OPTIMIZATION OF PARTITION WALL INFILLED PATTERN FOR MINIMIZING CARBON FOOTPRINT

A method that integrates parametric design interface with FEM analysis engine

HANMO WANG[1], ABHIMANYU GOEL[2] and ALEXANDER LIN[3]
[1,2,3]*National University of Singapore, Singapore.*
[1]*hanmo@u.nus.edu, 0000-0001-6856-3705*
[2]*ar.abhimanyugoel@gmail.com, 0000-0001-8908-9715*
[3]*bdgal@nus.edu.sg, 0000-0002-3773-7503*

Abstract. The term "biomimicry" has been discussed and studied for a long time in the research field. A triply minimal surface geometry called gyroid was found to have the potential to present lightweight but solid structures and possess good thermal insulation properties, thus possibly minimizing the carbon footprints during both building construction and operation stages. Therefore, this paper will deliver research on the physical properties of the gyroid structure at different scales and explore the feasibility of scaling microstructure onto a partition wall system, which seeks opportunities to set up an efficient connection between parametric modelling and finite element engine. The integration work allows evaluating the performance of the gyroid structure with various variables as the wall infillings, which supplies the critical information and assist the engineers in figuring out the ideal design candidates at a given condition. This workflow requires a parametric approach including Rhino and Grasshopper, a finite element analysis tool ANSYS APDL (ANSYS Parametric Design Language), and Excel to save the essential data for further use. This project studies the suitable scales of the selected gyroid structure and the technical details on the interoperability between the parametrical system and the structure analysis engine for performance-based optimization. The expected outcome is to provide a tool that assists designers in optimizing the building components at the early stage and finally enrich the methods of computer-aided design.

Keywords. Low-carbon Solution; Bio-inspired Solution; Design Simulation; FEA Method; Design Optimization; SDG 12; SDG 13.

1. Introduction

The term "biomimicry" has been used as a model in architectural design to guide and induce sustainability (Gamage and Hyde, 2012), which has been widely used to learn

the various abilities of biology, study their mechanism of action, as a way to carry out technical design, to improve existing or create new mechanical systems, instruments, and equipment, building structures and technological processes. Inspired by graphene materials, researchers at MIT developed a new triply minimal surface structure called "Gyroid." In their findings, this new structure has exceptionally high strength at a relatively high density based on the 3D graphene assembly, for which it has a density of 4.6% that of mild steel and is ten times as strong as mild steel (Zhao et al., 2017). Gyroid concrete is not only ultra-strong and lightweight but also has been experimentally proven to be insulated against heat (Piccioni et al., 2020), which makes it huge potential in building material and construction to minimize the emissions of both building construction and operation stages. As the construction technique evolves, the additive manufacturing technique has become an efficient tool for generating geometry-complex structures such as gyroid. Although advanced manufacturing technology has laid a solid foundation for transferring the bio-inspired pattern onto the building components, the lack of proper knowledge of the material's properties and the absence of appropriate simulation engines and building codes is still hindering the successful implementation of bio-inspired texture in building components.

One of the critical issues is the lack of an efficient linkage between the parametric modeling interface and corresponsive software for simulation, which separates the design and simulation work rather than integrates them. Some plugins based on finite element methods are available in parametric software like Rhino and Grasshopper. Still, the existing tools are designed explicitly for large-scale structures like beams, shells, etc. Besides, for evaluating the 3D heat transfer performance of the customized 3D model, there are no available plugins inside the parametric software. Therefore, to design and select the preferred infill pattern for further manufacturing, it is significant to set up a direct connection between the parametric software and the finite element analysis simulation engine. In this paper, we will explore the carbon emission capability of the partition wall system with the customized infill gyroid pattern by establishing and testing a linkage between parametric modelling and the finite element engine. Lastly, this research is a proof of concept for a multi-phase optimization scheme to design bio-inspired building components. The next stage will discuss a mature fabrication and construction for deploying this structure.

2. Main Workflow

This part will give a detailed illustration in constructing the basic gyroid model, writing the prepared text file of boundary conditions, interlinking the modelling interface with the simulation platform, and ultimately conducting the optimization and selection work based on the simulation results. A brief workflow is shown as a diagram below in Figure 1. The simulation engine in this project is ANSYS APDL based on finite element analysis (Thompson et al. ,2017), which is well recognized in the academic filed and widely used in various industrial domains. The programming language Gh_Python that is inbuilt within the parametric interface is as stable as the other languages such as Java, C++, and etc, which is able to pass and transport the essential information between ANSYS APDL and Rhino/Grasshopper.
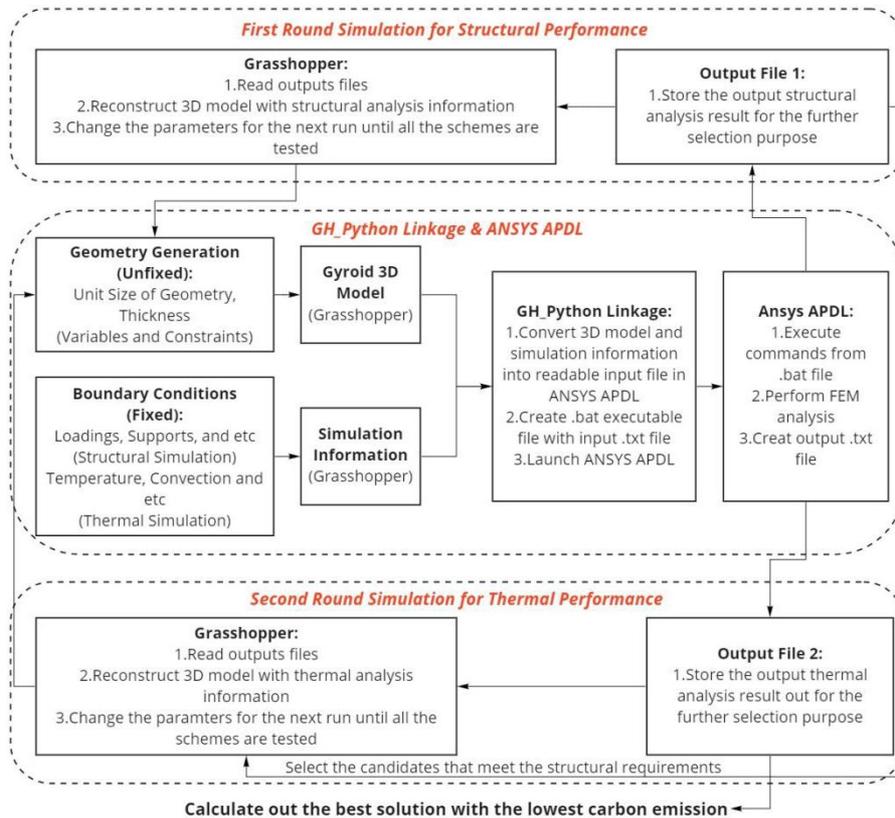
Figure 1. An image with a caption (Resource: author)

At the beginning, the 3D gyroid model is defined by several design variables, which are decided by the constraints in real applications. The prepared 3D model and the coded boundary condition files are sent to ANSYS APDL for analysis using Grasshopper Python as the linkage. When the simulation is finished in ANSYS APDL, the results are saved as text files and read in Rhino and Grasshopper for visualization and further optimization. In the process of the optimization procedures, the design candidates are arranged to go through two rounds of selection to obtain the best low carbon emission design. At the first round, the structural simulation starts to run under the given model and condition script, and the exported results from ANSYS APDL along with the input parameters are saved accordingly, after which the candidates meet the structural requirements are kept for the following stage selection. In the next round, the selected design schemes are sent to ANSYS APDL for the heat transfer simulation, and the results are retrieved and read from the saved file in Rhino and Grasshopper. Ultimately, the decision-making for the selection of a low carbon emission scheme is based on the material consumption that is stored in round 1 during the construction as well as the heat transfer that is stored in round 2 during the operation.

2.1. PARAMETRIC MODELING FOR GYROID STRUCTURE

To construct the gyroid structure in the parametric model, several ways can be used with the help of Grasshopper plugins. The standard method uses Millipede plugin coupled with a mathematical equation to construct the gyroid form where the gyroid structure is made of smaller meshes rather than the NURBS curves that use control points to build the geometry. Since the meshed geometry generated by Millipede is hard to be converted into a surface which takes less time to be imported into ANSYS, this paper is to use another optional way to construct the gyroid surface through NURBS rather than using a mathematical equation. Referring to Figure 2, the boundary of the surface patch is based on the six faces of a cube, and eight of the surface patch forms the cubic unit cell of a gyroid. Based on this, this paper proposes to construct the six surface patches first to create the ⅛ gyroid element, after which the ⅛ gyroid element is rotated and transformed to form the extra seven gyroid elements. This method is comparatively complex when constructing the needed gyroid structure, but the generated geometry can be utilized for efficiently exporting the IGES file.
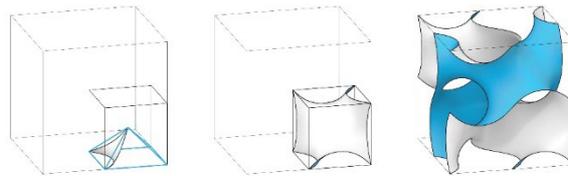

Figure 2. Formation of Gyroid Structure (Resource: (Whitehead, 2019))

Apart from understanding the mechanism of forming the gyroid structure in Rhino/Grasshopper, the size parameters also need to be decided. For the convenience of comparison of structural simulation results and reference experiment index, the project uses a 15 cm³ bounding box to contain the infill gyroid pattern that is to be analysed in ANSYS APDL in the next stage because the compressive strength of a specific concrete under the international standard is usually measured by using a 15 cm³ cube. Therefore, the size variables are defined for constructing the gyroid structure within this domain. Since the design is aimed at being applied in real construction in the future, both the print machine profile and the material property decide the value domain of the gyroid module. In this project, there are two variables: thickness and the gyroid structure's unit size. Considering printability by 3D printer, this project uses a bigger unit size domain containing 3.75cm, 5cm, 7.5cm, 15cm, which means that each side has 1 to 4 unit gyroid structures. For the thickness, it also uses a bigger unit size domain with consideration on the structural performance, in which it has five numbers containing 0.30cm, 0.35cm, 0.40cm, 0.45cm, 0.50cm. Therefore, the total test number is 20. The project didn't test as many design candidates as possible since the main target is to test the feasibility of this interoperability method and its efficiency to find better solutions to optimize the cellular structure size for the construction application.

2.2. DEFINING ESSENTIAL SIMULATION INFORMATION OF GYROID STRUCTURE

To complete the preparation work for the further simulation in ANSYS APDL, the necessary information needs to be input, including the material properties and the boundary conditions. Consequently, these values become the user inputs of the tool to accommodate the analysis under different requirements.

2.2.1. Material properties

According to EN 1992-1-1 (European Committee for Standardization, 2004), the C30 concrete (compressive strength class of 30 MPa) is used as the simulation material in this research since the standard material is easily used to judge if the designed gyroid structure meets the structural requirements in the simulation result analysis. The essential material properties for ANSYS APDL cover young's modulus, poisson's ratio, material density, and gravity, for which the proper property parameters are indicated in Table 1 below.

2.2.2. Structural simulation setup

The gyroid structure is modeled as one shell and is further meshed for the finite element analysis. There are many element types available in ANSYS APDL. Based on the field of application and the formulation of the element, shell 281 element is chosen. Since the simulation of a complete partition wall infilled with the gyroid pattern needs more computer power, the project uses a 15 cm³ cube as the test sample to replace a whole partition wall. Usually, the wall bottom bears the largest pressure, and this cube is proposed to be placed at the position of the wall bottom to undertake the loading from its top. In this project, the height of a partition wall is deemed as 3.0m high, which means that the 15 cm³ cube will bear the weight of 2.85m high volume above. The surface pressure is calculated by using the weight of the upper volume to divide the cube surface area. The volume of the loadings above is calculated using the Grasshopper volume function to obtain the weight loadings. Besides the loading from the external objects, self-weight is also considered. The horizontal shear force is ignored to simplify the analysis. The bottom surface of this cube will be constrained as support. Finally, the gyroid infill cube only undertakes vertical compressive stress and self-weight, for which the 1st principal stress is used to evaluate the simulation results. As long as the maximum stress from all the meshed elements is below 30MPa, the design scheme is viable for the construction and kept for the following selection. The detailed parameter information for the structural simulation is described in Table 1.

2.2.3. Thermal simulation setup

For the heat transfer analysis in ANSYS APDL, shell 131 is selected as the element type. In order to define the boundary conditions, two convective heat fluxes through non-solid materials are given on the front and back sides of the cube, and the heat transfer coefficients are specified according to the standard calculation methods: 20 W/m²·K for internal heat flux and 30 W/m²·K for external heat flux. The four sides of the cube are assigned an insulation boundary to represent the insulation condition, and the front and back of the cube are given a fixed temperature of 30 °C and 25 °C respectively. The simulation model directly considers heat conduction and uses the simplified method of an effective thermal conductivity of air proposed in NEN-EN-

ISO 6946 (NSAI, 2017) to replace convection and radiation. Since the internal wall has complex thermal transfer behaviour, including conduction, convection, and radiation, we use a simplified approach to convert these three heat transfer modes into one equivalent conduction for a convenient calculation. Under this method, there are two ways to set up the 3D thermal model. One option is to deem the air as a solid object attached to the shell structure, in which the solid "air" object is assigned a thermal mass element and uses the equivalent thermal resistance. Another method is to equate this geometry-complex form into a cube box according to NEN-EN-ISO 6946. The new conductivity of the equivalent cube box is calculated by combining the concrete thermal resistance and air thermal resistance. In this research, we used the second method for accelerating the simulation speed. Finally, to evaluate the insulation ability of the gyroid cube, a reference simulation is made where a solid 15 cm³ cube with the same material properties and external boundary conditions is given for the comparison test. The detailed boundary conditions for the thermal simulation are described in Table 1.

Name	Value	Unit
Youngs' modulus	3.2837e11	dyne/cm²
Poisson's ratio	0.2	N/A
Density	2.5	g/cm³
Gravity	9.806e2	cm/s²
Surface pressure	Loading weight/surface area	dyne/cm²
Internal/external temperature	20/30	°C
Internal/external heat flux	25/30	w/m²·k
Conductivity(concrete)	3e5	erg·s-1/cm·k
Conductivity(air)	Varies based on the shell thickness	erg·s-1/cm·k

Table 1: Parameters for simulation

2.3. INTEROPERABILITY BETWEEN PARAMETRIC DESIGN AND FEM ANALYSIS SYSTEM

There are two available ANSYS simulation engines, including Mechanical APDL and Workbench, for the analysis. In this paper, ANSYS APDL is used as the simulation engine behind the parametric interface. A grasshopper python plugin (Figure 3) helps to build up the connection between the parametric model and the simulation engine.
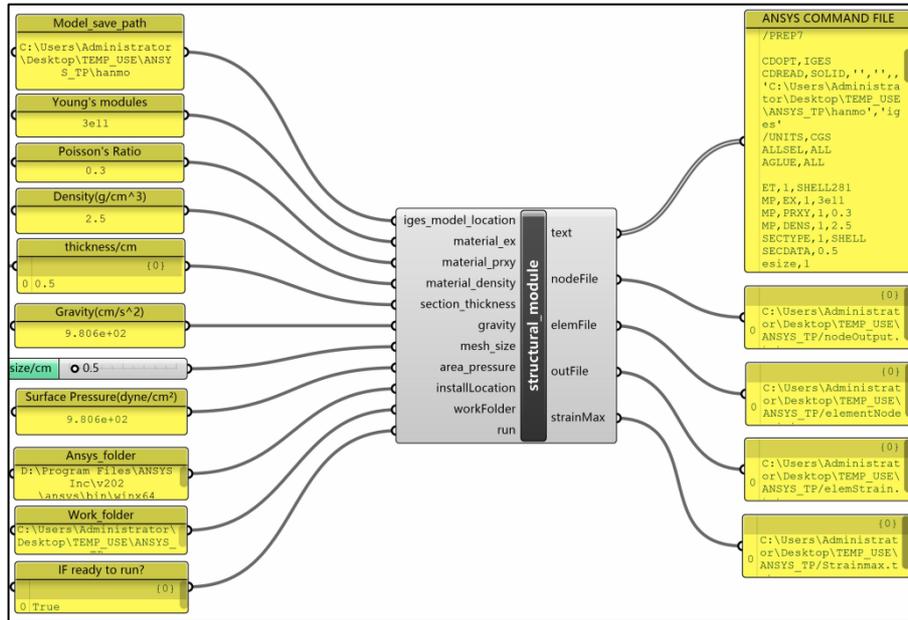
Figure 3. Grasshopper Python Linkage for Structural Simulation (Resource: author)

For a successful linkage between the parametric modeling and the ANSYS engine, the first step is reconstructing the 3D model in ANSYS APDL. For representing the geometry in ANSYS APDL, two ways are tested based on the precedent study. One solution is to mesh the gyroid structure in Rhino/Grasshopper and then use the mesh vertices and faces to represent the key points and areas in ANSYS APDL (Schoina, 2016). Another method is to export the whole model in IGES format and import the exported 3D model in ANSYS APDL (Magna, 2013). The plugin "Pancake" realizes the export work that converts the Grasshopper geometry into the IGES file directly in the work folder. After the preparation of the 3D model, the next step is to write the APDL command text file for the successful simulation in ANSYS APDL, which includes the necessary scripts like importing the geometry exported from Rhino and Grasshopper, setting up the material properties, meshing the volumes, constructing the boundary conditions, running the simulation (static analysis) and exporting the results as text in the work folder. The Grasshopper python plugin helps convert the command words above into the text file under the format of APDL. When the boolean toggle of this plugin is activated, the ANSYS APDL batch mode runs automatically to execute the tasks written inside the command file. For each time simulation, when the new result is calculated out and read in Rhino and Grasshopper, the plugin helps to tune the parameters, and the new simulation starts.

2.4. VISUALIZATION OF SIMULATION RESULTS

For the structural displacement visualization, the exported text files from ANSYS APDL include the deformed node coordinates, the nodes' number of each element, and

the maximum 1st principal stress of each element. To reconstruct the deformed geometry, the nodes' numbers of each element are used to rebuild the 8 - node polysurface with the exact coordinate of each node. According to the range of stress, one colour value domain is established, and each reconstructed polysurface is given a colour based on its stress value. Similarly, for heat transfer visualization, the nodes' numbers of each element are used to rebuild the meshed surface. But since the only concern is the heat flow through the cube, the visualized surface is the back surface of the cube. With each element's heat flux and area, we calculate each node's heat flow assigned to each reconstructed polysurface so that the heat flow performance is visualized through this 2D surface. Figure 4 shows the visualization interface for both structural and thermal results.
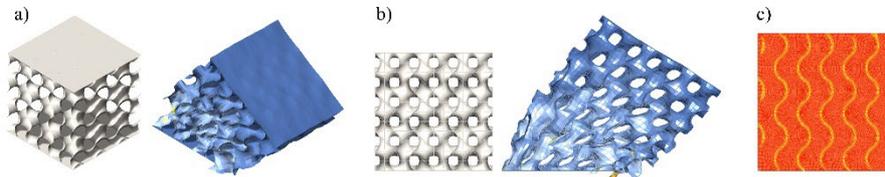

Figure 4. Visualization of Structure and Thermal Simulation (Resource: author) - (a) isometric view of deformation; (b) elevation view of deformation; (c) top view of thermal transfer

2.5. DATA COLLECTION AND PROCESSING

The structural analysis results are saved in the work folder through the Grasshopper plugin TT Toolbox in the first round. Apart from the simulation results, the geometry parameters, including the gyroid structure's unit size and thickness, are also saved. The simulation results are compared with the reference compressive strength 30MPa and filtered based on its structural performance when all the schemes are tested. All the schemes that meet the structural requirements in the first round simulation is saved. For the thermal analysis of the next round, the left-over schemes are sent to ANSYS APDL for the simulation, and the results are retrieved, after which the comparison work is made between the heat flow of the proposed schemes and that of the reference cube. Since this research aims to minimize carbon emissions during the construction and operation stages, an evaluation function is constructed to rank all the schemes. In the construction stage, the reduction in carbon emissions depends on how much material is used. The first evaluation index equals the weight of the saved material multiplied by the amount of carbon dioxide per unit weight. In the operation stage, a higher heat flow rate means more air conditioners are used to control the indoor temperature, increasing the total electricity consumption and indirectly inducing carbon emission. Therefore, we use the heat flow difference between the proposed schemes and the reference cube as the variable and convert it to the reduced carbon emission volume. We propose this elemental partition cube will be used in the next ten years. Then the second evaluation index equals the reduced carbon emission per unit time multiplied by ten years. Ultimately, the evaluation function equation is written as (1) below.

$$\text{Reduction} = W_{concrete} \times W_{contoCO2} + HF \times Time \times W_{HFtoCO2} \qquad (1)$$

Where:

$W_{concrete}$ = Weight of reduced concrete

$W_{contoCO2}$ = Weight of CO2 per concrete unit weight

HF = Amount of reduced heat flow per unit time

$W_{HFtoCO2}$ = Conversion ration from heat flow to weight of CO2

After the calculation, we select the top 5 schemes and present them in the below table 2 with their parameters.

Parameters/Number	Unit size (cm)	Thickness (cm)	Reduced weight (kg)	Reduced heat flow (J/s)	Carbon reduction by weight (kg)	Carbon reduction by heat flow (kg)	Total carbon reduction (kg)
1	3.75	0.75	3.41	0.78	0.28	15.85	16.10
2	3.75	0.7	3.61	0.76	0.25	15.51	15.77
3	5	0.85	3.68	0.72	0.27	14.71	14.97
4	5	0.9	3.48	0.72	0.25	14.66	14.91
5	5	0.8	3.88	0.70	0.28	14.26	14.54

Table 2: Calculated results for carbon reduction

3. Conclusion and Limitations

In this research, based on the simulation and the objective value calculations, it is found that the size of the infill pattern for an elemental cube of the partition wall has an effect on insulating the heat from the outdoor environment. The optimized scheme with the suitable shape parameters has the potential to be further developed into the real manufactured product to minimize carbon emission. Besides, the research's initial idea was to develop an approach that could help optimize and evaluate the structure design schemes by integrating the parametric interface with finite element software such as ANSYS APDL. However, validation work has not been conducted in this research. The hand calculation work and the physical test work need to be carried out to test the accuracy of this approach, which will be further discussed in the next stage of research. Besides, the structural and thermal simulation evaluation principles also ignore the potential factors affecting the results. In the structural analysis, the shear force was removed, and in the thermal analysis, the convection and radiation inside the wall were equated to the thermal resistance or conductivity. Therefore, a more standard and accurate evaluation method needs to be explored to regularize this process. Secondly, the efficiency of such a method is low since each simulation takes more than five minutes to finish to calculate the results and send back the data, which is time-consuming and makes the visualization effect unsmooth. Besides, this method is not applicable when it comes to a larger number of candidates for evaluation. Lastly, using the plugin requires knowledge including parametric modeling, python language, and ANSYS parametric design language, making it not friendly to designers who are weak

in one or more of them.

4. Recommendations and Future Works

A more systematic and accurate validation method needs to be conducted in the next stage of work, and the approach in this paper can be further improved and developed based on the information after validation. The current development of this design tool has a limitation in simulation speed. And consequently, only when the total test number is less, this method is applicable. Therefore, a surrogate or machine learning model needs to be developed for simulation acceleration rather than using the software engine as the simulation tool. Although the visualization is viable in this research, the speed of reconstruction work is based on the mesh density. When the mesh density is very heavy, the visualization process is unsuitable. A more efficient visualization method needs to be tested. Also, the plugin itself should be revised and more user-friendly to designers with less knowledge in the finite element field and python language field. It is essential to convert these parts into readable input modules. Lastly, as mentioned, this research is more of a proof of concept for a multi-phase optimization scheme to design bio-inspired building components, and therefore, further researches on fabrication and cost consideration of deploying this structure need to be discussed at the next stage.

References

European Committee for Standardization. (2004). *Eurocode 2: Design of concrete structures—Part 1-1: General rules and rules for buildings* (EN 1992-1-1). https://www.phd.eng.br/wp-content/uploads/2015/12/en.1992.1.1.2004.pdf

Gamage, A., & Hyde, R. (2012). A model based on Biomimicry to enhance ecologically sustainable design. *Architectural Science Review*, 55(3), 224–235. https://doi.org/10.1080/00038628.2012.709406

Magna, R., Reichert, S., & Knippers, J. (2013). Integrated design methods for the simulation of fibre-based structures. *Design Modelling Symposium Berlin*. https://www.researchgate.net/publication/292745702_Integrated_design_methods_for_the_simulation_of_fibre-based_structures

NSAI. (2017). *Building components and building elements—Thermal resistance and thermal transmittance—Calculation method* (ISO 6946: 2017). The National Standards Authority of Ireland. https://infostore.saiglobal.com/preview/is/en/2017/i.s.eniso6946-2017.pdf?sku=1961927

Piccioni, V., Turrin, M., & Tenpierik, M. J. (2020). A Performance-Driven Approach for the Design of Cellular Geometries with Low Thermal Conductivity for Application in 3D-Printed Façade Components. *Symposium on Simulation in Architecture and Urban Design, SimAUD2020.*

Qin, Z., Jung, G. S., Kang, M. J., & Buehler, M. J. (2017). The mechanics and design of a lightweight three-dimensional graphene assembly. *Science Advances*, 3(1), e1601536. https://doi.org/10.1126/sciadv.1601536

Schoina, S. (2016). *Performance-based form-finding and material distribution of free form roof structures* [Master Thesis, Delft University of Technology]. https://repository.tudelft.nl/

Thompson, M. K., & Thompson, J. M. (2017). *ANSYS mechanical APDL for finite element analysis.* Butterworth-Heinemann.