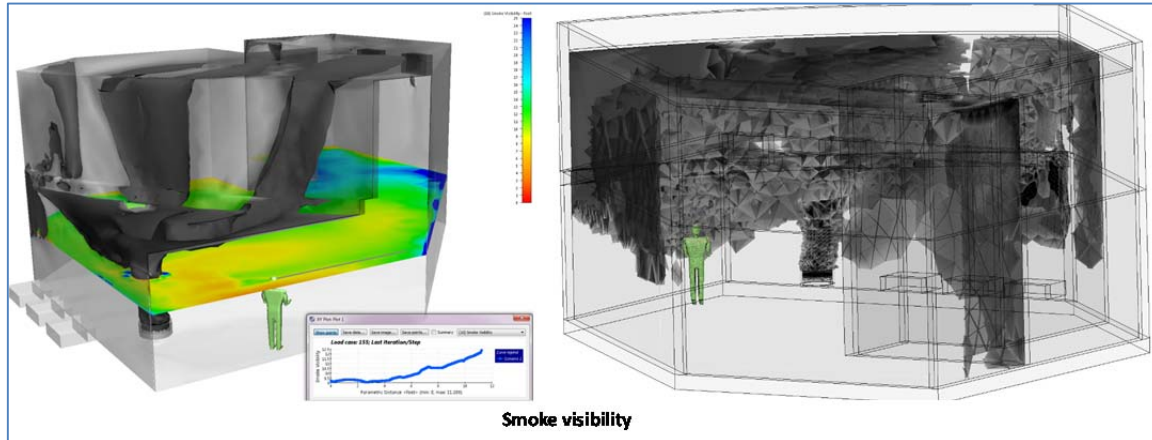
Figure 3.4.3O: The smoke visibility from Autodesk Simulation CFD user guide.

Humidity:

Autodesk Simulation CFD can simulate the condensation of liquid from a moist gas. Evaporation, however, is not supported. Select Humidity to simulate moist gas. Both the relative humidity and the condensed water can be visualized. Note that the condensation of a moist gas can be modeled by Autodesk Simulation CFD, but the evaporation of water into a gas stream cannot.

3.4.7 Criteria Group Workflow Post-Processing

Criterion 3.5.1: Visualization**:**   **ranking score = 5**

Visualization of results is required in every CFD simulation. The visualization reveals the performance of your design. They tell the user if the design satisfies its objectives. Results are essential for making informed design decisions. The Autodesk Simulation CFD results visualization environment contains a diverse assortment of tools designed to help you view, extract, and share your simulation results quickly, easily, and efficiently. The environment presents your results in multiple ways, including graphically-rich images, plots, and tabular data files. The results visualization environment is available both during and after the simulation. This seamless feedback keeps you informed at every stage of your simulation.

- Global
- Planes
- Iso Surface
- Wall Calculator
- Part
- Point
- Filter (in the drop-down menu)
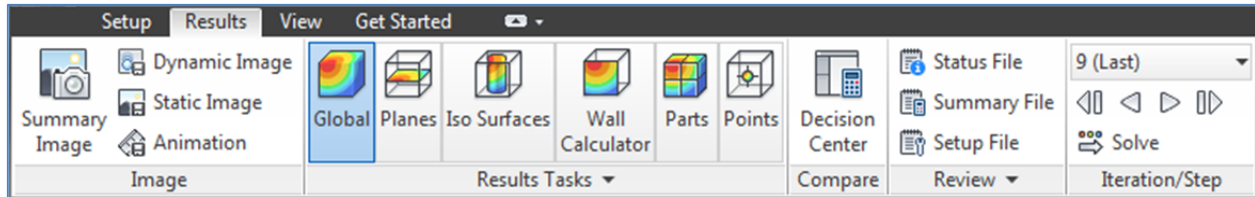- FEA Mapping (in the drop-down menu)

Figure 4.4.3P: Results tool bar from Autodesk Simulation CFD user guide.
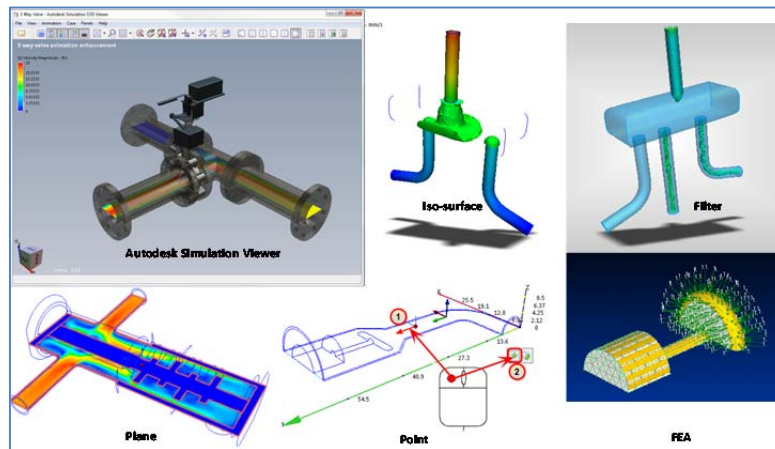
Figure 3.4.3Q: Examples of post-process from Autodesk Simulation CFD user guide.
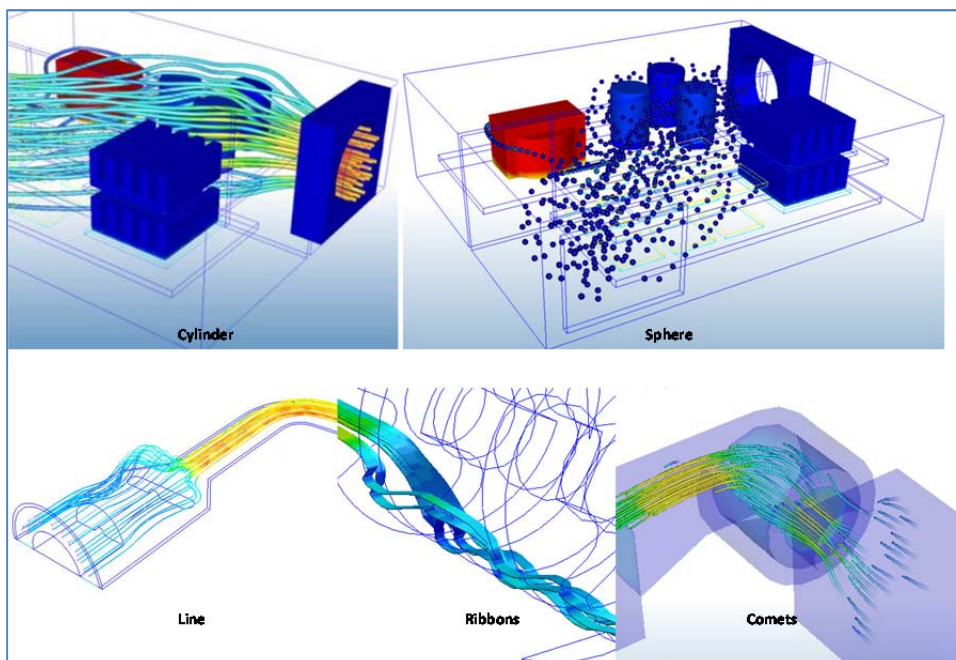
Figure 3.4.3R: Examples of particle trace from Autodesk Simulation CFD user guide.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                           School of Architecture, University of Hawaii

December 4, 2013   Page 111

Criterion 3.5.2: Quantitative analysis capabilities of components**:**   **ranking score = 5**

Autodesk Simulation CFD provides quantitative analysis through data queries and through special visualization tools, as such as below figures;

Plane Functionality: Planes provide a wide array of visualization functionality:
-   Planes graphically present results on cross-sections.
-   Planes provide a starting point for particle traces.
-   Planes allow extraction of bulk data through planar cross sections.
-   Planes serve as a basis for XY-plots.

Criterion 3.5.3: Comparative analysis (multi-scenarios)**:**   **ranking score = 5**

Autodesk Simulation CFD provides the post visualization in multi scenarios on the Decision Center where is a simple yet powerful environment for comparing design alternatives. Use it to identify the design that satisfies your design objectives. It is a powerful tool for:
-   Extracting specific results values
-   Comparing results from multiple scenarios

Figure 3.4.3S: Examples of plane functions and particle trace-seeding method from Autodesk Simulation CFD user guide.
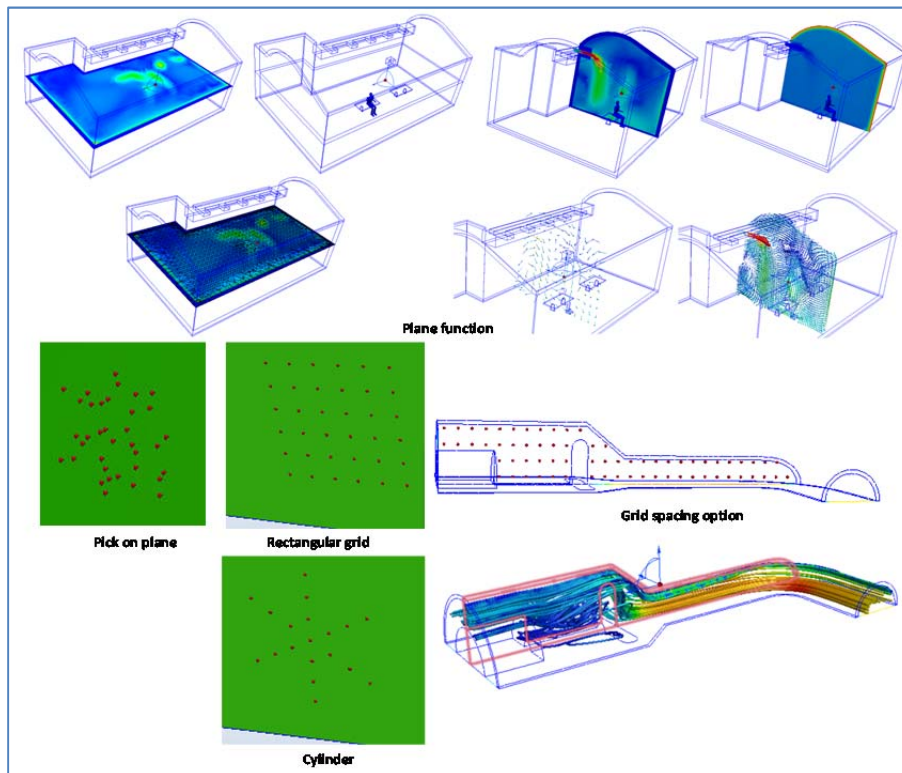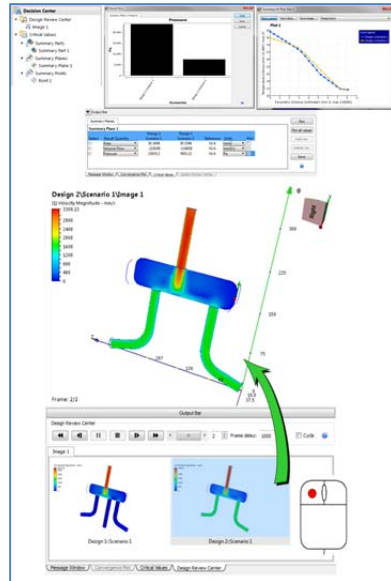
Figure 3.4.3T: Examples of Decision center from Autodesk Simulation CFD user guide.

3.4.8 Criteria Group Workflow-Customization / Special Functions

Criterion 3.6.1: Pedestrian comfort: **ranking score = 3**

Autodesk Simulation CFD does not provide a special functionality for pedestrian comfort assessment. However, pedestrian comfort refers to wind velocity at ground level, which Autodesk Simulation is able to predict and measure the velocity condition at grade level.

External Compressible Flow
External compressible flow can be classified into two categories:
-   Aerodynamic applications that are in open air (either at altitude or sea-level)
-   Aerodynamic applications in a wind-tunnel.

Criterion 3.6.2: Natural ventilation: **ranking score = 5**

Air movement in natural ventilation applications is generally the result of density gradients caused by temperature variations throughout the structure. Fans and other mechanical devices may be present, but do not dominate air movement. Many structures exhibit the combined contributions of ambient air passing through doors and windows with buoyancy-driven natural ventilation.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 113
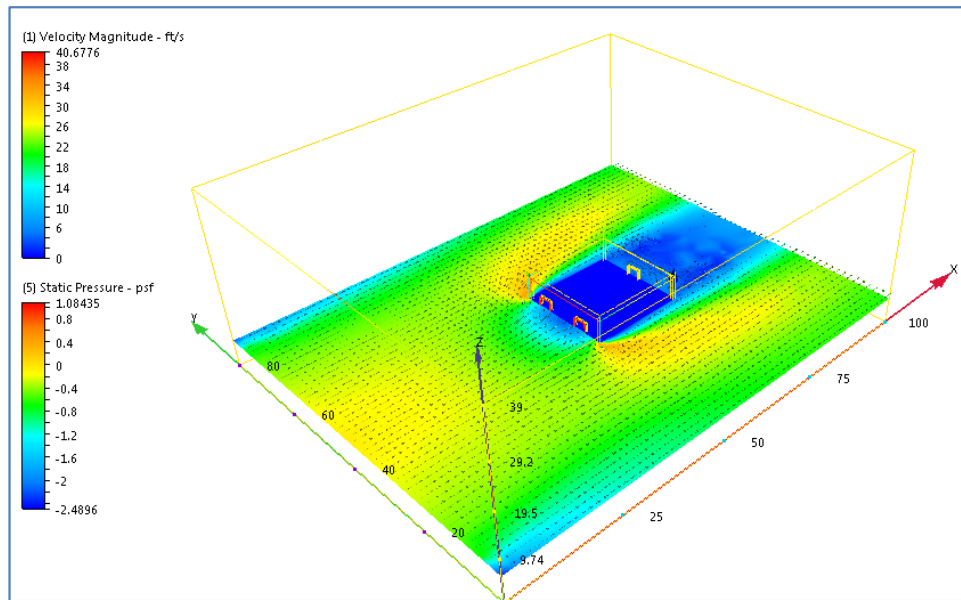
Figure 3.4.3W: Examples of velocity cut plane at ground level
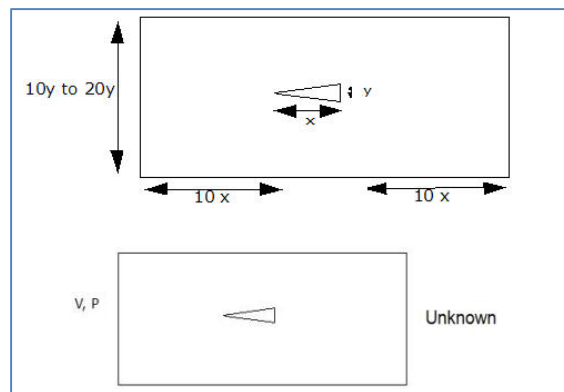
Figure 3.4.3X: Compressible Open air flow from Autodesk Simulation guide line.

- To define the inlet of the air volume, assign the wind speed as a Velocity boundary condition.
- To define the outlet of the air volume, assign Static Gage Pressure = 0.
- If the air region simulates a free-space environment (not a wind-tunnel), assign Slip/Symmetry to the top and sides of the region. (Do not specify a condition on the ground plane, because the air does not physically move along the ground plane.)
- Do not assign boundary conditions to windows and doors. Air flows in and out of the building based on the effects of wind and buoyancy.
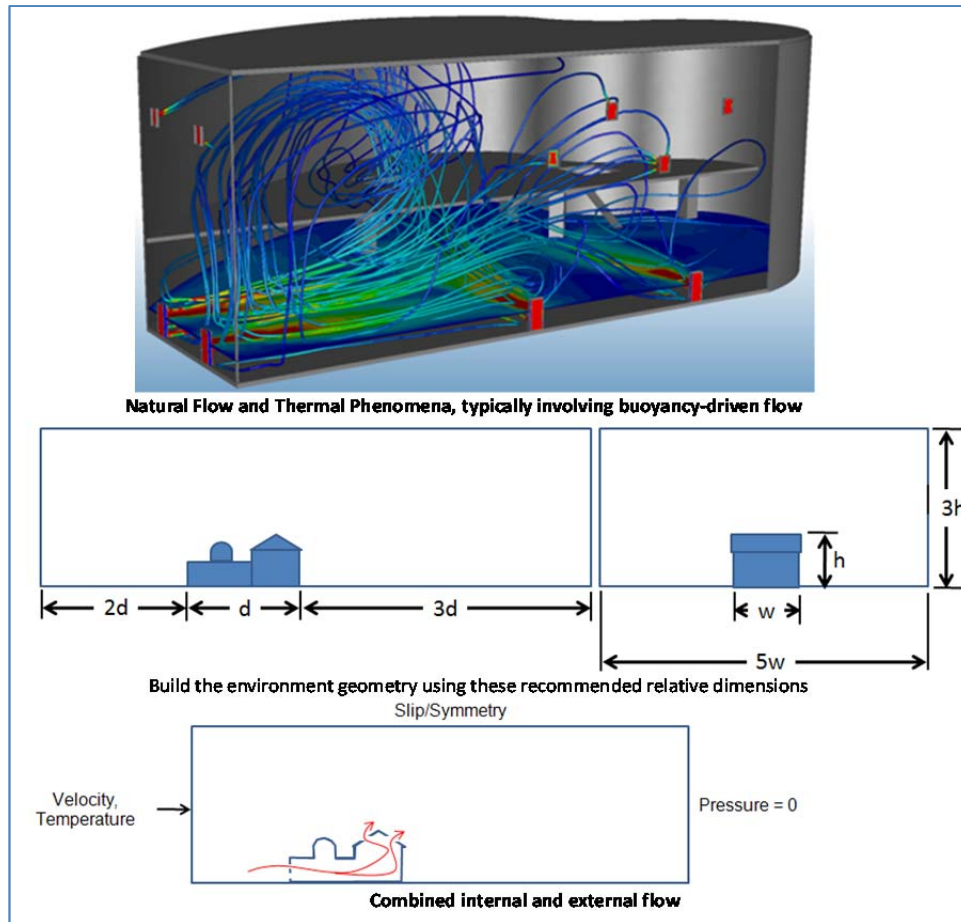
Figure 3.4.3Y: Natural ventilation feature from Autodesk Simulation guide line.

Criterion 3.6.3: Wind-driven rain (WDR):   **ranking score = 1**

Autodesk Simulation CFD does not provide a ready to use wind-driven rain model.

Criterion 3.6.4: Humidity: **ranking score = 5**

Autodesk Simulation CFD can simulate the condensation of liquid from a moist gas. Evaporation, however, is not supported. Autodesk Simulation CFD is able to demonstrate the amount of liquid condensed and the calculated field values of relative humidity can be viewed as results. The condensed liquid is calculated as a mixture fraction, i.e., the mass of the condensed liquid divided by the total mass of the liquid, vapor and carrier gas. The available results quantities for a humidity analysis are Relative Humidity and Percent Liquid. The Percent Liquid is plotted over a range from 0 to 1. A

value of 0 corresponds to completely dry air. A value of 1 indicates that liquid is condensing. Physically, a value of 1 indicates a liquid mist within the carrier gas.

Criterion 3.6.5: Wind load towards to building structure: **ranking score = 5**

Wind loading analyses simulate the air flow and resultant structural loading on buildings, large signs, and other structures. Examples include:
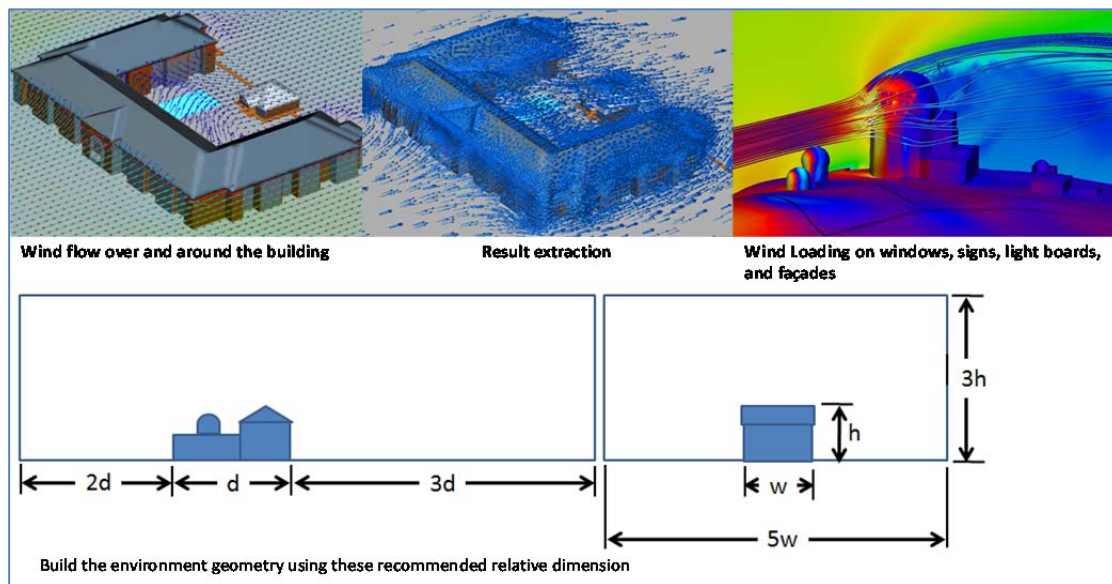
Figure 3.4.3Z: Example of wind load from Autodesk Simulation guide line.

Criterion 3.6.6: Pressure distribution along streamlines/pressure losses: **ranking score = 1**

Autodesk Simulation CFD guide does not provide any readily usable function or model to quantify pressure losses along streamlines.

Criterion 3.6.7: Conjugate heat transfer: **ranking score = 5**

For conjugate heat transfer analyses, the solid material conduction and the fluid convection are analyzed simultaneously. The type of fluid convection (natural, forced or mixed) determines the analysis parameters.

Natural convection occurs as a result of buoyancy-driven flow caused by density gradients due to temperature variations. Typical applications of natural convection include electronic systems that are either vented or completely sealed. These devices generally do not have fans or blowers. Instead, they

are cooled by buoyancy-driven flow that convects heat from heated components and conducts through the outer casing.

Natural convection within a sealed device is considered Internal. Natural convection around a device in a large enclosure or open environment is considered External. The analysis techniques for these two physical situations are a little different, and are explained in the Related Topics, below. Natural and free convection flows are largely dominated by buoyancy forces. The buoyancy forces are generated by density gradients which vary primarily with temperature since pressure gradients are relatively small in these flows. Natural convection flows may be laminar or turbulent.

Mixed convection: In many electronic cooling applications, the heated or cooled air is blown but may contain local temperature gradients that will cause some appreciable buoyancy effects. This type of heat transfer is known as mixed convection, since it has features of both natural and forced convection. There is not a good way to tell prior to the analysis if the heat transfer is mixed or forced.

3.4.9   Criteria Group: Support for Validation

Criterion 3.7.1: Validation document / example: **ranking score = 1**

It is difficult to find some Autodesk Simulation CFD references and sources of the validation from CFD application, wind tunnel, and on site measurement beyond Autodesk tutorials.

3.4.10   Criteria group: Cost of Software

Criterion 3.8.1: Commercial pricing model: **ranking score = 3**

Based on US.CAD commercial pricing model will cost $9,445 / per seat

Criterion 3.8.2: Academic pricing: **ranking score = 5**

Autodesk Simulation CFD 2014 will be charged about $250 per seat per year for one academic / commercial site license under ultimate suite from University of Hawaii at Manoa.

Criterion 3.8.3: Software includes all modules (no extra cost):   **ranking score = 5**

Even though, there are different features of Autodesk Simulation, all turbulence models are supplied with the CFD software package.

Criterion 3.8.4: Option to acquire sufficient seats at no extra costs: **ranking score = 1**

Based on US.CAD there is one seat per price if there is any additional seat will be charged by amount of required seats.

3.5. Ranking of CRADLE ScSTREAM:

This section provides a description of the capabilities of ScSTREAM software, and specifically the capabilities that are related to external CFD calculations.

3.5.1 Criteria Group: Value for the CFD software product to the Architecture and Building market

These ranking criteria are listed under Number 1 in the ranking matrix:

Criterion 1.1:   Benefit for external CFD applications to architecture   **ranking score = 5**

ScSTREAM offers a comprehensive functionality for external CFD applications. ScSTREAM offers a high quality simulation output for a relatively easy to understand GUI. The grid is a structured and self-adjusting grid and the requirements to obtain results follow a well-defined workflow in the GUI. The CFD results offer data of pressure and velocity distribution around the building, as well as other properties, such as the prediction of flow and thermal interactions and transport of particulate or dissolved matter.

ScSTREAM provides the practitioners in architecture or building industry as well as students high quality quantitative design assistance for common and more sophisticated flow and thermal problems that are relevant to the design and operation of buildings. The benefit to the building industry is that results of flow occurrences around building are readily obtainable. There appears to be an increasing readiness of the building and architecture industry to use scSTREAM for simulation of high-performance buildings.

Criterion 1.2:   Previous applications in architecture / urban design   **ranking score = 5**

The use of scSTREAM in external CFD applications appears to be well established, considering the currently relative small range of CFD allocations to architecture and buildings. The project team could identify several reported cases where the software has been used to investigate various technical issues related to building performance. Below are three cases for external CFD work that have been provided by CRADLE.
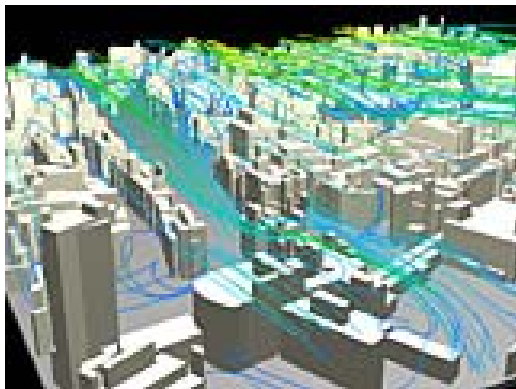
Example 1 of external CFD around buildings or in neighborhoods: The image on the left shows high rise communities for which wind patterns were analyzed to ensure good pedestrian comfort.
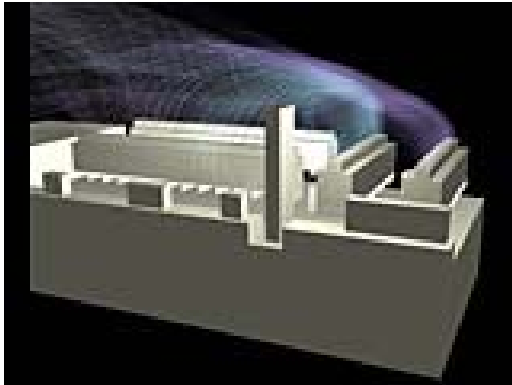
Example 2 of external CFD around buildings or in neighborhoods: The software analyzed the diffusion of plant stack gas and the exhaust heat from a building to the surrounding environment. The CFD analysis was used to predict the gas concentration at different altitudes in order to evaluate the efficiency of a specific cooling tower arrangement or the performance of exhaust of exterior air conditioning unit.

Example 3 of external CFD around buildings or in neighborhoods: The software analyzed wind conditions in valley-shaped terrains prior to construction. The software can import terrain geography from map data and calculate wind pressures on buildings and structures=.

Figure 3.5.1A: Example of wind pattern in external CFD (source: scStream web-site).

It must be noted, however, that external CFD applications in architecture are only at the beginning phase of wider use. The visualization of wind movement around buildings has been increasingly used in presentations by practitioners in the building industry. Besides these qualitative descriptions the use of quantitative means to describe fluid motion and resulting physical phenomena in or around buildings is experiencing a growing need. This increased interest of the architecture and building industry in high performance is supported by software packages, such as scSTREAM, that combine a high level of fluid analysis performance with a limited complexity of operating the software. Therefore the software offers a high benefit to the architecture field.

Criterion 1.3:   Previous mentioning of software product in scientific or applied professional articles
ranking score = 3

The software has been in use in Japan and Asia for a longer period than in the US. The project team received a list of articles that contain scSTREAM case studies where most of these articles were written in Japanese. There seems to be a limited availability of translation of these professional ad scientific articles into English. But the very fact that the software has received a sizable mention in technical and scientific publications speaks for the fact that scSTREAM appears to be well established for use by architecture and building professionals.

Criterion 1.4: Using graphical output for presentations   **ranking score = 4**

scSTREAM has a ready to use graphical interface that presents visual images of the distribution and quantity of external flow and thermal related phenomena around buildings. The software has a feature where high quality renderings of buildings can be overlaid by vector description of wind velocity and contoured slices for velocity and pressure. The handling of these graphics representation is easy and with some effort the "graphic-savvy" user can fuse other graphics presentations with the renderings produced by scSTREAM. Of particular interest is the import of streamlines into a graphical representations that were produced by other visualizations (and non CFD) applications.
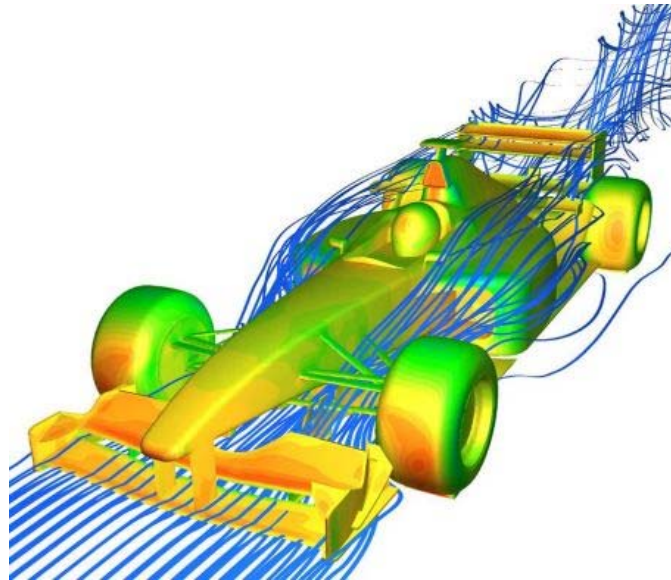
Figure 3.5.1B: Visualization of pressure distribution and overlaid streamlines around a highly variable geometry.

3.5.2 Criteria Group: Support by Vendor / Expert Network:

These ranking criteria are listed under Number 2 in the ranking matrix:

Criterion 2.1:   Technical support available: **ranking score = 5**

The technical support offered by CRADLE to the ERDL-HNEI CFD project team so far has been outstanding. Within the first weeks of cooperation the technical support has been very responsive in providing information, ranging from sending comprehensive training material and technical documentation to long support sessions on the phone.
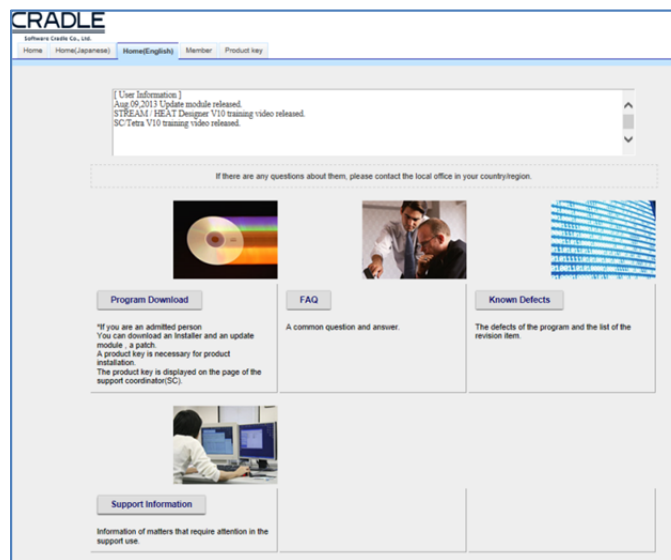
Figure 3.5.2:   Technical support option through the Design Builder website

Criterion 2.2:   Online tutorials available on demand: **ranking score = 2**

The software company offers a limited number of online tutorials. The tutorials, usually recorded webinars, can be obtained from the company. The tutorials are not posted on the support web-site at this moment.

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 4**

Online classes (synchronous training classes) are available. The online webcast are offered by the company on demand basis

<u>Criterion 2.4:</u>   Network of CFD professionals in architecture field: **ranking score = 3**

We could not identify a standing network of academics or professionals in architecture which uses scSTREAM for external CFD applications. Communication with US-CRADLE has indicated that the professional network in the US is in its initial stage. The software has had a US office for only 5 years (starting in 2008). The US-office has grown in the past years and with more use of CRADLE software in the US it is anticipated that the professional network will grow likewise.

<u>Criterion 2.5:</u>   Ease of working with academic support: *ranking score = 5*

The project team could not identify a specific academic technical support. The administrative academic support is accommodating and provides expeditious and very friendly service. The US manager of CRADLE has personally taken initiative to support our CFD team in acquiring a trial license for an unusually long period of two months. Usually trials are 1 or 2 weeks long.

<u>Criterion 2.6:</u>   User guide provides efficient and comprehensive information: **ranking score = 4**

The user guide of the software is very effective and comprehensive. The user guides are in static forma. An object hyper-liked user guide would be preferable, which means that every command is directly linked to the specific issue in the user guide. The CRADLE support has indicated that the object linked help is under development, but could not specify a time when a comprehensive object linked help function would be finally be available. The user guide contains all pertinent information to the level that is required to operate the external CFD module as well as some useful background information about the CFD process.

3.5.3 Criteria Group: Workflow-Ease of Use / GUI

These ranking criteria are listed under <u>Number 3.1</u> in the ranking matrix:

<u>Criterion 3.1.1:</u> GUI is easy to use for trained operator   *ranking score = 5*

The GUI is easy and intuitive to use by the trained operator. The program utilizes a so-called "Kicker" as the central gateway to all subsequent GUIs.

Figure 3.5.3A: scSTREAM Kicker GUI, first level controls and gateway to subsequent GUIs

The different process steps are controlled by the high-level buttons on the Kicker. For example:

Figure 3.5.3B: scSTREAM Kicker GUI, button to initiate preprocessing

The preprocessing process initiates the geometry, setting up the computational domain and other tasks that have to be completed in preparation of running the simulation. Once the pre-processing process step is initialized, a wizard with a series of interactive menus collects the required data for the flow problem. Two of the wizard menus are depicted below:
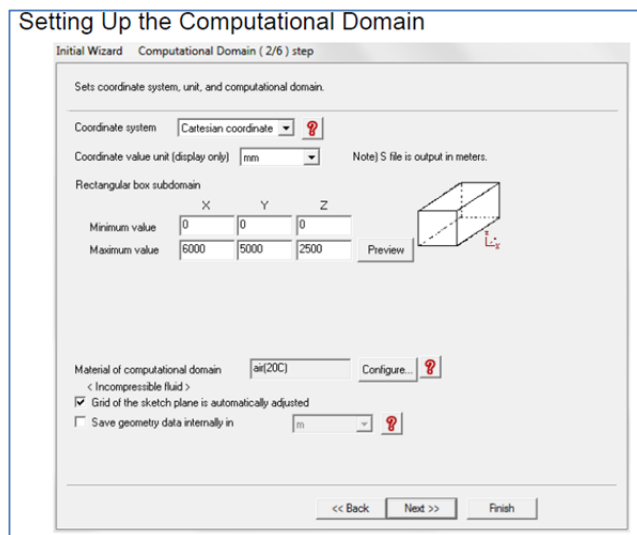
Figure 3.5.3C: scSTREAM pre-processing wizard menu; here setting up the computational domain

Criterion 3.1.2: User guide and online help is available for each command   *ranking score = 3*

The user guide and online help are effective. The subject linked help option is not implemented for all commands. According to
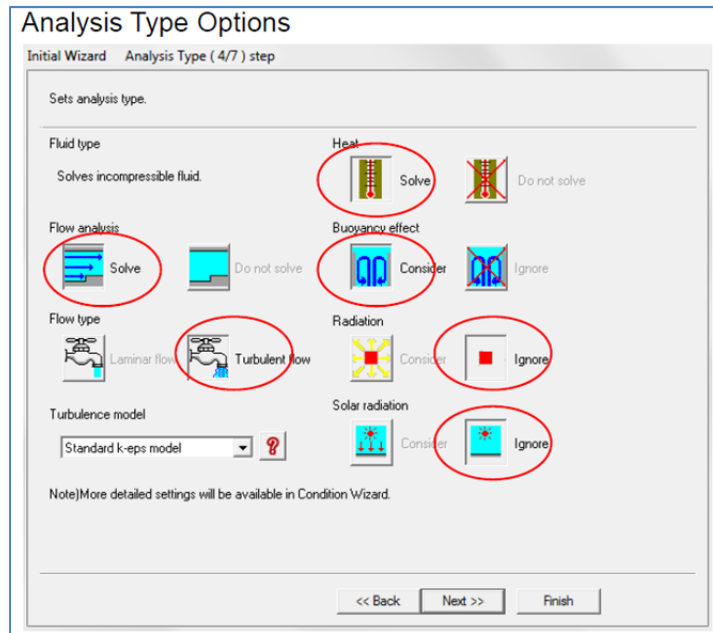
Figure 3.5.3D: scSTREAM pre-processing wizard menu; Analysis Type Options

3.5.4 Criteria Group: Workflow-Modeling / Import Geometry

These ranking criteria are listed under Number 3.2 in the ranking matrix:

Criterion 3.2.1:   Create geometry inside CFD application   *ranking score = 4*

The geometry modeler inside scSTREAM offers an effective way to create simple geometries. Since external CFD requires some degree of simplification of external geometry the internal modeler is sufficient and in the same way effective for simple and straight geometries. The software geometry modeler is not suitable for complex and especially curved building geometries. Standard roof forms can be created with entering a limited set of parameters. More complicated roof forms can be created with cubic elements that are combined (Boolean, merging of forms). DFX functions can greatly assist the geometry building.

Criterion 3.2.2:   Import geometry / repair surface before meshing   *ranking score = 3*

Importing 3D-geometry from external CAD programs follows a straight forward process. There is no repair tool in scSTREAM. As for analyzing the geometry, it is easy to check if the model is solid or sheets by using the parts icons in the parts tree in the Preprocessor interface ("Pre"). Also, if an error is detected in an imported file, an error message will be presented in the Message Window.

<u>Criterion 3.2.3:</u>   Support Revit Architecture Import geometry / repair surface before meshing

ranking score = 5

scSTREAM can readily import Revit files. In addition scSTREAM can import geometries from Solid Works and AutoCAD.

<u>Criterion 3.2.4:</u>   Support CADD format and Inventor: *ranking score = 1*

CAD geometry created with Inventor cannot be readily imported into scSTREAM

<u>Criterion 3.2.5:</u>   Support Rhino: *ranking score = 1*

CAD geometry created with Rhino cannot be readily imported into scSTREAM

3.5.5 Criteria Group: Workflow-Meshing

These ranking criteria are listed under <u>Number 3.3</u> in the ranking matrix:

<u>Criterion 3.3.1:</u>   Surface mesher   *ranking score = 4*

scSTREAM does not have a surface mesher function. The surface created in scSTREAM cannot be automatically improved and there are no analysis and /or surface editing functions to repair the surface. scSTREAM does not create a surface mesh. In scSTREAM the solution is discretized in a finite volume grid. Moreover, scSTREAM uses a non-uniform rectilinear Cartesian grid, which means that the grid lines are parallel with the major axes and the spacing between the grid lines enables non-uniformity.

Due to the strict rectilinear nature of the grid, grid cells that lie in regions outside of the domain required for calculation are „blocked-off" in order to cater for irregular geometries. The advantage of the Cartesian grin function of scSTREAM is the reported very good calculation performance, in regard to required time to reach a solution and the stability of the convergence. ScSTREAM has multiblock and zooming which allow the setting of different resolutions used in the computation domain. The higher resolution in areas where more precise flow phenomena are preferred creates a convenient, yet accurate and effective computational process. The illustrations below show the multi block and zooming functions.

The Multiblock meshing method is capable of increasing grid density in desired regions without using unnecessary elements in other regions, resulting in reductions in running time. But the aspect ratio of cells (between multi blocks) should not exceed a certain value, unique for the simulation in order to avoid performance of convergence and calculation accuracy.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                School of Architecture, University of Hawaii

December 4, 2013   Page 125
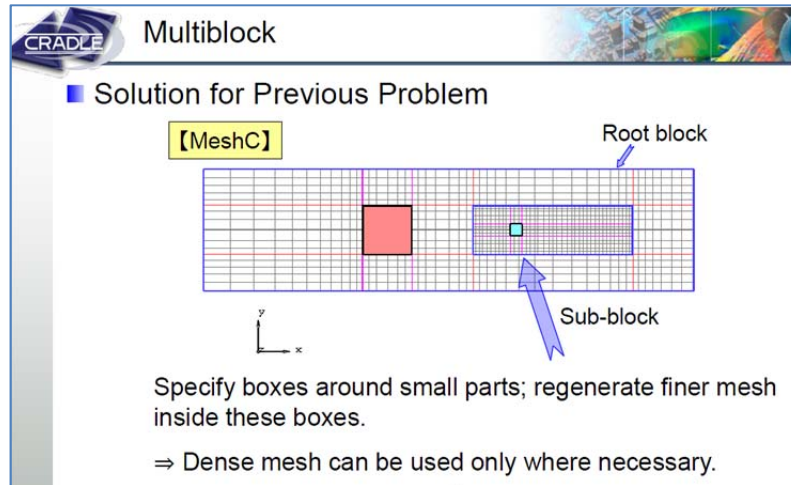
Figure 3.5.5A: scSTREAM Multiblock option
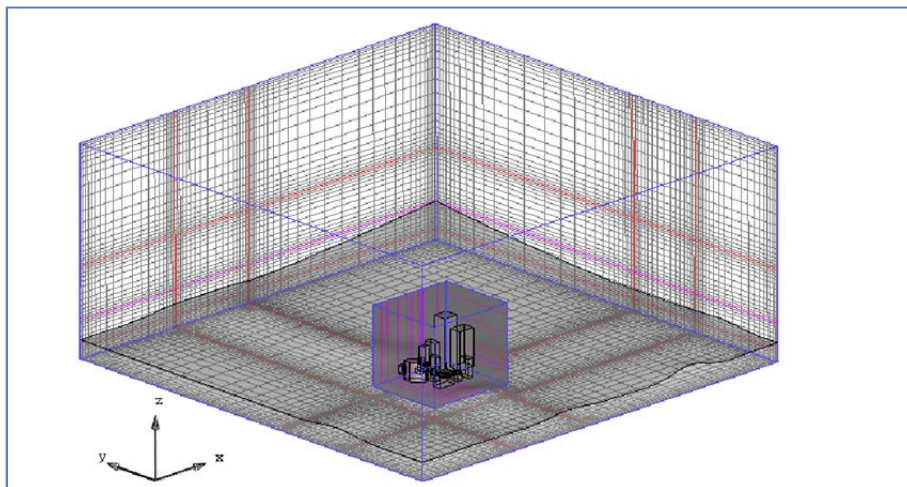
Figure 3.5.5B: scSTREAM Multiblock option

The image below shows that several blocks can be nested to achieve successively higher resolution around the specific building to be modeled, while preserving an appropriate resolution outside the specific area of interest.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                School of Architecture, University of Hawaii

December 4, 2013   Page 126

Figure 3.5.5C: scSTREAM Multiblock option

The Zooming (or mapping) function enables the user to link field parameters of different mesh sizes; it can link global large-scale analysis and local small-scale analysis. Zooming is a versatile function to calculate local occurrence in the context of larger fluid flow phenomena.

Supported conditions：
- Flow boundary
- Thermal boundary
- Initial Condition

Model variables：
- Velocity (components in X, Y, Z)
- Pressure
- Temperature
- Turbulent kinetic energy
- Turbulent dissipation rate
- Diffusive species concentration
- Heat transfer coefficient
- External temperature

When using the Multiblock and Zooming functions the user has to consider certain advantages and disadvantages. For example, Zooming is very efficient for changing a building geometry, since the changes require only a partial re-run of the simulation to get an update of the flow phenomena adjacent to the building. Zooming does not re-calculate the effects of the building on the far downstream; e.g. the effect of the building outside the zooming region. The Multiblock function requires a re-run of the entire simulation for the changed building geometry, while a re-run of the entire model also correctly shows effects of building on the far downstream regions.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                         School of Architecture, University of Hawaii

December 4, 2013   Page 127
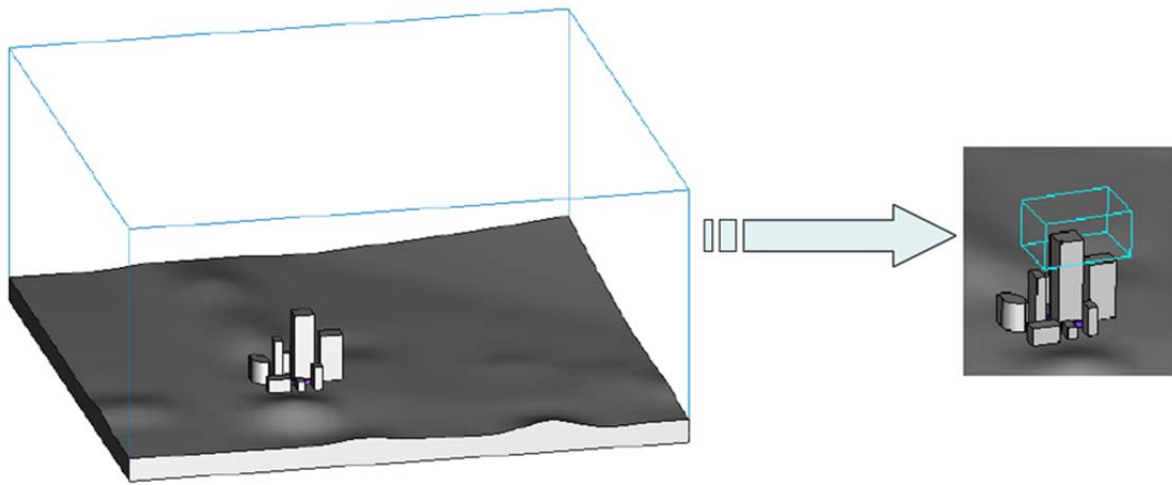
Figure 3.5.5D: scSTREAM Zooming function

Criterion 3.3.2:   Maximum cells and 2D-3D cell volume ranking *score = 5*

scSTREAM can use a large number of cells. The vendor indicated that 0 to 40 million cells usually do not create convergence and runtime problems.

Criterion 3.3.3:   Boundary layer types   *ranking score = 5*

ScSTREAM offers several very versatile functions to describe boundary layer types that are of interest in the building industry. The following mentions some of these useful functions. The GUI to invoke the different boundary conditions for external CFD analyses is straightforward and the simplicity of setting the external boundary conditions has the advantage of being very user friendly. The boundary conditions for the computational domain can be set at different stages in the work flow.

At the initial stage the selection of the boundary conditions is performed with the initial wizard in pre-processing. The image below shows the standard types of the computational domain.

scSTREAM has the capability to import land and building geometry to define the boundary layer close to the ground. Elevations of the site as well as building geometry are imported through appropriate CAD and GIS data exchange protocols. The building and the site geometry is then merged into one continuous geometry.
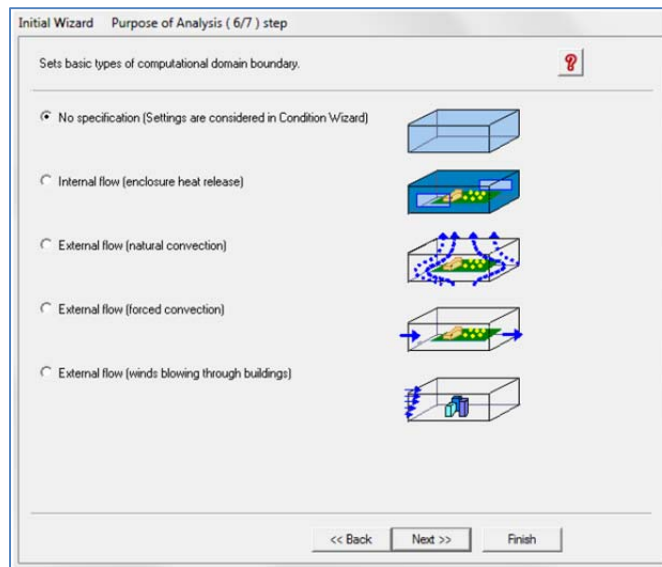
Figure 3.5.5E: scSTREAM Types of Computational Domain Boundaries
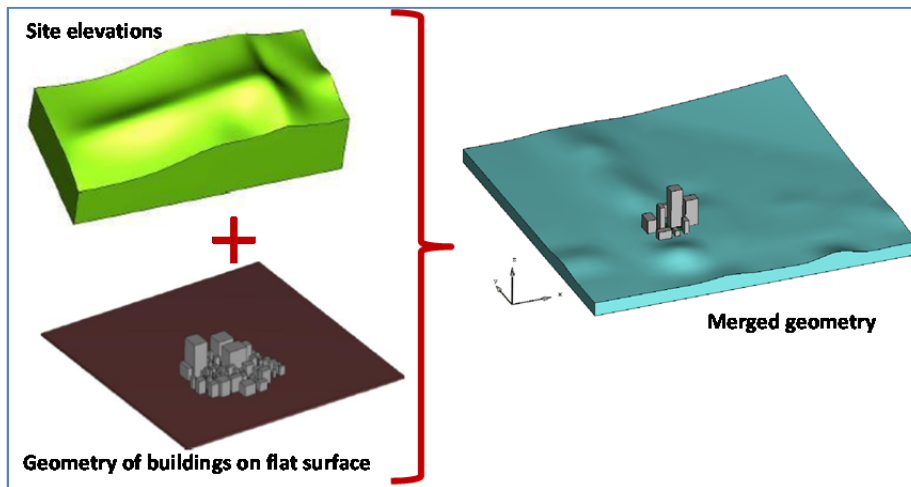
Figure 3.5.5E: scSTREAM Land and Building Geometry function to describe ground conditions of the computational domain

scSTREAM has a Power Law function to describe the inflow boundary condition as vertical profiles of velocity and turbulence density. For wind flowing in an urban area, the velocity profile can be expressed by a vertically displaced power law. Using this function, the flow field around buildings is defined by velocity and turbulence vertical distributions.
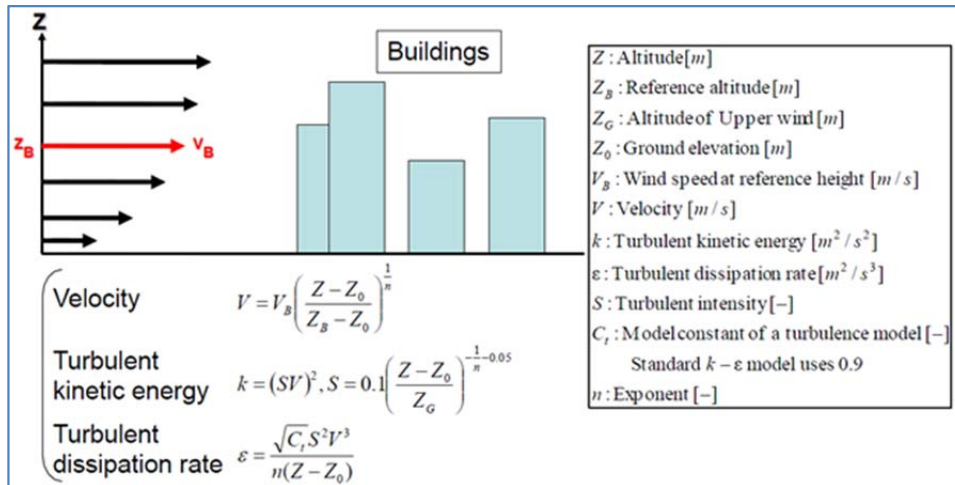
Figure 3.5.5F: scSTREAM Power Law function to describe the inflow boundary conditions

scSTREAM has a Forest Canopy Model function that considers aerodynamic resistance of canopy during an analysis. This function is very versatile to readily express the effects of canopy on the wind conditions around a building. The software considers air flows over, as well as through, a forest canopy. This results in volumetric pressure loss and an increase in turbulence.
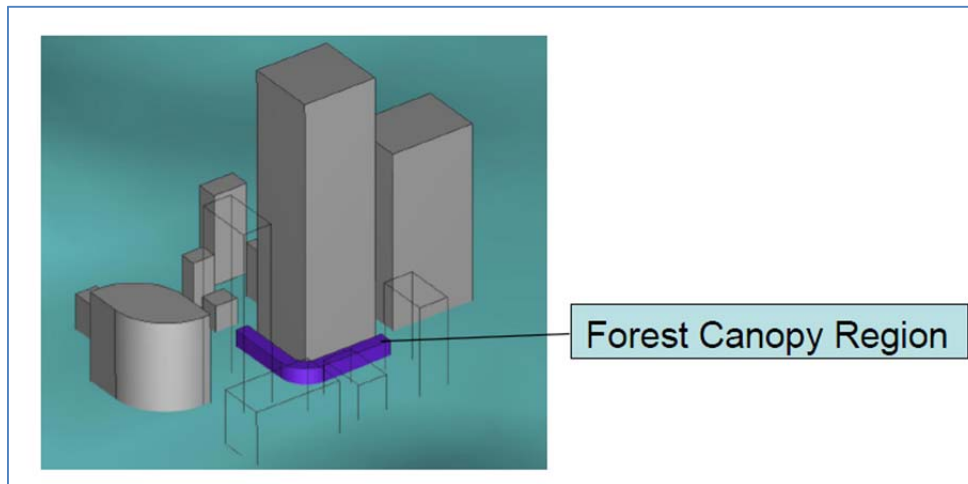
Figure 3.5.5G: scSTREAM Forest Canopy Model function enhances the geometry of the domain

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                              School of Architecture, University of Hawaii

December 4, 2013   Page 130
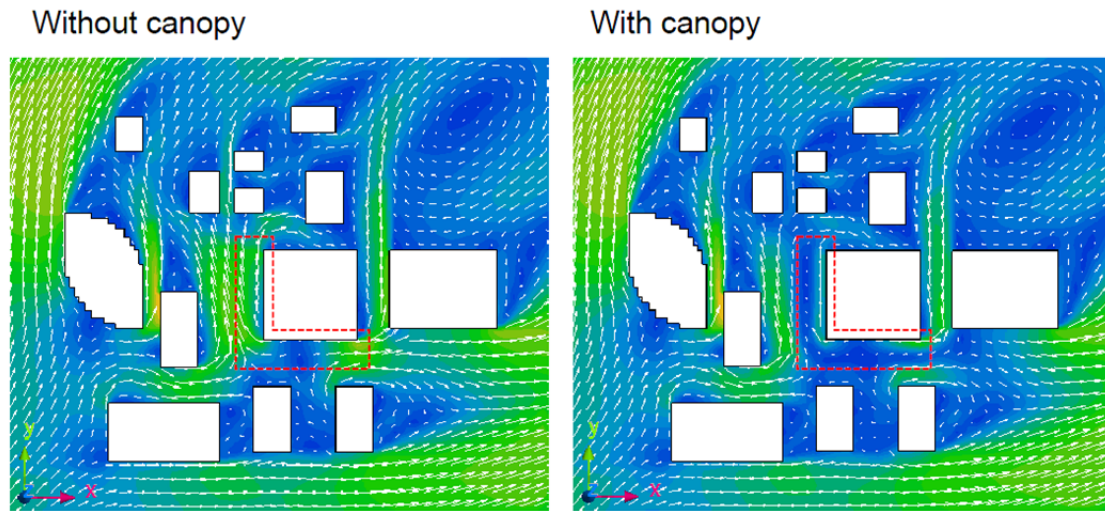
Figure 3.5.5G: scSTREAM Forest Canopy Model function; example of effects of canopy on flow regime close to the building

Criterion 3.3.4:   Structured and Unstructured meshing   *ranking score = 4*

scSTREAM uses a structured mesh. The conventional drawback of a structured mesh can be compensated for by the before-mentioned tools to define a non-uniform mesh.

Criterion 3.3.5:   Adaptive meshing capability   *ranking score = 4*

scSTREAM has several powerful functions (e.g. Multiblock and Zooming) to adapt the resolution of the mesh in areas that are of primary interest to the flow simulation around buildings.

3.5.6 Criteria Group: Workflow-Physics Models

These ranking criteria are listed under Number 3.4 in the ranking matrix:

Criterion 3.4.1:   Solvers - Coupled / decoupled   *ranking score = 5*

scSTREAM supports coupled numeric domains. The mesh can be defined with the Multiblock and/or zooming function.

Criterion 3.4.2:   Solver-Finite Element Method (FEV) / Finite Volume Method (FVM) *ranking score = 5*

scSTREAM uses Finite Volume Method (FVM) for the discretization of the computational domain.

Criterion 3.4.3:   Steady State/Transient State   *ranking score = 3*

scSTREAM can be used for steady -state as well as for unsteady simulations.

Criterion 3.4.4:   Turbulence models State   *ranking score = 1*

scSTREAM offers a range of turbulence models. The following is a list that has been reported by the company as standard turbulence models provided by the software product:

- Standard k-e
- RNG k-e
- MP k-e
- Linear low Reynolds no. model (AKN)
- Nonlinear low Reynolds no. model (AKN)
- LES (Smagorinsky)
- LES (Dynamic Smagorinsky)
- LES (WALE)
- LES (Mixed time scale model)

It should be noted that the software running performance is optimized through the use of the structured Cartesian mesh. Therefore use of LES turbulence models seems to be a valid option. For other software products the use of LES turbulence models would command such high computational resources which would surpass the project team's ability in computer performance. Testing LES models would be a very interesting and valuable option for the project team.

Criterion 3.4.5:   material   *ranking score = 5*

scSTREAM has a built-in library of generic materials and materials that can also be created and /or edited.

Criterion 3.4.6:   Customizable / special physics models   *ranking score = 4*

scSTREAM offers customization of special physics models.

3.5.7 Criteria Group: Workflow- Post Processing

These ranking criteria are listed under Number 3.5 in the ranking matrix:

Criterion 3.5.1:   Visualization   *ranking score = 5*

scSTREAM offers all standard CFD visualization techniques, including slices, contours, vector fields, 3D-contours, streamline visualization as well as animated visualization options. Below are some examples of visualizations provided by the vendor:
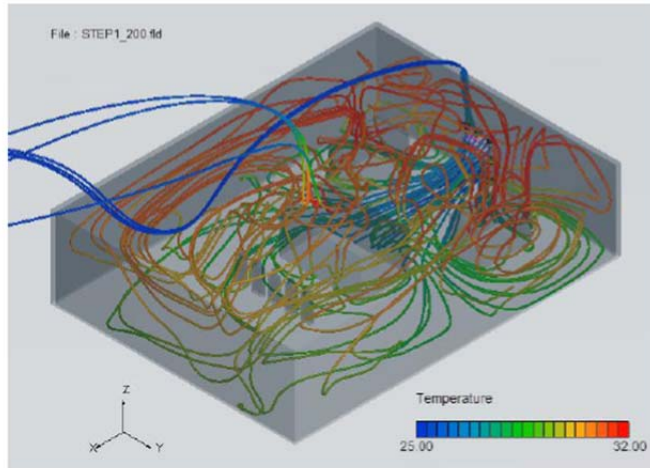
Streamlines (with temperature contours)
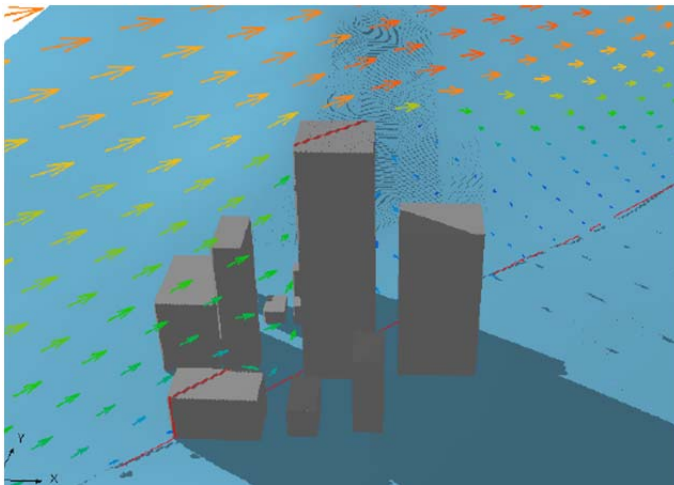
3D-vector field for the vertical velocity distribution around a group of buildings
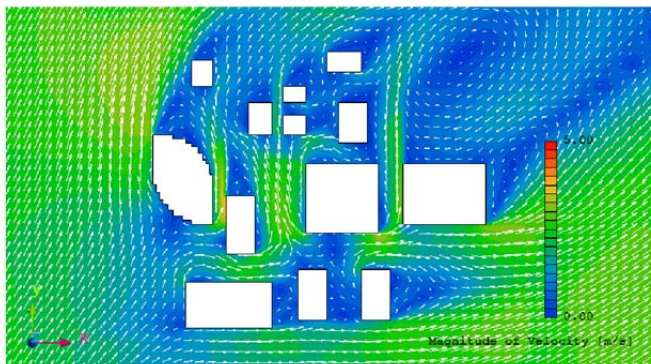
Combined 2D-velocity vector field (indicating direction) and velocity contours outside a group of buildings.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 133
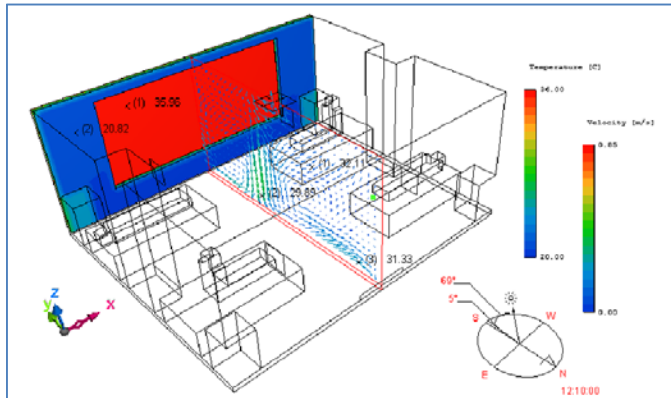
Combined contour surface map of solar irradiance on wall and velocity vector field of thermal induced velocity
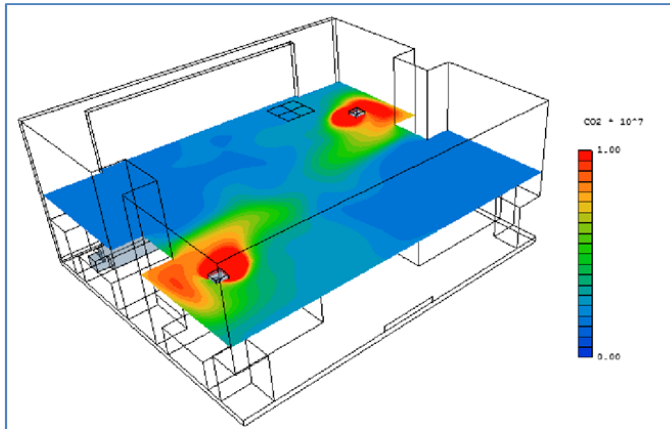
Horizontal slice with contours showing CO_2 concentration in a space.

scSTREAM provides visualization functions for a wide range of flow, thermal, mass distribution as well as related functions, such as:

- PMV
- PPD
- Comfort Indices
- Mean Radiant Temperature
- Operative Temperature

scSTREAM has a wide range special visualization function:

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 134
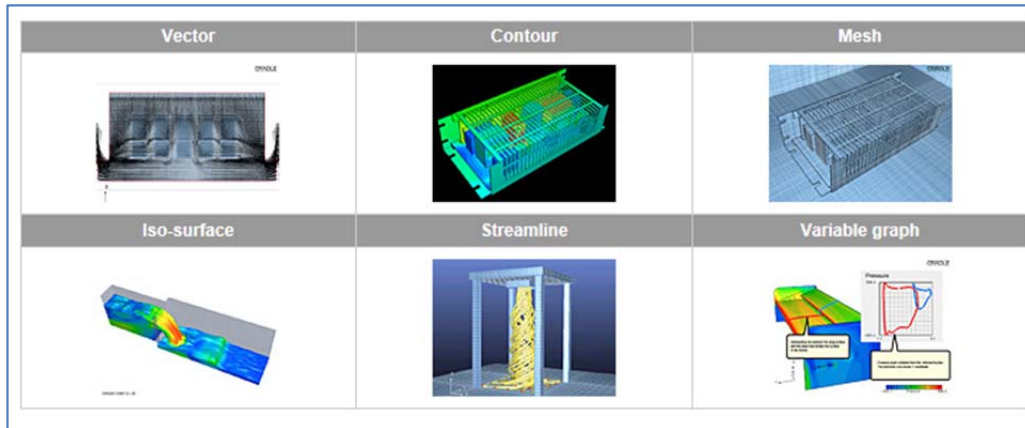
Figure 3.5.7A: scSTTREAM basic visualization functions
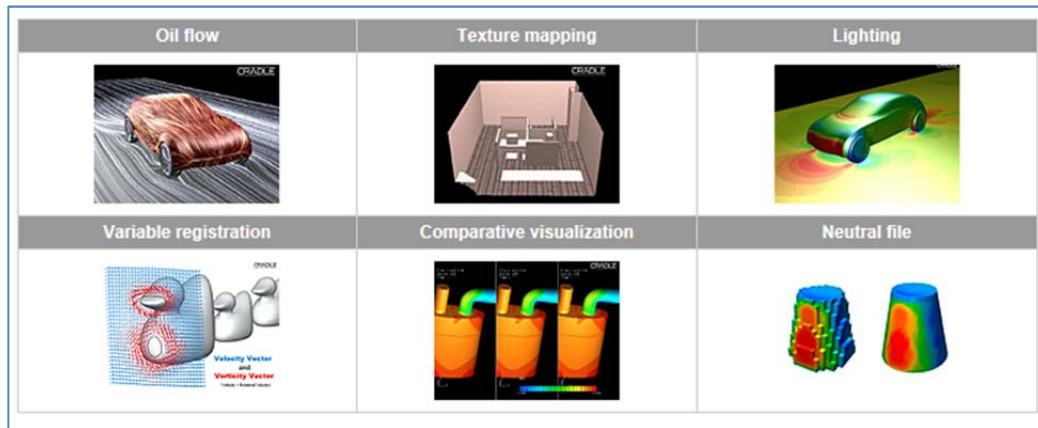
Figure 3.5.7B: scSTTREAM other static visualization functions
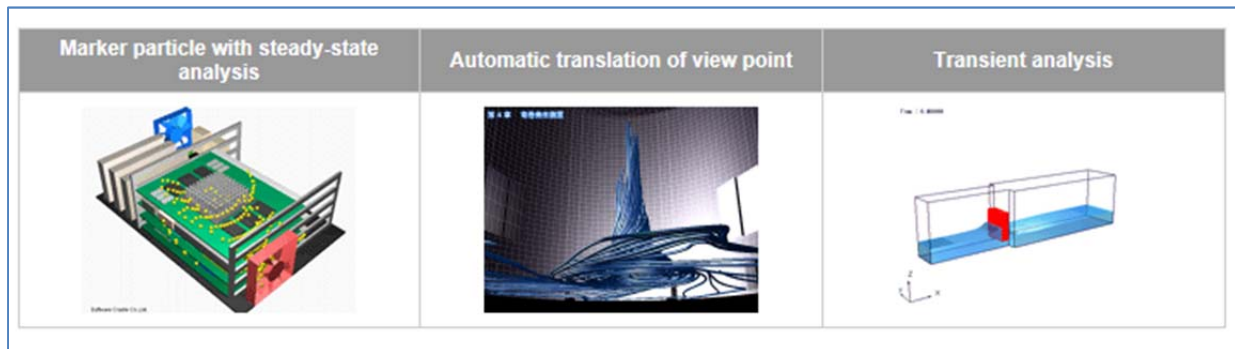
Figure 3.5.7C: scSTTREAM animation visualization functions

Criterion 3.5.2:   Quantitative analysis capabilities of components   *ranking score = 5*

scSTREAM offers versatile and easy to use tools to determine the quantities of certain properties of the flow field and resulting phenomena. The software allows the extraction of numerical values of all calculated data for every point inside the computational domain.

Criterion 3.5.3:   Comparative analysis (multi-scenarios)   *ranking score = 4*

scSTREAM offers some functions that can be used for comparative analysis.

**3.5.8 Criteria Group: Workflow-Customization / special functions**

These ranking criteria are listed under Number 3.6 in the ranking matrix:

Criterion 3.6.1:   Pedestrian comfort   *ranking score = 5*

scSTREAM has a function called "WindTool". This function provides an assessment index of the wind environment as proposed by Murakami et al. (Reference: Murakami et al., Investigation of statistical characteristics of wind at ground level and criteria for assessing wind-induced discomfort, Transactions of the Architectural Institute of Japan 325, 1983, pp.74-84.) In order to use different wind comfort standards the software can be customized.

Criterion 3.6.2:   Natural ventilation   *ranking score = 5*

scSTREAM has the ability to provide quantitative performance criteria for wind induced natural ventilation from using the results of external CFD. As stated before natural ventilation performance of spaces can be simulated through either coupled or decoupled domains. Below is an illustration of using scSTREAM with a decoupled geometry block.
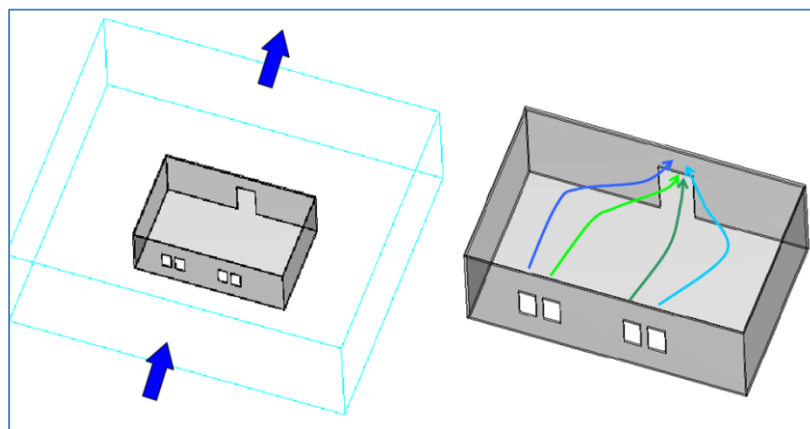
Figure 3.5.8B: scSTREAM calculating ventilation effectiveness through wind induced pressures fields

Criterion 3.6.3:   Wind-driven rain   *ranking score = 4*

scSTREAM has no readily available tool to assess wind driven rain. But the physical conditions can be set up to effectively model wind driven rain.

Criterion 3.6.4:   Humidity   *ranking score = 5*

scSTREAM has the ability to provide an assessment of the distribution of and changes in humidity. The Humidity and Dew Condensation (Evaporation) function can calculate relative humidity contours and dew condensation. This function is available for both external and internal CFD applications. The two images below show a case of changes in relative humidity and condensation

Relative humidity contour plots   Dew Condensation (after 10 mins.)
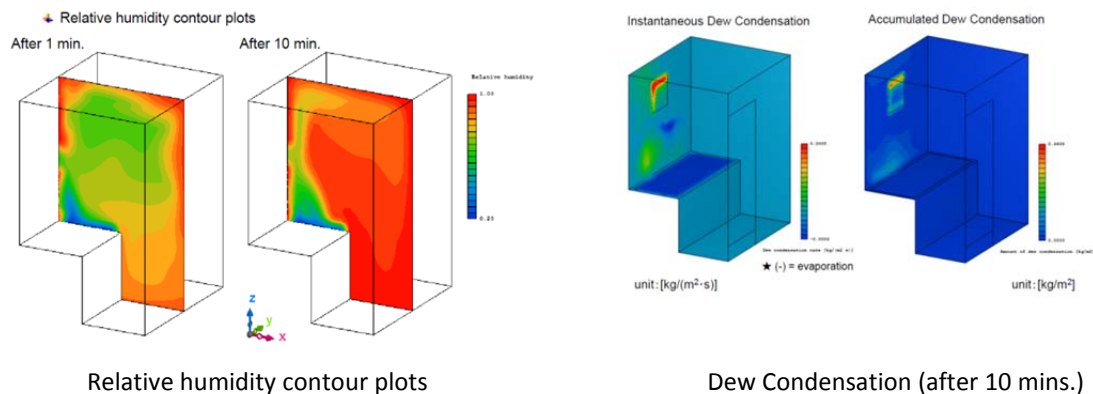
Figure 3.5.7C: scSTREAM Humidity, Dew Condensation (Evaporation) Functions

Criterion 3.6.5:   Wind load towards to building structure   *ranking score = 5*

scSTREAM provides comprehensive calculation of airflow and external wind induced pressure calculations can on buildings. The calculation of wind pressures on the building envelope is performed through common software applications and can be readily be carried out.

Criterion 3.6.6:   Pressure distribution along streamlines/pressure losses   *ranking score = 4*

scSTREAM has functionality to provide an assessment of the pressure distribution along streamlines or determine pressure losses   along a certain streamline. This option is important for the assessment of pressure losses throughout a building.

Criterion 3.6.7:   Conjugate heat transfer   *ranking score = 5*

scSTREAM can perform complex calculations that combine flow and thermal as well as diffusion processes.

3.5.9 Criteria Group: Workflow-Support for validation

These ranking criteria are listed under Number 3.7 in the ranking matrix:

Criterion 3.7.1:   Validation document / example   *ranking score = 5*

Cradle (vendor of scSTREAM) has indicated that a number of case studies can be provided that address validation of external CFD results.

3.5.10 Criteria Group: Ranking of *Cost of software*

These ranking criteria are listed under Number 4 in the ranking matrix:

Criterion 3.7.1:   Commercial pricing model   *ranking score = 3*

scSTREAM offers commercial pricing for its CFD that are within the comparable price ranges. The CFD functions are competitively priced, which include both external and internal CFD

Criterion 3.7.2:   Academic pricing   *ranking score = 3*

scSTREAM offers a significant discount for academic software use. The company has been very flexible in the pricing and the availability of software use in the early stages of the project.

 Criterion 3.7.3:   Software includes all modules (no extra costs)   *ranking score = 3*

scSTREAM comes with basically all functions that are needed. There might be additional modules that are useful to the project team (=application of that software in building related CFD simulations). It is not known at this point in time if academic pricing would include additional modules.

Criterion 3.7.4:   Option to acquire sufficient seats at no extra costs   *ranking score = 3*

It is not known at this point in time if an academic pricing would include additional seat for the use of students in education.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 138

3.6 Ranking of Cradle SC/Tetra:

3.6.1 Criteria Group Value of CFD Software Product for Architecture

Criterion 1.1:   Benefit for external CFD   **ranking score = 5**

Cradle scTetra can solve a wide a wider range of applications from mechanical/chemical engineering, automobile to internal and external airflow studies of buildings. SC/Tetra is featured with unstructured meshing capability for complex geometries and advanced turbulence models which are essential features in external CFD applications.

Criterion 1.2:   Previous application in architecture / urban design   **ranking score = 2**

Cradle SC/Tetra is currently used mostly in automotive, aerospace, marine, machinery, oil, gas, bio-medical industries. However, it is also claimed that one of its applications include analysis of architectural building climate control such as evaluating air-condition performance within spaces, heat storage performance, ventilation studies. However, the CFD team could not identify wider popularity of the software within architectural/urban design community, especially in the United States. The Cradle company has been making efforts to introduce their software including SC/Tetra to architectural / urban design industry.

Criterion 1.3:   Previous mentioning of software product in scientific or applied professional articles

ranking score = 1

Through searching ScienceDirect research sources, the author has not found evidence of using SC/Tetra in any journal of scientific or applied fields. The ranking of this criterion is assumedly based on this fact. However, further investigation on other sources should be taken.

Criterion 1.4:   Using graphics output for presentation   **ranking score = 5**

Cradle SC/Tetra offers a post-processor called Cradle Post-Processor to visualize the data from Cradle solver. It features with many output formats such as contour, vector, iso-surface, streamline, animation as well as many other advanced visualization functions.
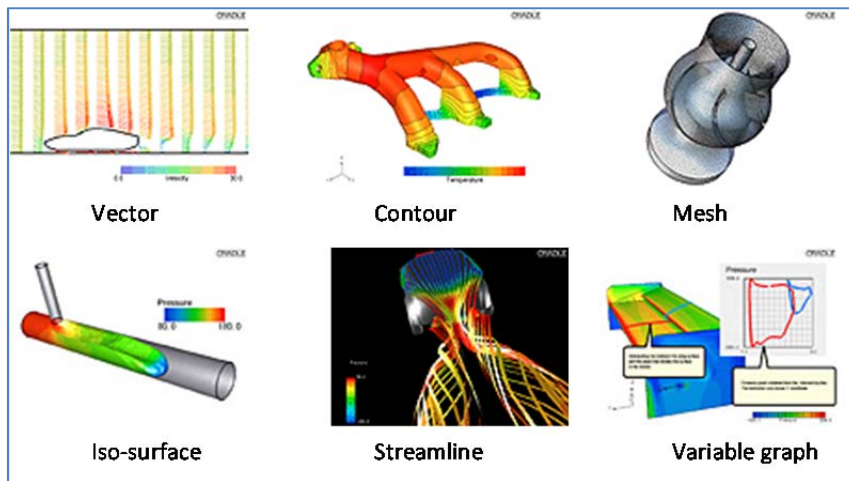
Figure 3.6.1:   Some examples of visualization capability offered by SC/Tetra
                 Source: http://www.cradle-cfd.com

3.6.2 Criteria Group Support by Vendor / Expert Network

Criterion 2.1: Technical support available: **ranking score = 3**

Technical support is available at the company website covering a wide range of different topics. However, this is only Japanese version and there is no English version available at this moment. The only way to use this resource is using the built-in Google translate utility to translate to English in the web browser. Other way to obtain technical support is to submit online technical support request.
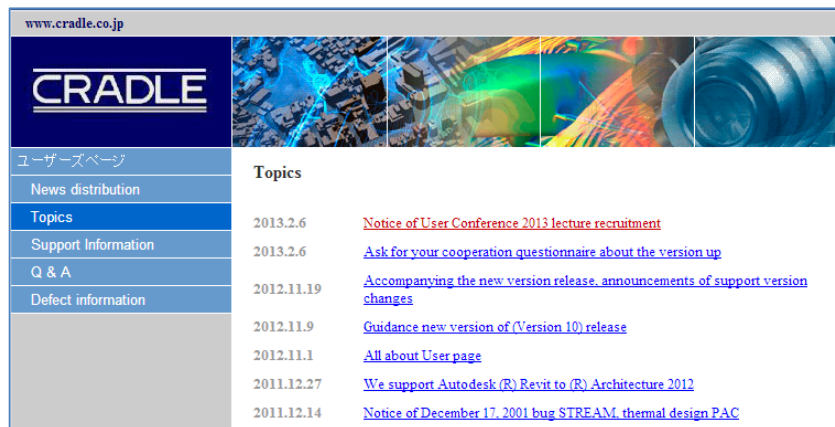
Figure 3.6.2A: Support information offered by Cradle. Source: http://www.cradle-cfd.com

<u>Criterion 2.2:</u>   Online tutorials available on demand   **ranking score = 4**

There are enough online tutorials available at the user support portal. They include a series of videos and PDF files covering from the fundamentals of CFD, basic tutorials of using all functionalities of the CFD applications. However, it is required to have license in order to log into the user support portal to access these resources. There is no public resource on Cradle SC/Tetra tutorial such as YouTube.
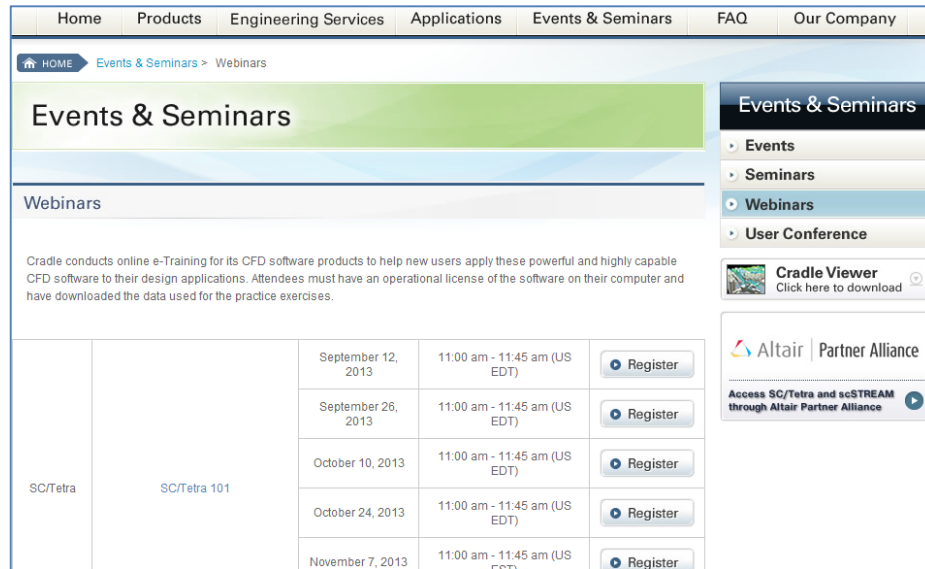
Figure 3.6.2B: Training tutorial materials can be downloaded from customer support portal of Cradle.
                          Source: http://www.cradle-cfd.com

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 5**

For beginners in CFD or people who would like to get familiar to Cradle CFD software, Cradle offers free webinars available at its website periodically. For example, SC/Tetra is introduced twice per month. More advanced courses however are introduced under seminars with cost. However, 85% discount is applied for academics.

Figure 3.6.2C: Introductory webinars are offered periodically by Cradle at no cost.
                   Source: http://www.cradle-cfd.com

Criterion 2.4:   Network of CFD professionals in architecture field: **ranking score = 1**

Cradle has been engaged in building a network of CFD professionals in the US. It has regularly organized many conferences each year since 1991. Several conference topics were related to architectural field, especially HVAC performance but most of the topics are related to other fields such chemical, electronic, gas, oil industries.

Criterion 2.5:   Ease of working with academic support: **ranking score = 5**

Cradle offers special pricing and support for academic users using and evaluating its software. The ERDL-HNEI CFD team has been quite satisfied with the support offered by Cradle.
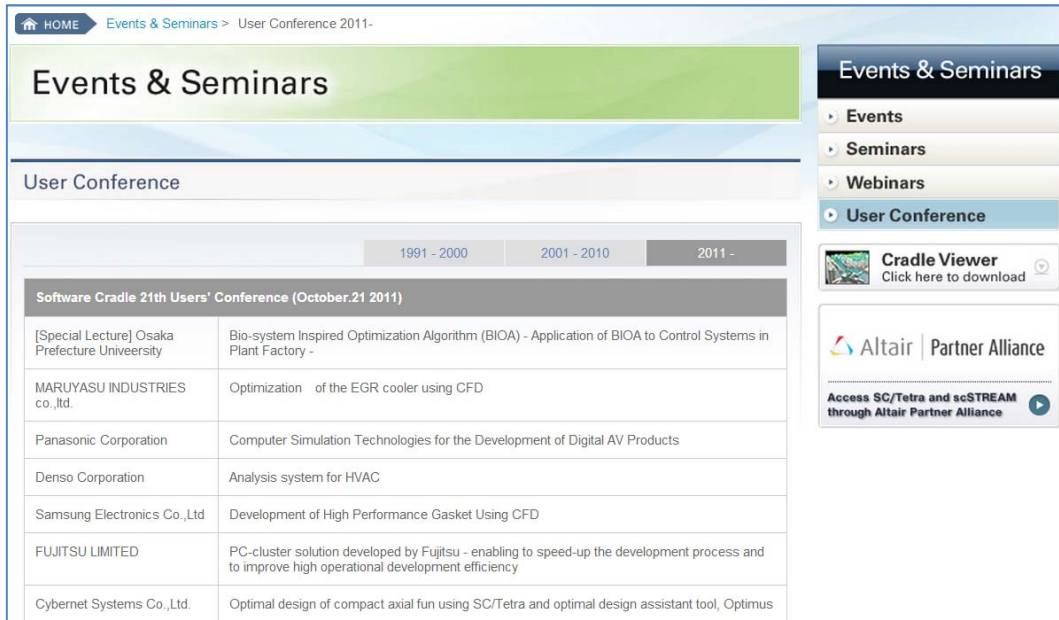
Figure 3.6.2D: Advanced seminars registration page. Source: http://www.cradle-cfd.com

<u>Criterion 2.6:</u>   User guide provides efficient and comprehensive information: **ranking score = 5**

Cradle SC/Tetra provides user guide along with its installation packages. The information is comprehensive and efficient for users with step by step instruction. However, this user guide also includes technical information which sometimes users might need for references.

Figure 3.6.2E: Technical manual, user guide, tutorials can be accessed easily at the Kicker toolbar.
Source: SC/Tetra Version 10 Introductory Seminar No. 1 (2013)

3.6.3 Criteria Group Workflow - Ease of use / GUI

Criterion 3.1.1: GUI is easy to use for trained operator   **ranking score = 5**

Cradle SC/Tetra provides user friendly GUI lining up with the CFD working flow which benefits to CFD operators to manage entire CFD process.
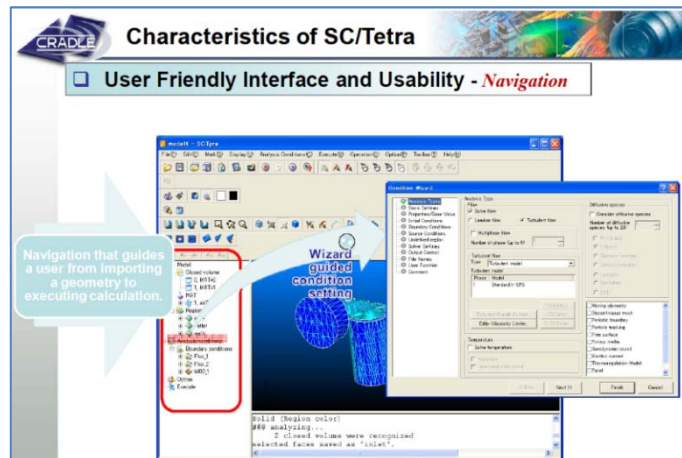
Figure 3.6.3A: SC/Tetra's GUI; Source: More innovation using SC/Tetra, the innovative CFD software (Yuya Ando, 2013)

Criterion 3.1.2: User guide and online help is available for each command   **ranking score = 4**

User guide is available at a quite comprehensive level. Users can refer to its available technical manuals available with the package after installation. These manuals include basics of CFD analysis introduction, basic and advanced exercises, preprocessor reference, solver reference, post-processor reference… Individual command can be explained at tool reference.
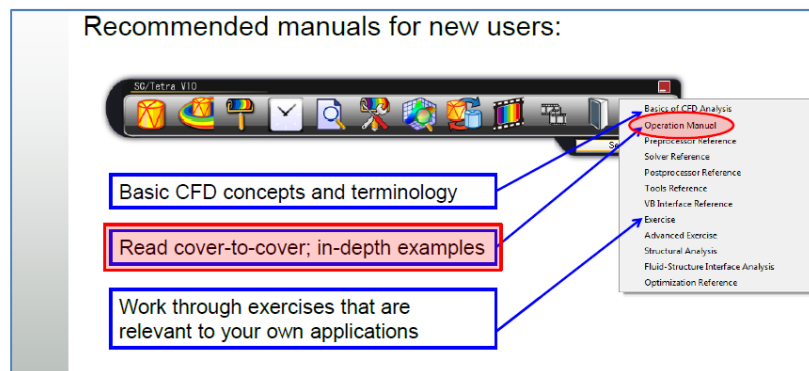
Figure 3.6.3B: User Guide assessable right at Kicker toolbar; Source: More innovation using SC/Tetra, the innovative CFD software (Yuya Ando, 2013)

3.6.4 Criteria Group Workflow – Modeling / Import geometry

Criterion 3.2.1: Create geometry inside CFD application   **ranking score = 1**

Cradle SC/Tetra does not have capability to create geometry inside CFD application.

Criterion 3.2.2: Import geometry / repair surface before meshing   **ranking score = 4**

Cradle SC/Tetra offers a robust capability to repair surface from imported geometry for meshing.
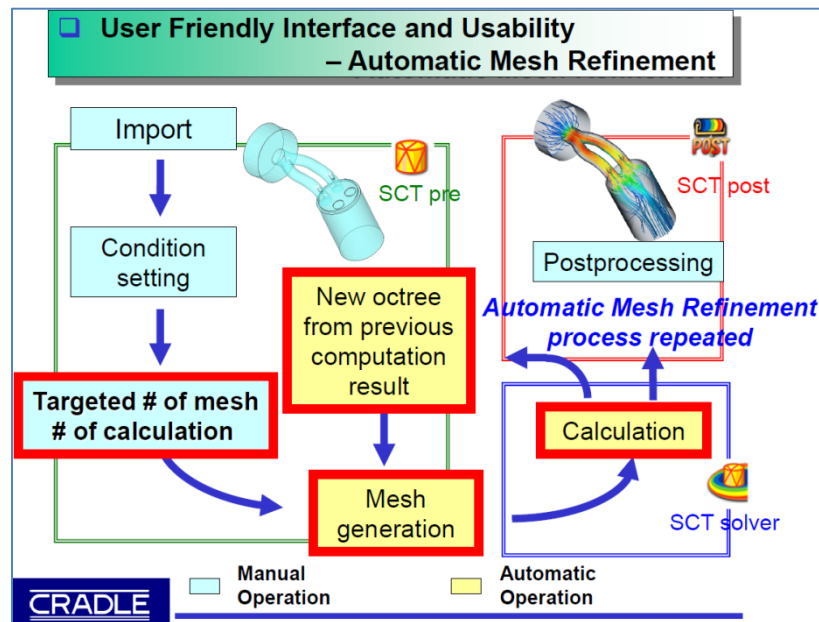
Figure 3.6.4A: Mesh preparation and preprocessing diagram; Source: SC/Tetra Short Course (Cradle, 2013)

Criterion 3.2.3: Support Revit Architecture   **ranking score = 2**

Cradle SC/Tetra does not directly support Revit format but architectural geometries from Revit products can be converted into other CAAD formats which can be then imported into SC/Tetra.

Criterion 3.2.4: Support CADD format and Inventor   **ranking score = 5**

Cradle SC/Tetra does not have capability to create geometry inside CFD application, however, it provides robust CADthru module to convert CAD data for CFD analysis.
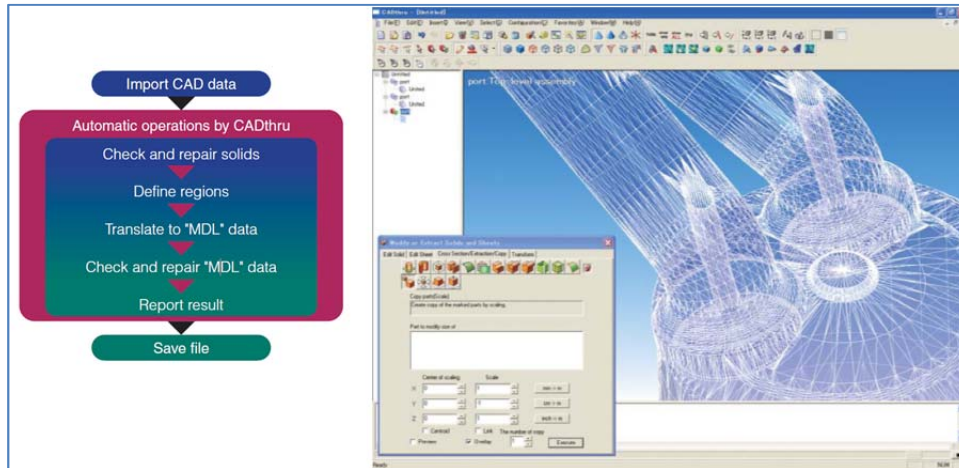
Figure 3.6.4B: Geometry importing workflow; Source: Cradle Production Guide (2013)

The following CAD formats are workable in SC/Tetra:

Figure 3.6.4C: List of supported formats for SC/Tetra; Source: Cradle Production Guide (2013)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                        School of Architecture, University of Hawaii

December 4, 2013   Page 146

Criterion 3.2.5: Support Rhino   **ranking score = 2**

Cradle SC/Tetra indirectly support Rhino3D format files by converting those files into other CAAD formats which can then be imported into SC/Tetra.

3.6.5 Criteria Group Workflow - Meshing

Criterion 3.3.1: Surface meshers   **ranking score = 1**

Cradle SC/Tetra provide robust surface wrapping meshing tool which allows for cleaning up geometries with holes, small edges and overlaps. This tool also is high efficiency in terms of using computer memory to do this task.
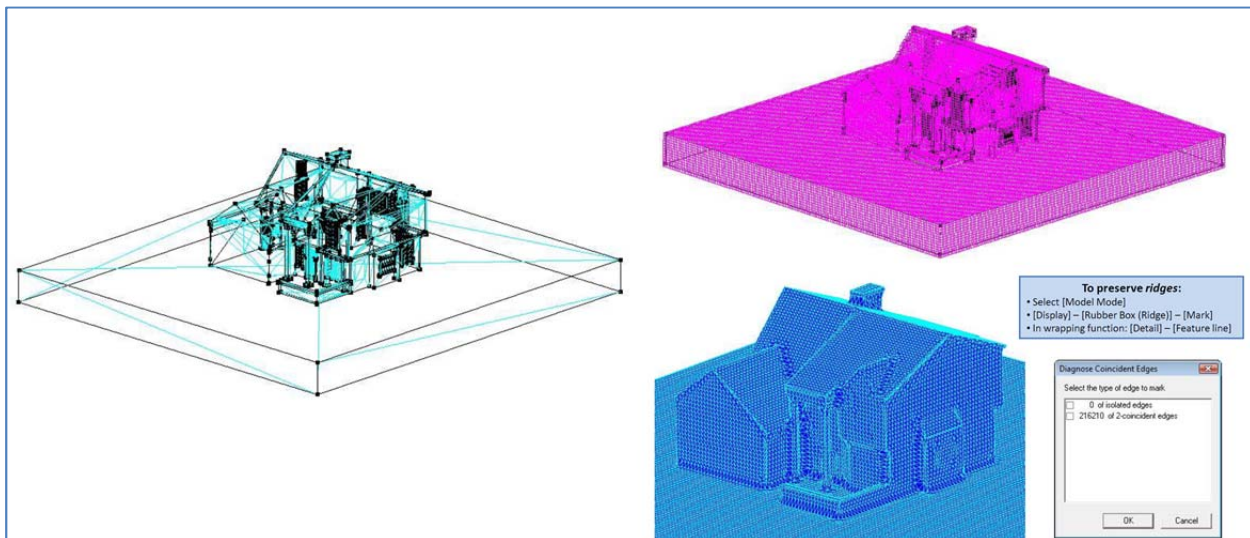
Figure 3.6.5A: Surface wrapping tool of SC/Tetra; Source: Cradle Production Guide (2013)

Criterion 3.3.2: Maximum cells and 2D-3D cell volume   *ranking score = 5*

Cradle SC/Tetra can handle a large number of cells depending on the availability of computer memory of the system. For example, a system with 64 GB of RAM memory can handle more than 300 million elements. Moreover, SC/Tetra achieves low memory usage and high speed computations by using cell vertex based scheme for the Finite Volume Method.

Criterion 3.3.3: Boundary types   *ranking score = 5*

Cradle SC/Tetra offers a wide range of boundary types which allow users to define the primary variables of the simulation. SC/Tetra's boundary types include the following features:

- Flow conditions: Velocity, volume flow rate, mass flow rate, pressure, air conditioner model, fan model, wave generation, wave dissipation
- Thermal conditions: fixed temperature, amount of heat generation, heat transfer coefficient, contact heat transfer coefficient
- Wall conditions: No-slip (stationary wall), free-slip (symmetry wall), log-law condition, power-low condition, surface roughness
- Pressure conditions: fixed pressure, pressure loss, porous media
- User-defined conditions: variable table, user-defined function (compilation required)

Criterion 3.3.4: Structured and Unstructured meshing   **ranking score = 5**

Cradle SC/Tetra provides unstructured meshing tool allowing for handling any kind of complex geometry. The supporting types of volume elements: tetrahedron, pentahedron (prism, pyramid), hexahedron, and cuboid.

Criterion 3.3.5: Adaptive meshing capability   *ranking score = 5*

Cradle SC/Tetra features with mesh adaptation which enables automatic optimization of the mesh size and density in regions where variations in flow and temperature are prominent. The user interface for this feature is very intuitive and the user only needs to set the targeted number of mesh elements.
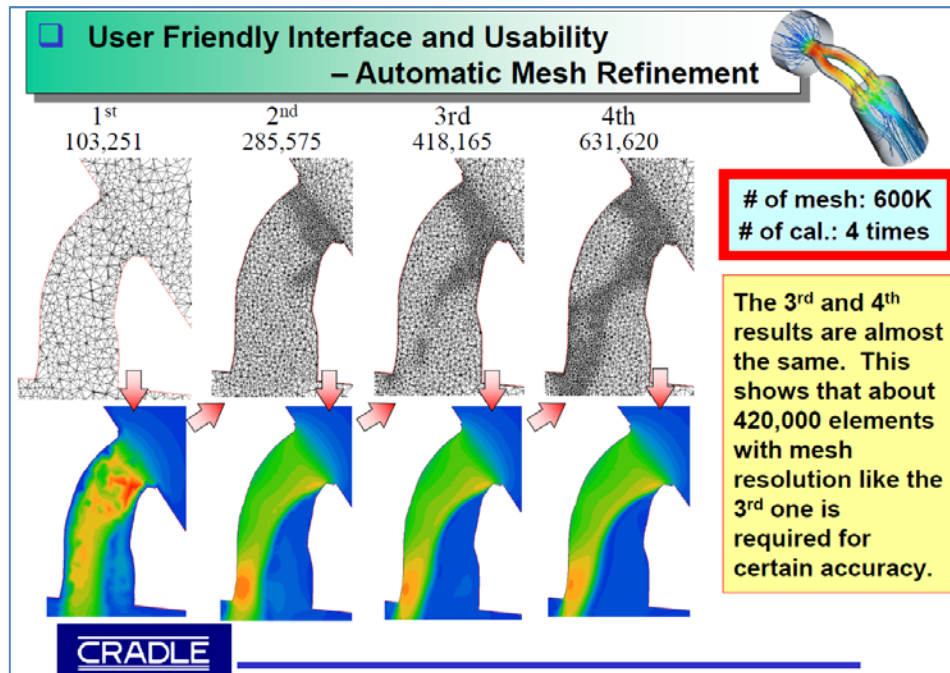
Figure 3.6.5B: Mesh adaptation example   Source: SC/Tetra Short Course (Cradle, 2013)

3.6.6 Criteria Group Workflow – Physics Models

Criterion 3.4.1: Solver-Coupled / decoupled   **ranking score = 5**

With robust meshing capability to handle any complex geometry with unlimited volume elements, SC/Tetra allows for modeling both internal and external airflow within a single computational domain with a highly flexible cell resolution at different areas of interest.

Criterion 3.4.2: Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)   **ranking score = 5**

Cradle SC/Tetra uses the Finite Volume Method (FVM) approach with cell vertex based scheme for numerical solution of the fluid dynamic equations.

Criterion 3.4.3: Steady state/Transient state   **ranking score = 5**

Cradle SC/Tetra offers steady state/ transient solution which allows for simulating a wide variety of time-dependent external airflow phenomena.

Criterion 3.4.4: Turbulence models   **ranking score = 5**

Cradle SC/Tetra features with the wide range of robust turbulence models as follows:
-    k-ε models (standard, RNG, MP, Linear low-Reynolds-number Non-linear low Reynolds-number, Realizable)
-    SST k-ω model
-    LES (Smagorinsky model, Dynamic Smagorinsky model, WALE model)
-    RNAS and LES hybrid turbulence model (VLES, DES)
-    Spalart-Allmaras 1-equation model
-    LKE k- kL-ω model

Criterion 3.4.5: Materials   **ranking score = 5**

Cradle SC/Tetra provides a comprehensive library of defined materials covering liquids, gases, and solids. Users also can add any user-defined / customizable materials for their models.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                School of Architecture, University of Hawaii

December 4, 2013   Page 149
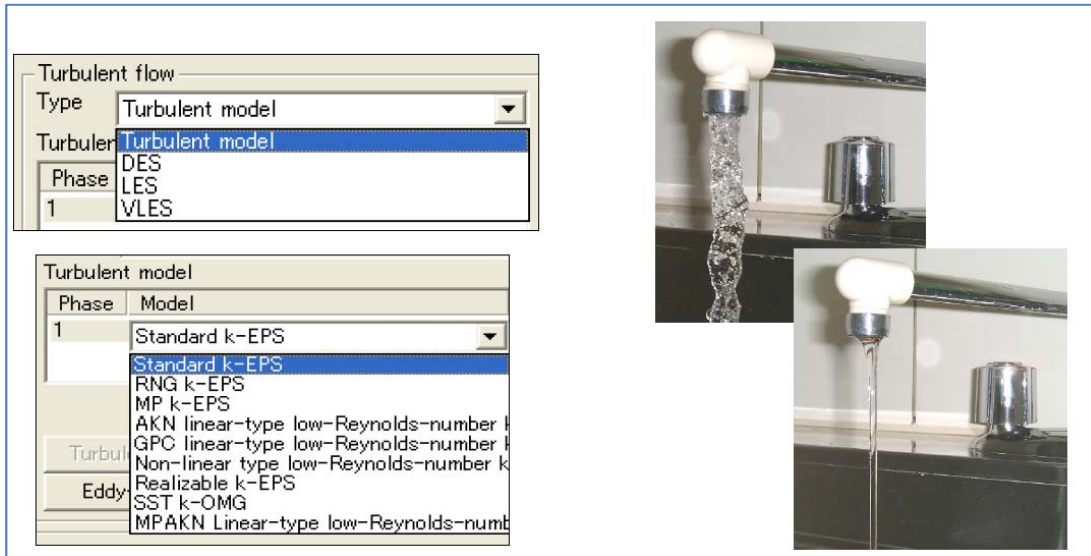
Figure 3.6.6A: Turbulence models supported by SC/Tetra; Source: SC/Tetra Short Course (Cradle, 2013)
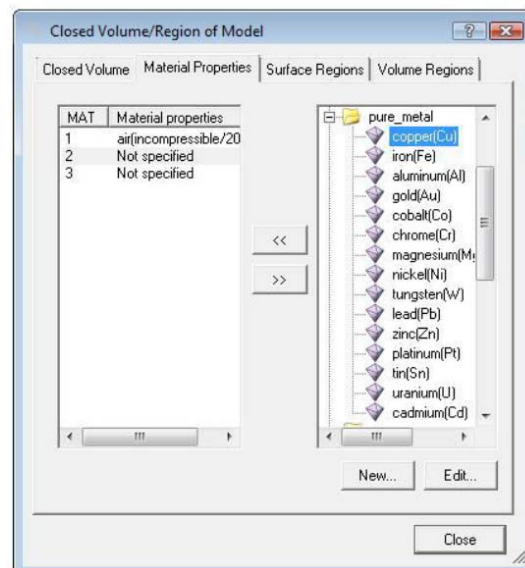
Figure 4.6.6B: Material selection dialogue; Source: SC/Tetra Short Course (Cradle, 2013)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                         School of Architecture, University of Hawaii

December 4, 2013   Page 150

<u>Criterion 3.4.6:</u> Customizable / special physics models **ranking score = 5**

Cradle SC/Tetra offer a wide range of many physic analyses such as thermal analysis (heat conduction, heat transfer, heat radiation, panel heat conduction, mean radiation temperature...) thermal comfort models (PMV, SET), humidity / dew condensation analysis (relative humidity, absolute humidity, dew condensation, humidity transfer in solid, vapor pressure under humidity analysis), reaction analysis (chemical reaction, combustion...) as well as user defined conditions.

3.6.7 Criteria Group Workflow – Post-processing

<u>Criterion 3.5.1:</u> Visualization **ranking score = 5**

Cradle SC/Tetra's visualization capabilities include basic features also found in other from other CFD applications, including vector plot, contour plot, streamlines, pathlines, isosufaces, oil flow, animation...
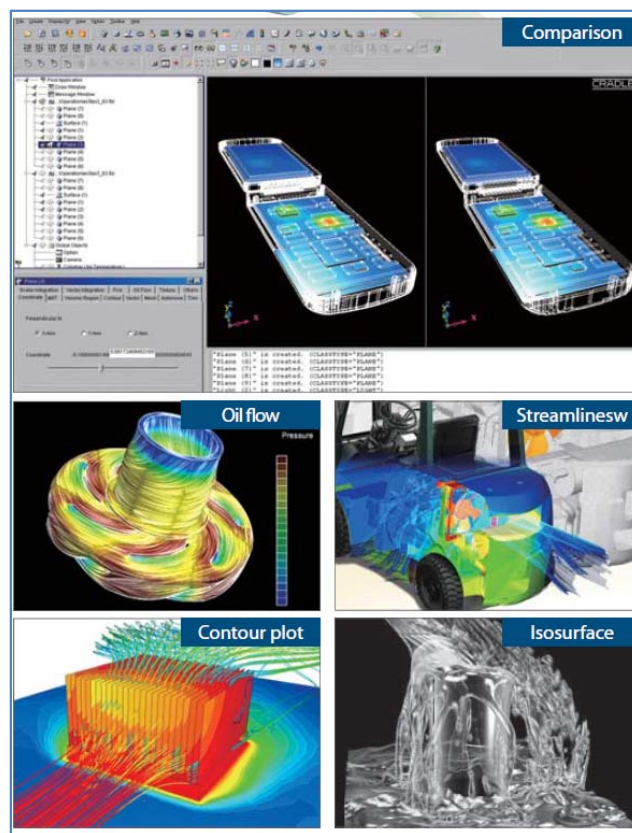
Figure 4.6.7: Examples of visualization features Source: Cradle Production Guide (2013)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
December 4, 2013   Page 151

<u>Criterion 3.5.2:</u> Quantitative analysis capabilities of components   **ranking score = 4**

Cradle SC/Tetra's post-processor provides a variety of advanced functions to analyze the calculation results. This is useful for CAE experts and for third parties viewing the results or other types of analysis.

<u>Criterion 3.5.3:</u> Comparative analysis (multi-scenarios)   **ranking score = 5**

Cradle SC/Tetra's post-processor allows for comparative analysis to different fluid dynamic problems. This capability is a powerful capability for analyzing different scenarios with less time and effort.

3.6.8 Criteria Group Workflow – Customizable / special functions

<u>Criterion 3.6.1:</u> Pedestrian comfort   **ranking score = 5**

Although Cradle SC/Tetra does not provide dedicated pedestrian comfort model, its flexible and user-defined function capability allows for any study of wind related pedestrian comfort.

<u>Criterion 3.6.2:</u> Natural Ventilation   **ranking score = 5**

Cradle SC/Tetra is capable to solve wind-driven and buoyance-driven natural ventilation by offering thermal comfort model (PMV, SET), humidity and dew condensation, coupled computational domain modeling capability, heat transfer – conduction, convection (natural and forced), radiation (solar, thermal), conjugate heat transfer, indoor air quality (mean age of air) and so on.

<u>Criterion 3.6.3:</u> Wind-driven rain (WDR)   **ranking score = 4**

Cradle SC/Tetra does not provide dedicated WDR model. However, user defined function feature from SC/Tetra can be used for this study.

<u>Criterion 3.6.4:</u> Humidity   **ranking score = 5**

Humidity transport can be modeled with a dedicated model inside SC/Tetra. This model can handle the following analyses: relative humidity, absolute humidity, dew condensation, humidity transfer in solid, vapor pressure under humidity analysis.

<u>Criterion 3.6.5:</u> Wind load towards to building structure   **ranking score = 1**

Cradle SC/Tetra does not offer wind-driven pressure distribution for building structure analysis.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
December 4, 2013   Page 152

<u>Criterion 3.6.6:</u> Pressure distribution along streamlines/pressure losses   **ranking score = 3**

Cradle SC/Tetra allows for setting up pressure loss of different obstruction or constrained geometries such as porous objects.

<u>Criterion 3.6.7:</u> Conjugate heat transfer   **ranking score = 5**

Cradle SC/Tetra provides heat transfer thermal analysis which allows for simultaneously simulating convective / turbulent fluid convection which can be seen in buoyance-driven ventilation.

3.6.9 Criteria Group Workflow – Support for Validation

<u>Criterion 3.7.1:</u> Validation document / example   **ranking score = 5**

Cradle SC/Tetra provides substantial documentation including a technical manual, a user's guide and validation exercises. These exercises were validated against more robust simulation such as Direct Numerical Solution (DNS) or experiments. These exercises also act as tutorials for users to learn how to set up their model properly.
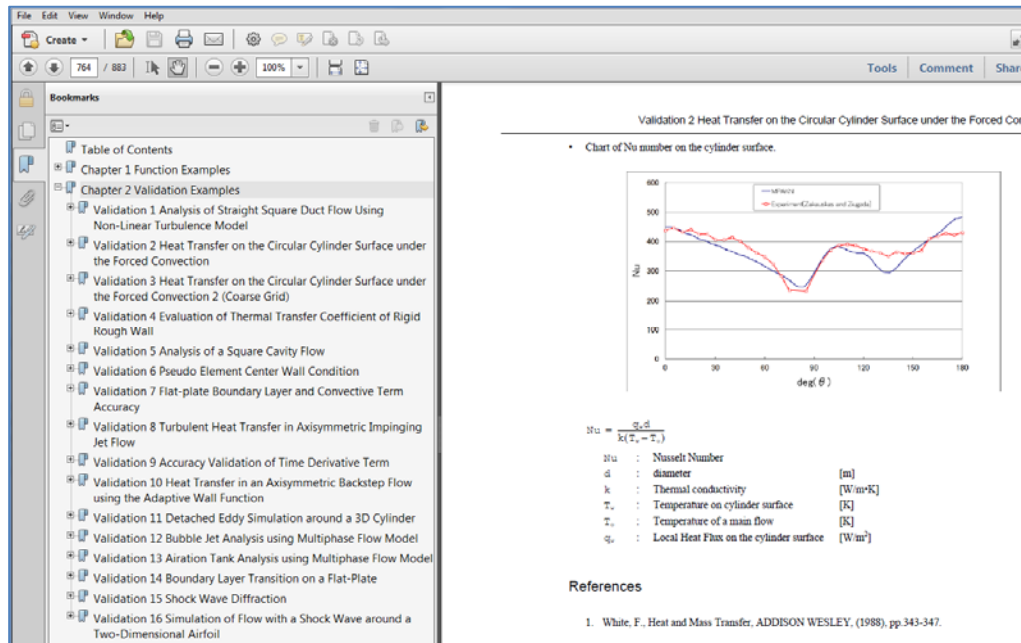
Figure 3.6.9:   16 Exercises of validation from SC/Tetra Version 10 User's Guide Exercise; Source: SC/Tetra Version 10 User's Guide Exercise (Cradle, 2012)

3.6.10 Criteria Group Cost of Software

<u>Criterion 4.1:</u>   Commercial pricing model   *ranking score = 3*

Need to be written after having the amount number.

<u>Criterion 4.2:</u>   Academic pricing   **ranking score = 4**

Cradle offers special discounts for academic users. For this study, the ERDL-HNEI CFD team received a substantial evaluation version of SC/Tetra packages for about two month at no cost academic license. However, there is no dedicated webpage on the website for academic license and support. Without direct contact to customer service personnel, user might have no idea on the actual cost for academic license.

<u>Criterion 4.3:</u>   Software includes all modules (no extra costs)   **ranking score = 3**

Cradle SC/Tetra package includes necessary modules for CFD modeling such as SSTpreprocessor, SCTsolver, SCTpost-processor, SCTjob for analysis job management, LFileView as viewer tool, SCTools for reprocessing calculation results data files, etc. However, the user might need to buy extra modules such as CADthru for preparing CAD data for CFD analysis.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 154

3.7 Ranking of Design Builder

This section provides a description of the capabilities of Design Builder software, and specifically the external CFD module that is part of the overall software package.

3.7.1 Criteria Group: Value for the CFD software product to the Architecture and Building Market

These ranking criteria are listed under <u>Number 1</u> in the ranking matrix:

Criterion 1.1:   Benefit for external CFD applications to architecture   **ranking score = 2**

Design builder offers an introductory level analysis capability for external CFD applications. Introductory can be understood as having a low sophistication of workflow and required understandings of CFD fundamentals. The grid is a structured and self-adjusting grid and the requirements to obtain results is very basic. The CFD results offer an idea about pressure and velocity distribution around the building.

What makes Design builder external CFD useful to the architecture practitioners or student is the readily obtainable visualization of pressure and velocity fields around the building. Since the main objective of Design Builder is energy simulations, building model geometries created for energy analysis purposes can also be used to describe a general wind pattern around the building.

The external CFD functionality of Design Builder is limited but represents an easy to learn and apply CFD application to assess wind movement around buildings.

 Criterion 1.2:   Previous applications in architecture / urban design   **ranking score = 1**

The use of Design Builder in external CFD applications appears to be quite limited. The project team could not identify any mentioning of use of Design Builder external CFD package in professional publications. It must be noted however that external CFD applications in architecture are only at the beginning of widespread use. The visualization of wind movement around buildings is often used in presentations. In these cases qualitative representation of wind patterns around buildings and structures can be given by the software.

Criterion 1.3:   Previous mentioning of software product in scientific or applied professional articles
ranking score = 1

The project team could not find any mention of Design Builder's external CFD software applications in the scientific (building science) and professional literature. This should not come as a surprise, since the external CFD module of Design Builder is mainly an easy-to-use building simulation tool for qualitative description of wind patterns around buildings.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 155

In the description of their external CFD package, Design Builder claims that *"DesignBuilder CFD has been independently tested against Pheonics, one of the most powerful and widely respected general purpose CFD packages available. The validation work, carried out by Northumbria University, shows that DesignBuilder is able to reproduce the same results as Pheonics given the same input data. This adds to confidence in the results produced by DesignBuilder CFD."*

Criterion 1.4:   Using graphics output for presentations   **ranking score = 3**

Design Builder has a ready to use graphical interface that presents visual images of the external velocity and pressure field around buildings. The software has a feature where high quality renderings of buildings can be overlaid by vector description of wind velocity and contoured slices for velocity and pressure. The handling of these graphics representation is very easy and with effort the user can fuse other presentations with the renderings produced by Design Builder.
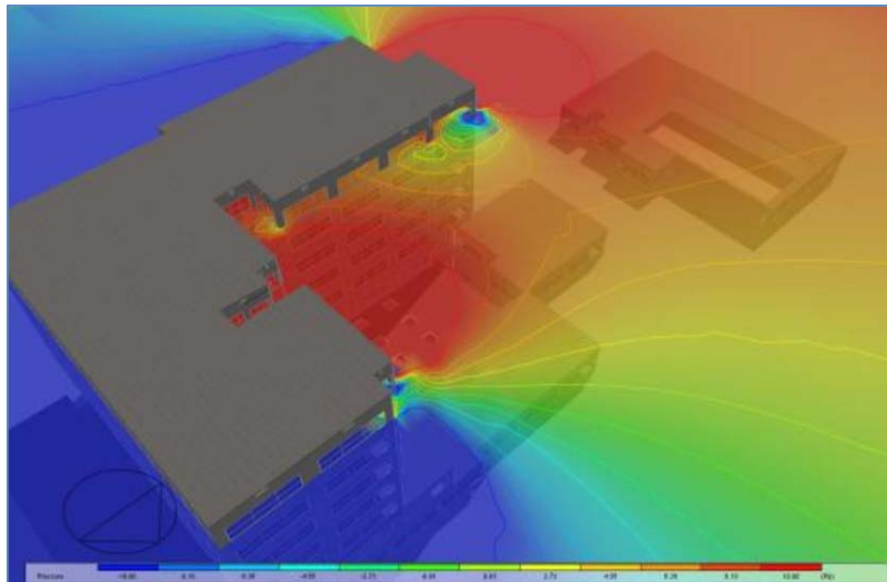
Figure 3.7.1:   Visualization of pressure distribution around buildings (Source Design Builder)

3.7.2 Criteria Group: Support by Vendor / Expert Network:

These ranking criteria are listed under Number 2 in the ranking matrix:

Criterion 2.1:   Technical support available: **ranking score = 3**

The technical support offers a chat section and also the possibility of submit a support ticket. Response time of the technical support is average. The quality of the overall service is above average. Direct

consultation is not available through the software company. The company provides a DesignBuilder Consultants Register where individual consultants (who are not part of Design Builder) can be contacted to provide support.
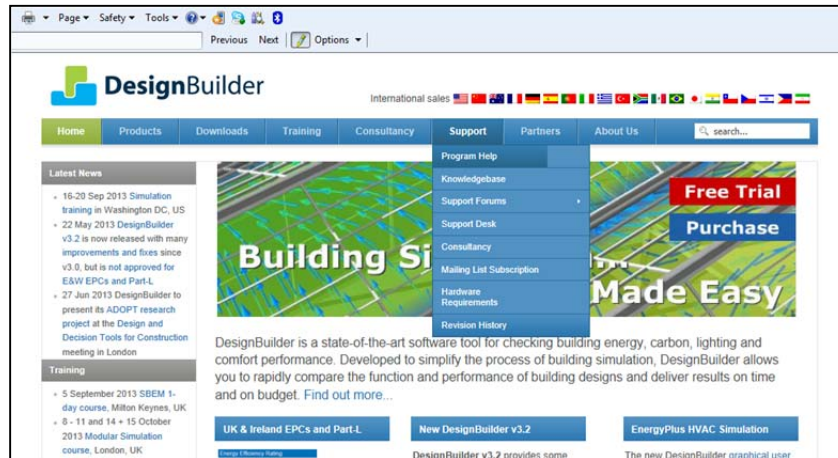
Figure 3.7.2A: Technical support option through the Design Builder website (Source Design Builder)

Criterion 2.2:   Online tutorials available on demand: **ranking score = 2**

The software company offers about 20 online tutorials (on-demand) that describes the entirety of the Design Builder functionality. The figure below shows a sample tutorial. In regard to CFD only 3 tutorials are offered and only one of which is related to external CFD. The students have to be registered to access the tutorials. The tutorials are archived webcasts and about 45 minutes long each and they show sample applications of external CFD applications. Due to the fact that the external CFD in Design Builder is very simplistic and easy to use, no in depth explanation is offered.

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 2**

On-line classes (synchronous training classes) are available but the company charges to take them. The company does offer some discount for academic clients but the entry barrier is therefore not insignificant.

Criterion 2.4:   Network of CFD professionals in architecture field: **ranking score = 3**

We could not identify a standing network of academics or professionals in architecture which uses Design Builder for external CFD applications. The Design Builder website offers an online forum where users address issues which are then answered by the technical support or are discussed by other users.
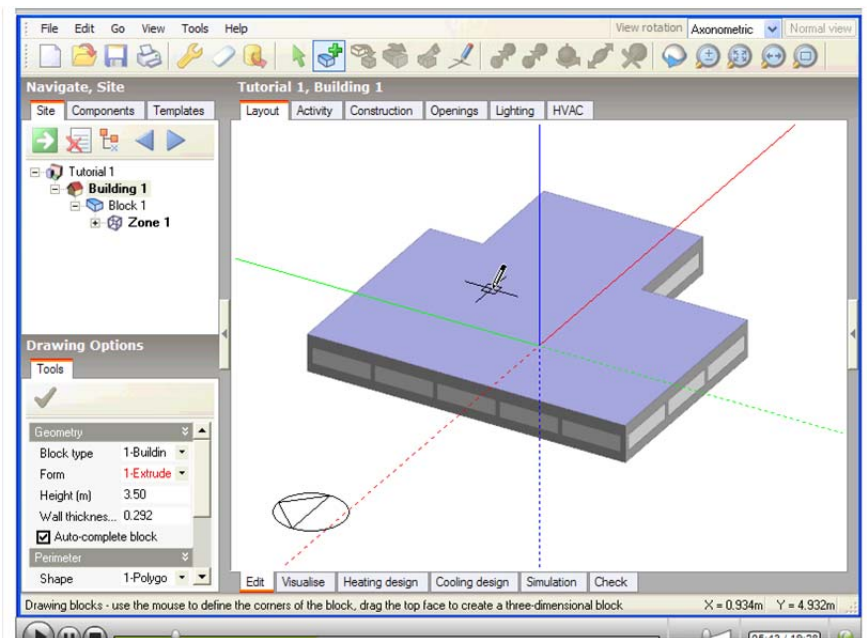
Figure 3.7.2B: Visualization of pressure distribution around buildings (Source Design Builder)

Figure 3.7.2C: Screen image of the forum for CFD (Source Design Builder)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                            School of Architecture, University of Hawaii

December 4, 2013   Page 158

<u>Criterion 2.5:</u>   Ease of working with academic support: *ranking score = 3*

The project team could not identify specific academic technical support. The administrative academic support is accommodating and provides expeditious service. The project team has first-hand experience in purchasing Design Builder academic licenses and the administrative person in charge of selling and supporting academic customers has been very helpful and supportive.

<u>Criterion 2.6:</u>   User guide provides efficient and comprehensive information: **ranking score = 5**

The user guide of the software is very effective and comprehensive. The user guide is object linked, which means that every command is directly linked to the specific issue in the user guide. The user guide contains all pertinent information to the level that is required to operate the external CFD module as well as some useful background information about the CFD process.

3.7.3 Criteria Group: Workflow-Ease of use / GUI

These ranking criteria are listed under <u>Number 3.1</u> in the ranking matrix:

<u>Criterion 3.1.1:</u> GUI is easy to use for trained operator   *ranking score = 4*

The GUI is very easy to use for the trained operator. In fact once the geometry is created the external CFD analysis only needs a handful of steps in order to start the simulation. The required GUI steps in starting a new external CFD analysis are limited to about <u>5 to 6 parameters</u> that have to be entered. If other than the default values are opted for the user requires significantly more knowledge and more steps in setting up the simulation. An analysis feature shows the aspect ratio of cells and suggests whether or not the CFD analysis will result in stable simulation.

The progress of the simulation is depicted in an interactive graph that shows development of residuals as the simulation time steps. After the simulation has converged the user can easily commence the post processing
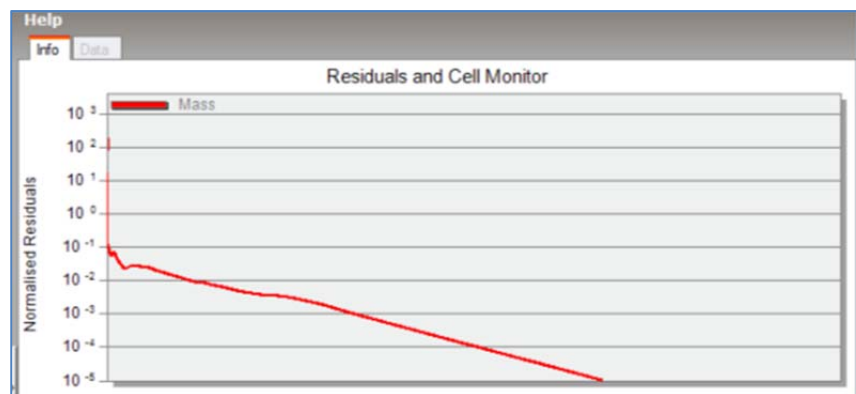
Figure 3.7.3:   Design Builder residual status screen (Source Design Builder)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
December 4, 2013   Page 159

<u>Criterion 3.1.2:</u> User guide and online help is available for each command   *ranking score = 5*

The user guide and online help are very effective. The help offers subject linked support at any phase of the work flow. This implies that at any stage in the work process the user can get support without leaving the simulation process.

3.7.4 Criteria Group: Workflow-Modeling / Import geometry

These ranking criteria are listed under <u>Number 3.2</u> in the ranking matrix:

<u>Criterion 3.2.1:</u> Create geometry inside CFD application   *ranking score = 5*

The geometry modeled in the Design Builder software offer a very effective way to create external geometries. Since external CFD requires some degree of simplification of external geometry the internal modeler is sufficient and in the same way effective. External walls can be easily created by automatic extrusions. Windows are created automatically. All standard roof forms can be created with entering a limited set of parameters. More complicated roof forms can be created with component blocks.

<u>Criterion 3.2.2:</u> Import geometry / repair surface before meshing   *ranking score = 1*

Importing geometry is possible but might involve several steps in the work process.

There is no option to repair the external surface before meshing. The mesh is a simple structured mesh that lacks sophistication and the combination of different cell forms. The mesh is rather surrounding the geometry and is not touching it, if the external surface is not perpendicular to the normal vector of the cubic cells.

<u>Criterion 3.2.3:</u>   Support Revit Architecture Import geometry / repair surface before meshing
                                                                                      ranking score = 2

DesignBuilder provides interoperability with BIM models through its gbXML import capability. This allows the user to import 3-D architectural models created in Revit, ArchiCAD or Microstation (and other 3-D drawing systems that support gbXML data exchange). Revit users can access DesignBuilder while working on their BIM models through the DesignBuilder Revit Plugin. This allows the model to be checked and analyzed without leaving Revit. An important feature of DesignBuilder gbXML import is the removal and repair of gaps between zone inner volumes, which means that models lose very little geometric information in the transfer from CAD to DesignBuilder. In fact, the imported and rendered

DesignBuilder model may look more realistic than it did in the CAD software! An example of a rendered image imported from ArchiCAD is shown below.

Figure 3.7.4:   Model imported as Revit and rendered in Design Builder (Source Design Builder)

Criterion 3.2.4:   Support CADD format and Inventor: *ranking score = 1*

CAD geometry created with Inventor cannot be readily imported into Design Builder

Criterion 3.2.5:   Support Rhino: *ranking score = 1*

CAD geometry created with Rhino cannot be readily imported into Design Builder.

3.7.5 Criteria Group: Workflow-Meshing

These ranking criteria are listed under Number 3.3 in the ranking matrix:

Criterion 3.3.1:   Surface mesher   *ranking score = 1*

Design Builder does not have a surface mesher function. The surface created in Design Builder cannot be automatically improved and there are no analysis and /or surface editing functions to repair the surface. Design Builder does not create a surface mesh. In Design Builder the solution is discretized in a finite volume grid. Moreover, Design Builder uses a non-uniform rectilinear Cartesian grid, which means that the grid lines are parallel with the major axes and the spacing between the grid lines enables non-uniformity. Due to the strict rectilinear nature of the grid, grid cells that lie in regions outside of the domain required for calculation are „blocked-off" in order to cater for irregular geometries.

Criterion 3.3.2:   Maximum cells and 2D-3D cell volume   *ranking score = 1*

DesignBuilder does not indicate the maximum number of cells that can be used in an external simulation. Experience has shown that with increasing number of cells in the grid, the performance, which means the time for running the simulation increases drastically.

Criterion 3.3.3:   Boundary layer types   *ranking score = 1*

The specification of boundary conditions for external analyses is relatively straightforward and just requires setting the building orientation, wind velocity and wind direction and this is done using a single dialogue box. The simplicity of setting the external boundary conditions has the advantage of being very user friendly. However, lacking more sophisticated means to describe inlet, outlet, lateral, ground and top conditions boundary diminishes the ability of the software to precisely predict external CFD wind phenomena.

Criterion 3.3.4:   Structured and Unstructured meshing   *ranking score = 1*

DesignBuilder only offers the option of a structured mesh. Consequently the grid gives rise to limitation in complexity.

Criterion 3.3.5:   Adaptive meshing capability   *ranking score = 1*

DesignBuilder offers a limited capability to edit the grid, which has to be initiated by the user. A default CFD grid is generated using the default grid spacing defined as part of the data set. The default grid may be edited using the 'Edit CFD Grid' tool which allows the user to change the spacing used for default regions, insert additional regions or remove previously inserted regions.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                             School of Architecture, University of Hawaii

December 4, 2013   Page 162
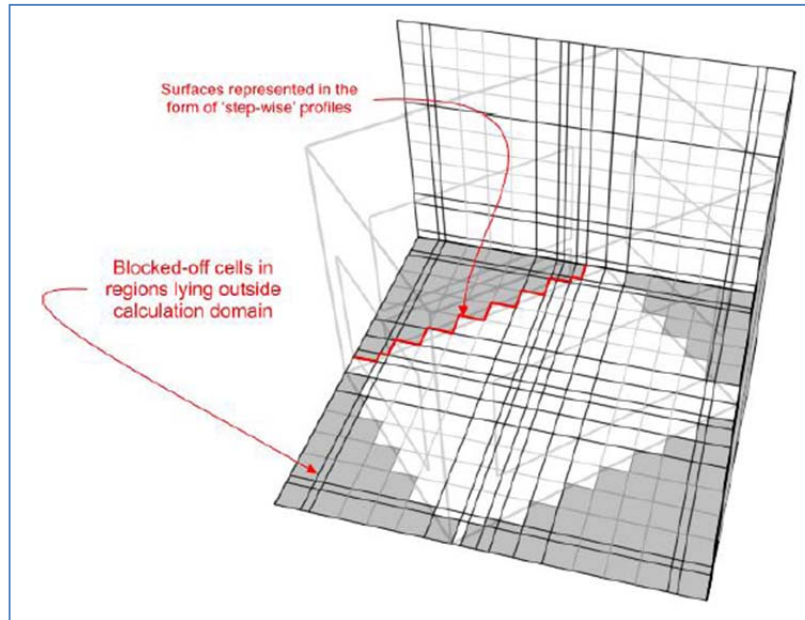
Figure 3.7.5A: Typical grid representation adjacent, non-axis parallel to surfaces (source Design Builder)
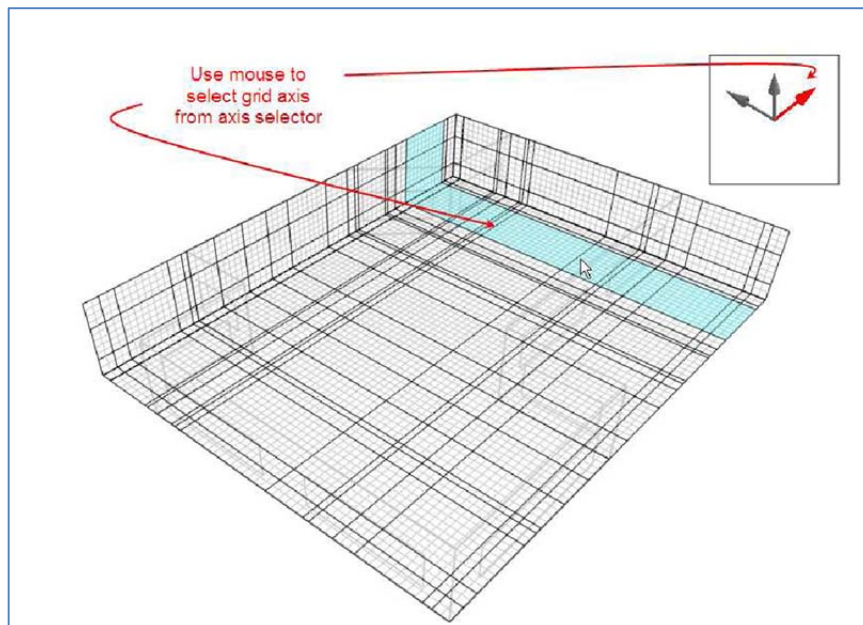
Figure 3.7.5A: Grid editing capabilities of Design Builder (source Design Builder)

3.7.6 Criteria Group: Workflow-Physics Models

These ranking criteria are listed under Number 3.4 in the ranking matrix:

Criterion 3.4.1:   Solvers - Coupled / decoupled   *ranking score = 1*

DesignBuilder cannot support coupled numeric domains. Furthermore DesignBuilder cannot display pressure coefficients on the surface of structures, since no surface mesh is created. Thus it is not easy to identify pressure coefficients at exactly the precise locations of the envelope. In essence Design Builder cannot easily determine decoupled domains since boundary conditions for internal CFD with DesignBuilder does not use pressure or stagnation point for openings in the envelope.

Criterion 3.4.2:   Solver-Finite Element Method (FEV) / Finite Volume Method (FVM) *ranking score = 3*

DesignBuilder uses a Finite Volume Method to solve the equation.

Criterion 3.4.3:   Steady State/Transient State   *ranking score = 2*

DesignBuilder uses only steady –state simulations.

Criterion 3.4.4:   Turbulence models   *ranking score = 1*

DesignBuilder only uses the k-e turbulence model. No other turbulence models are offered. Furthermore, the software does not allow altering the values of constant in the k-e turbulence model.

Criterion 3.4.5:   Materials   *ranking score = 1*

DesignBuilder does not offer setting of different materials, for example to define roughness conditions.

Criterion 3.4.6:   Customizable / special physics models   *ranking score = 1*

DesignBuilder does not offer customization of special physics models

3.7.7 Criteria Group: Post processing

These ranking criteria are listed under Number 3.5 in the ranking matrix:

Criterion 3.5.1: Visualization   *ranking score = 3*

The main mechanism for displaying results is the slice tool that allows the user to select a slice, along one of the main grid axes and perpendicular to it, within which can be displayed any of the selected results display plots. Velocity vectors are displayed as arrows, the length of which corresponds to the magnitude of the velocity and with applying a default vector scale factor. The maximum vector length is

the maximum length of a vector that will be displayed to prevent the display from becoming cluttered with excessively large vectors. 3-D contours allow the user to select any of the available variables for which a 3-D contour plot is required:
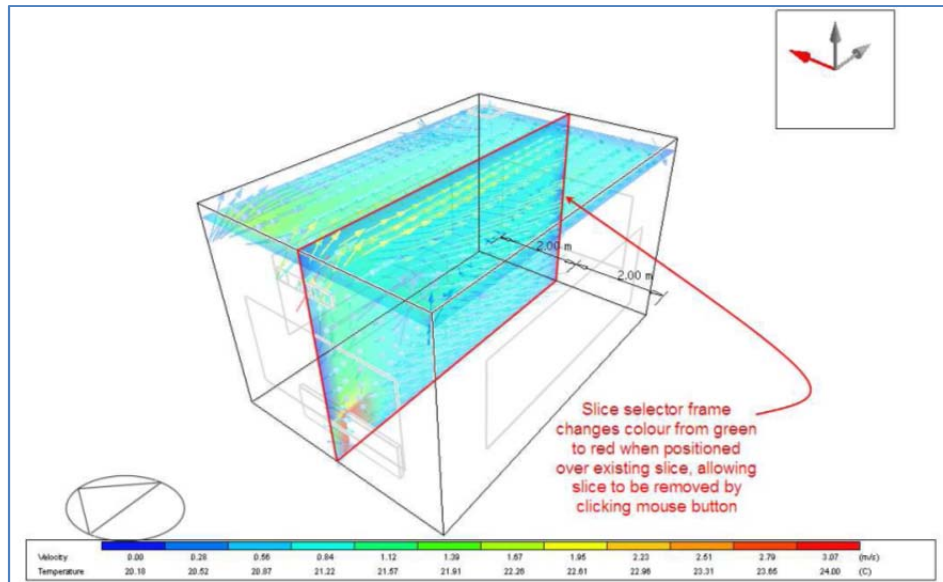
Figure 3.7.7A: Visualization using slice parallel to the three major coordinates (source Design Builder)
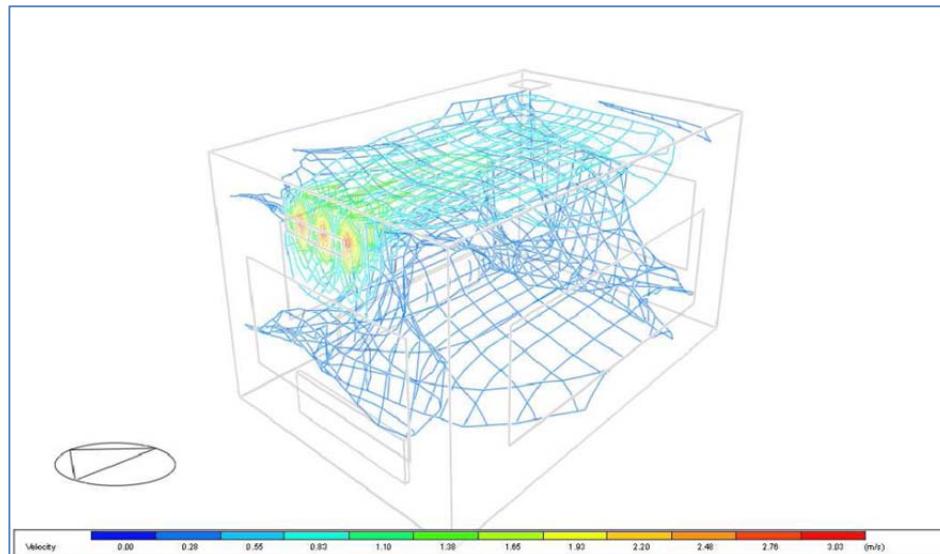
Figure 3.7.7B: Visualization using 3D-countours (source Design Builder)

Design Builder has a special application to visualize occupant comfort settings. Displayed values are:

- PMV
- PPD
- Comfort Indices
- Mean Radiant Temperature
- Operative Temperature

Design Builder has the ability to produce shaded renderings. The feature is very versatile and can also be used for the visualization of external CFD simulation results.

Criterion 3.5.2:   Quantitative analysis capabilities of components   *ranking score = 1*

DesignBuilder does not offer a versatile and easy to use tool to determine the pressure and velocity at specific locations in the numeric domain. There are indirect methods to identify pressure, velocity and other parameters at certain locations on slices. But these methods are cumbersome, not exact and difficult to use.

Criterion 3.5.3:   Comparative analysis (multi-scenarios)   *ranking score = 1*

DesignBuilder does not offer this function

3.7.8 Criteria Group: Workflow-Customization / special functions

These ranking criteria are listed under Number 3.6 in the ranking matrix:

Criterion 3.6.1:   Comparative analysis (multi-scenarios)   *ranking score = 2*

Design Builder has a limited ability to define pedestrian comfort by using qualitative representation of result of external CFD. The external flow field can be determined by placing appropriate slices.

Criterion 3.6.2:   Natural ventilation   *ranking score = 3*

Design Builder has the ability to provide quantitative performance criteria for wind induced natural ventilation from using the result of external CFD. However, DB does not utilize result obtained from the external CFD calculations in order to assess internal air movement inside the spaces. The natural ventilation calculations of internal spaces are based on boundary conditions that are calculated from Energy Plus simulations for multi-zonal model; the boundary conditions are then applied for the various openings in the building envelope.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                      School of Architecture, University of Hawaii

December 4, 2013   Page 166

<u>Criterion 3.6.3:</u>   Wind-driven rain   *ranking score = 1*

Design Builder does not have the ability to provide an assessment of wind driven rain.

<u>Criterion 3.6.4:</u>   Humidity   *ranking score = 1*

Design Builder does not have the ability to provide an assessment of the distribution of changes in humidity.

<u>Criterion 3.6.5:</u>   Wind load towards to building structure   *ranking score = 2*

Design Builder airflow and external pressure calculations can be used to determine the wind loading on buildings. However, Design Builder does not offer a ready tool or functionality to easily determine the magnitude of pressure coefficients on the building envelope. There is more elaborate process of extracting pressure coefficients for the specific Cartesian grid cells (for the applied slicing) through export of data to excel sheets. However, this process is time consuming and potentially does not use the correct cell coordinates.

<u>Criterion 3.6.6:</u>   Pressure distribution along streamlines/pressure losses   *ranking score = 1*

Design Builder does not have a ready function to provide an assessment of the pressure distribution along streamlines or determine pressure losses   along a certain streamline.

<u>Criterion 3.6.7:</u>   Conjugate heat transfer   *ranking score = 1*

Design Builder external CFD calculations are iso-thermal and therefore do not include thermal information.

3.7.9 Criteria Group: Workflow-Support for Validation

These ranking criteria are listed under <u>Number 3.7</u> in the ranking matrix:

<u>Criterion 3.7.1:</u>   Validation document / example   *ranking score = 2*

Design Builder has indicated that there are limited case studies that address validation of external CFD results.

3.7.10 Criteria Group: Cost of software

These ranking criteria are listed under <u>Number 4</u> in the ranking matrix:

Criterion 3.7.1:   Commercial pricing model   *ranking score = 3*

Design Builder offer the CFD package as a module of its basic simulation software or as a standalone licenses. In a network as many users can run the software as there are seats attached. The CFD functions are competitively priced, which include both external and internal CFD applications. The software packages are either perpetual licenses or can be obtained on an annual level.

Criterion 3.7.2:   Academic pricing   *ranking score = 5*

Design Builder offers a very competitive and (academic) customer friendly pricing for academic licenses. Design Builder offered outstanding service to the UHM School of Architecture when the school purchased an academic package. The project team had a very supportive service contact with the company, which highly regards academic outreach.

Criterion 3.7.3:   Software includes all modules (no extra costs)   *ranking score = 4*

For the academic license the software package came with all pertinent modules. As the project team understands, the commercial license requires the purchase for different modules.

Criterion 3.7.4:   Option to acquire sufficient seats at no extra costs   *ranking score = 5*

Design Builder offers a very attractive price to obtain licenses for the use by students. In effect Design Builder did not charge for 50 student seats (all full functionality packages). The student licenses are obtain as perpetual licenses but have to be renewed every year, which involves some administration actions but no extra costs. This feature of Design Builder is very attractive to use Design Builder as a simulation tool and for class room instructions.

3.8   Ranking of OpenFOAM (Open Field Operation and Manipulation):

3.8.1 Criteria Group: Value of CFD Software Product for Architecture

Criterion 1.1:   Benefit for external CFD   **ranking score = 5**

OpenFOAM offers many features capable to analyze complex fluid flows involving chemical reactions, turbulence, heat transfer, etc. With full range of turbulence models as well as flexible in meshing of complex geometries, capability to customize settings of atmosphere boundary layer, OpenFOAM receives the high ranking for an external CFD.

Criterion 1.2:   Previous application in architecture / urban design   **ranking score = 1**

OpenFOAM is the open-source CFD receiving high reputation of the open-source CFD user community. OpenFOAM has been used across most areas of engineering and science, from both commercial and academic organizations. However, its application in architecture / urban design is still limited to the best knowledge of the authors at the time this report written.

Criterion 1.3:   Previous mentioning of software product in scientific or applied professional articles
ranking score = 3

OpenFOAM has been found in many scientific and applied professional articles. However, its frequency is still lower in comparison to other commercial CFD software products.

Criterion 1.4:   Using graphics output for presentation   **ranking score = 3**

OpenFOAM uses open-source visualization software called ParaView for delivering a variety of graphic output for fluid presentation. However, the quality of these graphics is at average and ParaView is not dedicated CFD visualization software and is lacking intuitive tools.

3.8.2   Criteria Group Support by Vendor / Expert Network

Criterion 2.1:   Technical support available: **ranking score = 3**

Based on the information available on its website, even as free license cost open-source, OpenFOAM also provides a variety of commercial support including support for basic operation of OpenFOAM, assistance with CFD simulation, code customization and development. However, these supports are quite limited in comparison to other commercial CFD vendors. The response time for technical support has not been verified in this report.
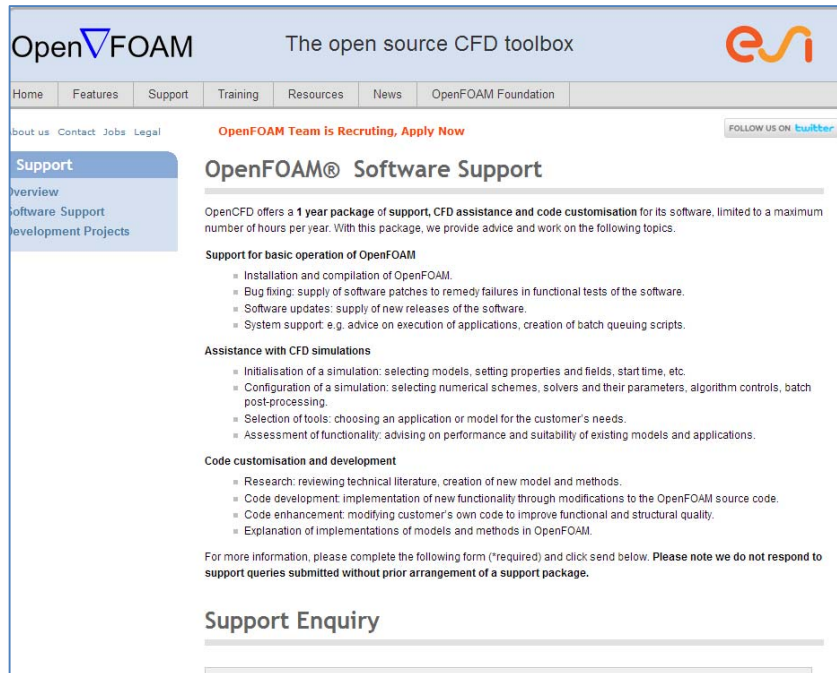
Figure 3.8.2A: Technical support options through the OpenFOAM website

Criterion 2.2:   Online tutorials available on demand   **ranking score = 2**

OpenFOAM only provides very brief tutorials in its manual documentation downloadable at its website. Other tutorial resources are also very limited online.

Criterion 2.3:   Online classes available (low entry barriers):   **ranking score = 2**

OpenFOAM offers training classes in person at its facilities worldwide. Moreover, these training courses require a fee ($1600 each). OpenFOAM also offers on-site training for companies or organizations requiring training for larger groups of people.

Criterion 2.4:.   Network of CFD professionals in architecture field: **ranking score = 1**

OpenFOAM has a lot of supports from user community sharing tutorials, updated information, trouble shooting and collaboration. However, user community network for CFD professionals in architecture field has not found at this time of this report written.
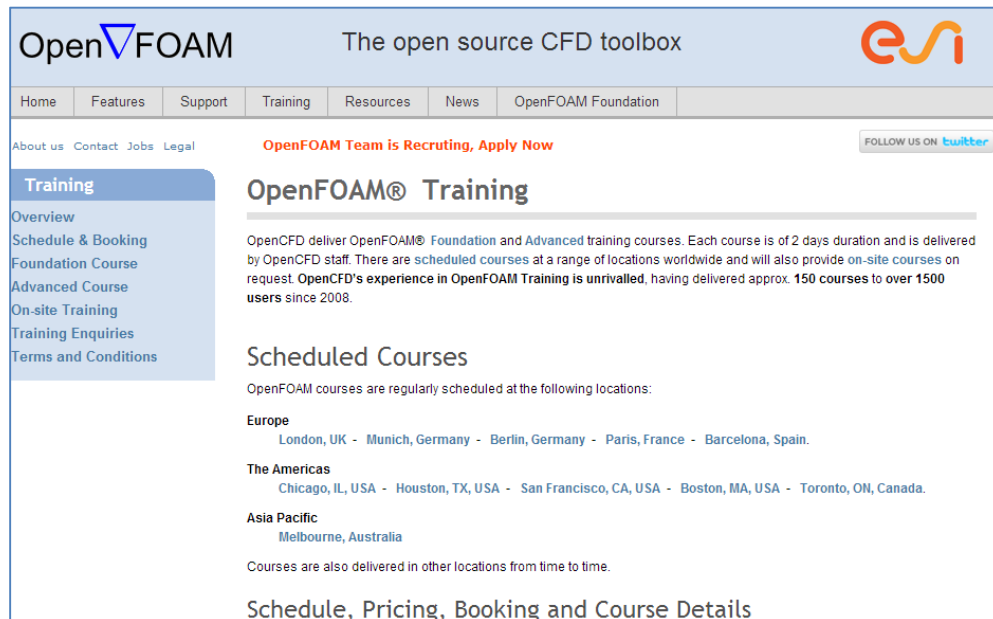
Figure 3.8.2B: Training options through the OpenFOAM website
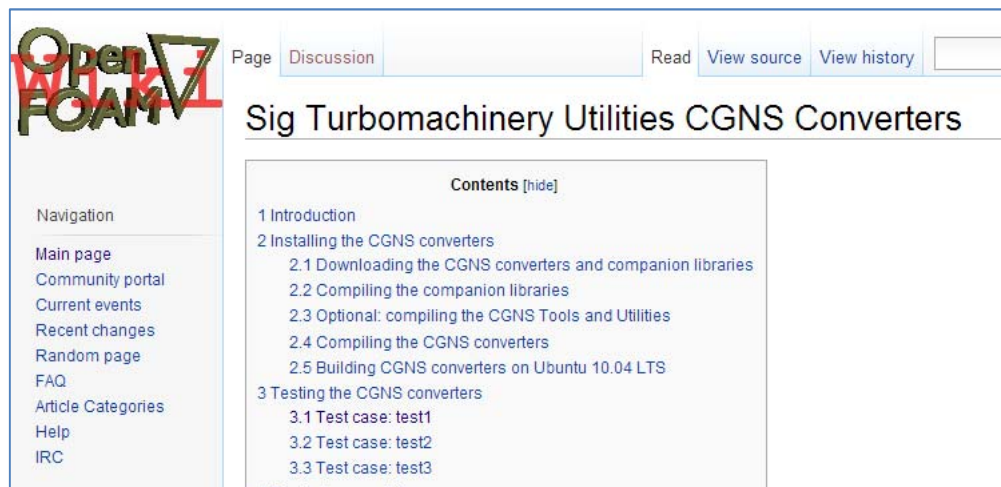
Figure 3.8.2C: OpenFoam wiki

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                       School of Architecture, University of Hawaii

December 4, 2013   Page 171
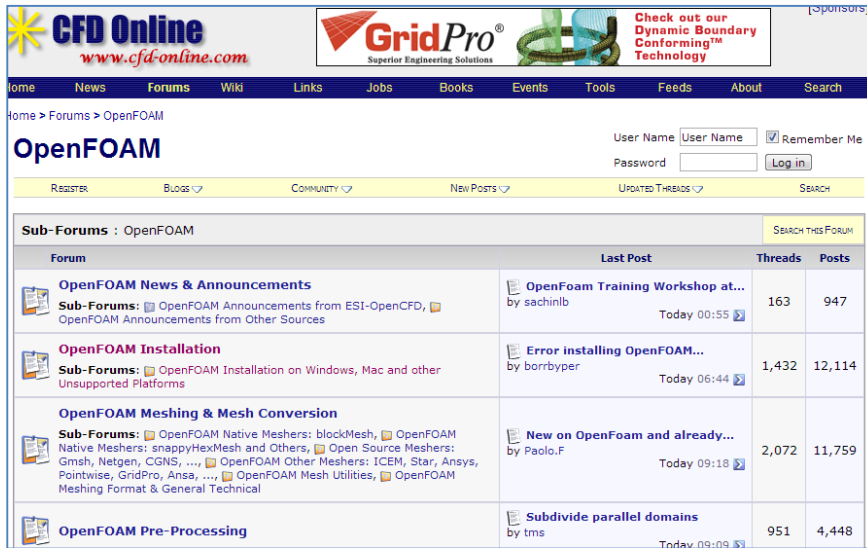
Figure 3.8.2D: OpenFoam user forum at www.cfd-online.com

Criterion 2.5:   Ease of working with academic support: **ranking score = 2**

Based on its website, OpenFOAM offers academic support services. However, the detail as well as quality of these services have not been investigated at this time of this report.

Criterion 2.6:   User guide provides efficient and comprehensive information: **ranking score = 2**

OpenFOAM provides a User guide. However, this documentation has insufficient details, making the learning curve very gradual. Moreover, the lack of maintained documentation makes it difficult for the new users.

3.8.3 Criteria Group Workflow - Ease of Use / GUI

Criterion 3.1.1: GUI is easy to use for trained operator   **ranking score = 1**

As open-source, OpenFOAM is absent of an integrated GUI which make it difficult for most users who are not familiar to command line based system link UNIX and Linux.

Criterion 3.1.2: User guide and online help is available for each command   **ranking score = 1**

OpenFOAM provides very limited user guide or off-line PFD copy of the user guide and online help at each command is not available at this time of this report written.
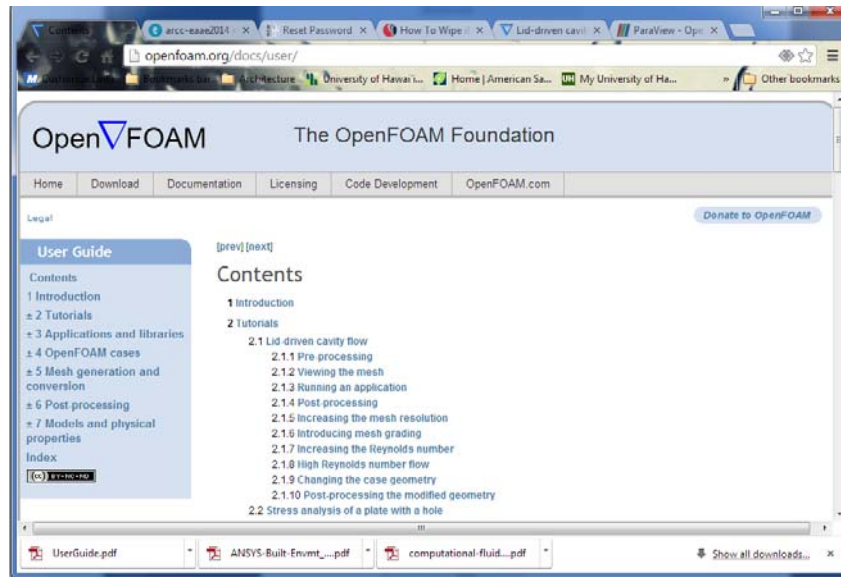
Figure 3.8.3:   Online user guide available at OpenFOAM website

3.8.4 Criteria Group Workflow – Modeling / Import Geometry

Criterion 3.2.1: Create geometry inside CFD application   **ranking score = 2**

OpenFOAM provides meshing functionalities to create geometry inside the CFD application. The software includes blockMesh, snappyHexMesh, exrudeMesh and polyDualMesh to handle different complexity levels of geometry. However, the lack of GUI for the software makes it difficult to use and requires a lot of effort and time.

Criterion 3.2.2: Import geometry / repair surface before meshing   **ranking score = 2**

OpenFOAM accepts meshes generated by virtually any type of major mesh generators and CAD systems such as ANSYS, FDX, Fluent,Gmsh, STAR-CD, etc. However, as a non-GUI system, OpenFOAM's does not offer a mesh repairing tools.
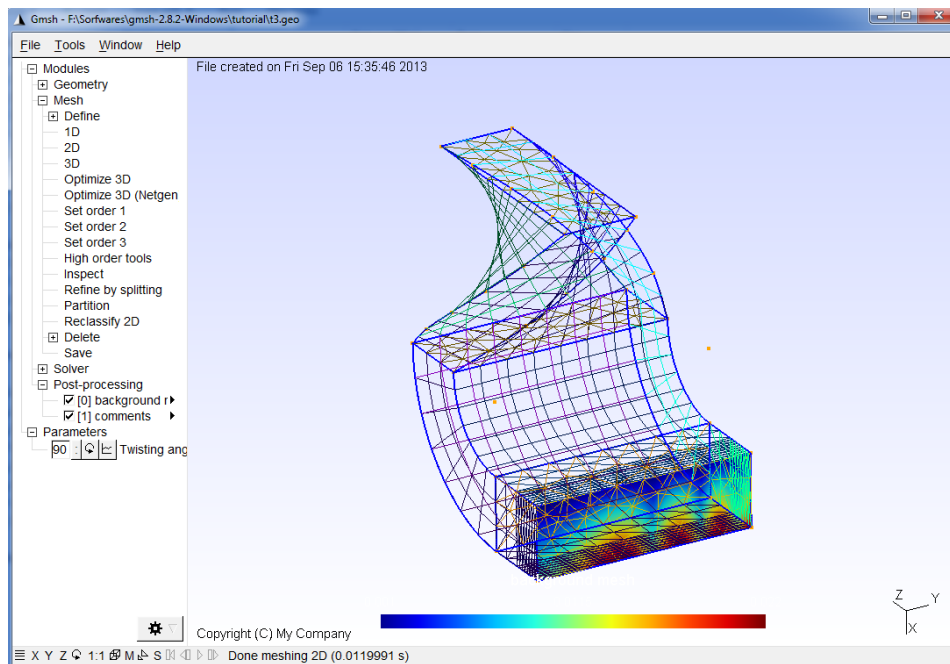
Figure 3.8.4:   Open-source GMSH geometry and mesh generators for meshes which can be imported into OpenFOAM

Criterion 3.2.3: Support Revit Architecture   **ranking score = 1**

OpenFOAM does not directly support geometry imports from REVIT.

Criterion 3.2.4: Support CADD format and Inventor   **ranking score = 1**

OpenFOAM does not support CADD format as well as geometry files created with Inventor.

Criterion 3.2.5: Support Rhino   **ranking score = 1**

OpenFOAM does not directly support Rhino 3D. However, the import of geometry created by Rhino is possible through conversion into STL format and then from STL to GMSH format, which can be finally converted into OpenFOAM. However, this complicated conversion process and the quality from the transition make this ranking is the lowest.

3.8.5 Criteria Group Workflow - Meshing

Criterion 3.3.1: Surface meshers   **ranking score = 1**

OpenFOAM does not support advanced surfaces mesher options such as the Surface wrapper, Surface remeshers, and automated surface repairing tools. At the present there is no support for embedded interfaces and refinement trees. Meshes can be imported from several third-party mesh generators.

Criterion 3.3.2: Maximum cells and 2D-3D cell volume   *ranking score = 5*

The CFD team is not aware of any limitation to the maximum allowable cells for OpenFOAM application. However, its meshing utility called snappyHexMesh can run in parallel to generate meshes of 100s of millions of cells, given sufficient computational resources.

Criterion 3.3.3: Boundary types   *ranking score = 5*

OpenFOAM offers a variety of boundary types classified into the following main categories; constraint boundary conditions, thermo-physical boundary conditions, inlet boundary conditions, coupled boundary conditions, generic boundary conditions, outlet boundary conditions and wall boundary conditions. These boundary conditions are sufficient to conduct the majority of fluid phenomena investigation with OpenFoam.
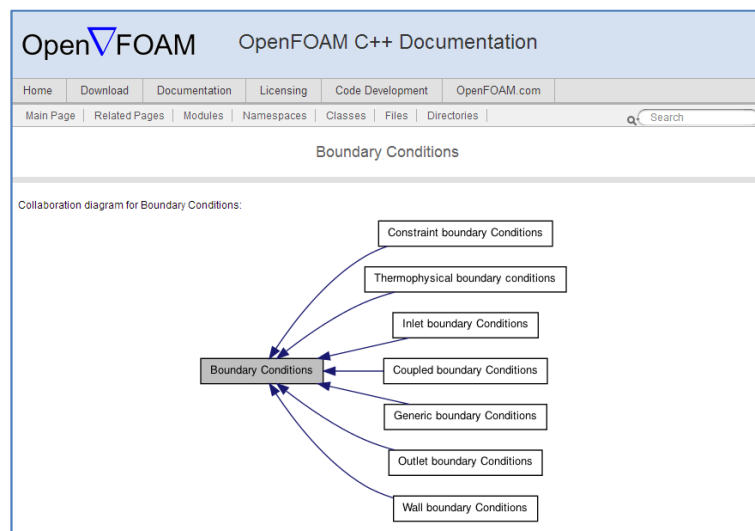
Figure 3.8.5: Main categories of boundary conditions covered by OpenFOAM

Criterion 3.3.4: Structured and Unstructured meshing   **ranking score = 5**

OpenFOAM supports unstructured meshes combined with a freedom of cell shape. This offers a great flexibility for the mesh generation. OpenFOAM provides several meshing tools such as blockMesh (hexahedra), snappy HexMesh (hexahedra), extrudeMesh (prism) and PolyDualMesh (tetrahedral and polyhedral meshing). OpenFOAM has several utilities that perform mesh checking and manipulation.

Criterion 3.3.5: Adaptive meshing capability   *ranking score = 1*

OpenFOAM has not provided dynamic refinement or adaptive meshing capability.

3.8.6 Criteria Group Workflow – Physics Models

Criterion 3.4.1: Solvers – Coupled / decoupled   **ranking score = 5**

OpenFOAM has the full range of customizable boundary conditions and flexible meshing tools to model "coupled" computational domain. The sizes of coupled domains differ from very small to very large scale. The basic advantage is that internal and external airflow can be solved simultaneously.

Criterion 3.4.2: Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)   **ranking score = 5**

OpenFOAM uses Finite Volume Method (FVM) for the numerical solution. This FVM method is appropriate to conduct flux calculation in numerical equation solving as well as to formulate arbitrary shaped cells (any number of faces and any number of edges) of unstructured meshes.

Criterion 3.4.3: Steady state/Transient state   **ranking score = 5**

OpenFOAM offers a full range of steady state and transient state solutions which allow for simulating a wide variety of airflow phenomena. These solvers for incompressible flow are briefly listed as follows: steady-state solver for incompressible turbulent flow of non-Newtonian fluids, incompressible LES solver for flow in a channel, transient solver for incompressible flow of Newtonian fluids on a moving mesh using the PIMPLE (merged PISO-SIMPLE) algorithm, large time-step transient solver for incompressible flow using the PIMPLE (merged PISO-SIMPLE) algorithm.

Criterion 3.4.4: Turbulence models   **ranking score = 5**

OpenFOAM offers a wide range of turbulence models including Reynolds-Averaged Navier-Stokes (RANS), Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and even Direct Numerical Simulation (DNS).

Criterion 3.4.5: Materials   **ranking score = 3**

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 176

OpenFOAM offers a variety of different thermo-physical models. The use of thermo-physical properties requires customization with specific models.

Criterion 3.4.6: Customizable / special physics models   **ranking score = 4**

OpenFOAM allow the use of customizable physics models including combustion (chemical reaction, fire, compressible premixed/partially-premixed combustion with turbulence modeling), heat transfer and buoyance-driven flows, etc.. Humidity absorption / dispersion, however, is not being offered in OpenFOAM yet.

3.8.7 Criteria Group Workflow – Post-processing

Criterion 3.5.1: Visualization   **ranking score = 3**

OpenFOAM uses ParaView, a reader module for OpenFOAM data from the open source visualization application. The graphic output from ParaView provides a wide range of different options from shaded and transparent surfaces, vectors, contours, particle traces and even animation. However, the transfer solution data from OpenFOAM to ParaView is not a seamless process and therefore requires time and effort.

Criterion 3.5.2: Quantitative analysis capabilities of components   **ranking score = 4**

OpenFOAM provides a wide range of tools for producing quantitative analysis. The range of available tools is extensive: e.g. files of sampled data can be written for graph plotting or other third party post-processor or analysis.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 177
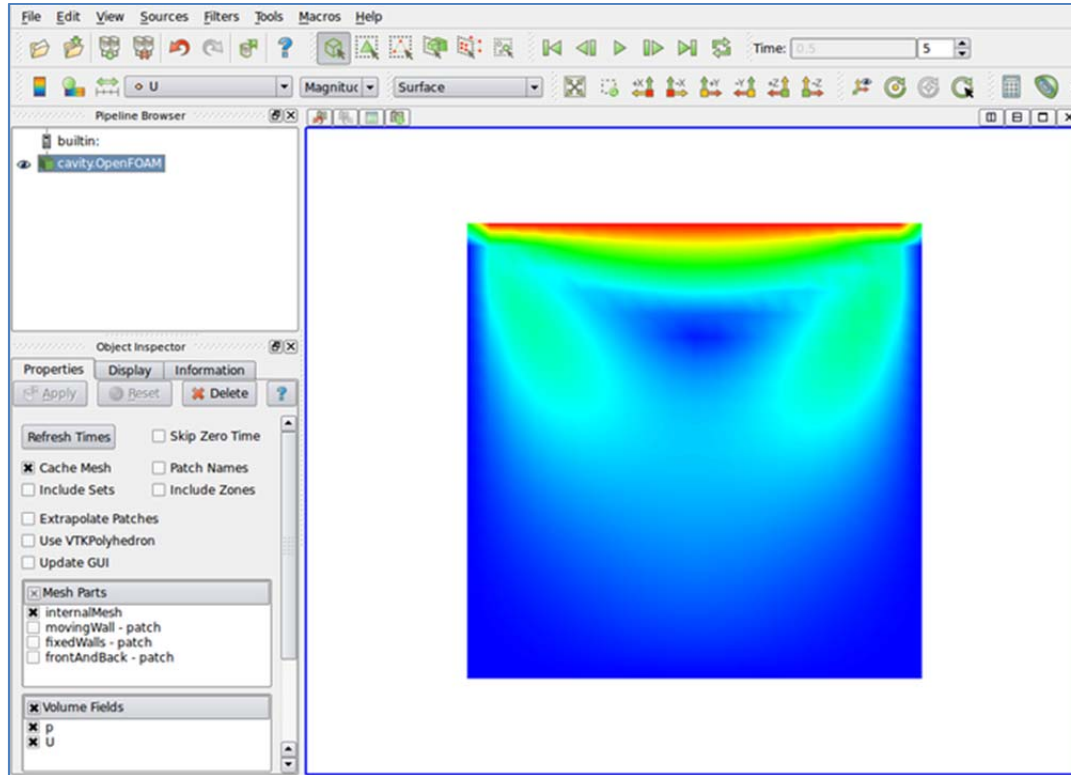
Figure 3.8.7:   ParaView - OpenFOAM's post-processor

Criterion 3.5.3: Comparative analysis (multi-scenarios)   **ranking score = 1**

OpenFOAM currently does not offer capabilities for comparative analysis.

3.8.8 Criteria Group Workflow – Customizable / Special Functions

Criterion 3.6.1: Pedestrian comfort   **ranking score = 5**

OpenFOAM can be used as simulation tool in modeling wind velocity related to pedestrian comfort.

Criterion 3.6.2: Natural Ventilation   **ranking score = 3**

OpenFOAM does not provide any specific user guide or resources regarding natural ventilation analysis.
However, the software capabilities can be used to assess highly complex external and internal airflow
for natural ventilation with significant effort and time. There is no special functions for mean age of air
and thermal comfort available from current versions of OpenFOAM's modules.

Criterion 3.6.3: Wind-driven rain (WDR)   **ranking score = 3**

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                       School of Architecture, University of Hawaii

December 4, 2013   Page 178

OpenFOAM does offer features which can be used to solve wind-driven rain phenomena.

Criterion 3.6.4: Humidity   **ranking score = 1**

Humidity analysis is not available as a dedicated function in OpenFOAM .

Criterion 3.6.5: Wind load towards to building structure   **ranking score = 5**

OpenFOAM offers modules for stress analysis of solids coupling with turbulence models to compute building surface pressure distribution used for wind loading analysis.

Criterion 3.6.6: Pressure distribution along streamlines/pressure losses   **ranking score = 3**

OpenFOAM offers post-processors capabilities for visualizing pressure losses along streamline.

Criterion 3.6.7: Conjugate heat transfer   **ranking score = 5**

OpenFOAM has a module that can analyze conjugate heat transfer between a solid and fluid regions. This module called chtMultiRegionFoam combines of heatConductionFoam and buoyantFoam.

3.8.9 Criteria Group Workflow – Support for Validation

Criterion 3.7.1: Validation document / example   **ranking score = 2**

As an open source, OpenFOAM has benchmarked against other commercial CFD software products and can been validated in on-site full-scaled measurement and small-scaled model wind tunnel measurements. However, this validation documentation is not available at a single source and therefore cannot be easily accessed.

3.8.10 Criteria Group Workflow – Cost of Software

Criterion 4.1:   Commercial pricing model   **ranking score = 5**

OpenFOAM is at no cost.

Criterion 4.2:   Academic pricing   **ranking score = 5**

OpenFOAM is at no cost.

Criterion 4.3:   Software includes all modules (no extra costs)   **ranking score = 5**

OpenFOAM is at no cost for any of its modules.

Criterion 4.4:   Option to acquire additional seats at no extra or small costs   **ranking score = 5**

OpenFOAM is at no cost.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                      School of Architecture, University of Hawaii

December 4, 2013   Page 180

3.9 Results of Initial Ranking for Seven Candidate CFD Software based on Vendor Documentation

This section presents the initial results of the ranking of the seven CFD software products which were evaluated solely based on CFD software product documentation.

The summary of the ranking results is provided in Table 3.9.1. and Figures 3.9.1 through 3.9.4.

The detailed analysis of the ranking for the seven CFD software products is shown on Table 3.9.2 (on the fold-out page following this page). given below.

Table 3.9.1 Summary of initial results of ranking based on CFD software vendor documentation

	CFD Software	total rank excl. costs	Criteria group 1	Criteria group 2	Criteria group 3
1	ANSYS - Fluent	65.1%	91%	58%	76%
2	STAR-CCM+	65.3%	63%	95%	77%
3	ACAD CFD	57.4%	46%	59%	79%
4	Cradle SD-Stream	61.8%	82%	71%	70%
5	Cradle SD-Tetra	60.5%	54%	67%	79%
6	Design Builder	24.3%	21%	47%	26%
7	Open-Foam	35.8%	43%	31%	45%

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                              School of Architecture, University of Hawaii

December 4, 2013   Page 181

Total ranking (excluding costs)

Software	Value
ANSYS - Fluent	~65%
STAR-CCM+	~65%
ACAD CFD	~57%
Cradle SD-Stream	~61%
Cradle SD-Tetra	~60%
Design Builder	~24%
Open-Foam	~35%

Figure 3.9.A: Results of initial ranking – Total ranking (excluding costs)

Criteria Group 1 -
Value for the CFD software product to the architecture and building market

Software	Value
ANSYS - Fluent	~91%
STAR-CCM+	~62%
ACAD CFD	~46%
Cradle SD-Stream	~83%
Cradle SD-Tetra	~55%
Design Builder	~22%
Open-Foam	~43%
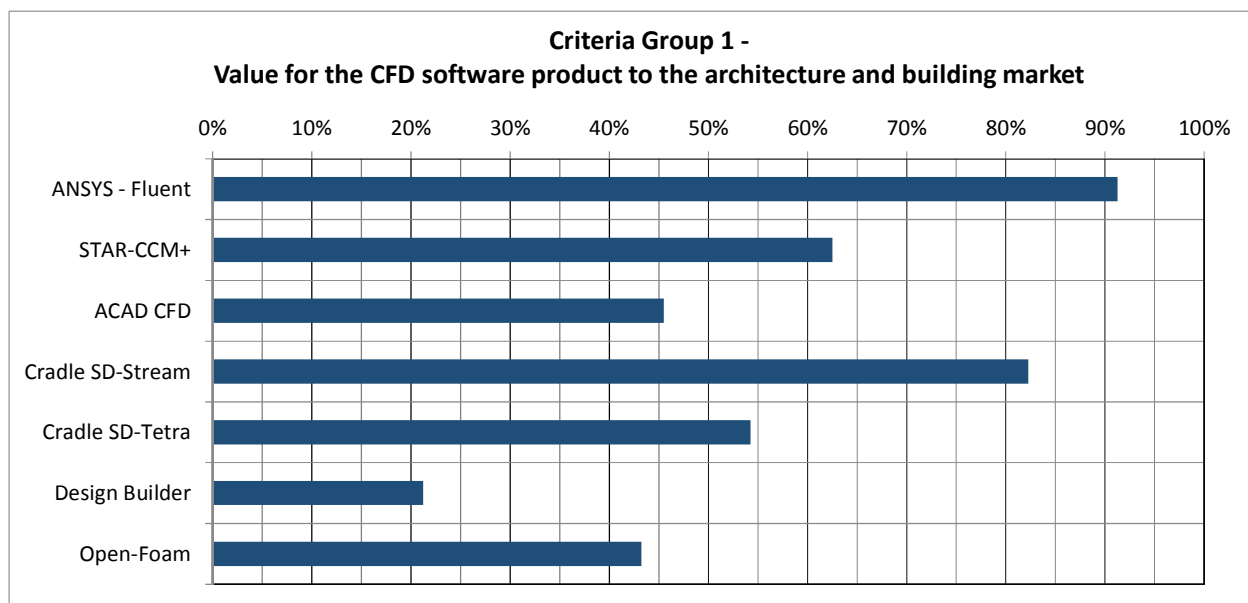
Figure 3.9.B: Results of initial ranking – Ranking for criteria group 1 - Value for the CFD software product to the architecture and building market

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                               School of Architecture, University of Hawaii

December 4, 2013   Page 182

Criteria Group 2 - Support by vendor/Expert Network
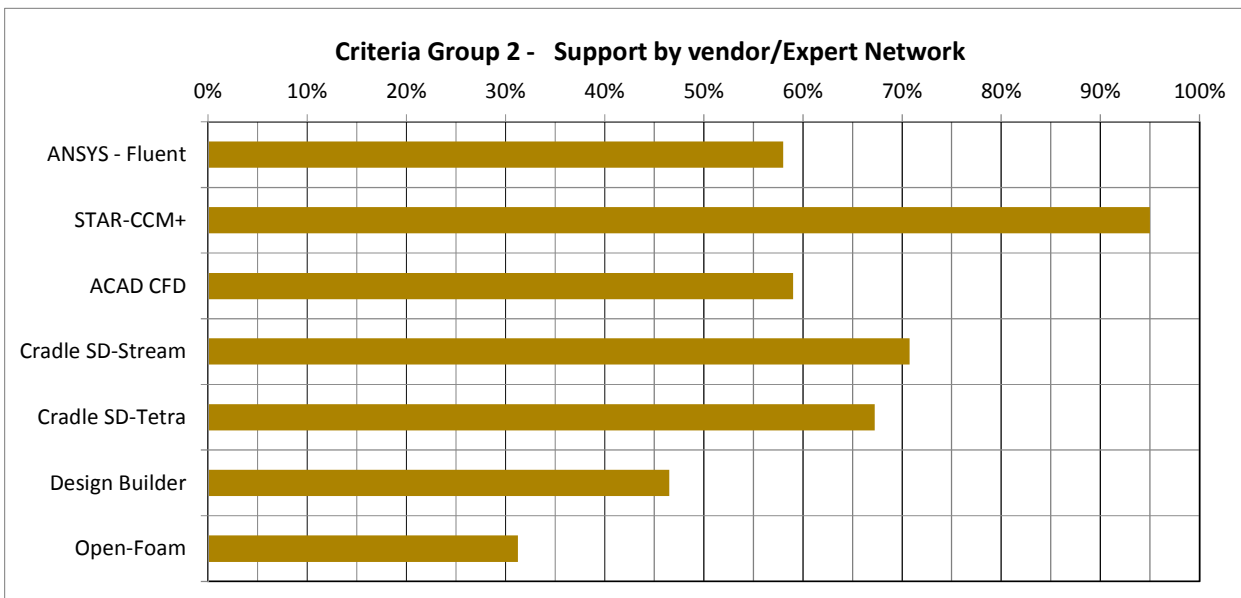
Figure 3.9.C: Results of initial ranking – Ranking for criteria group 2 - Criteria Group 2 - Support by vendor/Expert Network

Criteria Group 3 - Workflow

Figure 3.9D: Results of initial ranking – Ranking for criteria group 3 - Criteria Group 2 - CFD workflow

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 183

Ranking matrix for CFD software products for external CFD **DRAFT**

No.1	No.2	No.3	Description	weight first	weight second	weight third	weight overall	Fluent ranking	Fluent contr.	Fluent Points	STAR ranking	STAR contr.	STAR Points	ACAD ranking	ACAD contr.	ACAD Points	SD-Stream ranking	SD-Stream contr.	SD-Stream Points	SD-Tetra ranking	SD-Tetra contr.	SD-Tetra Points	Design Builder ranking	DB contr.	DB Points	Open-Foam ranking	OF contr.	OF Points
1			**Value for the CFD software product to the architecture and building market**	20%																								
	1		Benefit for external CFD applications to architecture		20%		4.0%	5	95%	3.8%	5	95%	3.8%	5	95%	3.8%	5	95%	3.8%	5	95%	3.8%	2	30%	1.2%	5	95%	3.8%
	2		Previous applications in architecture / urban design		35%		7.0%	5	95%	6.7%	3	50%	3.5%	1	5%	0.4%	5	95%	6.7%	2	30%	2.1%	1	5%	0.4%	1	5%	0.4%
	3		Previous mentioning of software product in scientific or applied professional articles		20%		4.0%	5	95%	3.8%	2	30%	1.2%	1	5%	0.2%	3	50%	2.0%	1	5%	0.2%	1	5%	0.2%	3	50%	2.0%
	4		Using graphics output for presentations		25%		5.0%	4	80%	4.0%	4	80%	4.0%	5	95%	4.8%	4	80%	4.0%	5	95%	4.8%	3	50%	2.5%	3	50%	2.5%
			sum second level >>>		100%		20.0%	test >>	18.3%		test >>	12.5%		test >>	9.1%		test >>	16.5%		test >>	10.9%		test >>	4.3%		test >>	8.7%	
2			**Support by vendor / expert network**	15%																								
	1		Technical support available		25%		3.8%	4	80%	3.0%	5	95%	3.6%	1	5%	0.2%	5	95%	3.6%	3	50%	1.9%	3	50%	1.9%	3	50%	1.9%
	2		Online tutorials available on demand		20%		3.0%	3	50%	1.5%	5	95%	2.9%	5	95%	2.9%	2	30%	0.9%	4	80%	2.4%	2	30%	0.9%	2	30%	0.9%
	3		Online classes available (low entry barriers)		20%		3.0%	2	30%	0.9%	5	95%	2.9%	5	95%	2.9%	4	80%	2.4%	5	95%	2.9%	2	30%	0.9%	2	30%	0.9%
	4		Network of CFD professionals in architecture field		15%		2.3%	2	30%	0.7%	5	95%	2.1%	1	5%	0.1%	3	50%	1.1%	1	5%	0.1%	3	50%	1.1%	1	5%	0.1%
	5		Ease of working with academic support		10%		1.5%	4	80%	1.2%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	4	80%	0.8%	2	30%	0.5%
	6		User guide provides the efficient information		10%		1.5%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	4	80%	1.2%	5	95%	1.4%	5	95%	1.4%	2	30%	0.5%
			sum second level >>>		100%		15.0%	test >>	8.7%		test >>	14.3%		test >>	8.9%		test >>	10.6%		test >>	10.1%		test >>	7.0%		test >>	4.7%	
3			**Workflow**	50%																								
	1		**Ease of use / GUI**		15%																								
		1	GUI is easy to use for trained operator			60%	4.5%	4	80%	3.6%	4	80%	3.6%	5	95%	4.3%	5	95%	4.3%	5	95%	4.3%	4	80%	3.6%	1	5%	0.2%
		2	User guide and online help is available for each command			40%	3.0%	3	50%	1.5%	5	95%	2.9%	5	95%	2.9%	3	50%	1.5%	4	80%	2.4%	5	95%	2.9%	1	5%	0.2%
			sum third level >>>			100%	7.5%	test >>	5.1%		test >>	6.5%		test >>	7.1%		test >>	5.8%		test >>	6.7%		test >>	6.5%		test >>	0.4%	
	2		**Modeling / Import geometry**		20%																								
		1	Create geometry inside CFD application			15%	1.5%	4	80%	1.2%	4	80%	1.2%	1	5%	0.1%	4	80%	1.2%	1	5%	0.1%	5	95%	1.4%	2	30%	0.5%
		2	Import geometry / repair surface before meshing			20%	2.0%	4	80%	1.6%	5	95%	1.9%	5	95%	1.9%	3	50%	1.0%	4	80%	1.6%	1	5%	0.1%	2	30%	0.6%
		3	Support Revit Architecture			30%	3.0%	3	50%	1.5%	1	5%	0.2%	5	95%	2.9%	2	30%	0.9%	2	30%	0.9%	1	5%	0.2%	1	5%	0.2%
		4	Support CADD format and Inventor			20%	2.0%	4	80%	1.6%	2	30%	0.6%	5	95%	1.9%	1	5%	0.1%	5	95%	1.9%	1	5%	0.1%	1	5%	0.1%
		5	Support Rhino			15%	1.5%	2	30%	0.5%	4	80%	1.2%	1	5%	0.1%	1	5%	0.1%	2	30%	0.5%	1	5%	0.1%	1	5%	0.1%
			sum third level >>>			100%	10.0%	test >>	6.4%		test >>	5.1%		test >>	6.8%		test >>	5.2%		test >>	4.9%		test >>	2.6%		test >>	1.4%	
	3		**Meshing**		20%																								
		1	Surface meshers			30%	3.0%	1	5%	0.2%	5	95%	2.9%	3	50%	1.5%	4	80%	2.4%	1	5%	0.2%	1	5%	0.2%	1	5%	0.2%
		2	Maximum cells and 2D-3D cell volume			10%	1.0%	5	95%	1.0%	5	95%	1.0%	3	50%	0.5%	5	95%	1.0%	5	95%	1.0%	1	5%	0.1%	5	95%	1.0%
		3	Boundary layer types			25%	2.5%	5	95%	2.4%	5	95%	2.4%	5	95%	2.4%	5	95%	2.4%	5	95%	2.4%	1	5%	0.1%	5	95%	2.4%
		4	Structured and Unstructured meshing			10%	1.0%	5	95%	1.0%	5	95%	1.0%	4	80%	0.8%	4	80%	0.8%	5	95%	1.0%	1	5%	0.1%	5	95%	1.0%
		5	Adaptive meshing capability			25%	2.5%	5	95%	2.4%	5	95%	2.4%	5	95%	2.4%	4	80%	2.0%	5	95%	2.4%	1	5%	0.1%	1	5%	0.1%
			sum third level >>>			100%	10.0%	test >>	6.8%		test >>	9.5%		test >>	7.6%		test >>	8.5%		test >>	6.8%		test >>	0.5%		test >>	4.6%	
	4		**Physics Models**		30%																								
		1	Solvers - Coupled / decoupled			15%	2.3%	5	95%	2.1%	5	95%	2.1%	4	80%	1.8%	5	95%	2.1%	5	95%	2.1%	1	5%	0.1%	5	95%	2.1%
		2	Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)			10%	1.5%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	3	50%	0.8%	5	95%	1.4%
		3	Steady State/Transient State			10%	1.5%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	3	50%	0.8%	5	95%	1.4%	2	30%	0.5%	5	95%	1.4%
		4	Turbulence models			35%	5.3%	5	95%	5.0%	5	95%	5.0%	5	95%	5.0%	1	5%	0.3%	5	95%	5.0%	1	5%	0.3%	5	95%	5.0%
		5	Materials			10%	1.5%	4	80%	1.2%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	1	5%	0.1%	3	50%	0.8%
		6	Customizable / special physics models			20%	3.0%	4	80%	2.4%	4	80%	2.4%	5	95%	2.9%	4	80%	2.4%	5	95%	2.9%	1	5%	0.2%	4	80%	2.4%
			sum third level >>>			100%	15.0%	test >>	13.6%		test >>	13.8%		test >>	13.9%		test >>	8.4%		test >>	14.3%		test >>	1.8%		test >>	13.1%	
	5		**Post processing**		5%																								
		1	Visualization			25%	0.6%	4	80%	0.5%	5	95%	0.6%	5	95%	0.6%	5	95%	0.6%	5	95%	0.6%	3	50%	0.3%	3	50%	0.3%
		2	Quantitative analysis capabilities of components			25%	0.6%	4	80%	0.5%	5	95%	0.6%	5	95%	0.6%	4	80%	0.5%	1	5%	0.0%	4	80%	0.5%	4	80%	0.5%
		3	Comparative analysis (multi-scenarios)			50%	1.3%	4	80%	1.0%	3	50%	0.6%	5	95%	1.2%	4	80%	1.0%	5	95%	1.2%	1	5%	0.1%	1	5%	0.1%
			sum third level >>>			100%	2.5%	test >>	2.0%		test >>	1.8%		test >>	2.4%		test >>	2.2%		test >>	2.3%		test >>	0.4%		test >>	0.9%	
	6		**Customization / special functions**		5%																								
		1	Pedestrain comfort			15%	0.4%	5	95%	0.4%	5	95%	0.4%	3	50%	0.2%	5	95%	0.4%	5	95%	0.4%	2	30%	0.1%	5	95%	0.4%
		2	Natural ventilation			25%	0.6%	5	95%	0.6%	5	95%	0.6%	5	95%	0.6%	5	95%	0.6%	5	95%	0.6%	3	50%	0.3%	3	50%	0.3%
		3	Wind-driven rain			10%	0.3%	3	50%	0.1%	5	95%	0.2%	1	5%	0.0%	4	80%	0.2%	4	80%	0.2%	1	5%	0.0%	3	50%	0.1%
		4	Humidity			15%	0.4%	1	5%	0.0%	3	50%	0.2%	5	95%	0.4%	5	95%	0.4%	5	95%	0.4%	1	5%	0.0%	1	5%	0.0%
		5	Wind load towards to building structure			5%	0.1%	5	95%	0.1%	3	50%	0.1%	5	95%	0.1%	5	95%	0.1%	3	50%	0.1%	2	30%	0.0%	5	95%	0.1%
		6	Pressure distribution along streamlines/pressure losses			20%	0.5%	5	95%	0.5%	2	30%	0.2%	1	5%	0.0%	4	80%	0.4%	5	95%	0.5%	1	5%	0.0%	3	50%	0.3%
		7	Conjugate heat transfer			10%	0.3%	5	95%	0.2%	5	95%	0.2%	5	95%	0.2%	5	95%	0.2%	5	95%	0.2%	1	5%	0.0%	5	95%	0.2%
			sum third level >>>			100%	2.5%	test >>	1.9%		test >>	1.8%		test >>	1.5%		test >>	2.3%		test >>	2.3%		test >>	0.5%		test >>	1.4%	
	7		**Support for validation**		5%																								
		1	Validation document / example			100%	2.5%	5	95%	2.4%	1	5%	0.1%	1	5%	0.1%	5	95%	2.4%	5	95%	2.4%	2	30%	0.8%	2	30%	0.8%
			sum third level >>>			100%	2.5%	test >>	3.1%		test >>	0.5%		test >>	0.4%		test >>	3.0%		test >>	3.1%		test >>	0.8%		test >>	1.2%	
			sum second level >>>		100%																									
4			**Cost of software (Not considered in DRAFT ranking)**	15%				(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)			(Not considered in DRAFT ranking)		
	1	1	Commercial pricing model			20%	3.0%	3	50%	1.5%	3	50%	1.5%	3	50%	1.5%	3	50%	1.5%	4	80%	2.4%	3	50%	1.5%	1	5%	0.2%
	2	2	Academic pricing			30%	4.5%	4	80%	3.6%	5	95%	4.3%	5	95%	4.3%	3	50%	2.3%	4	80%	3.6%	5	95%	4.3%	5	95%	0.2%
	3	3	Software includes all modules (no extra costs)			20%	3.0%	5	95%	2.9%	5	95%	2.9%	5	95%	2.9%	3	50%	1.5%	4	80%	2.4%	4	80%	2.4%	5	95%	0.2%
	4	4	Option to aquire sufficient seats at no extra costs			30%	4.5%	5	95%	4.3%	5	95%	4.3%	1	5%	0.2%	3	50%	2.3%	5	95%	4.3%	5	95%	4.3%	1	5%	0.2%
			sum second level >>>		100%		15.0%	test >>	12.2%		test >>	12.9%		test >>	8.9%		test >>	7.5%		test >>	12.0%		test >>	12.5%		test >>	0.8%	
			sum of first level >>>	100%				sum overall points >>	65.1%		sum overall points >>	65.3%		sum overall points >>	57.4%		sum overall points >>	61.8%		sum overall points >>	60.5%		sum overall points >>	24.3%		sum overall points >>	35.8%	
								Sum does not include costs			Sum does not include costs			Sum does not include costs			Sum does not include costs			Sum does not include costs			Sum does not include costs					
								ANSYS - Fluent			**STAR-CCM+**			**ACAD CFD**			**Cradle SD-Stream**			**Cradle SD-Tetra**			**Design Builder**			**Open-Foam**		

The ranking criteria is shown below:

ranking criteria

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

DRAFT

Table 3.9.2: Breakdown of all ranking criteria and contribution from particular software products

Part 4 Benchmarking of Candidate CFD Products Workflow Performance for External CFD

Following the initial ranking, which used vendor supplied documents, the CFD research team performed benchmarking workflow tests to obtain first-hand experience of the specific CFD software products. The following presents the results and conclusion of the benchmarking workflow tests that were conducted for six of the original seven candidate CFD software. The CFD software products which were tested are as follows:

- ANSYS - Fluent
- STAR-CCM+
- Autodesk Simulation CFD
- Cradle scStream
- Cradle SC/Tetra
- Design Builder

The CFD team had received temporary access to the six CFD software products and conducted the benchmarking tests within a period of six weeks. The conclusions obtained in the benchmarking tests were used to compile the final overall ranking of the candidate CFD software products.

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

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

Solver: All solver setting were selected to be similar for all six software CFD products. Since not all CFD software products provide the same sophistication in selecting certain physics parameters, the physics stetting, especially the turbulence model, was selected to accommodate the least sophisticated CFD software products. In essence this means that the test with the more sophisticated software products, such as Star-CCM+ and Fluent, used only basic settings and the quality of simulation results did not reflect the overall potential of these CFD software. Therefore,

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 185

the results of the benchmarking test should be viewed only in regard to the effectiveness of the workflow and not in terms of direct quantitative comparison of the results for velocity and pressure.

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

4.1 Framework for Conducting the Benchmarking Tests

The framework included the following parameters and setting:

4.1.1. Geometry File Type Format:

Both building geometry and adjacent mass models are originally created with Revit CAD architecture. Using Revit architecture is the raw model and geometry file that can be export to different file type formats, which are compatible with the candidate CFD software. The Revit CAD architecture's modeling interface is convenient for creating and modifying geometry models. The CFD team believes that Revit could be the primary modeling tool for CFD building model in the subsequent CFD project work.
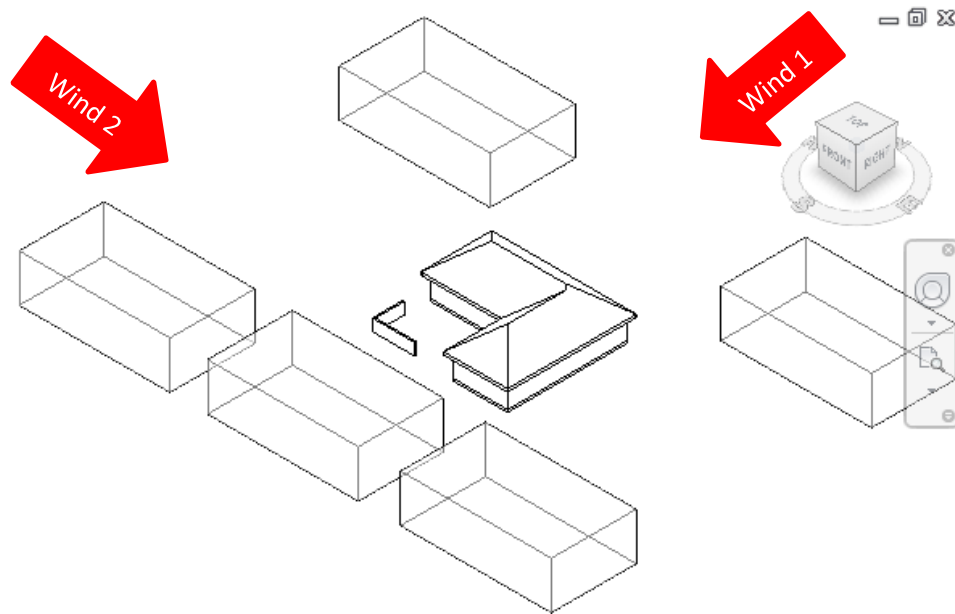
Figure 4.1.1: Given building geometry and massing adjacent models are created from Revit architecture

CFD softwares	1. Revit Architecture (Original) "RVT"	2. Revit export to CAD ".SAT"	3. Revit (.SAT) to Inventor ".IAM"	4. Revit export "gbXML"
ANSYS - Fluent		X	X	
STAR-CCM+		X	X	
Autodesk Simulation CFD	X	X	X	
Cradle SD-Stream		X	X	
Cradle SD-Tetra		X	X	
Design Builder	X			X

Table 4.1.1: Compare geometry file transition to the six candidate CFD software products.

4.1.2 Wind Approach Directions:

Two major wind approach directions (Figure 4.1.1) were selected for the benchmarking tests. The two wind directions are from the North (Wind direction 1) and from the West (Wind direction 2). The benchmarking workflow tests conducted simulations for both wind directions.

4.1.3   Atmosphere Boundary Layer (ABL):

The following parameters were used to describe the effect of the atmospheric boundary layer.

- K-Epsilon turbulence modeling in the ABL
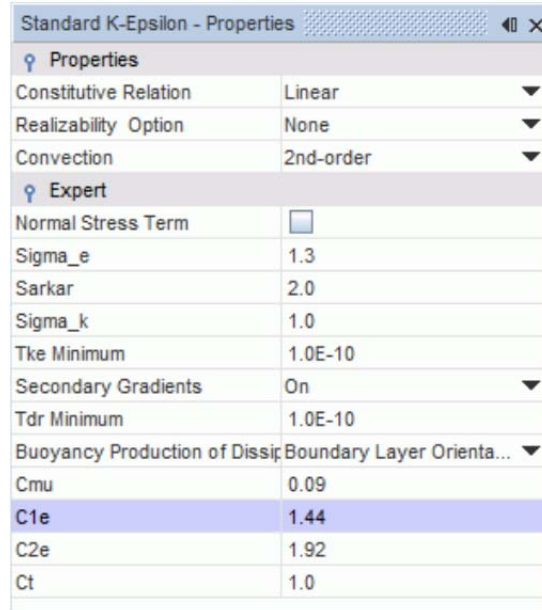
Figure 4.1.2: Turbulence modeling set up for ABL from Star-CCM+ user guide

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 187

- Ground roughness

Type of terrain	Roughness height z_0 [m]
Cities, forests	0.7
Suburbs, wooded countryside	0.3
Villages, countryside with trees, hedges	0.2
Open farmland, few trees	0.03
Flat desert, rough sea	0.001

Figure 4.1.3: Ground roughness for ABL from Star-CCM+ user guide

4.1.4 Boundary Condition

The dimensions of the computational domain were selected as 1,000 feet by 1,000 feet. This created a symmetrical virtual wind tunnel, in both main wind directions, regardless on upwind and downwind conditions. This approach had the advantage that CFD simulations could be initiated by merely redefining the boundary condition for inlet and outlet, depending on the wind direction.
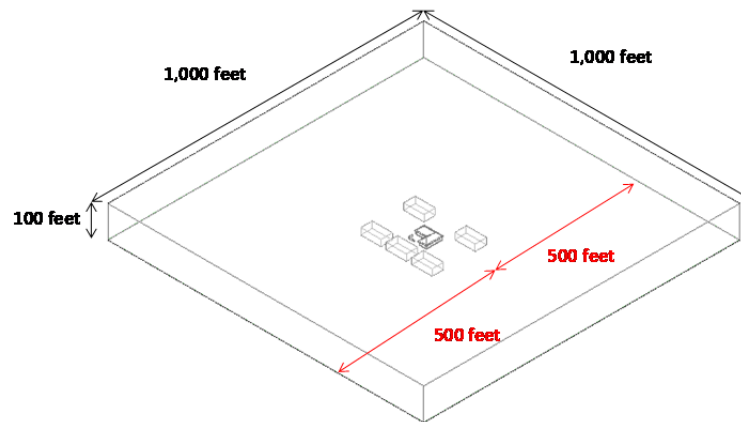
Figure 4.1.4: Example of axonometric view of boundary dimension

4.1.5 Boundary Layer Conditions

The boundary layer conditions are depicted in Figure 4.1.5, for both main wind directions.
- Descriptions of boundary layer types:
  - **Velocity inlet:** A velocity inlet boundary represents the inlet of a duct where wind velocity is known. The flow type is selected as incompressible internal flow.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 188

- **Wall:** A wall boundary is represented as an impenetrable, no-slip boundary for viscous flow simulations. The "Slip walls" are impenetrable but traction-free surface.
- **Pressure outlet:** The outlet boundary is defined as an outlet pressure boundary and the air flow is described as a compressible and internal flow.

- Inlet boundary condition: The inlet is defined as a velocity inlet with a wind speed of 7 meters per second
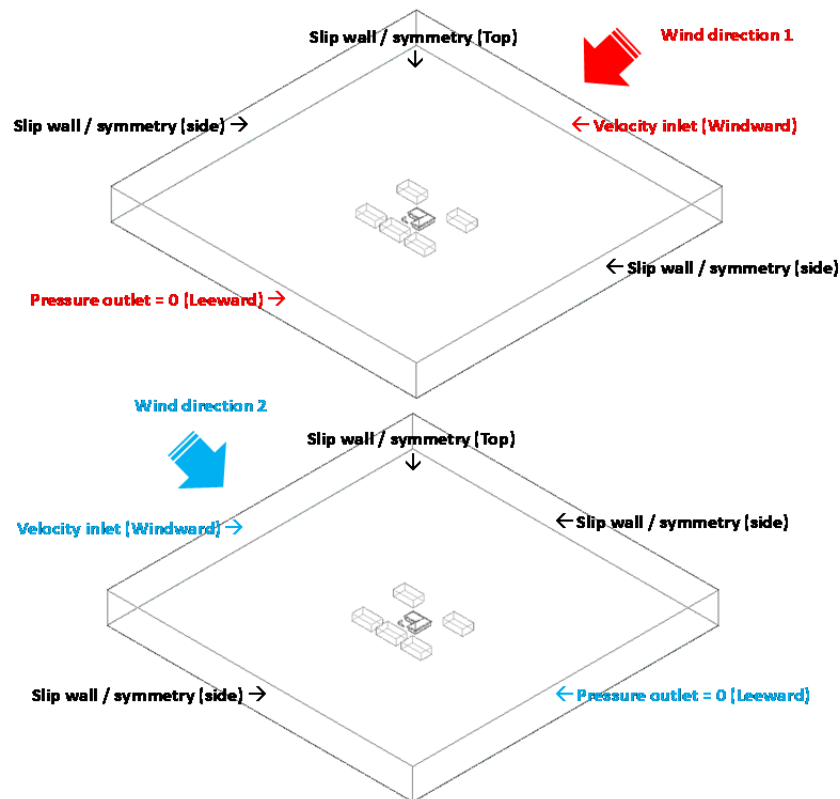
Figure 4.1.5: Example of wind directions and assigned boundary layer types

- The outlet boundary condition is defined by zero static gage pressure.

- Assign **Slip wall /Symmetry** to the top and sides of the region. (Do not specify a condition on the ground plane, because the air does not physically move along the ground plane.)
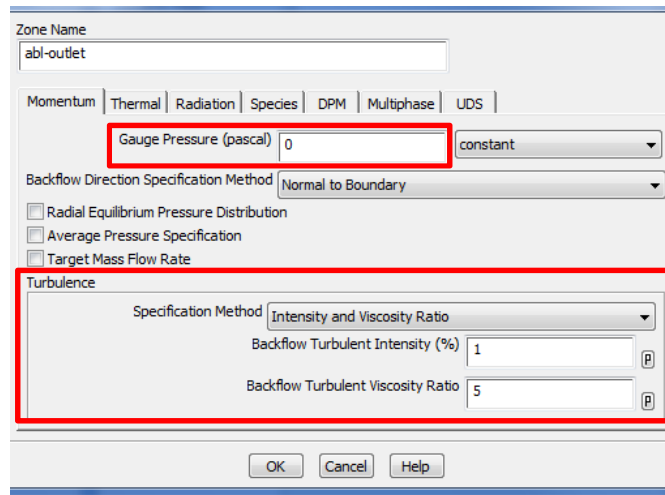
4.1.6 Surface Meshers:

The CFD team tried to use the most effective surface meshing tools of the CFD software tested. Since the meshing tools differed between CFD software products a range of meshing tools was sued, such as Surface Wrapper, Remesher, or Automatic surface repair. It is noted that not all candidate CFD software have mesh surface editing capabilities.

4.1.7 Structured and Unstructured Grid:

Since the CFD software differed in what mesh type was created the CFD team selected the most effective mesh type offers by the specific software. This included structured and unstructured girds. The CFD team furthermore used specific software tools to create varying resolution of the mesh around the area of interest. The tested CFD products provided different approaches to create varying resolutions and densities of the mesh.
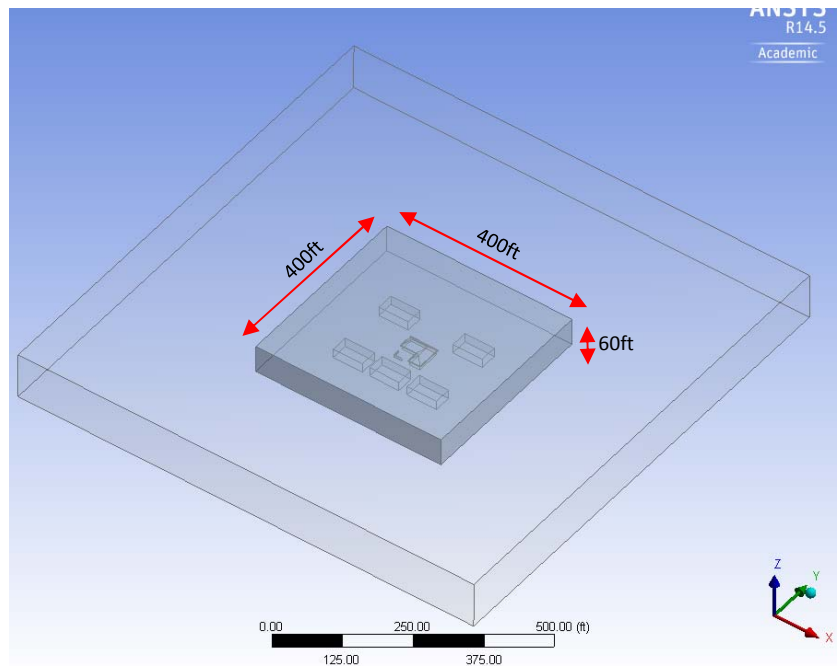
The higher solution area (cell size = 2m) 400'x400'x60'

Figure 4.1.6: Varying mesh resolution
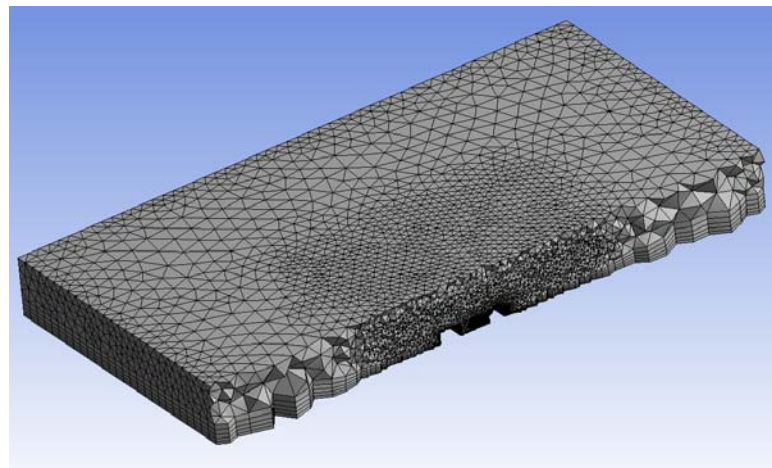
Figure 4.1.7: Depiction of volume mesh
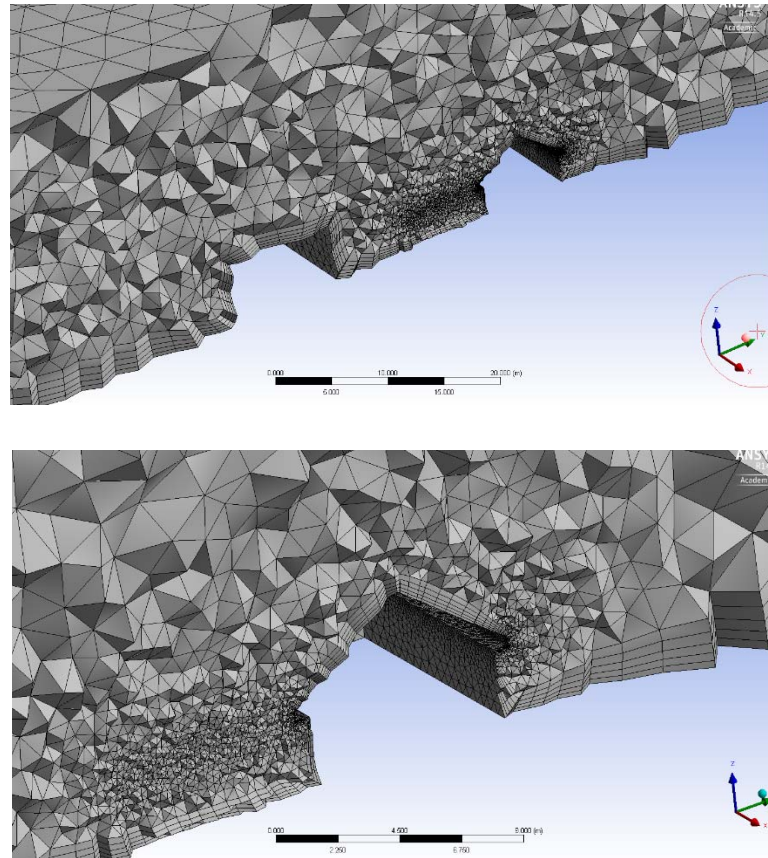
The minimum cell : 0.1m

The maximum cell: 10m

Cell size at L-shap building surface: 0.5m

Cell size at the focus area: 2m

Cell growth rate: 1.2 (20%)

The prism layer: 5 layers

Figure 4.1.8: Depiction of volume mesh around area of interest

4.1.8   Type of Simulation

Steady state simulation is used for all candidate CFD software.

4.1.9 Turbulence Model

Since not all candidate CFD software products provide a range of turbulence models, the basic use standard k-epsilon (k-ε) as used.
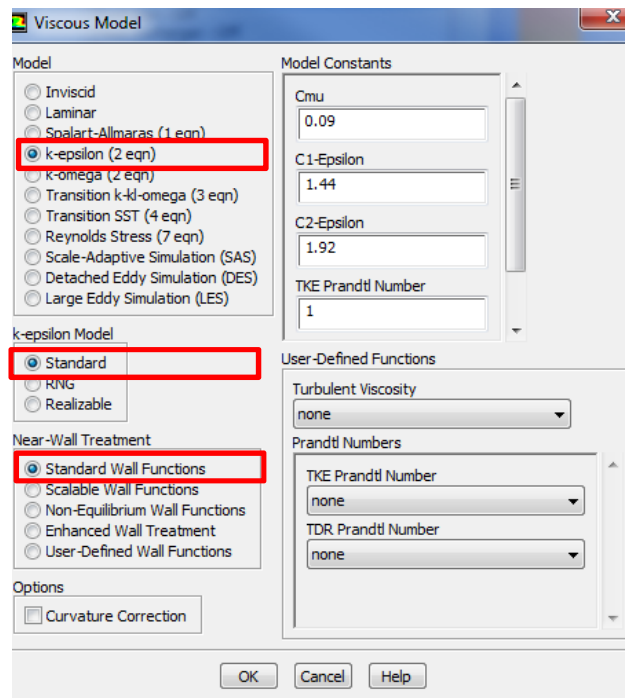
4.1.10 Materials:

Air is the only material that has to be defined with the following air properties:

Note that not all candidate CFD software products allow the setting of air properties

4.1.11 Solver Settings

The solver setting used was the SIMPLE scheme with second order upwind for momentum, turbulent kinetic energy and turbulent dissipation rate.
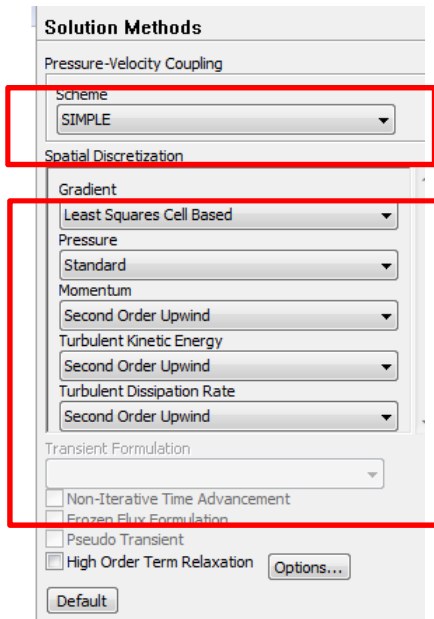
4.1.12 Convergence Criteria

Maximum step / Iterations were set to use a maximum of 1,000 iterations as the convergence criteria.
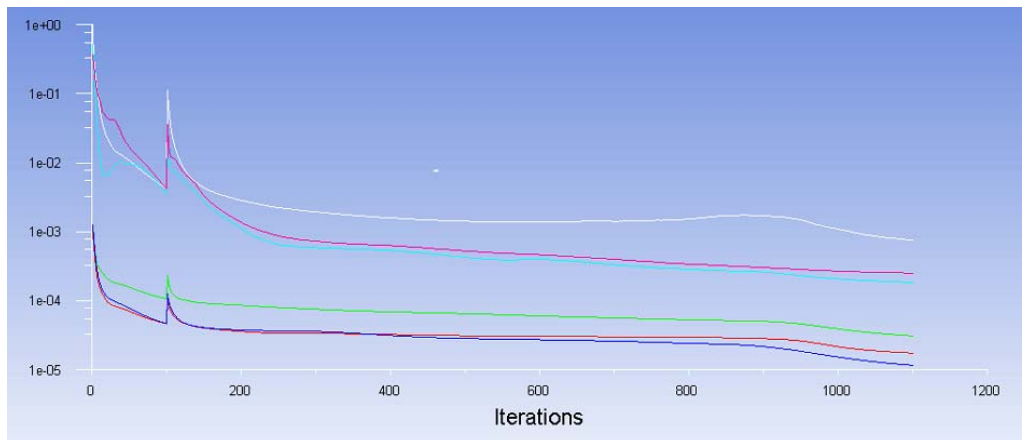
Figure 4.1.9: Typical convergence performance of solution

4.1.13   Post Processing

The CFD team tried to generate uniform post processing images for velocity and pressure using slices, surface renderings and streamlines. Figures 4.1.10 through 4.1.23 provide a guideline on post processing images that the CFD team was generating in the benchmarking workflow tests. Horizontal slices were generated a given elevation for all post processing visualizations.
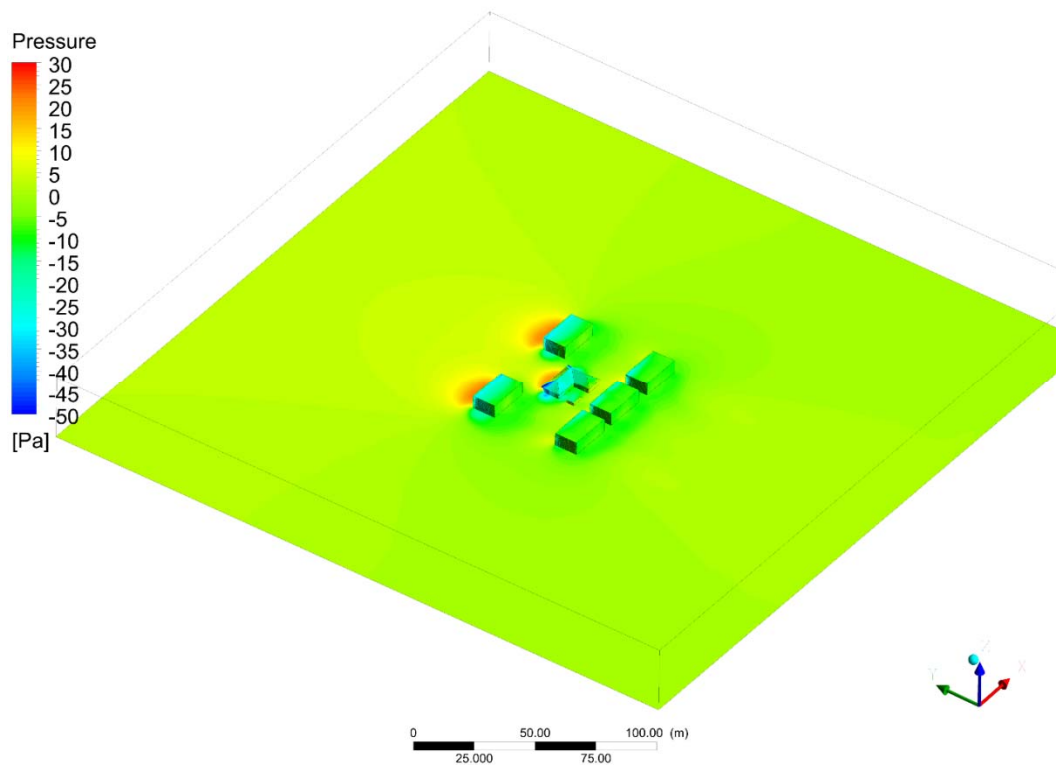
Figure 4.1.10:   Post-Processing - Pressure (static) distribution on building surface and the ground (looking from SW direction)
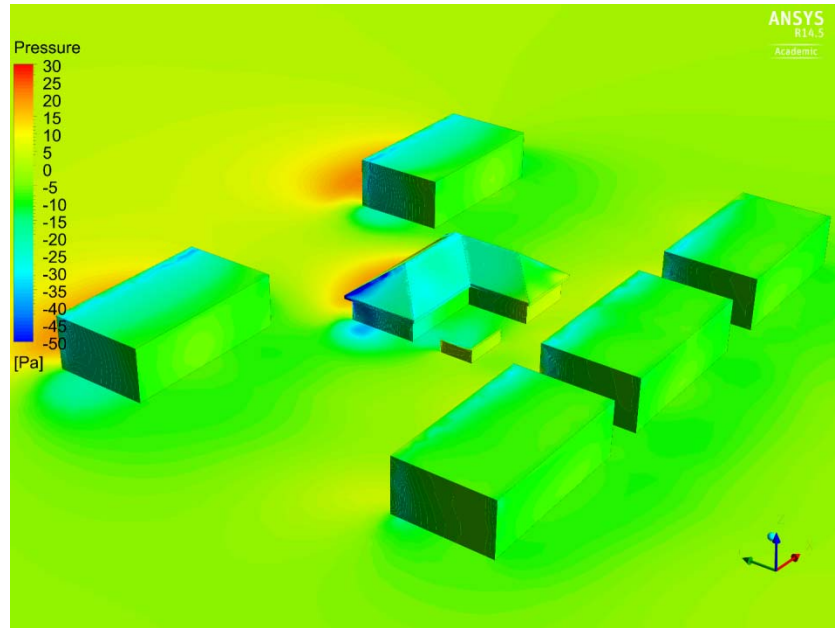
Figure 4.1.11:   Post-Processing - Pressure (static) distribution on building surface and the ground
(looking from SW direction) close-up view1
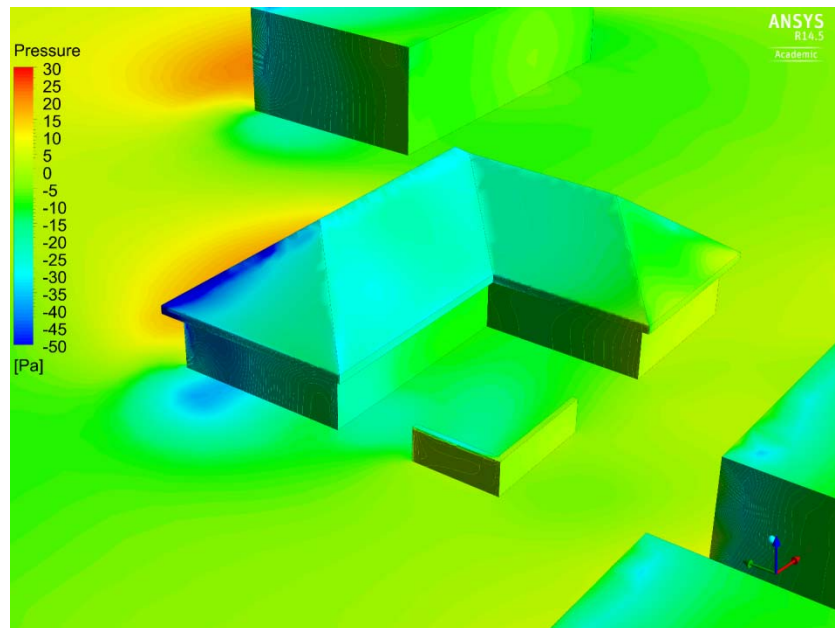
Figure 4.1.12:   Post-Processing - Pressure (static) distribution on building surface and the ground
(looking from SW direction) close-up view2
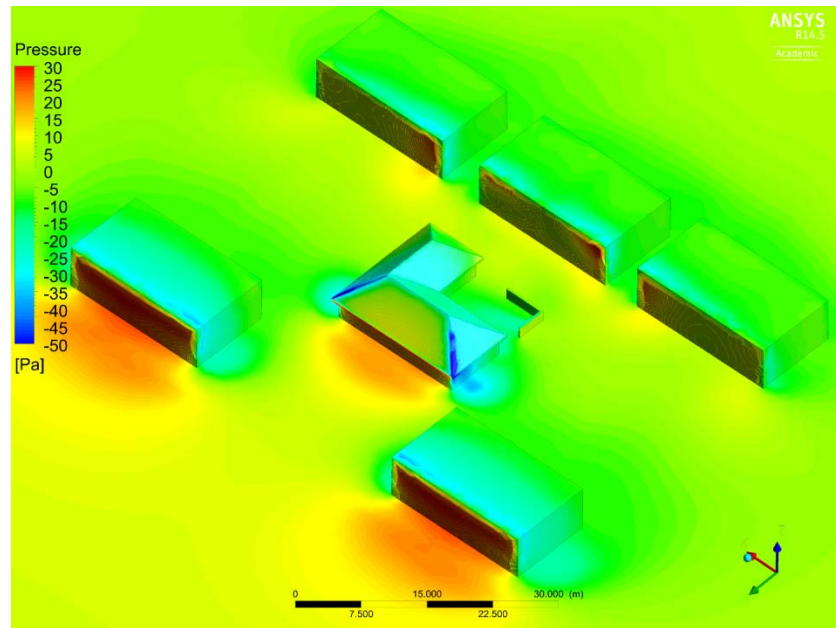
Figure 4.1.13:   Post-Processing - Pressure (static) distribution on building surface and the ground
(looking from NW direction) close-up view1
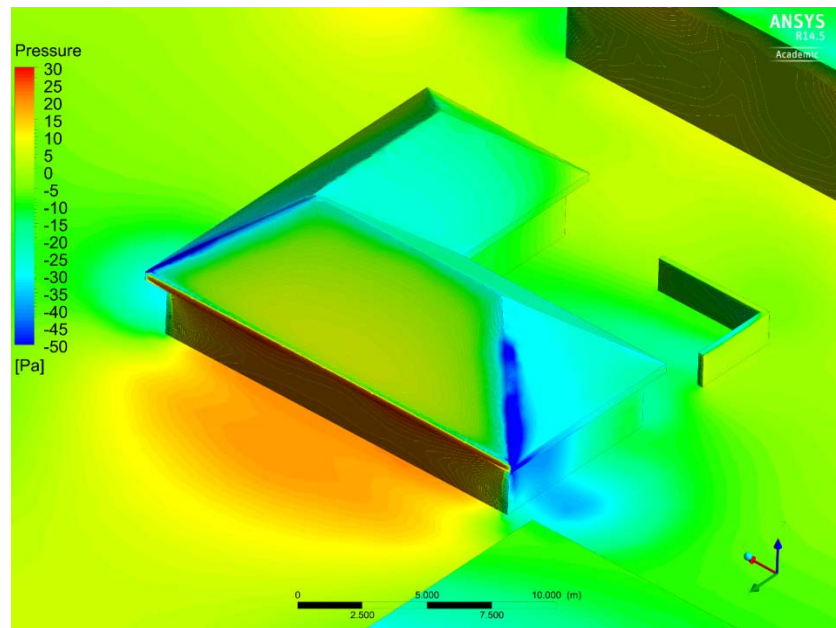
Figure 4.1.14:   Post-Processing - Pressure (static) distribution on building surface and the ground
(looking from NW direction) close-up view2

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                          School of Architecture, University of Hawaii

December 4, 2013   Page 198
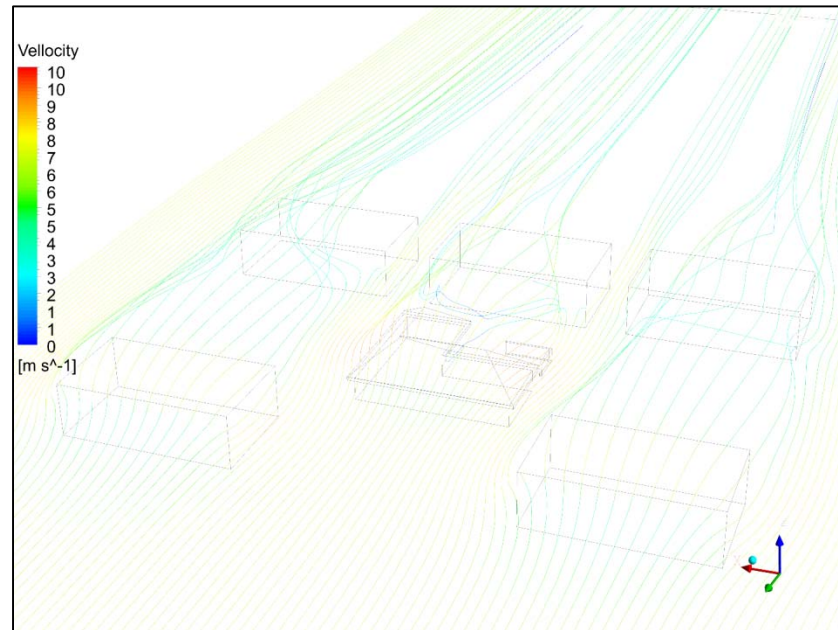
Figure 4.1.15:   Post-Processing - Provide image of velocity stream line
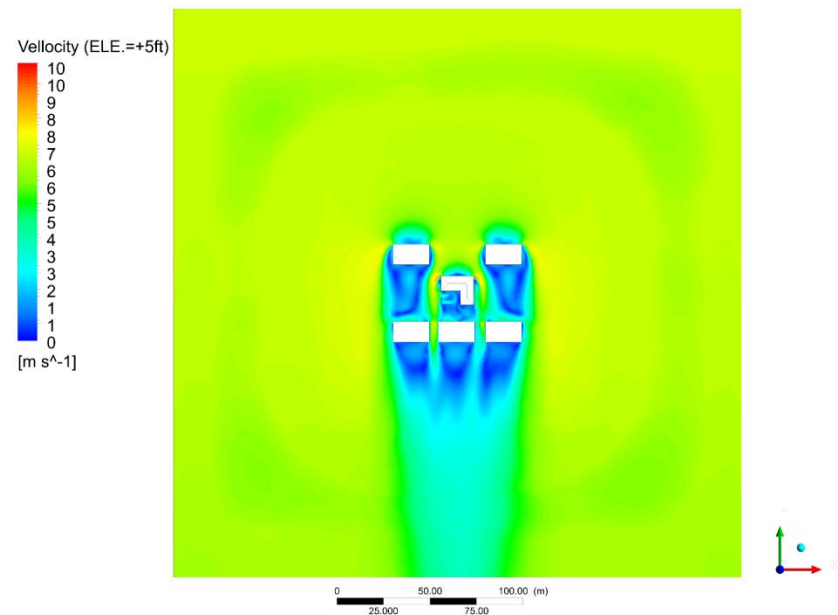
Figure 4.1.16:   Post-Processing - Wind velocity at the plane 5ft height level
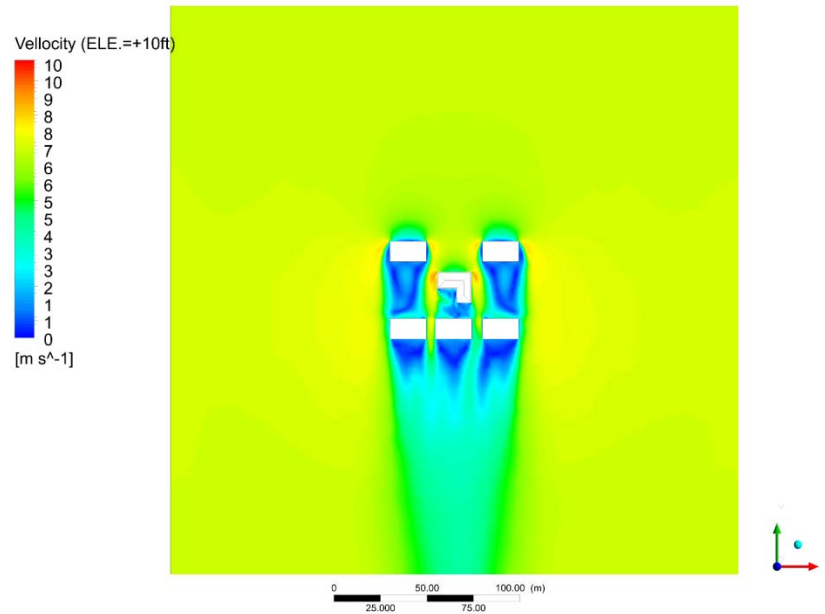
Figure 4.1.17:   Post-Processing - Wind velocity at the plane 10ft height level
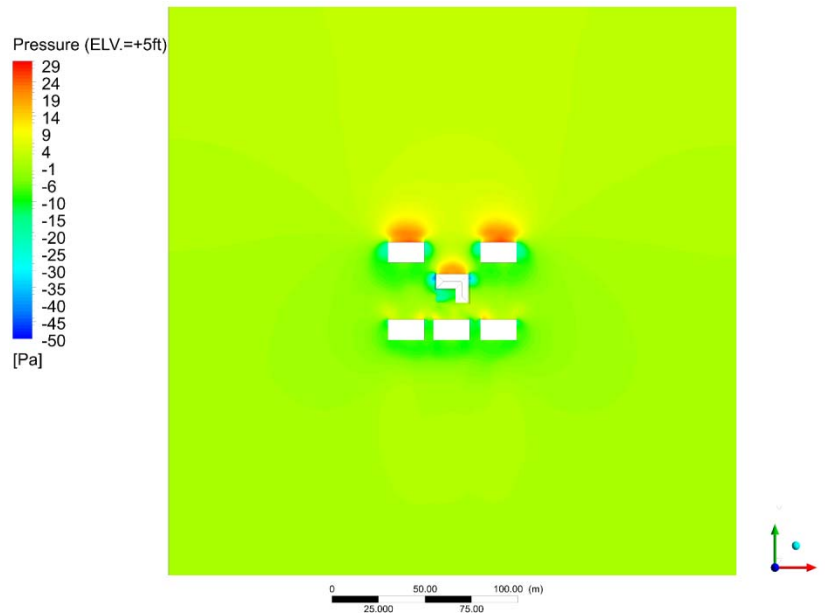
Figure 4.1.18:   Post-Processing - Wind pressure at the plane 5ft height level
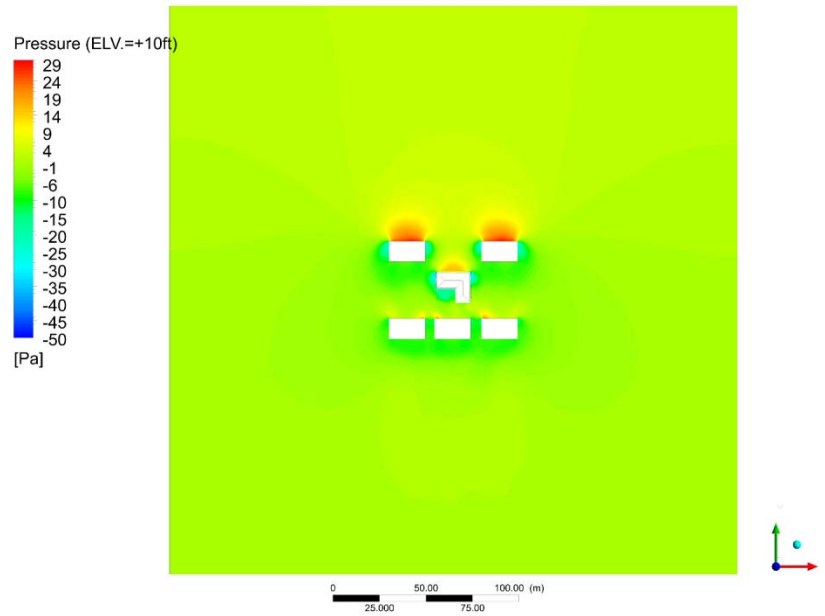
Figure 4.1.19: Post-Processing - Wind pressure at the plane 10ft height level
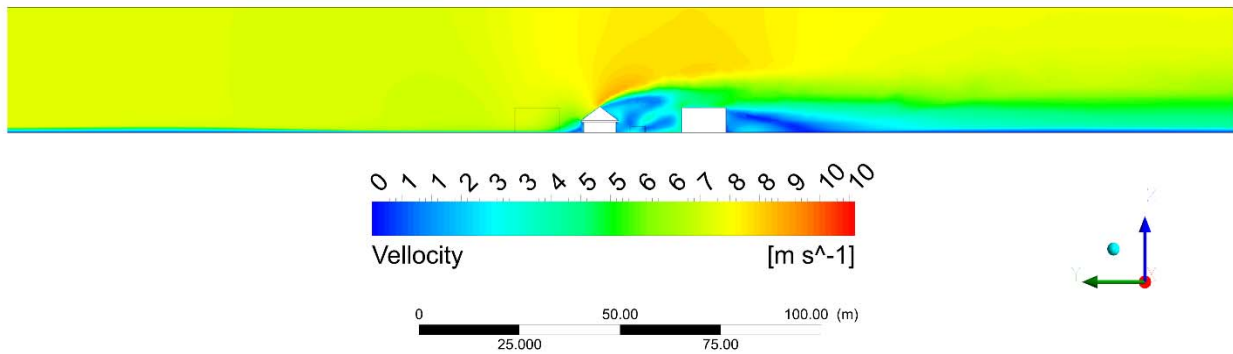
Figure 4.1.20: Post-Processing - Wind velocity at section (parallel to wind direction North South)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                     School of Architecture, University of Hawaii

December 4, 2013   Page 201
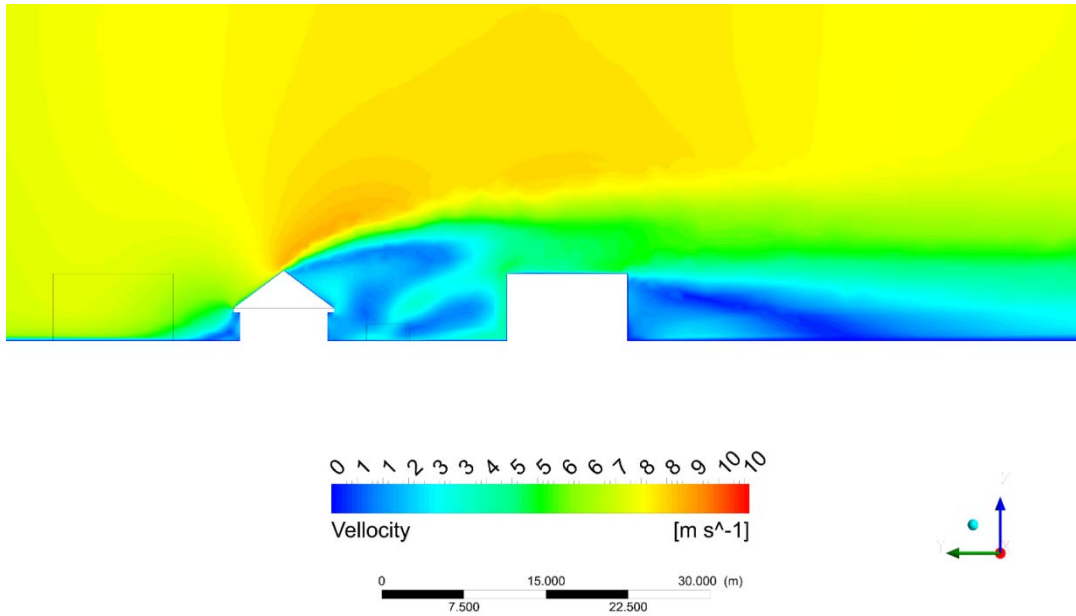
Figure 4.1.21: Post-Processing - Wind velocity at section (parallel to wind direction North South) – close-up view
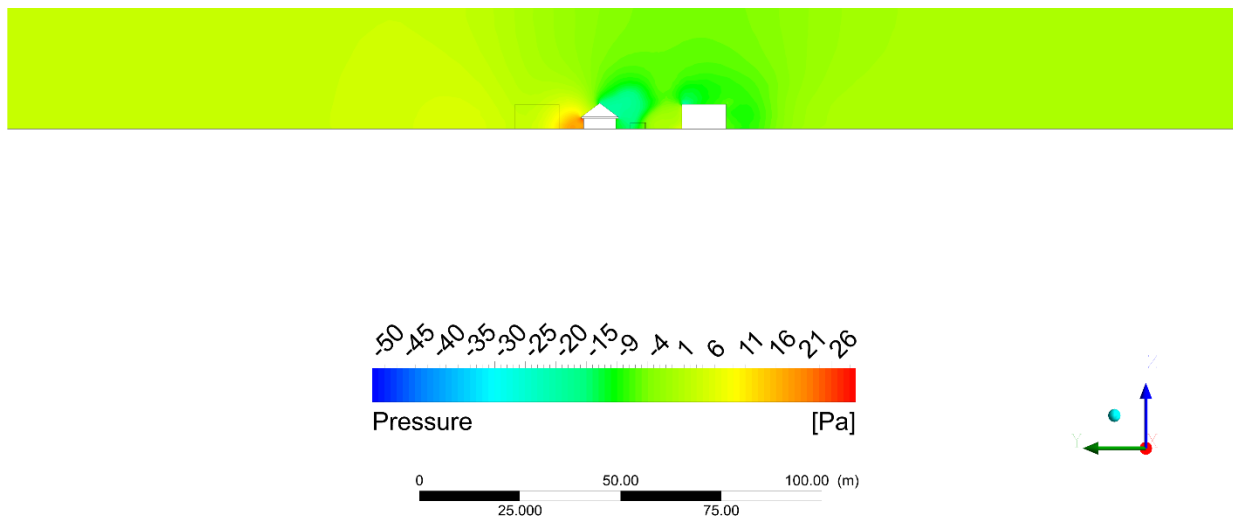
Figure 4.1.22: Post-Processing - Wind pressure at section (parallel to wind direction North South)

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 202
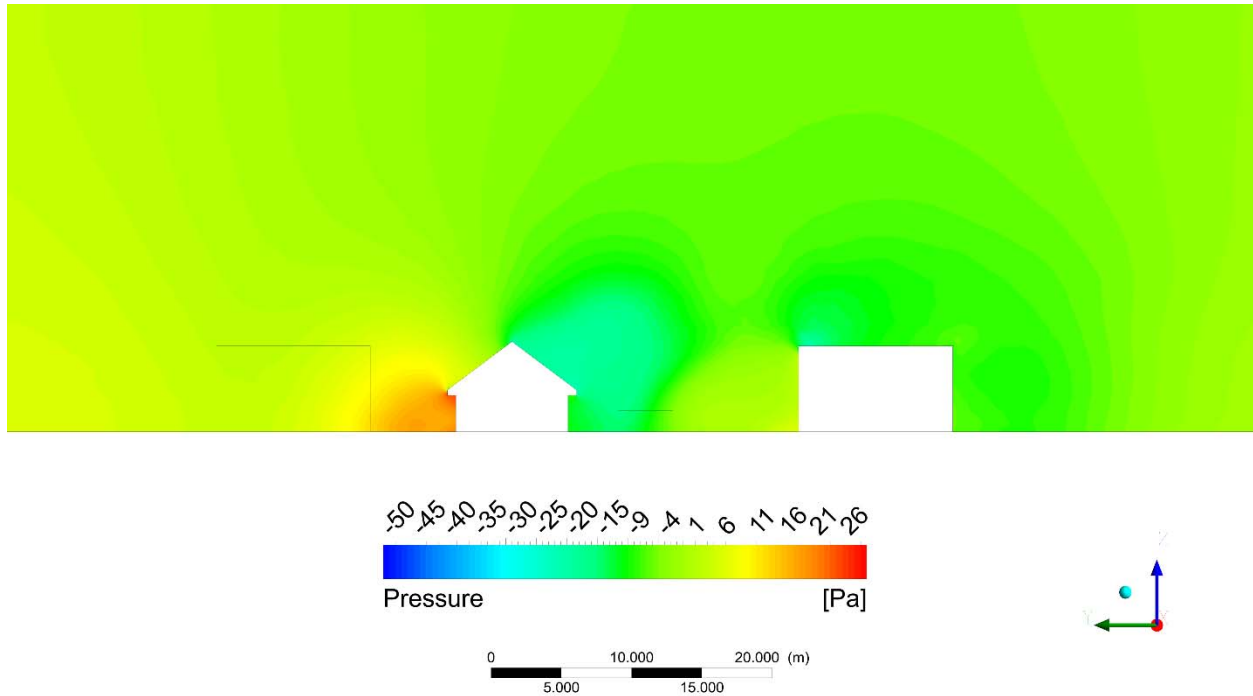
Figure 4.1.23: Post-Processing - Wind pressure at section (parallel to wind direction North South)

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 203

4.2 Benchmarking the External CFD Workflow for ANSYS – Fluent

This section presents the results and conclusions of the benchmark testing for external CFD for the software ANSYS – Fluent (Fluent). The section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These ten criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

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

   Working with ANSYS Fluent, one of the most impressive features is the WorkBench function which allows an effective organization of the simulation processes. Even though working on the very simple CFD test case, the workbench function helped to avoid creating, and therefore maintaining, numerous file types such as geometry files, meshing files, solution files and resultant data (Figure 4.2.1). The different files that are created during the workflow are replaced by a single WorkBench file for a CFD problem. WorkBench also provides good opportunities for parametric running with less effort.

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

   The CFD team tried to import CAD geometries as Inventor format files, which are available and conveniently converted from Revit models. However, the CAD import process was not successful using this file format. Instead, geometry files had to be converted from Inventor format into ACIS format (sat file). The Fluent internal geometry builder application "DesignModer" allows effective model geometry generation directly inside of Fluent. This internal function is convenient and especially powerful when using simple geometries and analyzing parametric cases. However, for architectural purposes, importing geometry from stand-alone 3D geometry modeler such as Autodesk Inventor is still a practical and preferred option.

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

   Fluent's mesher function works with unstructured meshing of tetrahedral and prims cells. This meshing tool is effective and powerful for setting up different resolution settings for specific objects or regions. The CFD team did not encounter difficulties in adding prism layers to the building's surfaces and ground and therefore the mesh generation process presents a logical and easy to

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 204

follow sequence of functions. The meshing generation for the test case with Fluent is shown in Figure 4.2.2 and Figure 4.2.3. Fluent also has the capability of adaptive meshing and converting the mesh from tetrahedral volume cells into polyhedral volume cells, but these meshing functions have not been tested for this test case and therefore, the CFD team could not determine how efficient the conversion process is.

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

The CFD team has experienced the Fluent's mesh generation application as an effective and intuitive process. The team believes that the mesh generation represents a significant contribution of the Fluent CFD software. Furthermore, the capability to visualize volume cell sections as well as statistics on volume cells is a helpful feature in determining the quality of the mesh.

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

Setting physics and boundary layer conditions for a CFD solution in Fluent is straight forward and effective. Fluent allows customizing any physics properties as well as boundary conditions. Fluent offers a wide range of spatial discretization schemes and different initializations as well as monitoring solution convergence.

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

In the test cases, the convergence after 1000 iterations is good without encountering problems that could have impeded the simulation process. The convergence plot of the test case is shown in Figure 4.2.4.

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

Yes, the convergence also could be improved if meshing was further refined. It took about two hour of computing time for the solution to converge.

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

During the time working on the test case with Fluent, the CFD team did not have encountered any significant difficulties. Sometimes the WorkBench function did not update correctly and the CFD team had to reconfigure the process instead.

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

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 205

Fluent's post-processor functionalities provide a wide range of visualization options as well as quantitative data queries. Most of the typical post-processing features typically offered by a commercial CFD packages are available in the software's CFD-Post function. However, overall the post processing tools are not as intuitive and user-friendly as was expected by the CFD team. When working in conjunction with BenchWork, the CFD-Post functions are efficient when doing comparative testing of multiple cases.

Some examples of the post-processing results for this benchmarking investigation are shown in the figures at the end of this section, including contour maps for pressure distribution on the ground and building surfaces for wind coming from the north (Figure 4.2.5 through Figure 4.2.10), contour maps showing pressure and wind velocity on the horizontal plane (e.g. slice) 10 feet above the ground (Figure 4.2.11 and Figure 4.2.12), the pressure and wind velocity on the section plane (Figure 4.2.13 and Figure 4.2.14), the vector map of wind velocity (Figure 4.2.15) and the over-laid contour map of pressure and velocity streamlines (Figure 4.2.16).

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

In general, Fluent is a powerful general purpose CFD code that offers high quality CFD simulation capabilities for a wide range of fluid and thermal processes. The analysis of external or internal airflow cases represents a small part of the wide range of possible fluid analysis option. Fluent requires an advanced skill set and solid understanding of CFD principles in order to implement more advanced features such as customable user-defined functions, especially for features that are of importance to the CFD team, including inlet boundary conditions for atmospheric boundary layer and wall functions.

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

During working on the test case, the CFD team did not need to contact ANSYS support desk for assistance. Instead, there is comprehensive documentation and numerous tutorials available on the ANSYS's user portal website. These training and reference assistance was sufficient for the CFD team to conduct the benchmarking test. The CFD team, however, could not find many resources relating to external airflow CFD in architecture/building fields.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 206

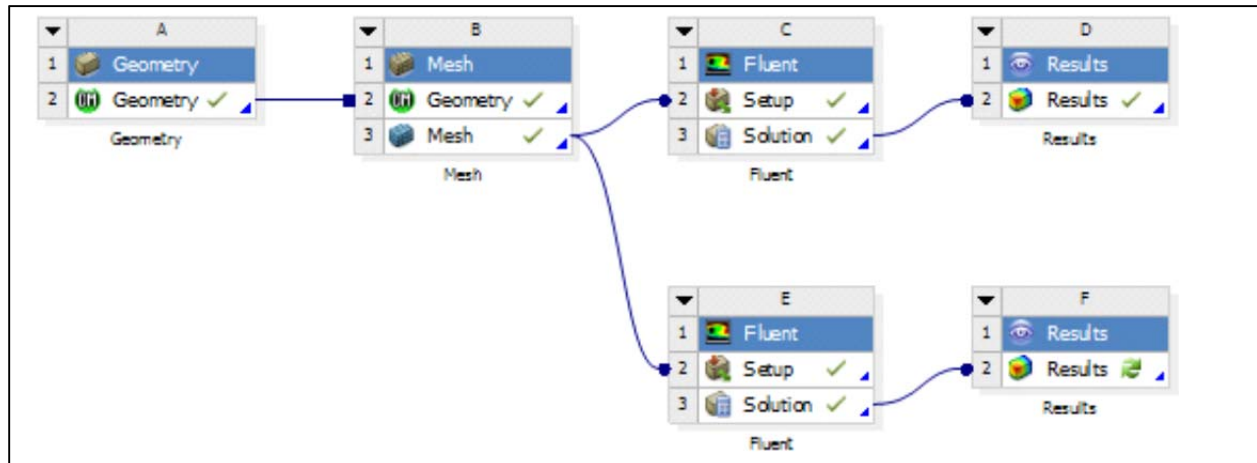**Figure 4.2.1 through 4.2.16 are shown below:**

Figure 4.2.1: The workflow of the test case simulating two wind directions using WorkBench (ANSYS Fluent).
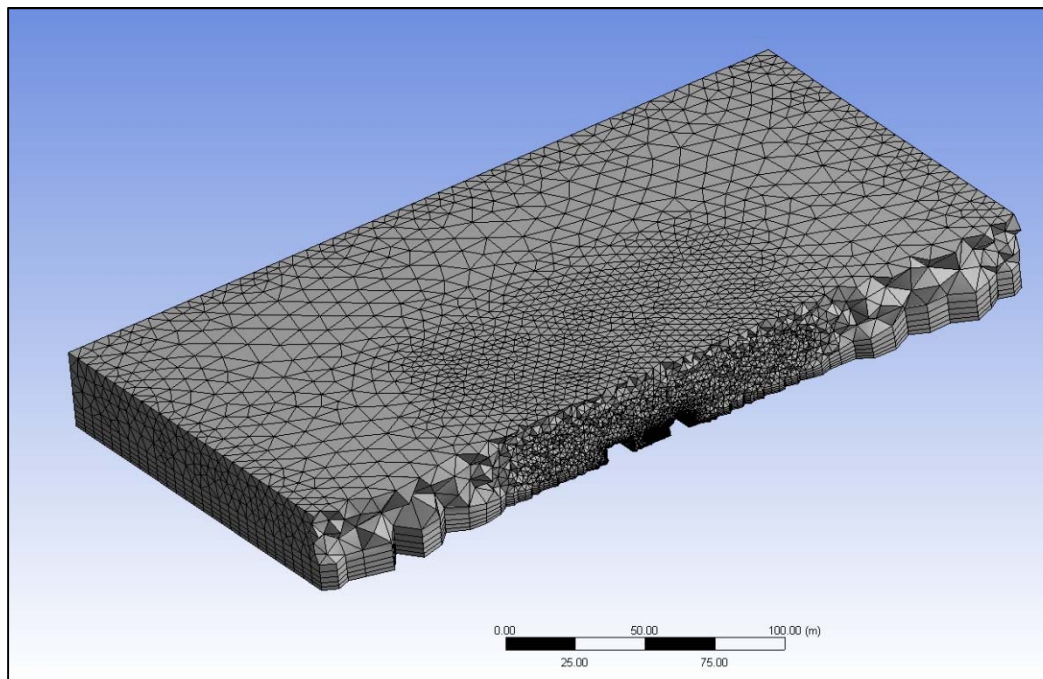

Figure 4.2.2: Cross section showing volume cells consisting of tetrahedrons and prism layers (ANSYS Fluent).
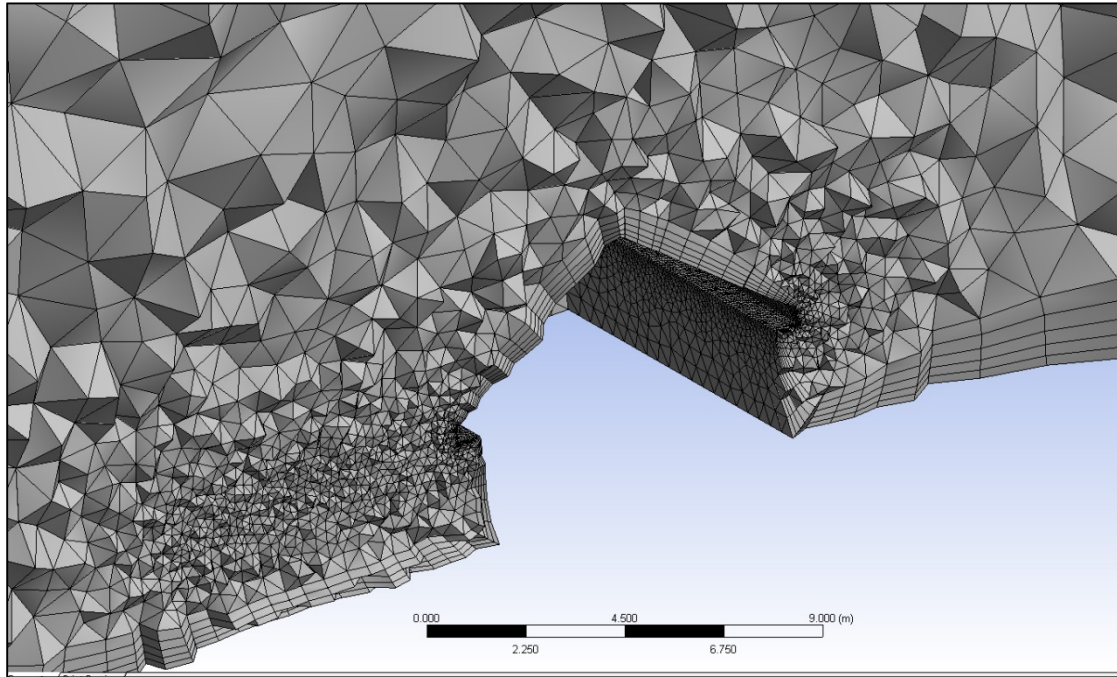
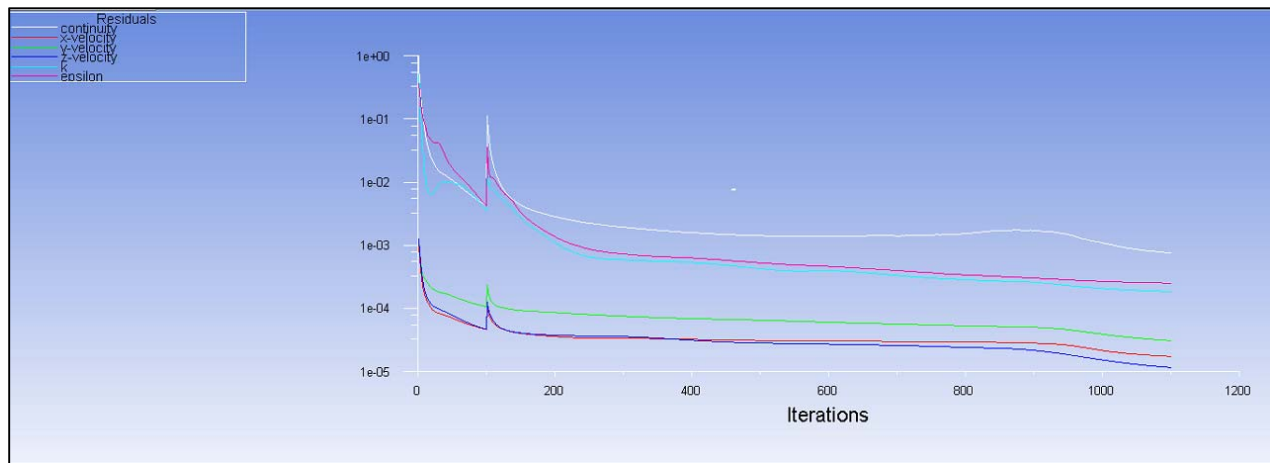Figure 4.2.3: Close-up cross section showing high density cells around the focused areas (ANSYS Fluent).

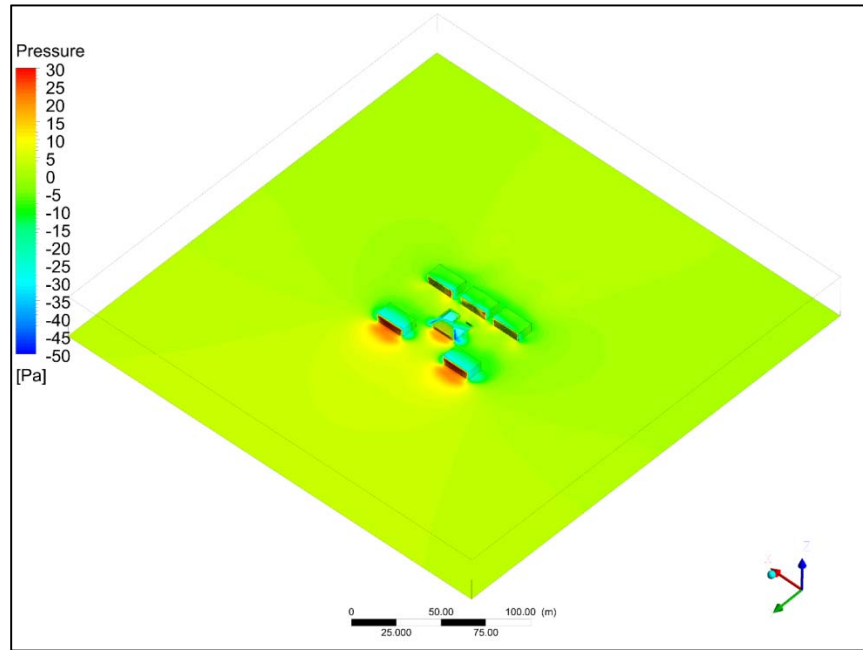Figure 4.2.4: Convergence plot (ANSYS Fluent).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 208

Figure 4.2.5: Pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).
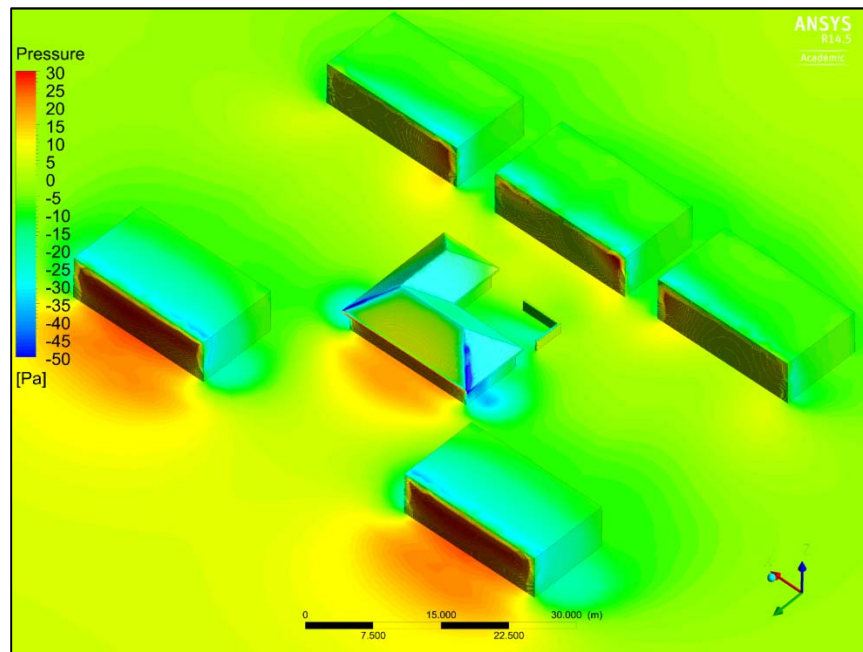
Figure 4.2.6: Close-up image of pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).
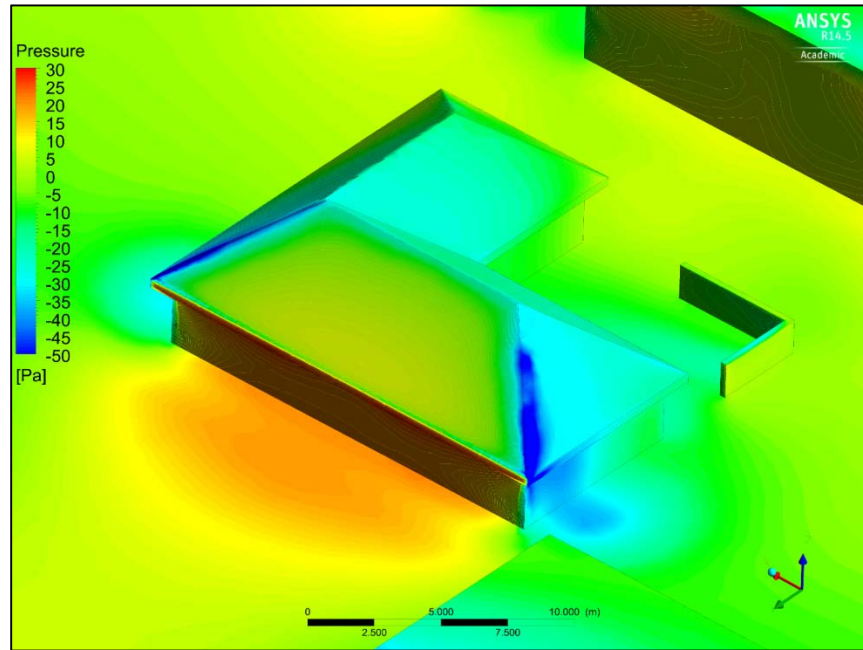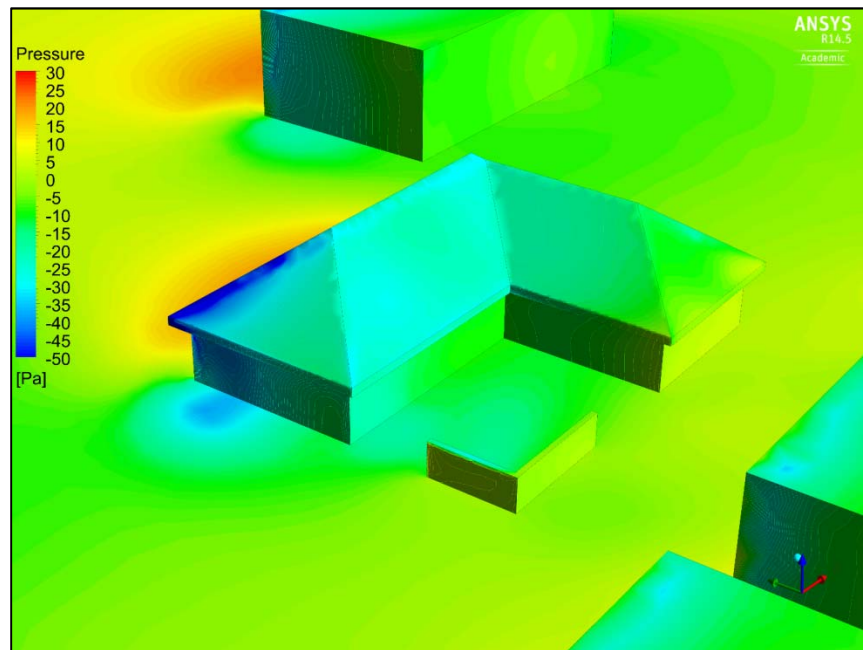
Figure 4.2.7: Close-up image of pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).
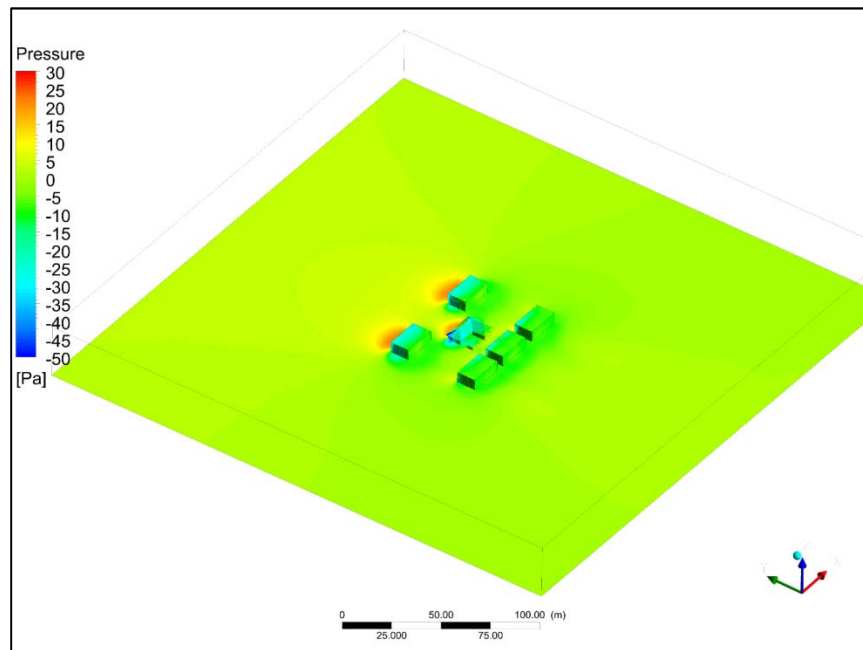
Figure 4.2.8: Pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).
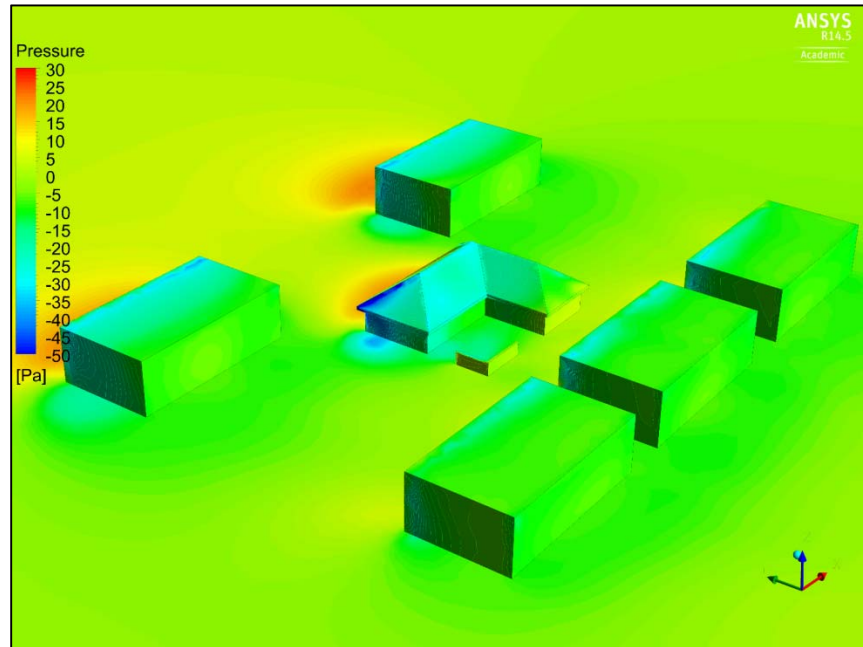
Figure 4.2.9: Close-up image of pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).

Figure 4.2.10: Close-up image of pressure (Pa) distribution on the ground and building surfaces (ANSYS Fluent).
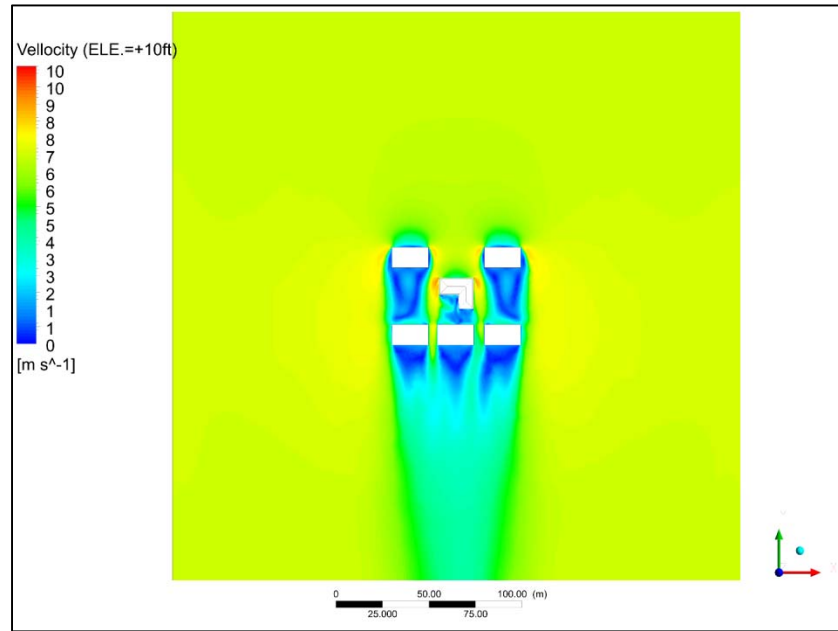
Figure 4.2.11: Velocity (m/s) contour map at the plane 10 feet above the ground (ANSYS Fluent).
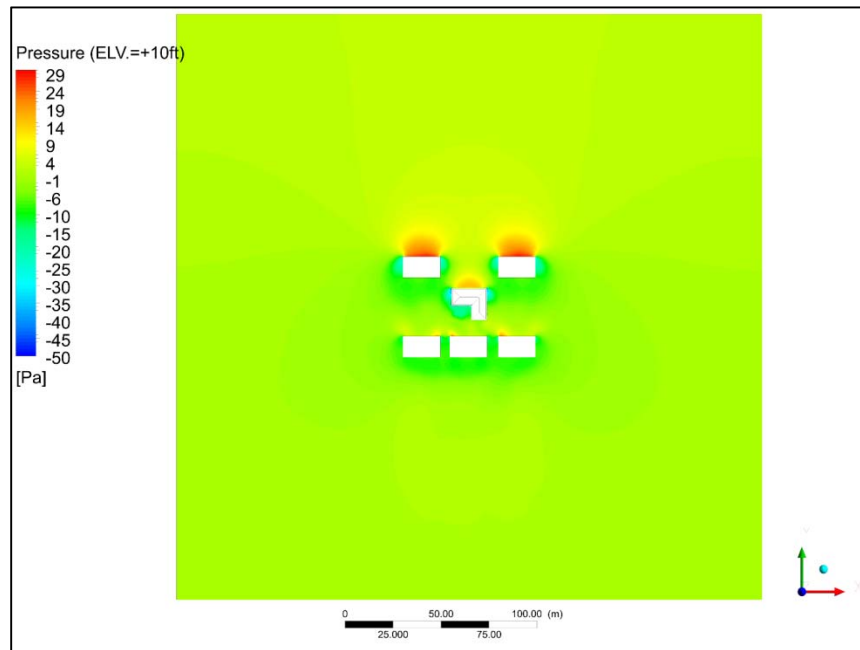
Figure 4.2.12: Pressure (Pa) contour map at the plane 10 feet above the ground (ANSYS Fluent).
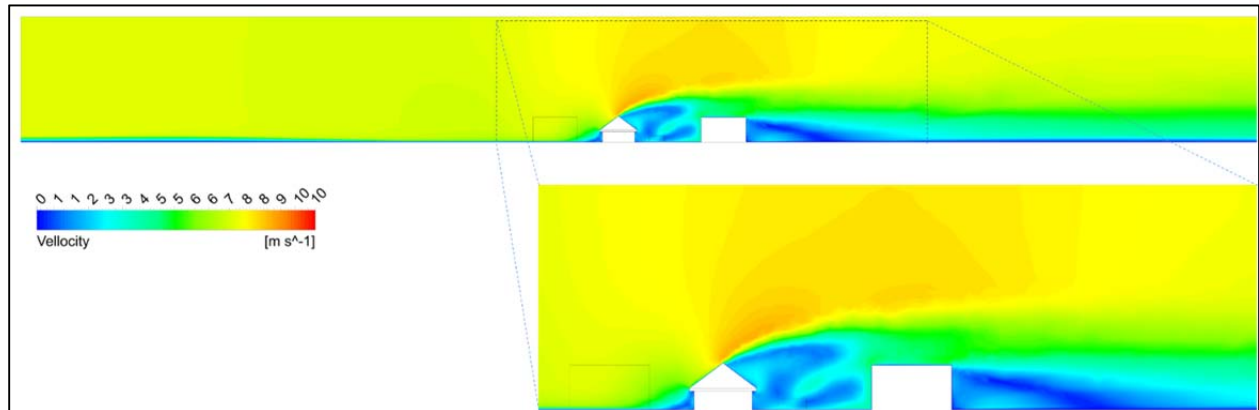
Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                         School of Architecture, University of Hawaii

December 4, 2013   Page 212

Figure 4.2.13: Wind velocity (m/s) contour map at the cross section (ANSYS Fluent)
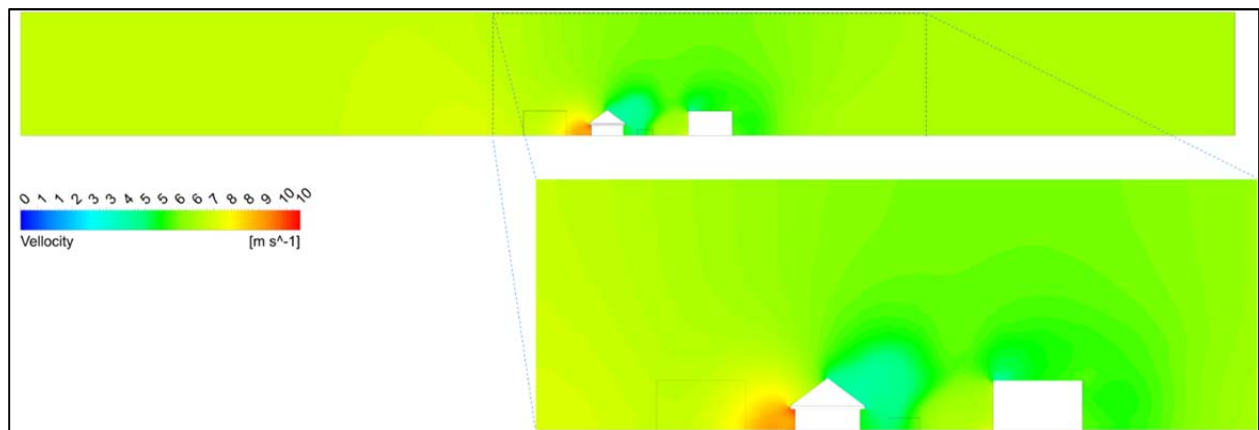
Figure 4.2.14: Pressure (Pa) contour map at the cross section (ANSYS Fluent)
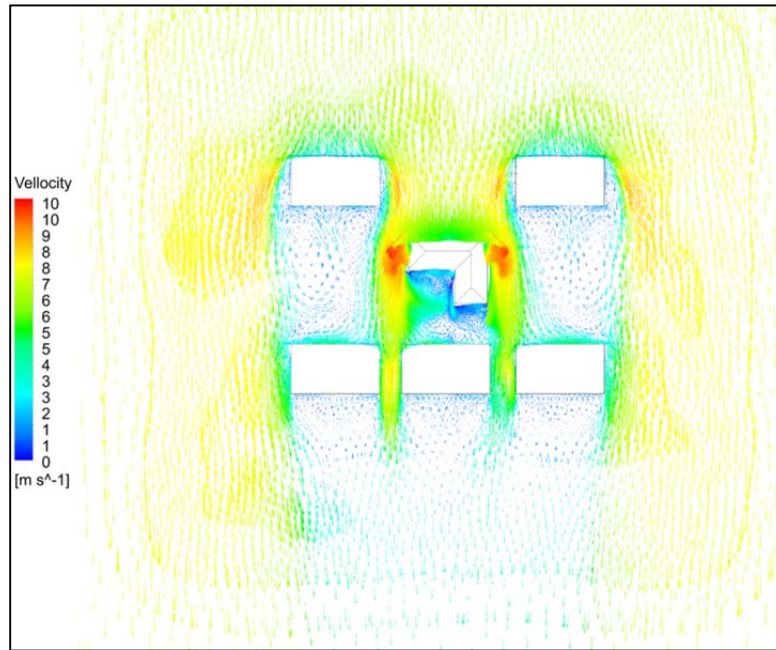
Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 213

Figure 4.2.15: Wind velocity (m/s) vector map at the plane 10 feet above the ground (ANSYS Fluent)
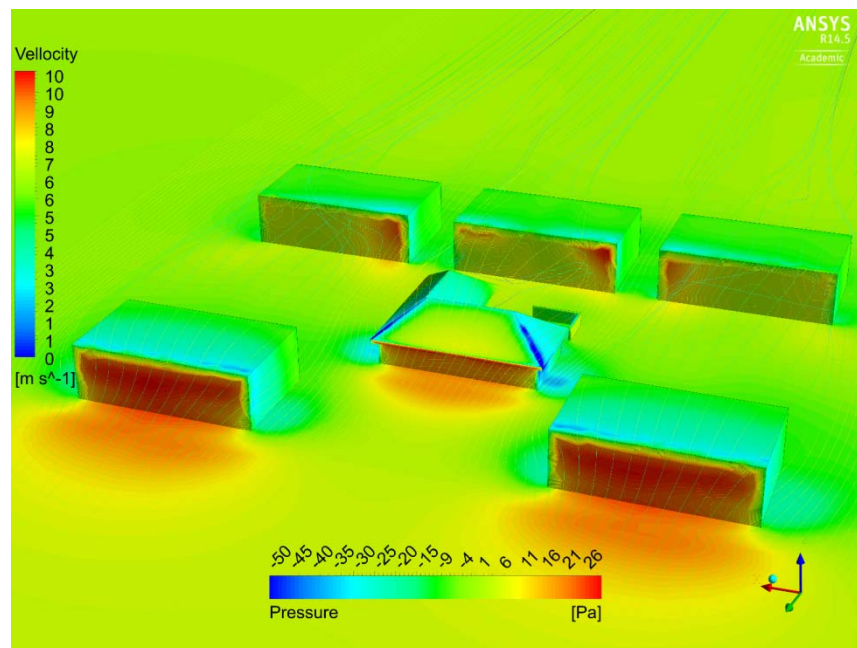
Figure 4.2.16: Pressure (Pa) distribution and wind velocity (m/s) streamlines (ANSYS Fluent)

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 214

4.3 Benchmarking the External CFD Workflow of Cd Adapco STAR-CCM+

This section presents the results and conclusions of the benchmark testing of external CFD for the software Cd Adapco STAR-CCM+ (Star-CCM+). This section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These eleven criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

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

   The GUI of Star-CCM+ 's GUI (Figure 4.3.1) is a java script interface which can load fast, use little computation resources and provide high resolution. The GUI is designed as a drop down browser menu that has a clean and simple interface and there is are unnecessary and distracting pop-up menus or windows. The drawback of the GUI, which is a significant shortcoming for new users, is that fact that the interface does not offer a clear work flow patterns. This means that the GUi does not offer a guided procession through the process of parameter input, which means the GUI lacks an indication of what are successive steps in the process. Without the guidance of the graphic pattern the user needs to know where to start and end in the workflow, and also where to input all valuables. The CFD team found it a bit difficult at the start to understand the advanced CFD workflow of Star-CCM+. Different from the other CFD software products Star-CCM+ requires the user to have an advanced understanding of CFD, which means the user is required to study user guide documents for better understanding of the workflow.

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

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

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

Star-CCM+ has an outstanding mesh generation tool. Meshes created in Star-CCM+ can be adjusted in regard to resolution control. The software can adjust grid resolution in the meshing process and generate an effective meshing structure in areas of interest. Setting up the mesh refinement is done by selecting a scaling parameter between 1 and 2, although there are some other parameters that need to be specified, such as target area or ratio (Figure 4.3.2).

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

Since the Star-CCM+ user interface and the way of navigating to any input factors and parameters are non-sequential within the menu browser (Figure 4.3.4), the user needs to know all necessary set-up details to avoid missing parameters and scripting steps. The mesh generation process is straight forward by first creating the surface and then the volume mesh. but successful meshing requires a closed surface. If the surface is defective, e.g. there are "holes" in the surface, effective repair surface functions can be used, such as the "Surface Wrapper". For complex model details, high mesh resolution, and targets with high details, the mesh set-up may require longer time to generate a quality surface mesh. There are different mesh structures and cell geometries which required a proper surface mesher that depends on the geometry type, but the most advanced is the polyhedral mesh (Figure 4.3.2).

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

Star-CCM+ uses all standard boundary condition layer types (Figure 4.3.3). The boundary conditions and domain size needed inside the software are significantly affected by the surface mesh, therefore surface extraction from the imported CAD geometry and possible repair of the surface is the key factor in STAR-CCM+. The software features a complex set-up of physics valuables which needs to be defined in the simulation set-up. The user needs a good understanding of the type of physical properties of investigated in the CFD analysis. The complexity of possible input parameters is both an advantage and a disadvantage of the software. An advantage is created since the software can simulate a broad array of fluid and thermal system phenomena. A challenge is created for the user since there is no "script" available in the software to guide the user though (industry) specific CFD analysis; such as external wind movement around buildings.

As an advantage to using CFD for external wind movements around buildings, the software offers a broad range of turbulence models (Figure 4.3. 5 and Figure 4.3.6).

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

Convergence performance of simulations was good throughout the benchmarking tests. Figure 7 shows a typical distribution of residuals over first 1000 iterations of the simulation. The fact that good convergence has been attained might be due to the somewhat simple configuration of the building model. Time required for the simulation could be an issue in the case of high grid resolution and small aspect ratio (small blockage ratio). Simulation were re-run, which required clearing prior simulation result after any modifications.

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

The simulation performance was good. Most simulations took between three and four 3-4 hours to accomplish the first 1000 iterations.

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

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

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

STAR-CCM+ features very good post processing tools both in regard to visualization and quantification. The simulation visualization is able to produce a range of visualization options with and without contour lines (Figure 4.3. 9 through Figure 4.3.12). The highest resolution of output image is 4000 x 3000 pixels (figure 8). Alphanumeric data of simulation results (such as velocity and pressure) can be obtained for a specific point, for lines and areas. This data can be exported to spreadsheets or as ACS characters.

When using the streamline visualization option "line integral convolution" in high resolution (Figure 4.3.13) runtime problems and in some cases file crashing occurred. The CFD team attributes this to hardware limitations and the team anticipates that these problems can be avoided when the software is installed on the newly acquired computer in the CFD lab. Overall the most useful tool for the CFD team has been as pressure distribution on building surface, pressure and velocity probe slice plane, velocity streamline, and velocity vector slice plane. These visualization techniques are very useful to depict distribution of wind occurrences and pressures on and around the building envelope (Figure 4.3.14).

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

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

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

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

The complexity of the software and the lack of a guided sequences in setting up the simulation and entering all required parameters requires a comprehensive understanding of user guides and specific descriptions for modeling, surface repair, and other set-up functions for the simulation. Star-CCM+ provides all required documents were which are very helpful and give the user sufficient guidance to set-up, run and post-process building simulations.

Figure 4.3.1 through Figure 4.3.14 are presented below:

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 218

Figure 4.3.1: Star-CCM+ graphic user interface (GUI). (the image on the right depicts the geometry of the benchmarking test)

Figure 4.3.2: Star-CCM+ shows mesh structure shows target area and refinement level around the buildings of interest
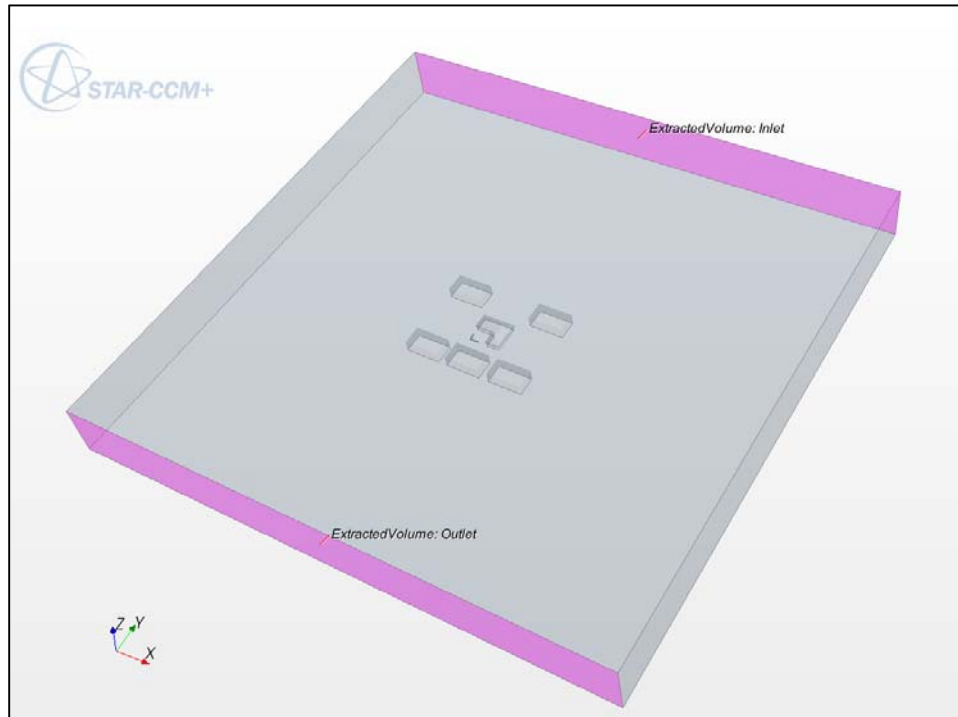
Figure 4.3.3: Star-CCM+ shows the assigned boundary conditions and layer types
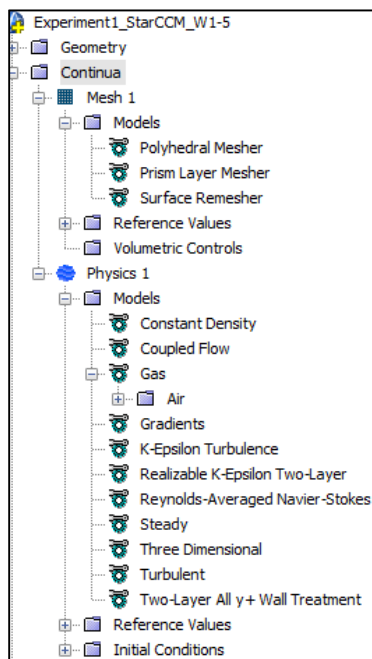
Figure 4.3.4: Star-CCM+ shows a continua functions for mesh model and physics drop down menu set up
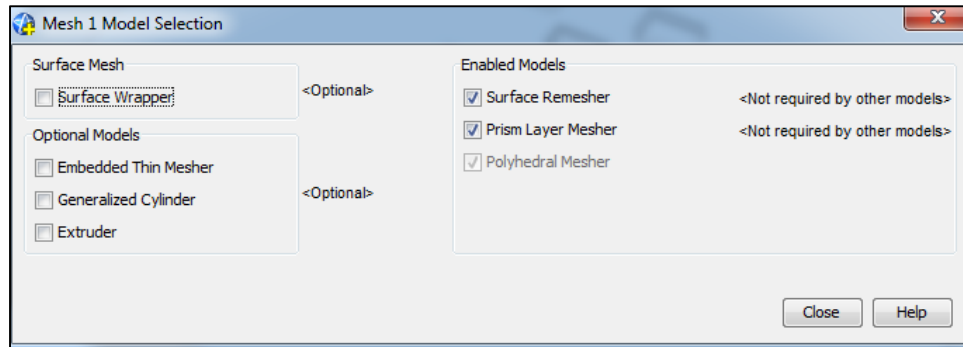
Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                           School of Architecture, University of Hawaii

December 4, 2013   Page 220

Figure 4.3.5: Star-CCM+ shows mesh model set up and selection
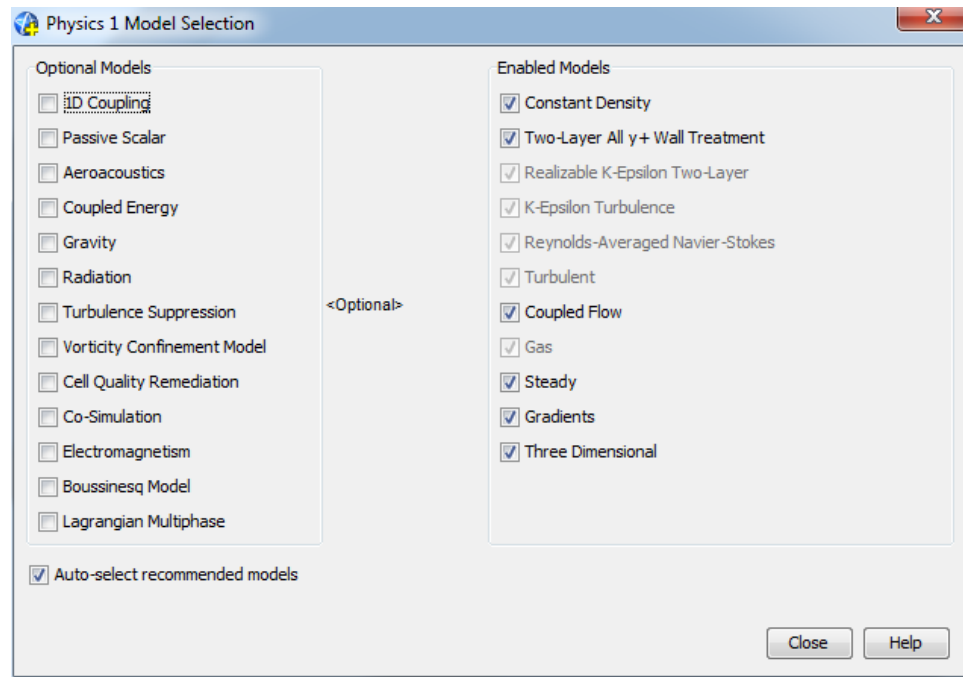

Figure 4.3.6: Star-CCM+ shows physics set up and selection
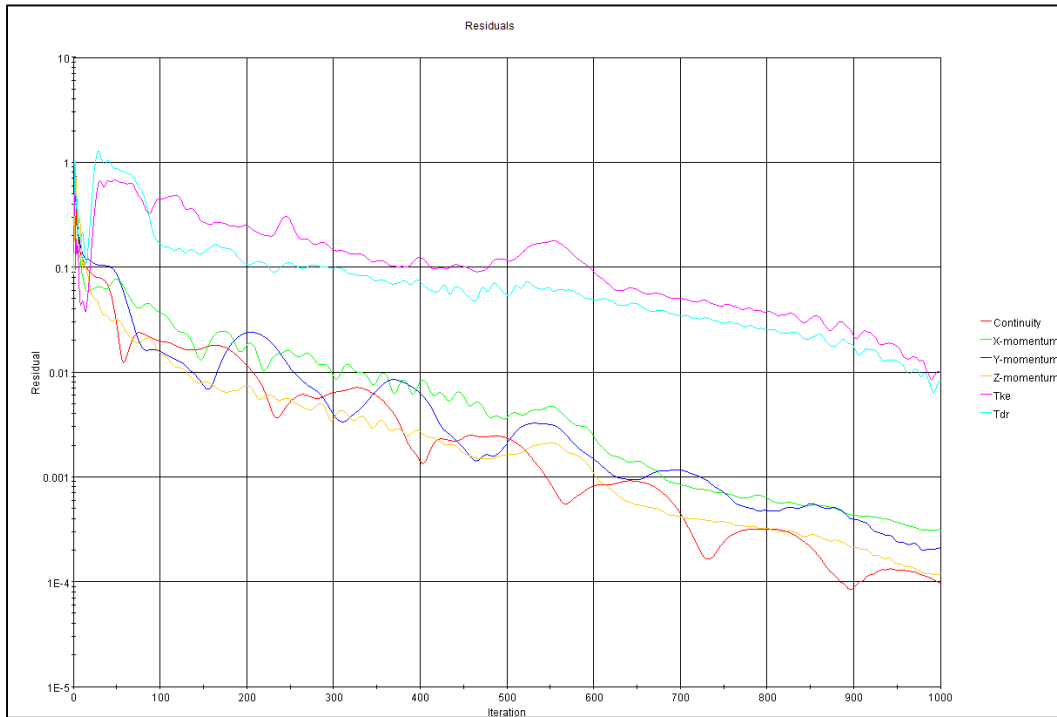
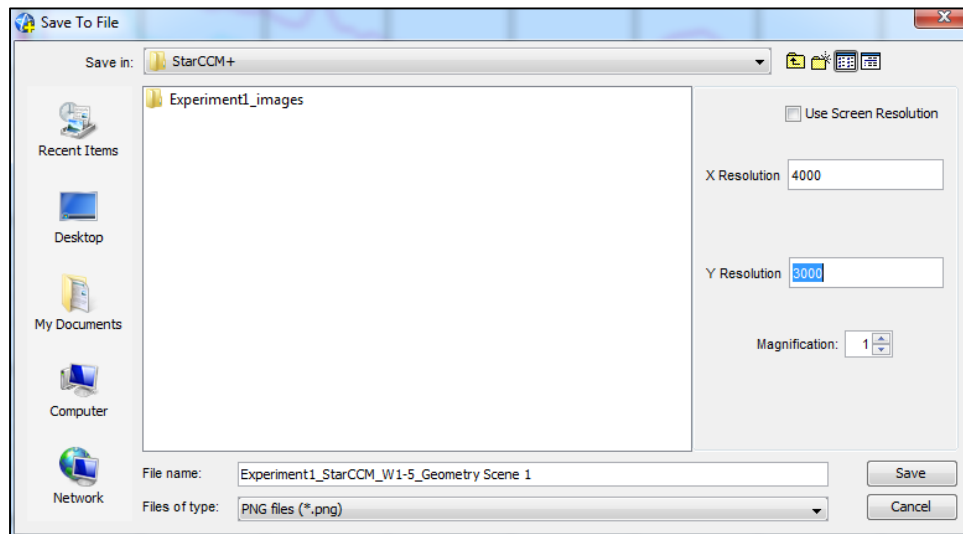Figure 4.3.7: Star-CCM+ residuals report depicts the convergence in the tests

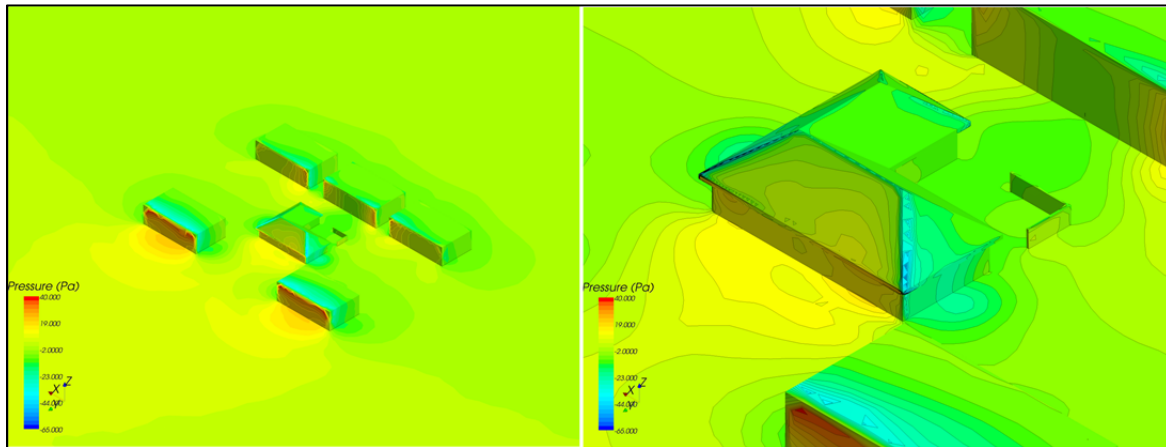Figure 4.3.8: Star-CCM+ shows saving output file format and resolution

Figure 4.3.9: Star-CCM+ shows the external CFD simulation result of pressure (Pa) distribution on building faces.
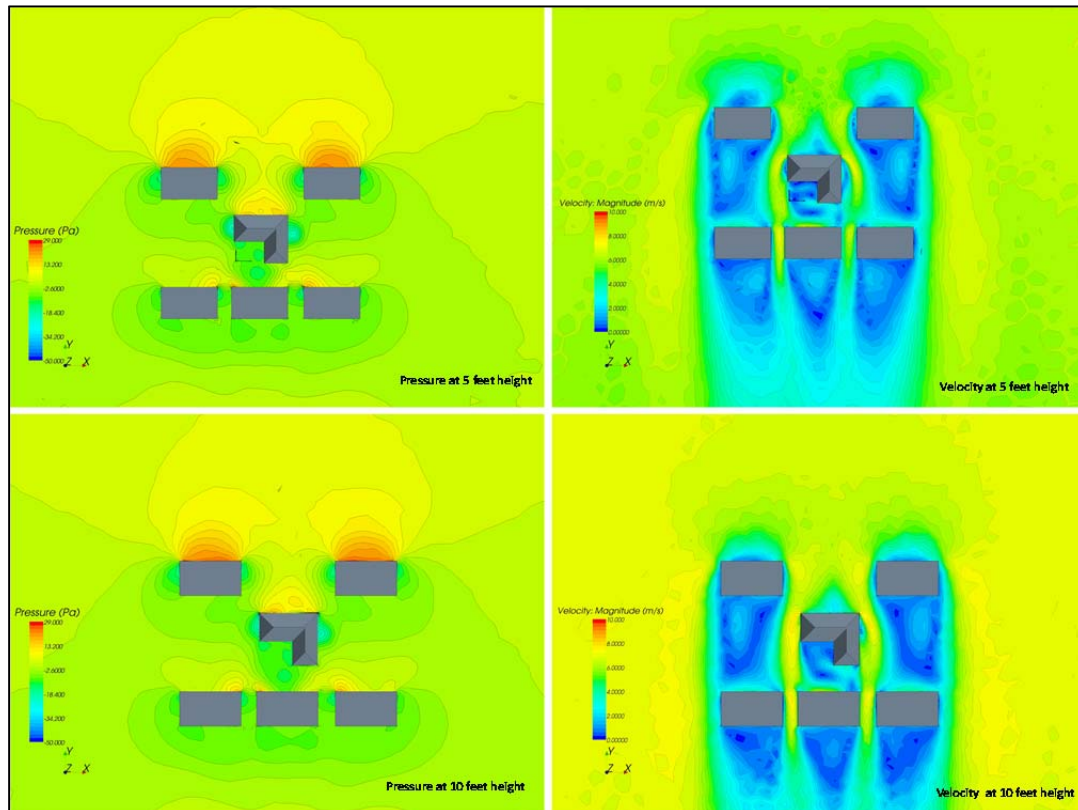

Figure 4.3.10:   Star-CCM+ shows the external CFD simulation result of pressure (Pa) and velocity (m/s) at the different slice plane height levels.
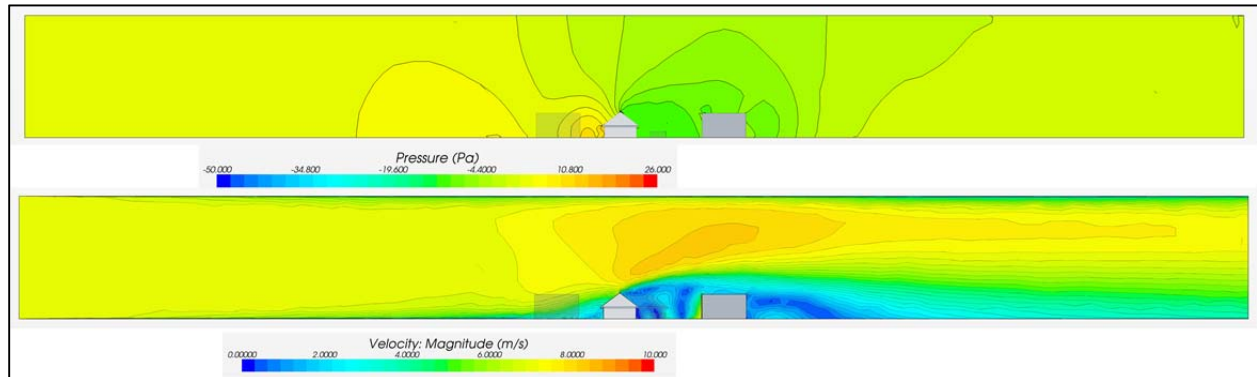
Figure 4.3.11:   Star-CCM+ shows the external CFD simulation result of pressure (Pa) and velocity (m/s) at the cross section plane.
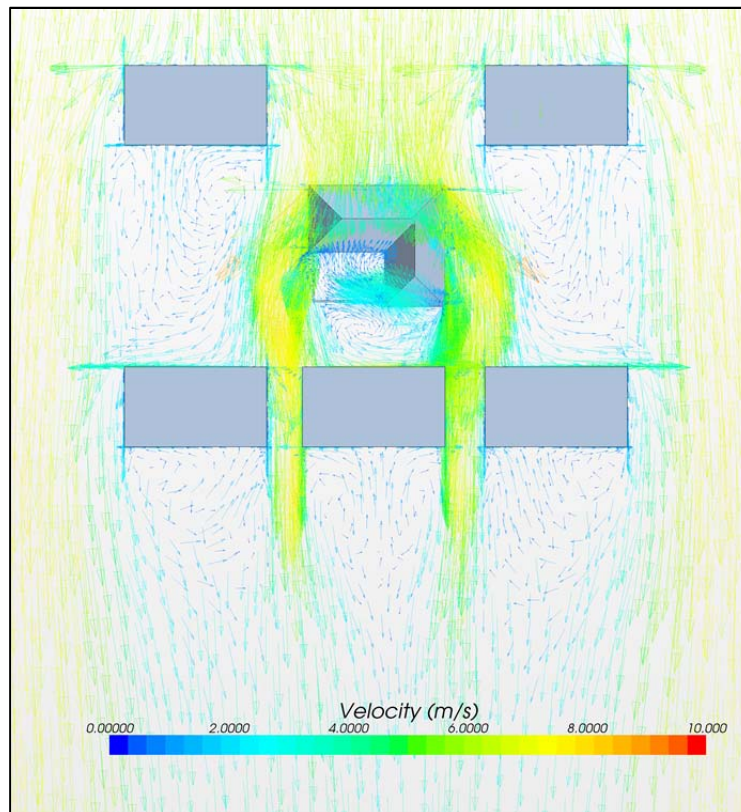

Figure 4.3.12:   Star-CCM+ shows the results of external CFD simulation with wind vector streamline at 10 feet height
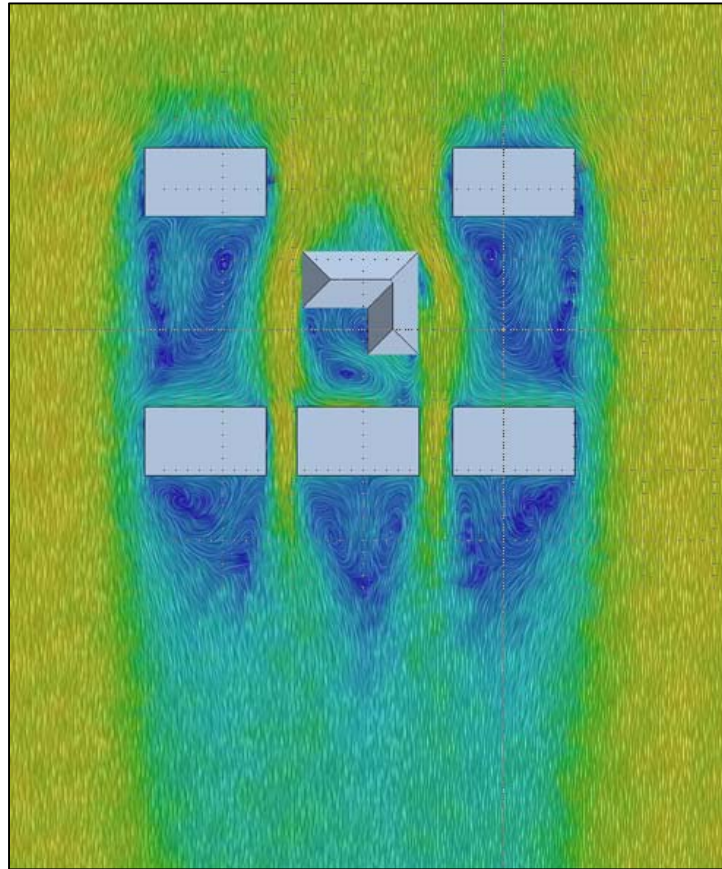
Figure 4.3.13:   Star-CCM+ shows the external CFD simulation of streamline options in high resolution of line integral convolution.
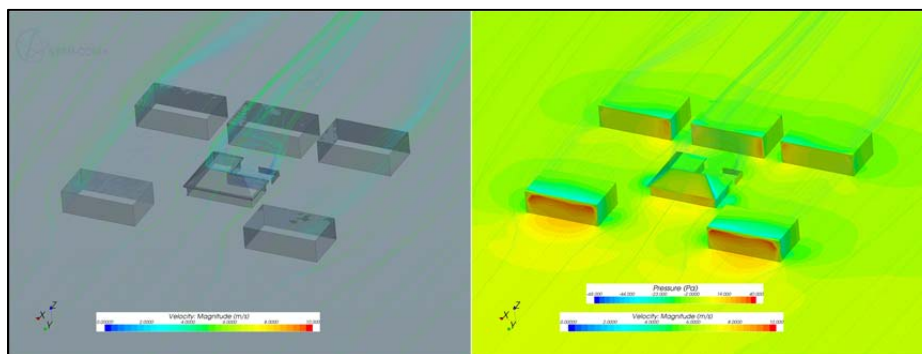

Figure 4.3.14:   Star-CCM+ shows the external CFD simulation of velocity streamline (m's) and pressure distribution on building faced (Ps)

4.4 Benchmarking the External CFD Workflow of Autodesk Simulation CFD

This section presents the results and conclusions of the benchmark testing of external CFD for the software Autodesk Simulation (ACAD CFD). This section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These eleven criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

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

    Autodesk Simulation CFD has a simple GUI (Figure 4.4.1) which reduces a complexity of simulation workflow for Autodesk Simulation CFD. The simplified GUI is further supported by a comprehensive help function and the GUI guides the user through the simulation process. The GUI guides the user through the process of navigation through the program, creating views, defining models, and building geometries. A major limitation of the software is that it doesn't have an internal function for the creation of geometries. The software overcomes this limitation by providing easy import and export functions that connect the CFD program with a range of Autodesk programs. In regard to creating geometry Autodesk Simulation CFD can import 3D-CAD geometries from any other Autodesk software, for instance Revit Architecture or Inventor Professional software.

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

    There are no modeling tools inside Autodesk Simulation CFD. Therefore third party modeling software tools or, preferably, Autodesk software products are required for 3D-geometry generation. One particular advantage of Autodesk Simulation CFD is the ability launch Revit Architecture or Inventor professional from within the CFD program. This facilitates the overall workflow of the CFD program (see Figure 4.4.2).

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

    The mesh generated by Autodesk Simulation CFD primarily consists of tetrahedral cells. In two dimensional models, elements are triangles (Figure 4.4.3). Geometry created with Autodesk CAD software maintains a good model quality upon import function. There is an automatic mesh repair function which is user friendly (Figure 4.4.4). The automatic repair has limitations and works best if the initial CAD model is of good quality. If the mesh has significant defects manually fixing the mesh surface would be required, which can be a time consuming process.

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 226

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

   The CFD team found a simple mesh generating set up (Figure 4.4.5) to be an easy and intuitive function, which results in effective and fast meshing. However, the quality of the mesh surface can be questionable.

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

   Since Autodesk Simulation CFD does not provide internal modeling tool CAD, geometry needs to be imported from either Revit architecture or Inventor professional. The CFD program requires defining the computational domain from within the Autodesk Simulation CFD. Autodesk Simulation CFD uses a finite element model which requires the definition of both physics (Figure 4.4.6) and materials (Figure 4.4.). Turbulence model is defaulted to K-epsilon (Figure 4.4.8). The physics properties of the models are limited to turbulence, roughness, air density, and etc.

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

   Defining the convergence criteria of the simulation (Figure 4.4.9) requires the selection for the number of iterations; whereby higher number of iterations usually increases the accuracy level of the results. For the test case 500 – 1,000 iterations were selected for the simulation. The solving user interface allows pausing and resuming the simulations, and also provide different design scenarios from different physics set up.

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

   The CFD team encountered several run-time difficulties. Simulation generally took about 2 hours for 5000 iterations.

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

   Solver is a simple set up (Figure 4.4.8). No problems were encountered unless scripting steps or missing materials were wrong.

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

   Autodesk Simulation CFD provides good visualization results (Figure 4.4.10 – Figure 4.4.12) in multiple ways, including graphically-rich images, plots and tabular data files. The visualization environment is available both during and after the simulation process. The simulation can be improved and expanded by using the Autodesk 360 view which can crate rich and (near) photo-

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 227

realistic images coupled with CFD results. A drawback is axis icon interfere in the visualization output. Autodesk Simulation CFD provides a several options of image file type formats. Unfortunately there is no option for adjusting image size and resolution.

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

Over all, Autodesk Simulation CFD provides an effective significant modelling integration with BIM software and other Autodesk software products. The CFD program guides user through successive steps of the modeling and simulation process. A very useful software function is the real time export or launch capability for model geometry creation and editing directly from the program. For example, Revit architecture can be automatic launched from within Autodesk Simulation CFD. The GUI of Autodesk Simulation CFD is intuitive and similar to all Autodesk software products. Most of the functions are easy to use and the GUI provides a productive interface to guide user through the subsequent steps of the simulation process. The simulation results provide good visualization with an easy set-up function.

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

Autodesk wiki provides some basic knowledge, trainings, tutorials, and videos available to user. Autodesk Simulation CFD help menu is very useful. Due to less complex functions, the software does not require any comprehensive user guide document. For users unfamiliar with CFD processes in general this can present a challenge.

Figures Figure 4.4.1 through Figure 4.4.12 are shown below:

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 228

Figure 4.4.1: Autodesk Simulation CFD graphic user interface (GUI).
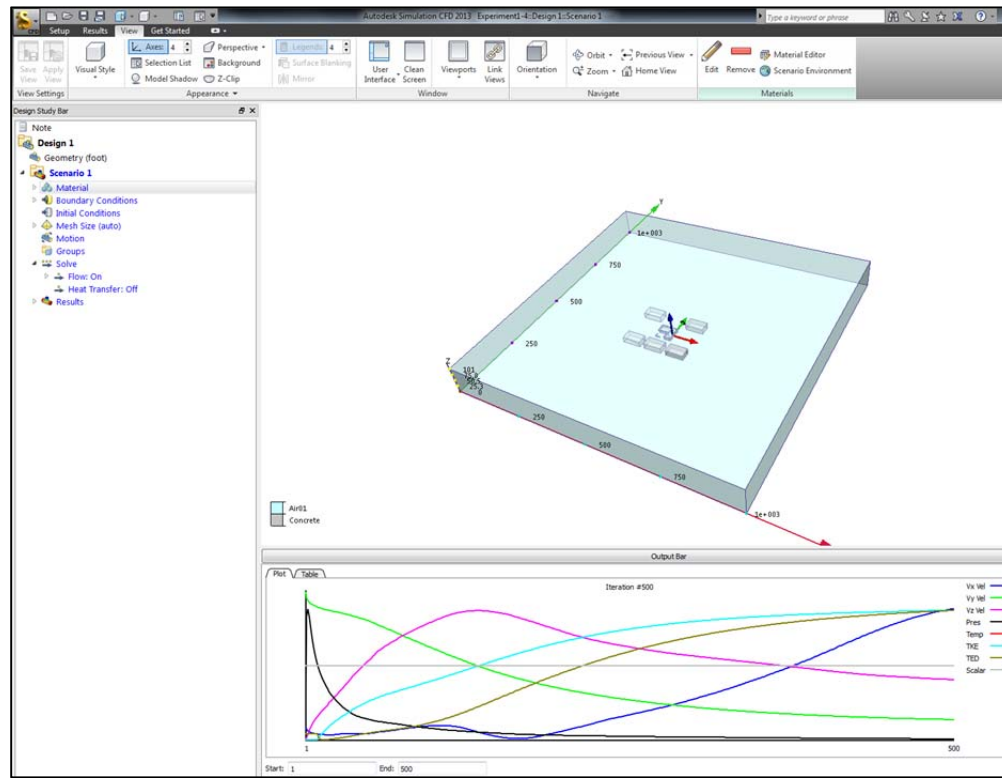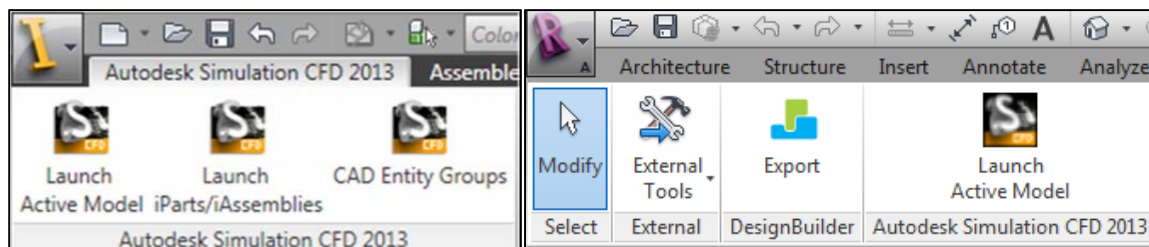

Figure 4.4.2: Autodesk Simulation CFD shows launching from CAD based model in both Inventory and Revit

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 229

Figure 4.4.3: Autodesk Simulation CFD shows tetrahedral mesh surface
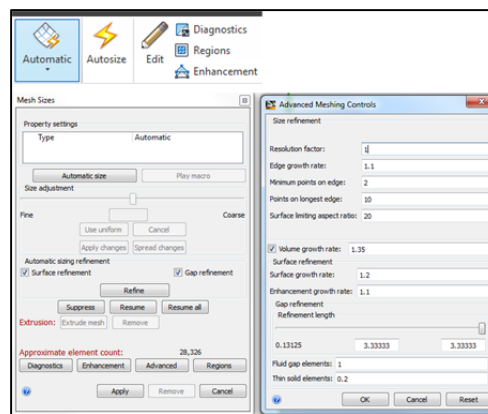
Figure 4.4.4: Autodesk Simulation CFD shows the auto and manual mesh refinement set up
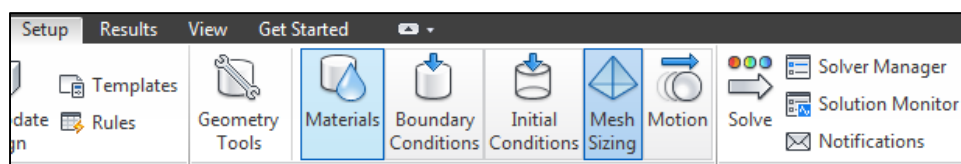
Figure 4.4.5: Autodesk Simulation CFD shows set up tool bar
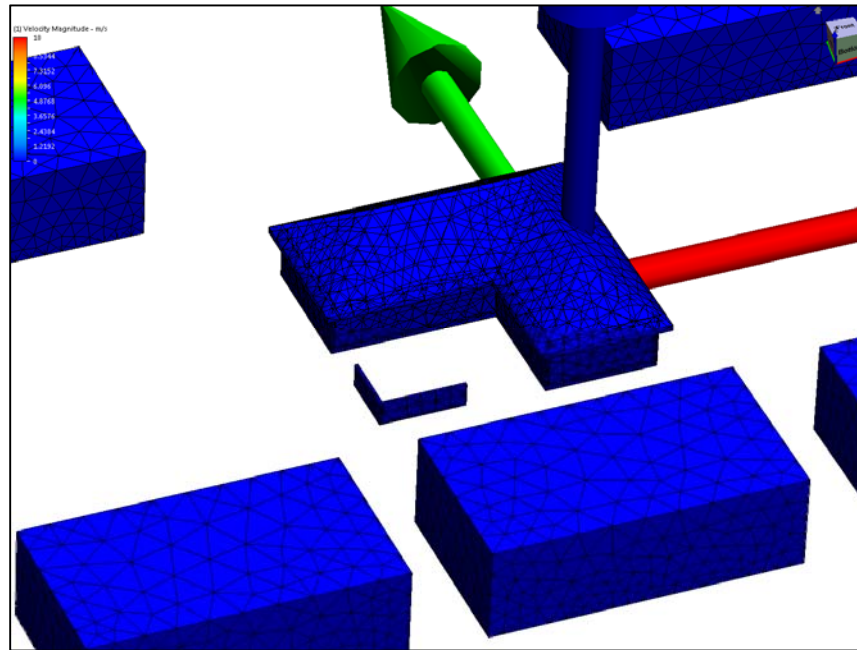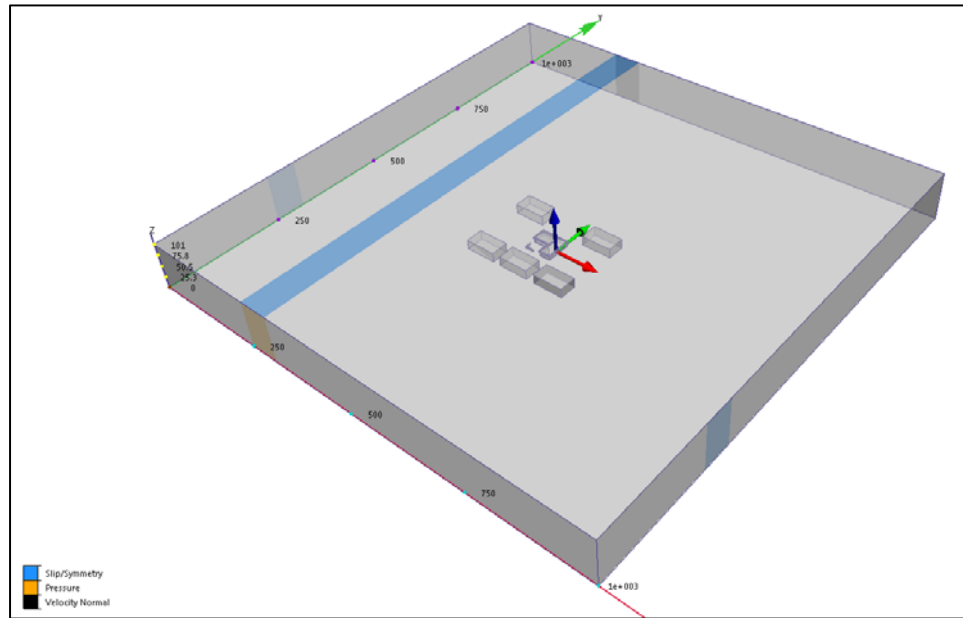
Figure 4.4.6: Autodesk Simulation CFD shows setting up boundary layer types
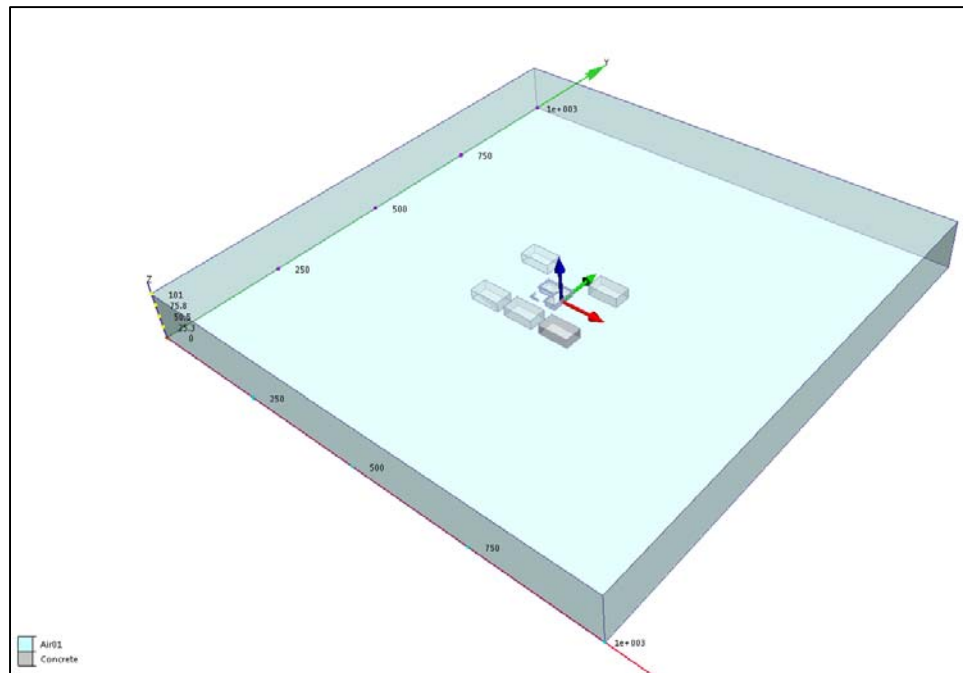
Figure 4.4.7: Autodesk Simulation CFD shows setting up basic materials

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                      School of Architecture, University of Hawaii

December 4, 2013   Page 231
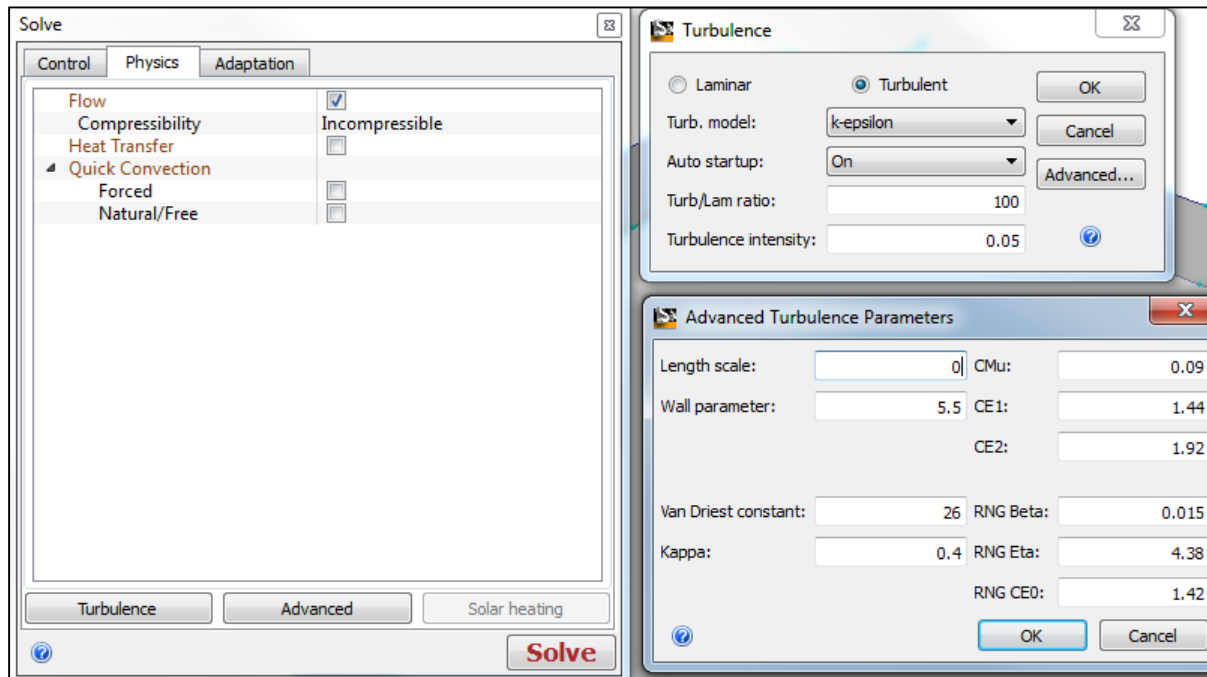
Figure 4.4.8: Autodesk Simulation CFD shows physic and turbulence set up
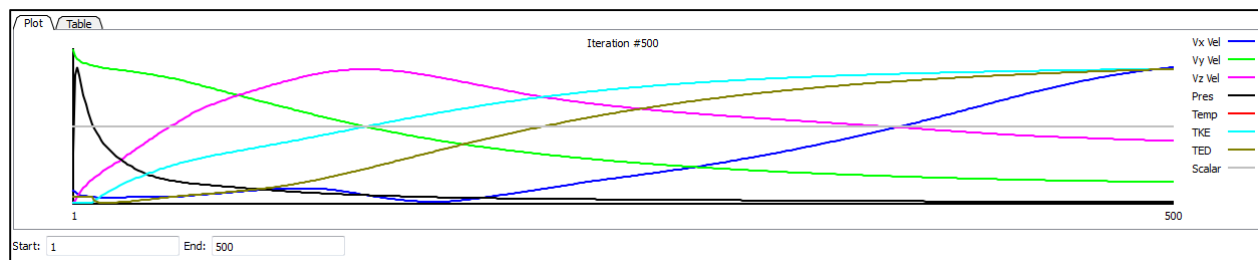
Figure 4.4.9: Autodesk Simulation CFD shows solution plot graph
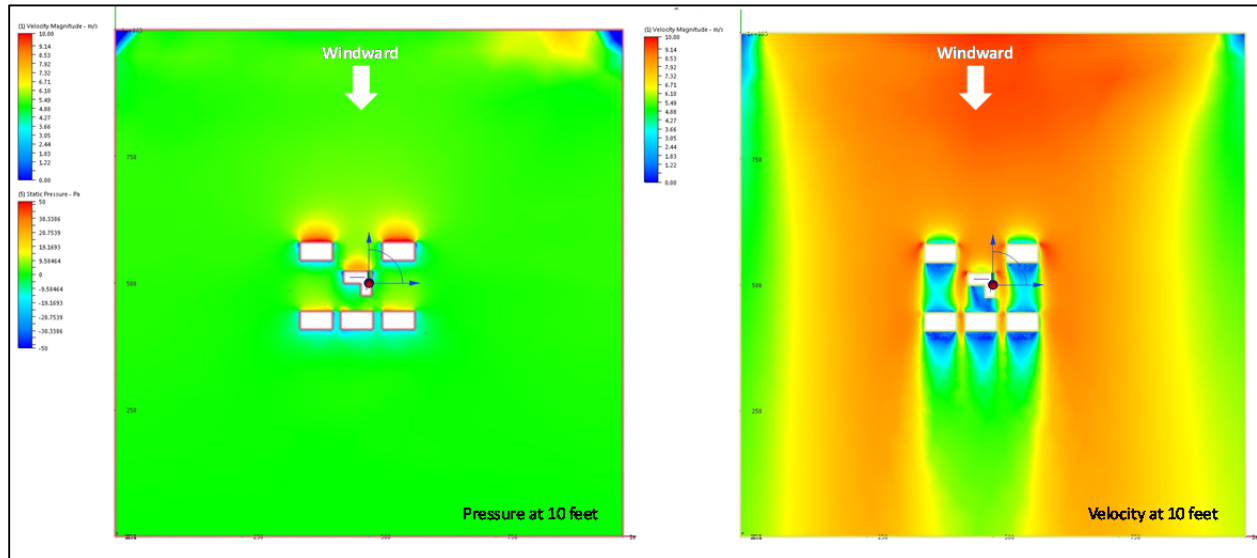
Figure 4.4.10: Autodesk Simulation CFD shows the simulation results of wind pressure (Pa) and velocity (m/s) on the section slice plane at 10 feet height.
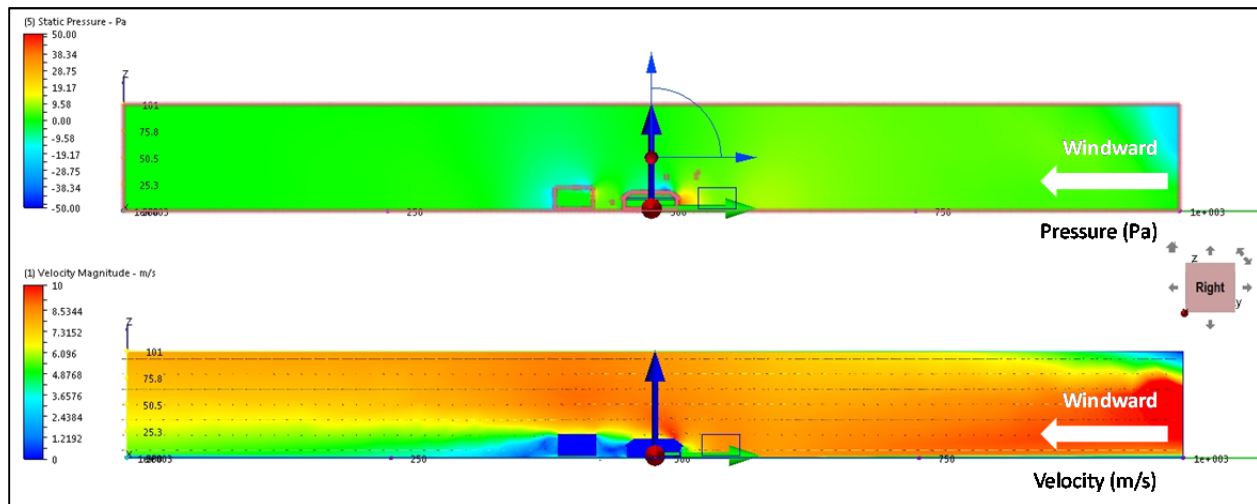
Figure 4.4.11: Autodesk Simulation CFD shows the simulation results of wind pressure (Pa) and velocity (m/s) on the cross section plane

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 233

Figure 4.4.12: Autodesk Simulation CFD shows the simulation results of wind velocity (m/s) on the tracing stream line.

4.5 Benchmarking the External CFD Workflow of Cradle scStream

This section presents the results and conclusions of the benchmark testing of external CFD for the software Cradle scStream (Stream). This section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These eleven criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

1. *When operating the CFD program what was the overall impression of the GUI (graphic user interface)*
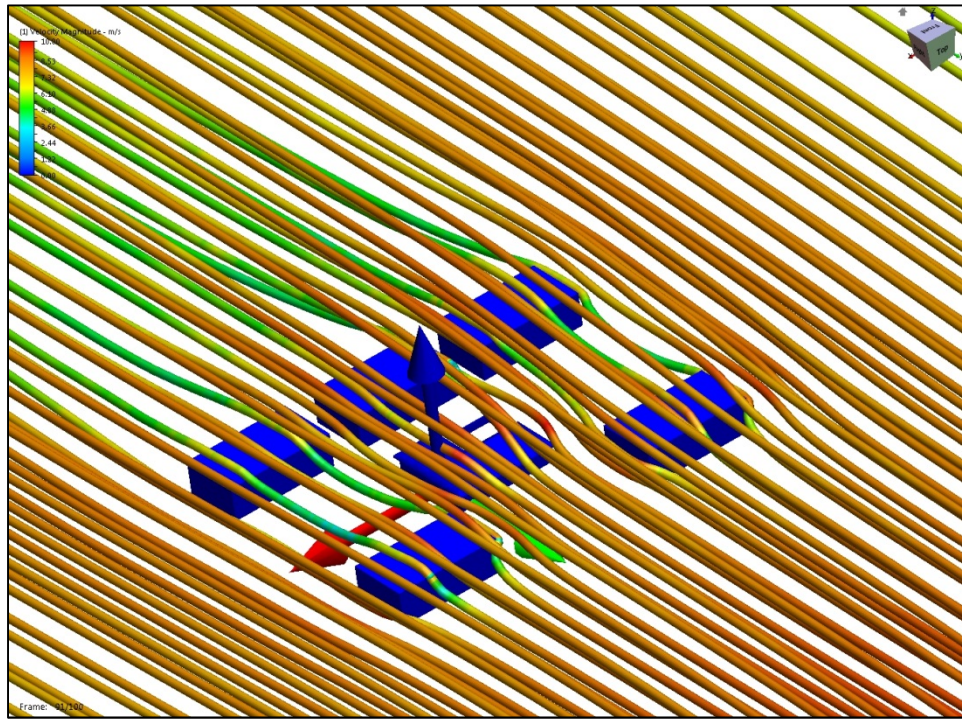
   Cradle Stream's GUI is similar to that of Cradle SC/Tetra, which allows users working on one of Cradle CFD software product to easily switch to the other Cradle software product. Cradle Stream' GUI is intuitive, providing a helpful initial wizard dialogue to guide users through necessary steps for setting up a simulation.

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

   Contrary to Cradle SC/Tetra, Stream allows to create geometric models directly inside the software. However, its modelling capabilities are not powerful enough to handle complicated geometries and therefore, importing STL format created from other CAD applications is a practical and preferred option. Since Cradle Stream is designed for external CFD, it offers a range of features facilitating the setting for external air boundary conditions.

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

   Since Cradle Stream uses a structured grid the meshing process in SC/Stream is significantly different from that of SC/Tetra. However, SC/Stream offers multi-blocks and automated meshing generation tools, which are quite flexible and represent convenient features, allowing for better meshing refinement in the area of interest. (Figure 4.5.1 and Figure 4.5.2).

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

   A very versatile mesh generation function is the "Multiblock" meshing function, which allows for meshing refinement at the regions of the interest. However, the structured grid mesh generation of SC/Stream requires a significant number of cells when working with complex geometries and large computational domain size of external CFD simulations.

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

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 235

Boundary condition and physics properties for external CFD can be easily modeled in Cradle Stream. Similar to Cradle SC/Tetra, Cradle Stream allows users to customize physic properties and user-defined functions. Moreover, and especially useful for external CFD applications, SC/Stream provides options for setting up terrain roughness factors, customizable wind profiles for boundary layer conditions. The vendor claims that the solver for the structured grid is using an effective code which accelerates the solution and compensates somewhat the fact that the number of cells in the structured grid is larger than in a comparable unstructured grid.

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

   The solution usually converged at the 1000th iterations threshold, which was selected as the convergence criteria. However, in the test simulation the distribution of residuals showed convergence performance that was not typical. (Figure 4.5.3). In order to improve convergence remedial actions were taken, including mesh refinement and adjustments in the boundary conditions. These corrective setting options resulted in better convergence, especially to for turbulence properties.

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

   Using a structured grid, Cradle Stream needs more volume cells in comparison to unstructured grids (e.g. Cradle SC/Tetra) to represent the complex geometries within the computational domain. Therefore, Cradle Stream needed more than five hours to achieve convergence after 1000 iterations while Cradle SC/Tetra took about three hours for the same number of iterations.

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

   All the internal functions of Cradle Stream are straightforward and the GUI is very intuitive to guide users through the required steps in the workflow. The CFD team did not encounter any significant operating challenges with GUI functions.

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

   Both Cradle Stream and Cradle SC/Tetra use the same post-processor tool which is very helpful for people using both Cradle CFD software packages. Some examples of post-processing results of the test cases include contour maps showing pressure distribution on the ground and building surfaces for wind coming from the north (Figure 4.5.4, through Figure 4.5.9), contour maps showing pressure and wind velocity at the plane 10 feet above the ground (Figure 4.5.10 and Figure 4.5.11), the pressure and wind velocity on the section plane (Figure 4.5.12 and Figure 4.5.13), and the over-laid contour map of pressure and velocity streamlines (Figure 4.5.14).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                                           School of Architecture, University of Hawaii

December 4, 2013   Page 236

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

    In general, Cradle Stream is a good basic CFD code for external CFD application. It features an intuitive GUI, which empowers the user with a steep learning curve. With many built-in features for external CFD solutions, Cradle Stream does not require much time to set-up and run a CFD simulation. Optimization tools are also available in Cradle Stream. These features, however, have not been thoroughly tested during the benchmarking tests.

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

    Similar to Cradle SC/Tetra, SC/Stream allows the use to access technical documentations and tutorials directly from the menu bar located in the software's GUI (Figure 4.5.15). The CFD team very much appreciated the technical support from the vendor for help in trouble shooting some issue that arose during the tests.

Figures Figure 4.5.1 through Figure 4.5.14 are shown below:
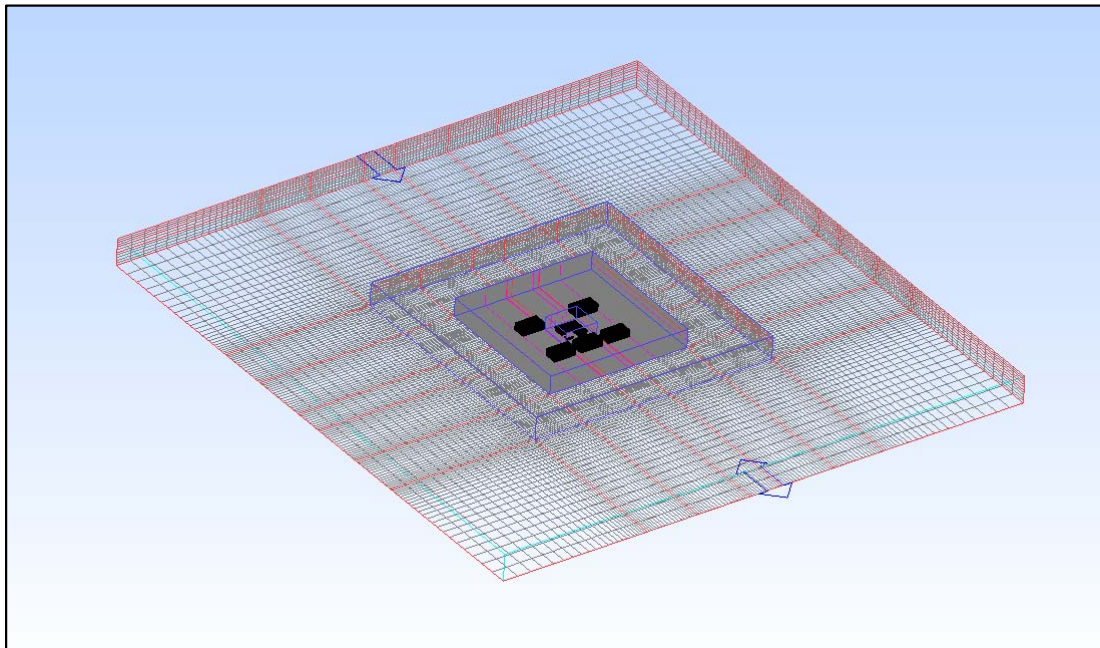
Figure 4.5.1: Multiblock meshing generation (Cradle Stream).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 237
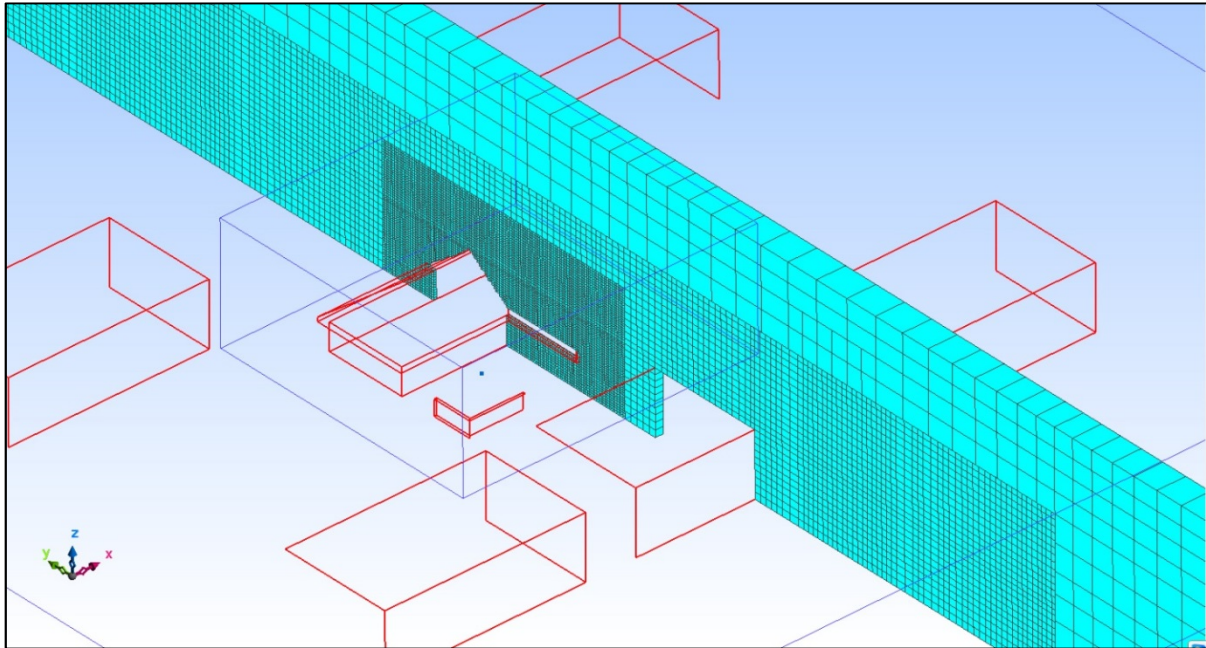
Figure 4.5.2: Cross section showing structured meshing generation (Cradle Stream).
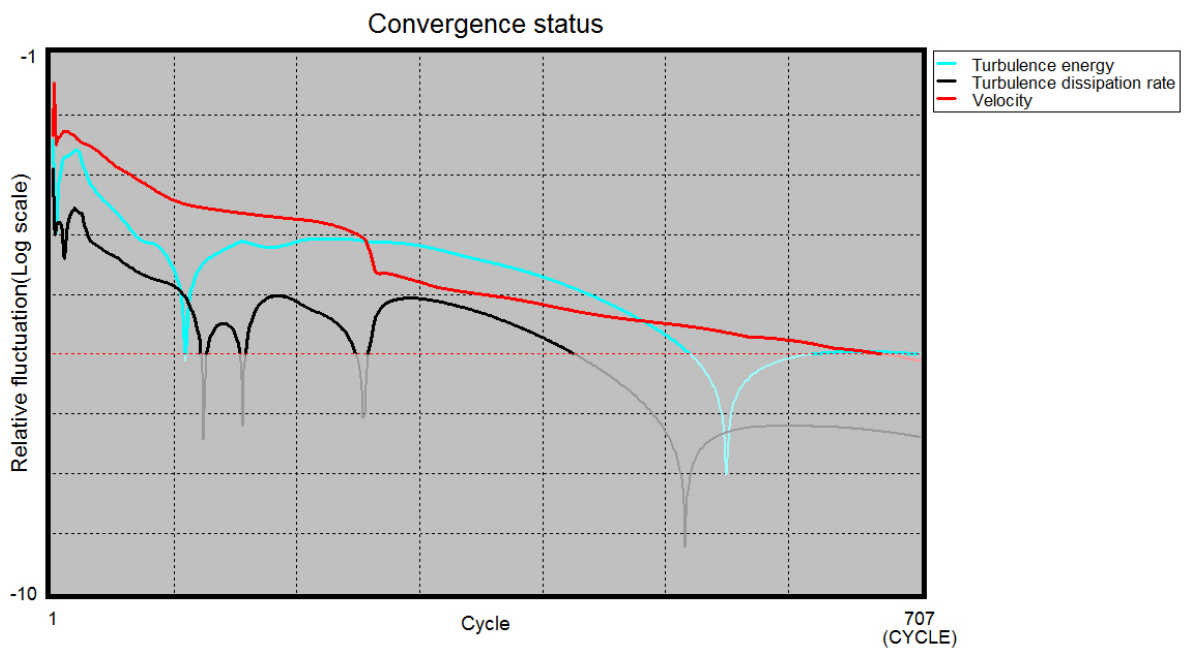
Figure 4.5.3: Residual plotting showing convergence status (Cradle Stream).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 238
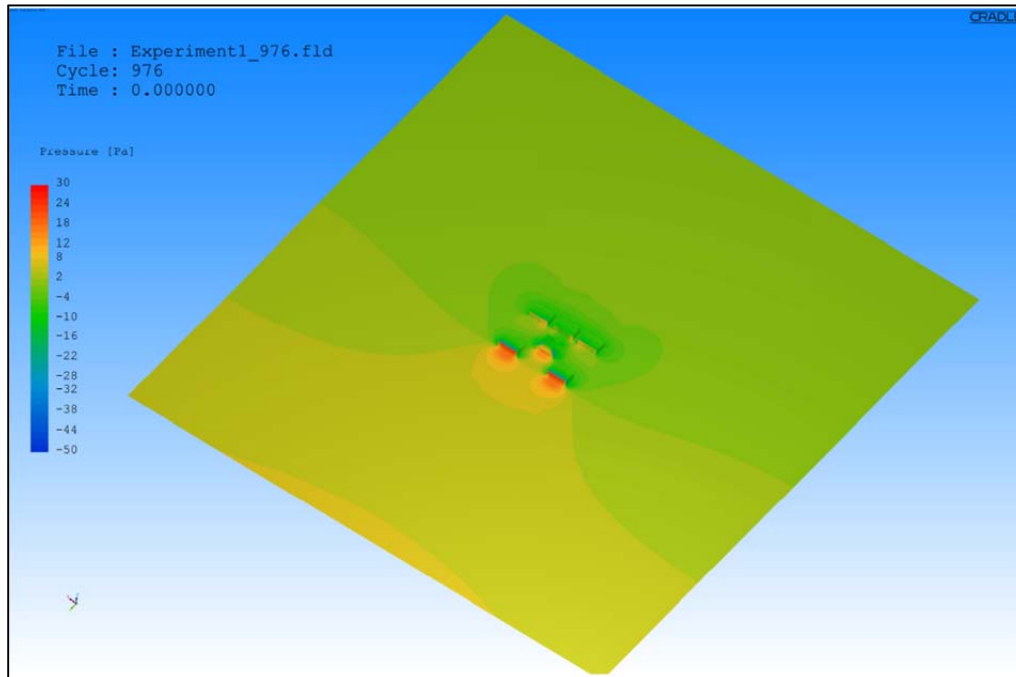
Figure 4.5.4: Pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).
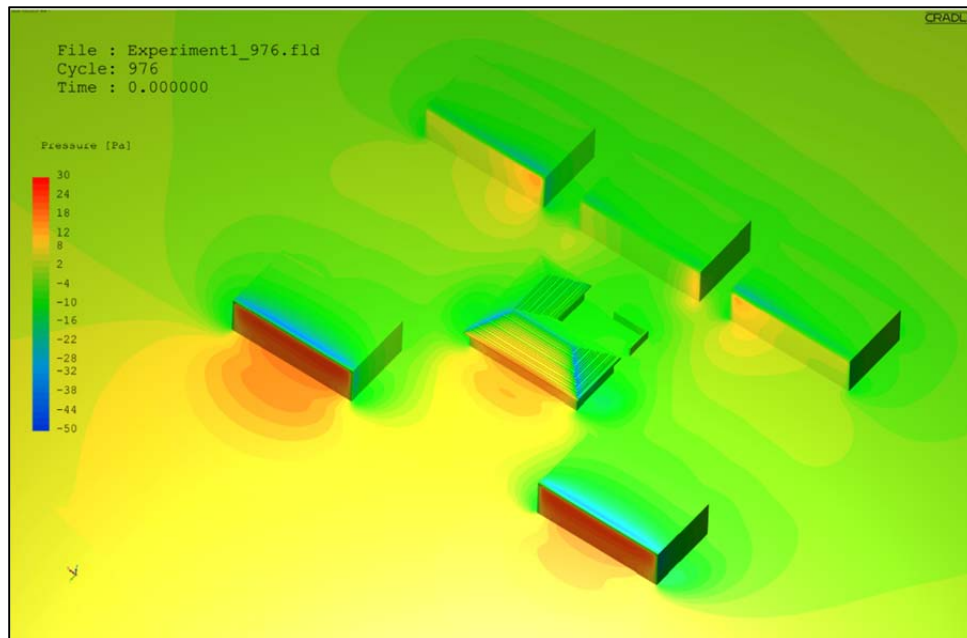
Figure 4.5.5: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).
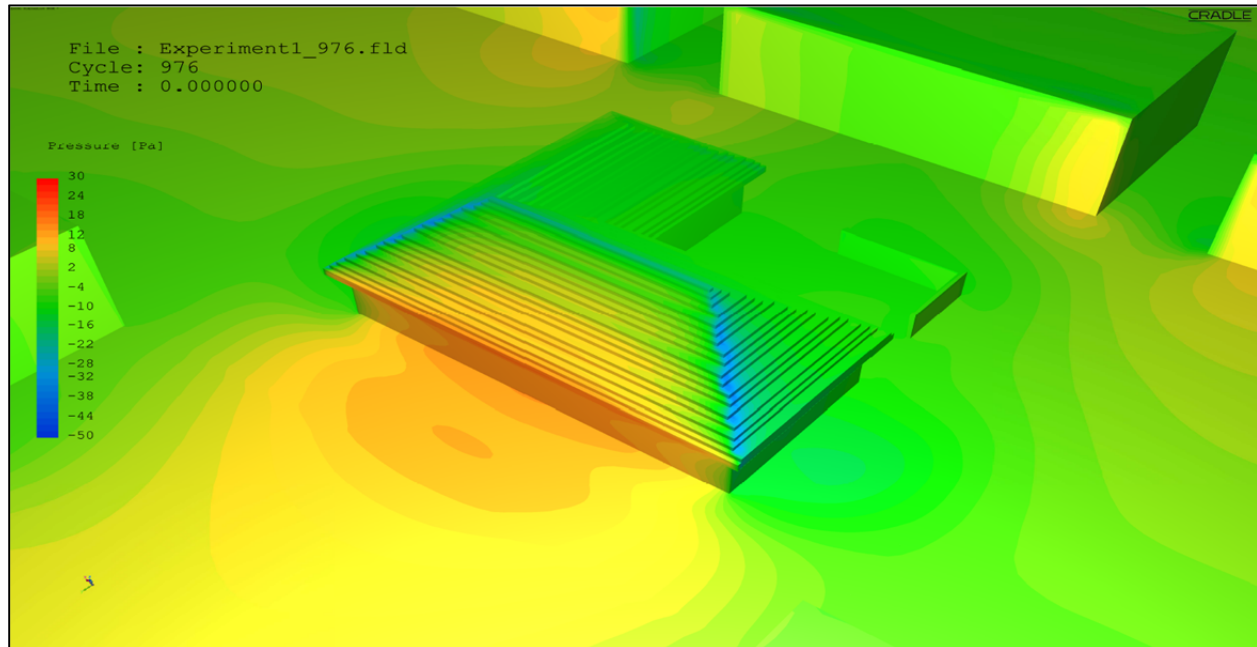
Figure 4.5.6 : Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).
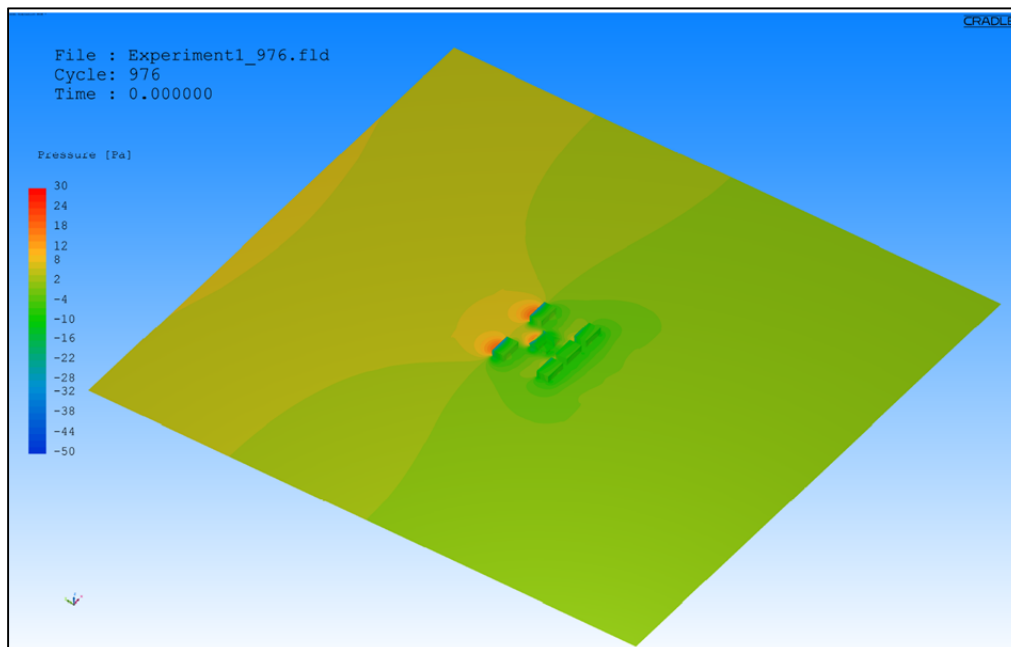
Figure 4.5.7: Pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).
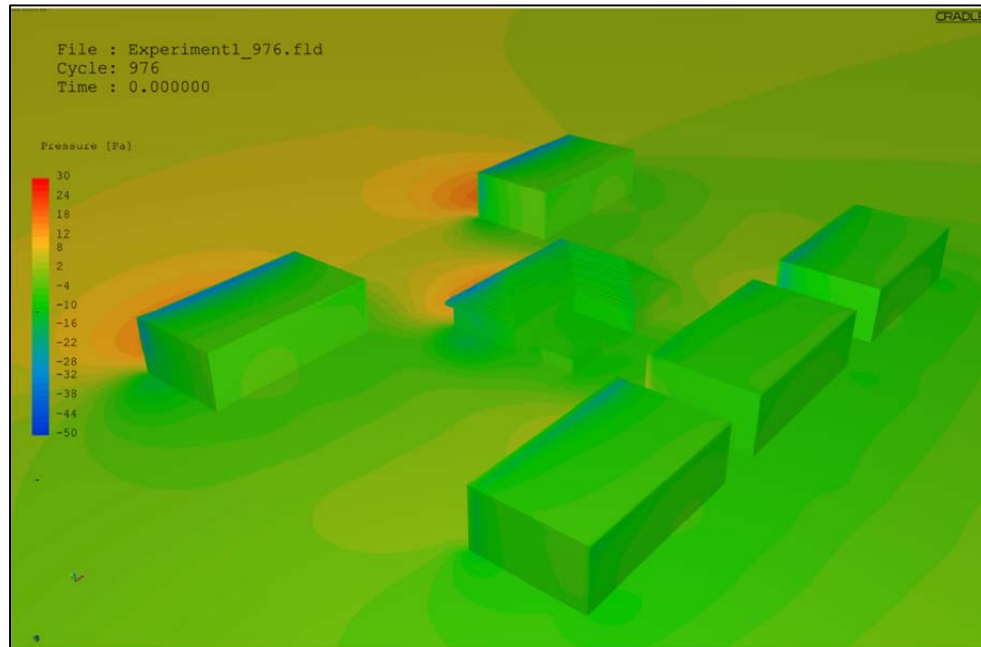
Figure 4.5.8: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 241
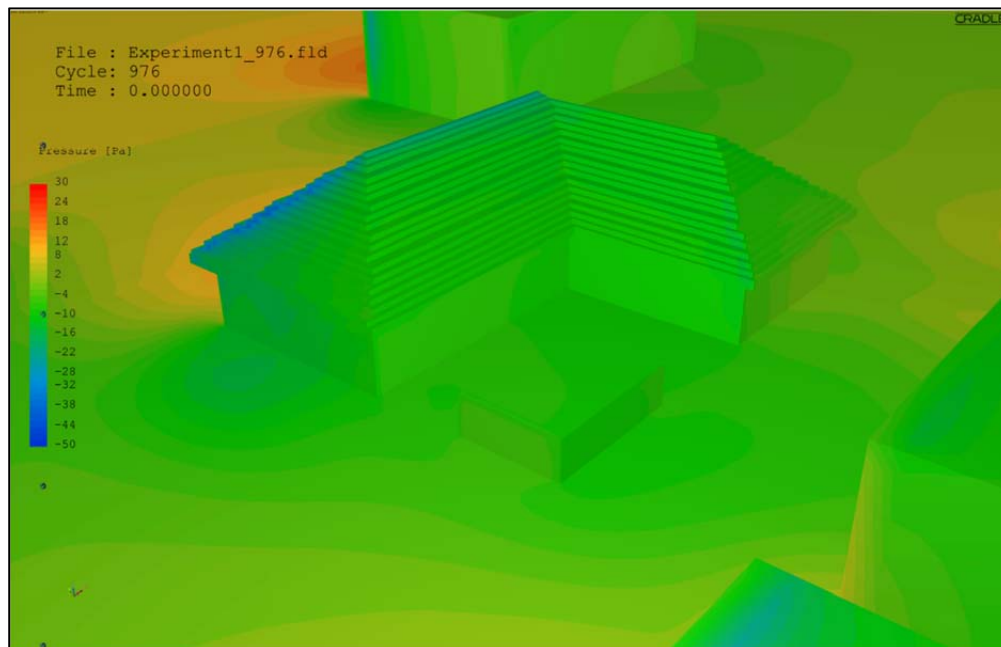
Figure 4.5.9: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle Stream).

Figure 4.5.10: Velocity (m/s) contour map at the plane 10 feet above the ground (Cradle Stream).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 242

Figure 4.5.11: Pressure (Pa) contour map at the plane 10 feet above the ground (Cradle Stream).
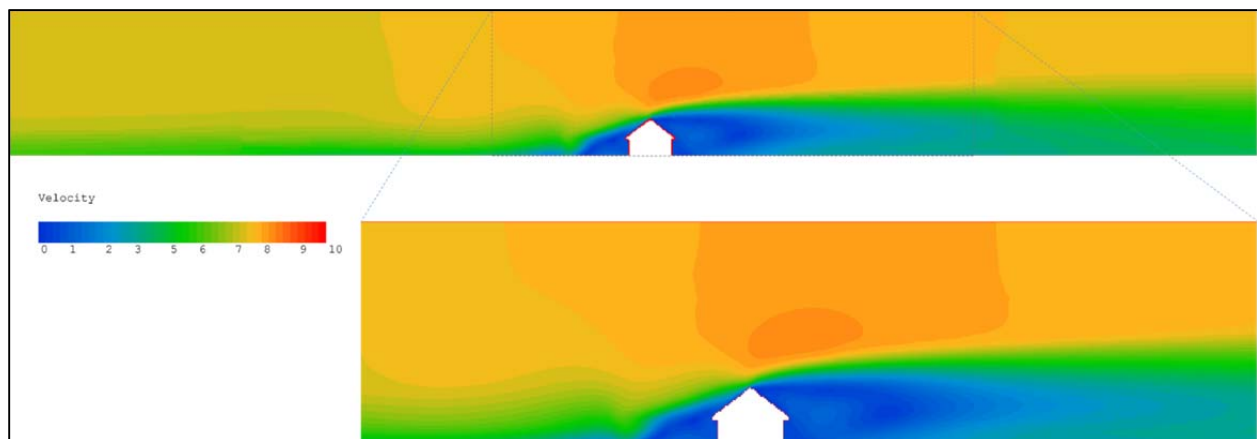
Figure 4.5.12: Wind velocity (m/s) contour map at the cross section (Cradle Stream).
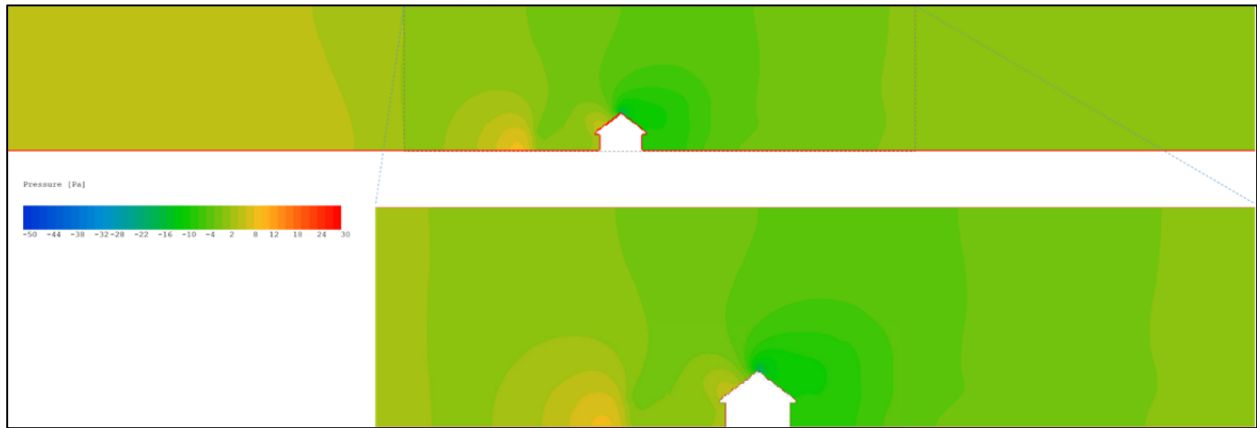
Figure 4.5.13: Pressure (Pa) contour map at the cross section (Cradle Stream).
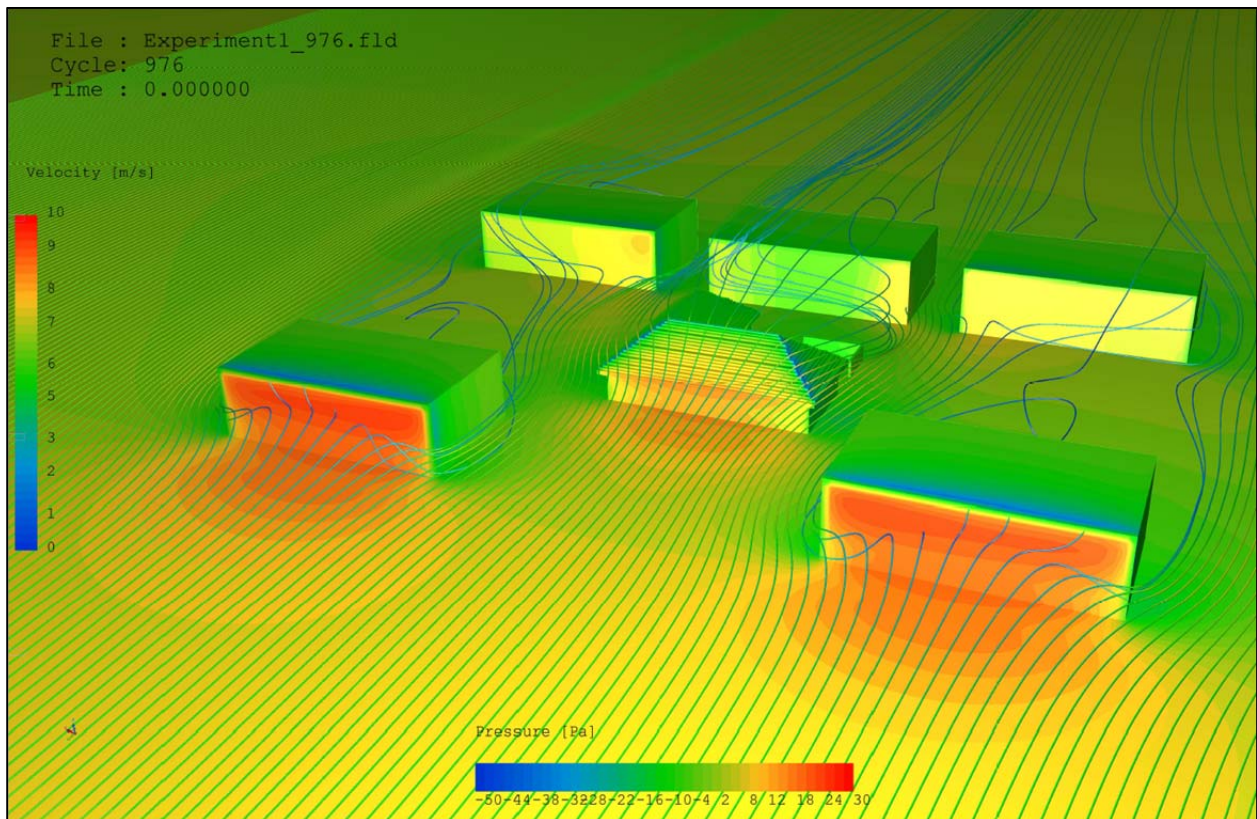
Figure 4.5.14: Pressure (Pa) distribution and wind velocity (m/s) streamlines (Cradle Stream).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 244

Figure 4.5.15: Tutorial and technical documents are available right at the menu bar for convenient accessibility (Cradle Stream).

4.6 Benchmarking the External CFD Workflow of Cradle SC/Tetra

This section presents the results and conclusions of the benchmark testing of external CFD for the software Cradle SC/Tetra (Tetra). This section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These eleven criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

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

   The significant features of Cradle's CFD applications are the CFD software company's effort to bring user friendliness and intuitiveness to CFD users. This is a very important criterion in choosing a CFD application for architecture/building industry. Cradle SC/Tetra application's GUI is organized in such a way that their components are lined up with successive steps in the CFD workflow. However, a somewhat shortcoming of the GUI is that big fonts and icons are hiding some of the working spaces and therefore causing problem when using smaller or single monitors.
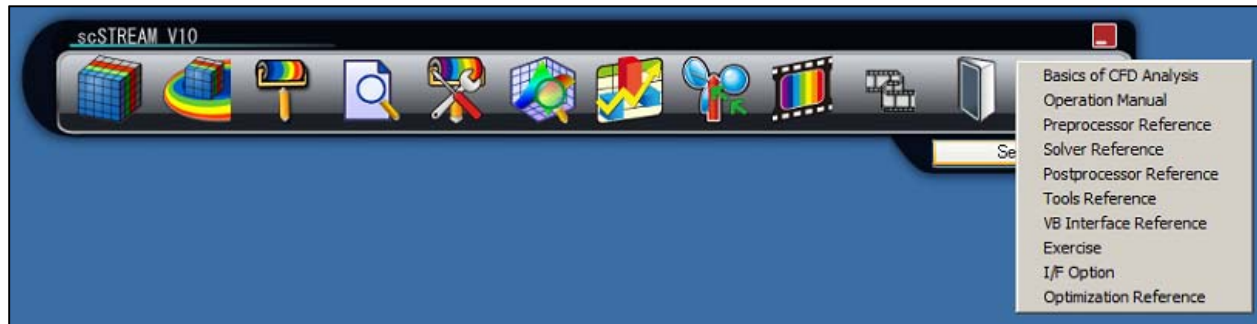
2. *Preprocessing: How effective was the conversion of the external 3D geometry data into the CFD software? Was it possible to read the 3D-geomerty into the CFD program easily; or did you use the CFD own 3D-geometry builder?*

   Cradle SC/Tetra does not have built-in modeler model to create 3d geometry from within the program. Instead, Cradle SC/Tetra supports STL file (STereoLithography) which is commonly used in many CAD software products for rapid computer-aid manufacturing. In the test case, the CFD team

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 245

converted the geometry file created in Autodesk Inventor into a STL file. The subsequent importing of geometry into Cradle SC/Tetra worked very well.

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

The meshing tool in Cradle SC/Tetra is effective and powerful in generating unstructured volume cells from the imported 3d geometries. The meshing process is quite effective and setting up different levels of meshing refinement, prism layers and other required steps in the meshing process were easily accomplished. (Figure 4.6.1 and Figure 4.6.2). SC/Tetra also allows for adaptive meshing refinement, however, the CFD team did not test this capability during the benchmarking.

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

The mesh generation procedure of Cradle SC/Tetra includes defining the octree file to represent the complexity of the geometry. Sometimes, the refinement levels of this functions did not work properly and required a considerable time and efforts for try and error fixes of the problem.

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

The process of setting up the boundary conditions and physical properties for the solver function was convenient and effective. SC/Tetra provides an easy way to work on this process with user-defined / customizable functions, material properties.

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

The test case was run for 1000 iterations and the CFD team encountered run time issues related to relaxation factor settings, which caused some solution convergence to fail. The technical support from Cradle was helpful in identifying and fixing the problem. After changing the relaxation factors, the solution converged after 1000 iterations but the residual plot showed a somewhat unsatisfactory convergence pattern. In order to solve these issues it would be necessary to improving meshing quality by means of meshing refinement. This, however, could not be done in during benchmarking due to time limitations. (Figure 4.6.3).

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

The solution met the convergence criteria after the 1000th run. After fixing the relaxation run time issue convergence was rather good. It also took about 3 hours for finishing each of the simulation runs.

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 246

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

Except the aforementioned issue with relaxation factor which caused solver interrupt, there was no other significant problem that the CFD team encountered.

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

Post-processor tools in SC/Tetra are quite intuitive to operate and they provide good visualization and assessment. Settings for legends, scale bars are colors are customizable and allow also for changing sizes and location. However, the CFD team found that the color rendering of the default ambient light scheme caused shading and fading effects which altered colors of the contours. This made it difficult at times to quantify velocities and pressures using the unobstructed color range provided in the legend. Some examples of the post-processing results from the test case include the contour maps showing pressure distribution on the ground and building surfaces for wind coming from the north (Figure 4.6.4 through Figure 4.6.9), contour maps showing pressure and wind velocity at the plane 10 feet above the ground (Figure 4.6.10 and Figure 4.6.11), the pressure and wind velocity on the section plane (Figure 4.6.12 and Figure 4.6.13), and the over-laid contour map of pressure and velocity streamlines (Figure 4.6.14).

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

In general, SC/Tetra is a multi-purpose CFD code which can be used for either internal or external airflow solutions. Since Tetra uses an unstructured mesh even complicated geometries, such as round buildings, can be effectively modeled. (The SC/Stream software, by virtue of using a structured grid, lacks the ability to model curved surfaces). The Human Body Thermoregulation module in SC/Tetra is especially suitable for studying internal airflow and thermal comfort. Optimization tools, important features of SC/Tetra, however, have not been tested during in subsequent test runs. A minor inconvenience, however, is caused by the numerous different file formats which are created and have to be maintained during the workflow. These files store results of certain simulation steps but the large number of files can confuse users.

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

During the test case, the CFD team easily accessed the technical documentations and tutorials right at the menu bar located in the software's GUI (Figure 4.6.15). The CFD team appreciated the

responsiveness of technical support from the Cradle Inc. for their help in trouble shooting several issues during the test case.

Figures 4.6.1 through 4.6.15 are shown below:
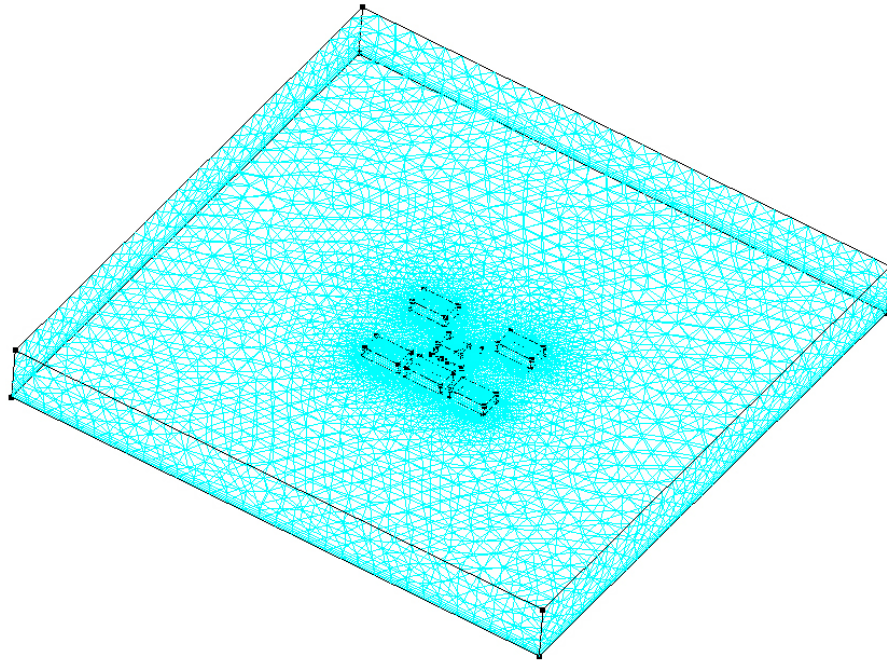
Figure 4.6.1: Tetrahedral volume cell type unstructured meshing generation with high density at the region of interest (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 248
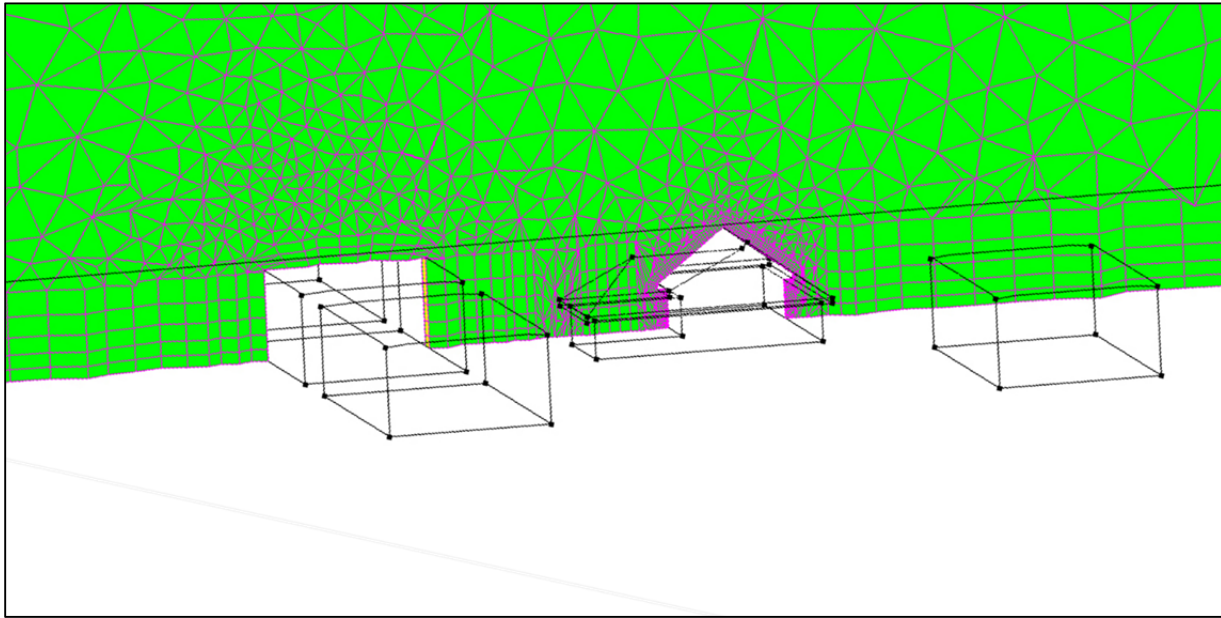
Figure 4.6.2: Cross section showing unstructured meshing generation with hexahedral volume cells and prisms wrapping around building surfaces and the ground (Cradle SC/Tetra).
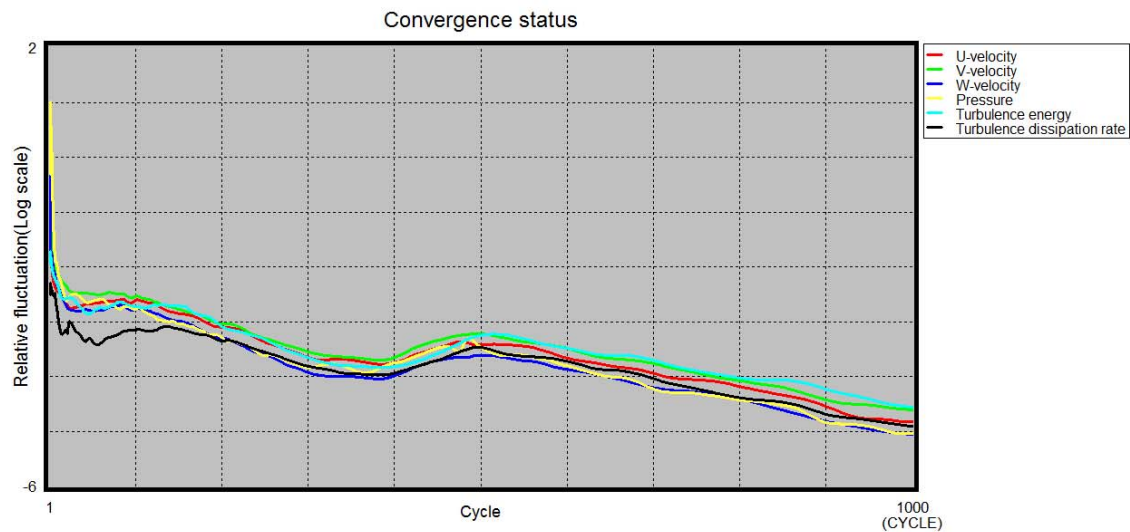
Figure 4.6.3: Residual plotting showing convergence status (Cradle SC/Tetra).
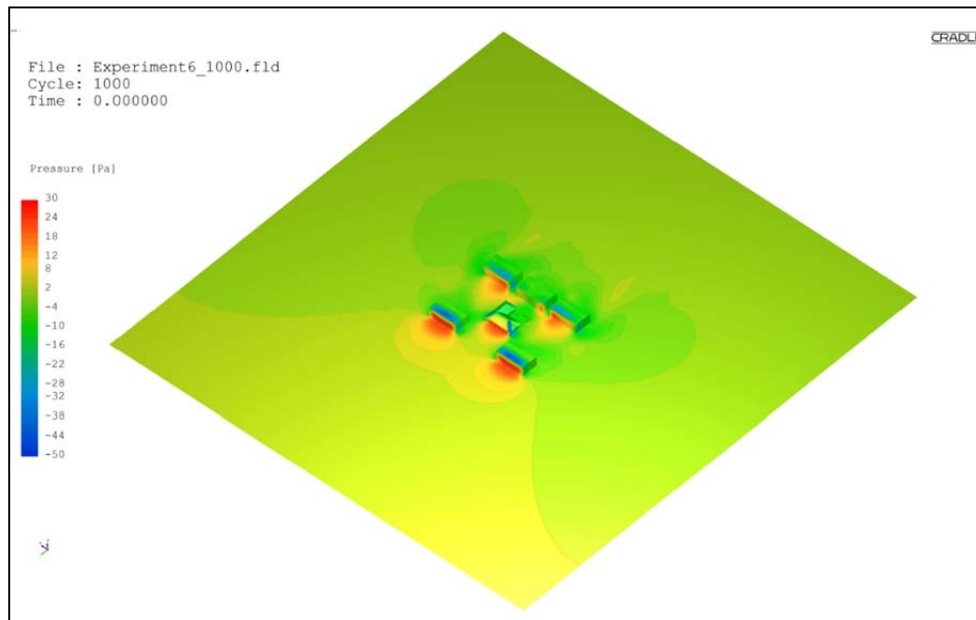
Figure 4.6.4: Pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).
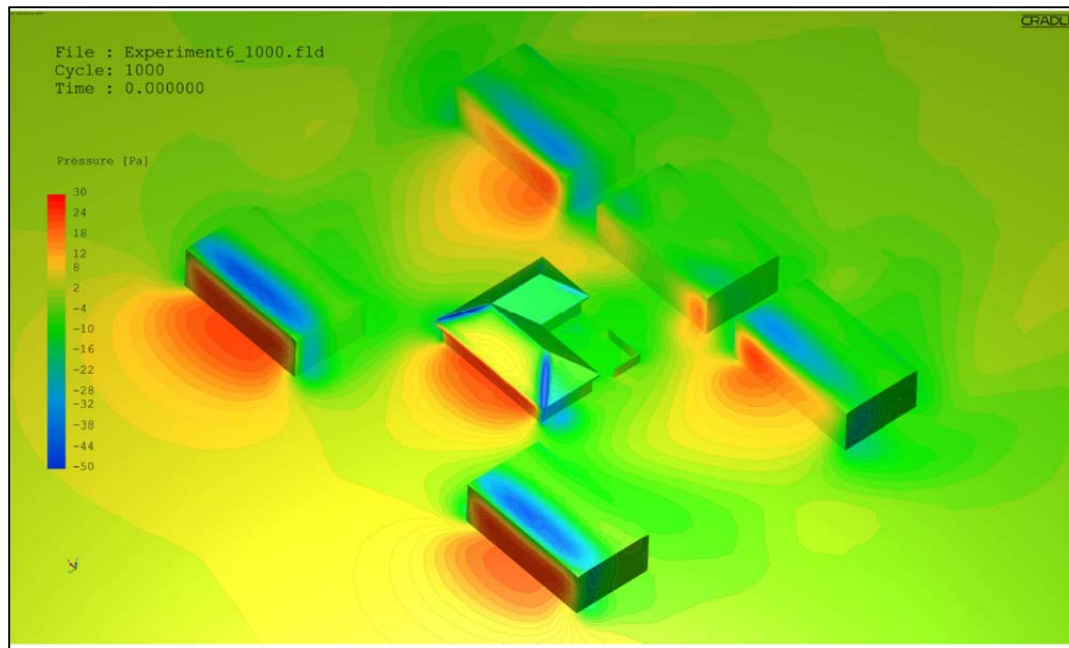
Figure 4.6.5: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 250
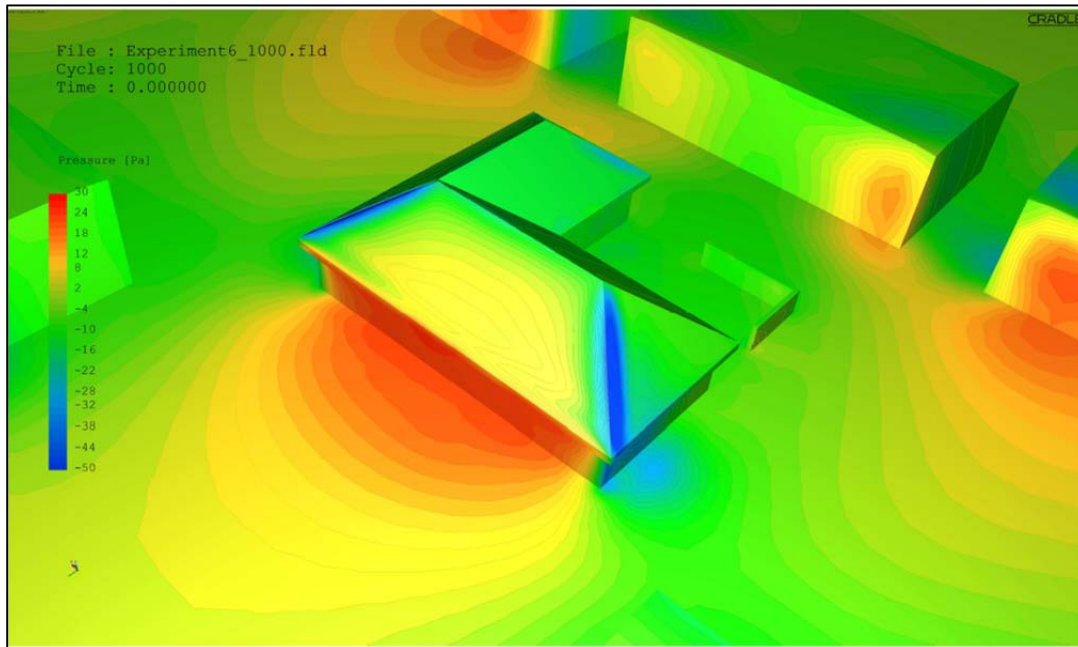
Figure 4.6.6: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).
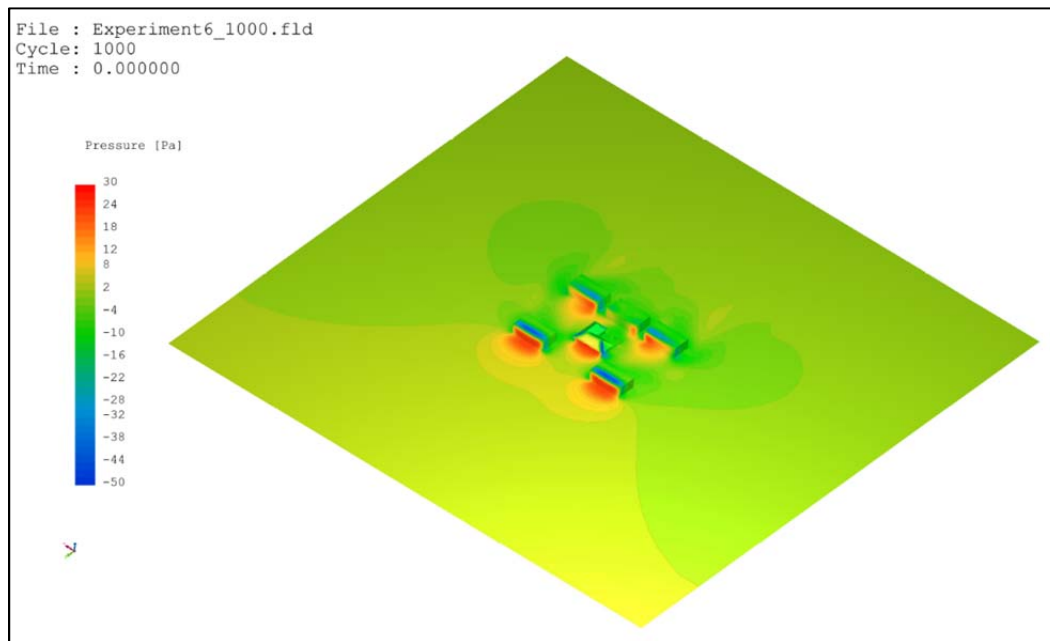
Figure 4.6.7: Pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                        School of Architecture, University of Hawaii

December 4, 2013   Page 251
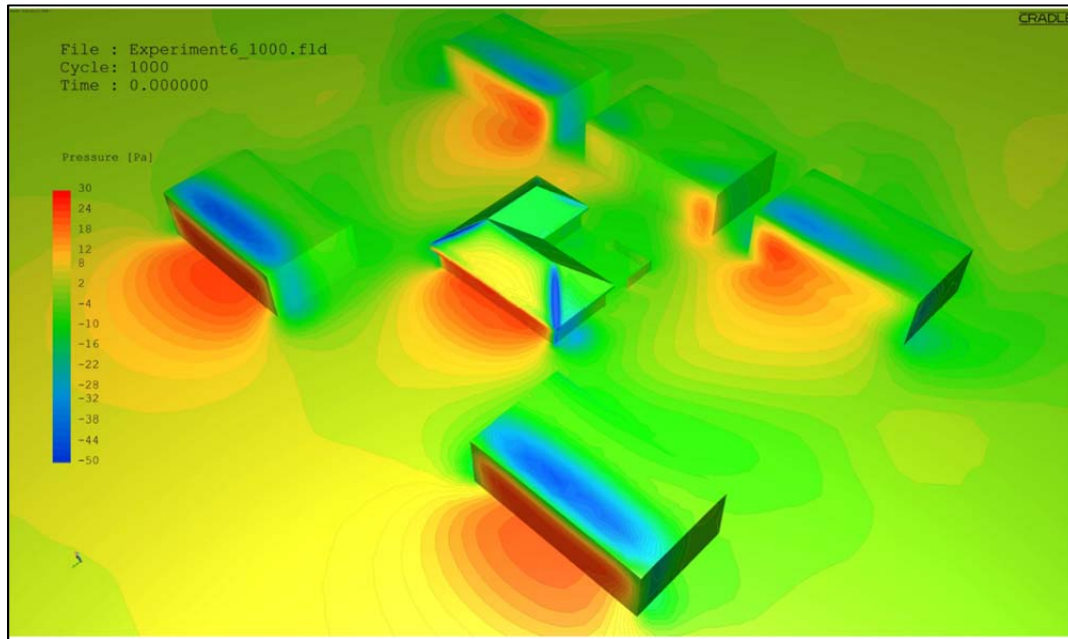
Figure 4.6.8: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).
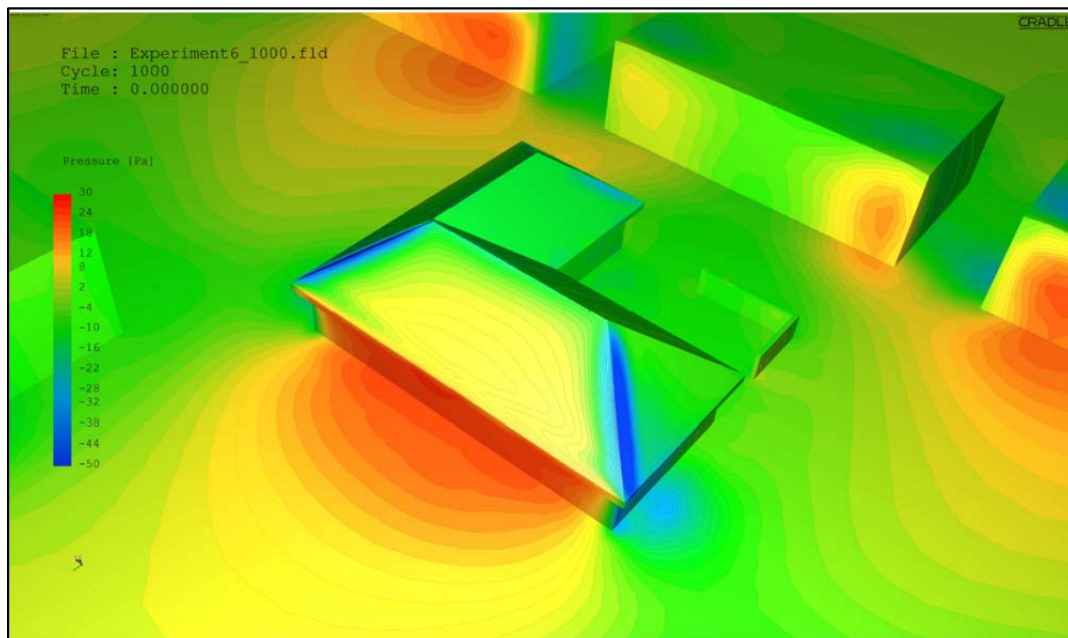
Figure 4.6.9: Close-up image of pressure (Pa) distribution on the ground and building surfaces (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                              School of Architecture, University of Hawaii

December 4, 2013   Page 252

Figure 4.6.10: Velocity (m/s) contour map at the plane 10 feet above the ground (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 253
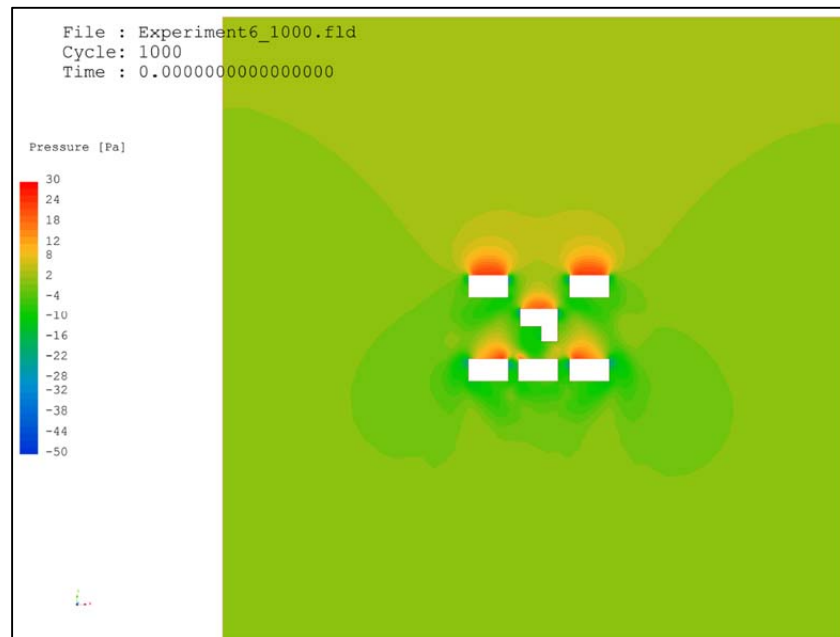
Figure 4.6.11: Pressure (Pa) contour map at the plane 10 feet above the ground (Cradle SC/Tetra).
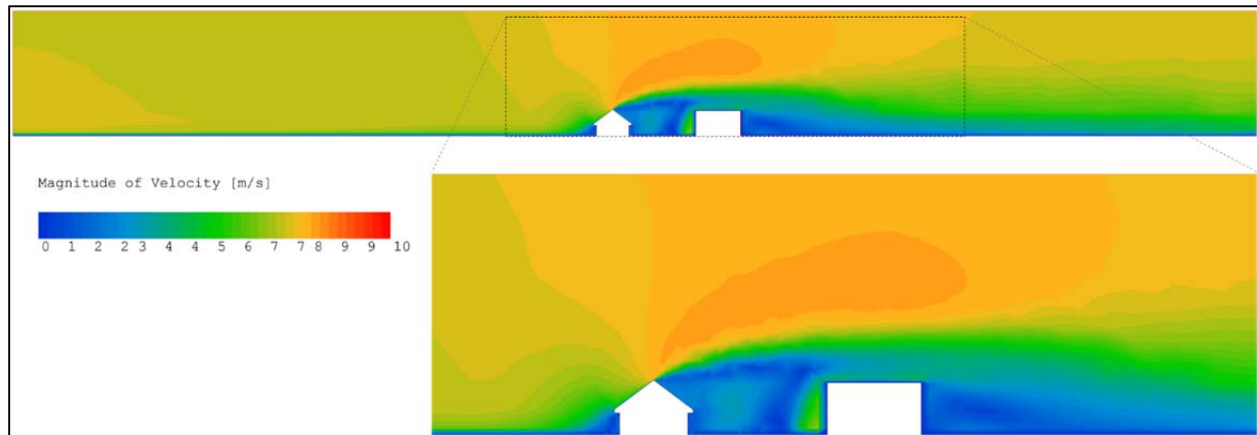
Figure 4.6.12: Wind velocity (m/s) contour map at the cross section (Cradle SC/Tetra).
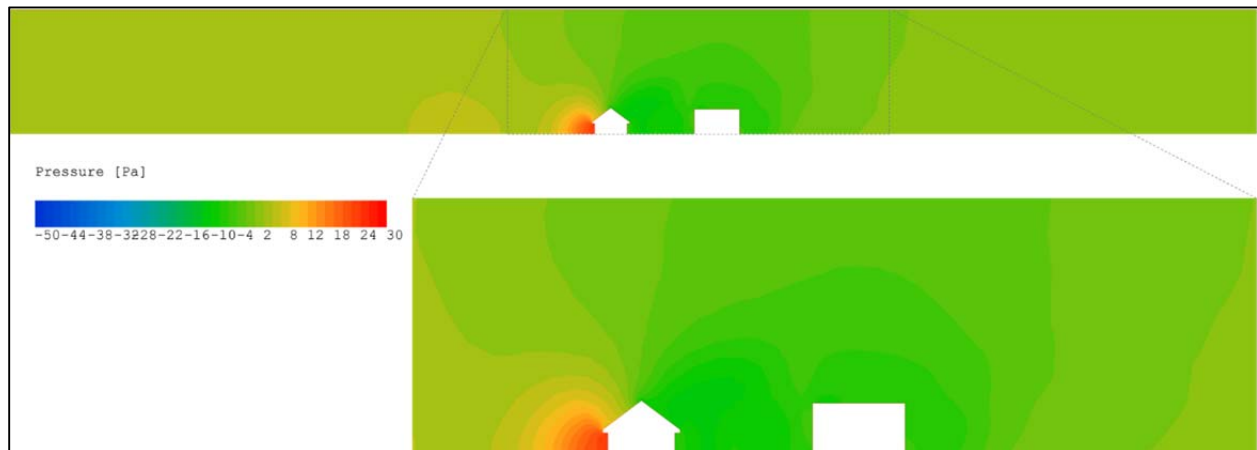
Figure 4.6.13: Pressure (Pa) contour map at the cross section (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

December 4, 2013

Page 254
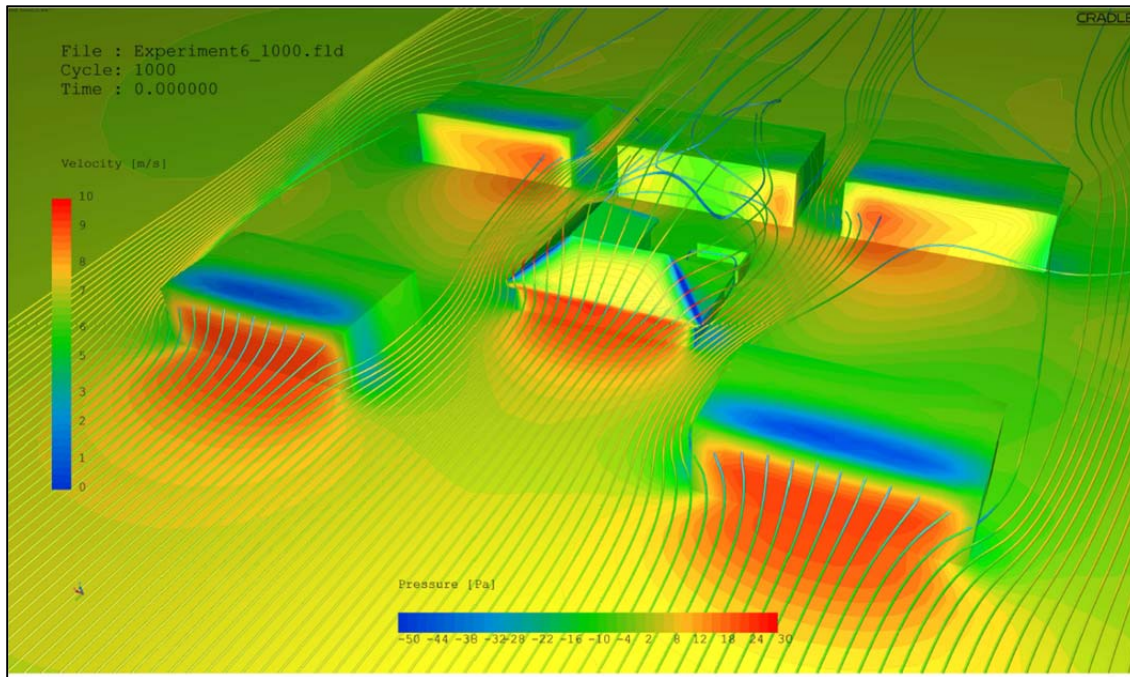
Figure 4.6.14: Pressure (Pa) distribution and wind velocity (m/s) streamlines (Cradle SC/Tetra).

Figure 4.6.15: Tutorial and technical documents are available right at the menu bar for convenient accessibility (Cradle SC/Tetra).

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013
Page 255

4.7 Benchmarking the External CFD Workflow of Design Builder

This section presents the results and conclusions of the benchmark testing of external CFD for the software Design Builder. This section provides an assessment of the software performance using eleven criteria / statements, 1 through 11. These eleven criteria / statements are identical for all six candidate CFD software in order to facilitate comparison of ease and effectiveness of external CFD workflow process.

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

   Design Builder provides a very simple set-up and intuitive interface for the external CFD workflow. Design Builder has designed the interface for external CFD across building industry users, with a short range of parameters that have to be provided by using intuitive functions and minimal drop down menu options. As an general observation, the CFD team finds it impressive that Design Builder is able to design a GUI within such a simple and easy to use graphic (Figure 4.7.1), while providing relevant input functions for the complex CFD calculation functions in the background.

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

   Design Builder allows import of 3D geometry from external CAD only through the gBXML format. Design Builder can integrate with several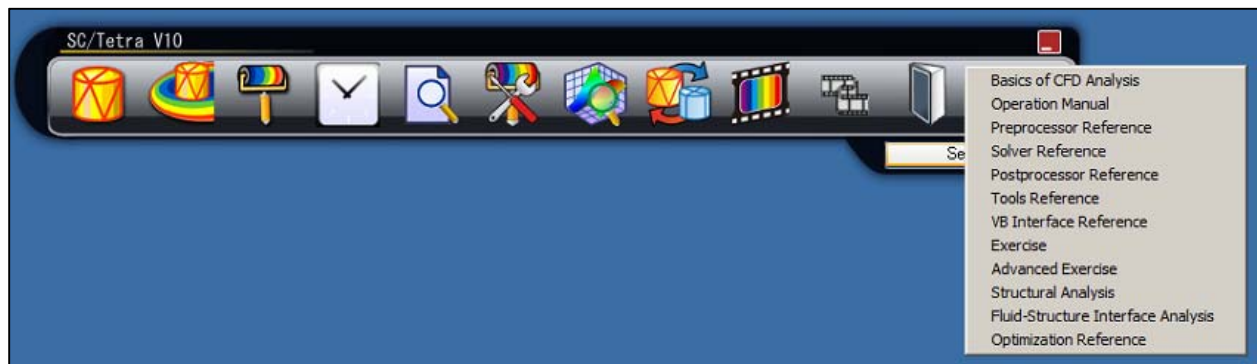 BIM systems, for instance, Revit Architecture (Figure 4.7.2). In order to minimize error, however, the CFD team finds it advantageous creating 3D geometry with the effective and user-friendly Design Builder internal modeler (e.g. the internal model builder is also used to create models for energy simulations). As a downside, the CFD team has found that Design Builder provides only generic building models and is lacking effective tools to create complex building geometries. Therefore, any complex or irregular building geometry modeled with the internal model will need to use simplifications of geometry.

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

   Design Builder does only provide a rather simplified mesh of uniform grid cells sizing (Figure 4.7.3). Design Builder provides only cubic volume cell types within a structured gird. Design Builder automatically adjusts the cell sizes to the building geometry, without a possible user interaction of identifying areas of interest where the mesh resolution is higher. As a result of this automatic cell size adjustment aspect ratio problems arise at times. This can negatively affect run time performance (Figure 4.7.4).

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 256

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

The CFD team has identified that the most effective functionality of Design Builder is the ability of the user to minimize efforts on mesh preparation and generation besides adjusting a basic cell size and boundary domain (Figure 4.7.5). On the other hand, the cell types produce uniform and structured grids. The downside of Design Builder is that it does not provide a significant opportunity of the user to set up grid details or chose physic in details. All of the more sophisticated CFD functions are hidden in the program backend and cannot be easily accessed or modified by the user.

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

Design Builder provides a basic set-up of boundary domain without asking the user to perform complex physic and mesh generating set-ups. K–epsilon turbulence is the default turbulence model (Figure 4.7.6), therefore all the user has to provide is cell–grid size, wind velocity, wind direction, and boundary domain factors (e.g. friction of the ground). Design Builder will automatically generate boundary layer type based on wind direction.

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

Design Builder provides simulation result even before the solution has sufficiently converged. Most of the unsuccessful convergences performance was due to complication of model geometry, geometry arrangements, or cell– grid size, which was too big.

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

With more complicated building configurations, CFD external simulation in Design Builder can take very long run-time. Several of the simulations carried out for benchmarking required between 12 and 24 hours for 1000 iterations (Figure 4.7.6).

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

Solver is a simple set-up but can take a long time depending on the complexity of the model.

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

Design Builder provides a basic slice plane visualization option and a 3D-contour option for visualizing results (Figure 4.7.7 through Figure 4.7.9). The visualization of results of external provides settings for pressure and velocity. Output allows export of image and video file with different screen

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

Page 257

resolutions. Results can also be exported as numeric data to spreadsheets. Overall the visualization is simple and produces basic output formats.

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

Overall, Design Builder provides a very effective GUI and intuitive tools which are easy and simple enough for a wide range of users, especially novices to CFD. The program has a simple set-up for geometry, meshing and physics compared to other CFD software products. The external CFD's workflow has a few steps and the set-up is easy compared to the achievable result. As a downside the CFD team observed slow simulation performance and long periods of time for solutions to converge.

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

Design Builder provides a outstanding help functions and online tutorials through the company's main website. Design Builder features one of the better training rich-media materials when compared to other CFD products.

Figures 4.7.1 through 4.7.9 are shown below:
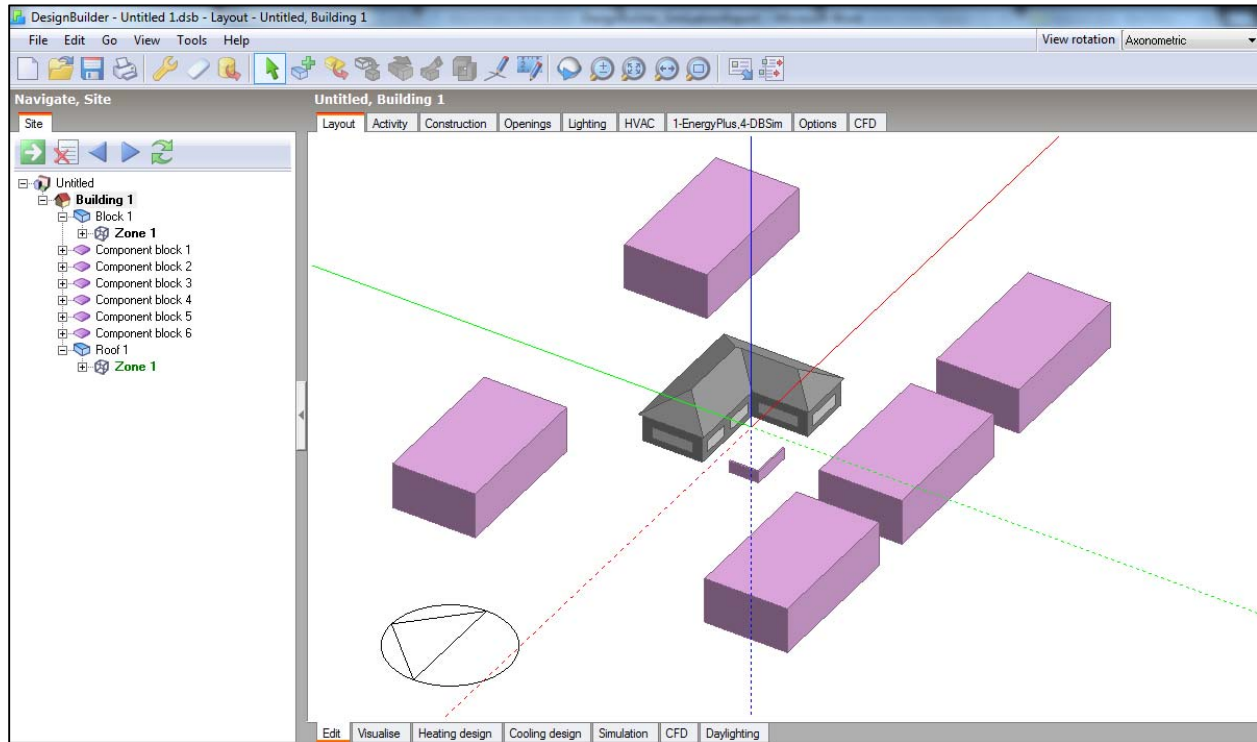
Figure 4.7.1: Design Builder graphic user interface
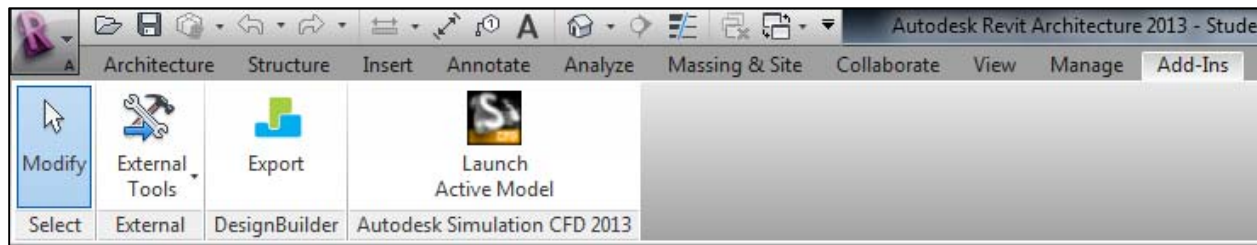
Figure 4.7.2: Design Builder is able to export from Revit Architecture

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013   Page 259
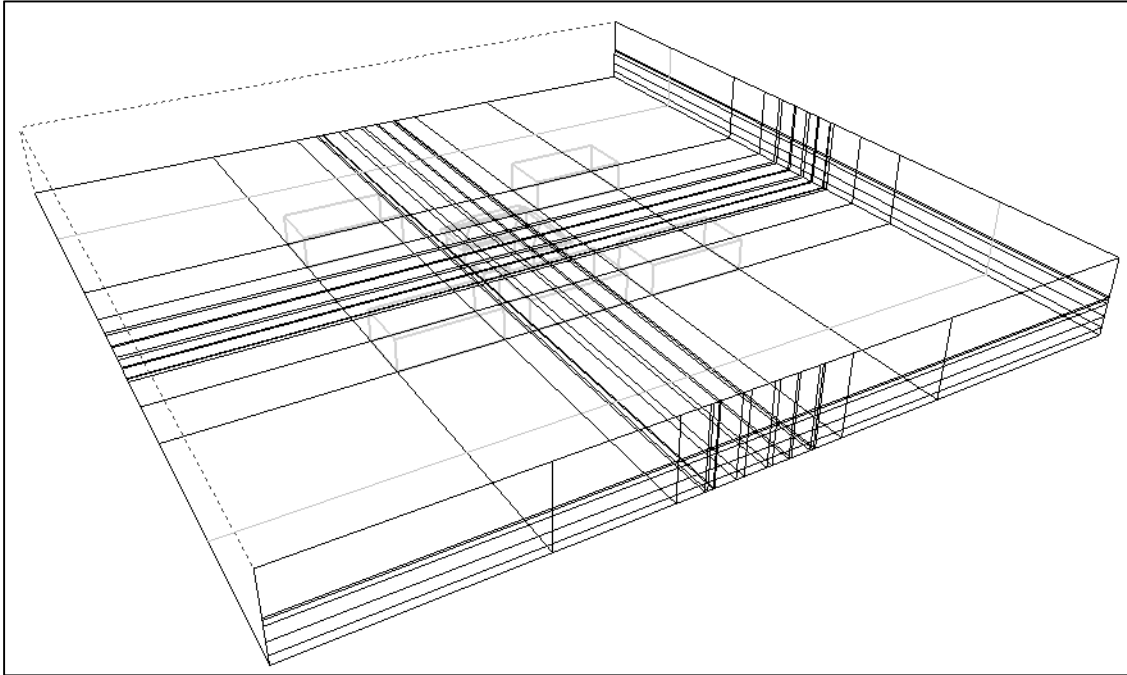
Figure 4.7.3: Design Builder shows grid and cell modification

Figure 4.7.4: Design Builder provides the aspect ratio for filtering any error and issue

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii

December 4, 2013

Page 260
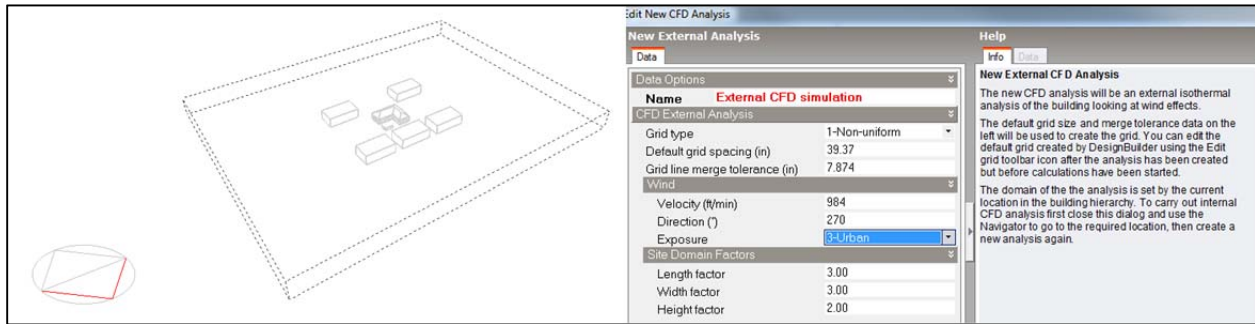
Figure 4.7.5: Design Builder shows basic the external CFD simulation set up
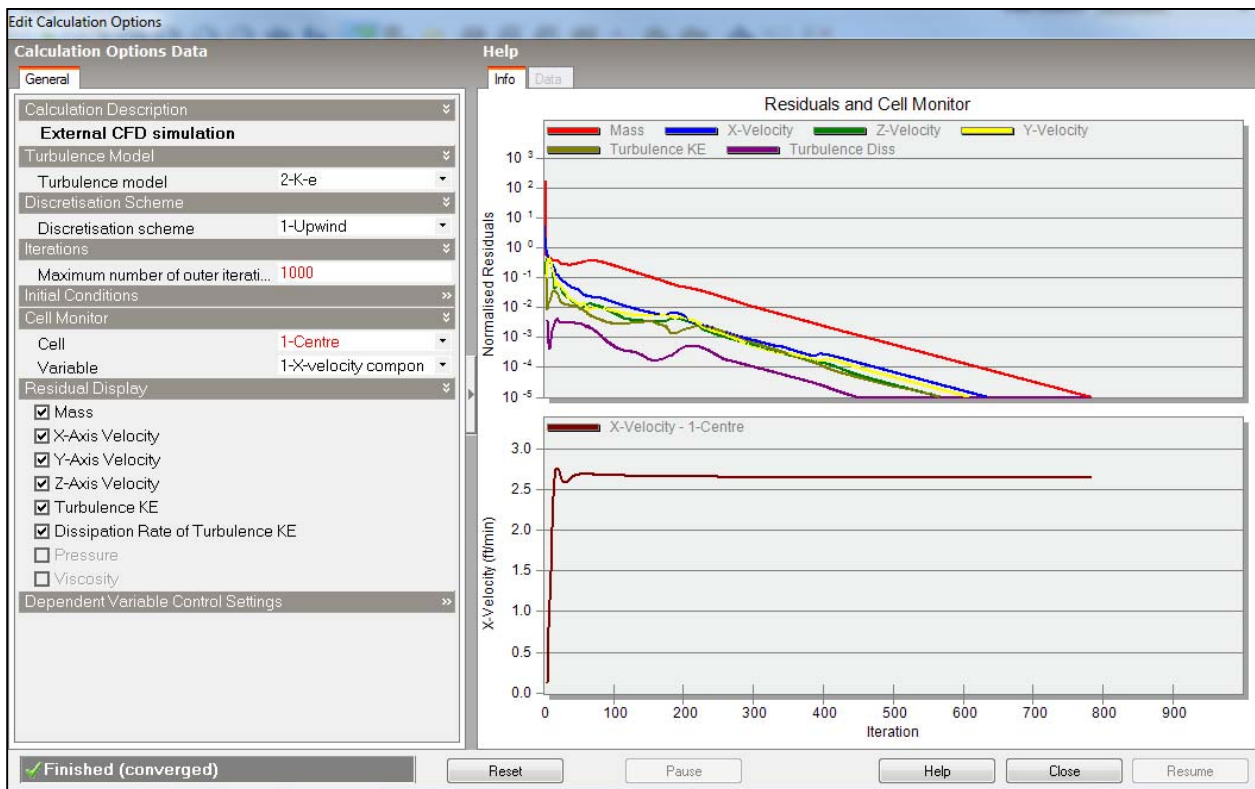
Figure 4.7.6: Design Builder shows basic turbulence, iterations set up, and residual – converged

Contract No.N000-14-13-1-0463

Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow

Environmental Research and Design Laboratory

School of Architecture, University of Hawaii

Page 261
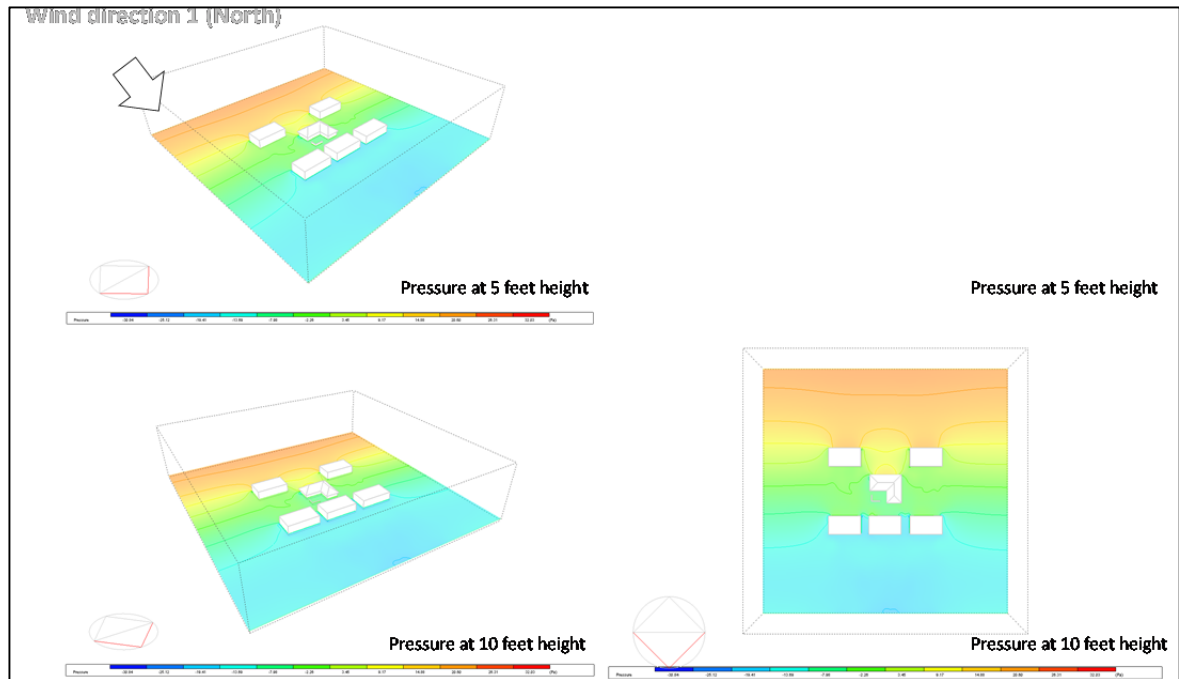
Figure 4.7.7: Design Builder shows the visualization of wind pressure (H$_2$O inch)at different height levels
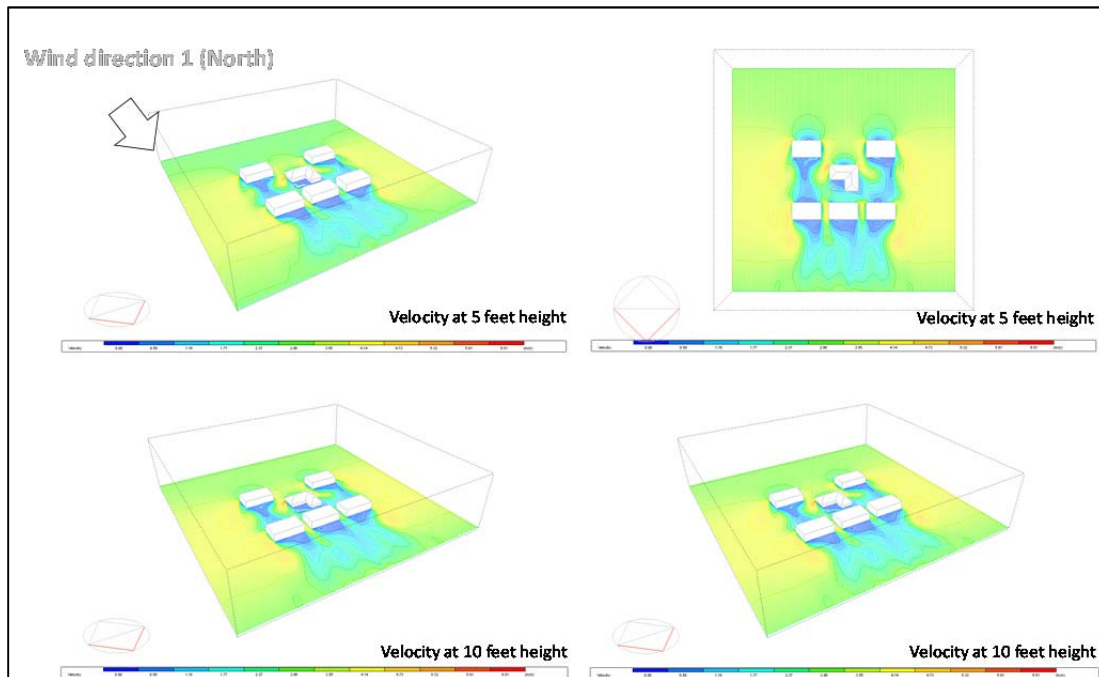
Figure 4.7.8: Design Builder shows the visualization of wind velocity (ft/mins)at different height levels

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                  School of Architecture, University of Hawaii

December 4, 2013   Page 262
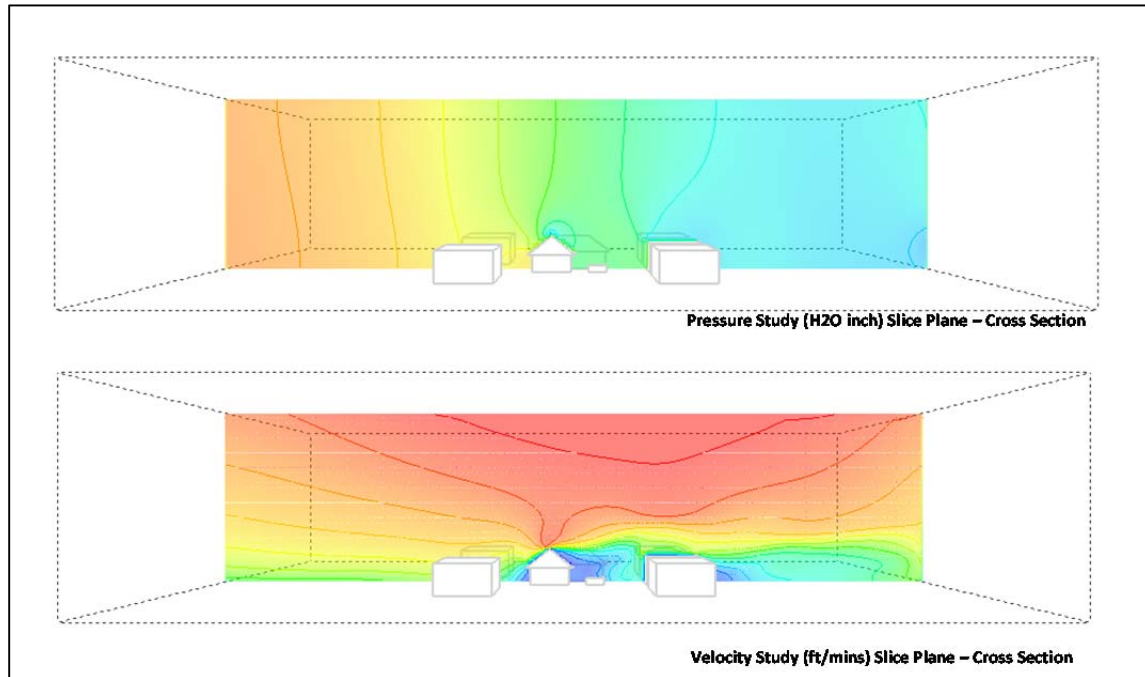
Figure 4.7.9:   Design Builder shows the visualization of wind pressure (H$_2$O inch) wind velocity (ft/mins)at the
cross section.

Part 5 - Selection of CFD Software Products for ERDL – HNEI Project Work

This section reports on the final findings of the workflow performance comparisons of the candidate CFD software products. While Part 3 of this report elaborates a preliminary overall ranking solely on the basis of the review of software product documentation, Part 5 presents the final overall ranking which includes conclusions drawn from actually benchmarking the candidate CFD software products. It should be noted that the final ranking includes six of the original seven CFD software products. The CFD research team came to the conclusion that the software Open Foam would not be considered or this CFD research project.

5.1   Final Overall Ranking of Candidate CFD Software Products

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

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

The summary of the final ranking results is provided in Table 5.1.1 and Figures 5.1.1 through 5.1.5.

The detailed analysis of the ranking for the seven CFD software products is shown on Table 5.1.2 (on the fold-out page following this page).

Table 5.1.1: Summary of the final ranking results for six candidate CFD software products

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

Contract No.N000-14-13-1-0463
Hawaii Natural Energy Institute

December 4, 2013

Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Environmental Research and Design Laboratory
School of Architecture, University of Hawaii
Page 265
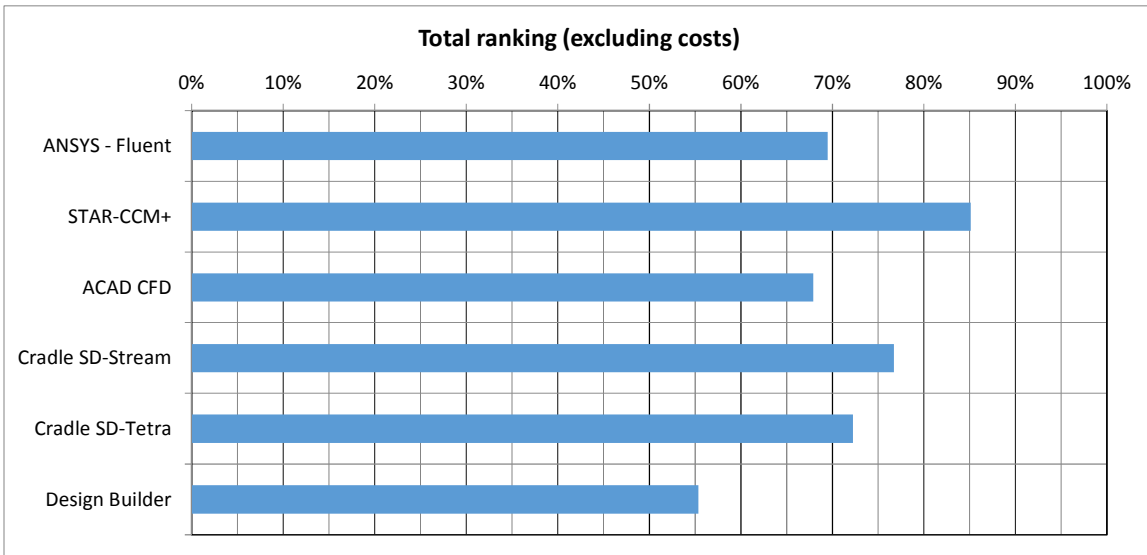
Figure 5.1.1:   Final overall (Total) ranking of six candidate CFD software products for ERDL-HNEI CFD research project

Figure 5.1.2:   Final ranking for ranking criteria group 1 of six candidate CFD software products for ERDL-HNEI CFD research
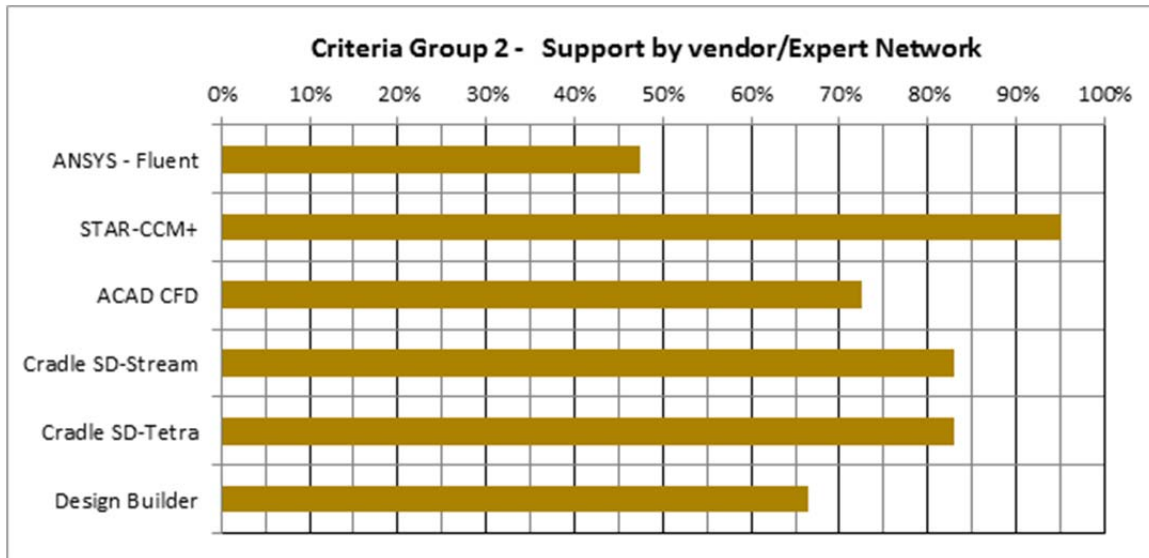
Figure 5.1.3:   Final ranking for ranking criteria group 2 of six candidate CFD software products for ERDL-HNEI CFD research
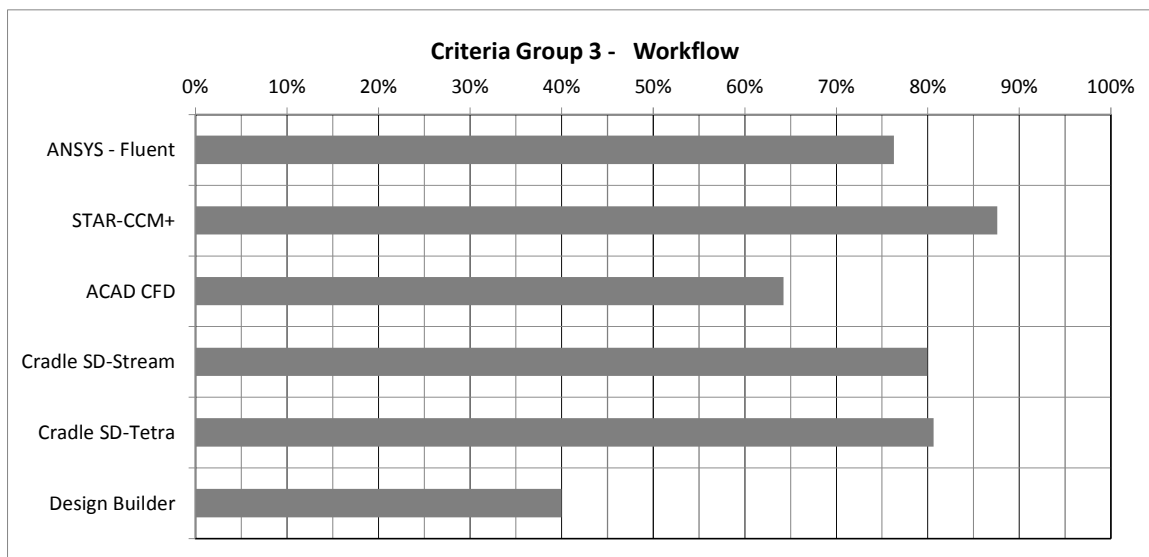
Figure 5.1.4:   Final ranking for ranking criteria group 3 of six candidate CFD software products for ERDL-HNEI CFD research

Contract No.N000-14-13-1-0463   Project Deliverable No. 2.2: FINAL Report - Establishing External CFD Workflow
Hawaii Natural Energy Institute   Environmental Research and Design Laboratory
                                                                                    School of Architecture, University of Hawaii

December 4, 2013   Page 267
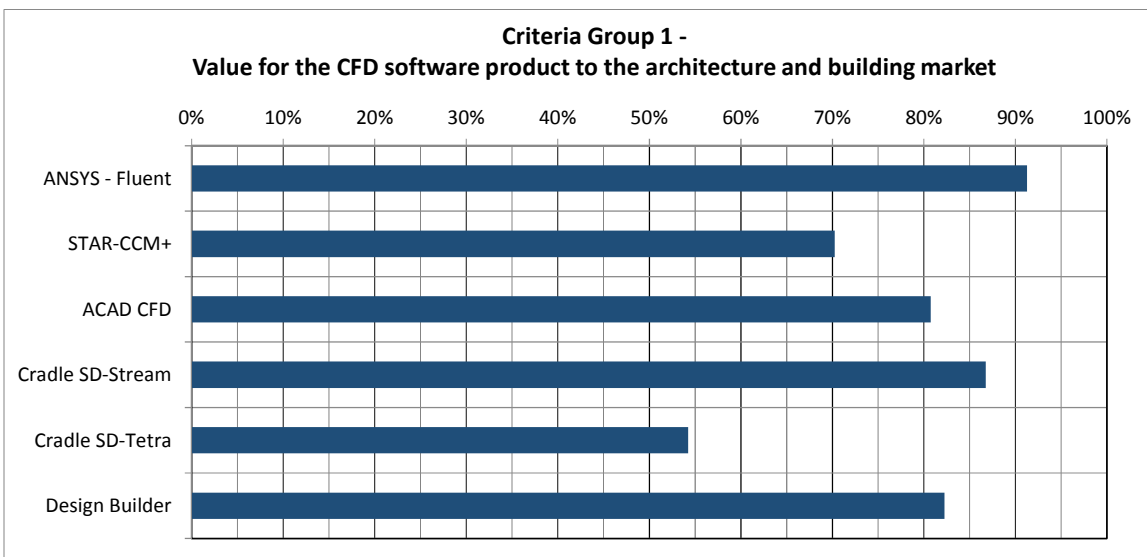
Figure 5.1.5:   Final ranking for ranking criteria group 4 of six candidate CFD software products for ERDL-HNEI CFD research
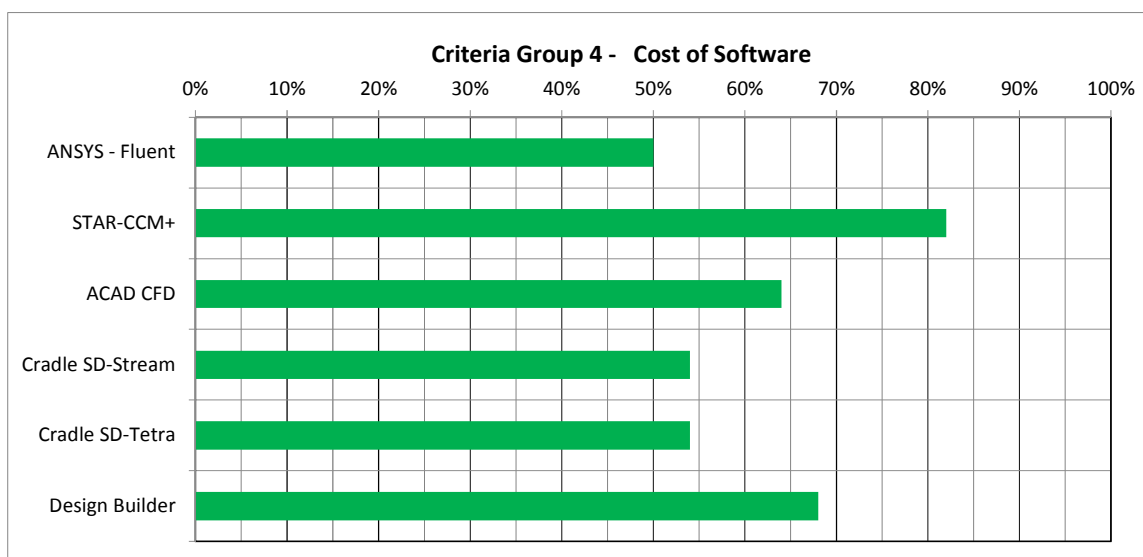
Table 5.1.2   Final Ranking matrix for CFD software products for external CFD

No.1	No.2	No.3	Description	w.first	w.second	w.third	w.overall	Fluent rank	Fluent %	Fluent pt%	STAR rank	STAR %	STAR pt%	ACAD rank	ACAD %	ACAD pt%	SD-Stream rank	SD-Stream %	SD-Stream pt%	SD-Tetra rank	SD-Tetra %	SD-Tetra pt%	DB rank	DB %	DB pt%
								1 to 5	% of 100%	Overall point %	1 to 5	% of 100%	Overall point %	1 to 5	% of 100%	Overall point %	1 to 5	% of 100%	Overall point %	1 to 5	% of 100%	Overall point %	1 to 5	% of 100%	Overall point %
1			**Value for the CFD software product to the architecture and building market**	15%																					
	1		Benefit for external CFD applications to architecture		20%		3.0%	5	95%	2.9%	5	95%	2.9%	5	95%	2.9%	5	95%	2.9%	5	95%	2.9%	5	95%	2.9%
	2		Previous applications in architecture / urban design		35%		5.3%	5	95%	5.0%	3	50%	2.6%	4	80%	4.2%	4	80%	4.2%	2	30%	1.6%	5	95%	5.0%
	3		Previous mentioning of software product in scientific or applied professional articles		20%		3.0%	5	95%	2.9%	3	50%	1.5%	3	50%	1.5%	4	80%	2.4%	1	5%	0.2%	3	50%	1.5%
	4		Using graphics output for presentations		25%		3.8%	4	80%	3.0%	5	95%	3.6%	5	95%	3.6%	5	95%	3.6%	5	95%	3.6%	4	80%	3.0%
			sum second level >>>		100%																				
2			**Support by vendor / expert network**	15%																					
	1		Technical support available		25%		3.8%	3	50%	1.9%	5	95%	3.6%	1	5%	0.2%	4	80%	3.0%	4	80%	3.0%	3	50%	1.9%
	2		Online tutorials available on demand		20%		3.0%	3	50%	1.5%	5	95%	2.9%	5	95%	2.9%	4	80%	2.4%	4	80%	2.4%	5	95%	2.9%
	3		Online classes available (low entry barriers)		20%		3.0%	2	30%	0.9%	5	95%	2.9%	5	95%	2.9%	4	80%	2.4%	4	80%	2.4%	3	50%	1.5%
	4		Network of CFD professionals in architecture field		15%		2.3%	2	30%	0.7%	5	95%	2.1%	5	95%	2.1%	4	80%	1.8%	4	80%	1.8%	3	50%	1.1%
	5		Ease of working with academic support		10%		1.5%	3	50%	0.8%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	4	80%	1.2%
	6		User guide provides the efficient information		10%		1.5%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%	5	95%	1.4%
			sum second level >>>		100%																				
3			**Workflow**	52%																					
	1		**Ease of use / GUI**		15%																					
		1	GUI is easy to use for trained operator			60%	4.7%	4	80%	3.7%	4	80%	3.7%	5	95%	4.4%	5	95%	4.4%	5	95%	4.4%	5	95%	4.4%
		2	User guide and online help is available for each command			40%	3.1%	3	50%	1.6%	5	95%	3.0%	5	95%	3.0%	4	80%	2.5%	4	80%	2.5%	5	95%	3.0%
			sum third level >>>			100%																			
	2		**Modeling / Import geometry**		15%																					
		1	Create geometry inside CFD application			15%	1.2%	4	80%	0.9%	5	95%	1.1%	1	5%	0.1%	4	80%	0.9%	1	5%	0.1%	5	95%	1.1%
		2	Import geometry / repair surface before meshing			20%	1.6%	4	80%	1.2%	5	95%	1.5%	5	95%	1.5%	4	80%	1.2%	4	80%	1.2%	3	50%	0.8%
		3	Support Revit Architecture			30%	2.3%	3	50%	1.2%	4	80%	1.9%	5	95%	2.2%	2	30%	0.7%	2	30%	0.7%	5	95%	2.2%
		4	Support CADD format and Inventor			20%	1.6%	4	80%	1.2%	5	95%	1.5%	5	95%	1.5%	5	95%	1.5%	5	95%	1.5%	1	5%	0.1%
		5	Support Rhino			15%	1.2%	1	5%	0.1%	4	80%	0.9%	1	5%	0.1%	2	30%	0.4%	2	30%	0.4%	1	5%	0.1%
			sum third level >>>			100%																			
	3		**Meshing**		20%																					
		1	Surface meshers			30%	3.1%	1	5%	0.2%	5	95%	3.0%	3	50%	1.6%	4	80%	2.5%	1	5%	0.2%	2	30%	0.9%
		2	Maximum cells and 2D-3D cell volume			10%	1.0%	5	95%	1.0%	5	95%	1.0%	4	80%	0.8%	5	95%	1.0%	5	95%	1.0%	2	30%	0.3%
		3	Boundary layer types			25%	2.6%	5	95%	2.5%	5	95%	2.5%	4	80%	2.1%	4	80%	2.1%	5	95%	2.5%	2	30%	0.8%
		4	Structured and Unstructured meshing			10%	1.0%	5	95%	1.0%	5	95%	1.0%	4	80%	0.8%	4	80%	0.8%	5	95%	1.0%	2	30%	0.3%
		5	Adaptive meshing capability			25%	2.6%	5	95%	2.5%	5	95%	2.5%	2	30%	0.8%	4	80%	2.1%	5	95%	2.5%	1	5%	0.1%
			sum third level >>>			100%																			
	4		**Physics Models**		35%																					
		1	Solvers - Coupled / decoupled			15%	2.7%	5	95%	2.6%	5	95%	2.6%	3	50%	1.4%	4	80%	2.2%	5	95%	2.6%	2	30%	0.8%
		2	Solver-Finite Element Method (FEV) / Finite Volume Method (FVM)			10%	1.8%	5	95%	1.7%	5	95%	1.7%	4	80%	1.5%	4	80%	1.5%	5	95%	1.7%	3	50%	0.9%
		3	Steady State/Transient State			10%	1.8%	5	95%	1.7%	5	95%	1.7%	4	80%	1.5%	4	80%	1.5%	5	95%	1.7%	3	50%	0.9%
		4	Turbulence models			35%	6.4%	5	95%	6.1%	5	95%	6.1%	3	50%	3.2%	4	80%	5.1%	5	95%	6.1%	2	30%	1.9%
		5	Materials			10%	1.8%	4	80%	1.5%	5	95%	1.7%	4	80%	1.5%	4	80%	1.5%	5	95%	1.7%	1	5%	0.1%
		6	Customizable / special physics models			20%	3.6%	4	80%	2.9%	5	95%	3.5%	4	80%	2.9%	4	80%	2.9%	5	95%	3.5%	1	5%	0.2%
			sum third level >>>			100%																			
	5		**Post processing**		5%																					
		1	Visualization			25%	0.7%	4	80%	0.5%	5	95%	0.6%	3	50%	0.3%	5	95%	0.6%	5	95%	0.6%	3	50%	0.3%
		2	Quantitative analysis capabilities of components			25%	0.7%	4	80%	0.5%	5	95%	0.6%	3	50%	0.3%	4	80%	0.5%	4	80%	0.5%	1	5%	0.0%
		3	Comparative analysis (multi-scenarios)			50%	1.3%	4	80%	1.0%	3	50%	0.7%	3	50%	0.7%	5	95%	1.2%	5	95%	1.2%	2	30%	0.4%
			sum third level >>>			100%																			
	6		**Customization / special functions**		5%																					
		1	Pedestrain comfort			15%	0.4%	5	95%	0.4%	5	95%	0.4%	3	50%	0.2%	5	95%	0.4%	5	95%	0.4%	3	50%	0.2%
		2	Natural ventilation			30%	0.8%	5	95%	0.7%	5	95%	0.7%	4	80%	0.6%	4	80%	0.6%	5	95%	0.7%	3	50%	0.4%
		3	Wind-driven rain			10%	0.3%	3	50%	0.1%	5	95%	0.2%	2	30%	0.1%	2	30%	0.1%	2	30%	0.1%	1	5%	0.0%
		4	Humidity			15%	0.4%	1	5%	0.0%	3	50%	0.2%	4	80%	0.3%	4	80%	0.3%	5	95%	0.4%	3	50%	0.2%
		5	Wind load towards to building structure			5%	0.1%	5	95%	0.1%	5	95%	0.1%	2	30%	0.0%	5	95%	0.1%	3	50%	0.1%	2	30%	0.0%
		6	Pressure distribution along streamlines/pressure losses			15%	0.4%	5	95%	0.4%	3	50%	0.2%	1	5%	0.0%	4	80%	0.3%	3	50%	0.2%	2	30%	0.1%
		7	Conjugate heat transfer			10%	0.3%	5	95%	0.2%	5	95%	0.2%	2	30%	0.1%	4	80%	0.2%	3	50%	0.1%	1	5%	0.0%
			sum third level >>>			100%																			
	7		**Support for validation**		5%																					
		1	Validation document / example			100%	2.6%	4	80%	2.1%	2	30%	0.8%	1	5%	0.1%	5	95%	2.5%	5	95%	2.5%	1	5%	0.1%
			sum third level >>>			100%																			
			sum second level >>>		100%																				
4			**Cost of Software**	18%																					
	1		Commercial pricing model		20%		3.6%	2	30%	1.1%	2	30%	1.1%	2	30%	1.1%	2	30%	1.1%	2	30%	1.1%	3	50%	1.8%
	2		Academic pricing		40%		7.2%	2	30%	2.2%	5	95%	6.8%	5	95%	6.8%	4	80%	5.8%	4	80%	5.8%	5	95%	6.8%
	3	3	Software includes all modules (no extra costs)		20%		3.6%	4	80%	2.9%	5	95%	3.4%	3	50%	1.8%	3	50%	1.8%	3	50%	1.8%	3	50%	1.8%
	4	4	Option to aquire sufficient seats at no extra costs		20%		3.6%	4	80%	2.9%	5	95%	3.4%	3	50%	1.8%	2	30%	1.1%	2	30%	1.1%	3	50%	1.8%
			sum second level >>>		100%																				
			sum of first level >>>	100%				**sum overall points >> 69.5%**			**85.1%**			**67.9%**			**76.8%**			**72.2%**			**55.3%**		

The ranking criteria is shown below:

ranking criteria

1	5%	This criteria is not achieved or is only achieved to a very limited extent'
2	30%	This criteria is somewhat achieved, but more than a little achievement
3	50%	The criteria is achieved on an average (50%) level
4	80%	This criteria is achieved to an above average level
5	95%	This criteria is totally achieved or to a very high extent

Table 5.1.2
Final Ranking matrix for CFD software products for external CFD

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

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

4. Star-CCM+ by Cd-Adapco
5. scStream by Cradle USA Inc.
6. SC/TETRA by Cradle USA Inc.

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

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

REFERENCES

All documents used in this report are derived from user guides or other documentation provided by the software vendor. For the sake of brevity the references are noted throughout the text rather than given in form of a reference list.

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