Optimisation of Complex Geometry High-Rise Buildings based on Wind Load Analysis

Estrado, Erron; Turrin, Michela; Eigenraam, Peter

Publication date
2020

Document Version
Final published version

Published in
Proceedings of the Symposium on Simulation for Architecture and Urban Design (SimAUD 2020)

Citation (APA)
Estrado, E., Turrin, M., & Eigenraam, P. (2020). Optimisation of Complex Geometry High-Rise Buildings based on Wind Load Analysis. In A. Chronis, G. Wurzer, W. E. Lorenz, C. M. Herr, U. Pont, D. Cupkova, & G. Wainer (Eds.), *Proceedings of the Symposium on Simulation for Architecture and Urban Design (SimAUD 2020)* (pp. 219-226). Society for Computer Simulation International (SCS).

Optimisation of Complex Geometry High-Rise Buildings based on Wind Load Analysis

Erron Estrado[1], Michela Turrin[2], and Peter Eigenraam[2]

[1]Delft University of Technology
Delft, Netherlands
erron_e@hotmail.com

[2]Delft University of Technology
Delft, Netherlands
{m.turrin, p.eigenraam}@tudelft.nl

ABSTRACT

Wind analysis for the structure of buildings is a challenging process. The increasing strength and frequency of wind events due to climate change only add higher demands. In addition, high-rise buildings are growing in number and include many of unconventional shape. Current methods used in practice for calculating structural wind response either do not account for these geometries, such as the Eurocode or are prohibitively time-consuming and expensive, such as physical wind tunnel tests and complex Computational Fluid Dynamics simulations. As such, wind loads are usually only considered towards the end of design. This paper presents the development of a computational method to analyse the effect of wind on the structural behaviour of a 3D building model and optimise the external geometry to reduce those effects at an early design phase. It combines Computational Fluid Dynamics (CFD), Finite Element Analysis (FEA), and an Optimisation algorithm. This allows it to be used in an early design stage for performance-based design exploration in complement to the more traditional late-stage methods outlined above. The method was implemented into a rapid and easy to use computational tool by combining existing plugins in Grasshopper into a single script that can be used in practice on complex shaped parametric high-rise building models. After developing the method and testing the timeliness and precision of the CFD, and FEA portions on case study buildings, the tool was able to output an optimal geometry as well as a database of improved geometric options with their corresponding performance for the wind loading allowing for performance-based decision-making in the early design phase.

Author Keywords

Computational Fluid Dynamics; Optimisation; Finite Element Analysis; Wind engineering; Parametric design; Computational design

ACM Classification Keywords

• Computing methodologies~Simulation evaluation • Applied computing~Architecture (buildings) • Applied computing~Computer-aided design

1. INTRODUCTION

The aim of this research was to develop a computational method to optimise a building's geometry based on the effect of wind on the structure that is sufficiently precise but also quicker than traditional methods such as wind tunnel testing. The method must be able to obtain rapid and dependable results so that it can be used in the early design phase to facilitate design decisions in situations where wind loading is, or should be, of concern. Thus, an array of design options is presented, and adjustments can be made when the detail level of the design is low rather than having to adjust internal structural members when the design is at an advanced stage leading to conflicts between structural, architectural, and mechanical layouts.

Forces due to wind can be critical for a building especially tall, slender towers and with the rise in technology building shapes are becoming increasingly complex. The main methods used in practice for calculating this structural effect of wind are by building codes, the Eurocode in Europe, wind tunnel testing and, to a lesser extent, in-depth CFD simulations but these methods have significant drawbacks.

The Eurocode (EN 1991-1-4:2005) however, only gives calculation methods for regular shapes such as rectangular or circular plan towers up to 200m high. Applying these methods to non-standard geometry buildings produces results that would undoubtedly be inaccurate as it is an approximation. Physical scaled-model wind tunnel testing is the most accurate method and can deal with any given shape. To obtain this accuracy, however, it requires a lot of effort from skilled technicians to properly set up the model, use the wind tunnel, and process the data. This can take a lot of time and expense. Thus, it is usually only done at a late stage for verification purposes.

Computational Fluid Dynamics (CFD) allows a user to simulate wind flow around a model and the effects on the body such as pressure and drag forces regardless of shape. While not so commonly used in professional practice for structural wind engineering it has been widely studied for many years [1-5]. These simulations can take a lot of time and computational power to complete especially when very

SimAUD 2020 May 25-27, Online

detailed meshes and more complicated settings are used to be as accurate as possible. Finite Element Analysis (FEA) software analyses the structural response of buildings and is widely used in practice. It can give very detailed results the level of which, and time needed for simulation depends on the software package and the settings used. Optimisation algorithms allow one to iteratively manipulate input variables until the resulting output value(s) of some algorithm or simulation reaches an optimum. Algorithms used for optimisation include metaheuristics which are widely applicable but require a lot of iterations to converge to an optimum [6], direct search methods which are efficient but not very robust [7], and model-based algorithms which use machine learning methods to give robust performance with few evaluations making it ideal for heavy simulations such as CFD and FEA [8].

The combination of these three technologies gives a Fluid-Structure Interaction based Optimisation (FSIO) method. This is a computational method that simulates wind flow and the resulting pressure on the façade of a building model and translates it to a structural Finite Element Model (FEM) that is analysed with FEA to obtain the resulting structural effects. The combination of CFD and FEA creates a Fluid-Structure Interaction (FSI) method. This is then combined with optimisation where the algorithm manipulates parameters that define the external geometry of a building model that is then analysed by the FSI algorithm and the output, in this case, the mass of the structure needed to resist deflection, is the objective that the optimisation minimizes.

This paper describes the development of this method by combining existing CFD, FEA, and optimisation software into a single computational tool. The testing of the method will focus not on absolute accuracy but on obtaining rapid results with enough precision that it can be used repeatedly and reliably in practice on 3D building models in an early design exploration stage as a complement to the more detailed traditional methods mentioned.

2. BACKGROUND

CFD is an iterative simulation based on the Navier-Stokes equations usually with turbulence model equations. It simulates the flow of a fluid, air in this case, and its impact on an object in its flow. There has been a lot of research into Computational Wind Engineering (CWE), i.e. the application of CFD for wind engineering problems, over the last three decades which is accelerated by the continued advancements in computer technology [2]. However, it is still not widely accepted by many codes as a definitive method for structural wind analysis.

For example, the Eurocode allows for "properly validated numerical methods" as a supplement to its calculation procedure [9] and the Architectural Institute of Japan (AIJ) has published a guide which gives advice on using CFD for wind engineering [4]. The Eurocode calculates wind force (F_w) using Equation 1 where $c_s c_d$ is the structural factor which accounts for the non-simultaneous action of peak wind pressures over the building face and the dynamic response of the structure due to turbulence, c_f is the force coefficient which accounts for the building shape, $q_p(z_e)$ is the peak velocity pressure at reference height z_e. A_{ref} is the reference area on which the force is acting. This equation requires the solution of up to 20 other equations and the reading of many values from tables and charts in the Eurocode which are often only provided for specific standard building shapes.

$$F_w = c_s c_d \cdot c_f \cdot q_p(z_e) \cdot A_{ref}$$

Equation 1: Wind force on a building according to EN1991-1-4:2005

The two most common types of turbulence model are Large Eddy Simulation (LES), which give time-dependent results, and Reynolds-Averaged Navier-Stokes (RANS), which give time-averaged results. Tamura, Nozawa [4] recommend LES for CWE purposes due to its accuracy, however, this is a very computationally expensive procedure. RANS models were used with a gust effect factor for along wind loads which gave results close to physical experiments. Clannachan, Lim [2] after testing various turbulence models on tall building models conclude that RANS is the most practical for typical structural engineering due its efficiency. While they conclude that CFD is not yet able to replace physical testing it has great potential complementary tool in early-stage structural design to assess a broad array of design alternatives.

CFD based optimisation, while applied extensively in aerospace, is rare in building design [10]. This, however, would have great benefits over typical trial and error approaches for finding optimal shapes. However, optimisation only compounds the computational expense of CFD thus, it only practical if a CFD evaluation takes at most a few hours [11]. Bernardini, Spence [11] used surrogate-model based optimisation along with 2D CFD and to find the optimum cross-section of a high-rise building that minimises lift force and drag coefficient with minimal optimisation evaluations. Chronis, Turner [12] used Fast Fluid Dynamics, a lower order CFD method that is faster but much less accurate, combined with evolutionary optimisation to

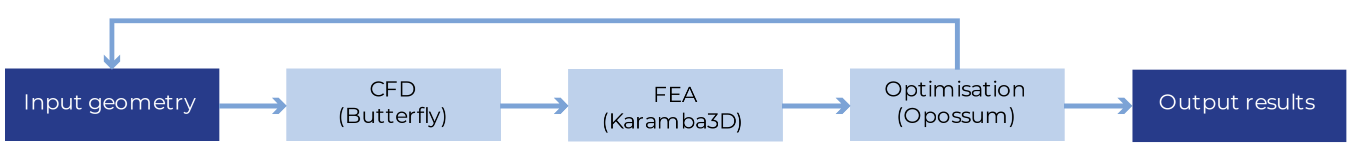

Figure 1: FSIO method flowchart

optimise the shape of a free-form surface based on surface pressure.

There are some shortcomings in CWE techniques compared to traditional wind tunnel tests particularly in modelling the complex airflows induced by buildings [2]. Nonetheless, the benefits of CWE over time-consuming and expensive physical tests continue to inspire more research in the field.

3. RESEARCH METHODOLOGY

The research methodology is divided into four parts: development, CFD validation and sensitivity analysis, FSI validation, and optimisation testing.

These tests were done using various case study buildings. The Commonwealth Advisory Aeronautical Research Council (CAARC) Standard Tall Building Model was used for CFD tests as there are results from physical wind tunnel tests to compare to [13]. In order to verify the method's effectiveness for non-standard geometry buildings, three such models of existing buildings were also used. These are the Absolute World Towers by MAD Architects, Jiangxi Nanchang Greenland Central Plaza by SOM, and Ardmore Residence by UN Studio. Each possesses unique geometric features that test the robustness of the tool and method.

3.1. Development

In this study, the proposed FSIO method (Figure 1) was implemented in Grasshopper, the visual scripting plugin for Rhinoceros3D. It involved the combination of 3 plugins: Butterfly, Karamba3D, and Opossum, with custom scripting in order to produce an easy to use tool where a parametrically defined building model can be input to obtain a collection of variations of that model based on structural performance under wind load.

Butterfly is a Grasshopper plugin and python library developed by Mostapha Sadeghipour Roudsari as part of the Ladybug Tools suite [14]. It allows users to run OpenFOAM CFD simulations from within the Grasshopper environment for cases pertaining to building design such as outdoor airflow. OpenFOAM is a free open source CFD software that has been widely used and validated [15]. Karamba3D is a parametric FEA plugin for Grasshopper developed by Clemens Preisinger in cooperation with Bollinger und Grohmann ZT GmbH. [16]. Opossum, developed by Thomas Wortmann [17] is a model-based optimisation algorithm. It uses Radial Basis Functions (RBF) to generate a response surface thus reducing runtime particularly for heavy simulations [8]. These plugins were selected based on their wide use, previous validation, quick performance, and ease-of-use.

The Butterfly portion was set up according to an outdoor airflow simulation. Modifications were made for the automatic setting of mesh cell size and size of the domain based on the building dimensions. Probe points for sampling pressure on the façade are also created automatically. An algorithm was written to translate the resulting static mean pressure values on a mesh of the building facade to vectors for point loads and moments which Karamba applies at 6 points to a fixed upright beam formed at the centreline of the building using the dimensions of its structural core creating the FEM. The analysis outputs results for deflection, mass of the structural model, forces, etc. which can be used as the objective for Opossum. The input variables are the numerical sliders controlling the shape of the model being analysed.

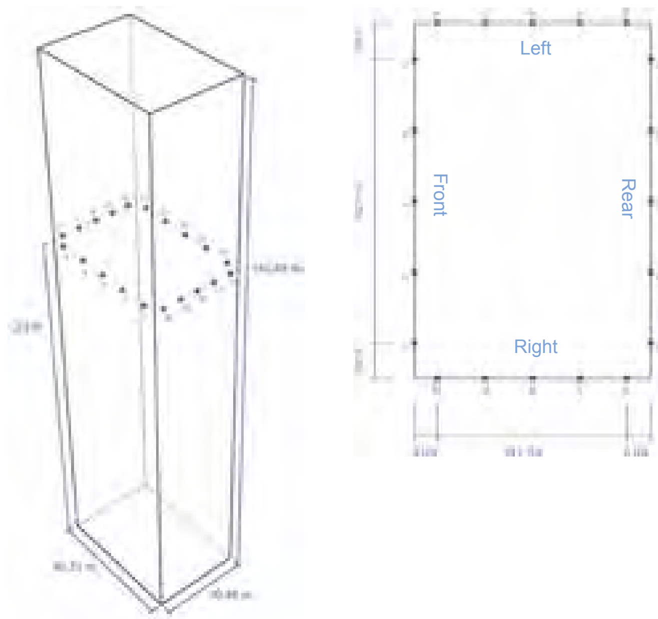

Figure 2: CAARC model CFD setup

Figure 3: Absolute Tower model CFD setup

3.2. CFD Validation and Sensitivity

A validation study was performed to ascertain the ability of Butterfly to give results within a degree of uncertainty to physical tests. An absolute deviation of ±20% is deemed acceptable for an early stage method as this [18]. For the comparison, the CAARC Standard Tall Building Model was used for which physical wind tunnel test results were obtained from Meng, He [18]. The digital model was set up just as the physical model where pressure coefficient (C_P) was measured at 20 points around the façade at 2/3 of the height and compared to those of the physical wind tunnel tests. The CFD domain size was 900 x 600 x 400 m (Figure 2) and the number of CFD iterations was set at 30000.

Three RANS turbulence models were tested: standard $k - \varepsilon$, Realizable $k - \varepsilon$ [19], and RNG $k - \varepsilon$ [20]. The $k - \varepsilon$ models solve equations for the turbulent viscosity of the flowing air by calculating the kinetic energy, k, and the turbulent dissipation rate, ε [2]. Other settings were kept constant. The resulting C_p obtained was compared between each of the turbulence models and the wind tunnel results at each pressure tap but also the overall shape of the graph was examined to determine its precision in capturing results at each area of the building. Lastly, these results were considered along with the time taken for the simulation to determine how efficient it is.

After a suitably accurate turbulence model was chosen from the validation a sensitivity analysis was performed. This involves changing one parameter while keeping others constant to see how much the simulation time can be reduced while maintaining sufficient precision and accuracy. In this case, the number of CFD iterations and mesh cell size were tested. To evaluate how much the number of iterations could be reduced and its effect on simulation time, simulations were done with 30000, 10000, and 5000 iterations using the chosen turbulence model. All other settings remained constant. The C_p was compared for each simulation to determine deviation. To test the effect of mesh cell size on precision and timeliness, simulations were done reducing cell sizes starting from a value equal to the length of the shortest side of the building divided by 10 [21] then successively reduced by a chosen value of $\sqrt{2}$ based on common practice [22]. This gives resolutions of Coarse, Medium, Fine, Super Fine and XXFine as defined in Table 1.

The building model used was the Absolute Tower. To check precision C_P was measured at 30 points around the façade to capture the effect of its complex geometry (Figure 3). The domain has the following dimensions: windward = 3H, Leeward = 10H, sides = 2.3H, and top = 2.3H where H is the height of the building [21]. Wind speed was set at 30 m/s corresponding to a violent storm on the Beaufort scale and terrain category was chosen for an urban site (roughness length = 1m).

3.3. FSI

This test sought to compare the values obtained from the Eurocode (EN) method to those obtained from the FSIO method implemented in the script. The Absolute Tower and Nanchang Tower models were used in this test. The wind force (F_w) at 6 heights along the building was calculated using the EN method and the FSIO method and compared. As the EN only gives guidance for regular shapes it was chosen to assume values for a circular cross-section for this calculation as these building models have a smooth cross-sectional shape. Basic wind velocity was taken as 30m/s, roughness length of 1m and A_{ref} was set at 1m^2. The areas to which the F_w would be applied was obtained in Grasshopper by finding the areas around the point loads of the building perpendicular to the wind flow. This was done for each of the tested case study buildings and the loads compared to those from the FSIO method.

3.4. Optimisation

The aim of these tests was to determine the extent to which structural performance due to wind can be optimised by making relatively small changes to the geometry of the building. The Ardmore Residence model was used for this study. The model was set up parametrically so that its geometry could be manipulated by the optimisation algorithm. The floor plan shape was built as a curve to be modified by three parameters. Two design variables control the position of each of the wings along the main body. A third slider modifies the edges of these wings from straight to a more angled position. From this curve outline, the massing was extruded to the 136m height of the building. In addition, the building was rotated 45 degrees so the wind would impinge on the building off-axis (Figure 4).

As CFD is very computationally expensive simulation settings were chosen in order to minimize the time for a single iteration. Thus, a coarse mesh setting was used and max iterations for Butterfly was set to 2000.

The output objective was the mass of structural material needed to resist the wind force. This was determined by modelling the core of the building as a hollow concrete (C25/30) beam approximately the dimensions of the core of the existing building.

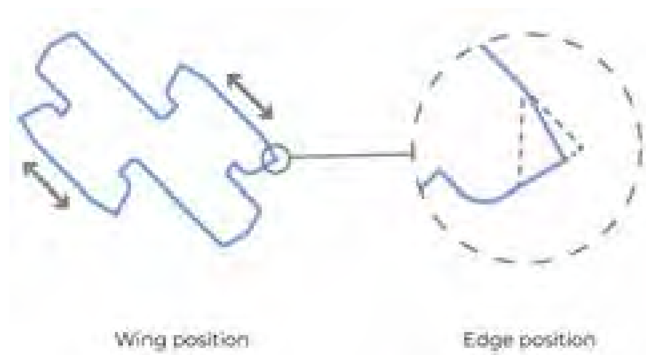

Figure 4: Ardmore Residence model parametric variables

The Optimize Cross-section component of Karamba was used to automatically select from a list of cross-sections for the core varying in thickness (length and width kept constant) that satisfies the deflection limit which was set at 0.66m based on an initial FSI simulation. The core of the building measured 11m by 7m and 0.4m thick. The lists of cross-sections range from 0.10m to 0.59m thick in 0.01m intervals.

In Opossum the RBFOpt optimisation algorithm was used and the max number of iterations was set at 100. The geometry created at each optimisation iteration and its corresponding objective value was recorded in Grasshopper to create a database of design options.

4. RESULTS

4.1. CFD Validation and sensitivity

The RNG $k-\varepsilon$ turbulence model showed the closest C_P values to the physical wind tunnel results (WT) from Meng, He [18] followed by standard $k-\varepsilon$ (Figure 5). The Realizable $k-\varepsilon$ model gave unrealistic C_P values the reason for which was not found and thus was omitted.

From Figure 5 the C_P values of all models are quite close at the front face of the building (pressure tap 1 - 5). On the sides and rear (6 – 20) the values deviate more. RNG $k-\varepsilon$, while with an absolute deviation of 25%, deviates much less from the wind tunnel values compared to standard $k-\varepsilon$ which has a more rounded graph shape. This is likely due to the poor performance of standard $k-\varepsilon$ in predicting regions of flow separation like the building rear and edges [4].

Timewise, RNG $k-\varepsilon$ took 42.6 hours to complete, standard $k-\varepsilon$ took 41.7 hours and realizable KE, 37.4 hours. RNG $k-\varepsilon$, therefore, proved to be the best choice due to its level of precision in capturing the flows at different areas of the building while taking not much more time than other models.

The timeliness of the RNG $k-\varepsilon$ model was further evaluated in the sensitivity analysis. At 10000 and 5000 CFD iterations, the C_P values obtained were nearly identical to that of the 30000-iteration simulation done previously while taking only 15.7 hours and 6.95 hours respectively to complete the simulations. From plotting a graph of the residuals, i.e. the scaled errors of calculated values between successive iterations, the values are sufficiently below the accepted threshold [21] at all tested numbers of iterations used in the validation study (Figure 6). Table 1 shows the settings used, the time taken, and absolute deviation at front and rear of the building from MAD_5, which is assumed to be the most accurate, for each run at differing mesh sizes. XXFine took the most time with 20.3 hours. The coarse mesh size has the lowest time of 5.7 hours. Absolute deviation increases as cell size decreases with the exception of the front at Medium size which is highest. At Fine resolution, the time is greatly reduced while deviation is still around 10% making it an ideal choice to balance time and accuracy for an early-stage study.

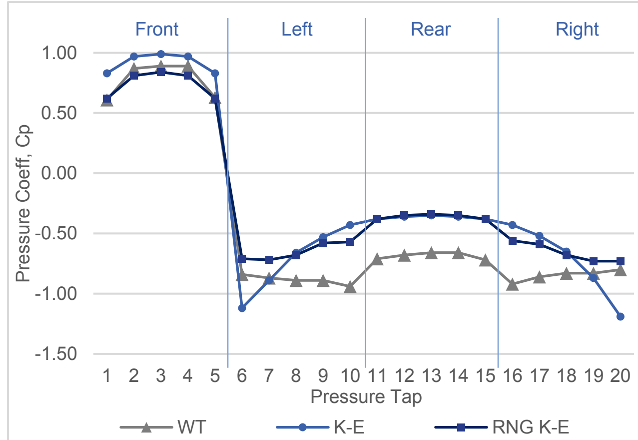

Figure 5: Graph of C_p at each pressure tap from each turbulence model and wind tunnel (WT) for CAARC model

Figure 6: Residuals graph of RNG $k-\varepsilon$ simulation

Test	MAD_1	MAD_2	MAD_3	MAD_4	MAD_5
Resolution	Coarse	Medium	Fine	Super Fine	XXFine
Cell size (m)	4.18	2.96	2.09	1.48	1.08
no. of cells	176545	236050	346647	525640	732422
Time	5.7h	6.3h	8.6h	14.5h	20.3h
Iterations	10000	10000	10000	10000	10000
Dev. Front	18.46%	28.08%	8.71%	0.77%	N/A
Dev. Rear	17.63%	15.36%	10.19%	1.46%	N/A

Table 1: CFD cell size sensitivity analysis results

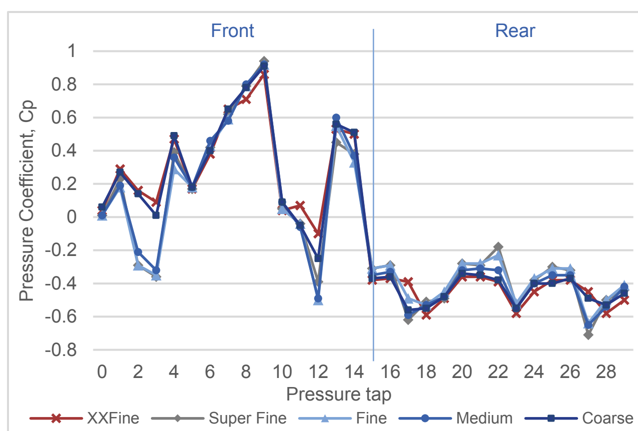

Figure 7: CFD mesh cell size sensitivity analysis

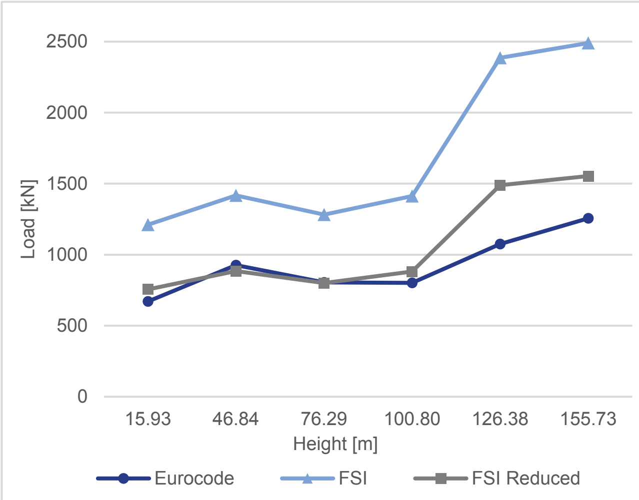

Figure 8: Absolute Tower EN/FSI comparison

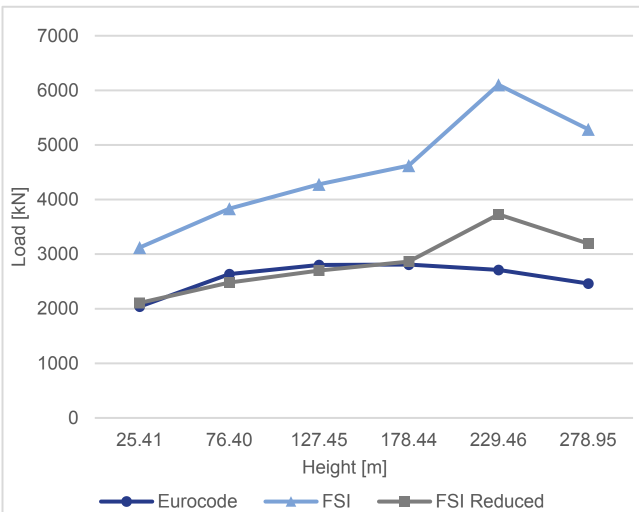

Figure 9: Nanchang Tower EN/FSI comparison

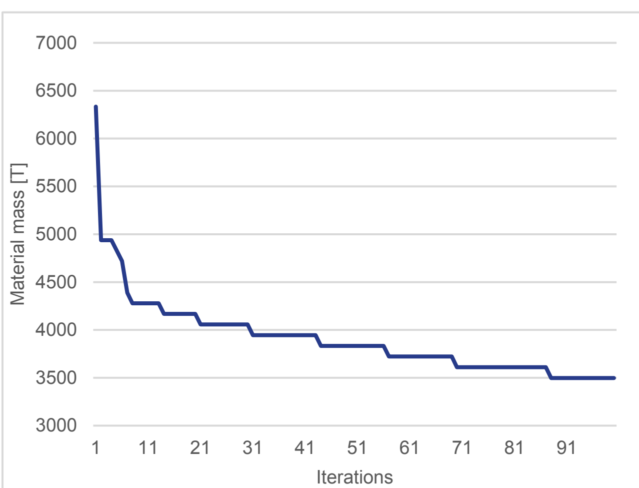

Figure 10: Mass of structural material per iteration

From Figure 7, while there is some deviation the C_p values generally follow the same trend. The exception being around the edges of the building near the twist (pressure tap 1 - 4 and 10 - 12) which can be reasonably assumed to be caused by issues in generating the mesh at those points for Medium and Fine resolution. The result shows that between mesh sizes precision can be maintained. Thus, at an early stage or for optimisation purposes a coarser, faster setting can be used to obtain quick and repeatable results that can reliably inform early-stage decision making. A finer resolution is suitable for a later stage as a final check.

4.2. FSI Validation

The results for the Absolute Tower model are shown in Figure 8. The EN numbers begin to rise then fall with respect to the decrease in the perpendicular area near the middle of the tower then rise again to a maximum value of 1256kN. The values from the FSI method follow a similar pattern but are much higher as CFD with RANS turbulence models like the RNG $k - \varepsilon$ simulate the mean static pressures across the entire surface. In reality, wind flow in the atmospheric boundary layer is more random and peak pressures do not occur simultaneously over a structure [23]. To account for this the FSI values were multiplied by the structural factor, c_sc_d, and force coefficient, c_f, from the Eurocode (FSI Reduced) [9].

As seen in Figure 8, the reduced values from the FSI simulation are now closer in line with those from EN calculations. The discrepancy in values above 100m is accounted for by the unique geometry of the building. At the top of the tower, FSI values are higher than EN as this point has the broad side of the elliptical plan thus causing more pressure than the EN (which assumes a circular plan) has calculated. This is more pronounced since wind speed is highest near the top.

For the Nanchang Tower model, the EN values follow a smooth curve. For the FSI values, the values smoothly increase until a height of 178.44m then jump at 229.46m. This is likely due to the high wind speed and concave façade at this point which leads to a higher pressure as the air would have difficulty flowing around the building at this point. The value then drops back down at the highest point where the wind can then flow over the top of the building (Figure 9).

4.3. Optimisation

Figure 10 shows the gradual reduction of the objective over each optimisation iteration. A 24% reduction in the material mass was achieved by reducing the building's structural core thickness from 0.40m to 0.30m after 100 iterations. The model-based approach of Opossum seems to have sufficiently converged after the relatively low number of 100 iterations but perhaps could have run longer to confirm since the final line of the graph is quite short.

Figure 11 shows the original vs the resulting optimum geometry, surface pressure from CFD, and FEA results after completion of the optimisation test using Opossum. The

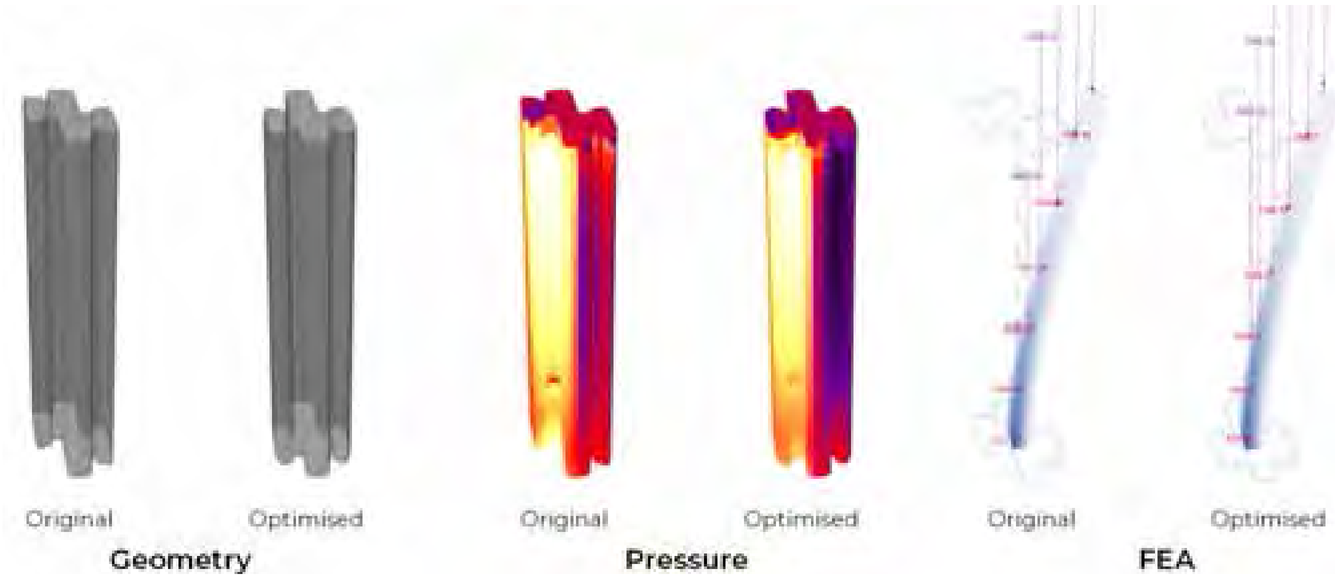

Figure 11: Original and optimised geometry, CFD, and FEA results of Ardmore Residence model

Figure 12: Excerpt of the resulting geometries and the corresponding objective values (mass)

lower wing is moved to the front resulting in a more symmetrical cross-section. The edges were also pulled to a sharper angle. This reduces the flat pressure inducing wall area on the windward of the building and allows the wind the flow more smoothly around the building. The reduced pressure results in lower forces on the core of the building requiring less thickness to resist deflection. The FEA results were also checked with hand calculations. r

This optimal shape is quite different from the original. Using the data recorded from Grasshopper a database of results was created where each resulting shape of the building and its corresponding structural mass calculated by the FSIO method was displayed (Figure 12). This allows a user to make decisions based on performance and architectural factors.

5. CONCLUSION

This paper outlined the development and testing of a Fluid-Structure Interaction based Optimisation (FSIO) method used to optimise the geometry of complex-shaped high-rise building models based on wind structural response in the early design phase. The aim was to create a computational method to aid in performance-based design exploration as a complement to more traditional methods of structural wind analysis used in the late phases of building design.

The first part involving the validation and sensitivity analysis of the CFD simulations showed that Butterfly using the RNG $k - \varepsilon$ turbulence model was the best performing choice. The validation showed it was able to capture variations in pressure around different areas of the building which is essential for complex geometries in close accordance to physical wind tunnel results. The sensitivity analysis showed that simulation time can be greatly reduced, from 42.6 hours initially to 5.7 hours at most by doing fewer CFD iterations, which had little to no impact on the accuracy, and by using a coarser mesh which has a greater impact on accuracy with about 18% deviation for the coarsest mesh. It follows a similar trend in values over the building area and thus can reliably show relative improvement between simulations of different geometries if CFD settings remain constant.

The second test involved comparing the results from the developed FSI method (the combination of CFD and FEA) to results obtained by Eurocode (EN) calculations. It showed that the FSI results offer near exact values with the EN procedure when combined the structural factor and force coefficient values. The differences in values can be concluded to occur due to the unique characteristics of the geometry of the building showing the FSIO method's applicability to non-standard shapes while the EN assumes a standard shape. The values obtained could possibly be lower if a finer mesh and higher number of iterations for the CFD simulation was used. These were kept low in the tests to save time.

In the optimisation test, the algorithm reduced the objective, mass of structural material, by 24% solely by manipulating the building geometry to reduce wind pressure. However, the optimal geometry was quite different from the original. The database produced contained the geometry made at each optimisation iteration with its corresponding objective value. This can be used to make better decisions for geometry in situations where wind is critical. While the complete FSIO run took 2 days to compute it is believed to still be more efficient and valuable than Eurocode calculations, wind tunnel tests, and detailed CFD simulations in the early stage due to the large number of options it can produce directly by a designer or engineer at no additional cost.

Overall, the method, particularly in the form of a Grasshopper-based tool, is best suited in the form-finding stage of design for complex towers in high wind situations. It can be used to further optimise an initially chosen geometry by making small geometric adjustments to increase wind performance while not completely deviating from the desired form. A final more thorough CFD or wind tunnel test should be done later to verify safety and conformance with code.

Further work should look at increasing the accuracy and speed of the method. Additional CFD software, turbulence models, and meshing procedures could be tested. In addition, the testing of multiple wind directions can have a large impact on the results. The choice of optimisation algorithm in this project was based on research however, tests on different algorithms could be done to find the best performing option for this specific use case.

ACKNOWLEDGEMENTS

Special thanks to Andreja Andrejevic of OMRT for guidance on the computational development of the method, Dr Adelya Doudart de la Grée and Christoph Waibel for support on the CFD aspects of the project, Baojie He for providing the wind tunnel data, and Hans Hoogenboom for support in running the simulations.

REFERENCES

1. Colombo, E., A. Grassano, and F. Perotti, *Comparison of numerical and experimental simulations used to investigate the wind-structure interaction of a skyscraper.* 2006.
2. Clannachan, G., et al., *Practical Application of CFD for Wind Loading on Tall Buildings*. 2009.
3. Fransos, D. and A. Lo Giudice, *On the use of computational simulation in the determination of wind loads on structures: design experiences and food for thought*. 2015.
4. Tamura, T., K. Nozawa, and K. Kondo, *AIJ guide for numerical prediction of wind loads on buildings.* Journal of Wind Engineering and Industrial Aerodynamics, 2008. 96(10): p. 1974-1984.
5. Chang, D. *Aerodynamic Performance Driven Form-Generation for Skyscraper Design*. 2013. Berlin, Heidelberg: Springer Berlin Heidelberg.
6. Wortmann, T. and G. Nannicini, *Black-box optimisation methods for architectural design*. 2016.
7. Evins, R., *A review of computational optimisation methods applied to sustainable building design.* Renewable and Sustainable Energy Reviews, 2013. 22: p. 230-245.
8. Wortmann, T., et al., *Advantages of surrogate models for architectural design optimization*. Vol. 29. 2015. 471-481.
9. NEN, N.S.I., *Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions*, in *NEN-EN1994-1-4:2005*. 2005, NEN.
10. Thévenin, D. and G.b. Janiga, *Optimization and computational fluid dynamics*. 2008, Springer Verlag: Berlin.
11. Bernardini, E., et al., *Aerodynamic shape optimization of civil structures: A CFD-enabled Kriging-based approach.* Journal of Wind Engineering and Industrial Aerodynamics, 2015. 144: p. 154-164.
12. Chronis, A., A. Turner, and M. Tsigkari, *Generative fluid dynamics: integration of fast fluid dynamics and genetic algorithms for wind loading optimization of a free form surface*, in Proc. SimAUD 2011 p. 29-36.
13. Melbourne, W.H., *Comparison of measurements on the CAARC standard tall building model in simulated model wind flows.* Journal of Wind Engineering and Industrial Aerodynamics, 1980. 6(1): p. 73-88.
14. Sadeghipour Roudsari, M. and M. Pak, *Ladybug: A parametric environmental plugin for grasshopper to help designers create an environmentally-conscious design.* Proc. of BS 2013 p. 3128-3135.
15. *OpenFOAM*. 2019 [cited 2019; Available from: https://www.openfoam.com/.
16. Preisinger, C., *Linking structure and parametric geometry.* Architectural Design, 2013. 83(2): p. 110-113.
17. Wortmann, T., *Opossum: Introducing and Evaluating a Model-based Optimization Tool for Grasshopper*. 2017.
18. Meng, F.-Q., et al., *Sensitivity analysis of wind pressure coefficients on CAARC standard tall buildings in CFD simulations.* Journal of Building Engineering, 2018. 16: p. 146-158.
19. Shih, T.H., et al., *A New k-(Eddy Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation*. Vol. 24. 1994.
20. Yakhot, V., et al., *Development of turbulence models for shear flows by a double expansion technique.* 1992. 4(7): p. 1510-1520.
21. Franke, J., et al., *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment*. Vol. 732. 2007. 51.
22. Roache, P.J., K.N. Ghia, and F.M. White, *Editorial Policy Statement on the Control of Numerical Accuracy.* Journal of Fluids Engineering, 1986. 108(1): p. 2-2.
23. Cook, N., *Designers' Guide to EN 1991-1-4 Eurocode 1 - Actions on Structures, General Actions, Part 1-4: Wind Actions*. 2007: ICE Publishing.