Trends in the application of CFD for architectural design

Soo Jeong Jo[1], James Jones[1], Elizabeth Grant[1]

[1]Virginia Tech School of Architecture + Design, Blacksburg, Virginia

ABSTRACT: This paper is an overview of the trends in the application of Computational Fluid Dynamics (CFD) in architectural design. This paper aims to identify the current trends of CFD-related research in the architectural field and questions how CFD may interact with architectural design practice. To achieve the research objectives, a thorough literature review was conducted with two steps. First, relevant data were collected from journals and conference proceedings. The collected data were categorized according to the detailed topic of literature, such as designing HVAC and building envelope systems, evaluating indoor climates, simulating outdoor airflow, and developing early-stage designs. Based on the developed categories, we studied the trends of the CFD-related paper submissions for the International Building Performance Simulation Association (IBPSA) international conferences from 1997 to 2015. The results showed that the amount of CFD-related research has been constantly growing due to the increase of case-based research, and because the design process itself (including decision-making) has been an active research topic recently. Through in-depth literature review and trend analysis, we found that CFD-related research has been evolving in interaction with architectural design practice, and that the boundary of the research has expanded from evaluation of built environments to include the early stages of design.

KEYWORDS: CFD, architecture, design, application, literature review

1.0. INTRODUCTION

Computational Fluid Dynamics (CFD) is a numerical methodology used to simulate the movement of fluid including airflow. By using partial differential equations, CFD calculates the properties of airflow such as direction, speed, pressure, and temperature. Compared to conventional hand calculation or wind tunnel testing, CFD provides prompt predictions concerning airflow. While CFD has been widely used in aerospace and automotive engineering since the 1970s, it also has a great potential for architectural design. The potential may include structural load testing to include resistance to lateral winds and wind uplift forces, and design for natural ventilation in buildings. Airflow has a direct impact on designing a building form, opening, or space.

The close relationship between architectural design and airflow may explain why airflow diagrams consistently appear in architectural design competitions and discourses. However, some airflow diagrams in architectural design are driven by the architects' intuition, which is often different from reality. For example, the Tjibaou Cultural Center in New Caledonia is a well-known building touted as an ecological design, but researchers proved that the airflow in and around the building was different from the designed path (Wu et al. 2011). Also, Parkinson (2015) reported that Londoners complained to the city about the unintended ground level wind generated by the Walkie Talkie Tower. These papers show that airflow does not behave as predicted in many cases. Airflow is unpredictable because detecting turbulence generated by the wind in interaction with obstacles is a complex process. In other words, intuition may not be sufficient to achieve credible prediction of airflow. Wind tunnel tests can compensate for this problem, but due to their cost and required time, few architectural projects with critical airflow issues include wind tunnel tests.

Utilizing CFD could improve the situation because it offers "meaningful data visualization" which can help architectural designers to make better design decisions (Roudsari and Pak 2014, 3128). Due to its benefits, CFD was introduced in the architectural field in the 1990s (Zhai 2006, 311). In its infancy, CFD-related research in the architectural field focused on numerical methods and mechanical systems. However, the characteristics of architectural design have since been reflected in the research, and accordingly, the research direction has evolved. CFD-related research now has various trends influenced by its interaction with the architectural field.

This paper aims to reveal the current direction of CFD studies in architectural research. Through the review of various research trends, we also question how CFD interacts with architectural design practice. Specifically, we focus on the needs of architectural designers, and the reflection of their needs in CFD-related research. In brief, the objectives of this study can be summarized as follows:

- Reveal the trends in CFD-related studies for architectural research
- Find the interactions between CFD and architectural design practice

To achieve these goals, papers addressing a CFD-related topic published in renowned building research journals and conferences were collected and reviewed in depth to define the categories of major research trends. The categories defined by this process were used to identify the flow of CFD-related research trends in the architectural field.

2.0. BACKGROUND

CFD was introduced in the architectural field in the 1990s, and the number of CFD-related studies has been constantly growing since 1997 (Zhai 2006, 311). The growth implies that researchers recognized the potential of CFD, and thus, an interest in CFD-related topics permeated the architectural field. However, there have been barriers to implementing CFD in architectural practice. Den Hartog, Koutamanis and Luscuere (2000) pointed out the lack of case-based studies that allow architectural designers to be connected with CFD. Their arguments are reminiscent of Nigel Cross' (2006) statement that designers learn when they test solutions in a real-world setting, whereas scientists learn through analytic studies. Since architectural designers are trained to visualize information and produce projects, a case-based approach with visualized data may best speak their language.

The lack of knowledge concerning fluid mechanics principles has been one of the barriers as well. Using CFD requires knowledge of the underlying principles that can be translated into effective procedures. But, concepts in fluid mechanics can be difficult for architectural designers, who are often visually oriented, to understand. Passe and Battaglia pointed out the literature gap between engineering and architectural design; various references on CFD exist, but the existing literature addresses the issue "in a very technical manner" containing "mathematical formulae many architects are ill-equipped to incorporate for lack of knowledge, patience, or time" (2015, 6). To overcome the barriers to implementing CFD in the architectural field, researchers have focused on case-based studies and simulation methods for simplifying the tedious simulation process. Those efforts have become major trends of CFD studies in architectural research. These trends will be discussed further in the Results section of this paper.

3.0. METHODS

Document analysis, also known as content analysis, stems from research "that uses archival materials as data" (Silverman and Patterson 2014, 95). The strengths of this method are the ease of data access and the opportunity for replication. Through the document analysis process, the information in various formats is reduced, reorganized, and interpreted by the researcher. Therefore, document analysis is not a simple review of existing materials, it is a creation of data interconnections and a movement toward "meaningful understanding" (Groat and Wang 2013, 246). Similarly, Okoli and Schabram (2010) asserted that literature review using document analysis methods may become a stand-alone study with new findings or provide an overview of certain topics for other researchers or practitioners.

Using the document analysis method, Zhai (2006) investigated the trends of CFD studies in building research by tracking the number of CFD-related papers presented at the International Building Performance Simulation Association (IBPSA) international conferences from 1985 to 2003. Similarly, Blocken et al. explored outdoor airflow simulations and categorized the literature into four types: "pedestrian wind environment around buildings"; "wind-driven rain on buildings"; "convective heat and mass transfer coefficients at exterior buildings services"; and "air pollutant concentration distributions around buildings" (2009, 489-493). Likewise, document analysis was the main research method of the study presented in this paper. Moreover, the methodology of Zhai and Blocken et al. were referenced to structure our study into data collection, categorization, and data analysis stages.

The data collection was conducted in two steps, defining categories and checking trends. First, we collected literature through search engines including Google Scholar, Science Direct, and Engineering Village. The keywords for the search were "CFD," "Computational Fluid Dynamics," "airflow," and "wind." After selecting papers with titles or abstracts having these keywords, redundant or irrelevant papers were eliminated. An irrelevant paper, here, meant that its methodology did not include CFD. For example, if a paper discussed airflow issues without using CFD, it was eliminated. A thorough literature review was conducted to define categories of CFD-related studies in architectural research. The literature review was summarized with the following categories and subcategories:

- Support for the application of CFD
- Application of CFD
    - HVAC system analysis
    - Indoor climate analysis
    - Outdoor airflow analysis
    - Building envelope analysis
    - Design decision-making process

While Zhai (2006) defined four CFD-relevant trends from 1985 to 2003 focusing on the mechanism of CFD, such as simplification of the CFD interface and development of modeling methods, the categorization above focuses on the application of CFD similar to the literature review by Blocken et al. (2009). This difference in categorization is due to the significant increase of case-based application research after Zhai's studies in 2003, while the mechanism of CFD was the main research topic prior to 2003.

After setting the categories, the second step was checking the trends of the studies based on the categories defined in the first step. The sample of the second step was limited to papers submitted for the International Building Performance Simulation Association (IBPSA) international conferences from 1997 to 2015. Within this boundary, the number of papers that belonged to each category was counted each year to reveal the research trends. The reason for setting this limit to conference proceedings was that conference proceedings are more time-sensitive, thus, they reflect ongoing studies promptly. Although journal articles are often considered more refined versions of these studies, they usually require some time for publication.

The reasons for setting the limit to IBPSA international conferences were: 1) to confine the size of sampling for clarity; 2) to utilize the association's abundant publications and open-source database from 1985. IBPSA is a major association specializing in building simulation. This association holds biannual international conferences that are among the best-known events in the building simulation field (Zhai 2006, 311). Therefore, observing the research topics presented at the IBPSA international conferences can demonstrate the trends of building simulation studies, which include CFD. The reason for limiting the time of publication after 1997 was that there were insufficient data before 1997 since the growth of CFD-related research started in 1997 (Zhai 2006, 311).

4.0. RESULTS AND REFLECTION

The collected papers were categorized into two main streams: support for the application of CFD and the application of CFD in architectural projects. These two streams are explained in section 4.1 and 4.2.

4.1. Support for the application of CFD

Researchers have argued that the complexity of CFD has interrupted the dissemination of the tool in the architectural field and have responded to this problem with various solutions (Passe and Battaglia 2015, Kim 2014, Kaijima et al. 2013, Menacha-B and Glicksman 2008, Broderick and Chen 2001). The solutions can be summarized as: 1) numerical solutions; 2) interface development; 3) creation of guidelines.

Numerical solutions are the traditional type of CFD research focused on equation models. In architectural research, *Chen and Srebric* (2000) simplified the process of CFD by improving Reynolds Averaged Navier-Stokes (RANS) equation models and Large Eddy Simulation (LES) models. Similarly, Zhao et al. (2001) developed a new "zero equation turbulence model" which yields closer calculation results to experiments. With the enhancement of computing power, the numerical research boundary has extended to algorithms and computing systems. Zhang et al. (2010) suggested a prototype mesh generation tool for the CFD simulation of architectural projects; Zuo and Chen (2010) adopted the fast fluid dynamics (FFD) model utilizing the Graphic Processing Unit (GPU) for CFD simulation instead of the existing system based on the Central Processing Unit (CPU). Numerical solutions can reduce simulation time and enhance the simulation accuracy, thus, making CFD more palatable and accessible to architects.

Researchers have also developed various interfaces of CFD that are simplified and graphically oriented for architectural designers. Broderick and Chen developed a software package called Simplified CFD Interface (SCI). The purpose of the software development was to provide a "public domain program that allows architects and building engineers to use CFD without excessive training" (2001, 1). To create a user-friendly tool, the authors emphasized three elements that should be simplified: the settings for the boundary condition, the interface, and the compatibility with architectural graphic software. To achieve these goals, the authors reduced mesh options and set realistic default values for simplifying the boundary-setting process.

The efforts to develop a simplified interface have continued after the publication of the SCI. Malkawi et al. (2005) introduced a platform where CFD, Genetic Algorithm (GA), and visualization are incorporated. Their study adopted an “iterative approach” for supporting the design process (2005, 33). Moreover, Menacha-B and Glicksman developed CoolVent software that required only “bulk characteristics” of the building and allowed users to adjust the boundary conditions according to the type of the target site (2008, 133). In a similar context, Kajima et al. (2013) developed a toolkit for the visualization of CFD. In the toolkit, the authors attempted to connect CFD and architectural design software. Likewise, the Autodesk company released a built-in CFD feature in Revit software (Kfouri 2012).

Setting the best practice guidelines for CFD is also a solution suggested by researchers (Tominaga et al. 2008, Franke et al. 2010, Kim 2014, Den Hartog, Koutamanis, and Luscuere 2000, Schmid and Burrell 2004). Hartog et al. introduced CFD as a “new technique in building design” (2000, 165). After stating various possibilities for using CFD for indoor climate analysis, the authors explained each step of the simulation. Also, Schmid and Burrell (2004) explained basic information about CFD and described the procedure of the simulation with an example project. Both of these articles allowed readers to see the simulation procedure from a non-expert’s perspective, thus, the articles could serve as CFD guidelines for architectural designers.

Expanding the range of CFD application, Tominaga et al. (2008) suggested a CFD guideline for simulations at the urban scale, which was adopted by the Architectural Institute of Japan (AIJ). Similarly, Franke et al. proposed a guideline for the simulation of urban environments, particularly for “micro-scale obstacle-accommodating meteorological models” (2010, 1). On the other hand, Kim (2014) used a qualitative approach to set up a CFD guideline specifically for architectural designers by utilizing Star-CCM+ software. Kim pointed out the lack of qualitative research on CFD and emphasized the necessity of guidelines enabling architectural designers to easily apply CFD in their projects. His immersive case studies demonstrated how the knowledge gained through literature review could be applied to real-world projects.

The literature in this stream shows that CFD-related studies in architectural research tend to consider the characteristics of the architectural design. For example, Broderick and Chen (2001) set the default values for the simulation to mitigate the uncertainty of the early-design stage. Malkawi et al. (2005) observed the iterations in the architectural design process and adopted an iterative approach for their algorithm. All these efforts demonstrate how the features of architectural design can be reflected in CFD-related research.

4.2. The application of CFD in architectural projects

CFD can support analysis of a built environment as well as decision-making in design. Researchers have conducted case-based studies to show these capabilities of CFD and have demonstrated realistic outcomes. In case-based research on CFD, indoor climate analysis linked with HVAC systems used to be the most prevalent topic. However, new movements in the architectural field have extended the scope of CFD-related studies to include particular subjects such as natural ventilation, vernacular architecture, and human interactions with indoor air. Outdoor CFD analysis and its application in the design decision-making process are also novel trends.

Indoor climate study has been a popular research topic since the emergence of CFD in architectural research. Sinclair (2001) studied the movement of fire smoke in an atrium space by using CFD to find fire-prone areas in the building. Ji et al. (2007) also worked on an atrium space, but focused on natural ventilation in the space. Asfour and Gadi (2008) simulated natural ventilation in a domed roof, a typical element in classical architecture. Kristianto et al. (2014) studied natural ventilation in the Minahasa traditional house in Indonesia by simulating airflow around its raised floor. In contrast to this trend, other researchers concentrated on heating and cooling systems rather than natural ventilation. Webb (2013) evaluated the Under Floor Air Distribution (UFAD) system of an office building by using CFD and energy simulation; Moustafa and Aripin (2014) tested a pottery water wall system for cooling a place in Luxor, Egypt. Taking another approach, Malkawi and Srinivasan (2005) included the users as a part of their simulation and measured human interactions with airflow in a built environment by utilizing CFD.

CFD outdoor airflow analysis is also a major trend in architectural research. For example, Gousseau et al. (2011) tested pollutant dispersion in downtown Montreal, and Montazeri et al. (2013) simulated the airflow around a 78m high-rise tower in downtown Antwerp to evaluate a façade design of the tower. Wu, Hung, and Lin (2013) studied the wind environment of a community space in a residential complex in Taiwan. By using CFD, Wu, Hung, and Lin proved that adding a plaza around the building perimeter would create more exterior cooling. In their research, the results of the simulation were used to support the authors’ suggestion to design more open space for future projects.

Wu et al. (2011) also utilized CFD to detect a problem and suggest a design improvement. The authors simulated the air movement in and around Renzo Piano's Tjibaou Cultural Center building in New Caledonia. Then, the simulation results were compared to the diagrams drawn by the architect during the early stages of design. Wu et al. found that the airflow in the simulation was different from the diagram. Emphasizing the need for a more accurate tool to support the architectural design process, Wu et al. suggested an improvement to the Tjibaou Cultural Center design, which could change the airflow to the direction that the architect initially intended (2011, 2795). This study is notable because: 1) CFD simulation was compared to an intuitive, diagrammatic analysis; 2) CFD simulation was the catalyst to move from the problem identification to the design solution.

With a further movement toward the early stages of architectural design, Van Hoof et al. (2011) utilized their simulation results to set up a guideline for designing a stadium. The authors created twelve design options for the roof and stands of a stadium based on the Club AZ football stadium plan. Then, by using CFD, they measured the flow of the wind and the stand area wetted by the wind-driven rain to find an optimized shape for the stadium stands and roof.

In a similar context, Janssen, Bloken, and Hooff studied the influences of a high-rise building on the wind comfort of pedestrians in an urban setting. Based on field measurement data and CFD simulation results, the authors detected the points with a high probability of wind discomfort (2013, 1922). After determining the problematic points, they suggested adding a canopy to the building entrance, which could mitigate the discomfort. Providing the wind condition of each canopy design option, the authors utilized CFD to find the optimum size of the building canopy (2013, 1923). In this simulation, the authors removed the turbulence at the east pedestrian access by increasing the canopy size. Van Hoof et al. and Janssen, Bloken, and Hooff started to engage CFD in the creative design process, whereas the previous role of CFD was closer to an evaluation tool for existing conditions.

In recent research, architects tend to use CFD more actively in the design decision-making process (Kaijima et al. 2013, Guo, Liu, and Yuan 2015, Kozlovsky and Grobman 2017). Kaijima et al. (2013) introduced an experimental project that used CFD in the early stages of design. The authors designed a bus-stop canopy in Singapore, and by manipulating the shape of the canopy, they intended to generate a natural airflow. By utilizing CFD, Kaijima et al. optimized the shape of the canopy. While Kaijima et al. focused on the building form in relation to CFD, Guo et al. (2015) worked on space planning in interaction with CFD. The authors performed simulations for three space planning options for an art gallery in Guangzhou. Based on the simulation results, Guo et al. found the best design option for natural ventilation. In another example, Kozlovsky and Grobman (2017) applied CFD to their seawall design decision-making process. They contended that esthetics should be considered further in seawall design since seawalls are significant components in landscape and urban design as well as functional protectors from the sea. For them, CFD was a decision-making tool to find a design that met both functional and esthetic needs.

4.3. Trends of CFD applications in architectural research

The CFD-related research topics presented at the IBPSA international conferences from 1997 to 2015 were counted and categorized as shown in Table . In the table, the "support for the application of CFD" category includes simplification of the CFD modeling process and coupling with other tools. The studies in this category tend to focus on simplification of the simulation process by developing numerical methods, scripting, and interfaces. The "application of CFD in architectural projects" category includes testing conditioned or naturally ventilated environments, or specific façade and roof systems, as well as the decision-making process supported by CFD.

According to Table , the number of CFD-related research topics has been constantly growing. This growth has occurred mainly due to the increase of case-based research in which CFD is applied to evaluate architectural systems or projects. While the analyses of indoor and outdoor environments are the mainstream of case-based research, the recent emergence of design decision-making research is remarkable.

This table has limitations in that it is based on one data source and that relevant studies having a title or abstract without any of the selected keywords could be excluded from the count. Therefore, the counted numbers in the table cannot be considered absolute, rather, this table is intended to provide an overview of the research trends when comparing the numbers relatively.

Table 1: The number of CFD-related topics presented at the IBPSA international conferences from 1997 to 2015.

Year\Topic	Application of CFD in Architectural Projects	Total

	Support for Application of CFD	HVAC System	Indoor	Outdoor	Building Envelope	Design Process	Total	
1997	2	1	2	0	0	0	3	5
1999	2	0	0	2	0	0	2	4
2001	6	0	0	0	0	0	0	6
2003	5	1	1	2	1	0	5	10
2005	3	3	2	2	2	0	9	12
2007	4	0	4	4	2	0	10	14
2009	4	0	4	4	0	0	8	12
2011	4	2	6	2	0	3	13	17
2013	4	1	7	7	0	0	15	19
2015	4	1	4	4	2	3	14	18

5.0. CONCLUSION

CFD-related research in the architectural field has been evolving during the last decade. When CFD was introduced, it was largely situated within the discipline of engineering and not intended to be accessible to most architectural designers. However, the interaction with architectural design has expanded the boundary of the research. As a result, the current CFD-related research tends to be more interdisciplinary, and closer to architectural design practice. The research trends dealing with the simulation process itself have developed from a focus on numerical methods to include additional topics such as simplified interface development. Similarly, indoor climate research utilizing CFD has been diversified with studies that simulate vernacular buildings along with studies that address other emerging issues in the architectural field. Furthermore, the improvement of computing power enabled researchers to expand their study area to include CFD outdoor analysis, allowing iterative building designs to be tested through simulation. As in the study by Janssen, Bloken, and Hooff (2013), the increased interaction between CFD and architectural design brought about the involvement of the simulation results in the design decision-making process. In other words, the trends of CFD-related studies in architectural research have been expanding from quantitative to qualitative across the boundary of engineering and architectural design. Figure 1 summarizes these general trends of CFD-related research in the architectural field.

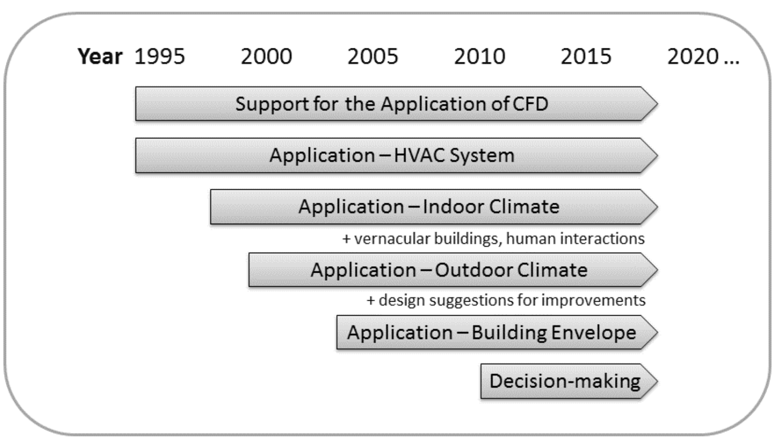

Figure 1: The expansion of CFD-related research areas in the architectural field.

These research trends show the potential of CFD as a tool for the early stages of design. Different from the conventional practice of using CFD to evaluate completed designs, these trends imply that CFD may be actively involved in the design decision-making process. The involvement of CFD may change the flow of the architectural design process as well.

In summary, we attempted to identify the major trends of CFD-related research in the architectural field by conducting an in-depth literature review and by analyzing the streams of relevant conference publications. Questioning how CFD research trends interact with architectural design practice, we found that the needs of current practice were reflected in the evolution of the research trends. Moreover, the boundary of the research has expanded to include early-design considerations, showing the potential of CFD as a tool for the early stages of design. As a future research direction, this paper could be improved by collecting data from further conferences, or could be developed into recommendations in connection with building codes.

REFERENCES

Asfour, Omar S, and Mohamed B Gadi. 2008. "Using CFD to investigate ventilation characteristics of vaults as wind-inducing devices in buildings." *Applied Energy* 85 (12):1126-1140.

Blocken, Bert, Ted Stathopoulos, Jan Carmeliet, and Jan Hensen. 2009. "Application of CFD in building performance simulation for the outdoor environment." Eleventh International IBPSA Conference.

Broderick, Charles R, and Qingyan Chen. 2001. "A simple interface to CFD codes for building environment simulations." Proceedings of Building Simulation, Seventh International IBPSA Conference.

Chen, Qingyan, and Jelena Srebric. 2000. "Application of CFD tools for indoor and outdoor environment design." *International Journal on Architectural Science* 1 (1):14-29.

Den Hartog, JP, A Koutamanis, and PG Luscuere. 2000. "Possibilities and limitations of CFD simulation for indoor climate analysis."

Franke, Jörg, Antti Hellsten, HA Schlunzen, and Bertrand Carissimo. 2010. "The Best Practise Guideline for the CFD simulation of flows in the urban environment: an outcome of COST 732." The Fifth International Symposium on Computational Wind Engineering. Chapel Hill, North Carolina, USA.

Gousseau, P, Bert Blocken, T Stathopoulos, and GJF Van Heijst. 2011. "CFD simulation of near-field pollutant dispersion on a high-resolution grid: a case study by LES and RANS for a building group in downtown Montreal." *Atmospheric Environment* 45 (2):428-438.

Groat, Linda N, and David Wang. 2013. *Architectural research methods*: John Wiley & Sons.

Guo, Weihong, Xiao Liu, and Xu Yuan. 2015. "A Case Study on Optimization of Building Design Based on CFD Simulation Technology of Wind Environment." *Procedia Engineering* 121:225-231.

Janssen, Wendy, Bert Blocken, and TV Hooff. 2013. "Use of CFD Simulations to Improve the Pedestrian Wind Comfort around a High-Rise Building in a Complex Urban Area." 13th Conference of International Building Performance Simulation Association.

Ji, Y, MJ Cook, and V Hanby. 2007. "CFD modelling of natural displacement ventilation in an enclosure connected to an atrium." *Building and Environment* 42 (3):1158-1172.

Kaijima, Sawako, Roland Bouffanais, Karen Willcox, and Suresh Naidu. 2013. "Computational fluid dynamics for architectural design." *Architectural Design* 83 (2):118-123.

Kfouri, Emile. 2012. "CFD Integrated into Revit." Autodesk Accessed 02/21. http://autodesk.typepad.com/bpa/2012/12/cfd-integrated-into-revit.html.

Kim, Daeung. 2014. "The Application of CFD to Building Analysis and Design: A Combined Approach of an Immersive Case Study and Wind Tunnel Testing ", Virginia Polytechnic Institute and State University.

Kozlovsky, Roy, and Yasha J Grobman. 2017. "The blue garden: coastal infrastructure as ecologically enhanced wave-scapes." *Landscape Research* 42 (5):439-454.

Kristianto, Mandau A, N Agya Utama, and Andhy Muhammad Fathoni. 2014. "Analyzing indoor environment of Minahasa traditional house using CFD." *Procedia Environmental Sciences* 20:172-179.

Malkawi, Ali M, and Ravi S Srinivasan. 2005. "A new paradigm for Human-Building Interaction: the use of CFD and Augmented Reality." *Automation in Construction* 14 (1):71-84.

Malkawi, Ali M, Ravi S Srinivasan, K Yi Yun, and Ruchi Choudhary. 2005. "Decision support and design evolution: integrating genetic algorithms, CFD and visualization." *Automation in construction* 14 (1):33-44.

Menacha-B, Maria-Alejandra, and Leon Glicksman. 2008. "Coolvent: A multizone airflow and thermal analysis simulator for natural ventilation in buildings." *IBPSA-USA Journal* 3 (1):132-139.

Montazeri, H, Bert Blocken, WD Janssen, and Twan van Hooff. 2013. "CFD evaluation of new second-skin facade concept for wind comfort on building balconies: Case study for the Park Tower in Antwerp." *Building and Environment* 68:179-192.

Moustafa, Moustafa Anwar, and Srazali Aripin. 2014. "CFD evaluation of the pottery water wall in a hot arid climate of Luxor, Egypt." *Journal of Green Building* 9 (4):175-189.

Okoli, Chitu, and Kira Schabram. 2010. "A guide to conducting a systematic literature review of information systems research."

Parkinson, Justin. 2015. "The problem with the skyscraper wind effect." *BBC News Magazine*.

Passe, Ulrike, and Francine Battaglia. 2015. *Designing Spaces for Natural Ventilation: An Architect's Guide*: Routledge.

Roudsari, M, and Michelle Pak. 2014. "Ladybug: A Parametric Environmental Plugin for Grasshopper to Help Designers Create an Environmentally-Conscious Design." 13th International Conference of the International Building Performance Simulation Association (BS2013), Chambery, France.

Schmid, Ferdinand, and Galen Burrell. 2004. "CFD Analysis Challenges in Building Simulation." SimBuild 2004, IBPSA-USA National Conference, Boulder, Colorado, August 4-6, 2004.

Silverman, Robert Mark, and Kelly L Patterson. 2014. *Qualitative research methods for community development*: Routledge.

Sinclair, Ray. 2001. "CFD simulation in atrium smoke management system design/Discussion." *ASHRAE Transactions* 107:711.

Tominaga, Yoshihide, Akashi Mochida, Ryuichiro Yoshie, Hiroto Kataoka, Tsuyoshi Nozu, Masaru Yoshikawa, and Taichi Shirasawa. 2008. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of wind engineering and industrial aerodynamics* 96 (10):1749-1761.

Van Hooff, Twan, Bert Blocken, and M Van Harten. 2011. "3D CFD simulations of wind flow and wind-driven rain shelter in sports stadia: influence of stadium geometry." *Building and Environment* 46 (1):22-37.

Webb, Matthew. 2013. "Building energy and CFD simulation to verify thermal comfort in under floor air distribution (UFAD) design." Proceedings of 13th International IBPSA Building Simulation Conference, Chambéry, France.

Wu, Kang Li, I An Hung, and Hsien Te Lin. 2013. "Application of CFD Simulations in Studying Outdoor Wind Environment in Different Community Building Layouts and Open Space Designs." Applied Mechanics and Materials.

Wu, Yu-Chou, An-Shik Yang, Li-Yu Tseng, and Chin-Lung Liu. 2011. "Myth of ecological architecture designs: Comparison between design concept and computational analysis results of natural-ventilation for Tjibaou Cultural Center in New Caledonia." *Energy and buildings* 43 (10):2788-2797.

Zhai, Zhiqiang. 2006. "Application of computational fluid dynamics in building design: aspects and trends." *Indoor and built environment* 15 (4):305-313.

Zhang, Rui, Yongjie Zhang, Khee Poh Lam, and David H Archer. 2010. "A prototype mesh generation tool for CFD simulations in architecture domain." *Building and Environment* 45 (10):2253-2262.

Zhao, Bin, Xianting Li, and O Yan. 2001. "Simulation of indoor air flow in ventilated room by zero-equation turbulence model." *JOURNAL-TSINGHUA UNIVERSITY* 41 (10):109-113.

Zuo, Wangda, and Qingyan Chen. 2010. "Fast and informative flow simulations in a building by using fast fluid dynamics model on graphics processing unit." *Building and Environment* 45 (3):747-757.