IOP Conference Series: Earth and Environmental Science

PAPER • **OPEN ACCESS**

Airflow Modelling Software Development for Natural Ventilation Design - Green Building Environment Simulation Technology

To cite this article: Hee Joo Poh *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **238** 012077

IOP Conf. Series: Earth and Environmental Science **238** (2019) 012077 doi:10.1088/1755-1315/238/1/012077

Airflow Modelling Software Development for Natural Ventilation Design - Green Building Environment Simulation Technology

Hee Joo POH[1,*], Pao-Hsiung CHIU[1], Hoang Huy NGUYEN[1], George XU[1], Chiet Sing CHONG[2], Lai Tee LEE[3], Ken PO[4] , TAN Phay Ping[4], Nyuk-Hien Wong[5], Ruixin LI[6], Sui Fung LEE[6], Ngian Chung WONG[6]

[1] Institute of High Performance Computing, Agency for Science, Technology and Research (A*STAR), 1 Fusionopolis Way, #16-16 Connexis, Singapore 138632
[2] Right Dimension Pte Ltd, 73 Upper Paya Lebar Road, Centro Bianco #08-01L,Singapore 534818
[3] Latitude Architect , Unit 05-01, Mayo Street, Singapore 208315
[4] Building System & Diagnostics Pte. Ltd, No. 18 Boon Lay Way #07-107, Tradehub 21 Singapore 609966
[5] Department of Building, National University of Singapore, 4 Architecture Drive, Singapore 117566
[6] Building and Construction Authority of Singapore, 52 Jurong Gateway Road, #12-01, Singapore 608550
[*] Corresponding author email: pohhj@ihpc.a-star.edu.sg

Abstract. We developed natural ventilation simulation tool to provide an integration of BIM and CFD simulation in one simple, time efficient and cost effective solution. There are three main working procedures in this *GrBEST* developmental work. Firstly, we work with an Architect to prepare a clean BIM-REVIT model (IFC output) for *GrBEST* flow solver. A modelling protocol for REVIT base geometry creation was developed to help create a clean BIM-REVIT model (IFC output), allowing for seamless conversion algorithm to convert the primitive REVIT IFC file for *GrBest* flow solver. Secondly, a unique geometry handling tool to perform BIM conversion to CFD input file has also been developed. It involves the development of a new triangular mesh repair solution that automatically rectifies common geometrical and topological errors that are inherent in the processes of CAD modelling and simulation. Finally, we use an in-house CFD code and customize it for building ventilation simulation. *GrBEST* Flow Solver consists of UNIX executable modules to conduct meshing and incompressible CFD computation activities. Solver enhancement with immersed boundary method for building geometry adaptation has also been developed. This *GrBEST* software could promote building designers and architects to implement good natural ventilation strategies during conceptual planning and master design stage, and therefore resulting in larger impact towards urban sustainability effort in new town development.

1. Introduction

The Singapore Building and Construction Authority (BCA) Green Mark Scheme (BCA, (2018)) was launched in 2005 and the latest version being evolved to promote passive and climatic responsive building design. Following up from this national effort, CFD simulation for natural ventilation study has becoming more popular among designers and consultants in Singapore. However, constraints associated with CFD studies have often prevented practitioners from bringing apparent value to building projects for the following reasons: 1) complexity, 2) turnaround time, 3) software cost and 4) hardware cost.

1) Complexity

Modelling and simulation tools are complex to use and usually require the domain knowledge and expertise of a CFD specialist. The requirement to obtain significant knowledge in a short time frame sometimes prevents architects who are involved in the design of a development from undertaking the study themselves. Moreover, engineers performing detailed CFD analyses are usually uncertain about the level of simplification required for modelling purposes without undermining the original design intent of the building.

2) Turnaround Time

The architectural design process evolves quickly and hence requires the airflow modelling and simulation analysis to be conducted in a timely manner for it to be relevant.

3) Software Cost

Commercially available software is generally costly to acquire and hence typically only well capitalized specialist companies, rather than smaller enterprises, can afford investment in applications and the necessary licences.

4) Hardware Cost

The computational time required to perform airflow simulation is often dependent on computational hardware, including, processing, memory, and data transfer resources. The larger and faster a CFD simulation, the greater the hardware cost required.

2. Literature Review

The very first work to use CFD for the simulation of airflow within building environments was made in the 1970s (Nilesen, 1974). Since then, CFD has been widely adopted by the architectural, engineering, and Heating, Ventilating and Air-Conditioning (HVAC) industries in building design, especially with the advancement of computational resources. Following this rapid development, the CFD community has formulated a design handbook and guidelines for the application of CFD to indoor and outdoor environmental airflow analysis (Tominagaa, Y, 2008, Blocken,B, 2012). The validation of CFD simulation results, based on certain test case studies, has been pursued extensively for both outdoor (Murakami, S., 2006) and indoor (Chen, Q, 2002) airflow studies. The difficulty in converting architectural geometry to a CFD-specific watertight computational domain necessary for mesh generation, has also been addressed (Zhang, R, 2010). In addition, research has been conducted on customizing CFD software for flow simulation in the urban built environment (Broderick III, 2000, Karola, A., 2002). Following up from this literature, a summary on the four challenging issues for developing and customizing a CFD software application to perform large scale flow simulation in the urban built environment are identified as follows:

1. Architectural geometry conversion to a watertight CFD-compatible geometry
2. Application usage workflow to reflect CFD modelling and simulation tasks
3. An efficient and accurate CFD solver for building airflow simulations
4. Comprehensive validation of CFD results using test case data.

3. Motivation and Objective

The Green Building Environment Simulation Technology (GrBEST) project aims to address the above constraint by developing an intuitive and cost-effective airflow modeling software for usage by the green building industry. The software enables master planners, architects, sustainability consultants, BCA Green Mark officers and general green building practitioners to perform timely CFD analyses for detailed green building conceptual design and assessment.

We seek to develop the GrBEST package into a robust, efficient and user friendly airflow simulation tool for green building applications for both Residential Buildings (RB) and Non Residential Buildings (NRB), with the following features and attributes:

i. Collaborate with architect in order to prepare a "clean" BIM-Revit model (IFC output) for GrBEST flow solver
ii. Geometry conversion on BIM Revit model - Development of a new triangular mesh repair solution that automatically rectifies common geometrical and topological errors that are inherent in the processes of CAD modelling and simulation
iii. Web-based GrBEST client for model submission
iv. Immersed Boundary method for Building Geometry Adaptation - new solution could be computed much faster than that for the re-meshed model

Figure 1 below summarizes the work flow of GrBEST, with automated process from geometrical modelling towards simulation and assessment.

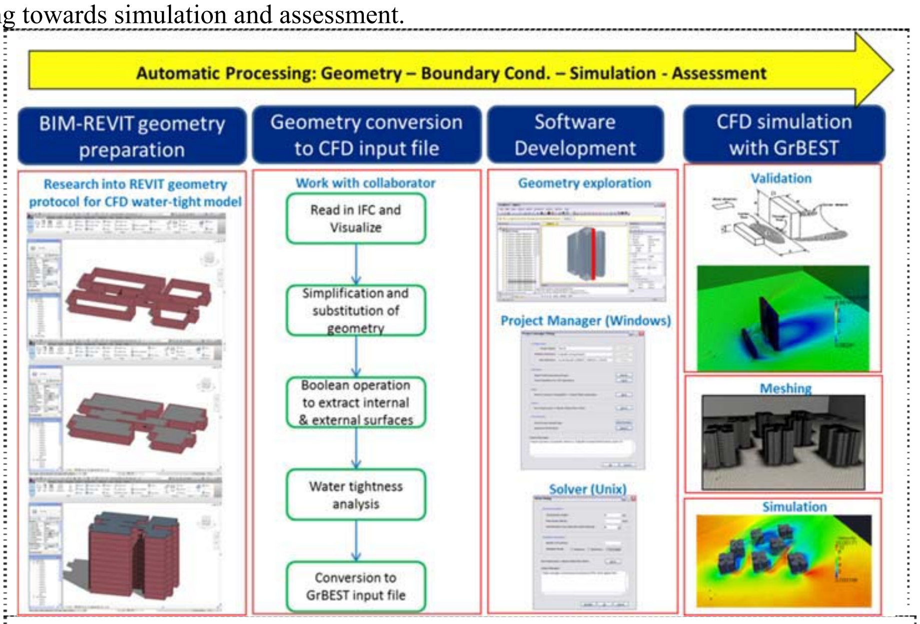

Figure 1. Work flow of GrBEST process.

4. Results & Discussion

Preparation of clean BIM-REVIT model (IFC output) for GrBEST flow solver

Our team work together with architect to develop AUTODESK REVIT 2016 Modeling for Computational Fluid Dynamics User Manual in order to generate a clean and readable building geometry data for CFD mesh generation. It provide step by step tools to guide a user how to develop a clean BIM-Revit Model (IFC output), which help the user to perform seamless conversion algorithm to convert the primitive REVIT IFC file for GrBEST flow solver. Figure 2 shows the BIM-Revit geometrical modelling protocol to simplify the building envelop with surrounding building in order to facilitate the external wind flow simulation.

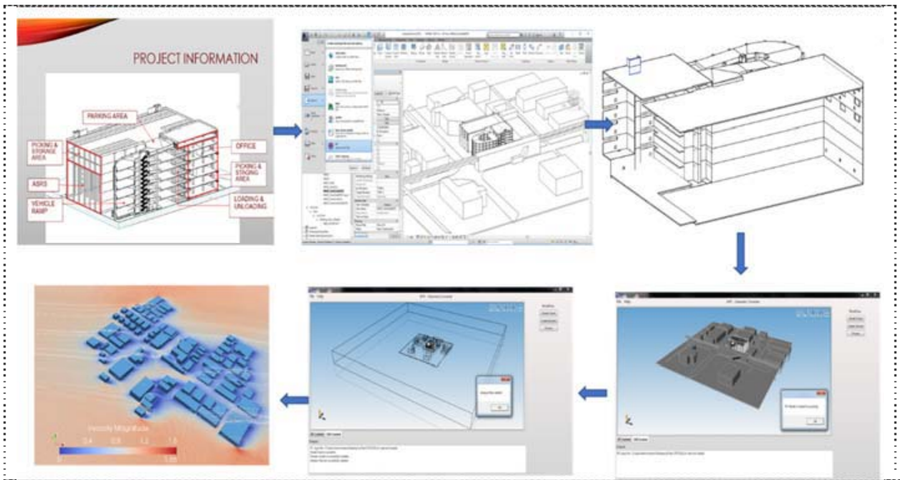

Figure 2. BIM-Revit geometrical modelling for GrBEST CFD simulation.

Geometrical conversion of BIM-REVIT model for GrBEST flow solver

Research work has been carried out to improve the "mesh-ability" of the IFC output model from the geometrical conversion module of GrBEST. New mesh-based model optimization algorithms are developed to improve the triangulated BIM models in terms of building facades and external domain. It automatically rectifies common geometrical and topological errors that are inherent in the processes of CAD modelling and simulation. This methodology would reduce intensive manual intervention in the repair and re-creation of simulation models and thus provide a more robust conversion process. Figure 3 shows the overall process of the input BIM model converting into GrBEST solver-ready model.

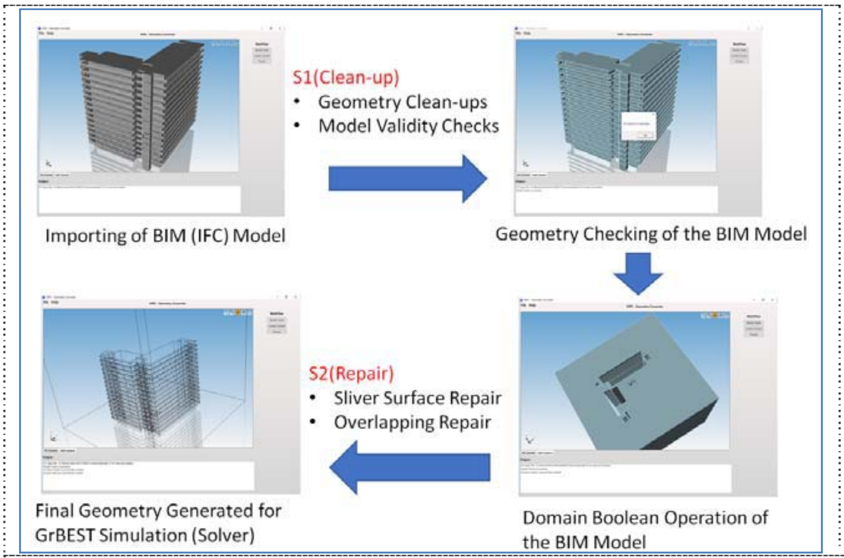

Figure 3. Overall BIM model enhancement and conversion process for GrBEST simulation.

Generation of web-based *GrBEST* client for model submission

A Web-based GrBEST client for model submission is developed. It's web-based application allows the easier integration of additional solver modules future R&D development. In addition, it does not require installation needed on user side, except for a downloadable tool for model verification and visualization before submission of BIM models for simulation. In Figure 4(a), a user can create simulation projects based on simulation type, e.g. Estate air-flow simulation, simulation of Turbulence models, modelling using immersed boundary method. The user will then attach a project with the BIM model and the related simulation parameters (non-graphical) before submitting it for simulation. This application can support multiple submissions and results will be updated on the user's dashboard. Figure 4(b) & (c) shows the automatic BIM model to solver model conversion and the running of simulation in the Linux solver when the simulation model is ready, respectively.

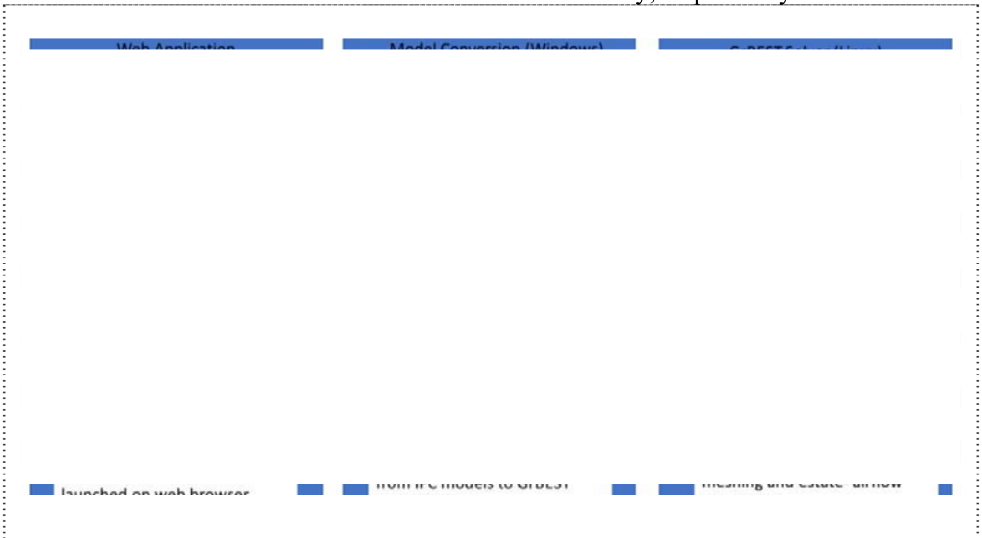

Figure 4. GrBEST web-based application.

Immersed Boundary method for Building Geometry Adaptation

For conventional CFD calculations, it is always necessary to generate the meshes in computational domain. If the domain contains complex geometry, especially for complicated building shape for natural ventilation analysis, it will take a lot of effort to generate the meshes. Our proposed immerse boundary method (IBM) can help to reduce re-meshing time associated with the changing of building geometry. Our IBM code has been developed and tested with external flow simulation carried out with Architecture Institute of Japan (AIJ) test case study (Yoshie, R, 2005).

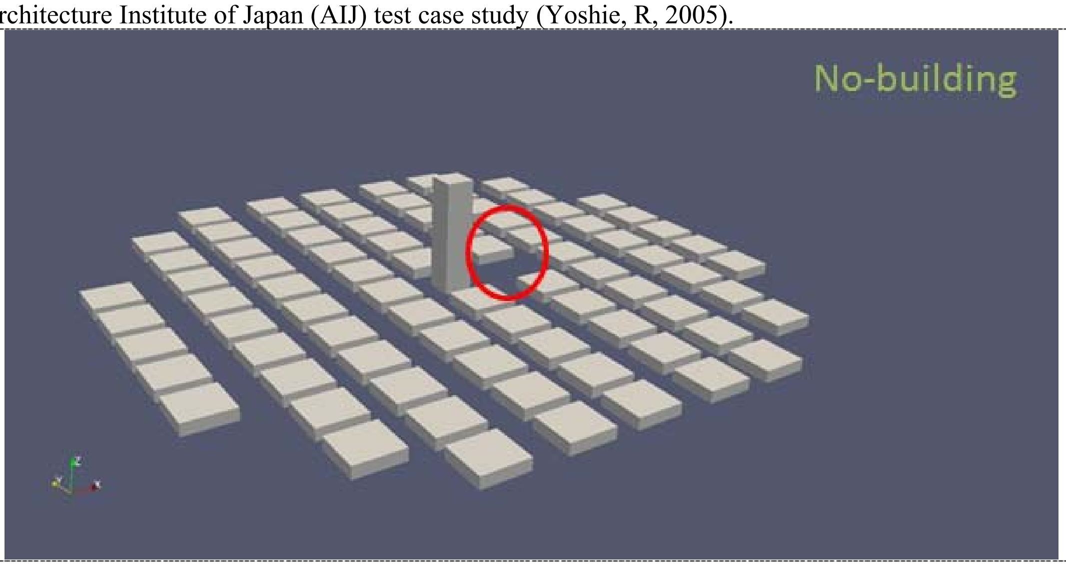

Figure 5. Schematic diagram of AIJ test case study on the building geometry.

In our simulation study, we model the flow past clusters of building with a building block which is behind the high-rise building by using the immersed boundary method (See Figure 5). With our present immersed boundary approach, (See Fig. 6), user just need to input the STL files as the newly introduced building blocks. The massive time for mesh generations can be reduced.

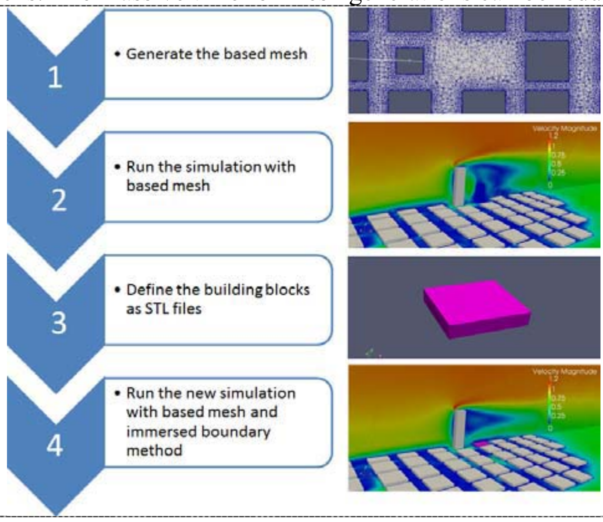

Figure 6. Advantage of immerse boundary method developed in GrBEST.

Comparison on the velocity field simulation with immersed boundary method and conventional method is shown in Figure 7. It can be seen that good agreements with between this two methods have been clearly established, hence the applicability and efficiency is further confirmed.

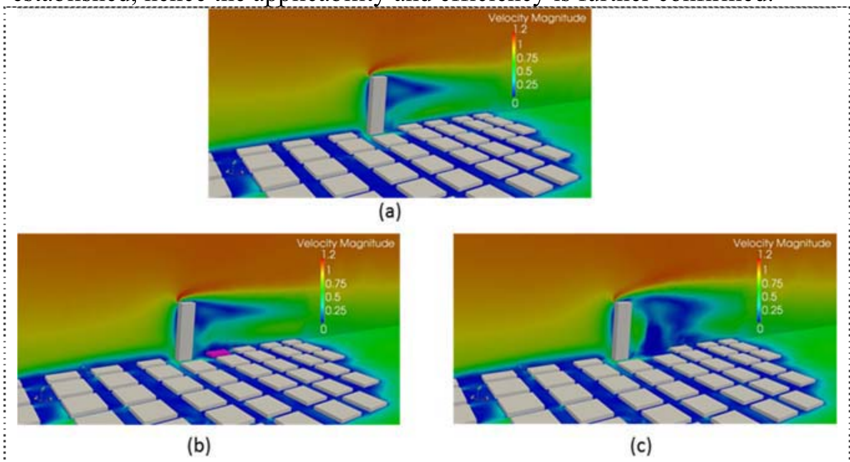

Figure 7. Predicted results using present GrBEST solver. (a) Convectional method; (b) With immersed boundary method; (c) Without immersed boundary method. Note that purple block is not explicitly included during the simulation.

5. Conclusion

We develop the airflow modelling software in order to provide an integration of BIM and CFD simulation in one simple, time efficient and cost effective solution. It is an enabling technology for the planners, architects, ESD consultants and general green building practitioners to widely adopt CFD software for green building conceptual and detailed design. It is the Computational Fluid Dynamics (CFD) airflow modelling and simulation coupled with the geometry input from BIM-REVIT. CFD enables airflow simulation over an estate landscape and within the building interior to be conducted as a design optimization and assessment tool towards achieving a comfortable naturally ventilated environment in buildings in the Tropics. More importantly, this *GrBEST* technology can increase the productivity in Green Building design cycles and provide cost-effective and time-efficient CFD simulation software.

References

[1] Building and Construction Authority Green Mark Scheme, https://www.bca.gov.sg/greenmark/green_mark_buildings.html (accessed in 22 Oct 2018).
[2] Nielson, P.V., Flow in Air Conditioned Rooms, Ph.D. thesis, Technical University of Denmark, Copenhagen, 1974.
[3] Tominagaa, Y., Mochidab, A., Yoshiec, R., Kataokad, H., Nozue, T,. Yoshikawaf, M., and Shirasawac, T., AIJ Guidelines for Practical Applications of CFD to Pedestrian Wind Environment Around Buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96, 1749 – 1761, 2008.
[4] Blocken,B. and Gualtieri,C., Ten Iterative Steps for Model Development and Evaluation Applied to Computational Fluid Dynamics for Environmental Fluid Mechanics, *Environmental Modelling & Software*, Vol. 33, 1 – 22, 2012.
[5] Murakami, S., Indoor/Outdoor Climate Design by CFD based on the Software Platform, *International Journal of Heat and Fluid Flow*, Vol. 25, 849–863, 2004.
[6] Chen, Q. and Srebric, J., A Procedure for Verification, Validation, and Reporting of Indoor Environment CFD Analyses, *International Journal of HVAC&R Research*, Vol. 8(2), 201-216, 2002.
[7] Zhang, R., Zhang, Y., Khee, P.L. and Archer, H.D., A Prototype Mesh Generation Tool for CFD Simulations in Architecture Domain, *Building and Environment*, Vol. 45, 2253 – 2262, 2010.
[8] Broderick III, C.R. and Chen, Q., A Simple Interface to Computational Fluid Dynamics Programs for Building Environment Simulations, *Indoor Built Environment*, Vol. 9, 317–324, 2000.
[9] Karola, A., Lahtela, H., Hänninen, R., Hitchcock, R., Chen, Q., Dajka, S. and Hagström, K., BSPro COM-Server—Interoperability between Software Tools using Industrial Foundation Classes, *Energy and Buildings*, Vol. 34, 901 – 907, 2002
[10] Yoshie, R., Mochida, A., Tominaga, T., Kataoka, H., Yoshikawa, M., 2005. Cross Comparisons of CFD Prediction for Wind Environment at Pedestrian Level around Buildings. Part 1 Comparison of Results for Flow-field around a High-rise Building Located in Surrounding City Blocks. The Sixth Asia-Pacific Conference on Wind Engineering (APCWE-VI) Seoul, Korea, September 12-14, 2005

Acknowledgments

This research is funded by the Republic of Singapore's National Research Foundation (NRF) through Building and Construction Authority (BCA)'s Green Innovation Cluster (GBIC) R&D Grant, and MND Research Fund's 2-stage Innovation Grant (iGrant).