VALIDATION OF COMPUTATIONAL FLUID DYNAMICS (CFD) PLATFORMS FOR THE EARLY STAGES OF ARCHITECTURAL DESIGN

Abstract

Airflow simulation using computational fluid dynamics (CFD) during the architectural design process can support decision-making in the early stages. However, due to cost and complexity, the use of CFD is limited to consultant engineers or researchers, and not readily used by architectural designers. With the growing power of computers, researchers have developed more user-friendly platforms for CFD that target the early stages of design. The goal of this study is a validation of the platforms through a comparative analysis. Autodesk CFD and Flow Design are selected for this study due to their simplicity and interoperability. The results of the study demonstrated that Autodesk CFD and Flow Design simulations showed agreement with the experiments despite the simplicity of the software.

Authors

Soo Jeong Jo, James Jones,
and Francine Battaglia
Virginia Tech, Louisiana State University,
University at Buffalo

Keywords

Airflow, architectural design, building performance simulation, CFD, early-stage

Introduction

Computational fluid dynamics (CFD) refers to numerical methods to solve the complex three-dimensional nonlinear partial different equations that govern the movement of fluids. In the building industry, CFD has been used for various engineering purposes. CFD also has potential as a creative design tool such as manipulating wind loads that influence the formative process of a building and designing natural ventilation that affects facade design. However, the users of CFD in the industry have been limited to engineers or researchers. Not many designers experienced CFD because of the required computer memory, storage, and time for the simulation, as well as the complexity of the simulation process, which requires extensive knowledge (Kaijima et al., 2013; Kim, 2014; Passe & Battaglia, 2015).

To make CFD more accessible, there have been continuous efforts to simplify the simulation process (Broderick & Chen, 2001; Menacha-B & Glicksman, 2008; Roudsari & Pak, 2014). Early-design stages are a series of decision-making (De Wit & Augenbroe, 2002; Lawson, 2006; Macmillan et al., 1999) in which lots of information is undecided. This implies that simulation accuracy may be compromised here if the simulation results can support the decision-making.

In response, light-version CFD tools have been developed recently, which offers enhanced speed and user-friendly interfaces. However, utilizing the software without understanding the limitations may cause misleading results (He & Passe, 2015; Holzer, 2017). Therefore, the weakness of the tools must be carefully assessed and understood by the users. This paper presents a validation of two CFD platforms. Although validation of software has been a popular topic, not many studies on the early-stage simulations are available. The goal of this paper is to provide support for the further use of CFD in architectural design.

Methodology

Among the tools for early-stage simulations, this study evaluates Autodesk CFD and Flow Design. Autodesk CFD offers a plug-in for Revit, a well-known Building Information Modeling (BIM) software. Since BIM is growing fast in the current building industry, the interoperability with BIM makes the software promising. Flow Design is also a notable platform due to its simplicity. Although Flow Design does not have elaborate functions, its automated process allows untrained users to utilize CFD and receive prompt feedback. The comparison between the experimental results and the simulation results is the major focus of this study.

Comparative analysis is an important step in the validation of the simulation and models employed. Researchers validated various CFD software by comparing their simulation with wind tunnel test (WTT) measurements (Jiang, et al., 2003; Kim, 2014; Santiago et al., 2007; Waibel et al., 2017; Wang et al., 2018). While most scholarly articles focused on the validation of conventional CFD software such as Fluent, Vogel (1984) studied on Autodesk CFD by duplicating a measurement of turbulent flow with the software.

The WTT conducted by Karava et al. (2011) was selected for this study. The geometry of the WTT was an acrylic building model in 1/200 scale with openings on the windward and leeward sides. The dimension of the wind tunnel was 0.9 m × 1.54 m × 0.48 m, and the location of the model was: 1) 3H from the inlet to the windward wall; 2) 15H from the leeward wall to the outlet; 3) 5H for lateral sides, where H is the height of the model. The dimensions of the building are shown in Figure 1. The configurations of the openings in the simulated five cross-ventilation cases are shown in Figure 2. Each case has different placement of the openings on the windward and leeward sides.

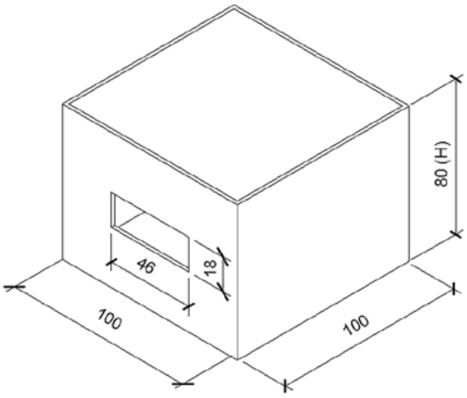

Figure 1: Dimensions of the building (unit: mm).

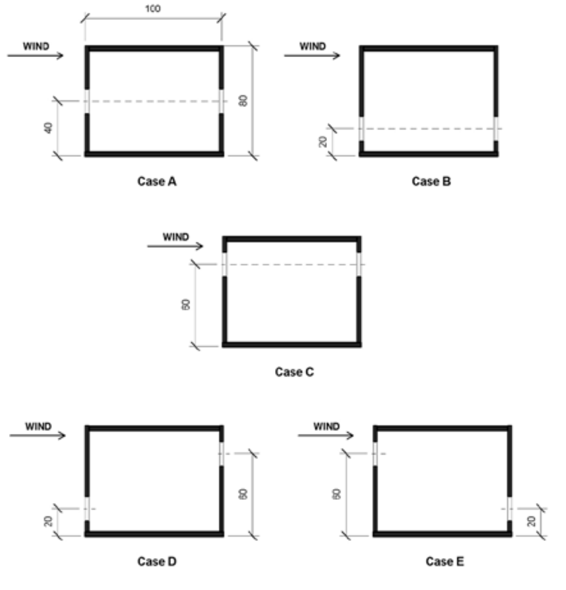

Figure 2: 2D view of the cross-ventilation cases (unit: mm).

The work by Ramponi and Blocken (2012) was the reference for the simulation settings in this study. The authors replicated the experiment by Karava et al. (2011) to validate their simulation approach using Fluent. The works herein followed their solver settings: steady-state 3D Reynolds-Averaged Navier-Stokes (RANS), and SST k-omega turbulence model. Although the large eddy simulation (LES) approach is known as a more accurate method, the RANS approach was selected after considering its efficiency and popularity in cross-ventilation studies. A mesh containing 580,000 grid cells was created for the simulation. The bottom domain surface was modeled as “open terrain” with friction, and the incoming wind speed at the building height (80 mm) was 6.97 m/s. Figure 3 illustrates the vertical profile of the inlet wind speed U and turbulence intensity Iu measured in the experiment. The vertical profile of the wind follows the logarithmic law in Equation 1.

$$U(z) = \frac{u^*}{\kappa} \ln\left(\frac{z+z0}{z0}\right)$$

u^*	friction velocity (0.363)
κ	von Karman constant (0.42)
z	height

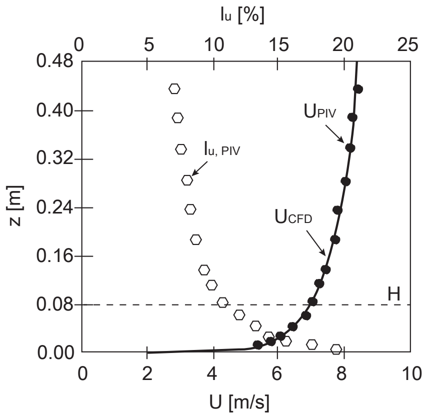

Figure 3: Vertical profile of the inlet wind speed and turbulence intensity.

In contrast to Autodesk CFD software, Flow Design limited the ability to select the solver methods and models, and the meshing process and material properties were automated without user-input. Furthermore, users could specify only a uniform mean speed at the inlet boundary. Therefore, Flow Design could not completely represent the referenced configuration. Moreover, one lateral wall of the building model had to be removed to see the interior airflow because sectional views of the results were not supported. Although this treatment may cause errors, it has been a popular indoor simulation method in the community of designers. Thus, it was included in this study to explore the consequences of this process.

For the data analysis, qualitative and quantitative methods were used. For the qualitative study, the predicted flowfield was presented in the form of velocity contour plots to compare with the vector plots of the airflow from the experiments. For the quantitative study, profiles of air velocity along the center axis of the opening were processed from the simulations. After overlapping and comparing the data from the CFD simulation and the experiment, 45 data points were extracted from each data set. The relationship between the simulation and the experiment was quantified by calculating the Pearson correlation coefficients. Correlation is a linear relationship between two data sets, and the Pearson correlation coefficient (r) is a statistical method to measure the level of correlation. The correlation coefficient varies from -1 to 1, where 1 is a perfect linear relationship, -1 is a perfect negative linear relationship, and 0 is no relationship. This method is useful for verifying the similarity between two data trends.

The reason for focusing on the trends of the data rather than the absolute accuracy would be the specificity of the early stages of design. In the early-stage simulations, it would be impossible to obtain precise simulation results as many parts of the simulation input would be undetermined. In contrast, the prediction of the changes due to the new design would be the most important requirement because the goal of the early-stage simulation is to test the impacts of different design options.

Results and Discussions

The results of the WTT and simulations are presented in Figure 4 at a section through the center-planes of the openings. In the WTT images in Figure 4, darker regions indicate higher air velocity. The air velocity near the openings (within 15 mm from the inlet, 5 mm from the outlet) is missing due to the errors caused by the reflections. The second and third columns in Figure 4 are contours of velocity magnitude from the simulations. The results of each case are as follows:

— **CASE A:** Both simulations showed agreement with the experiments for the roof and leeward side and predicted the separation along the roof. However, only Autodesk CFD correctly predicted the interior building flow.
— **CASE B:** The gradual flow separation along the roof was correctly predicted by Autodesk CFD. Flow Design predicted a more immediate separation and did not show the interior flow attaching to the ceiling.
— **CASE C:** The separation on the roof was replicated in both simulations, but only Autodesk CFD predicted downward flow at the outlet. Both simulations showed agreement with predicting the interior building flow.
— **CASE D:** Both simulations were similar to the experiments, but Flow Design did not predict the interior flow attaching to the building floor.
— **CASE E:** Autodesk CFD better predicted the interior and exterior flow. Flow Design could not predict the flow separation along the roof or the interior flow attaching to the ceiling.

In Figure 5, only Autodesk CFD results were compared to the experiments for cases A, B, and C because Flow Design did not have a customized x-y plot function. The graphs present the normalized air velocity, Ux / Uref (Uref = 6.97 m/s), measured at the center height of the openings. The plane for the measurement is indicated by the dashed line on the right side of the graph. The x-axis is the distance from the inlet to the direction of the outlet and normalized by D = 100 mm, which is the depth of the building.

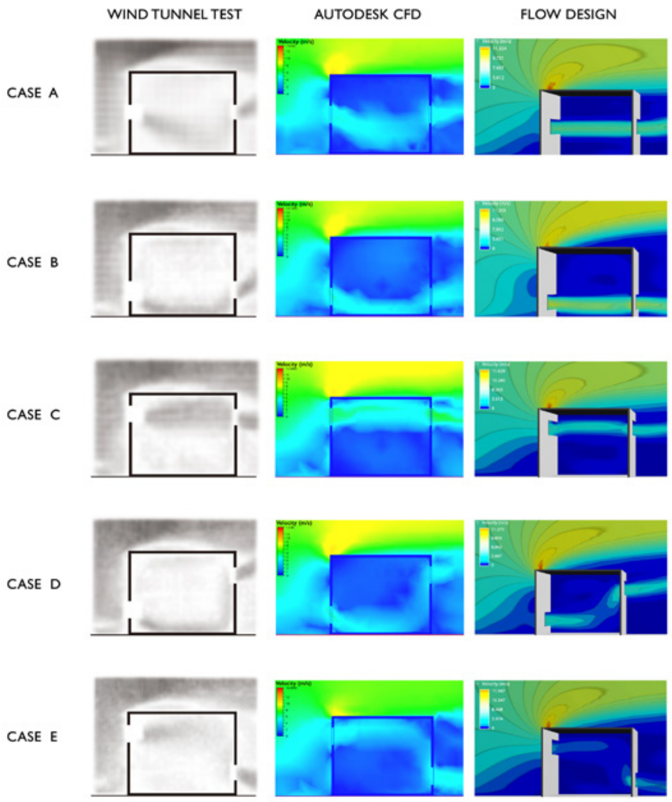

Figure 4: Comparison of WTT velocities with simulations using Autodesk and Flow Design.

Although the values from the simulations tended to be higher than the experiments, the overall trends show agreement. Table 1 summarizes the correlation coefficients of the simulated cases, which quantify the similarity of the trends between two data sets. Cases A and B, simulated by Ramponi and Blocken (2012) using Fluent, and the correlation coefficients are included in Table 1. Overall, Autodesk CFD simulations showed a reasonable similarity with the experiments, and the average correlation coefficient was around 75%.

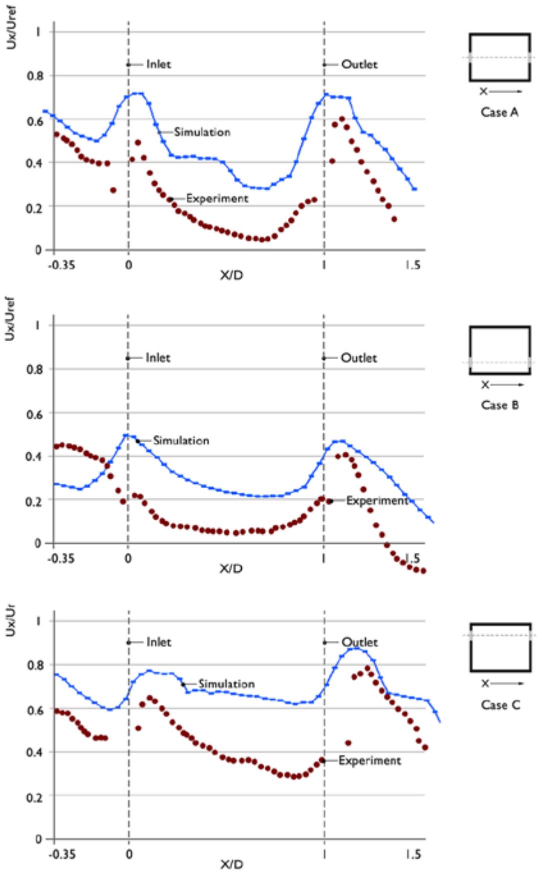

Figure 5: Velocity profiles—WTT (red dots) vs Autodesk CFD simulation (blue line).

	Autodesk CFD	Fluent
Case A	87%	95%
Case B	56%	72%
Case C	80%	data not provided

Table 1: Correlation Coefficient between WTT and CFD Simulations.

Conclusion

This study compared the airflow predictions from Autodesk CFD and Flow Design with experiments and showed the strengths and limitations of each CFD platform. In general, Autodesk CFD had good agreement with the experiments and performed better than Flow Design. Flow Design was also limited in specifying methods and parameters. However, Flow Design simulation showed reasonable agreement with the experiments. Also, its affordable cost and user-friendly interface that does not require an advanced level of expertise may compensate for its limitation.

Although the CFD simulation platforms introduced in this study had limitations compared to expert-version tools, the utilization of light-version CFD software is increasing in the community of designers. This study can be continued to evaluate other CFD platforms for designers. Since the development of light-version CFD software is a relatively new area in the market, the software options are currently in a state of flux. In the rapid-changing trends of simulation tools, third-party evaluations would support designers to find appropriate CFD platforms for their early-stages of design. The significance of this study is to show the strengths and limitations of different simulation tools that will ultimately help designers understand the potential and weakness of their simulation tools.

Acknowledgement

We thank Dr. Demetri Telionis who provided helpful comments on this paper.

References

Broderick, C. R., & Chen, Q. (2001). *A simple interface to CFD codes for building environment simulations.* Paper presented at the Proceedings of Building Simulation, Seventh International IBPSA Conference.

De Wit, S., & Augenbroe, G. (2002). Analysis of uncertainty in building design evaluations and its implications. *Energy and Buildings, 34*(9), 951–958.

He, S., & Passe, U. (2015). Architectural student's attitude towards building energy modeling: a pilot study to improve integrated design education. *Consultant, 12,* 13.

Holzer, D. (2017). *Optimising human comfort in medium-density housing via daylight and wind simulation.* Paper presented at the 22nd International Conference of the Association for Computer-Aided Architectural Design Research in Asia (CAADRIA), Hong Kong.

Jiang, Y., Alexander, D., Jenkins, H., Arthur, R., & Chen, Q. (2003). Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics, 91*(3), 331–353.

Kaijima, S., Bouffanais, R., Willcox, K., & Naidu, S. (2013). Computational fluid dynamics for architectural design. *Architectural Design, 83*(2), 118–123.

Karava, P., Stathopoulos, T., & Athienitis, A. (2011). Airflow assessment in cross-ventilated buildings with operable facade elements. *Building and Environment, 46*(1), 266–279.

Kim, D. (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.

Lawson, B. (2006). *How designers think: the design process demystified.* Routledge.

Macmillan, S., Steele, J., Austin, S., Spence, R., & Kirby, P. (1999). *Mapping the early stages of the design process—a comparison between engineering and construction.* Paper presented at the International Conference on Engineering Design, Munich.

Menacha-B, M.-A., & Glicksman, L. (2008). Coolvent: A multizone airflow and thermal analysis simulator for natural ventilation in buildings. *IBPSA-USA Journal, 3*(1), 132–139.

Passe, U., & Battaglia, F. (2015). *Designing Spaces for Natural Ventilation: An Architect's Guide:* Routledge.

Ramponi, R., & Blocken, B. (2012). CFD simulation of cross-ventilation flow for different isolated building configurations: validation with wind tunnel measurements and analysis of physical and numerical diffusion effects. *Journal of Wind Engineering and Industrial Aerodynamics, 104,* 408–418.

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

Santiago, J. L., Martilli, A., & Martín, F. (2007). CFD simulation of airflow over a regular array of cubes. Part I: Three-dimensional simulation of the flow and validation with wind-tunnel measurements. *Boundary-layer meteorology, 122*(3), 609–634.

Vogel, J. C. (1984). Heat transfer and fluid mechanics measurements in the turbulent reattaching flow behind a backward-facing step.

Waibel, C., Bystricky, L., Kubilay, A., Evins, R., & Carmeliet, J. (2017). *Validation of Grasshopper-based Fast Fluid Dynamics for Air Flow around Buildings in Early Design Stage.* Paper presented at the IBPSA International Conference.

Wang, J., Zhang, T., Wang, S., & Battaglia, F. (2018). Numerical investigation of single-sided natural ventilation driven by buoyancy and wind through variable window configurations. *Energy and Buildings, 168,* 147–164.