Structural assessment methods for architectural façade elements in cross-disciplinary collaboration between architects and structural engineers

D. QUAN
Xi'an Jiaotong-Liverpool University, Suzhou, China

Z. GAO
Imperial College London，London, The United Kingdom

C.M. HERR
Southern University of Science and Technology, Shenzhen, China

D. LOMBARDI
Xi'an Jiaotong-Liverpool University, Suzhou, China

J. XIA
Xi'an Jiaotong-Liverpool University, Suzhou, China

ABSTRACT: This paper explores structural simulations in the context of freeform facade design, with a particular focus on approaches in architectural and engineering collaboration. The paper presents a structural simulation process for parametrically designed non-standard facade geometry inspired by Erwin Hauer's sculptural work which is manufactured utilizing ultra-high-performance concrete cast in robotically 3D-printed formwork. Two typical structural performance analysis methods are employed, in Abaqus and Karamba3D, which respectively illustrate the engineering and architectural perspectives. The results and configurations of prior physical testing provide indications on analysis of results from two simulation methods. The two simulation methods offer detailed insights into engineering and architectural simulation processes and the distinct viewpoints taken by structural engineers and architects in collaborative design processes. Based on the analyzed case, the paper concludes by discussing the prospects for the advancement of structural analysis strategies for non-standard structural components in collaborative workflows involving structural engineers and architects.

1 INTRODUCTION

Contemporary architectural façade design frequently adopts intricate and parametrically variable shapes that can easily be modeled digitally. However, such designs can present challenges to structural engineers in terms of fabrication as well as determining the structural performance of façade components. As self-supporting building elements, façades require sufficient load-bearing capacity to resist relevant mechanical and external loads such as wind, earthquake, and blast loading. Accordingly, structural simulation plays a crucial role during façade design to meet both architectural and structural design requirements.

A considerable number of studies have investigated existing finite element analysis approaches to accurately predict the structural behavior of architectural components (Othman and Marzouk, 2018). However, the versatility of these analysis methods remains limited in the context of freeform architectural components as they are mostly applicable to 2.5-dimensional and regular panel-based shapes. In parallel, recent developments in digital architectural design methods address structural performance and its optimization as part of architectural design but also encounter challenges such as the required precision of simulation (Solhmirzaei and Kodur, 2017). Despite the need to address the question of accuracy in structural simulations in cross-disciplinary design processes, few studies have considered the differences between architectural and engineering approaches to conceptual design in general (Herr, 2018) and structural design in particular (Barazzetti et al., 2015).

This paper examines selected methods of structural assessment with a focus on the different viewpoints arising between architecture and engineering. To this end, we discuss the design process of a freeform façade element and report on related experiments conducted in the form of architectural structural simulations and finite element analysis. The façade element analyzed in this case study is based on a parametrically designed freeform facade geometry developed from the sculptural concrete works of Erwin Hauer (Hauer, 2017). Façade components are fabricated with ultra-high-performance fiber-reinforced concrete (UHPFRC) that is cast in robotically printed formwork. The study focused on Abaqus (Abaqus, 2020) and Karamba3D (Preisinger, 2018) as two typical analysis methods of structural performance employed in structural engineering and architectural contexts. The outcomes and arrangements of previous physical testing inform the analysis of the results obtained through two structural simulation methods, with our discussion in this paper focusing on differences between engineering and architectural simulation approaches. This paper examines the different perspectives taken by structural engineers and architects in the collaborative design process. It concludes with a discussion of the potentials of structural analysis strategies for freeform structural components in collaborations between structural engineers and architects.

2 COLLABORATION BETWEEN ARCHITECTS AND STRUCTUAL ENGINEERS

Cross-disciplinary collaboration between architects and structural engineers has been highlighted as a rich source of architectural as well as structural quality and innovation. Collaborations between architects and structural engineers in many international design practices have contributed to the design and construction of renowned projects (Olsen and Mac Namara, 2014; Rappaport, 2007). Cross-disciplinary collaboration is becoming more widespread, partly due to digital technologies such as BIM and digital fabrication, which offer new toolsets supporting alignments of workflows (Herr, 2020).

In previous accounts of typical workflows involving architects and engineers, early conceptual design decisions were typically determined by the architect. After the architect had completed the initial drawings, the structural engineer typically responded by checking compliance with structural requirements or by suggesting structural adjustments. As a result, the architect was involved in the entire design process, while the structural engineer focused their attention towards the later stages of the design, thus creating a linear design process. However, Bosia and Kara (2016) emphasize that the process of structural design is creative and should be thought of as iterative rather than linear –unlike architectural design, which is widely considered to be circular and iterative in nature. While the widely used linear design approach greatly speeds up the design process, it also leads to a significant lack of creativity and innovation in engineering structures (Herr, 2018). Many prominent design institutions and research teams in structural engineering are now integrating architectural approaches into structural designs. Meanwhile, architects who prioritize a more technically focused creativity are joining these teams as collaborative partners, resulting in numerous joint projects (Herr, 2022). Detailed accounts of such innovative collaborative workflows are however rare as they take place in practice-focused settings and are rarely documented in publications. In this context, the following sections report and discuss detailed insights deriving from a particular cross-disciplinary collaborative workflow, with a focus on the role of structural assessment methods in this process.

3 STRUCTURAL ASSESSMENT METHODS IN ARCHITECTURE AND ENGINEERING

This section presents an overview of the structural assessment techniques that are commonly utilized by architects and structural engineers, and critically evaluates the advantages and limitations of analytical methods and Finite Element Analysis based parametric methods.

3.1 *Analytical Methods*

Analytical methods can support the design of structural components. In general, these methods utilize analytical formulations that apply mostly to simple linear elastic problems and result in closed-form solutions, which can often be solved by hand (Chang, 2015). Strength of materials, energy methods, and linear elasticity are often included in the calculations. These methods however also rely on simplifying assumptions: For example, it is often assumed that materials are isotropic and elastic, all deformations are small, and the stress is correlated with strain linearly (Chang, 2015). Consequently, using analytical models can lead to somewhat inaccurate results and thus structural engineers cannot use them for structural design directly to avoid safety issues. Additionally, simplifications can lead to larger errors when the loading scenarios and structural system are very complicated, as has been reported for example in offshore structures (Fu, 2018).

3.2 *Finite element analysis and parametric tools enabled structural analysis*

The Finite Element Method (FEM) is a numerical technique, which can subdivide complex structures into discrete elements and simulate solutions to boundary value problems for partial differential equations (Welch-Phillips et al., 2020). By using Finite Element Analysis (FEA), structural behaviors of building elements can be simulated, such as load deflection curves. With the help of FEA, many structural analysis tasks can be conducted more accurately compared to analytical methods, such as problems related to structural dynamics (Chang, 2015). As the discrete elements can be arranged flexibly to form volumes, FEA can be applied to structural problems with non-standard geometries (Jagota et al., 2013), in particular in the context of digital design.

A multitude of FEM simulation tools have been developed to run within parametric design systems. Frequently adopted tools, such as Karamba3D (Preisinger, 2018) and Millipede (Michalatos, 2014), are created as parametric and associative modeling systems within the Grasshopper plug-in ecosystem of Rhinoceros (McNeel, 2014). Further advancements in structural simulation, specifically with regards to parametric design technology, have been demonstrated through the development of tools such as StructuralComponents (Cook and Cook, 2005) and Salamander (Jeffries, 2018), which demand the creativity of structural engineers and support from architects while utilizing these tools. The distinctive character of design projects created utilizing these tools often generates challenges when introducing them to conventional design and construction workflows. The majority of parametric and associative design systems emphasize geometric complexity, but few of them support a multidisciplinary approach. In instances where structural simulation is applied to intricate architectural designs, designers can reap substantial benefits of parametric control as parametric modeling methodologies can be integrated into the structural design process. This facilitates the rapid generation of structural alternatives, construction models, and iterative analysis of the proposed structures (Rolvink et al., 2010).

4 A COLLABORATIVE PROCESS BETWEEN ARCHITECTS AND STRUCTURAL ENGINEERS IN THE DESIGN OF PARAMETRIC FAÇADE ELEMENTS

4.1 *Collaboration workflow overview*

The adoption of a fully parametric and associative design approach necessitates a transformation of the design culture for all stakeholders involved in the design process (Bosia and Kara, 2016). Once adopted, the approach enables iterative processeses in design workflows and accounts for interrelationships with other disciplines. Figure 1 documents a collaborative process between architects and structural engineers in the design process of parametric façade elements as observed in this study. The architects took the lead in conceptual design and proposing material specifications, while the collaborating structural engineers took charge of material design and performing assessments of material characteristics. Taking the engineers's insights into account, the architects engaged in a simultaneous iterative design process, comprising geometry design, conceptual structural design, physical prototyping, and physical load testing. Conceptual structural design and physical loading tests relied heavily on effective communication and suggestions from the structural engineers. Following the initial design, the architects continued to adapt the geometry alongside the development of fabrication techniques, while the structural engineers further optimized structural efficiency in parallel to achieve the structural performance goals and target geometry.

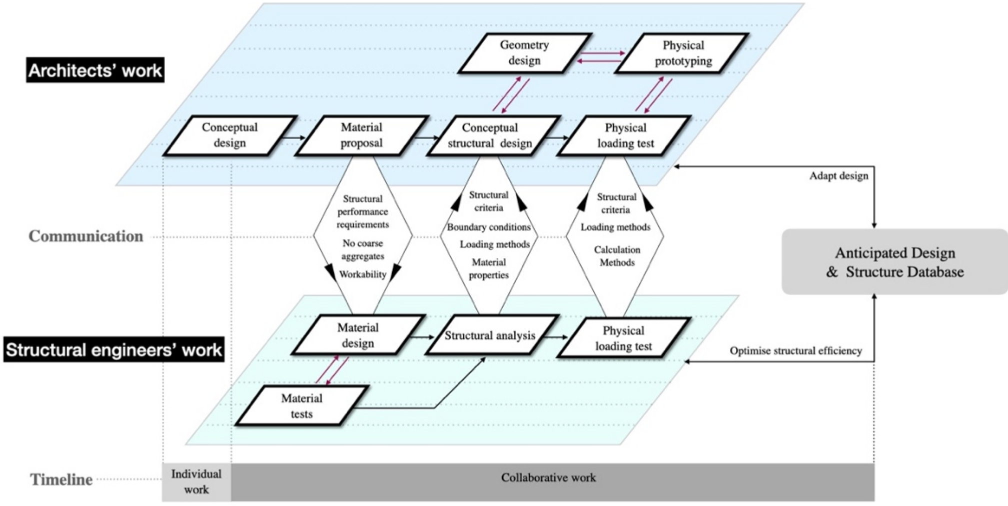
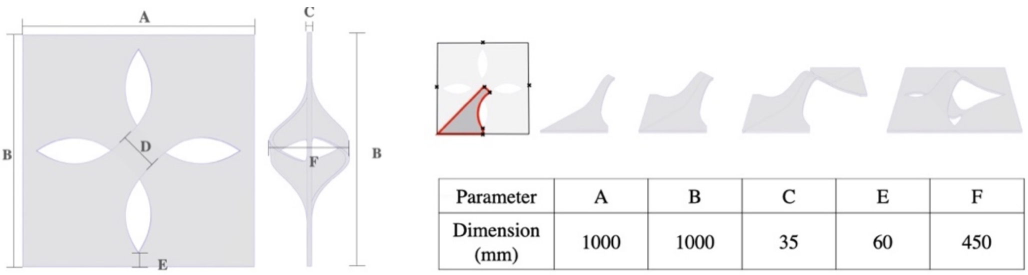


Figure 1. The cross-disciplinary collaborative workflow in the case study

4.2 *Parametric freeform façade geometric design*

The interlocking non-standard façade geometry shown in Figure 2 (left) forms the design focus of the workflow used to illustrate the collaborative structural assessments in this study. The parametric geometry is developed as a standard module and defined through the specification of vertices, boundaries, and surfaces. This allowed for the establishment of parameters, variables, and their mutual geometric relationships and constraints. The modular façade design is illustrated in Figure 2 (right), which presents the module generation logic and the specific variables (A-F) of the prototype geometry. The next sections discuss the structural performance assessment of the prototype through digital simulations. The module thickness (C) is the primary variable that has been optimized to achieve both high structural and material efficiency.



Parameter	A	B	C	E	F
Dimension (mm)	1000	1000	35	60	450

Figure 2. Parametric modelling logic and prototype variables

4.3 *Conceptual structural simulation in Karamba3D*

Given that the designed façade elements should be self-loading and withstand wind loads on high-rise buildings, a series of structural assessments were conducted through physical and digital simulations. These assessments informed the geometric design and fabrication of the façade elements at an architectural scale. At a previous stage of this study, the structural performance of small-scale prototypes was explored through physical testing (QUAN et al., 2022). The result of these tests is shown in the load-deflection curve (Figure 3, second from the left). The curve was taken as a reference to check if the full-scale façade element is safe under the ultimate load of the small-scale specimen. Then, a conceptual structural simulation was conducted by architects on the full-scale prototypes (dimensions in the table shown in Figure 2) in Karamba3D. The aim of this test was to validate the model by comparing the maximum displacement in the simulation with the maximum displacement in the physical test.

The main material properties of UHPFRC are shown in Table 1. Supports at the four corners of the façade panel were set to prevent displacement in the three directions as well as all three rotations. Point loads with a total value of 6.5KN were applied on 12 nodes on the center of the arched part as shown in Figure 3. As the total force applied on the façade panel is 6.5 KN, the load on each node is 0.54 KN. The 3D model was discretized and imported into the structural simulation as a shell structure with a thickness of 35mm. The result of the analysis shows that the maximum displacement is around 0.7mm, which is less than 2mm the maximum displacement in the physical testing. It indicates that this prototype will not be damaged under this loading condition. To further optimize the structural efficiency, the prototype thickness was decreased to 25mm for the same simulation condition. However, the maximum displacement in 25mm thickness case is up to 2.1mm. This indicates that, the protype may be damaged under the loading conditions. These Karamba3D simulations offer architects a conceptual understanding of structural behaviors while exploring geometric potentials in the façade element design. These simulations also serve as the foundation for effective communication and collaboration and with structural engineers.

Table 1: Main material properties of UHPFRC

Young's Modulus	Poisson's ratio	Tensile Strength	Compressive Strength
45000 MPa	0.2	7 MPa	150 MPa

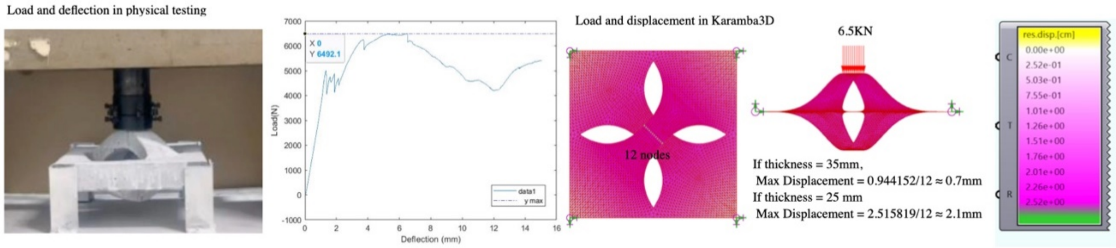


Figure 3. Conceptual structural simulation in Karamba3D

4.4 *Simulation in ABAQUS*

Using ABAQUS, the structural engineers conducted a Finite Element Analysis to obtain the displacement and stress distribution of the façade panel. The FEA results were then compared with the ones from Karamba3D to further check thickness reliability. The façade model was first constructed in Rhinoceros and then imported into Abaqus as a .sat file. The boundary conditions are the same as the ones in the Karamba3D simulation. A Concrete Damage Plasticity (CDP) model was used to model the concrete in Abaqus. Tensile damage was also included in the simulation. Material properties remained consistent with previous research and simulation in Karamba3D. For the meshing, quadrilateral elements are used, and the resultant element number is 3934.

The results of displacement and stress distribution are shown in Figure 4. Overall, the maximum displacement is around 3.6 mm for the façade panel, which occurs in the middle of the arch part. As indicated in Figure 4, no tensile damage was observed under the applied loading. Wind load was calculated based on the Eurocode, resulting in a total wind load of around 1.6 KN, which is smaller than 6.5 KN and indicates that the façade panel will be safe under wind loads.

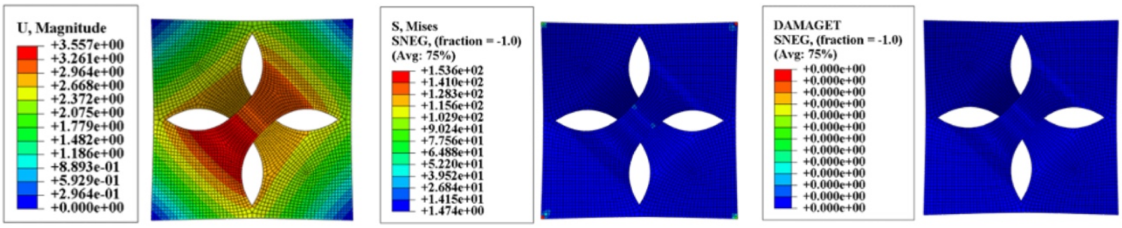


Figure 4. Simulation results in Abaqus

5 SUMMARY

This paper presents structural simulations of freeform façade design in two typical structural analysis software packages, Abaqus and Karamba3D. While the simulations provided indicative insights on optimized thicknesses for façade panels and improved the material efficiency of the designed façade panel, they also offered insights into the collaborative cross-disciplinary workflow between architects and structural engineers. Based on a parametric and associative design approach, this workflow incorporated structural design expertise, enabling structural engineers to make informed decisions and supporting effective communication with the collaborating architects. The workflow developed in this project also allowed the architects to synchronize design decisions with the constantly adapted material and geometric variables while meeting structural requirements. As a result, the overall design flexibility was enhanced throughout the collaborative design process as a range of design parameters and decisions could be assessed within short time frames. Based on the experimental insights, this paper contributes a material-efficient method to design non-standard concrete façade elements with high structural performance. The iterative work process presented further illustrates the advantages of flexible and efficient collaboration between architects and engineers.

6 REFERENCE

ABAQUS. 2020. *Abaqus user manual* [Online]. Available: https://www.3ds.com/support/hardware-and-software/simulia-system-information/abaqus-2020/ [Accessed 29 March 2023].

BARAZZETTI, L., BANFI, F., BRUMANA, R., GUSMEROLI, G., PREVITALI, M. & SCHIANTARELLI, G. 2015. Cloud-to-BIM-to-FEM: Structural simulation with accurate historic BIM from laser scans. *Simulation Modelling Practice and Theory,* 57**,** 71-87.

BOSIA, D. & KARA, H. 2016. *Design engineering refocused*, John Wiley & Sons.

CHANG, K.-H. 2015. Chapter 7 - Structural Analysis. *In:* CHANG, K.-H. (ed.) *e-Design.* Boston: Academic Press.

COOK, B. & COOK, K. 2005. *StructuralComponents* [Online]. Available: https://www.structuralcomponents.net [Accessed 29 March 2023].

FU, F. 2018. Chapter Eight - Design of Offshore Structures. *In:* FU, F. (ed.) *Design and Analysis of Tall and Complex Structures.* Butterworth-Heinemann.

HAUER, E. 2017. *Still Facing Infinity: Sculpture by Erwin Hauer*, Images Publishing Dist Ac.

HERR, C. M. 2018. Creativity in cross-disciplinary collaborations between architects and structural engineers in China. *Proceedings of IASS Annual Symposia.* Boston: International Association for Shell and Spatial Structures (IASS).

HERR, C. M. 2020. Design cybernetics in support of cross-disciplinary collaboration: educating the next generation of Chinese architects and structural engineers. *Kybernetes,* 49**,** 2109-2124.

HERR, C. M. 2022. Sublimating Tectonics of Architecture: Innovations in Creative Structural Engineering. *In:* KANAANI, M. (ed.) *The Routledge Companion to Ecological Design Thinking.* New York: Routledge.

JAGOTA, V., SETHI, A. P. S. & KUMAR, K. 2013. Finite element method: an overview. *Walailak Journal of Science and Technology (WJST),* 10**,** 1-8.

JEFFRIES, P. 2018. *SALAMANDER 3* [Online]. Available: https://www.food4rhino.com/en/app/salamander-3 [Accessed 29 March 2023].

MCNEEL, R. 2014. Grasshopper 3D. *URL: https://www. grasshopper3d. com/3.*

MICHALATOS, P. 2014. *Millipede* [Online]. Available: https://grasshopperdocs.com/addons/millipede.html [Accessed 29 March 2023].

OLSEN, C. & MAC NAMARA, S. 2014. *Collaborations in architecture and engineering*, Routledge.

OTHMAN, H. & MARZOUK, H. 2018. Applicability of damage plasticity constitutive model for ultra-high performance fibre-reinforced concrete under impact loads. *International Journal of Impact Engineering,* 114**,** 20-31.

PREISINGER, C. 2018. Karamba3D. *Vienna: Bollinger und Grohmann ZT GmbH.*

QUAN, D., HERR, C. M., LOMBARDI, D., GAO, Z. & XIA, J. 2022. Prototyping Parametrically Designed Fiber-reinforced Concrete Façade Elements Using 3D Printed Formwork. IASS 2022 Symposium affiliated with APCS 2022 conference, 2022 Beijing. Proceedings of the IASS 2022 Symposium affiliated with APCS 2022 conference.

RAPPAPORT, N. 2007. *Support and Resist: Structural engineers and design innovation*, Monacelli Press.

ROLVINK, A., VAN DE STRAAT, R. & COENDERS, J. 2010. Parametric structural design and beyond. *International Journal of Architectural Computing,* 8**,** 319-336.

SOLHMIRZAEI, R. & KODUR, V. 2017. Modeling the response of ultra high performance fiber reinforced concrete beams. *Procedia engineering,* 210**,** 211-219.

WELCH-PHILLIPS, A., GIBBONS, D., AHERN, D. P. & BUTLER, J. S. 2020. What Is Finite Element Analysis? *Clinical Spine Surgery,* 33**,** 323-324.