Fall 2014

Computational Fluid Dynamics for Early Stage Architectural Design

Russell Wisniewski
Portland State University

Corey T. Griffin
Portland State University, cgriffin@pdx.edu

Mark Stroller
Portland State University

Lona Rerick
Portland State University

Erika Colvin
Portland State University

See next page for additional authors

Authors
Russell Wisniewski, Corey T. Griffin, Mark Stroller, Lona Rerick, Erika Colvin, Jeanne Jameson, and Stephen Colin

Computational Fluid Dynamics for Early Stage Architectural Design

Portland State University
YOST GRUBE HALL ARCHITECTURE

Russell Wisniewski
Corey Griffin
Mark Stroller
Lona Rerick
Erika Colvin
Jeanne Jameson
Stephen Colin

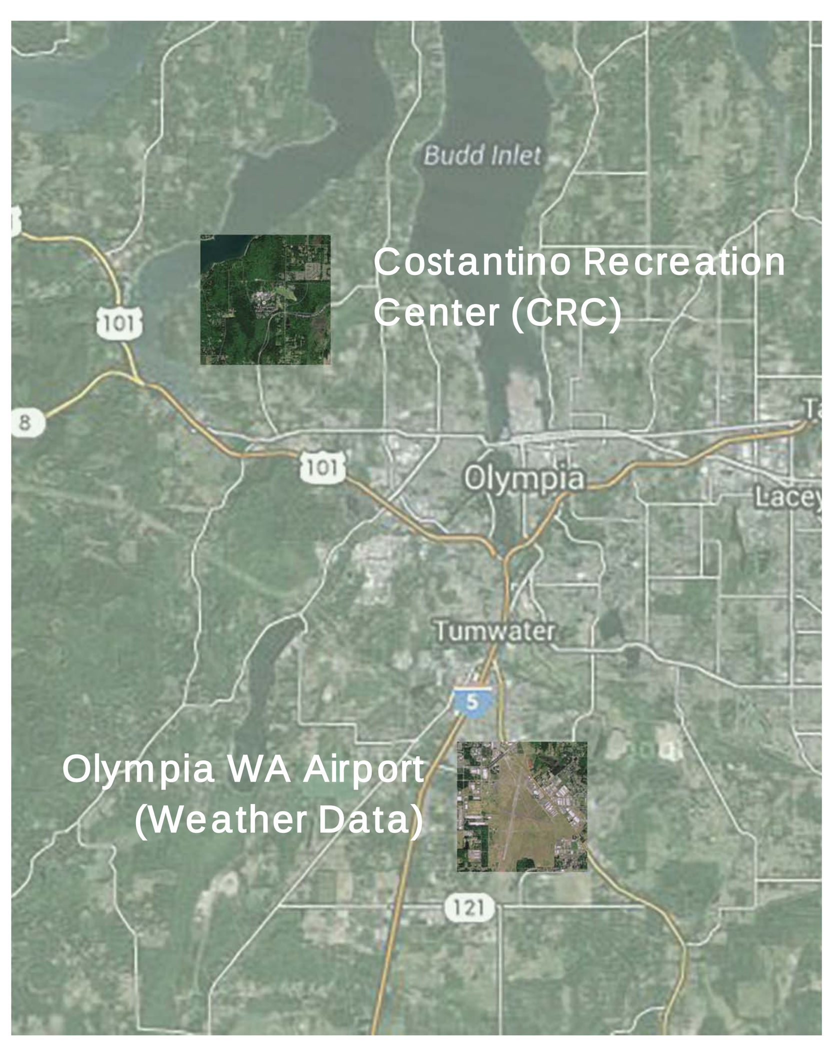
Introduction

This project set out to evaluate the validity and usefulness of CFD within the contemporary Design Practice. Initially attempting to analyze a specific design it became clear a more general study was needed. Analyzing natural ventilation is an infinitely complex process, to determine the level of simplication required for use involved breaking the natural process down and analyzing the pieces seperately and then together. By using this process it become clear how the process can be used by architecture firms within their design process.

Project Background

The Evergreen State College is located in Olympia, WA.
@ 46.97° N, 122.9° W elev 200 ft.

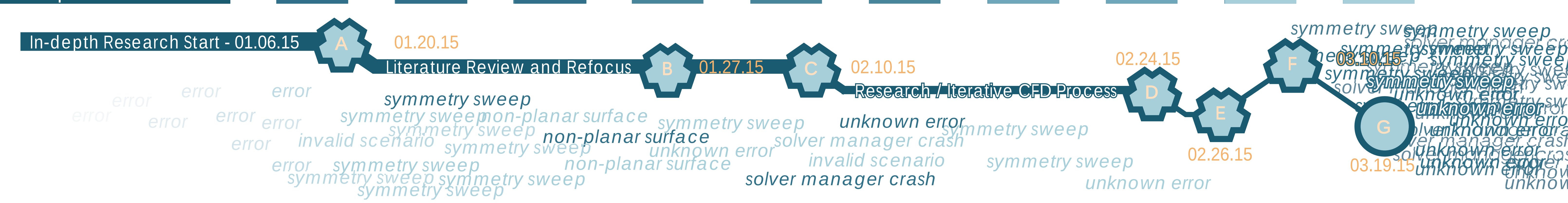
The Costantino Recreation Center (CRC) houses the offices for all head coaches, athletic training, and recreation and athletics administrative staff. The gymnasium holds three full-size basketball and four full-size volleyball courts, and can seat up to 1,500 for events. It was last re-modeled in 1990 to add a gymnasium, multi-purpose exercise room, office space and wellness center, and general recreation use by students, faculty, staff and members of the community.



Project Timeline

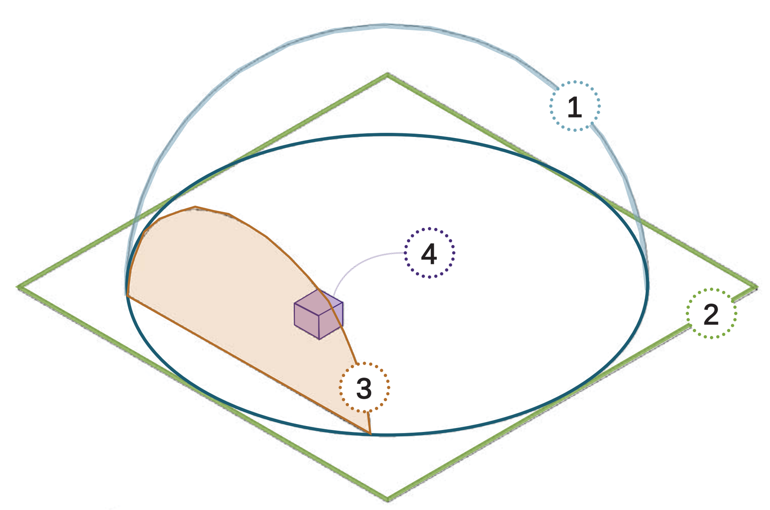
Project Start - 09.29.14 | Preliminary Research Complete - 12.11.14

1. CFD Case Study: CRC
2. CFD Literature Review
3. Wind Driven CFD Analysis
4. Bouyancy CFD Analysis
5. Combined CFD Analysis
6. Template Development



Significant Events: A - First team meeting, Discover Project is discontinued, lost one design team member, B - Decide to conduct Survey on "Rules of thumb" as basis for study, C - Lose another team member at YGH, Redirect away from survey in another direction, D - Suffer signifcant setback in development of integrated research model, lose another team member, E - Success, scenarios don't crash, F - Scenarios begin crashing again, G - Project Finshes.

Method

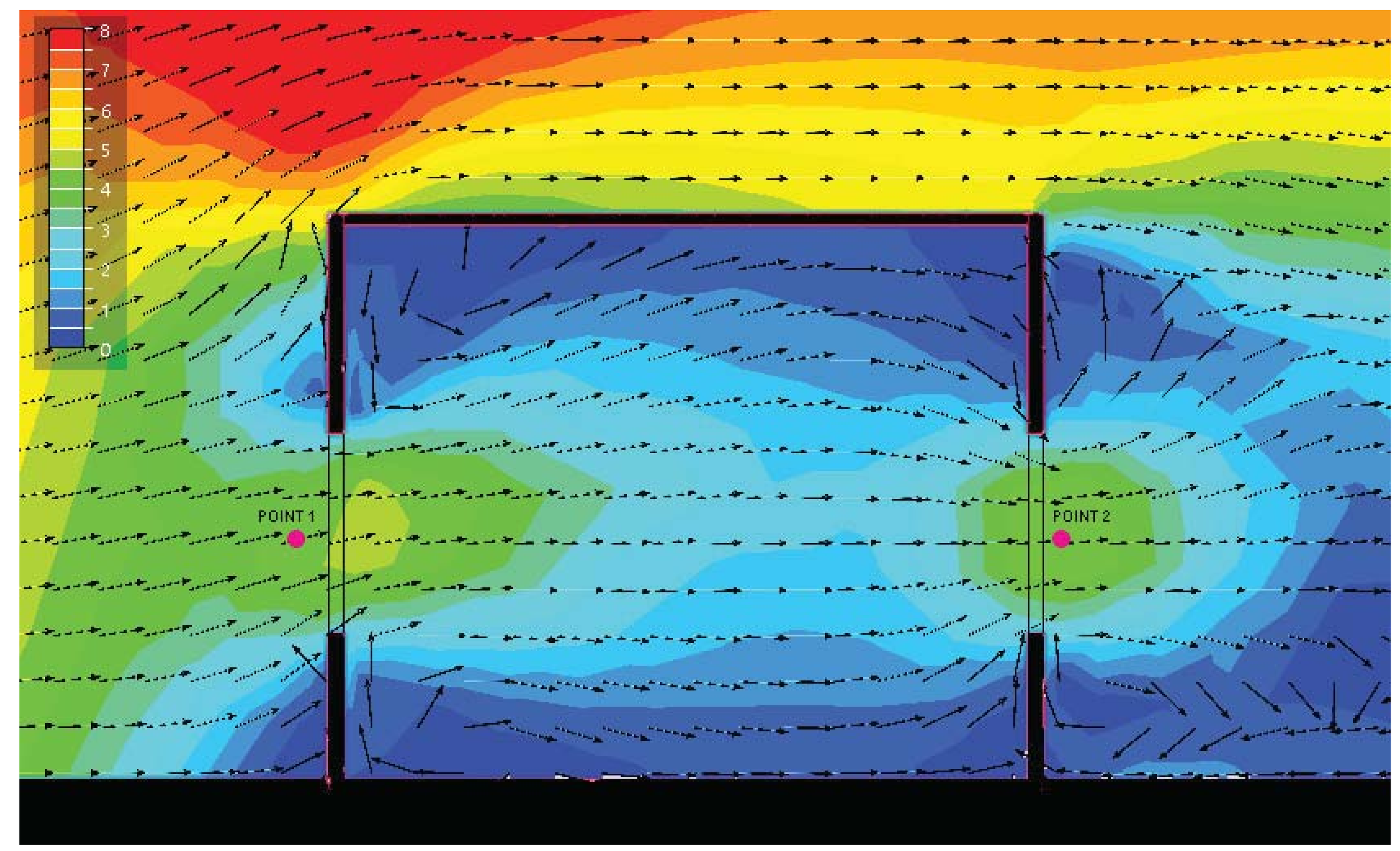


Simulation Model Setup. 1- Air Volume, 2- Ground Plane, 3- "Wind Wall, 4- Building location.

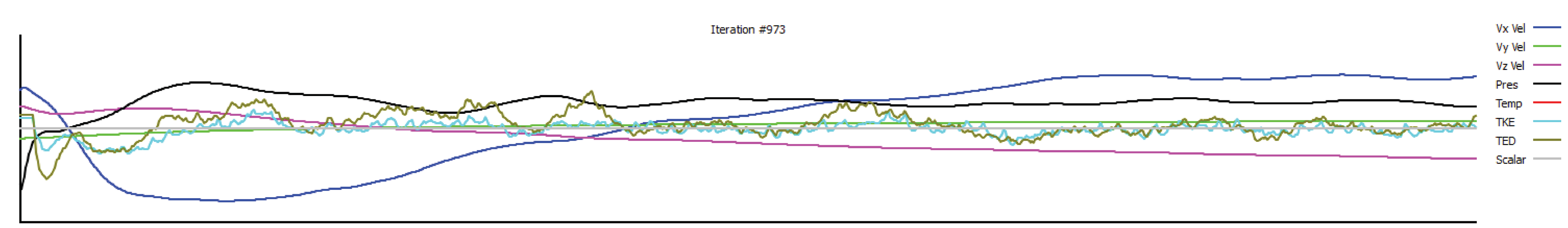
The method used to understand the effectivenesss and accuracy of CFD can be summed into a gradual expansion of complexity. The first iteration dealt with gain accurate understandings of given wind variables, which was based on YGH's the Evergreen State College Recreation Center. Once wind conditions seemed stable, the test was reiterated by using a generic model. The wind conditions and siting of the structure were used to ensure the accuracy of the wind as a standalone test on a generic model based on the study by Ramponi & Blocken (2012) . Then radiant heat and heat transfer were used to accurately reflect bouyancy driven ventilation. A generic model, set with the same conditions as the Recreation Center, with a solar chimney was used to see if the predicted results occured. Finally, the methods were combined on the same generic model with the chimney to observe the result of combined forces. Once the results were analyzed, the project progressed by attempting to develop templates that address the needs of the architecture firm to be able to test design iterations on real world projects.

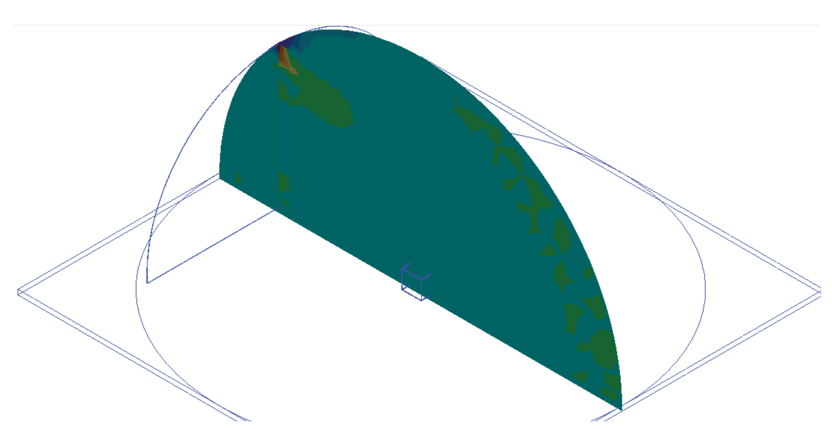
Results

The available results were both graphical and numerical in nature. The graphical data allows for an intuitive understand of relative scale among calculated data. This type of data applies well to the architectural profession, as it provides the flow quantity and direction generic terms as a basis for understanding design. By placing analysis points within the model, relevant numerical data could be extracted at each point. By reviewing the data, the study show what would be expected to happen for wind driven ventilation based on a basic understanding of physics and design "rules of thumb." In Figure 2, the results from the comparison of Simulation CFD and the previous generic study building can be seen graphically. The flow directions and speeds correspond to the data achieved in the previous study along the centerline of the opening (Ramponi & Blocken, 2012). For the purposes of the study, the available results were more compelling and useful graphically than numerically.

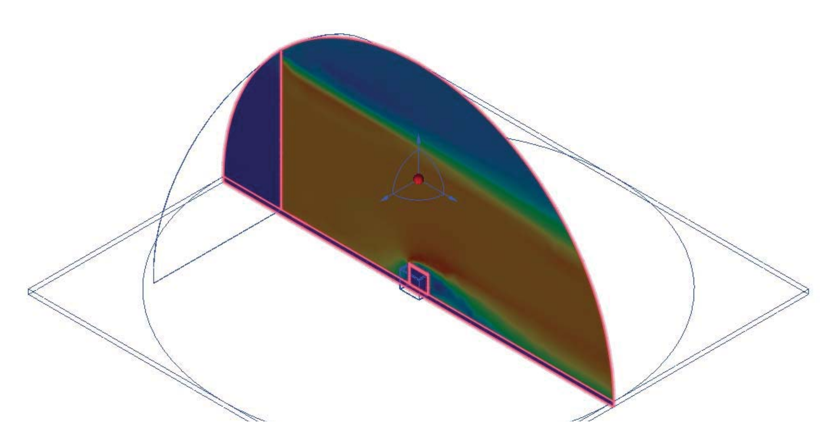


Above: Results of wind driven ventilation comparison study. Point 1 was placed centered at the inlet and Point 2 the outlet. The results are in wind velocity in m/s with the vectors indicating direction of travel in the XZ Plane. **Below:** Convergence plot of numerical data.

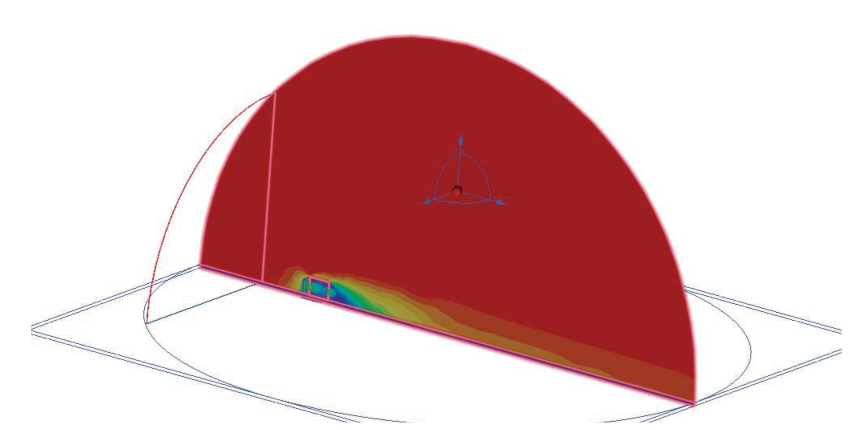




Only successful test of wind and buoyancy environment with Simulation CFD.



First successful test of the "wind wall" concept within Simulation CFD.



A wind study working in the mixed environmental setting within Simulation CFD.

Analysis

From a functional standpoint, the program worked to understand wind and buoyant ventilation conditions separately well. In climates that are dominated by one or the other, this tool should work successfully, with the assumption that the answer will not be completely accurate to actual conditions. In locations where there is not a significant force in one direction, or the other, the software becomes much less useful. Additionally, by verifying the CFD with architectural "rules of thumb" it is clear that they are based on the science and mathematics that drive the CFD program. This then indicates that firms using "rules of thumb" are already starting at a decent baseline for ventilation design without the program.

Conclusions

As the program develops these technical and training issues may be addressed, but as it stands Simulation CFD is a tool that works accurately and presents well, but is unusable at the architectural firm level. In continuing this research there are several points that need to be addressed. First, the errors and assumptions found within the program need to be fully understood and optimized as required. Secondly, a post-occupant study of a space with both wind and buoyancy driven ventilation should be conducted to verify the combined results within the program. If the post-occupant study proved the validity, or relative accuracy, of the calculations, then the next steps go from a research study to implementation within architectural firms. Because of the interface with existing 3D modelling software, it would be logical to develop templates for the firms.