The Journal of the IASS welcomes contributed Technical Papers pertaining to the design, analysis, construction, and other aspects of the technology of all types of shell and spatial structures. In addition, Project Descriptions about realizations of innovative or noteworthy spatial structures are particularly solicited. All Technical Papers and Project Descriptions are open to written discussion, which will be published, possibly together with a response from the author(s). Contributors need not be members of the IASS.

Requirements and instructions for submitting Technical Papers, Project Descriptions and Discussions are presented on the inside of the back cover of each issue and also appear on the IASS website.
Announcements

IASS Symposium Announcement 2019
103
Upcoming Events
104

Technical Papers

Case Study on a Spherical Glass Shell
I. El Hayek, M. Engelmann, O. Friedreich and B. Weller
105

Predimensioning Sub-Space for Spoke-Wheel Roofs
A. Cabello
119

Jack Christiansen's Cylindrical Concrete Shells
T. S. Sprague
131

Design and Analysis of an Adaptive Bending-Active Plate Gridshell
M. C. Phocas, K. Alexandrou and C. Zakou
141

Shaping Forces; Review of two Bridge Design Methodologies towards Architectural and Structural Symbiosis
152


IASS Secretariat: CEDEX-Laboratorio Central de Estructuras y Materiales
Alfonso XII, 3; 28014 Madrid, Spain
Tel: 34 91 3357491; Fax: 34 91 3357422; https://iass-structures.org
journal@iass-structures.org; iass@iass-structures.org
Printed by SODEGRAF ISSN:1028-365X Depósito legal: M. 1444-1960
Purpose of the IASS

The continuing development of design, analysis and construction techniques of shell and spatial structures has resulted in an increasing fund of information of practical interest to architects, engineers, and builders. The IASS, founded in 1959 by Eduardo Torroja and a number of other prominent pioneers in the field, has as its goal the achievement of further progress through an interchange of ideas between all those interested in lightweight structural systems such as lattice, tension, membrane, and shell structures. To this end, the IASS organizes annual Symposia and occasional Colloquia, fosters the activities of several technical Working Groups and sponsors the publication of their reports, and publishes this Journal four times yearly.

Membership Benefits and Fees

All those interested in any aspect of design, analysis and construction of spatial or shell structures, as well as those interested in the research into their behavior, are welcome as members of the IASS. The benefits of membership include:

- Reduced registration fees at IASS Symposia and Colloquia
- Participation in the Working Groups of the Association
- A copy of newly published reports of every Working Group
- A subscription to this Journal of the IASS

The current annual membership fees (in euros) are:

- **Individuals** who share the aims of the IASS: €85
- **Students** at recognized institutions of higher learning: €10
- **Collective**: organizations or firms which share the aims: €425

Currenty Active Working Groups

3. Technical Expert Group on Cooling and Solar Updraft Towers
4. Technical Expert Group on Masts & Towers
5. Concrete Shell Roofs
6. Tension & Membrane Structures
8. Metal Spatial Structures
12. Timber Spatial Structures
13. Computational Methods
15. Structural Morphology
16. Historical Spatial Structures
17. Environmentally Compatible Structures
18. Teaching of Shell and Spatial Structures
20. Advanced Manufacturing and Materials

Further Information

The Association is governed by its Executive Council, elected by the members. The Council, in turn, elects the officers and appoints chairs of the standing committees and Working Groups.

For additional information about the Association and for membership application forms please visit the IASS website or contact the Secretariat of the IASS (see bottom of previous page).
INTRODUCTION

Lightweight structural systems have been the focus of the IASS since its foundation in 1959. Among them, structural membranes are perhaps the prime example of lightness and performance; they have been the topic of the Structural Membranes Conference series since 2003.

Tension and membrane structures, shells and spatial structures show, better than any other structural type, that shape and stress, form and force, are deeply interrelated. This relationship reaches well beyond purely static aspects and has an impact on performance, design, environment and aesthetics of structures in multiple fields of application.

Form and Force 2019 incorporates the IASS Symposium 2019 (60th Anniversary Symposium) and the 8th International Conference on Textile Composites and Inflatable Structures – Structural Membranes 2019. Following the customary celebration of the IASS Symposium every ten years in Spain and the Structural Membranes Conference every four years in Barcelona, the organizers have found an outstanding opportunity to merge both communities into a joint international conference that aims to provide a forum for state-of-the-art contributions and fruitful discussion.

Form and Force 2019 is organized by Carlos Lázaro (UPV, Spain), Kai-Uwe Bletzinger (TUM, Germany) and Eugenio Oñate (CIMNE/UPC, Spain).

THEME AND TOPICS

The conference will cover all aspects related to material, design, computation, construction, maintenance, history, environmental impact and sustainability of shell, spatial, tension and inflatable structures in all fields of application.

SYMPOSIUM DATES AND VENUE

Form and Force 2019 will be held on 7 – 10 October 2019 at the Crowne Plaza Hotel, located in the heart of Montjuïc, in downtown Barcelona, with very good connections to the airport and the city centre. An attractive combined accommodation-registration package will be offered to all participants.

CALL FOR PAPERS

Engineers, architects, researchers and everybody interested in exploring the interrelation between design, analysis, performance, environmental footprint and aesthetics of structures are encouraged to submit their abstracts and participate in Form and Force 2019.

DEADLINES (TENTATIVE)

Abstract submission 31 January 2019
Notification of abstract acceptance 28 February 2019
Paper submission 30 April 2019
Notification of paper acceptance 31 May 2019

SOCIAL PROGRAM

Social events will be organized for the participants and accompanying persons including a welcome reception, a gala dinner, and other events.

FURTHER INFORMATION

For further information please visit the conference website: http://congress.cimne.com/Formandforce2019
### UPCOMING EVENTS

<table>
<thead>
<tr>
<th>EVENT</th>
<th>DATE &amp; PLACE</th>
<th>WEBSITE</th>
</tr>
</thead>
<tbody>
<tr>
<td>International Conference on construction research-Eduardo- Torroja-Architecture, Engineering and Concrete / AEC Where do we come? Where we are going?</td>
<td>21-23 November 2018 Madrid, Spain</td>
<td><a href="http://www.fundacioneduardotorroja.org">http://www.fundacioneduardotorroja.org</a></td>
</tr>
</tbody>
</table>

*For more information and links: [https://iass-structures.org/Events](https://iass-structures.org/Events)*
CASE STUDY ON A SPHERICAL GLASS SHELL

Ibrahim EL HAYEK¹, Michael ENGELMANN², Otfried FRIEDREICH³ and Bernhard WELLER⁴

¹M.Sc., former M.Sc student at the Institute for Building Construction, Technische Universität Dresden
²Dr.-Ing., former researcher at the Institute for Building Construction, Technische Universität Dresden, Josef Gartner GmbH, Gartnerstraße 20, 89423 Gundelfingen, Germany, m.engelmann@permasteelisagroup.com
³Dipl.-Ing., AXIS Ingenieurleistungen ZT GmbH, Rainergasse 4, 1040 Vienna, Austria, friedreich@axis.at
⁴Prof. Dr.-Ing., Technische Universität Dresden, Institute of Building Construction, August-Bebel-Straße 30, 01219 Dresden, Germany, bernhard.weller@tu-dresden.de

Editor's Note: Manuscript submitted 21 April 2017; revision received 16 October; accepted 8 November 2017. This paper is open for written discussion, which should be submitted to the IASS Secretariat no later than December 2018.

DOI: https://doi.org/10.20898/j.iass.2018.196.874

ABSTRACT

Transparency is a social demand – in both politics and architecture. Therefore, outstanding glass constructions are often located in buildings of political significance. We raised the question whether it is feasible to design a 26.0 m spherical glass shell to cover the assembly hall of the Austrian Parliament in Vienna. This structure will be post-tensioned by means of a cable net to prevent a decompression of the glass. The paper introduces in the full numerical design process: global form finding, a research on available joint designs and the calculation of the minimum cable force that is necessary to ensure that there is no tension in the glass at all. Additionally, a buckling analysis as a function of the joint stiffness ensured a stable structure. Based on this we proposed a method on how to cover a post-breakage scenario. It was concluded that it is feasible to design ambitious full glass shells while using the glass as a load-bearing material.

Keywords: glass shell, structural glass, numerical modelling, post-breakage

1. INTRODUCTION

1.1. Glass shells

Glass is known as a brittle material and is usually used as a cladding element while the load is carried by a substructure made of another material like steel. However, the high stiffness of glass, its compressive strength and the continuous improvement of glass manufacturing techniques allow for a wider use of the material. Double-curved spherical shell structures which carry load primarily by in-plane compression forces show considerable potential for the use of glass not only as an outside skin but also as the main load-bearing element.

This principle has been used and applied in several projects; in 1998, a self-supporting glass dome with a diameter of 12.3 m and an arch rise of 2.5 m was presented at the “glasstec” trade fair in Düsseldorf, Germany.[1] The dome was realised with a system of equilateral triangles projected onto the spherical surface and a three-way arrangement of steel cables running beneath it. From 2002 to 2004 at the Delft University of Technology, a glass dome was designed and built. The dome had a diameter of 5 m and consisted of plate trapezoidal panels that were connected to each other using glue aluminium strips. [1] Additionally, in 2004, a closed spherical shell was realised at the Institute for Lightweight Structures and Conceptual Design (ILEK) at the University of Stuttgart, Germany. The glass dome rests on a titanium ring and has a diameter of 8.5 m and a rise of 1.76 m. The glass panes are double curved and connected to each other with epoxy adhesive. [2]

Today, transparency is a political demand as well and thus materialised by glass structures and open, transparent designs in governmental buildings. As an outstanding example, the Austrian government decided to rehabilitate its neoclassical parliament.
building in Vienna, Austria, in 2014. The 350-million-euro project included to covering of the “Nationalratssitzungssaal” (large assembly hall) with a 26.0 m spherical glass shell. The planning team of Jabornegg & Pálffy and AXIS engineers proposed a load-bearing glass shell in order to both flood the room with natural light and to provide a viewpoint for the visitors and citizens. (Figure 1)

![Figure 1: Visualisation of the spherical glass shell of the “Nationalratssitzungssaal” in Vienna, Austria. ©GP Jabornegg & Pálffy_AXIS](image)

The idea was to realise a two layered layout where to top-layer consists of rectangular glass panels while the inner layer comprises a post-tensioned steel cable web to fully compress all glass elements and joints during all possible load combinations.

### 1.2. Resulting questions

In order to be able to build a glass spherical shell, many questions must be answered in advance. First of all: Is it possible to create a glass shell geometry in a way that meets the architect’s wishes and, simultaneously, is both cost-competitive and structurally stable? Generating a requested cutting pattern shape from mathematically defined or free-form surfaces was and still is difficult with the existing CAD programmes. Therefore, a method had to be found for the glass shell to be able to create a desired cutting pattern. Second, the previous glass shells were not able to span large areas since the amount of tension forces in both joints and glass itself increased. However, cables can be used to allow other members in the structure to be differently arranged, so as to make the best use of their individual material properties.

Moreover, a very important aspect of the glass shell is the connection detailing between the glass panels. Hence, the question was raised whether it was possible to implement a suitable modelling technique to describe the physical characteristics of joints in a finite element model.

Furthermore, the glass shell has to be watertight and resist all types of loads, symmetric and asymmetric loads, as well as imperfections and temperature.

The last challenge to be mentioned here is the glass itself. The brittle behaviour needs to be controlled. The possibility of failure of one or more glass panels needs to be considered in the design. Thus, a course of action on how to consider partial failure of glass in the model has to be studied as well.

Thus, a feasibility check was executed as part of a master’s thesis at the Institute of Building Construction at the Technische Universität Dresden. [3] This paper summarises its findings and contributes to research into creating and building a spherical glass shell structure that can overcome most of the issues the previous shells had.

### 1.3. Course of action

In chapter 2, possible methods for generating the geometry of the glass shell are suggested. Two glass shells are obtained, the first with equal distances between nodes and the second with rectangular, planar glass elements. Generating the geometry is implemented by using the Chebyshev-net method, which was chosen and implemented using the advanced algorithmic modelling tool Grasshopper. Part 3 presents the functional requirements for designing the connection and a list of joint details found in the literature. It also devises a suitable method for modelling the joints between the glass panels using spring stiffness, and proposes two connection details for the finite element model. Part 4 shows the derivation and the assumptions of the permanent and variable loads for the glass shell model according to the Eurocode. In chapter 5, the finite element model of the glass shell is developed using the finite element software Dlubal RFEM. Furthermore, the amount of pre-stressing of cables is
determined and the influence of pre-stressing and types of joints on the glass shell is shown. Chapter 6 predicts the behaviour of the glass shell when a glass panel breaks and shows the assumed broken panels according to linear buckling analysis. Finally, part 7 summarises the work and provides a concept for a mock-up and detail drawings of its joints and bearing.

2. FORM FINDING

2.1. Basic considerations

The geometry of the glass shell was obtained and developed according to the Figure 3. These drafts are considered as the initialising point for further improvement and optimising of the spherical glass shell geometry. The ground plan shows a raster of 1 m spacing in both directions and therefore the glass panels should have dimensions of 1 m x 1 m. In the vertical plan, one can see that the glass shell should have a height of 2.8 m and a radius of 36.4 m.

The shell geometry consists of the glass panels, the bar spacers and the cables (see Figure 2). The glass panels are considered the main parts of the shell structure because they are the actual load-carrying elements. The bar spacers are used in order to create connections between glass panels and cables, in a way that they will always act in compression.

According to [1] as many of the panes as possible should have rectangular corners to keep production costs low. The glass industry applies a cost premium of between 30 and 50 percent for cutting triangular panes on top of the price for cutting rectangular sheets. This premium is necessary to cover the greater waste and longer cutting times involved. It was also mentioned that the cost of double-curved glass can be ten times that of flat rectangular glass. Thus, double-curved glass panels are very expensive and not considered cost-competitive in this specific case. As a result, the main focus in this section will be on how to create glass shell geometry with rectangular plate panels. In fact, a spherical shell with perfectly rectangular and planar plates is not possible. The given flat geometry however allows for a considerable close solution. Therefore, the following section includes two options of panel sizing. [4], [5]
2.2. Chebyshev-net method for glass shells

As mentioned in the previous sections, there is no software which has the ability to generate a shell structure with plane elements with specified edges length. Consequently, a new method had to be found that would aid in the design process of a plate shell structure. The Chebyshev-net method was found in [4]. With this method, grid shells with equal distances between nodes and non-planar elements can be generated. However, after implementing the proposed method, it was extended to create a shell structure with plane elements.

To find a grid of equidistant points in a 2D space, a general point 0 is used to draw two orthogonal lines A and B (Figure 5). The desired edge length L is defined by drawing a circle (circle 0) with the radius L originating from point 0; then intersecting the circle with curves A and B to define point 1 and point 2. At the intersection points, circle 1 and circle 2 are drawn with the same radius as circle 0. The circles’ intersection defines point 3. The procedure is repeated for the remaining three quadrants to create a grid. The 3D extension of the 2D method implies a tri-dimensional geometric construction. Meaning, a set of spheres were used instead of circles.

The implementation of this approach resulted in a glass shell composed of 518 curved panels but with equal edges length of 1 m and an angle range of 87°-93° (Figure 6, left). Here, three possible design principles arise. Firstly, using flat square panels, an offset of maximum 2 mm at the corners of the glass panels will result. Secondly, using twisted glass panels, a common cut lines at the panels joints without offset will arise. Finally, in comparison with the given plans a noticeable deviation of 28 cm at the circumference will be the result.

In an effort to diminish this, [3] proposes the use of flat panels but with different edge lengths (Figure 6, right). The panel’s edges have a length range between 970-1030 mm where the angles between panel’s edges are exactly 90°. An inclination of the cut edge may be compensated by an adjustable construction as introduced in section 3.

2.3. Final glass shape

The Chebyshev-net method was implemented in the parametric tool Grasshopper. Two options resulted from stepwise refinement: first, equal distances between nodes, so that all panel edges have a length of 1 m and, second, right-angle edges and a slight difference in the length of the glass panels edges.

The connection between the glass panels and the cable mesh was done using the bar spacers. These spacers have the direction of the normal vectors on the spherical surface at the intersection of the glass panels (see Figure 7). The spacers have a length of 10 cm. The cable mesh is then produced by connecting the end points of the bar spacers.
3. DESIGN OF JOINTS

3.1. State of the art in linear glass-to-glass joints

The critical part of designing the glass shell is the joint between the glass panels. Thus, when constructing the joint connections in glass shell structure, numerous interdependent factors and functional demands must be taken into account.

Load-bearing behaviour

The connection detail must be able to resist the dominating loads (self-weight, snow and wind; both in symmetric and asymmetric distribution). The detail must show high compressive strength capability since the shell structure is mainly loaded in compression. In addition, it must be able to transfer the loads between the glass panel edges without stress concentration. This asks for a suitable glass contact material that meets the requirements of structural glazing. DIN 18008-3 [7] recommends Polyamid, Polyoxyethylene (POM), Polyetheretherketone and Polysulfone.

Installation

Bringing together and assembling of the glass panels has to be as simple as possible. The connection detail has to be able to take any tolerances that may appear on site. In addition, transport, mounting and maintenance have to be considered.

Architecture

Architectural design intention and connection appearance are of very high importance. The main idea of the glass shell structure is that the glass panels carry the loads and the connection detail should transfer the loads between the panels. Therefore, if the connection detail looks large and heavy, it will appear as if it was a common steel-tube structure and the basic concept of a glass shell will be mislaid.

Since it is difficult to design a connection detail without knowing how the glass shell reacts under different connection types (rigid, semi-rigid and flexible), materials and connections were looked into, which had been tested in experiments and studies. The results in terms of recommendations concerning materials and connections will be presented in the next section.

Knot connection detail

The Institute of Building Construction at the Technische Universität Dresden is involved in the development process of transparent space grid structures ([10], [11]). The idea behind these structures is to replace all steel chords in the compression layer with in-plane-loaded glass elements. In 2009, a transparent space grid structure was built above the inner courtyard of the “Reichstagspräsidentenpalais” adjacent to the “Reichstag” in Berlin, Germany ([11], Figure 8). The load transfer between the glass panes is ensured by stainless steel knots at the corners with the high-loaded blocking material POM-C GF25 as shown in [10], [11] and [12].

Figure 7: Normal vectors to design the bar spacers

Figure 8: “Reichstagspräsidentenpalais”, Berlin, Germany
Folded plate structure connection detail

At the Technische Universität Wien, a new connection for folded plate structures was developed ([8], [9]). Figure 10 presents the components in a clear manner. Thus, the connection is suitable for triple laminated glass panels, where a metallic connection plate is laminated and inserted into the interlayer. The connecting plates consist of two parts and are joined by rivets to allow for thickness tolerances. Moreover, the cavity between the connection plates is filled with epoxy-grout, which gives the connection more stability. Finally, weather protection and covering of the connection can be completed by using sealing or silicone joints.

Glued-in plate connection

Another connection detail found in literature is the joint modelled by Bagger [13]. The detail was used to model the joints in a plate shell structure at the Technical University of Denmark. It is a glued-in plate joint as shown in Figure 11 and consists of an aluminium strip embedded into the glass facet edge using epoxy adhesive DP 490 by 3M.

Friction connection

As an alternative construction, a friction connection was used to model the plate shell structure in [13] (Figure 12). The detail contains two continuous profiles (aluminium or steel) that are clamped around the edge of a single monolithic sheet of glass. The friction interlayer between the continuous profile and the glass sheet can be either EPDM or Klingersil C-4500 (Klinger) interlayer.

Glued butt joint

Ultimate slenderness and transparency could be achieved by connecting the glass panels with a glued butt joint, like the joining method used in the glass dome at ILEK in Stuttgart. The glued butt joint consists of the epoxy DP 490 that is used to connect the double curvature glass panel and is illustrated in Figure 13.
3.2. Modelling of joints

The body in finite element analysis is considered as a continuous domain and the stress is a continuous function of the strain. Therefore, surfaces touching each other on one line are rigidly connected to it. However, the glass shell consists of rectangular plate panels which are connected to each other with joints. This means there is a variation of the stiffness at the panels’ edges. To realise the possibility of modelling this stiffness difference, a suitable method will be applied.

Mainly, a simple representation of the physical connection detail is in focus since modelling the detail with every geometric feature and using precise material properties would be unrealistically cumbersome. It is thought more valuable to use a simpler model which can be easily handled. As stated in [13], a simple description of the stiffness components can be arranged. Thus, a linear relationship is supposed to occur between the various types of movement in the connection and its resistance, so that the connection detail’s stiffness components can be described by linear springs, translational and rotational. Finally, the full shell rests on hinged line supports around the circumference of the structure.

The joint between the glass panels will be modelled in a way that the contribution of all important parts of the structure can be included in the finite element model. Therefore, the line bonds between glass panels will be modelled by means of springs stiffness’s $k_m$, $k_n$, $k_{s,i}$, and $k_{s,o}$ (see Figure 14).

![Figure 14: Connection detail replaced by linear springs. Definitions according to [15]](image)

In order to obtain the characteristic properties (springs stiffness) of the connection joint between the glass panels, a connection detail has to be designed and experimentally tested. In the previous section, different connection details were found in the literature, which were also experimentally tested to obtain their stiffness parameters, and they are summarised in Table 1 below. Every opaque connection is unwanted in a transparent glass structure, both aesthetically and structurally. Therefore, Table 1 comprises a theoretical pure-glass joint as a reference as this demonstrates the ideal connection.

<table>
<thead>
<tr>
<th>$k_m$</th>
<th>$k_n$</th>
<th>$k_{s,i}$</th>
<th>$k_{s,o}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[kN]</td>
<td>[kN/mm²]</td>
<td>[kN/mm²]</td>
<td>[kN/mm²]</td>
</tr>
<tr>
<td>[8]</td>
<td>35</td>
<td>3,6</td>
<td>0,33</td>
</tr>
<tr>
<td>[14]</td>
<td>16</td>
<td>5,0</td>
<td>1</td>
</tr>
<tr>
<td>[13]</td>
<td>100</td>
<td>0,5</td>
<td>0,08</td>
</tr>
<tr>
<td>[13]</td>
<td>5</td>
<td>0,02</td>
<td>0,02</td>
</tr>
<tr>
<td>[2]</td>
<td>129</td>
<td>6,9</td>
<td>1,9</td>
</tr>
<tr>
<td>glass</td>
<td>8450</td>
<td>280</td>
<td>108</td>
</tr>
</tbody>
</table>

3.3. Preferable joint option

To find a preferable option, we considered the following aspects:

- The criteria and functional demands described in section 3.1 (load-bearing behaviour, installation and architecture).
- The behaviour of the glass shell geometry in which the detail connection should have the ability not only to take compressive forces ($k_n$) but also the resultant moments ($k_m$) at the panels’ edges.
- The need of available connection stiffness parameters in order to model them in the finite element.
- The possibility of extending the proposed connection at the corners of the glass panels when designing the final details by connecting it with bar spacer and cable.

After extensive detailed research in [3], we propose two solutions for the given architectural design.

The first suggested connection detail A) with its stiffness parameters will be based on the combination of the plastic block material POM-C GF 25 from [12] and the friction connection detail with a KlingerSil C-4500 friction interlayer from [13]. The connection detail consists of an
aluminium profile with a Klingsersil interlayer and the plastic block POM-C GF 25 in between (Figure 15), which leads to a maximal stiffness $k_n$ and $k_m$.

The Klingsersil was chosen since it showed good relaxation and weather-resistance. [13] Additionally, the stiffness of Klingsersil is significantly higher than that of EPDM, and it can provide higher in-plane shear and moment resistance. Moreover, the plastic block POM-C GF 25 was mainly selected to increase the in-plane normal stiffness making use of its normal stiffness. Furthermore, the material is commonly known in structural glass design and thus will be accepted.

The second proposed connection detail B) from [8] and [9] was designed for folded glass plate structures (see section 3.1). Hence, it makes sense to use it in a shell structure where its bending moment capacities will be used as a backup load path only. Additionally, its stiffness parameters were already obtained by experimental tests and can be used for modelling the joint of the given spherical glass shell. Table 2 presents the stiffness parameters for the two proposed connections for the finite element model. As a result, not only the proposed connection details can be examined but also the rigidity of the joints on the overall behaviour of the glass shell.

### Table 2: Stiffness parameters of the proposed connections for modelling the joint in FE model

<table>
<thead>
<tr>
<th></th>
<th>$k_m$ [kN]</th>
<th>$k_n$ [kN/mm²]</th>
<th>$k_{s,i}$ [kN/mm²]</th>
<th>$k_{s,o}$ [kN/mm²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A)</td>
<td>100</td>
<td>12</td>
<td>0.08</td>
<td>1.0</td>
</tr>
<tr>
<td>B)</td>
<td>35</td>
<td>3.6</td>
<td>0.33</td>
<td>0.048</td>
</tr>
</tbody>
</table>

### 4. NUMERICAL MODELLING OF THE SHELL STRUCTURE

#### 4.1. Load assumptions

The glass shell was modelled with symmetrical and asymmetrical snow and wind loads according to the Eurocode for dome structures ([16], [17]), where the characteristic value of snow on the ground $s_k$ is 0.85 kN/m² and the peak velocity pressure $q_p$ is 0.8 kN/m². As a result, eight load combinations (CO1-8) were applied to the shell structure to assess the limit state bearing capacity (ULS) and the serviceability limit state (SLS). [3]

#### 4.2. Determination of cable force

One of the major issues to be addressed when modelling the glass shell structure is how to handle the amount of pre-stressing of the cables. Therefore, an analysis was carried out in three phases. In phase 1, the shell structure was applied to the finite element model with cables and bar spacers but without pre-stressing. In this way, it can be realised in which load case the cables will be loaded in compression or in tension. In phase 2, the shell structure was modelled without cables and tested under all load combinations with the two proposed connection details. This phase was applied to locate the places with tension forces in the shell structure. In phase 3, the amount needed for pre-stressing the cables by means of a pre-stress load case for the tendons in the model was estimated in order to fully decompress the shell (based on the results obtained from phase 1 and 2). Moreover, the flowing of pre-stressing forces was presented (Figure 16). According to this approach the glass shell structure will be free of tension forces and only loaded by compression.

![Illustration of the flowing of pre-stressing forces in the shell structure](image)
5. RESULTS

5.1. Deformation

The overall structural behaviour of a plate shell structure shares an important characteristic with a smooth shell structure. The load is transformed into in-plane forces and is thereby carried to the supports by shell action. This entails a high structural efficiency since the in-plane stiffness of a thin walled structural surface is many times greater than the bending stiffness and this leads to very small displacements of the glass shell structure.

Figure 17 illustrates the deformation of the glass shell under all load combinations. It presents how the shell reacts when external loads combined with pre-stressing are applied to it. By pre-stressing, self-weight and symmetrical snow load, the shell deformation is always downward while by applying asymmetrical snow or wind loads, the shell responds with upward and downward deformations.

Figure 17: Deformation of the glass shell, dark red color showing maximal downward deformation and dark blue showing maximal upward deformation (def. scale: 350) for friction connection from Figure 15

In Figure 18, the maximum global displacements of the shell for the two proposed connection joints with and without pre-stressing for all load combinations are shown. It was noticed that by pre-stressing the cables, the upward displacement of the shell is decreased while the downward displacement increases. Moreover, the shell with friction connection with POM-C GF 25 revealed less deformation as with the rivet connection for load combinations CO1, CO2, CO6, CO7 and CO8. This is due to the fact that the friction connection is more rigid than the rivet connection since the rotational and normal stiffness for friction connections are higher than that for rivets.

Figure 18: Maximum global deflection of the shell with the two proposed connection details with and without pre-stressing for eight load combinations

On the other hand, in the case of asymmetrical snow loads CO3, CO4 and CO5, the rivet connection showed less deformation than the friction connection with POM-C GF 25. This is due to the low in-plane shear stiffness of the friction connection since the in-plane shear stiffness will play an important role in transferring the forces in the case of asymmetrical snow loads.

5.2. Effect of cable force and joint on the glass shell

Phase 2 in section 4.2 showed that tension forces only appeared in asymmetrical load combinations. Thus, load combinations CO3 and CO8 were chosen to represent how the tension forces in the plate glass shell were diminished when pre-stressing is applied. Moreover, these load combinations are the worst cases for asymmetrical snow and wind loads respectively and thus the maximum internal forces are found there. Furthermore, CO2 (symmetrical load) was also selected to show how pre-stressing affects the shell when no tension forces in the shell structure appeared. For each load combinations, the in-plane normal forces for the two proposed
connections along the edges of the glass panes are focused on. For load combinations CO2, CO3 and CO8, the joints group A in Figure 19 was chosen because the largest in-plane normal forces in the structure occur there.

**In-plane normal force**

Load combination CO2 without the pre-stressing load did not show any tension forces in the shell. Therefore, by pre-stressing the cables, the in-plane normal force was shifted up as shown in Figure 20(a). Moreover, it can be seen that the axial in-plane force is largely independent of the connection stiffness parameters although the rotational and axial stiffnesses of the friction connection are greater than those of the rivet connection.

The asymmetrical snow load CO3 showed only tension force near the edge of the shell and by pre-stressing the cables; the connection joint was free of tension force. The asymmetrical snow loads consist of different load amplitudes on each facet. The axial in-plane forces in a facet are not anymore in equilibrium. Hence, the resulting in-plane force in the facet will be transferred to the supporting plates by in-plane shear along some of the edges. Since the in-plane shear stiffness for friction connection is lower than that for the rivet, less in-plane shear forces can be transferred through the edges of the plates and thus higher in-plane normal forces fluctuations were noticed as shown in Figure 20(b).

The asymmetrical wind load combination CO8 showed the maximum tension force approximately equal to 30 N/mm without pre-stressing. The tension force was caused by the upward deformation of the shell. However, by applying 40 kN pre-stressing to the cables, it was noticed that the tension force was diminished (see Figure 20(c)).

![Figure 19: Maximum in-plane normal forces found in the marked (red) glass edges](image)

![Figure 20: In-plane normal force for CO2, CO3 and CO8 along joints group A (according to Figure 19)](image)

6. **POST-BREAKAGE ASSUMPTIONS**

6.1. **Description of the behaviour when a glass panel fails**

The glass shell structure is an innovative structural system and seldom used. Thus, when it comes to glass being used as the main load-bearing element and also as an overhead glazing, many questions arise. The upmost important one for a structural engineer is: “What if a glass panel breaks?” This question is of course valid and must be taken into consideration beforehand.

The glass used for the shell panels is laminated glass – single glass sheets laminated by means of an autoclave process using a plastic interlayer material. The first reason for this is that when a glass panel breaks the broken pieces will not fall down and will stay in place and be adhered to the film interlayer. The second reason is that the panel will have remaining stiffness after the laminated glass is broken. However, this capacity depends on both the fragmentation of the glass and the type of the interlayer material used. Therefore, when all layers of the glass panel break a sagging effect similar to a cloth will occur if toughened glass with PVB
interlayer is used. [18] This needs to be prevented at all times. On the other hand, annealed glass will break in larger pieces and will have remaining stiffness for a short period of time during which the building can be evacuated. Consequently, a combination of toughened and annealed glass was proposed. Furthermore, it must be borne in mind that the load-bearing capacity also relies on the joint type because the panel may slide from the joints after breakage of the laminated safety glass. Hence, the joints must be able to cope with the larger rotation and deflection of the edges of the glass panels to prevent them from falling out.

Additionally, even during a post-breakage scenario the fragmented glass will be in compression. During the experimental study for the glass roof project at the “Reichstagspräsidentenpalais” (see Figure 8), it was observed that a fragmented laminated glass sheet remained stable. [11] However, this effect is still under research but may include some interesting potentials for future glass shells.

On the outside of the glass shell structure, a protective layer of 6 mm of fully tempered glass is also applied. This layer will serve as a shielding layer from external factors. This means that when the shell is hit by an outside impact this panel will be damaged first, with a high chance that both structural panels will survive unfractured. Furthermore, a very important aspect is the replacement of a broken panel. Since the glass shell structure is still in use also after the failure of one of its panels, repairing the glass shell has to be possible. In the current proposed design of the joints, removal and replacement should not be a problem. However, this has to be checked when building a structural mock-up of the glass shell structure.

6.2. Course of action on how to consider partial failure of the glass in the model

Since the integrity of the glass shell system depends on every single facet, it will be actually difficult to locate which panel will break first. However, the main aim of this section is to find a method on how to consider partial failure of the glass in the shell model. Shell structures are very sensitive to stability failure because of their thin-walled thickness and their high in-plane compression forces. In addition, the membrane stiffness is generally greater than the bending stiffness. If the shell is loaded in a way that most of its membrane strain energy from compression forces is converted into bending strain energy, the shell may fail dramatically in a process called buckling. [19]

Buckling occurs when compressive membrane forces are strong enough to reduce the bending stiffness to zero for a physically possible deformation mode. This effect is called stress stiffening or geometric stiffening, and it is the stiffening or weakening of a structure due to its stress state. The effect of stress stiffening is calculated by generating a geometric stiffness matrix \( K_G \). The geometric stiffness matrix \( K_G \) is developed according to the internal stresses obtained from the static analysis from the reference loading. The buckling loads are then calculated as part of the second load steps, by solving an eigenvalue problem:

\[
(K_e - \lambda[K_G])V = 0
\]

where \( K_e \) is the elastic stiffness matrix of the whole structural system, \( \lambda \) are the eigenvalues at which buckling takes place and \( V \) are the eigenvectors or the buckling shapes corresponding to the eigenvalues.

As an example, Figure 21 shows the resulting eigenvector for the load combination CO2 for the case with friction connection with POM-C GF 25. As a result, according to these eigenvectors and at the locations of the maximum normed displacement, we were able to locate the places where less energy is needed to buckle the glass shell. There, the glass may fail. The assumption furthermore included that a maximum of four panels will fail. These panels were accordingly removed from the model. The results from this model are illustrated in Figure 22, where local deflection effects around the perimeter of the broken panels are shown. At a distance of four to five adjacent panels, the curvature remains considerably smooth.

This course of action needs to be conducted for the other load cases and for a varying number of failed panels to find the worst-case scenario. Additionally, this structure needs to be checked for instability by conducting a second round of eigenvalue calculation. Additional considerations on partial failures of the shell were addressed in [20].
Figure 21: The first eigenvector associated to the lowest eigenvalue $\lambda = 3.24$ for load combinations CO2 (top) and the assumed broken glass panels in red colour (bottom)

Figure 22: Global deflection of the shell for a situation with four broken panels derived from CO2 eigenvalue $\lambda = 3.24$

7. CONCLUSIONS - SUMMARY - OUTLOOK

This paper shows that the design of a spherical glass shell with glass panels as the main load-bearing elements looks very promising. The shell consists of rectangular plane glass elements, is free of tension forces, resists symmetrical and asymmetrical loads and is watertight.

The development of the spherical glass shell geometry with plane glass elements for the desired cutting pattern was feasible by adapting the Chebyshev-net method. The method was implemented using the parametric tool Grasshopper and in this way it was able to facilitate the generating of both the glass shell geometry and the finite element model. Two glass shell geometries were obtained. At this stage, only the shell with rectangular glass plane elements was used for the finite element model.

Another important characteristic of the glass shell is the detail connection between the glass panels. Various detail connections were found in several research studies. Two of them were then proposed for the global finite element model, in accordance with fundamental criteria. The glass panes were modelled by shell elements in the finite element software RFEM, which included both bending and in-plane forces. The two connection details were modelled using line hinges. The stiffness of these line hinges was chosen, so that they describe the physical connection detail’s resistance to rotations around the joint line (rotation stiffness) as well as axial and shear displacements of the joint (normal and shear stiffness). However, a future research may include a sensitive study on the reaction of the global and local structure as a function of the continuous stiffness parameter.

The glass shell showed resistance to symmetrical and asymmetrical wind and snow loads. The resultant tensile forces were taken up by a pre-stressing cable mesh above the glass shell. This required an amount of pre-stressing equal to 40 kN, so that the glass shell was free of tension forces. However, the amount of pre-stressing also led to higher compression forces in both the joints and the glass panes, particularly in the area that was free of tension forces or in the load cases where no tension forces appear like in self-weight and symmetrical snow load. Additionally, this mode of loading will cause creeping of the plastic blocking materials within the joint. Thus, the shell will sag over time. Limiting the stress in the blockings will reduce this issue to a minimum. Moreover, the properties of the details as well as the global structure is sensitive to a change in temperature. Therefore, a detailed study on the effect of thermal loads on the load-bearing behaviour is mandatory in a full structural design.
Moreover, it was noticed that the rotational and the normal stiffness are the most important factors of the joint in case of symmetrical loads. Thus, both higher rotational stiffness and normal stiffness led to lower displacement of the shell. Nevertheless, in case of asymmetrical snow loads it was found that the in-plane shear stiffness of the joint was dominant. Thus, the higher the in-plane shear stiffness, the lower the displacement.

Furthermore, by using fully tempered glass with a thickness of 19 mm as the main load-bearing glass panel it was clear that the rate of utilization of the glass tensile strength of 6% is not critical. With the aid of the calculation, a buckling factor of 2.89 for asymmetrical snow load was determined. As imperfection was not included in the thesis, the thickness of the glass panels will be more probably determined by its buckling behaviour rather than by strength limitations. So, the shape of the imperfection, augmented by realistic and validated data on the expected quantity values needs future attention.

Using the corresponding eigenvector, the authors of this paper were able to locate the places in the shell where glass panels may fail. This result was used to derive a post-breakage scenario where a number of panels were assumed to fail structurally. The result of this assumption was a “perforated” shell as a new basis for a second step of stress and deflection calculation. The stability in case of a broken panel scenario will be a topic for future research as it is crucial for a safe structure.

In summary, the paper presented a structural concept and a novel design model for a glass shell that makes use of the stiffness and strength properties of the material. This will pave the way for structural glass designs that are more transparent, material efficient and safe.

ACKNOWLEDGMENTS

The master’s thesis in [3] was awarded the “Günther-Grüning-Preis 2016” during the formal diploma awarding of the Faculty of Civil Engineering at the Technische Universität Dresden. The authors want to thank the jury and the “Landesvereinigung der Prüfingenieure für Bautechnik Sachsen” (Association of Inspection Engineers in the State of Saxony).

REFERENCES


PREDIMENSIONING SUB-SPACE FOR SPOKE-WHEEL ROOFS

ADRIAN CABELLO
adrian.cabello@tensys.com; acg.adrian@gmail.com

Editor's Note: Manuscript submitted 29 May 2017; revisions received 29 October and 20 December 2017. This paper is open for written discussion, which should be submitted to the IASS Secretariat no later than December 2018.

DOI: https://doi.org/10.20898/j.iass.2018.196.875

ABSTRACT

Spoked-wheel roofs are increasingly becoming the most common method to cover wide areas such as stadia, yet little research has been completed in this field of engineering where experience and practice lead the design. This paper sheds light on this structural typology by proposing an engineering tool to help in the pre-dimensioning and early stage identification of some key structural and geometrical constrains.

The structure is reduced to a fundamental 2-dimensional problem and two systems of equations are established based on both isostatic and hyperstatic equilibriums. When the condition is met that all cable elements are subject to a minimum tension a predimensioning sub-space can be defined to be intersected by tailored limitation planes (such as critical buckling load, ultimate resistance of the material or deflection limits) and by the suitable target prestress of the structure.

The proposed tool is presented together with a strategy to facilitate the predimensioning of both circular and oval spoked-wheel roofs, with oval roofs often desirable for stadia that accommodate sports with a field aspect ratio greater than unity. These strategies have been validated through worked examples, comparing pre-dimensioning results with finite element linear and non-linear analyses.

Keywords: Spoke-wheel, stadium, lightweight structures, cable structures, steel, membrane roofs

1. INTRODUCTION

The past big sport events reflect that spoked-wheel roofs are becoming popular. There are two main reasons for that: its shape perfectly suits the usage requirements of that kind of buildings and its noteworthy structural efficiency gives rise to saving in material compared to other traditional truss systems. The latest point has been stated by several authors like Schlaich, Bergermann [1], Buchholdt [2], Majowiecki [3], Gonzalez Quelle [4].

Since its first appearance in 1989, when the Olympic Stadium of Rome [5] showed for the first time such a roof in large scale and with all its characteristic elements, it has gone through an intense and fruitful evolution and optimization, which is explained in the work of Kutterer [6]. Some interesting recent researches on the field are Masubuchi [7] and Sandalo [8].

Despite the advantages its design still demands engineering teams on a high degree of specialization and the know-how is not widely disseminated. The purpose of the current study is, in part, contributing to bridge that gap and make more accessible the design of this kind of structures.

This paper explains a design tool useful in the predimensioning of spoked-wheel roofs defined by 2 compression rings + 1 tension ring or by 1 compression ring + 2 tension rings. The first typology is fully developed and case studies are investigated based upon it, but the same procedure can be applied to the second typology. The nomenclature used in this work is: TR = Tension Ring; UCR = Upper Compression Ring; LCR = Lower Compression Ring.

The ring action on a spoked wheel roof consists on transferring the external loads to the outer and inner
rings getting a tension-compression system very effective to cover wide spaces. The governing equation of the ring action (Fig. 1) is the Barlow’s formula [9]:

\[ 2T = q \cdot D = 2 \cdot q \cdot r \Rightarrow T = q \cdot r \]

which, in a regular polygon, can be reinterpreted as:

\[ T_H = T \cdot 2 \cos \omega \] (1)

Figure 1: Constant load on a pressure vessel (left) [10], leading to the ring tension force (centre) [10] and its analogy to a spoke-wheel system (right)

The efficiency of the spoke wheel depends on the required in-plane shape of the roof. The idyllic shape is circular and it can be applied for example in arenas, but stadia usually demand an oval shape where the efficiency of the system decreases.

Where the radius is smaller the ring action improves but where it is larger the performance declines and some dimensions of the structure might need to be modified. The parameters affected by the changes in the curvature, for a given prestress force, are:

- Axial stresses in the rings increase when the radius increases:

  \[ \sigma = \frac{q \cdot r}{A} \]

- Tangential extension (\( \delta_o \)) of the ring increases when the radius increases [10]:

  \[ \delta_o = \int_0^{2\pi} \varepsilon \, dx = \frac{\sigma}{E} \int_0^{2\pi} dx = \frac{q \cdot r}{A \cdot E} \cdot 2\pi \cdot r = \frac{2\pi q r^2}{A E} \]

- Radial extension (\( \delta_r \)) of the ring increases when the radius increases:

  \[ \delta_r = \frac{\delta_o}{2\pi} \]

- Rings transfer less axial force to the spokes when the radius increases (fig. 2):

Figure 2: Different radial force \(|\overrightarrow{T_{H2}}| > |\overrightarrow{T_{H1}}|\) for a constant tension \(|\overrightarrow{T_2}| = |\overrightarrow{T_1}|\)

2. ISOSTATIC EQUILIBRIUM AND PRE-DIMENSIONING SUBSPACE

The first target in the design criteria will be that the cable elements of the structure must remain in tension, i.e. not going slack, under the loads \( W^c \) considered in the serviceability state. It is a conventional approach, as suggested by Bergermann with graphic statics [11]. To that end, a proper geometry has to be investigated. It is a 2 dimensional problem.

The parameters of the structure can be identified if it is taken a vertical section (fig. 4) showing the upper compression ring, the lower compression ring, an upper spoke (hanger), a lower spoke and the tension ring.

Loads are applied to the cladding, typically fabric or another lightweight material and, from there, they are assumed equally distributed to the lower compression ring and tension ring. By means of simplification and, assuming that the vertical position of the outer cladding connection may vary, the vertical load \( W^c \) has been considered horizontally projected as per typical snow loads. Throughout the text “\( W \)” (capital letters) corresponds to the load per unitary area and “\( w \)” (lower case) to the load per unitary length of spoke (fig. 4).

Assuming \( w \) as uniformly distributed along the spoke is a simplification on the safe side which is valid for cases with large radius or many spokes. Alternatively, where a precise definition of the load is required, \( w = (2 w_{in} + w_{out}) / 3 \) as per fig. 3 has to be adopted in the rest of the formulation explained from this point forward.
When it is considered wind uplift $w^-$ the prestress in the tension ring should provide enough tension to the hanger cable. On the other hand, when it is considered downwards pressure $w^+$, enough tension in the lower radial cable should be guaranteed. $C_i$ are the radial components of the force in the compression rings. $T_H$ is the radial component from the tension ring providing tension to the spokes.

If the equilibrium of the node is analysed, it can be found an isostatic structure to be solved easily:

$$\sum H = 0 \Rightarrow T_H = F_2 \frac{L}{\sqrt{H_2^2 + L^2}} + F_1 \frac{L}{\sqrt{H_1^2 + L^2}}$$

$$\sum V = 0 \Rightarrow \frac{w^- L}{2} + F_2 \frac{H_2}{\sqrt{H_2^2 + L^2}} = F_1 \frac{H_1}{\sqrt{H_1^2 + L^2}}$$

It is a linear system of equations. If it is solved for $F_2$:

$$F_2 = \frac{T_H - \frac{w^- L^2}{2 H_1}}{\sqrt{H_1^2 + L^2}} \left(1 + \frac{H_1}{H_2}\right)$$

Figure 3: Trapezoidal load, as opposed to u.d.l. defined in figs. 4 and 5, when a precise definition is sought

F₂ is positive, according to the sign convention, as long as the second term of the equation is positive as well.

Denominator $\Rightarrow$ always $> 0$

Numerator $\Rightarrow$

$$T_H - \frac{w^- L^2}{2 H_1} > 0 \Rightarrow T_H > \frac{w^- L^2}{2 H_1}$$

On the other hand, if downwards loads are considered and the system is solved for $F_1$ an equivalent equation would be found for the lower radial cable:

$$F_1 = \frac{T_H - \frac{w^+ L^2}{2 H_2}}{\sqrt{H_2^2 + L^2}} \left(1 + \frac{H_2}{H_1}\right)$$

$$\text{Hence: } T_H > \frac{w^+ L^2}{2 H_2}$$

The same equations 3 and 5 are still valid if the system is mirrored to generate a compression-tension-tension wheel (fig. 5). The application of the external load along the vertical strut (top, middle or bottom) is not relevant for the equilibrium, apparent after doing the sum of moments respect to the compression ring. Equations 3 and 5 would be rephrased in this case as:

$$T_{H_{1,2}} > \frac{w^+ L^2}{2 H_2} \text{ and } T_{H,1} > \frac{w^- L^2}{2 H_1}$$

Figure 4: Section of the roof showing the radial cables, column, upper compression ring (UCR), lower compression ring (LCR) and tension ring (TR), together with the forces used in the isostatic equilibrium (in red) and the main geometrical parameters $H_1$, $H_2$, L (in blue)

Figure 5: Section of the roof showing the radial cables, vertical strut, compression ring, upper tension ring and lower tension ring, together with the forces used in the isostatic equilibrium (in red) and the main geometrical parameters $H_1$, $H_2$, L (in blue)

With equations 2 and 4 it is known at every moment the force that the cables take for a given $T_H$ component in the node, and therefore $T$ according
to (eq. 1). Formulae 3 and 5 are basically the same with a different sign of the load. That equation can be plotted, for an absolute value of load, leading to the subspace for predimensioning (fig. 6). Every value above the surface guarantees the fulfillment of the aforementioned criteria of minimum tension in the cables.

It is clear from (fig. 6) that, for values of $H_i$ above 10-15m, the influence of the span turns lesser in order to find the value $T_{H/w}$ (typically KN/(KN/m) = m).

The tension $T$ claims for a second approach. The value given so far doesn’t provide all the information required to define the prestress level of the structure since it corresponds to the output prestress, different from the input prestress.

3. HYPERSTATIC EQUILIBRIUM

When the 2-dimensional problem is integrated into the real structure it becomes a hyperstatic system of springs where every element’s stiffness plays a role. It is necessary to find the input prestress leading to the output prestress and, consequently, the set of forces defined in the isostatic equilibrium. The direct stiffness method [12] is suitable to find the exact solution.

A portion of the stadium can be isolated and modelled (fig. 7), as long as it includes the rings, because it is assumed the structure follows a polar symmetry. The length of the rings’ bars should be half of the distance between ring nodes to achieve the symmetry conditions. After that, an equivalent ring bar, coplanar with the spokes, can be found by means of eq. 1 provided that they keep the same stiffness $EA/L$.

The nomenclature chosen for the bars is defined in table 1. All connections are considered articulated. Two translational degrees of freedom are assigned to each node.
Table 1: Definition of the bar elements for the direct stiffness method model

<table>
<thead>
<tr>
<th>Bar</th>
<th>Name</th>
<th>Location</th>
<th>Transformation matrix</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Lower radial cable (hanger)</td>
<td>From node 1 to 3</td>
<td>$T_1 = \begin{bmatrix} \cos\beta &amp; \sin\beta &amp; 0 &amp; 0 \ 0 &amp; 0 &amp; \cos\beta &amp; \sin\beta \end{bmatrix}$</td>
</tr>
<tr>
<td>2</td>
<td>Equivalent tension ring (T.R.)</td>
<td>From node 2 to 3</td>
<td>$T_2 = \begin{bmatrix} \cos\alpha &amp; -\sin\alpha &amp; 0 &amp; 0 \ 0 &amp; 0 &amp; \cos\alpha &amp; -\sin\alpha \end{bmatrix}$</td>
</tr>
<tr>
<td>3</td>
<td>Equivalent upper compression ring (U.C.R.)</td>
<td>From node 3 to 4</td>
<td>$T_3 = T_4 = T_5 = \begin{bmatrix} 1 &amp; 0 &amp; 0 &amp; 0 \ 0 &amp; 0 &amp; 1 &amp; 0 \end{bmatrix}$</td>
</tr>
<tr>
<td>4</td>
<td>Equivalent lower compression ring (L.C.R.)</td>
<td>From node 2 to 5</td>
<td>$T_6 = \begin{bmatrix} 0 &amp; 1 &amp; 0 &amp; 0 \ 0 &amp; 0 &amp; 0 &amp; 1 \end{bmatrix}$</td>
</tr>
</tbody>
</table>

Element stiffness matrix in local coordinates:

$$K_i' = \frac{A_i E_i}{L_i} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

The vector of external forces $F_e = [0, 0, 0, 0, T_h, V^\pm, 0, 0, 0, 0, 0]$ and the vector of nodal displacements $\bar{d} = [d_1, 0, d_3, d_4, d_5, d_6, 0, 0, 0, 0, 0]$. The relationship below leads to a system of linear equations that can be solved for the nodal displacements.

$$F_e = K_i' \bar{d}$$

Eventually, these displacements can be used to obtain the internal forces in the members via $F_i = K_i' T_i \bar{d}$

These individual stiffness matrices can be expressed instead in global coordinates by $K_i' = T_i^T K_i T_i$. The stiffness matrix of the system is given by the assemblage of the previous $K_i$ matrices. It is built up as follows:

$$K = \begin{bmatrix} K_1' & K_2' & \cdots & K_6' \\ K_1' & K_2' & \cdots & K_6' \\ \vdots & \vdots & \ddots & \vdots \\ K_1' & K_2' & \cdots & K_6' \end{bmatrix}$$

It yields to the following system of equations which can be easily solved for the values $F_i$:

$$F_1 = \frac{A_i E_i}{L_1} (-d_1 \cdot \cos\beta + d_5 \cdot \cos\beta + d_6 \cdot \sin\beta)$$

$$F_2 = \frac{A_i E_i}{L_2} (d_3 \cdot \cos\alpha - d_4 \cdot \sin\alpha - d_5 \cdot \cos\alpha + d_6 \cdot \sin\alpha)$$

$$F_3 = \frac{A_3 E_3}{L_3} (-d_3)$$

$$F_4 = \frac{A_4 E_4}{L_4} (-d_4)$$

$$F_5 = \frac{A_5 E_5}{L_5} (-d_1)$$

$$F_6 = \frac{A_6 E_6}{L_6} (-d_4)$$

(7)
The values $F_1$ and $F_2$ coincide with those obtained by the isostatic equilibrium.

However, the system, as it is defined, gives the output set of forces but the input prestress is yet unknown. If the displacement $5$ is expressed as function of $(T_H + \Delta T_H)$ by means of eq. 6: $\delta_5 = f(T_H + \Delta T_H)$ and then it is combined with $\Delta T_H = A_3 E_3 / L_3 (\delta_5)$ from eq. 7, the system of two equations can be solved for $\Delta T_H$. Hereinafter, making use of eq. 1, the input prestress force of the structure can be defined as:

$$\text{PS} = (T_H + \Delta T_H) / 2 \cos \omega$$  \hspace{1cm} (8)

4. PRACTICAL DESIGN

The interaction of the structure with the predimensioning sub-space, based on the isostatic equilibrium, together with the equations governing the hyperstatic problem have been programmed in Matlab so that the tool becomes practical and results can be tested against a commercial finite element modelling software (GSA).

Two worked examples are shown to test the Matlab tool. The results given by the procedure afore described match numerically those found by any FE software under linear analysis. However, they will differ under a geometric non-linear analysis. It has been used dynamic relaxation [13] to track the nonlinearity.

An analysis of the source and importance of these differences between linear and non-linear results is out of the scope of this paper but the non-linear results are brought here because they represent much better the real roof.

The proper choice of cross-sections, geometrical dimensions and global prestress of the structure depends dramatically on the experience of the designer; consequently this tool is to be used iteratively to tune the design in compliance with the relevant code specifications. This exercise aims to achieve a good visualization and understanding of all the parameters rather than strictly optimizing the structure to the highest degree.

The FE models created to compare the results have the following characteristics: the cables are modelled with bar elements, able to work in tension-compression and bending. Each LCR node is supported on rollers releasing the horizontal displacement along its corresponding spoke axis. This condition is very important to let the rings shrink/expand and achieve the ring action.

4.1. Worked example 1: circular spoked-wheel roof

The first design example is a circular spoke-wheel (fig. 8) with the following geometrical properties, which are considered as given even though one of them, the spacing of the spokes, could be included in the optimization study to reduce the forces in the radial cables.

![Figure 8: Top and perspective view of the worked example 1](image)

Inner radius $R = 48\text{ m}$; spacing between spokes at $TR = 10\text{ m}$; Span $L = 48\text{ m} \Rightarrow A_1 \text{ bay } = 710\text{ m}^2$; $W^+ = 1.74 \text{ kPa}$; $W^- = -0.65 \text{ kPa} \Rightarrow w^+ [\text{KN/m}] \approx W^+ / A / L$.  

The predimensioning subspace is used to obtain the height of the columns ($H_1$ and $H_2$), the prestress level and the minimum sizes of the elements. The procedure is as follows: starting values for the 6
different cross-sections and for H₁ and H₂ are chosen by the designer. Depending on the experience, they will be closer or further from the final values. Sections chosen are those from table 2. H₁ = 3m and H₂ = 13m have been taken as a compromise between length of the columns (therefore raising amount of material and buckling risk) and tension in the TR (raising the cost of the cable/s).

By solving equations 3 and 5, one can find out which load governs, i.e. which one demands the higher Tₜ so that any cable goes slack. Figure 9 shows in red the solution for the governing equation, in this case, corresponding to the uplift (eq. 3), requiring (Tₜ/w⁻)min = 378 m⁻¹. The concomitant equation is (eq. 5). The blue line points to (Tₜ/w⁺)min = 141 m⁻¹, based on the governing Tₜ, obtained previously. The closer both results are to the predimensioning surface, the more optimized the structure is. If a smaller value of H₂ is taken then downwards loads are likely to govern the design. The current structure, as it is defined, shows slight excess of tension in the TR when downwards forces are applied.

The rest of member forces, displacements and input prestress (shown in table 3) can be obtained by solving the hyperstatic equilibrium equations from chapter 3. Those equations are indeed very useful to impose and visualize constrains in the predimensioning subspace. For instance, it is useful to define the critical buckling force in the compression elements and relate it to Tₜ. To proceed with the columns:

\[
d₄ = -\frac{\pi^2 E \cdot I_{\min}}{k (H₁ + H₂)^2} \]

where k is the effective length factor. The value d₄ is now a starting parameter in equation 6, which is solved for Tₜ this time. So forth with UCR and LCR (fig. 10).

Other useful constrains would be relating Tₜ to the strength capacity or to admissible displacements of the TR.

The solution found by the red and blue lines must remain in/above the dimensioning sub-space and below their relative limit planes. The closer are the planes to the solution, the more optimized the structure is. Note that the optimization of an element for the load in one direction may be impeded by the load in the opposite direction, to be considered as well.

The following limit planes (fig. 11) are found for the cross-sections from table 2. In this case the UCR is the first element to buckle at Tₜ/w = 259m⁻¹ = 1.8 (Tₜ/w⁻)min. This value is obtained for F_{crit, UCR} = -14.77 MN (taking k = 1). By following this approach it is not verified the global buckling capacity of the structure, which may be governing. The classic formulation for circle rings under radial pressure [14] can provide a feeling of the global stability of the compression rings, bearing in mind this results in a conservative upper bound which neglects the stiffness provided by the attached columns. Applied to the current scheme, it would be F_{crit} = 3EI/R².

Table 2: Element properties, elastic moduli and minimum moment of inertia

<table>
<thead>
<tr>
<th>Cross-section</th>
<th>E  [N/m²]</th>
<th>I_{min} [cm⁴]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lower Radial Cable</td>
<td>60,000</td>
<td>165000</td>
</tr>
<tr>
<td>Hanger Cable</td>
<td>150,000</td>
<td>165000</td>
</tr>
<tr>
<td>TR</td>
<td>150,000</td>
<td>165000</td>
</tr>
<tr>
<td>UCR</td>
<td>200,000</td>
<td>200,000</td>
</tr>
<tr>
<td>LCR</td>
<td>500,000</td>
<td>500,000</td>
</tr>
<tr>
<td>Columns</td>
<td>300,000</td>
<td>300,000</td>
</tr>
</tbody>
</table>
Figure 11: Predimensioning subspace (example 1) constrained by the limit planes defined in fig. 10. Alternative limit planes based on strength or displacements may be used.

Once the $(T_h/w)_{\text{min}}$ is chosen, the hyperstatic equilibrium equations can be worked out to get the “Input PS” and forces from table 3, 'Matlab tool' column. The symbol prime (’) is used to refer to forces already transformed through eq.1. $d_6$ is the vertical displacement of the TR and it is monitored as well since it represents an important behaviour of the structure.

Provided that Input PS, loads and geometry are known, results can be cross-checked with commercial FE software (fig. 12). As announced before, differences come only when it is compared to the geometric non linear analysis. Maximum deviation in forces takes place in the TR and reaches 4%. In addition, it moves a 9% less than predicted by the linear case.

Table 3: Worked example 1. Comparison of results between the predimensioning tool and FE software (linear and non-linear)

<table>
<thead>
<tr>
<th></th>
<th>Matlab tool</th>
<th>GSA Linear</th>
<th>GSA Non linear</th>
</tr>
</thead>
<tbody>
<tr>
<td>PS_Input F1 [KN]</td>
<td>42392</td>
<td>17511</td>
<td>18190</td>
</tr>
<tr>
<td>PS_Output F1 [KN]</td>
<td>17511</td>
<td>17511</td>
<td>3600</td>
</tr>
<tr>
<td>F1 [KN]</td>
<td>3678</td>
<td>3678</td>
<td>240</td>
</tr>
<tr>
<td>F2 [KN]</td>
<td>0</td>
<td>0</td>
<td>-1114</td>
</tr>
<tr>
<td>F4' [KN]</td>
<td>0</td>
<td>0</td>
<td>-17511</td>
</tr>
<tr>
<td>F5' [KN]</td>
<td>0</td>
<td>0</td>
<td>-17080</td>
</tr>
<tr>
<td>F6 [KN]</td>
<td>0</td>
<td>0</td>
<td>-59</td>
</tr>
<tr>
<td>d6 [m x 10^{-4}]</td>
<td>9400</td>
<td>9400</td>
<td>8570</td>
</tr>
</tbody>
</table>

Figure 12: Diagram based on Table 3

4.2. Worked example 2: oval spoked-wheel roof

As soon as the footprint is not circular but elongated in one direction the structural system loses efficiency and so does the predimensioning tool but, unfortunately, many stadia need to take an oval shape to accommodate the running track or stands.

When the polar symmetry is lost the three-dimensional interaction of all structural elements increases. In-plane bending moments tend to appear in the compression rings. Due to the fact that different column heights are required at different axes, the compression rings are not uniformly horizontal and a vertical component of the axial compression force turn up in the steel rings, distorting the schematic flow of forces from fig.7.
As an example, it has been taken for the first three spokes on the axis east-west the same radius and span as per the previous exercise. Then, angles between spokes are reduced gradually, according to table 4, until a maximum radius equal to 141m is defined for the last three spokes at axis north-south (Fig. 13). Keeping the spoke spacing constant (10m) compression rings are traced, in plan, as an affine transformation of the TR, starting from E-W axis. It implies that span at E-W axis is 48m, equal to the first example, but span at N-S axis is 45m.

<table>
<thead>
<tr>
<th>Spoke</th>
<th>∆°</th>
<th>H2 [m]</th>
<th>H1 [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axis E-W</td>
<td>1</td>
<td>12.3</td>
<td>13.0</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>11.4</td>
<td>13.0</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>10.5</td>
<td>13.7</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>9.7</td>
<td>14.4</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>9.0</td>
<td>15.1</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>8.3</td>
<td>15.7</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>7.7</td>
<td>16.2</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>7.1</td>
<td>16.7</td>
</tr>
<tr>
<td></td>
<td>9</td>
<td>6.6</td>
<td>17.2</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>6.1</td>
<td>17.6</td>
</tr>
<tr>
<td></td>
<td>11</td>
<td>5.6</td>
<td>18.0</td>
</tr>
<tr>
<td>Axis N-S</td>
<td>12</td>
<td>5.2</td>
<td>18.0</td>
</tr>
</tbody>
</table>

The procedure suggested to define the required prestress in the roof together with the element sizes and heights \( H_1, H_2 \) is to study the sections at the two main axes. Once a solution is found for both cases, where the capacity of the structural elements is not exceeded, the higher value of prestress can be adopted for all the TR and a graduation of heights can be defined for the remaining axes, as defined in table 4. An alternative is to achieve the same output PS in both axes so that the TR is optimized.

Axis East-West is defined equal to the first example. Axis North-South is studied with the predimensioning tool, leading to the following results in fig. 14.

Column heights chosen at axis N-S are \( H_1 = 5.5 \text{m} \) and \( H_2 = 18 \text{m} \). Uplift case is also governing at this axis. Monitoring the limit plane of the columns was not critical at axis E-W (fig. 11) but it is critical for axis N-S (fig. 14). On the other hand, it seems that the UCR at axis N-S is quite oversized for the uplift case (not that much for downwards loads) but it can...
barely be optimized if results from figure 11 are taken into account. Both diagrams need to be checked through the design process.

Besides, before a compromise decision is taken, stresses in the structural elements need to be verified. The set of forces corresponding to the N-S axis are shown in table 6. An input $PS = 30$MN is found for the TR, leading to $22.7$MN, what is higher than the force found for axis E-W (table 5). Nonetheless, it is less than $F_{Rd} = 4 \times 6990 = 27960$ KN and the design is valid.

The following figures show the results of the Pre-dimensioning tool in the “Matlab” column for axes E-W (table 5, fig. 15) and N-S (table 6, fig. 16). The outcome of this column for axis E-W is the same as example 1 but the interesting point is to check if once the oval FE model is built, deviations from the predimensioning tool are still admissible. Maximum deviations reach a 5% at the TR, between the tool and the non-linear model.

Displacements predicted are smaller. They are quite high, almost 1m, but still in the range expected in this kind of highly elastic structures. Figure 16 displays the results of the N-S axis, where deviations from the expected results rise up to 18% in this case, reflecting a more inefficient yet useful response of the predimensioning tool.

The final PS introduced in the TR can be either the maximum required after investigating both extreme cases (maximum and minimum radius) or a graduation similar to those defined in table 3. The real PS in the structure is applied defining different cutting lengths for each element. Figure 17 shows the result from applying an input $PS = 30006$ KN all along the TR. In doing so, maximum vertical displacement of the TR is around 0.7m, at E-W axis.

Table 5: Worked example 2, axis E-W. Comparison of results between the predimensioning tool and FE software (linear and non-linear)

<table>
<thead>
<tr>
<th>$PS_{\text{Input}}$ [KN]</th>
<th>Matlab tool</th>
<th>GSA Linear</th>
<th>GSA Non linear</th>
</tr>
</thead>
<tbody>
<tr>
<td>$PS_{\text{Output}}$ ($F_6'$) [KN]</td>
<td>42392</td>
<td>17511</td>
<td>18370</td>
</tr>
<tr>
<td>$F_1$ [KN]</td>
<td>3678</td>
<td>3726</td>
<td>3578</td>
</tr>
<tr>
<td>$F_2$ [KN]</td>
<td>0</td>
<td>12</td>
<td>338</td>
</tr>
<tr>
<td>$F_4'$ [KN]</td>
<td>0</td>
<td>62</td>
<td>-1434</td>
</tr>
<tr>
<td>$F_6'$ [KN]</td>
<td>-17511</td>
<td>-17660</td>
<td>-16830</td>
</tr>
<tr>
<td>$F_6$ [KN]</td>
<td>0</td>
<td>-3</td>
<td>84</td>
</tr>
<tr>
<td>$d_6$ [$m \times 10^{-4}$]</td>
<td>9400</td>
<td>9360</td>
<td>8250</td>
</tr>
</tbody>
</table>

Table 6: Worked example 2, axis N-S. Comparison of results between the predimensioning tool and FE software (linear and non-linear)

<table>
<thead>
<tr>
<th>$PS_{\text{Input}}$ [KN]</th>
<th>Matlab tool</th>
<th>GSA Linear</th>
<th>GSA Non linear</th>
</tr>
</thead>
<tbody>
<tr>
<td>$PS_{\text{Output}}$ ($F_6'$) [KN]</td>
<td>30006</td>
<td>22720</td>
<td>18750</td>
</tr>
<tr>
<td>$F_1$ [KN]</td>
<td>1622</td>
<td>1403</td>
<td>1457</td>
</tr>
<tr>
<td>$F_2$ [KN]</td>
<td>0</td>
<td>-71</td>
<td>-78</td>
</tr>
<tr>
<td>$F_4'$ [KN]</td>
<td>0</td>
<td>-119</td>
<td>319</td>
</tr>
<tr>
<td>$F_6'$ [KN]</td>
<td>-22720</td>
<td>-18740</td>
<td>-19220</td>
</tr>
<tr>
<td>$F_6$ [KN]</td>
<td>0</td>
<td>28</td>
<td>31</td>
</tr>
<tr>
<td>$d_6$ [$m \times 10^{-4}$]</td>
<td>5522</td>
<td>3700</td>
<td>3935</td>
</tr>
</tbody>
</table>
5. CONCLUSIONS

The predimensioning subspace is proved to be a simple and hands-on tool to assist in the predimensioning of spoked-wheel roofs. The accuracy of the tool is sufficient to make it suitable for concept design stage, as it is shown in the examples herein. Nonetheless it should be emphasised the importance of a final modelling where aspects such as unbalanced loading or global stability can be thoroughly investigated, since they may well govern a final design.

The tool can be suited to meet a target tension force in the radial cables instead of the minimum tension. In addition, equations from chapter 3 can be obtained in a similar fashion to describe a compression-tension-tension system (fig. 5).

Last but not least, the author trusts this study will contribute to the global understanding of spoke-wheel roofs and, in doing so, pay tribute to the family of lightweight structures.

ACKNOWLEDGEMENTS

The author is sincerely thankful to Mr. Jorge Chenevey (SBP) for encouraging this research by suggesting the first idea in the mid of 2014.

REFERENCES


JACK CHRISTIANSEN’S CYLINDRICAL CONCRETE SHELLS

Tyler S. SPRAGUE, PE., Ph.D.
Assistant Professor, Box 355720, Department of Architecture, University of Washington
Seattle, Washington 98195, tyler2@uw.edu

Editor’s Note: Manuscript submitted 15 August 2017; revision received 16 February 2018; accepted 4 March. This paper is open for written discussion, which should be submitted to the IASS Secretariat no later than December 2018.

DOI: https://doi.org/10.20898/j.iass.2018.196.883

ABSTRACT

This article examines the early thin shell concrete designs of the structural engineer Jack Christiansen (1927-2017), a 2016 recipient of the Eduardo Torroja Medal. With no proper training in shell behavior, Christiansen started his career designing cylindrical concrete shells based on the 1952 American Society of Civil Engineers (ASCE) Manual 31. This manual, and its approach to solving indeterminate behavior, both directed Christiansen’s early design and provided a framework for significant creative work outside its bounds. His designs of long, spanning shells and short, arching shells (between 1954 and 1958) were adapted to a variety of architectural spaces, utilizing emerging structural methods like prestressing. These designs constitute the first era of Christiansen’s career, and set the stage for more varied shell geometries to come.

Keywords: historic structures, concrete shells, cylindrical shells, modernism, indeterminate analysis methods

1. INTRODUCTION

Jack Christiansen’s (1927-2017) designs are a significant contribution to the global legacy of thin shell concrete in the 20th century. When compared to other designers, Christiansen found unique success with his shell structures – practicing well into the 1970s, in the Pacific Northwest region of the United States, and often in extremely long-span conditions. In acknowledgement of his career achievements, Christiansen was awarded the Eduardo Torroja Medal at IASS Tokyo in 2016.

Christiansen began his career, however, at a much smaller scale. Having never taken courses in shell design, Christiansen had to learn the structural behavior and construction practicalities of shells on his own – using published industry resources to get his start. In 1952, the publication of the American Society of Civil Engineers (ASCE) Design Manual 31, titled “Design of Cylindrical Concrete Shell Roofs” launched Christiansen’s first venture into shell design. This Manual provided a simplified method of design for clearly defined shell geometry, and introduced Christiansen to the structural mechanics of shells. [1] While prescriptive in its geometry, Christiansen was able to quickly understand the behavior of cylindrical shells and soon began exploring a wide variety of configurations and variations. In a short amount of time, Christiansen took his designs beyond the bounds of the Manual, utilizing prestressing tendons, vierendeel trusses and other structural methods that indicate his emerging creativity in shell design.

This article will describe Christiansen’s design process - connecting the engineering analysis outlined in Manual 31 to the several shell projects Christiansen executed, and describing the variations he explored. These early explorations set the stage for Christiansen’s shells designs to come.

Note: The units used in this paper (English units) are consistent with Christiansen’s manner of practice in the United States.

2. ASCE MANUAL 31

Jack Christiansen moved to Seattle in 1952, after graduating from the University of Illinois in Architectural Engineering (1949), Northwestern University in Structural Engineering (1950) and professional experience in Chicago. The same year that Christiansen began working in Seattle, the American Society of Civil Engineers (ASCE)
published its first “Manual of Practice” addressing concrete shells. The Manual was titled “Design of Cylindrical Concrete Shell Roofs”. [2] Anton Tedesko had previously introduced cylindrical thin shell structures to the United States in the 1930s. [3] In 1946, the ASCE organized a subcommittee on thin shell design, with the intent of simplifying the structural analysis of this type of shell, in hopes of encouraging its widespread use. The members of the committee were distinguished engineers: Charles S. Whitney (chairman), Hans H. Bleich, Alfred L. Parme, Mario G. Salvadori, and Herman Schorer. This manual was largely written by Parme, a structural engineer in charge of technical publication and design for the Portland Cement Association. The Manual contained design information that had been developed, patented and implemented in Germany (by Carl Zeiss, U. Finsterwalder, and Franz Dischinger and others) and then adapted for use in the United States. [4] This publication provided American structural engineers with a series of tables and charts that drastically reduced the time required to analyze and design a cylindrical concrete shell. This Manual provided just the boost that Christiansen needed to start his exploration in to thin shell concrete.

2.1. Geometrical Definition

The Manual presented the geometry of the cylindrical barrel-vault in a simple, straightforward way with only a few variables. By definition, the cylindrical vaults have a constant radius ($r$) that defines the curvature of the shell, as well as a typically constant thickness, $t$. Shell thickness is generally determined by practical construction concerns, and for Christiansen’s projects, only varied slightly (between 2.5” and 3.25”) for widely different configurations. The arc of each constructed shell is thus a segment of a circle, defined by an angle in degrees measured equally from the centerline of the arc to either side. This arc is then extruded longitudinally, allowing the shell to span in the perpendicular, longitudinal direction, $l$.

As the Manual described, an important way of categorizing the different types of barrel-vaulted shells is through a comparison of the longitudinal span of the vault to the radius of its curvature ($l/r$). If the span is large compared to the radius, this ratio will go up, and the shell is determined to be a long shell with behavior more similar to a beam (carrying loads through bending between vertical supports). An approximate ratio for long shells is an $l/r$ of 3. If the length is short with respect to the radius, it is called a short shell ($l/r$ less than 1.0) and has behavior more similar to an arch (generating thrust between buttressed supports). Shells with a ratio in between were called intermediate shells, with behavior between that of a beam and arch. Other ratios of geometrical proportions become important in shell behavior, like $r/t$.

![Figure 1: Cylindrical Shell diagram](image)

2.2. Analysis Method

With no formal instruction in shell design, Christiansen used this Manual to learn the techniques necessary for design for himself. The Manual was divided in to two parts – the first dealing with the practical design of cylindrical shells, and the second providing the mathematical basis for the design tables. The objective of the Manual was to facilitate the task of designing shells, enabling “the engineer to study a shell problem without spending innumerable hours on tedious numerical computation.” [2, p. vii]

The fundamental approach of the guide is to address the fundamental indeterminacy of a shell structure, by first solving the determinate membrane stresses and adding in the effect of boundary conditions. The Manual acknowledged that the calculation of the boundary conditions and their effects creates the
difficult mathematical problem of shell design. The manual aimed to simplify this analysis through the use of tables and charts in the place of computation.

Figure 2: Cylindrical Shell parameters [2]

The design of thin shells is divided into two stages, with different sets of tables: 1) determine the internal stresses and the edge forces created by the surface loading on the basis of membrane analysis, then 2) determine the stresses due to boundary condition effects. Final stresses in the shell are found by adding up those found in the determinate state and those induced by end restraints. The general procedure of shell design often reduces to a problem of supplying necessary forces at the edges for equilibrium.

3. SHELL DESIGNS

This method of analysis allowed the separation of the curved cylindrical shell from the edge beam and column conditions within design. This method allowed Christiansen to approach each individual project as a two-part process: the shaping of the shell geometry to match each project, and the sizing of the edge beam to satisfy structural demands. As he changed the span-to-radius ratio and the physical dimensions of a structure, he was able to create distinctly different types of architectural spaces. Soaring arches could create high vaulted space overhead, while long barrels could create more linear, horizontal spaces. Simply by adjusting the proportion, the cylindrical shell could become many different things. Once these dimensions were set, Christiansen could define the edge beam and stiffener locations appropriately.

With a strong background in structural analysis, he easily navigated the geometry and force calculations. More importantly, however, Christiansen began to understand the spatial possibilities of the simple form and how to integrate them into architectural projects.


Christiansen’s first opportunity to design a thin shell concrete structure came in March 1954 in the design of the Green Lake indoor pool facility. The City hired the local architect Daniel E. Lamont & Lester Fey to design the pool facility as an eastern addition to the 1920s Green Lake Field House. Known for their innovative work in reinforced concrete, Lamont & Fey hired Christiansen’s firm, the W. H. Witt Company as structural engineers and Christiansen saw an opportunity. The primary design issue at hand was how to enclose and cover the swimming pool space in a low-cost manner, while relating the structure to the earlier field house.

The swimming pool was 75’ long and 42’ wide. In order to enclose the surrounding walkways and support spaces, Christiansen needed his shell to cover a space 110’ long and 60’ wide. With the length established (110’), Christiansen needed to decide what radius of shell to use, and determine the basic action of the shell. Considering the interior height, building profile and acoustics, Christiansen chose a shell with a 54’ radius, swept through a 68-degree arc (34-degrees on each side of the centerline). With these proportions, the shell had a span-to-radius ratio of roughly 2.0, making it an intermediate shell by the definitions in the design guide. Christiansen defined the shell to be 3 ¼ inches thick.

Four corner columns raised the entire shell 20’ to allow for entrances and windows. Concrete end walls stiffened each end of the vault. Following the analysis procedure from Manual 31 for a simply-supported shell with edge beams, the edge forces from the shell required edge beams 10 inches thick and 38 inches deep. These beams span between columns in the long direction. Concrete walls enclose the space below.
In addition to filling a long-desired civic need, the construction of the Green Lake pool attracted a great deal of local attention. *Seattle Times* reported that the vault was an “unusual ‘egg-shell’ roof”, the “first Seattle building of this design.” [5] Other articles celebrated the pool as “a new type for this area.” *Pacific Architect and Builder* stated that the “cylindrical barrel thin shell concrete roofing has made its Seattle debut.” [6] With this project, Christiansen had executed his first thin shell vault in the Puget Sound area.

![Figure 3: Greenlake Pool, exterior](Seattle Municipal Archives Photograph Collection, Item #29161)

3.2. Seattle School Warehouse (Long Shell - 1956)

After the success of the Green Lake pool, Christiansen saw an emerging potential for thin-shell concrete. Coming in under budget, it became clear that thin-shell concrete was indeed a low-cost solution for enclosing large areas – but not every space could be covered with a single barrel-vaulted shell. If Christiansen was going to expand its use, he would have to explore many different configurations and proportions – ones that matched different types of architectural spaces. He also knew that this complexity would require more detailed construction methods. Building the formwork for the Green Lake pool had been a major component of the cost, only to have it disassembled and thrown away after casting the shell. If a form could be built and reused, then multiple shells could be cast at only minimal additional cost. Christiansen had learned from the Green Lake pool experience, and began to seek out other opportunities to use thin-shell concrete.

Instead of the single vault like the Green Lake pool, Christiansen designed a series of multiple thin-shell vaults with very different proportions than the shell he designed before. Each vault covered a space 33 feet wide (half of before) but extended much longer (128 feet in length). The shell had a constant radius of 25,’ giving it a span-to-radius ratio of more than 5 – making it a long shell according to the ASCE Manual. As a result of these proportions, the edge beam forces became much larger and requiring a deeper section – 10" x 60" on the exterior, and slightly larger on the interior. Like the previous project, 8” end walls stiffened each shell extending from the top of the shell arc to the top of the columns.

By designing multiple, simply supported shells, Christiansen could create an expansive interior space. Working with the architect John Maloney (1896-1978), Christiansen came up with a scheme of eight concrete vaults (four vaults wide, two vaults long) to enclose the main warehouse, with the majority of columns around the perimeter and only one row of columns down the middle. Despite the multiple shells, Christiansen only had to design for two cases: an interior shell condition and an edge condition – both clearly defined by the Manual.

![Figure 4: Seattle School Warehouse, original drawing set, shell definition, sheet S-3](Seattle Public Schools Archive)

The multiple shells provided not only a new type of space, but also an opportunity for a reuse of formwork that could lower the cost of construction.
even more. The warehouse was sent out to bid to several area contractors, and the District chose the Howard S. Wright Company. Upon reviewing the bid proposal, the District found that the estimated cost to build the proposed warehouse was so low, the District could afford to expand the number of constructed vaults from eight (as seen in the original drawing) to fourteen (as was built), and still remain under budget. The Howard S. Wright Company planned to build only a single form, and use it fourteen times over the course of several weeks. The repetitive nature of Christiansen’s design and a carefully coordinated construction process meant the District could get more space for its money.

For the music and arts building, Maloney and Christiansen used a similar approach as the Seattle School District warehouse. The building used multiple barrel vaults in sequence – covering an area 30’ wide and 60’ long, giving them proportions of an intermediate shell. This arrangement naturally divided the building into 30-foot bays, which accommodated individual classroom/practice spaces well. As with the School District Warehouse, Maloney detailed the shell to extend beyond the perimeter of the enclosure – expressing the 3-inch thickness of the shell as a significant feature of the building’s appearance.

To cover the long-span gymnasium, Maloney and Christiansen used a different configuration. They significantly increased the radius of the vault to 141’ to span 175-feet in width, while effectively shortening its length to only 19-feet. This length dimension measured the distance between thicker, stiffening ribs (16” wide by 24” deep), which arched the width of the vault. These ribs acted similar to the edge beams of the longitudinal vault – collecting the loads from the shell, and channeling them to the foundation elements and ground. The shell segment between the ribs stopped short of the ground – allowing for doorways, windows and a vertical enclosure plane at the base. Five of these vaulted segments (shell between ribs) together make up entire gymnasium – a total length of almost 100 feet. With the top of the vault 36’ off the ground, the gymnasium could accommodate sporting events and large assemblies of people. Maloney and Christiansen had created a distinctly different kind of space through the simple variations of the barrel vault geometry.

3.3. Ellensburg High School (Intermediate & Short Shells - 1956)

In Christiansen’s next project, he explored the spatial quality of the cylindrical vault even further. Working on an Arts and Music building, a gymnasium and workshop building for a new high school in Ellensburg, Washington, Christiansen and Maloney investigated a variety of spanning solutions. At this point, both Maloney and Christiansen knew that by significantly varying the radius of the shell with respect to its length, while also elongating different dimensions, the designers could create significantly different types of spaces. In terms of the Design Guide, the project would have a combination of short, intermediate, and long shells – utilizing their spatial characteristics to the fullest.
4. VARIATIONS FROM THE MANUAL

These three projects introduced Christiansen to the range of possibilities of short, intermediate and long shell configurations. The analysis procedures outlined in Manual 31 gave Christiansen the confidence to design a variety of cylindrical shell forms by changing the geometrical parameters. Though the analysis process was simplified, it did not limit Christiansen’s capability to extend the method to include other structural forms and he began to see how the edge condition of each shell could be treated differently. The conceptual division of the shell design into shell and edge beam condition also provided Christiansen with opportunities to innovate.

With only a few shell designs completed, Christiansen began including prestressing tendons in the edge beams of his long shells, replacing the edge beam with vierendeel trusses, and introducing double curvature in the longitudinal direction. Christiansen was still able to use the methods of Manual 31 to capture the indeterminate behavior of these increasingly complex shell forms.

4.1. Wilson Junior High School (Long Shells - 1956)

In 1956, Christiansen brought together his cylindrical thin shell concrete structures with prestressing techniques for the first time in a gymnasium and classroom space for the Yakima Junior High School (again with John Maloney).

Christiansen designed two sets of shells, with cylindrical geometry and prestressing in the edge beams. One set of vaults covering a gymnasium - measured 41-feet wide and 135-feet long with a radius of 40 feet. The edge beam is 8” wide by 60” deep – still significantly deep, but thinner than the comparable shell designed for the Seattle Schools Warehouse. The other set of shells – covering classrooms – were also long shells. These shells spanned 90 feet, but only had a radius of 18 feet.

Christiansen’s used post-tensioning to deliver additional force to the concrete edge beams at the base of the vault. For the 135-foot span, he detailed six strands, tensioned to deliver at total of 660,000 lbs. to each edge beam (245,000 lbs. for the smaller span, in parenthesis to follow). This prestressing force was centered 5'-0" (4'-4") below the centerline of the vault, providing the counteracting moment. Manual 31 provided the required moment needed to support the shell edge, so instead of designed a mild steel reinforced beam to take the load, Christiansen simply used a prestressed design.

Additionally, in order to amplify the effect of the prestressing, Christiansen also introduced modest curvature (for the entire vault) in the longitudinal direction at an extremely large radius of 762’ (431’). This curvature gave additional rise to the center of the beam, increasing the eccentricity of the prestressing force. While the curvature was small, this variation shows how Christiansen is continuing to change the way he designs shells and marks his first use of double curvature in a shell project.

While the edge beams used in the Yakima high school were not significantly shallower (ie. less deep), they mark an important set in Christiansen’s modification of the prescribed design methods. Prestressing gave Christiansen a new set of parameters to incorporate into his shell designs – one not include in Manual 31 – and opened new possibilities as well.

In 1957, Skilling and Christiansen publicized their work, presenting “Prestressing of Cylindrical Concrete Shell Structures” at the national conference for the American Concrete Institute (held in Seattle, November 4-6). [7] In this presentation they presented the advantages of prestressing cylindrical concrete shells and their methods used. While other shells in England were utilizing pre-stressing, Christiansen and Skilling
believed these shells to be the first in the United States to have a cast-in-place, post-tensioned edge beam. [8]

Figure 8: Wilson High School Gymnasium [MKA Archive]

4.2. St. Edwards Church (Long Shell - 1956)

In addition to prestressing, Christiansen was interested in integrating other techniques into his thin shell concrete barrel-vaults and exploring its use in more sensitive works of architecture. In 1956, John Maloney was commissioned to design a new Catholic church in the Hilman City neighborhood, south of downtown Seattle. Even though Maloney had designed thin shell concrete for warehouses and schools, he now sought to use a thin shell concrete vault for a sacred, religious space.

Figure 9: St. Edwards Church, exterior

Inspired by the thin lines of the shell as a medium of Modern architecture, Maloney designed the primary sanctuary space to be enclosed by a single, cylindrical shell. As he had done with the School District Warehouse and Ellensburg School, he extended the vault beyond the end walls and over the longitudinal walls on each side. With this treatment, the shell seemed to wrap the building, embracing the space within, becoming the dominant feature of the church design.

This simple, sensitive design required a unique type of shell. The shell is 43’ wide (with a radius of 37’) and planned to extend 153’ in length – one of the longest spanning vaults in the United States. Yet, Maloney sought greater transparency and light through windows around the base of the shell. The typical solid edge beams (that this length of shell would traditionally require) did not align with the architectural priorities of the project.

To accommodate, Christiansen again innovated at these edge conditions. Responding to Maloney, Christiansen designed large openings within the depth of the edge beam, transforming it into a vierendeel trusses. The proportioning of the truss was structurally conservative, but architecturally expressive. The top and bottom chords were each 3’ deep, separated by large 7’ tall openings, with vertical members 4’-9” wide, spaced to create openings over 11’ wide. The entire beam was 1’-0” thick, with mild reinforcement.

These openings were filled with stained glass, allowing filtered light into the sanctuary space along its entire length. These edge members carried the loads from the shell as well as the lower roofs over the aisles, allowing clear-span access to the

Figure 10: St. Edwards Church, shell definition [MKA Archive]
sanctuary space from the connecting buildings. The end walls stiffened the vault, but were also accentuated with stone mosaics, depicting the patron saint, St. Edward.

The shell for St. Edwards Parish attracted national attention from the American Concrete Institute for its long span and unique edge beam design. More importantly however, the shell was the defining spatial element of a carefully designed community church – an architectural/structural component of a quiet, reflective space. In this project, the thin-shell concrete vault was used not only for its economy or structural performance, but also its space-shaping ability.

5. ANALYSIS

The large number of cylindrical shells that Christiansen designed in a relatively short amount of time lend themselves to a high-level comparison and analysis of their characteristics. Twelve shell projects, including those discussed above, have been selected for further study. A table comparing the basic dimensional properties of these shells is included in Table 1.

5.1. Short vs. Long Shells

Christiansen utilized the ability of Manual 31 to analyze both short (arching) and long (spanning) shells. The same parameters of design (radius, width, and length) could define structural sections with widely varying spatial qualities. The profiles of these shells are shown in Figure 11.

This figure is drawn with all shells sharing a common center point of a sphere, or origin. This figure reveals the variation in profile that comes from changing the radius of the shell and the width independently. The top three curves correspond to the short shells of the Pioneer Gymnasium, the Ellensburg Gymnasium, and the Boeing Hangar. With relatively short dimension between stiffening ribs in the longitudinal direction, these arcs resemble the dominant profile of each structure. The smaller radius shells then refer to the long and intermediate shells and deserve a closer, independent study.

Table 1: Christiansen Cylindrical Shell Projects

<table>
<thead>
<tr>
<th>Structure</th>
<th>Date</th>
<th>Type</th>
<th>Width (ft)</th>
<th>Length (ft)</th>
<th>r (ft)</th>
<th>t (in)</th>
<th>Ø (degrees)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Greenlake Pool</td>
<td>1954</td>
<td>Intermediate</td>
<td>60</td>
<td>110</td>
<td>54</td>
<td>3.50</td>
<td>34</td>
</tr>
<tr>
<td>Seattle School Warehouse</td>
<td>1954</td>
<td>Long</td>
<td>33</td>
<td>130</td>
<td>25</td>
<td>3.00</td>
<td>41</td>
</tr>
<tr>
<td>Ellensburg Gym</td>
<td>1955</td>
<td>Short</td>
<td>175</td>
<td>19</td>
<td>141</td>
<td>2.75</td>
<td>38</td>
</tr>
<tr>
<td>Ellensburg Classroom</td>
<td>1955</td>
<td>Intermediate</td>
<td>30</td>
<td>60</td>
<td>23</td>
<td>2.75</td>
<td>41</td>
</tr>
<tr>
<td>Yakima Gym</td>
<td>1956</td>
<td>Long</td>
<td>41</td>
<td>135</td>
<td>40</td>
<td>3.00</td>
<td>31</td>
</tr>
<tr>
<td>Nile Temple</td>
<td>1956</td>
<td>Long</td>
<td>16</td>
<td>58</td>
<td>16</td>
<td>2.50</td>
<td>30</td>
</tr>
<tr>
<td>Boeing B-52 Flight Hangar</td>
<td>1956</td>
<td>Intermediate</td>
<td>135</td>
<td>140</td>
<td>105</td>
<td>3.00</td>
<td>40</td>
</tr>
<tr>
<td>Forest Ridge Gym</td>
<td>1957</td>
<td>Long</td>
<td>18.5</td>
<td>68</td>
<td>17.8</td>
<td>2.50</td>
<td>31</td>
</tr>
<tr>
<td>Pioneer Classrooms</td>
<td>1957</td>
<td>Intermediate</td>
<td>28</td>
<td>28</td>
<td>36</td>
<td>3.00</td>
<td>23</td>
</tr>
<tr>
<td>Pioneer Gym</td>
<td>1957</td>
<td>Short</td>
<td>180</td>
<td>19</td>
<td>124</td>
<td>3.00</td>
<td>47</td>
</tr>
<tr>
<td>St. Edwards Church</td>
<td>1958</td>
<td>Long</td>
<td>43</td>
<td>153</td>
<td>36.5</td>
<td>3.50</td>
<td>36</td>
</tr>
</tbody>
</table>

Figure 11: Shell Geometry Comparison
5.2. Long and Intermediate Shell Depth

As described above, however, in long and intermediate shell structures, the height of the shell (defined as the distance of the crown of the shell above the edge) is only part of the structural system. Edge beams, both along the exterior of a structure and in between shell vaults, were a crucial part of the analysis of cylindrical shells. Conceptually, these edge beams provided a solution to an otherwise indeterminate structure.

Therefore it is not surprising that a figure comparing the height of the shell to its span (Figure 12) reveals only a shallow, linear relationship. This trend does not appear to acknowledge the increasing difficulty of long-span structures.

Yet a figure, which includes the depth of the edge beam as part of the overall structural depth, significantly changes the profile of the trend line. (Figure 13) The high point in the figure refers to the St Edwards Church – with a 153’ span, only 7 feet of height from the shell, but 13 feet deep vierendeel trusses as edge beams.

6. CONCLUSION

Between 1956 and 1958, Christiansen designed over 23 thin shell barrel vaults around Washington State, all slightly varied in their size and configuration. As shown, these structures ranged in use from swimming pools, to airplane hangers, to churches, with a spatial quality that resonated with the modern architects in the Northwest at the time.

The success of the structural form is even more remarkable knowing that they all originated from a single structural design Manual. The clarity of descriptions within Manual 31, both of fundamental shell behavior and practical design process, was able to guide Christiansen’s analysis of basic shell forms and enable expansion beyond its own bounds. These cylindrical shells defined the first era of Christiansen’s work, and set the stage for even more varied use of geometry and structural analysis to come.

PHOTO CREDITS

Photos cited to “MKA Archive” refer to an image archive, referenced courtesy of Magnusson Klemencic Associates, Seattle, WA.

REFERENCES

[1] Tedesko, Anton, Shells 1970 – History of Outlook, Concrete Thin Shells, Medwadowski, Stefan Jerzy, Schnobrich, W. C., Scordelis, Alex C., and ACI-ASCE Committee 334, American Concrete Institute, 1971, pp. #1-#8.


“ACI Regional Meeting in Seattle”, *ACI Newsletter - Journal of the American Concrete Institute*, Vol 29, No. 3., September 1957, pp. #3-#7. The abstract for the presentation “Prestressing of Cylindrical Concrete Shell Structures” is published on p. #7.


DESIGN AND ANALYSIS OF AN ADAPTIVE BENDING-ACTIVE PLATE GRIDSHELL

MARIOS C. PHOCAS¹, KRISTIS ALEXANDROU² and CHRISTINA ZAKOU³

¹Associate Professor, Department of Architecture, Faculty of Engineering, University of Cyprus, 75 Kallipoleos St., 1678 Nicosia, Cyprus, mcphocas@ucy.ac.cy
²Research Associate, Department of Architecture, Faculty of Engineering, University of Cyprus, 75 Kallipoleos St., 1678 Nicosia, Cyprus, kalex01@ucy.ac.cy
³Research Assistant, Department of Architecture, Faculty of Engineering, University of Cyprus, 75 Kallipoleos St., 1678 Nicosia, Cyprus, czakou01@ucy.ac.cy

Editor’s Note: Manuscript submitted 9 December 2016; revisions received 20 November 2017, 4 January 2018 and 19 April, accepted 23 April. This paper is open for written discussion, which should be submitted to the IASS Secretariat no later than December 2018.

DOI: https://doi.org/10.20898/j.iass.2018.196.861

ABSTRACT

Adaptive structural systems possess the ability of geometrical and mechanical adjustment with regard to changing functional, loading, or environmental conditions. In particular, gridshells following bending-active principles in their deformation behavior, require a comprehensive approach in dealing with aspects of form-finding and adaptive structural behavior. The respective framework provides systems of multi-curvature configurations with transformability capabilities that are implemented through a step-by-step process of an initially non-deformed planar grid. In the current paper an adaptive bending-active plate gridshell is proposed. This is a proposal of a structural concept and not of a specific structural/architectural solution. It consists of an array of paired elastic member stripes, which combined with interconnecting telescopic bars and cables of variable length connecting the supports, operate the necessary deformation in providing the system’s planar expansion and erection. The topology of the structure and its configurability are initially digitally investigated. Twelve different cases have been developed, based on control parameters of the supports’ displacement and the telescopic tubes’ length modification. A particular system configuration has been realized in a small-scale model, and further investigated following a sequential nonlinear static Finite-Element Analysis (FEA), in its form-finding and load-deformation behavior.

Keywords: Adaptive structures, Gridshells, Bending-active plate structures, Form-finding, Finite-Element Analysis

1. INTRODUCTION

Adaptive Architecture refers to building structures, whose certain parts may perform transformations, in order to adjust to external stimuli, without reducing the overall structural integrity and stiffness. An adaptive structure might seek to enhance its geometrical and mechanical properties, in order to respond to varying functional, loading and environmental conditions. The sharp technological progress in mechanics, electronics and robotics has been the main cause of an increasing growth of practical applications of adaptive structures over the past years [1].

Relevant applications of adaptive architecture target temporary spaces that are enabled through transformable structures. These require standardized members and modular components of low self-weight, transportability and low package volume, as well as fast erectability and easy connectivity. Engineering precedents for the development of transformable structures include meanwhile commonly known systems, such as deployable tensegrity and scissor-like systems [2]. Still, both typologies provide in principle configurations that are limited between an open and closed state, without actually succeeding in real adaptiveness as to varying external conditions or requirements.

Strained gridshells, based on the alternative use of inherently flexible materials enable a unique and
ingenious way in achieving adaptation in form [3]. The utilization of soft materials in architecture, initially realized by using high elastic modulus timber in shell shape structures, comprises the driving agent of the design process, providing unique architectural forms and free-form constructions [4-6]. Soft materials constitute mechanisms with autonomous, adaptive physical behavior. Compared to technical linkage systems, bending-active mechanisms replace local hinges by elastic deformations of their members and thus distribute the acting forces over a wider area in which bending takes place [7]. Bending-active members carry the imposed loads visually and therefore reflect directly their load-deformation behavior. During the form-activation process, the residual forces caused by active bending increase the overall stiffness of the structure. By selecting the right material and actuation type, bending-active structures can act as real-time adaptive ones with inherent elastic capacity attributes [8]. The application of bending-active plate members has been demonstrated with a pavilion prototype construction with birch plywood lamellas at the University of Stuttgart in 2010 [9] and a Biomimetic Media Façade with flexible shading elements for the Thematic Pavilion at Expo 2012 in Yeosu, South Korea [10].

Historically, the pioneering work by Frei Otto, Edmund Happold and Rolf Gutbrod gave first insights with regard to design aspects of strained gridshells [11, 12]. Initially flat lattice maps of perpendicularly cross-fastened timber laths were used, that created thin enough sections, in order to bend and therefore reach the required global system shape. Indicative project is the Manheim multi-hall constructed in Germany by Frei Otto in 1975 [13]. The structure was assembled in planar state and bent, in order to obtain its overall shape, by using the ‘push-up’ technique, i.e. the structure was lifted upwards through static framework. A further more recent example is the Undulating Downland gridshell constructed in the United Kingdom by Edward Cullinan Architects and BuroHappold in 2002 [14]. The shell was initially configured in planar form as a network of two layers of timber laths running in two directions, and placed on scaffolding support at approximately mid-height of the estimated form-found system’s height. Once the form-found shape was obtained through a ‘pull-down’ erection process, all nodes were strengthened with bracings for minimizing post-buckling and preventing post-scissoring effects. Nowadays further construction techniques exist, among others, the ‘pull-up’ technique, applied via the use of scaffolding infrastructural support, cranes and robes, or large inflatable balloons installed beneath the non-deformed grid, forcing the overall assembly to deform and reach its target shape [15, 16].

Due to the nonlinear material properties and deformation behavior of the elastic members of bending-active systems, a critical design parameter is the form-finding process. Meanwhile, several form-finding calculation techniques are applied, among others, dynamic relaxation, particle-spring system and FEA, each in different stages of development and with respective different amount of preciseness. The dynamic relaxation method can be very useful, especially in the preliminary design stage, since the members can be designed as vectors, reducing thus the simulation complexity. Particle-spring system on the other hand, has excellent application for preliminary bending behavior simulation by using axial springs on each consecutive element node. FEA is considered as the most precise method, due to the fact that the specific mechanical characteristics of the elastic members are taken into account during the calculations [17, 18]. In principle, the simulation of any form-finding process using FEA requires that the analyses of the actuated for erection and force-driven deformation of the structure are considered in the same modelling environment, following a step-by-step nonlinear approach [19]. This ensures that structural residual stresses can be identified and stored in all stages of the analysis in investigating the exact behavior of the system.

In shifting the design aim towards a controlled form-finding process, alternation of the primary components is necessary. This may refer to the boundary support conditions at the periphery, the actuation type or deformation technique of the system, the geometric and mechanical characteristics of the elastic members, or the topology of the non-deformed gridshell and its joints [3, 20]. Along these lines, a bending-active plate gridshell is proposed in the current paper, which incorporates in its form-finding process control deformation mechanisms through the elastic properties of the primary members, the length modification of the cables connecting the supports and the elastic members’ coupling telescopic bars. Thus, the term “adaptive” referred to for the current system, addresses issues of deformability at element and system level towards
the achievement of shape transformation and uniform stress distribution in the members of the system. The components and geometry of the system investigated, such as the telescopic bars connecting the bending-active stripes and the system grid applied in the case study, are to be understood as variables of the structure concept proposed, instead of definite parameters of a specific structure solution. Therefore, this is a proposal of a structural concept and not of a specific structural/architectural solution. In the next section of the paper, the composition of the structure is described. Subsequently the configurability of the structure is investigated in digital simulations and a physical small-scale model, and its load-deformation behavior, through FEA.

2. BENDING-ACTIVE PLATE GRID SHELL

The bending-active members of the structure define its main shape and are responsible for the stability of the overall system throughout the process of erection. The elastic members are fastened together and interconnected through a secondary system of telescopic bars (Fig. 1). The structure is first constructed as a flat closed surface of elastic stripes, arranged with their strong axis vertically. The stripes are interconnected in pairs, and their deployment in form of continuous ellipsoid units is performed by lengthening of the internal telescopic bars, which induce bending to the elastic members, creating a flat system expansion as shown in Fig. 2. The elastic members’ units have openings of 0.5 m and lengths of 7.5 m. The resulting overall grid creates a rectangular ground plan in its flat form with dimensions of 7.5 x 3.5 m and 16 support points at the ends. The structure is then actuated for erection through a system of cables with adjustable length that connect opposite supports respectively. Due to the eccentric anchoring of the cables, by decreasing their length, the structure lifts-up and obtains its single or double curved shape, based on the actuation scenario. The supports enable free displacements on the x and y-axis, in order to enable the system to reach its form-found shape. A single corner support is fixed to the ground, in order for the system to maintain a stable reference point. Peripheral telescopic bars are applied for further deformation control, after the structure has been stabilized.

The bending-active stripes have elastic material properties, with cross section dimensions of 200 x 10 mm. The joints are made of steel plates in form of rings of 8 mm thickness. Steel rings support also the telescopic bars (Fig. 3). The latter are made of
Figure 4: System force distribution and peripheral telescopic bars’ application cases

aluminum hollow sections of 30 mm diameter and may obtain lengths between 300 to 600 mm. The peripheral bars are assembled alongside of the elastic members, with same cross section as the internal ones. In the design of the connections between the elastic members and the telescopic bars, a ball bearing is inserted in a steel envelope welded on the elastic members’ ring. The joints ensure relative rotations of the compression members of up to 50°.

The supports allow horizontal displacements of the structure during its erection, except from a single one, which is anchored to the ground. The elastic members are hinge connected to the supports. The cables have diameter of 12 mm, and their length is adjusted by a hydraulic control mechanism with pulley on the supports.

3. DIGITAL MODELING

In a preliminary investigation, the system has been modeled with the computational design software Grasshopper (plug in of Rhino®), and simulated in its form-found shape using Kangaroo Physics (add-on of Grasshopper) [21, 22]. While Grasshopper is based on algorithmic logic and enables easy determination of any of the geometrical parameters, physical properties can be assigned to the developed geometry in Kangaroo plug-in, which is based on the particle-spring system simulation technique. The interactive analysis enables the visualization of possible configurations, based on the distribution of external forces acting on the supports, representative for respective cables’ shrinkage, and the geometrical modifications of the telescopic bars (Fig. 4). In
principle, the external force distribution on the supports influences the erection of the system and its configuration. In a second assessment stage, the contribution of the peripheral bars to the structural configurations is perceived by using linear spring elements.

3.1. System Behavior

Initially, the application of external forces on the structural supports has been examined in three different distribution cases. Case 1 concerns a uniform distribution of forces on all 16 supports, which creates a horizontal displacement of the supports on the x-axis. In case 2, the forces are equally distributed on the four corner supports only, inducing again a horizontal displacement on the x-axis. In case 3, the forces are applied diagonally on the four corner supports, inducing a movement towards the center of the system, while displacing the supports on both x- and y-axis. The external forces lift the elastic stripes up and create an arched shell (Fig. 5). Initiating with a flat form with system span L, two other cases of L/2 and L/4 span are investigated in this stage.

The second parameter that was taken into account, and applies to all three cases of force distribution, is the application of the telescopic bars with regard to the external periphery of the structure. In case A, the behavior of the system with no periphery bars is investigated. In case B, extra telescopic bars are applied between the supports on the y-axis for maintaining constant distance. In case C, peripheral telescopic bars are only placed on the x-axis. Case D is a combination of cases B and C with application of peripheral telescopic bars in both horizontal directions of the system. In all cases, the telescopic bars’ length is modified after the structure has been lifted-up, in order to investigate their influence with regard to the curvature and the compatibility of the system. In a first sub-stage, the length of the internal bars is increased by l/2, with l, the initial length of the members of 500 mm. In a second sub-stage, alongside with the length of the internal bars, the length of the peripheral bars is also increased by l/2.

The control parameters of external force distribution and telescopic bars’ activation yield 24 different system configurations. The results of the parametric analysis prove that the application of external forces is determinant for the erection and configurability of the gridshell. In cases 2 and 3, where the forces are applied on the four corner supports only, nonsymmetrical configurations develop, due to the flexibility of the structural members to provide uniform system deformations. Especially in case 3 with diagonal direction of the forces acting on the supports, these move towards the center with irregular behavior, leading to nonsymmetrical structural shapes.

The peripheral telescopic bars applied on the y-axis, as in cases B and D, help the structure to keep its symmetry. This is mainly due to the fact that the bars keep the supports at constant distance. The telescopic bars placed on the x-axis have limited contribution to the symmetry of the configurations obtained. They solely support the two external stripes in balancing the curvature, without providing further improvement. In addition, they cause the peripheral edges of the structure to lift-up. With regard to the telescopic bars between the stripes, the most noticeable differentiation is the size of the units’ openings. A length increase of the internal bars causes bending deformations at the top of the structure in span direction, along with an upwards deformation of the edges of the arch. Furthermore, it increases the length of the gridshell on the y-axis, except from the support points.

4. PHYSICAL MODEL

A simplified physical model in scale 1:10 was built for verification purposes of the erection process. Bending-active stripes with cross section dimensions of 20 x 1 mm were interconnected in pairs, and deployed on horizontal plane through assemblage of round steel bars of 4 mm diameter and constant length as shown in Fig. 6. These were connected to the elastic members through eye bolts that allowed rotation of the former during erection. In this stage, the supports slid in span direction (x-axis) while...
allowing rotations of the primary members connected as shown in Fig. 7. Finally, the peripheral bars were implemented. The final configuration obtained refers to the form-found state of the system. The physical model demonstrated the kinematics of the system on one side, but indicated also the significance to the bending-active members stabilization at the form-found state of the system.

The geometrical characteristics and dimensions of the members are identical to the previously investigated digital model. The elastic members are modeled as beam elements, and divided in the longitudinal direction into a number of segments of 50 mm length each, so as to increase the analysis resolution. These are assigned to elastic material properties PTFE (Polytetrafluorethylene) of high elastic modulus, $E = 2.5$ GPa and a relatively high yield strength, $\sigma_{yd}$, of 48 MPa. These values have been used in the analyses, whereas a linear stress-strain behavior of the material has been assumed. Furthermore, the bearing capacity of the elastic members is not calculated realistically, due to the lack of assessment of lateral-torsional buckling. The cables are defined as cable elements with single segment partition, in order to avoid sagging effects. These are assigned to prestressed steel Y1770 (EN1992) of 195 GPa elastic modulus and 1520 MPa yield strength. In the structure’s reconfiguration, the free supports may roll on the x and y-axis. At its form-found state, the supports are then restrained using beam elements with strong sections as anchors, in order for the system to keep its lifted form for the next stages of transformation.

FEA of the system is based on the third order theory and takes into account geometrical nonlinearities, as well as large system deformations. The material selected for the elastic members has a linear elastic stress-strain behavior to allow the analysis to focus on the geometrical aspects of the active formation process. Sequential step-by-step structural modifications are followed through the parametric text editor module Teddy, in order to gradually investigate the structure’s deformations in obtaining the final form-found position. The iterations are set to 200 and the tolerance, to 0.5 for each analysis step. The tolerance limit is set with relative high value, in order to avoid excessive computing time. In each consecutive analysis step, the residual stresses stored by the members and the updated geometry of the system are taken into consideration.

5. STATIC ANALYSIS

A model of the gridshell of case D1 used in the digital simulation stage has been created with the FEA software program SOFiSTiK and investigated using the text editor module Teddy [23, 24]. The selected case D1 includes all possible configurational transitions of the system. The model is recreated in flat, non-deformed shape, in order to investigate the system’s load-deformation behavior in every stage of erection and transformation processes.
5.1. System Transformation Stages

The static analysis of the bending-active gridshell throughout the deformation process is divided in three main stages with certain sub-stages in-between, as clarified in Fig. 8.

![System deformation stages diagram](image)

**Figure 8: System deformation stages diagram**

The first stage refers to the ‘planar deformation’ and considers the expansion of the units by gradually increasing the internal bars’ length at an initial value of 400 mm. The ‘erection approach’ follows in stage 2. In this stage, the reduction of the lower cables’ length, connecting the supports in span direction, modification of the cables’ length in the analysis is achieved gradually, in respective steps of 10-100 mm, in order to avoid nonlinear convergence errors.

In the final ‘morphology optimization process’ of stage 3, further modification of the telescopic bars’ length is induced, in order to examine their influence on the system’s deformation behavior. This stage is divided into four sub-stages of gradual corresponding length modifications, stage 3.1 and 3.2 for the internal bars, as well as 3.1.1 and 3.2.1 for the peripheral bars. The telescopic bars’ length increases by 1/4, i.e. up to 500 mm in stage 3.1 and 3.1.1, and 1/2, i.e. up to 600 mm in stage 3.2 and 3.2.1, whereas l is their initial length (Fig. 9).

**- Erection stage**

The erection approach is decisive for the form-finding of the structure in the transformation process. This stage is divided in three sub-stages of gradual cable shrinkage for a more accurate investigation of the lifting-up progress and the consequent deformations of the elastic members of the structure. The shrinkage of the cables connecting the supports in span direction induces respective displacement of the supports along the x-axis. In each step the cables’ length is reduced by 800 mm resulting to a final length reduction of 2.4 m. In the three sub-stages of gradual cable shrinkage, the respective cable forces amount to 1.13, 0.88 and 0.82 kN. A maximum respective value of 1.9 kN was registered at the beginning of the erection stage. The here from induced compression in the members is necessary for their deformation in the erection of the structure.

**Table 1: Maximum internal forces of the bending-active members (N_x, V_z, M_y, M_z) and telescopic bars (N_{tx}) in the planar deformation and erection process**

<table>
<thead>
<tr>
<th>Stage</th>
<th>N_x [kN]</th>
<th>V_z [kN]</th>
<th>M_y [kNm]</th>
<th>M_z [kNm]</th>
<th>N_{tx} [kN]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-1.10</td>
<td>0</td>
<td>0</td>
<td>0.12</td>
<td>0.10</td>
</tr>
<tr>
<td>2(a)</td>
<td>1.80</td>
<td>-1.28</td>
<td>-0.70</td>
<td>0.15</td>
<td>8.10</td>
</tr>
<tr>
<td>2(b)</td>
<td>2.40</td>
<td>-1.25</td>
<td>-0.69</td>
<td>0.18</td>
<td>2.00</td>
</tr>
<tr>
<td>2(c)</td>
<td>2.80</td>
<td>4.24</td>
<td>-1.08</td>
<td>0.21</td>
<td>2.30</td>
</tr>
</tbody>
</table>

![System transformation stages](image)

**Figure 9: System transformation stages**
The critical bending deformations of the elastic members, concentrate at mid-span, close to the periphery of the structure. Additional deformations occur at the edges of the structure, where the bending-active stripes undergo rotations upwards. The maximum internal forces of the elastic members and telescopic bars are included in Table 1.

The maximum axial force of the elastic members, \( N_x \), increases as the structure is lifted-up, reaching a maximum value of 2.80 kN in the final sub-stage 2(c). The maximum value corresponds to an increase of 16 and 180 % compared to the respective values in sub-stage 2(b) and 2(a) respectively. The maximum axial force in tension develops at the center of the peripheral elastic stripes. The maximum shear force, \( V_z \), increases in sub-stage 2(c) to 4.24 kN, and is 35 % higher compared to the two previous sub-stages. The maximum axial force, \( N_{tx} \), of the telescopic bars develops in stage 2(a), and amounts 8.10 kN. The maximum bending moments of the elastic members increase gradually during erection reaching maximum values of 1.08 kNm for \( M_y \) and 0.21 kNm for \( M_z \). The maximum bending moment \( M_z \) has a constant increase of 0.03 kNm in each sub-stage.

- Elastically deformed stage

Modification of the telescopic bars’ length aims at the development of an appropriate control technique for the structure configurability. Lengthening of the telescopic bars reduces the intense distortions of the elastic members at mid-periphery. When the system reaches the stage of ‘morphology optimization process’, the tension forces of the elastic members increase, especially those close to the supports. This is due to the fact that the internal telescopic bars’ length increase with fixed system supports causes torsional deformations of the elastic members near the supports. By lengthening the peripheral telescopic bars, uniform deformations succeed, reducing thus local torsions of the elastic members. Following lengthening of both the internal and peripheral bars, the maximum tension force of the elastic members develops at mid-span, decreasing in this way the double curvature and the distortions of the elastic members at mid-periphery. The numerical results for all transformation stages and respective system deformations, measured on the middle bending-active stripe at mid-span, are presented in Tables 2, 3.

The maximum axial force value, \( N_{tx} \), increases by 350 % from stage 2 to stage 3, from 2.8 to 12.8 kN. Following stage 3, the maximum axial force of the elastic members has lower fluctuations. As the length of the telescopic bars increases by 1/2 and 1/4 (stages 3.1 to 3.2 and 3.1.1 to 3.2.1) the axial force, \( N_x \) of the elastic members decreases, in contrast to the bending moment \( M_z \), which in the same cases presents a slight increase. Lengthening of all telescopic bars (3.1.1 and 3.2.1) induces lower axial forces in the elastic members, in comparison to stages 3.1 and 3.2, where the transformations apply only to the internal bars. A general observation concerning the axial forces of the elastic members is that these are stressed in tension except in stage 1. Finally, by lengthening the internal telescopic bars as well as the peripheral bars, the maximum bending moment values, \( M_y \) and \( M_z \), of the elastic members decrease by 4 %.

### Table 2: Maximum internal forces of the bending-active members \( (N_x, V_z, M_y, M_z) \) and telescopic bars \( (N_{tx}) \) and system height \( (h_z) \) in transformation stages

<table>
<thead>
<tr>
<th>Stage</th>
<th>( N_x ) [kN]</th>
<th>( V_z ) [kN]</th>
<th>( M_y ) [kN]</th>
<th>( M_z ) [kN]</th>
<th>( N_{tx} ) [kN]</th>
<th>( h_z ) [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-1.10</td>
<td>0</td>
<td>0</td>
<td>0.12</td>
<td>0.10</td>
<td>0.10</td>
</tr>
<tr>
<td>2</td>
<td>2.80</td>
<td>4.24</td>
<td>-1.08</td>
<td>0.21</td>
<td>2.30</td>
<td>2.43</td>
</tr>
<tr>
<td>3.1</td>
<td>-12.80</td>
<td>-1.27</td>
<td>-0.73</td>
<td>0.28</td>
<td>5.30</td>
<td>2.13</td>
</tr>
<tr>
<td>3.1.1</td>
<td>10.10</td>
<td>-1.35</td>
<td>-0.70</td>
<td>0.25</td>
<td>27.90</td>
<td>2.19</td>
</tr>
<tr>
<td>3.2</td>
<td>9.80</td>
<td>1.43</td>
<td>-0.72</td>
<td>0.38</td>
<td>5.20</td>
<td>1.95</td>
</tr>
<tr>
<td>3.2.1</td>
<td>4.90</td>
<td>1.25</td>
<td>-0.70</td>
<td>0.26</td>
<td>6.10</td>
<td>2.12</td>
</tr>
</tbody>
</table>

### Table 3: System configurations through telescopic bars’ length modifications

<table>
<thead>
<tr>
<th>Stage</th>
<th>System_( x-x )</th>
<th>System_( y-y )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>![System_( x-x ) 3.1]</td>
<td>![System_( y-y ) 3.1]</td>
</tr>
<tr>
<td>3.1.1</td>
<td>![System_( x-x ) 3.1.1]</td>
<td>![System_( y-y ) 3.1.1]</td>
</tr>
<tr>
<td>3.2</td>
<td>![System_( x-x ) 3.2]</td>
<td>![System_( y-y ) 3.2]</td>
</tr>
<tr>
<td>3.2.1</td>
<td>![System_( x-x ) 3.2.1]</td>
<td>![System_( y-y ) 3.2.1]</td>
</tr>
</tbody>
</table>
Load-deformation behavior

Once the form-found state of the system has been obtained, the analysis aims at evaluating the bending-active members’ behavior under constant vertical loads of 0.27 kN acting on the nodes, Tables 4, 5. The point loads acting on the 52 nodes of the system derive from a uniform distribution of a surface load of 1 kN/m² acting on the system. In stage 2, when the structure is lifted-up, the application of the vertical loads seems to entirely eliminate the tension forces in all members of the structure. Furthermore, the application of vertical loading to the structure presents some differentiations with regard to the internal forces’ values compared to the form-finding stages of the analysis. The vertical load causes a decrease of the maximum axial force of the elastic members that remains constant with 40% in stages 3.1, 3.1.1 and 3.2. In stage 3.2.1 there is a limited only reduction of the axial force. The maximum bending moments of the elastic members fluctuate between 0.1 to 0.7 kNm. The height at mid-span of the structure seems to have a small increase after loading. This can be justified by the fact that the vertical load pushes the structure downwards and at the same time, the middle area tends to slightly divert upwards.

In following, the structure’s response in stage 2 has been verified under horizontal point loads acting on the nodes. The point loads applied refer to a uniform distribution of a surface load of 1 kN/m² acting on the system in span direction. The maximum axial force, \( N_x \), developed in the elastic members amounts only to 2.3 kN and the maximum shear force, \( V_z \), 0.75 kN. The maximum bending moments, \( M_y \) and \( M_z \), amount to 0.67 and 0.21 kNm respectively. Furthermore, the maximum horizontal deformation of the system, measured on the middle bending-active stripe at mid-span, amounts only to 11 mm.

### Table 4: Maximum internal forces of the bending-active members \( (N_x, V_z, M_y, M_z) \) and system height \( (h_z) \) in transformation stages under vertical loading

<table>
<thead>
<tr>
<th>Stage</th>
<th>( N_x ) [kN]</th>
<th>( V_z ) [kN]</th>
<th>( M_y ) [kNm]</th>
<th>( M_z ) [kNm]</th>
<th>( h_z ) [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>-4.30</td>
<td>-0.91</td>
<td>-0.86</td>
<td>0.20</td>
<td>2.45</td>
</tr>
<tr>
<td>3.1</td>
<td>-5.10</td>
<td>-1.45</td>
<td>-0.72</td>
<td>0.25</td>
<td>2.12</td>
</tr>
<tr>
<td>3.1.1</td>
<td>6.00</td>
<td>-1.38</td>
<td>-0.71</td>
<td>0.28</td>
<td>2.20</td>
</tr>
<tr>
<td>3.2</td>
<td>5.80</td>
<td>-0.96</td>
<td>-0.62</td>
<td>0.31</td>
<td>1.97</td>
</tr>
<tr>
<td>3.2.1</td>
<td>-4.10</td>
<td>-1.09</td>
<td>-0.60</td>
<td>0.28</td>
<td>2.13</td>
</tr>
</tbody>
</table>

6. CONCLUSION

The current paper refers to the design, analysis and configurability issues of an adaptive bending-active plate gridshell. This is a proposal of a structural concept and not of a specific structural/architectural solution. The investigation focused on the configurability and load-deformation behavior of the structure following preliminary simulations and numerical analysis of a particular case study. The gridshell proposed consists of interconnected PTFE stripes placed in parallel and with their strong axis vertically. The stripes are additionally joint together with telescopic bars. These define the topology of the flat grid through induced openings of the elastic units, and are furthermore responsible for deformation control of the gridshell after its form-found shape has been obtained. The structure is actuated for erection through cables with variable length that connect the supports.

In the preliminary particle spring system investigation conducted with the computational programs Grasshopper and Kangaroo Physics, 24 different system configurations have been obtained, based on the distribution of external forces to the supports and the peripheral telescopic bars’ implementation and length modification. A uniform distribution of external forces in terms of cables’ shrinkage connecting opposite supports in span direction, combined with implementation of both, internal and peripheral telescopic bars, has been selected as the case study in a physical small-scale model and for verification with the numerical
analysis, based on the maximum flexibility this case offered compared to the others.

The form-finding process and load-deformation behavior of the structure have been investigated in discrete steps of FEA with the SOFiSTiK software program. The form-finding of the system is divided into three main analysis stages. The first stage refers to the planar deformation of the elastic members, the second stage, to the erection of the gridshell and the third stage, to the modification of the telescopic bars’ length. In the erection stage, the system develops a double curvature. By increasing the length of the telescopic bars between the elastic members, the curvature in transverse direction is minimized, while the system reaches its self-equilibrium with improved local deformations of the elastic members. In this stage, the maximum axial forces increase substantially, in contrast to the bending moments of the elastic members. Following vertical loading of the gridshell in its different configuration states, the internal forces in the elastic members decrease substantially and any tension forces in the elastic members eliminate. Horizontal loading of the system does not induce higher internal forces in the elastic members. The maximum deformation of the middle bending-active stripe at mid-span amounts to L/375 and 0.5%h under the vertical and horizontal loading applied to the system respectively. At this point, it should be also noted that the analysis of the gridshell in the current paper is based on the assumption of a linear elastic stress-strain behavior of the PTFE material, in order to focus on the geometrical aspects of the active formation process. Furthermore, the bearing capacity of the elastic members is not calculated realistically, due to the lack of assessment of lateral-torsional buckling. Shell-type finite elements, instead of beam elements used in the current FEA, would be necessary for identifying lateral-torsional buckling of the members. For application purposes of the concept, the shell should be further modified with regard to the grid and dimensions of the members, in satisfying the strength requirements of the actual elastic members to be applied.

The proposed bending-active gridshell demonstrates at first place the advantages of deployability, self-erection, geometrical transformability and self-stabilization. The simplicity of the geometrical form and constructability of the specific system make it a promising solution for lightweight and adaptive structures.

REFERENCES


SHAPING FORCES; REVIEW OF TWO BRIDGE DESIGN METHODOLOGIES TOWARDS ARCHITECTURAL AND STRUCTURAL SYMBIOSIS

Joris SMITS1a, Peter EIGENRAAM1a, Rafail GKAIDATZIS1a, Dirk Rinze VISSER2a, Kaitlin WONG2b and Stephan WASSERMANN-FRY2c

1aMSc, Delft University of Technology, Julianalaan 134, 2628 BL Delft, the Netherlands, J.E.P.Smits@tudelft.nl, P.Eigenraam@tudelft.nl, R.Gkaidatzis@tudelft.nl
2a MSc CEng RC,2bBE (Hons), 2cBSc, BuroHappold Engineering, 17 Newman Street, London, W1T 1PD, United Kingdom, Katie.Wong@BuroHappold.com, Stephan.Wassermann-Fry@BuroHappold.com, Dirk.Visser@BuroHappold.com

Editor's Note: Manuscript submitted 13 December 2017; revision received 11 March 2018; accepted 23 April. This paper is open for written discussion, which should be submitted to the IASS Secretariat no later than December 2018.

DOI: https://doi.org/10.20898/j.iass.2018.196.884

ABSTRACT

This paper investigates the symbiotic relationship between the architectural appearance of a bridge and the structural design. The research is done by reviewing and comparing the design methodology employed by the first author in the conceptualization of two of his bridges; an early work from 1997 and a recent work from 2017. The review of the early work describes a design methodology that could be described as intuitive design, whereas the later work is the result of computational from-finding and optimization. Parallels are drawn and the historical development of the toolbox of the architect and the engineer is described. The paper analysis the way the two designs were achieved by looking from the perspective of the architect and that of the engineer, two disciplines that nowadays closely work together on the design of a bridge. The paper concludes by identifying the key design considerations to achieve a beautiful yet structurally sound bridge. The question whether beauty can be the sole result of a rational design process towards the most efficient form according to the laws of mechanics, is addressed. This paper demonstrate the belief that when it comes to the design of a bridge, architecture and structure, form and force, are involved in an interdependable and symbiotic relationship.

Keywords: Bridge design, Architecture, Structural design, Optimization, Parametric design, Form-finding, Concrete, FRP

1. INTRODUCTION

Over the past two decades architects have found their way into the practice of bridge design, a field of expertise that was formerly considered the sole domain of structural engineers. Ever since the 90’s a strong growth of the involvement of architects in bridge design has taken place. The beginning of this new era of architectural bridge designs is clearly marked by the realization of the Alamillo Bridge, built for the '92 Seville Expo, by the world famous architect and engineer Santiago Calatrava. His design for a cable stayed bridge stands out for the massive pylon which, by its backward inclination, formed a counterbalance to the forces from the cable stays, thus creating a bold demonstration of the forces at play. At the same time the Alamillo Bridge was a defiance to traditional bridge designers demonstrating that the easiest way to design a cable stayed bridge was not necessarily the only way, and that structurally sound solutions can also be found in a not-so-straightforward approach. Ever since the Alamillo Bridge a closer collaboration between architects and structural engineers has resulted in many beautiful and well-integrated bridge designs all over the world. The downside to this development is that at the same time a lot of farfetched bridge designs have also seen the light of day.
What are the key design considerations to achieve a beautiful and yet structurally sound bridge? Does a structure always need to follow the most efficient form, according to the laws of mechanics and/or finance? Or is there such a thing as symbiosis between Form and Force, a way of working that ensures that the final result becomes greater than the sum of its parts?

Two bridge designs from the author, one marking the beginning of his career in 1997 and the second one only recently accomplished, demonstrate the belief that structure and architecture are involved in a symbiotic relationship. One cannot be successful without the other. Just how successful this interaction is, forms the subject of this paper.

2. SHAPING FORCES

In 1997 W. Zalewski and E. Allen wrote the book ‘Shaping Structures’ for students of Architecture and structural engineering. In the preface it is written that ‘The essence of structural design is to shape each structure to respond effectively to the forces that it must withstand and to the human activities that it nurtures’ [1]. It is interesting to compare Zalewski’s theory with the well-known trinity Venustas, Firmitas and Utilitas described by the influential Roman architect/engineer Vitruvius (80-25 BC) in “de architectura” [2]. Zalewski’s theory addresses both force and utility, or as Vitruvius would put it, Firmitas and Utilitas. However, the aesthetic dimension, Venustas, has been left out of the equation. Or rather an assumption seems to have been made that a structure that responds effectively to the forces and to human needs is intrinsically beautiful.

The title Shaping Forces is based on the well-known adage ‘Form follows Force’; the assumption that an architectural design that follows a path of structural logic also holds a greater aesthetic value. But what exactly is structural logic, and how can it be achieved? There are of course many design methodologies that lead to a structurally logic bridge.

One method is to pursue a minimal use of materials for the required program and load case by following the path of the loads to the foundations in such a way that the least amount of material is used. A very popular approach among academics and professionals nowadays is achieved through computational design using advanced parametric form-finding and optimization software like Grasshopper, Karamba and Kangaroo [3].

One has to acknowledge that these types of form-finding and optimization software are in fact nothing more than a tool to achieve structural logic. The method behind it is not new. An eminent pioneer in this field was Heinz Isler who used his ‘frozen towel’ technique to create poetic natural shells. He states: “One does not actually create the form; one lets it become, as it has to according to its own law.” [4]

Before that Antoni Gaudi used his now famous inverted chain model to find the most efficient vaulted shape for the Sagrada Familia.

A third way of deriving architectural form through structural ideals relies on greater design intuition. Instead of letting the form create itself, such as Isler did (Figure 1), a skilled designer with a profound understanding of structural mechanics and a fine sense of aesthetics can accomplish good results. They can shape a structural geometry based on the functional constraints, a befitting architectural typology that fits the context and an understanding of the forces and materials used. It is this intuitive way of determining a structure that is demonstrated in the first case study on the Navel Bridges in chapter 3.
3. NAVEL BRIDGES IN NIEUW VENNEP

The design of the Navel Bridges in Nieuw Vennep, the Netherlands, is a clear demonstration of the authors’ conviction that structure and architecture are involved in a symbiotic relationship (Figure 2). The Navel Bridges were designed and drafted in 1999 at his architectural office at a time when he was freshly graduated from both the School of Architecture as well as the School of Civil Engineering in Delft [5].

When planning a new thoroughfare road in a new suburb of Nieuw Vennep, the authorities at first considered making a two-short-span bridge, one span over the canal and the other for a bicycle underpass directly adjacent to the bridge.

The first step in the design process was to combine the bridge and the tunnel into one structure spanning both water and bike passage, thus increasing the spaciousness and transparency under the road and improving the perception of the bicyclists of being protected (Figure 3). The chosen material to achieve this span was in situ concrete. This had to do with the specific urban context of the surroundings and the wishes of the municipality to have a sturdy design with little maintenance issues. It was argued that two larger span bridges could be built within the budget if they would be identical (although rotated 180 degrees from each other) and could share the same formwork. An alternative in prefabricated concrete beams was dismissed because both the client and the architect wanted a unique design with a homogenous sculptural appearance that would benefit the identity of the entirely new town.

Second step in the design process was to determine the soffit level underneath the structure, both for bicycles and pedestrians as for boats and ice skaters, to determine the height and alignment of the ceiling. The thoroughfare road was allowed to raise by one meter locally. As it turned out the ceiling needed to be at its highest above the bicycle path, as an optimization between the vertical alignment of the path and the most slender part of the bridge deck.

The short span caused a visual disruption of the recreational water in the park, while at the same time the bicycle passages faced issues of poor visibility on the surroundings.

Figure 2: One of two Navel Bridges in Nieuw Vennep, The Netherlands

Figure 3: Initial proposal for a culvert and an underpass
The resulting elevation of the bridge now showed a vaulted arch with an asymmetrical profile (Figure 4). The asymmetry of the profile determined the static scheme of a clamped connection on the side of the abutment near the water, and a rolling hinge near the bicycle path. Whilst the landing on the side of the bicycle path was relatively slender, a very massive piece of concrete appeared above the water. Therefore, the third step in the design process was to eliminate the surplus of concrete by creating a cavity between the deck and the vault (Figure 5). Sharp inner corners in the concrete cavity were avoided to allow for a fluent flow of stresses, reducing concentrated areas of high stress, and to avoid cracking in the corners. The resulting shape was a combination of a straight, flat slab for the motorized traffic and an arch beneath, which merged with the slab as it rose up vertically. Statically speaking, it is not entirely correct to speak of an arch, as it does not receive any vertical loads after separating from the deck, other than its own weight. One could also see it as a slanted pillar under the deck.
The fourth step in the design process was to further reduce the amount of concrete by tapering the sides of the bridge deck as well as the arch under a 45 degree angle (Figure 7). This resulted in a much lighter appearance, the cavity became shorter and thus more transparent when seen at an oblique angle.

and daylight penetration under the bridge, on the bicycle path and on the water improved greatly.

The design could have stopped there as a pleasing architectural space under the bridge had formed. However, it was soon realised that further weight savings and greater elegance could be achieved by further opening up the vault and the cavity. The fifth and last step, therefore, was to create another cavity at a 90 degree angle to the first cavity along the longitudinal span of the bridge (Figure 8).

A T-junction of cavities was created, splitting the arched vault into two separate arches and opening up unexpected perspectives through these cavities to the surroundings (Figure 6, previous page).

The bridge was completed by designing matching parapets out of concrete and stainless steel. The bridge was accessible to motorised traffic, so the parapets, which acted as side walls, were required to be robust in design. In addition, the design consciously accommodated for unhindered views of the river for drivers. This was achieved by employing low walls with cavities, which succinctly tied in with the overall design of the bridge. The stainless steel railing was kept light and simple with short posts mounted directly on top of the concrete.

4. THE SHARC, BERLIN 2017

Eighteen years after completing the design of the two Navel Bridges, the author’s university team participated in the design competition for a new footbridge for the International Footbridge 2017 Berlin conference, together with the London based offices of BuroHappold (Figure 9). The accepted conference paper focussed on the final product and images, not on the design process itself. The current paper is a review of the used methodology to get to the final design.

The ShArc is a hybrid structure that combines the characteristics of a shell with those of an arc, hence ShArc. The design was created using a computational design methodology by means of parametric software and scripting. The form-finding methods that was used was not limited to optimizing solely the structural behaviour, it was also used to improve the aesthetical and functional design. The iterative design process that was used resulted in a good balance between these three aspects, which interacted in a symbiotic relationship.

Pioneers like Gaudí, Isler and Frei Otto worked on form-finding through physical models such as suspended chain models, frozen textile and soap films. This form of physical form-finding results in one solution for the specific situation. Thereby not taking into account other load combinations that will also be applied during the lifespan of the bridge. The design methodology employed for the ShArc equally started from a unilateral form-finding model, translating self-weight and additional equally distributed loads into a shape that is convenient for the distribution of loads.

The difference however with the methods of physical modelling, described above, is that the authors did not stop when the first correct shape for the constraints was found. The team proceeded
to investigate ways to alter the initial form-finding geometry in order to comply better with different load cases, functional requirements and aesthetical requirements. For this purpose, a next step was made by introducing specific additional loads so that the result of the form-finding would include an solution for other loads than equally distributed. This way the curvatures of the sides, the steepness of the slopes and the transparency of the overall design could be controlled intuitively. For this purpose a script was developed allowing for adaptation of the shape of the model, and showing immediate feedback on structural behaviour. The specific steps for the design of the ShArc in Berlin are now further described.

4.1. Conceptual design

Like all designs, this design was based on an initial idea. The concept is to create more than just a bridge from A to B; but rather a bold design that will become a destination in itself and a cultural landmark for the metropolitan city of Berlin. Instead of the two linear bridges with a medium span, as the program suggested, it was decided to create one bridge connecting the three landings resulting in a tripod-shaped bridge, located at the confluence of the river “Spree“ and the “Landwehrkanal”, connecting the downtown districts Charlottenburg and Moabit, would serve both purposes (Figure 10).

The goal was to span both the Spree and the canal with one fluent and single span structure, each of the three bridge members being reciprocally restrained by the other two. Another design decision was that the deck of the bridge had to be a fluent arc above the water; shallow enough for pedestrians to be able to walk on top of it, but high enough to allow ships to pass under, and for the arc to act in compression. At the confluence of the three bridge members, the bridge should provide a public platform where people can enjoy panoramic views of the surroundings.

It should be mentioned that although the materialisation and detailing of the structure is not a subject for this paper, as it focusses on form...
finding, it was necessary to consider such details in order to conceive a feasible design. The longest span of the bridge is approximately 170m. To achieve the desired fluent and fluid appearance without additional supports, materialisation was rather important. The ShArc is conceived and engineered from a composite sandwich structure material, which is formed from Fibre Reinforced Polymer (FRP) outer layers with a foam or honeycomb-core. Sandwich structures can be created from various combinations of outer layer and core materials, enabling flexibility in the design, with the outer layers designed to resist bending and axial stresses and the core to resist shear [7]. For the ShArc, it was proposed to use Glass Fibre Reinforced Plastic for the outer layers and a foam core material. These materials were chosen for their high compressive and shear strength properties, as well as low self-weight [8]. High compression strength also boded well for the arch-like structure incorporated into the design, as pure arches behave in pure compression, and in turn relieving the induced bending stresses in the structure and reducing deflections; through the design process, the optimum arch radius was evaluated, trading off the structural implications and functionality of the bridge. In order to reduce the deck weight further, and to maintain the open character, it was decided to create an opening in the deck at the junction of the three “legs”, directing the flows of pedestrian around the void. Reducing the weight of the deck had structural benefits by reducing deflections and high stresses in these areas. Another advantage of FRP is that it can be easily moulded, which allows the process of creating the curvaceous deck to be considerably easier than traditional materials.

Figure 11: 3D printed Daedalus Pavilion at the GPU

Figure 12: First sketches for Berlin; introducing a double surface within a shell structure
Initially the inspiration for the shape came from the 3D printed prototype of the Daedalus Pavilion, presented at the GPU Technology Conference in Amsterdam (Figure 11). The pavilion has an intricate shape with a double layered deck in the central part, consisting of an upper and a lower deck crossing over in a void (figure 12). Although this idea was later abandoned, as it proved impossible to create enough distance between the two decks for a person to be able to walk underneath, it lead to further use of the parametric script as the main design tool. Therefore pursuing the development of the idea proved to be crucial to the process of the design. It challenged the authors to use form finding to manipulate the shapes intuitively in a way that was aesthetically pleasing and not necessary resulting in a structural optimal shape. In this first step this was merely a convenient side effect and later implemented as part of the final design. The shape was manipulated by differentiating the ratio between loads and stiffness in the form finding model.

4.2. Digital form finding

Digital form-finding was used for exploring various possible geometries for this complex bridge. Therefore Grasshopper and Kangaroo were used. Both well-known and often applied software for form-finding. The first rough model of the desired shape allowed to create a model with surfaces that overlap in the middle (Figure 13). Having physical modelling in mind it would be impossible to create this shape when starting from a single membrane because at the centre there are two layers on top of each other.

For the initial model two ways to influence the geometry of the structure were applied; differentiating the stiffness within the membrane as if it was non-uniform in different directions but also in different areas of the entire model. A second method was to influence the shape of the structure by including line loads at the perimeter of the model, additionally to the equally distributed loads, which created more curvature within the cross section. This second method was predominantly used to fine-tune the shape of the bridge.

Working with a parametric script provided a great amount of design freedom. It allowed to intuitively modify the model. For example, it became much easier to move the landing areas along the quays in order to create bridge members that were more equal in length, and thus had less steep ramps.

4.3. Physical form-finding

Form-finding within a digital environment proved to be very powerful while allowing for a great amount of flexibility to modify the model. However, for structural purposes it does not necessary provide insight in the structural behaviour. Furthermore numerical models provide quantitative information which does not necessary lead to qualitative information. Therefore, parallel to creating the parametric script, physical form-finding tests were performed using experiments in fabric, paraffin and gypsum (Figure 14). Physical models provide insight. Stiff and flexible parts can be identified quickly by applying loads by hand. Since models are often fragile the weaker parts break when the model is subjected to less gentle pushing. Thereby they are easily identified as well. Also the overall shape provides a reference for the shape resulting from form-finding within a digital environment. Physical modelling provides a context to think about the consequences of different shapes and boundary conditions or where to apply stiffening measures. The cutting pattern for example influences on the resulting shape, as does the positions of the three support points. The type of fabric also influences the shape. A microfiber cleaning cloth was used which has uniform stiffness in all directions.

The results provided a reference for the shape that was form-found digitally, using only a distributed load that represented self-weight. Also, by having a physical model, structurally weak places could be identified quickly. Therefore the model had to be damaged, but multiple models could be made easily. One essential flaw that was particularly demonstrated was that the model had a tendency to become flat in the perpendicular direction of the span. This made it sensitive for asymmetric loading, introducing bending in the structure.
Making a physical model created awareness of the aspects influencing the digital model. E.g. the initial layout, stiffness of the membrane in multiple directions and positions of the supports. Also, a physical model provides a sense of scale to the designer. Something that in a digital model is easily lost.

4.5. Reflection on the performance

Both the physical models and the parametric model demonstrated one weakness in the shape. The shell was initially optimized for one load case only; that of an equally distributed load, the self-weight. However, as different load cases do not result in the same deformed shape, and will deflect according to their own load take-down (Isler), the physical model showed to be weak when subjected to asymmetric loading. Since the model was rather slender it was expected that other load combinations than self-weight could have significant effect. The first models resulted in a rather flat cross-section of the deck along the entire span of the bridge. The flatness wasn’t well suited to resist the asymmetric loading; A flat geometry behaves in a beam-like manner, so induces higher bending stresses. Therefore it was required to design for more resistance to bending stresses in certain areas. In other words, the second moment of area of the cross section had to be increased at certain points. This can be done by increasing the structural height. This was applied in a gradual manner reducing towards the supports (Figure 15).

Manipulating the shape of the shell also altered the stress patterns throughout the structure. This resulted in two different curvatures for the top and bottom of the bridge. The thickness of the top and bottom FRP layers are able to be adjusted to suit the stresses in the panels throughout the structure.

4.6. Elaborating the parametric model

Based on early findings the parametric model was modified in Kangaroo to create more resilience to bending and buckling. In order to increase the second moment of area the sides of the cross section were curved upwards by adding virtual line loads along the perimeter (Figure 16). This unequal distributed load was in total approximately similar in magnitude to the total load of the equally distributed load. Therefore the overall shape of the model was similar but now with a curved cross section.

Another influence on the curvature of the deck was the shape of the supports at the bridge abutments. While a straight line support (and resultant flat deck) only offers limited stiffness, a concave line extends the ‘half-pipe’ (and more moment-resisting) section through to the pier. This would have consequences for the distribution of stresses which should be taken into account later. For
aesthetic reasons the walking surface at the landings changed from sinclastic to anticlastic. By raising the edges the second moment of area also increased at the area of transition.

Figure 17: Subdivision of deck (left) and grid (right)

For modelling purposes the model was subdivided into nine surfaces, each one subsequently divided in a grid and diagonals (Figure 17). This set-up provided means to differentiate the stiffness in different directions.

Figure 18: A grasshopper script was written to monitor the resulting slopes

While the authors were constantly modifying the geometry, realisation came that one of the most important functional requirements of the bridge, the slope percentage, was also constantly changing. In order to get visual feedback on the slope percentage from the model an addition to the script was made to provide direct feedback on actual slopes Figure 18). This became a useful tool to balance fulfilling requirements of different nature, namely structural, aesthetic and functional.

During the process of form-finding, one of the challenges was to prevent the shape of the bridge from wrinkling. This can be seen in Figure 21. Here the abutment remains fixed while the edges of the bridge are curved up- and inward. This proved to be problematic for both practical and structural reasons. A wrinkled surface would not be a good surface for further modelling. Structurally, a wrinkled surface would not transfer loads in a distributed way. The analytical model therefore had to be tidied up before analysis, rendering the process less efficient. The wrinkling effect could be prevented by changing the properties of the line elements in the parametric model by setting a rest length smaller than the original length. This would be equivalent to changing the length of the springs during the form finding.

Figure 19: Wrinkling near the abutment

Like all shells, slenderness comes at a price. The bridge had the tendency to buckle laterally at the edges of the deck. Stiffening these edges by adding a separate edge beam was undesirable from an architectural point of view. Instead, the edges were strengthened by curling them, very much like a water lily leaf, to create thicker flanges.

4.7. Use of FEM with the Grasshopper script

The parametric set-up that was used allowed direct communication between the Grasshopper model and the FE analysis software (Robot). This meant that all FE input data could be centrally-controlled...
via the GH interface, and all resultant FE output data (deflections and stresses) fed back to the script.

Relying on a rigorous node and panel-numbering system, each panel is individually selectable and properties controllable, allowing us to assign specific panel thicknesses and loads, model large-scale gradients of loading-scenarios, and effectively analyse a more realistic cross-sectional geometry.

FEM was used to determine areas for perforations. As with the concrete bridge design method mentioned before for the Navel Bridges in Nieuw Vennep, from the FEM analysis we were able to determine the areas of high and low stress. This ultimately indicated the areas where redundant material had been provided, informing the locations and sizes of the perforations. It was essential to implement the numbering system in the script. In such a way the tool could be used for both intuitive modification and structural analysis.

In order to optimize the structure and use the material to its full potential, an iterative Grasshopper form-finding process is adopted. The sandwich composition of glass-fibre outer layers with honeycomb interior provide only limited out-of-plane shear capacity. Instead, the doubly-curving geometry of the ShArc allows shear transfer to the curving edges to be transmitted to the piers via the parapets and hull, which we need to encourage the Kangaroo relaxation algorithm to converge to.

Weighting functions are developed for each of the utilization factors as follows (Figure 20). This is to promote a high utilization in-plane (approaching 0.8), a low out-of-plane shear utilization and as low deflections as possible. The functions are at this point not academic and mainly serve comparative purposes.

\[
\begin{align*}
    f(UF_t) &= f\left(\frac{\tau_{zz,i}}{\tau_{max}}\right) = \frac{k_1}{\tau_{zz,i}} \\
    f(UF_\sigma) &= f\left(\frac{\sigma_{v,Mises,i}}{\sigma_{max}}\right) \\
    &= \left(k_2\left(1 - \frac{\sigma_i}{\sigma_{max}}\right)\right)^{-1} e^{-\ln\left(k_2\left(1 - \frac{\sigma_i}{\sigma_{max}}\right)\right)^2} \\
    UF_\delta &= \frac{\delta_i}{\delta_{lim}} \\
    UF_i &= UF = \frac{1}{3} \sum (f(UF_t) + f(UF_\sigma) + UF_\delta)
\end{align*}
\]

With:

- \(\tau_{zz,i}\) = out-of-plane shear at each node
- \(\tau_{max}\) = out-of-plane shear capacity
- \(\sigma_{v,Mises,i}\) = in-plane stress at each node
- \(\sigma_{max}\) = in-plane stress capacity
- \(\delta_i\) = deflection at each node
- \(\delta_{lim}\) = deflection limit
- \(k_{i=1,3}\) = weighting factors

The average value of the utilization factors is calculated, and used as a utilization factor singular to each node (\(UF_i\)). The utilization factors for each node can then be plotted against their location relative to the bridge, either in a 2D plot graph or directly over the 3D geometry. As a result, areas of low material utilization can be identified rapidly and highlighted for required geometric alterations.

Shown are node utilization factors (\(UF_i\)) along the y-axis, with their relative position to the centre of the bridge shown along the x-axis (Figure 21 and 22). As can be seen between A and C, significant improvement can be made to make the material fully-utilized. A low point is shown at C, close to the centre, where the high stress concentrations associated with the piers are not as typical. The slight arch of the bridge and wide deck width presumably prevent the high flexural stiffness’s typically associated with the mid-spans of structures.

The step at B is a result of an averaging function that we used to avoid the high local stresses at the corners where the two tracks meet around the opening, which we deem will require constructive mitigation and so deem to be unrepresentative.
Future scope for research include expanding the number of utilization factors to include factors such as susceptibility to dynamic excitation, material quantity, gradient of the bridge deck and visual prominence/height of the bridge above the water.

Ultimately, a global fitness criteria is to be created ($UF_{glob,j}$), which provides a reading for the working efficiency of a specific geometry for all affecting parameters. As geometric input parameters are altered, the resultant utilization factors can then be tested for their improvement to the global utilization ($UF_{glob,j+1} > UF_{glob,j}$) and the change retained or discarded as appropriate.

With sufficient computing power, this approach can be extended to the use of an evolutionary solver, with the geometric variables as input parameters and $UF_{glob,j}$ as the fitness criteria. As the analysis runs through multiple iterations, it is expected the analysis will converge toward a material and structural optimum.

5. CONCLUSIONS

1. In order to achieve symbiosis between architecture and structure in integral bridge design architects and structural engineers must be willing to overcome the current division between the work of the architect and the work of the structural engineer and get rid of the classical hierarchy.

2. A pure and self-contained form of bridge design is possible when the designer observes a degree of self-restraint to stay within the boundaries of the forces at play. A bridge design must follow the laws of static, allowing minimal manoeuvre space for frivolity. This way each design visualises its own display of forces, showing nothing more than itself.

3. At the same time it is important to acknowledge that a bridge design cannot be simplified as a mere display of forces. A coherent design is just as much influenced by thorough response to the boundary conditions imposed by the context, the choice of material, the building process and the maintenance and financing of the bridge. A beautiful optimization design has little added value to society if it is impossible to build, maintain or finance.

4. Today, the need to carry out experiments and physical tests with scale models is put into question with the ability to use the computer as a tool for optimization and a way to search for new forms. But how useful is the computer really? In an interview with Juan Maria Songel in 2010 Frei Otto stated “The computer can only calculate what is already conceptually inside of it; you can only find what you look for in computers. Nevertheless, you can find what you haven’t searched for with free experimentation.” [6]

5. Although the tools have changed over the last 18 years, the methodology and the design parameters have remained the same.

6. Just like design methods in the pre-computational period, computational design allows for intuitive design. Through parametric models and graphic scripts, an interactive design process can be created that is open to both architects and structural engineers.
7. Parametric design allows for exchange of disciplines in a multidisciplinary process.

8. A parametric model allows control over aspects that are hard to influence in a physical way.

REFERENCES


INFORMATION FOR CONTRIBUTORS
TO THE JOURNAL OF THE IASS

The Journal of the IASS welcomes contributions on topics related to shell and spatial structures as described on the inside of the front cover.

Contributions must contain material previously not published, or not readily available to the members of the Association. Submission of a manuscript shall be interpreted as an assertion by the author(s) that it is not being considered for publication elsewhere. If the contribution contains materials bearing a copyright, author(s) must submit documentation from each copyright holder granting permission to reproduce such material in the Journal, and the material must be properly acknowledged by reference citations and/or with credits in the captions of photos, figures and tables.

The journal is published in full color. Therefore, color photos and the use of color in figures, tables and equations are encouraged.

All contributions should be submitted in electronic form, as a pre-formatted Microsoft Word .doc file or .docx file. Upload through the Journal’s ScholarOne website: https://mc04.manuscriptcentral.com/iass_journal.

To use this website, authors will have to create an account for their first submission, or login if they already have an account. After logging in, authors will be guided by the on-line system through the various steps of the submission process, with the option to find help on the top of the login page and to select “Author User Guide” for downloading the file: Author.pdf that has detailed instructions to authors.

All material submitted for publication shall be evaluated as to editorial and technical content by a rigorous peer review process that enlists member and non-members of the Editorial Committee. The Editor-in-Chief, in consultation with the Editorial Committee, reserves the right to accept or reject any manuscript. Typically, the review process provides constructive suggestions for revisions of manuscripts to render them acceptable for publication.

The copyright for a Technical Paper or Project Description usually remains with the author(s), but they may elect to assign the copyright to the IASS. As a prerequisite for publication, a completed and signed IASS Copyright Declaration must be uploaded during the review process or prior to publication. This form may be downloaded from https://mc04.manuscriptcentral.com/iass_journal, on the top of the login page, click on the button: Instructions and Forms.

Authors of accepted contributions will receive free of charge an electronic reprint in the form of a protected but printable .pdf file after publication of the article in the Journal. Be also aware that such a protected file will be downloadable to any member or subscriber from the online journal site on the members-only portion of these webpages. And the public will be able to purchase protected .pdf files of individual articles from: http://www.ingentaconnect.com/content/iass/jiass.

An MS Word template and instructions for Technical Papers and Project Descriptions can be downloaded from: https://mc04.manuscriptcentral.com/iass_journal on the top of the login page, click on the button: Instructions and Forms.

TECHNICAL PAPERS. It is required that authors use the template to compose their contributions directly in the two-column form for publication with equations, tables, and figures & photos embedded in the manuscript document and with figure/photo captions and table titles included. If some equations or figures are too wide for the two-column format, they may extend across the width of the page as long as the exterior margins of the template are maintained.

The maximum length of Technical Paper manuscripts shall be fourteen (14) pages, including figures and tables. Brevity is most strenuously encouraged.

PROJECT DESCRIPTIONS. The principal criterion for an acceptable Project Description is that the project should have innovative or distinctive features that are clearly described. In addition, the description should not exceed four (4) pages, should conform to the format of the template, and should include clear photos and/or drawings of the project as well as brief explanatory text.

As an exception to the length limit of 4 pages, if there are very significant technical innovations represented by one or more projects, it would be appropriate for the contributor(s) to submit, instead of a Project Description, a full-length Technical Paper about these innovations using the project(s) as example(s) of implementation.

DISCUSSIONS. Discussion of any Technical Papers and Project Descriptions published in the Journal of the IASS is encouraged and, after review, these will be published, possibly together with a response from the author(s). The deadline for submission of the written Discussion is six months from the date of publication of the original contribution. The maximum length of a discussion of a Technical Paper should be six (6) pages, while a discussion of a Project Description should not exceed two (2) pages.

References cited in the original contribution should not be repeated, but should be cited by the original number; while new references should be numbered to continue the original sequence. The Editor-in-Chief, in consultation with the Editorial Committee, reserves the right to accept or reject any Discussion, but such rejection is reserved for unusual cases of non-professional dialogue.