Final year project / Master thesis
presented at the INSA Strasbourg

Active-Bending Hybrid Structures

Parameter optimisation of restraining membrane

Fabienne GARTI
Civil Engineering

Berlin, 12 Juin 2015

Tutor INSA: Prof.Dr.Chazallon
Tutor UdK: Prof. Dr.-Ing.Gengnagel

KET Lehrstuhl für Konstruktives Entwerfen und Tragwerksplanung
Prof. Dr.-Ing. Gengnagel - Universität der Künste Berlin
Acknowledgements

I would like to address my sincere acknowledgements to:

My master thesis tutor, Prof. Dr.-Ing. Christoph GENGNAGEL, thanks to I joined his research laboratory, KET, and who had been sharing his experience during the whole project.

KET- Department for Structural Design and Technology of the University of Arts Berlin, to let me work in their offices, and to have provided me with all necessary equipment and comfort in a unique atmosphere.

The researchers M. Sc. Michel SCHMECK and M. Eng. Gregory QUINN, for their pleased help, and always enthusiastic presence and availability during this master-thesis research.

¹Chair of Structural Design and Technology
²University of the Arts Berlin, Germany
Abstract

Actively bent hybrid structures are an emerging set of structural systems in the field of lightweight construction. They combine active-bending compression members in interaction with tensile restraining membranes. Active-bending is a way to easily create curved structures from initially straight elements. Advantages of this method are simplified production, transportation and assembly. However, the bending process introduces residual stresses into the bent element. To increase load-bearing capacity of the whole structure, a restraining membrane is added. The scope of this master thesis is to study the effect of membrane fibres orientation, stiffness and pre-stress on a simple planar hybrid structure, within the SOFiSTiK® finite element software environment. It was shown that for the analysed structure, more pre-stress has a negative effect on the load-bearing capacity, that an anisotropic membrane with a horizontally stiffer direction provides a better behaviour compared to an isotropic stiff membrane, and finally that diagonal membrane fibres significantly increase the load-bearing capacity of the structure.

Keywords

Hybrid structure, Active bending, Restraining membrane, FEM (Finite Elements Method).

Résumé

Les structures hybrides en flexion active sont un ensemble émergent dans le domaine de la construction légère. Elles combinent des éléments comprimés en flexion active, en interaction avec des membranes contraignantes, en tension. La flexion active est un moyen efficace de produire des structures incurvées, à partir d’éléments initialement rectilignes. Les avantages de cette méthode sont une production, des transports et un assemblage simplifiés. Toutefois, le processus de flexion entraîne des efforts internes importants dans les éléments fléchis. Pour augmenter la capacité de chargement de la structure entière, une membrane stabilisante est ajoutée. L’objectif de ce projet de fin d’études est d’étudier l’effet de l’orientation des fibres de la membrane, de sa rigidité et de sa précontrainte sur une structure hybride plane simple, à l’aide du logiciel d’éléments finis SOFiSTiK®. Il a ainsi été montré que pour la structure étudiée, la précontrainte a un effet négatif, qu’une membrane anisotrope avec les fibres les plus rigides orientées horizontalement est un meilleur choix qu’une membrane rigide isotrope, et enfin que des fibres orientées diagonalement augmentent significativement la résistance de la structure.

Mots-clefs

Structure hybride, Flexion Active, Membranes Contraignantes, MEF (Méthode des Eléments Finis).
List of abbreviations:

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAD</td>
<td>Computer Aided Design</td>
</tr>
<tr>
<td>GFRP</td>
<td>Glass Fibres Reinforced Polymer</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Modelling</td>
</tr>
<tr>
<td>LC</td>
<td>Load Case</td>
</tr>
<tr>
<td>SSD</td>
<td>SOFiSTiK® Structural Desktop</td>
</tr>
</tbody>
</table>
Table of figures:

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stress level calculation into a bent beam [1]</td>
</tr>
<tr>
<td>2</td>
<td>Simulation of the roofing in SOFiSTIK® Source: <a href="http://www.spectrum.com/">http://www.spectrum.com/</a></td>
</tr>
<tr>
<td>3</td>
<td>Testing of the 11 x 12 meter span structure in Stuttgart Source: <a href="http://www.spectrum.com/">http://www.spectrum.com/</a></td>
</tr>
<tr>
<td>4</td>
<td>Material Equilibria Installation - ICD Stuttgart Source: <a href="http://icd.uni-stuttgart.de/?p=7636">http://icd.uni-stuttgart.de/?p=7636</a></td>
</tr>
<tr>
<td>5</td>
<td>GFRP profiles, one yielded with apparent glass fibres Source: Fabienne Garti</td>
</tr>
<tr>
<td>6</td>
<td>Typical pultrusion line Source: <a href="http://www.fibrotec.es/">http://www.fibrotec.es/</a></td>
</tr>
<tr>
<td>7</td>
<td>PVC coated polyester fabric Source: <a href="http://www.sattler-global.com">www.sattler-global.com</a></td>
</tr>
<tr>
<td>8</td>
<td>Basic element of the tower</td>
</tr>
<tr>
<td>9</td>
<td>Illustration of the parametric study Source: Max Marshall</td>
</tr>
<tr>
<td>10</td>
<td>1:4 scaled physical models (height around 2m)</td>
</tr>
<tr>
<td>11</td>
<td>3D scan of tower number 4 superposed to virtual model</td>
</tr>
<tr>
<td>12</td>
<td>Membrane 2D patches</td>
</tr>
<tr>
<td>13</td>
<td>Testing of the digital generation of patches on 1:4 physical models</td>
</tr>
<tr>
<td>14</td>
<td>Construction of the 1:1 model</td>
</tr>
<tr>
<td>15</td>
<td>Fabrication of the first 1:1 patch</td>
</tr>
<tr>
<td>16</td>
<td>Hybrid tower prototype exposed at the Danish Design Museum</td>
</tr>
<tr>
<td>17</td>
<td>Gridshell studied by Elisa Lafuente Hernández</td>
</tr>
<tr>
<td>18</td>
<td>Tested planar GFRP grid [1]</td>
</tr>
<tr>
<td>19</td>
<td>Construction stages for an active-bending hybrid structure</td>
</tr>
<tr>
<td>20</td>
<td>P-delta effect for a clamped column</td>
</tr>
<tr>
<td>21</td>
<td>Initial system</td>
</tr>
<tr>
<td>22</td>
<td>Flow chart representation of the simulation approach using SOFiSTIK</td>
</tr>
<tr>
<td>23</td>
<td>Geometry dawn in Rhinoceros 3D®</td>
</tr>
<tr>
<td>24</td>
<td>Initial exported geometry</td>
</tr>
<tr>
<td>25</td>
<td>Orientation of QUAD elements</td>
</tr>
<tr>
<td>26</td>
<td>Beam bending iterative process through shortening of a virtual cable</td>
</tr>
<tr>
<td>27</td>
<td>Beam instabilities with iterations ran up to load case no.74</td>
</tr>
<tr>
<td>28</td>
<td>Programming for the beam-bending step</td>
</tr>
<tr>
<td>29</td>
<td>Beam stresses graphical evaluation</td>
</tr>
<tr>
<td>30</td>
<td>Not updated coordinate systems after form-finding</td>
</tr>
<tr>
<td>31</td>
<td>Updated local coordinate system after form-finding</td>
</tr>
<tr>
<td>32</td>
<td>Programming for the stresses output and geometry update</td>
</tr>
<tr>
<td>33</td>
<td>Hem sleeve with rope as edge element [8]</td>
</tr>
<tr>
<td>34</td>
<td>Rope edge with sewn-on webbing [8]</td>
</tr>
<tr>
<td>35</td>
<td>Detail of links between beam and membrane</td>
</tr>
<tr>
<td>36</td>
<td>Bad link implementation consequence</td>
</tr>
<tr>
<td>37</td>
<td>Programming for the linking step</td>
</tr>
<tr>
<td>38</td>
<td>Cable force relation with membrane pre-stress and radius [6]</td>
</tr>
<tr>
<td>39</td>
<td>Visualisation of crushed QUAD elements under pre-stress without point A support</td>
</tr>
<tr>
<td>40</td>
<td>Additional beam curvatures due to a too high pre-stress of 0.5 kN/m</td>
</tr>
<tr>
<td>41</td>
<td>Instability due to a pre-stress of 0.09 kN/m, under a load of 1 kN</td>
</tr>
<tr>
<td>42</td>
<td>Programming for the pre-stressing step</td>
</tr>
<tr>
<td>43</td>
<td>Relaxing step effect</td>
</tr>
<tr>
<td>44</td>
<td>Programming for the relaxing step</td>
</tr>
<tr>
<td>45</td>
<td>Position of punctual axial load</td>
</tr>
<tr>
<td>46</td>
<td>Loading step effect</td>
</tr>
<tr>
<td>47</td>
<td>Programming for the loading step</td>
</tr>
<tr>
<td>48</td>
<td>Vector representation for nodal displacement [mm]</td>
</tr>
<tr>
<td>49</td>
<td>Vector visualisation for membrane forces [kN/m]</td>
</tr>
</tbody>
</table>
Figure 50: Bending moment My [kN.m] ................................................................. 51
Figure 51: Beam Von Mises stresses [MPa] ............................................................... 51
Figure 52: Systems after the pre-stressing process ................................................. 52
Figure 53: Initial system tested without membrane ................................................. 52
Figure 54: Beam maximal Von Mises stress during increasing axial loading for different pre-stresses .... 53
Figure 55: Beam maximal bending moment My under increasing axial loading for different pre-stresses ........ 53
Figure 56: Node A displacement under increasing axial loading for different pre-stresses .................. 54
Figure 57: Maximal membrane force under increasing axial loading for different pre-stresses ........ 54
Figure 58: Orientation of anisotropic membrane ....................................................... 56
Figure 59: Beam maximal Von Mises stress during increasing axial loading for different degrees of membrane stiffness ................................................................. 57
Figure 60: Maximal bending moment My in the beam under increasing axial loading for different degrees of membrane stiffness ......................................................... 57
Figure 61: Node A deflections under increasing axial loading for different degrees of membrane stiffness ...... 58
Figure 62: Maximal membrane force under increasing axial loading for different degrees of membrane stiffness .............................................................................................. 58
Figure 63: Node A and maximum deflections under increasing axial loading for diagonally and longitudinally oriented fibres ................................................................. 60
Figure 64: Maximal membrane force under increasing axial loading for diagonally and longitudinally oriented fibres .............................................................................................. 60
Figure 65: Beam maximal Von Mises stress during increasing axial loading for diagonally and longitudinally oriented fibres ................................................................. 61
Figure 66: Maximal bending moment My in the beam under increasing axial loading for diagonally and longitudinally oriented fibres ................................................................. 61
Figure 67: Principal membrane forces orientation for longitudinally and diagonally oriented fibres ............ 62
List of tables:
Table 1: Five membrane types  Source: www.sattler-global.com ................................................................. 15
Table 2: SOFISTIK software organisation ........................................................................................................ 24
Table 3: GFRP specifications ................................................................................................................................ 30
Table 4: PVC anisotropic membrane specifications .............................................................................................. 30
Table 5: Results for first load case for different pre-stresses .................................................................................. 55
Table 6: Beam max. Von Mises stress due to the pre-stressing process ................................................................. 55
Contents

1. Context ................................................................................................................................. 10
   1.1. Active-bending hybrid structures ...................................................................................... 10
   1.1.2. Materials ...................................................................................................................... 13
   1.1.3. Problematic .................................................................................................................. 15
   1.2. Application on a hybrid tower prototype with CITA institute in Copenhagen ................. 16
   1.3. Previous research on the effect of a restraining membrane ............................................ 20

2. Methodology ............................................................................................................................ 22
   2.1. Numerical analysis tool .................................................................................................... 22
   2.2. Active-bending hybrid structures simulation principles .................................................... 22
   2.3. SOFiSTiK® working .......................................................................................................... 23
      2.3.1. Software organisation ................................................................................................. 23
      2.3.2. SOFiSTiK® parameters: theoretical background ......................................................... 24

3. Case study ................................................................................................................................ 28
   3.1. Settings ............................................................................................................................. 28
      3.1.1. Geometry ..................................................................................................................... 28
      3.1.2. Materials ..................................................................................................................... 29
   3.2. Modelling ........................................................................................................................... 31
      3.2.1. Modelling steps ............................................................................................................ 31
      3.2.2. Geometry input ............................................................................................................ 33
      3.2.3. Beam form-finding ...................................................................................................... 35
      3.2.4. Stresses output and geometry update ............................................................................ 38
      3.2.5. Membrane-beam links implementation ....................................................................... 41
      3.2.6. Membrane pre-stress ................................................................................................. 44
      3.2.7. Relaxation .................................................................................................................... 46
      3.2.8. Loading ....................................................................................................................... 48
   3.3. Parameters of the study ...................................................................................................... 50
   3.4. Results ............................................................................................................................... 50
      3.4.1. Output visualisation and analyse ................................................................................ 50
      3.4.2. Influence of membrane pre-stress ............................................................................... 51
      3.4.3. Influence of membrane stiffness ................................................................................ 56
      3.4.4. Influence of membrane fibres orientation ................................................................. 59
Introduction

German architect Frei Otto, pioneer in the lightweight construction field, has won this year the prestigious Pritzker Architecture Prize. This award pays homage to his life work. One of his major contribution consists in exploring and developing construction with modern fabric roofs over tensile structures.

Such lightweight structures, use minimal amount of material because of their optimal shape. Moreover, due to their production and construction process, they allow large economical and time savings. Those properties advance lightweight structures as a relevant answer to current ecological, social and economic construction issues.

Active-bending structures follow the principle of continuous rods progressively bent up to a defined position and curvature. The final realized shape results from the form-finding of the structure itself.

In order to increase the load-bearing capacity of such structures, a restraining membrane is connected to the actively-bent elements. A shape stabilisation effect then occurs, through the interactive force equilibrium between the two components. In order to evaluate and analyse this balance, the finite element method is used.

The scope of this master-thesis is to investigate the influence of the restraining membrane parameters on the load-bearing capacity of an active-bending hybrid structure. A planar simple active-bending hybrid structure will be analysed for varying parameters under loading, thanks to the finite element method.
1. Context

1.1. Active-bending hybrid structures

1.1.1. Active-bending hybrid structures characteristics

Structural aspects
Active-bending hybrid structures are composed of elastically bent rods, working in interaction with restraining membranes.

The initially straight beams are progressively bent into a curved shape. The structure needs to stay in the linear elastic domain until the desired curvature radius is reached. In order to fulfil this high limit strain condition, the chosen material must have a low elastic Young’s modulus.

The shaping process of those bent elements generates stresses in the beam. Those are called the post-bending residual stresses, and follow the relation:

\[ \sigma = \frac{E \cdot z}{R} \]

with \( \sigma \) being the stresses in the beam, \( E \) its Young modulus, \( z \) the height between the edge and the neutral axis of the beam, and \( R \) its curvature radius (Figure 1).

Figure 1: Stress level calculation into a bent beam [1]
To keep the residual stresses at an acceptable amount, the variable \( z \) must also stay small. Only slender beams are therefore used.

The low cross-sections of profiles provide them with very weak axial and bending stiffness, respectively \( E \cdot A \) and \( E \cdot I \), with \( A \) being the section area, and \( I \) the inertia of the profile cross section. Active-bending members are therefore extremely sensitive to buckling. To prevent this phenomenon from happening, a shape stabilisation is achieved through the implementation of a tensile membrane, restraining the slender compression members.

The idea behind this principle is to avoid further bending forces under external impact. The membrane restrained rod therefore receives only additional axial forces.

**Constructive aspects**

The principle of elastically bending initially straight elements, as a self-formation process, presents numerous advantages.

First, the use of initially straight elements facilitates handling, transportation and storage of material. Additionally, the fact that rods are continuous avoids a lot of assembling steps. Compared to curved structures, the construction and erection process is substantially simplified and shortened: rods can first be pre-assembled as a planar grid on the ground, before being erected. The erection methods then also require a reduced equipment, which is another reason for time and costs savings.

Also for constructive aspects, the choice of membrane and beams degrees of stiffness should take into account the more difficult handling and implementation in case of high values.

In order to avoid wrinkles due to compression in the membrane, the structural fabric is pre-stressed during implementation.

**Examples**

The Marrakech membrane roofing is an innovative membrane structure, which supporting structure comprises 7.5 m long, elastically-bent fibreglass rods (in pockets). The 11 x 12 m span was successfully tested. The patio roofing is now installed in Marrakech. This structure was developed with HTF Stuttgart, ITKE and the engineering office of Julian Lienhard, Str.ucture (Figure 2 and Figure 3).
Figure 2: Simulation of the roofing in SOFISTIK®
Source: http://www.struction.com/

Figure 3: Testing of the 11 x 12 m span structure in Stuttgart
Source: http://www.struction.com/
Material Equilibria Installation was developed by the Institut for Computational Design (ICD) of Stuttgart, as a hybrid structure installation exploring the interaction between active-bending rods and tensile membrane (Figure 4).

![Material Equilibria Installation - ICD Stuttgart](http://icd.uni-stuttgart.de/?p=7636)

**Source:** http://icd.uni-stuttgart.de/?p=7636

1.1.2. Materials

**Beam**

Various materials can be used for active-bending, from timber laths (Multihalle Mannheim, 1975, Germany) to paper tubes (Shigeru Ban’s Japan Pavilion at the Expo 2000, Hannover, Germany) [1]. But a particularly adapted one, regarding its mechanical properties, is the Glass Fibres Reinforced Polymer (GFRP).

As seen in the former paragraph, actively-bent beams must present a high limit strain and at the same time being strong enough to remain stable.

GFRP is a composite polymer, made of an epoxy resin, mixed with generally around 60% glass fibres (Figure 5). This material therefore combines the high flexural strength of glass fibres, with the low elastic modulus of the polymer. The flexural strength varies between 250 and 450 MPa, and its modulus of elasticity between 12 to 41 GPa, depending on the amount of reinforcing fibres [3]. Those properties make GFRP a particularly well-suited material for active-bending.
Moreover, due to its industrial production process, “pultrusion” (Figure 6), various diameters, lengths and mechanical properties can easily be produced in an economical and effective way, with constant properties. This parameter flexibility allows a precise adjustment of the rods mechanical properties.

Figure 5: GFRP profiles, one yielded with apparent glass fibres
Source: Fabienne Garti

Figure 6: Typical pultrusion line
Source: http://www.fibrotec.es/
Membrane fabric

The most commonly used structural skins are PVC-coated polyester fabrics (Figure 7). Fibres are woven in orthogonal directions (warp and weft) and coated. Depending on the coating thickness, shear stiffness can be controlled, while weaving type and density change strength properties of the fabric.

It leads to five common types differing in terms of stiffness and strength which properties are described in the Table 1.

![PVC coated polyester fabric](Source: www.sattler-global.com)

<table>
<thead>
<tr>
<th>Table 1: Five membrane types</th>
<th>Type I</th>
<th>Type II</th>
<th>Type III</th>
<th>Type IV</th>
<th>Type V</th>
</tr>
</thead>
<tbody>
<tr>
<td>Threads in warp and weft</td>
<td>9/9</td>
<td>12/12</td>
<td>10,5/10,5</td>
<td>14/14</td>
<td>14/14</td>
</tr>
<tr>
<td>Yarn density (dtex)</td>
<td>1100</td>
<td>1100</td>
<td>1670</td>
<td>1670</td>
<td>2200</td>
</tr>
<tr>
<td>Total weight (gr/m²)</td>
<td>800</td>
<td>900</td>
<td>1300</td>
<td>1300</td>
<td>1450</td>
</tr>
<tr>
<td>Tear strength (N/5 cm/warp/weft)</td>
<td>3000/3000</td>
<td>4400/3950</td>
<td>7450/6400</td>
<td>7450/6400</td>
<td>9800/8300</td>
</tr>
<tr>
<td>Tear growth resistance (N, warp and weft)</td>
<td>310/350</td>
<td>520/580</td>
<td>1100/1400</td>
<td>1100/1400</td>
<td>1600/1800</td>
</tr>
<tr>
<td>Low flammability</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
<td>yes</td>
</tr>
</tbody>
</table>

1.1.3. Problematic

After previously exposed mechanical considerations, the main problematic of active-bending hybrid structures can be set.

Due to the very low cross-sections of actively bent elements, the structure is extremely sensitive to buckling. However, increasing the diameter of rods to get more stiffness would risk reaching their yield point. Therefore, the challenge consists in getting a stable structure, with a rod in a very unstable state. A restraining membrane is thus added to the system, as a shape stabilising element.

The problematic is consequently to know which parameters of the membrane have to be set, in order to optimise the shape stabilising effect of the membrane, and to provide the structure with a higher load-bearing capacity.

The scope of this master-thesis is to analyse the influence of the membrane stiffness, fibres orientation and pre-stress on the load-bearing capacity of a simple planar hybrid structure.
1.2. **Application on a hybrid tower prototype with CITA institute in Copenhagen**

The construction of an 8 m high hybrid tower prototype was realised during a collaboration workshop in Copenhagen, with the Centre for Information Technology and Architecture of the Danish Royal Academy of Arts (CITA), and the Chair of Structural Design and Technology of the University of Arts of Berlin (KET).

The basic element of the tower is a single spatial actively-bent GFRP beam (Figure 8). Their assembling shape the structure of the tower. An added restraining membrane is the last element composing the hybrid tower.

![Figure 8: Basic element of the tower](image)

The active-bending hybrid system tested in this master thesis is a simplified similar component, the aim being to test it behavior under different membrane parameters, in order to potentially optimise the tower.

The aim of the workshop was a computational and physical research of different active-bending hybrid structures design proposals, for an architectonical purpose.

A mass-spring based tool implemented in a design pipeline allowing easy handling to carry the experimental study. This was developed thanks to the algorithmic modelling plug-ins for Rhinoceros 3D – Grasshopper with Kangaroo – were first used for a parametric study. Variations of defined parameters were tested on Grasshopper, from a basic model. Various beam lengths, number of feet, and number of floors were implemented by different working groups of people (Figure 9).
Four design proposals chosen for their variety were then modelled in a 1:4 scale, with 3 mm diameter GFRP beams (Figure 10). Assembling challenge for GFRP active-bending structures was experienced. During the assembly process it was notably observed that irregular grid-geometries lead to tower prototypes with higher stiffness. However, bending forces are increased, and connecting the bent beam together began very difficult.

To validate the accuracy of the digital form finding process, 3D-scanning of those 1:4 models was performed, to compare geometries of virtual models defined in Grasshopper, to physical models. The scope of this comparison was to test the exactitude of the form-finding computational tool. Differences
visible on Figure 11 could be explained by the definition of rods self-weight and stiffness, which may need to be adjusted. Indeed, only a relative virtual stiffness can be input in the form-finding method (mass-spring) used in the design process, whereas Finite Element Analysis (FEA) uses real material properties (see paragraph 2.3.2. for more precisions).

![Figure 11: 3D scan of tower number 4 superposed to virtual model](image)

The second step of the workshop was the computational generation and physical fabrication of the membrane patches. The 3D shape of the membrane has to be unrolled in 2D patches, to allow their cutting in the planar fabric. A Grasshopper algorithm was used to define patches from towers 3D models (result in Figure 12).

![Figure 12: Membrane 2D patches](image)

Matching between digitally generated patches and physical models was again assessed (Figure 13). Perfect fitting was not verified, the algorithm had thus to be reviewed.
Last phase of the workshop was the construction of a 1:1 model of 8 m height, with GFRP beams diameters from 6 to 12 mm. Research on fabrication of the fabric pockets where beams should be slipped was also carried. A first 1:1 patch cutting and sewing was started but not finished before the end of the week.
Because those models were only geometrically and not structurally scaled, they are not precise enough to draw definite structural conclusions. Apart from extensive testing of the digital modelling pipeline and the evaluation of the precision, assembly strategies were tested to erect the tower prototype without scaffolding. The different physical prototypes imparted an impression of the softness of actively bent structures, which was prior to the workshop only experienced in the digital world.

The hybrid tower prototype was further developed after this workshop and is now exposed at the Danish Design Museum (Figure 15).

**Figure 15: Hybrid tower prototype exposed at the Danish Design Museum**

1.3. Previous research on the effect of a restraining membrane

Elisa Lafuente Hernández has worked for her thesis at KET [3] on an GFRP elastic gridshell (Figure 16).

**Figure 16: Gridshell studied by Elisa Lafuente Hernández**

In order to study the effect of a restraining membrane on the load-bearing capacity of this structure, she has isolated and simplified a basic element of the whole structure.
She has thus tested the restraining effect of a membrane on a simple planar grid, made of GFRP profiles, under a horizontal loading (Figure 17).

**Figure 17: Tested planar GFRP grid [1]**

**System definition**
A polyester cloth with PVC/PVDF coating of type III has been used for the membrane. Profiles are made of glass fibre reinforced plastics, with a modulus of elasticity of 25000 MPa and have a tubular section, with a diameter of 20 mm and a thickness of 3 mm. They are connecting with allowed scissoring.

**Test variables and assessment criteria**
The system was loaded from 0.0 to 1.3 kN.

Test variables are membrane fibres orientation (diagonal and longitudinal warp and weft fibres), membrane stiffness (isotropic E=500, 1000, 1500 and 2000 MPa), and membrane pre-stress (isotropic pre-stress of 0.1, 1.0, 1.5 and 2.0 kN/m). The assessment criteria is the point A nodal displacement.

**Results**
It has been demonstrated that the diagonally oriented membrane provides much higher shear stiffness to the grid than the longitudinally oriented.

Membranes with higher stiffness offer a stronger reduction of the grid’s deformations.

With a higher pre-stress level, the membrane carries a certain level of forces in the diagonal receiving compression, thus deformations are more strongly reduced.

**Connection with present master thesis**
This study has tested the influence of a restraining membrane on a grid GFRP structure.

This master-thesis tests the influence of the same membrane parameters, but on an active-bending structure. The scope is to study the interaction between the active-bending member and the membrane.
2. Methodology

In order to study the influence of membrane parameters in an active-bending hybrid structure, stresses and forces are measured on a loaded simple hybrid structure (described in paragraph 3.1.). To be able to establish a force equilibrium between the bent beam interacting with the membrane, the finite element method (FEM) is used.

2.1. Numerical analysis tool

Form-finding of membrane structures can be realised with computational methods which do not take into account material stiffness properties, as mass-spring models. In active-bending hybrid structures, contrary to the membrane, the bent beam owns a bending stiffness.

Therefore, the analysis method must take into account the material stiffness, to deal with the interaction between the bent beam stresses and the membrane forces. The Finite Element Analysis (FEA) is based on the stiffness matrix method. Resultants and deformations are calculated on the real structural system, with real stiffness properties, under loads or deformations.

In the case of active-bending hybrid structures, large deformations and stresses have to be evaluated, in particular due to the bending process.

Hence, approaches using iterative load steps and third order theory are combined with FEA, to quantitatively analyse the structure. This study was carried thanks to the finite element software SOFiSTIK®.

2.2. Active-bending hybrid structures simulation principles

Active-bending hybrid structures FEM simulation needs particular steps to be performed. Those are explained here, the detailed simulation being described in paragraph 3.2.

Active-bending hybrid structures construction process

Active-bending hybrid structures are composed of two principal elements having very different behaviours: a bent beam and a membrane. The construction process of active-bending hybrid structures induces stresses in the elements. In order to get accurate values to analyse the system, the simulation of the whole construction process has to be performed. This process is described in Figure 18.
Figure 18: Construction stages for an active-bending hybrid structure

**Beam bending**
During the bending process, high stresses are induced in the beam, due to the large deformations imposed to it. To perform this simulation in a FE environment, the “elastic cable approach” developed by Julian Lienhard [4], was used. It consists in the use of a cable to pull the end of the beam to its end position. The stiffness of the cable is drastically reduced and incrementally pre-stressed in order to shorten it in length. This process allows a progressive bending of the beam.

**Membrane pre-stressing**
After having linked the membrane to the beam, pre-stressing of the fabric has to be performed. The membrane stiffness is also drastically reduced for this step. A homogeneous pre-stress in X and Y directions is then applied on the surface. After removing all additional supports, the beam and the membrane with full stiffness can find their equilibrium.

### 2.3. SOFiSTiK® working

#### 2.3.1. Software organisation
Sofistik® is a German Finite Element Analysis (FEA) and CAD software, used for a wide range of applications, from bridges and geotechnics to lightweight structures. It is mainly popular in German-speaking countries.

The software is composed of various programs, used in the different steps of the modelling. Modules specifically used for this project are presented in the following table. All those functions are gathered in the SSD interface: Sofistik Structural Desktop. The software organisation is exposed in Table 2.

Teddy is the first one of those. It is one of the pre-processing modules, used to input the structural system, loads, etc. This text editor allows defining geometries by using Sofistik script language CADINP. It is based on a classical programming vocabulary, where for example loops can be implemented.
The graphical and text editor input tools of the package SSD also allow a wide variety to create the model. Compatibility with 3D-CAD software as AutoCAD or Rhinoceros 3D, and with programming plug-in as Grasshopper, through specific interfaces, support the development of numerous tools.

Table 2: SOFiSTiK software organisation

<table>
<thead>
<tr>
<th>Preprocessing</th>
<th>Processing</th>
<th>Postprocessing</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEDDY</td>
<td>AQUA</td>
<td>Design:</td>
</tr>
<tr>
<td>Parametric input</td>
<td>Material and cross sections</td>
<td>AQB Beam elements</td>
</tr>
<tr>
<td></td>
<td></td>
<td>AQBS pre- and post-tensioned beams and composite sections</td>
</tr>
<tr>
<td></td>
<td></td>
<td>BEMESS slab, wall and shell elements</td>
</tr>
<tr>
<td></td>
<td></td>
<td>BDK Lateral torsional buckling</td>
</tr>
<tr>
<td>RHINO INTERFACE</td>
<td>SOFILOAD</td>
<td>Plotting / Visualisation</td>
</tr>
<tr>
<td>Graphical pre-processing</td>
<td>Load generator</td>
<td>ANIMATOR Interactive system viewer</td>
</tr>
<tr>
<td></td>
<td></td>
<td>WINGRAF/GRAFIX Interactive result plotting</td>
</tr>
<tr>
<td></td>
<td></td>
<td>URSULA viewer for print-result</td>
</tr>
<tr>
<td>SOFIMSHA</td>
<td>ASE</td>
<td>structural FE meshing, import-export and modification of FE</td>
</tr>
<tr>
<td>structural FE meshing, import-export and modification of FE</td>
<td>3D FE solver (linear, nonlinear, dynamics)</td>
<td></td>
</tr>
<tr>
<td>SOFIMSHC</td>
<td></td>
<td>automatic FE meshing</td>
</tr>
</tbody>
</table>

2.3.2. SOFiSTiK® parameters: theoretical background

Type of analysis: third order theory (TH3)

A system of equations is linear when it can be transformed in a linear combination of variables. Translated in the vocabulary of static equilibrium, it means that if the external loads applied on a linear system are multiplied by a factor n, resultant displacements and stresses should also be multiplied by the same factor. If not, the studied system is non-linear [4].

If stresses calculation in a column submitted to an axial compression force is carried without taking into account the insignificant deformations, it is a linear first order calculation.

Because active-bending structures are submitted to large deformations, rotations and stresses, geometrical non-linear behaviour has to be taken into account. This non-linear behaviour is based on second and third order theories.
If deformations of the beam are still little, but non insignificant, bending of the end of the column leads to an eccentricity of the applied force. If the stresses calculation is carried again on the deformed structure, it will lead to a new deformation. When those successive deformations diverge, the buckling phenomenon occurs: an amplification effect is implemented (Figure 19). It is called the **P-δ effect**: large deformations and rotations appear and have to be taken into account to correctly analyse the structure. It is the **second order** theory.

The third order theory includes the P-delta effect, and also takes into account the geometrical system modification, like snap-through, length modification for big deformations and behaviour after buckling. A calculation software is needed to apply the **third order** theory.

![Figure 19: P-delta effect for a clamped column](image)

In SOFiSTiK®, the option TH3 have to be implemented in the calculation system definition, to analyse it according to third order theory.

Mathematically, the geometric non-linearity has an effect on the stiffness matrix definition. Indeed, the deformed system is considered to calculate the static equilibrium. Therefore, the stiffness matrix is not constant anymore.

“In geometric nonlinear analysis, the stiffness matrix is generally referred to as tangent stiffness matrix \( K_T \) which can be split into the elastic stiffness matrix \( K_e \), the initial displacement stiffness matrix \( K_u \) and the geometric stiffness matrix \( K_g \), which is computed based on the stress state of the previous equilibrium iteration” [4] (Equation 1).

**Equation 1: equilibrium equations or a geometrically non-linear system [4]**

\[
(K_e + K_u + K_g) \cdot u = F
\]

This update of the geometric stiffness matrix \( K_g \) based on the previous equilibrium stress state, explains the need for an incremental bending of the beam, thus the use of the elastic cable approach.
**Non-linear iteration method: residual forces**

Following explanations are issued from SOFiSTiK® documentation [5].

![Image of residual forces during iterations](image)

**Figure 20: Visualisation of residual forces during iterations**

Non-linear calculation is based on an iterative method. After every iteration step, displacements and thus stresses are calculated for the new equilibrium step. If non-linear effects as plasticising, cracks, or any other occur, equivalent nodal loads are generated. Because those generated additional nodal loads are not anymore in equilibrium with the system, the remaining **residual forces** are applied as additional loading during the next iteration step. A new calculation of deformations and stresses is calculated with this next iteration step, generally closer to the final equilibrium result. The calculation is successful when the residual forces drop under a certain value. Residual forces can be visualised during iterations (Figure 20).

**Tolerance limit of the iteration (TOL)**

A tolerance limit for the residual forces can be defined with the record SYST. For a positive value (<1), the highest nodal load is multiplied by the value of TOL. For example, for TOL=0.001, if a maximum nodal load of 200 kN is calculated, the tolerance limit for the residual forces is = 200 * 0.001 = 0.2 kN. It means that if that as a result of the current iteration, the maximum residual force is smaller than 0.2 kN, the iterative process is interrupted.

Therefore, the smaller the tolerance, the better approximation of the result. A tolerance factor value of 0.06 was used for simulations. When no convergence could be reached, a tolerance factor value of 0.1 was used.

A number of iteration ITER can also be defined, to define the number of iteration. If no convergence is reached, a higher number of iteration can be input.
Export criteria: meshing properties

Meshing properties of structural elements can be defined during exportation of the system through the Rhinoceros 3D® SOFiSTiK® interface.

Finite element type can be chosen, between quadrangle or triangle elements. Calculations are less precise, and thus much faster, with triangle elements. In order to reduce instabilities and calculation time, the whole study was performed with triangle elements. Those finite elements will be called QUAD elements in the rest of the paper. A maximum allowed length of an element edge of 0.07 m, and without any edge refinement also allows a coherent and fast simulation.
3. Case study

The studied planar hybrid structure was consciously chosen for its simplicity and the subsequent ability to focus on the test parameters, which will be described in paragraph 3.3. In the following paragraphs the different phases of the simulation are described. After explaining the principle the physical equivalent and the code for the calculation will be expounded.

3.1. Settings

Initial settings of the case study are described in this paragraph.

3.1.1. Geometry

In order to analyse the interaction between a restraining membrane and an actively bent beam, a planar hybrid structure was chosen (Figure 21).

The system is composed of a 10 mm thick 2 m long beam (C-D on the illustration), bent up to a defined position. A 1.0 mm thick membrane is linked to the bent beam. The membrane is also fixed with a hinged support (B). Its edges are maintained with two 3 mm diameter edge cables.

The beam end point (A) final position is deliberately chosen. Yet, as seen in paragraph 1.1.3., one problematic of active-bending hybrid structures is to solve the extreme buckling sensibility. The aim of the membrane is precisely to stabilise the beam. Because of the bent beam shape, a horizontal punctual load can be applied in point A, to activate the structure as wanted. Indeed, this load tends to...
further bend the beam, and the membrane is mobilised to stabilise the structure by reducing its displacements.

Curvature of membrane edges represents the equilibrium-shape of the membrane and the edge cable forces. Edge cables are initially input with a defined curvature radius (see paragraph 3.2.5. about pre-stress modelling). Their radius were arbitrary implemented with the value of 1.34 m, which draw a coherent curvature regarding the global structure dimensions.

Cross-sections diameters of the beam and of the edge cables were chosen to simulate a realistic system. On the one hand, the beam diameter of 10 mm is low enough to keep the residual stresses due to the bending process at an acceptable level, and high enough to be stable. For the edge cables, a diameter of 3 mm avoid a too high influence of those edge cables on the load-bearing capacity of the structure.

3.1.2. Materials

In order to carry a representative case study, commonly used materials for active-bending structures and membrane structures have been chosen. The bent beam is made of glass fibres reinforced polymer (GFRP). The membrane fabric is a polyester cloth, with a PVC coating (Type III) (see paragraph 1.1.2. for more information about those materials).

Materials definition is performed in SSD thanks to an included interface, where various material types can be selected. For each material type, default mechanical properties can be individually modified. The interface replaces manual input in module AQUA to define materials and cross-sections of structural elements, and module SOFIMSHA which defines QUAD elements of the membrane.

Following Table 3 and Table 4 detail GFRP and membrane material input in SSD.
Table 3: GFRP specifications

<table>
<thead>
<tr>
<th>Item</th>
<th>Description / GFRP</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Category: General materials</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Type: Elastic Material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>EP</td>
<td>Elastic modulus parallel to fibre</td>
<td>25000</td>
<td>N/mm²</td>
</tr>
<tr>
<td>G</td>
<td>Shear modulus</td>
<td>3000</td>
<td>N/mm²</td>
</tr>
<tr>
<td>E90</td>
<td>Elastic modulus normal to fibre</td>
<td>9000</td>
<td>N/mm²</td>
</tr>
<tr>
<td>GAM</td>
<td>Unit weight (density)</td>
<td>19.0</td>
<td>kN/m³</td>
</tr>
<tr>
<td>MUE</td>
<td>Poisson's ratio yz</td>
<td>0.23</td>
<td>-</td>
</tr>
<tr>
<td>FT0</td>
<td>Tensile strength parallel to the fibre</td>
<td>250</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FT90</td>
<td>Tensile strength normal to the fibre</td>
<td>50</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FC0</td>
<td>Compressive strength parallel to the fibres</td>
<td>250</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FC90</td>
<td>Compressive strength normal to the fibres</td>
<td>90</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FV</td>
<td>Shear strength at centre (shear force)</td>
<td>25</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FVR</td>
<td>Shear strength at the edge (torsion)</td>
<td>25</td>
<td>N/mm²</td>
</tr>
<tr>
<td>FVB</td>
<td>Shear strength for plate bending</td>
<td>25</td>
<td>N/mm²</td>
</tr>
</tbody>
</table>

Table 4: PVC anisotropic membrane specifications

<table>
<thead>
<tr>
<th>Item</th>
<th>Description / Polyester cloth, PVC coated fabric</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Category: Structural materials</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Type: Polyester Fabric</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Thickness</td>
<td>A value of 1 mm is input to define the geometry</td>
<td>0.8 ± 1</td>
<td>mm</td>
</tr>
<tr>
<td>GAM</td>
<td>Unit weight (density)</td>
<td>12</td>
<td>kN/m³</td>
</tr>
<tr>
<td>EP</td>
<td>Elastic modulus parallel to fibre</td>
<td>1000</td>
<td>N/mm²</td>
</tr>
<tr>
<td>E90</td>
<td>Elastic modulus perpendicular to fibre</td>
<td>500</td>
<td>N/mm²</td>
</tr>
<tr>
<td>MUE</td>
<td>Poisson’s ratio</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>G</td>
<td>Shear modulus</td>
<td>30</td>
<td>N/mm²</td>
</tr>
</tbody>
</table>
3.2. Modelling

3.2.1. Modelling steps

Form-finding of the actively-bent beam, membrane linking, pre-stressing, and loading of the hybrid structure are divided in several phases. Those are performed in the FE software SOFISTiK® through its specific programming modules such as ASE and SOFIMSHA. Those modules and their roles have been described in paragraph 2.3.1.. A flow chart specifies the different steps of the modelling (Figure 21). Each stage will be explained in detail in the following paragraphs.

All illustrations included in this paragraph concern a system with diagonally oriented fibres, a E-modulus of 1000 MPa for an isotropic membrane, under pre-stress of 0.01 kN/m.
Figure 22: Flow chart representation of the simulation approach using SOFiSTiK
3.2.2. Geometry input

The initial geometry is input from the 3D CAD software Rhinoceros 3D® (see Figure 23). Curves, points and surfaces are drawn. SOFiSTiK® plug-in for Rhinoceros 3D® allows to add structural properties to the geometry and to export these structural points, beams and quad-elements into the SSD database. Elements cross-sections are selected, from those defined in SSD. They attribute a material and a diameter/thickness to each structural element. Meshing type and density, support conditions and orientation of elements are the other used export parameters.

Implementation of the system geometry necessitates two stages. First is the single beam bending. Its bent geometry is then imported back in Rhinoceros 3D®, to define the membrane shape, perfectly fitting with the bent beam curvature.

**Beam geometry input**

The initially straight beam of two meters long is drawn, with fifty support nodes. This line and the points are set through SOFiSTiK® Rhino plug-in as structural elements. “Centric beam” type is selected for the beam.

Support nodes are correlated with the beam subdivision into fifty 4 cm long elements. Subdivision number is chosen on the one hand not too high in order to perform a rapid simulation, on the other hand significant, to ensure a dense enough meshing of the beam.

Displacements in Y direction and out-of-plane rotations around X and Z axis are restricted for the fifty support nodes, in order to hold the system in plane (O,X,Z).
Membrane and edge cables geometry input

Once the beam is form-found (process detailed in next chapter), the membrane can be drawn with the correct geometry. Edge (A-C) follows the bent beam curvature. The two other edges are implemented with a curvature radius, representing the shape of the membrane after pre-stressing (refer to chapter 3.2.5. for details about pre-stressing). Two steel cables of 3 mm diameter are implemented at those edges. A hinged support hold the membrane at its corner (B). Cables are defined as “cable elements”, and the surface as a “membrane” in Rhinoceros 3D®.

Fifty nodes corresponding to the fifty nodes of the beam are set on the edge of the membrane to be linked with the beam. Those points will be used for the linking process (see paragraph 3.2.4. about the linking process).

Export criteria

Meshing of each structural curve and surface can be defined individually in the SOFiSTiK® interface for Rhinoceros 3D®. The beam needs a fifty elements division, to fit with the implemented support nodes. The cable element needs to stay as a single element, in order to be pre-stressed and shortened. All the other elements are individually defined with an automatic meshing.

The global system is then exported as a regular triangular mesh, with a density of 0.07 m and without any refinement (see paragraph 2.3.2. for explanation on density). The complete exported structure is shown on the following illustration (Figure 24).

Figure 24: Initial exported geometry
Two different QUAD elements orientation are chosen in the Rhino interface, to simulate orientation of membrane fibres (Figure 25).

![Longitudinally oriented QUAD elements](image1)

![Diagonally oriented QUAD elements](image2)

**Figure 25**: Orientation of QUAD elements

### 3.2.3. Beam form-finding

**Physical realisation**

As experienced during the workshop in Copenhagen (chapter 1.2.), slender GFRP beams can be bent manually. Once each support point fixed, the force used to bend it can be released, to let stresses in the beam relax, and let the beam find a new equilibrium state (form-finding).

**Implementation in SOFiSTIK®**

Significant residual stresses are induced in the beam during the bending process. Because of the large deformations imposed to the beam, convergence for calculation of the restrained stresses is not possible. The idea is thus to divide this deformation into small steps.

A progressive loading have to be performed in SOFiSTIK®. The “elastic cable approach” developed by Julien Lienhard [9] was used: a single-element cable is implemented with very low stiffness properties, such that under increasing load increments of pre-stress it shortens in length. Convergence to null for this virtual highly elastic cable is not possible. Therefore, the calculation process is aborted as soon as the end of the bent beam has find its final position. For this case study, it occurs after 49 iterations (see Figure 26 below).
**Programming**

The code is visible on the next screen shot (Figure 28). The SOFISTiK® module TEMPLATE is called to implement some variables which are used in the next module. The module ASE incrementally apply the pre-tension in the highly elastically virtual cable.

A loop was implemented, which increases pre-tension of the virtual cable, linked to the beam and fixed at the other end. This process results in the shortening in length of the virtual cable, thus to the bending o the beam linked to it. In order to be able to shorten the cable as close as possible of null, its stiffness has been reduced to a very low level, through the division by a factor of 1000.

Each group of elements (membrane, beams, and cables) is referred to with a group number, so material and force properties can be manipulated for each group during the simulation process. Group 30 **GRP 30** refers to the virtual cable, and the stiffness factor **FACS** has a value of 0,001. Pre-tension is applied on the highly elastic virtual cable through the command **PREX**. Its value is 0.06 kN, and multiplied by the loop variable **#n**. This variable is the one of the **LOOP** implemented to progressively increase the amount of pre-tension. The key point of this loop is the reuse of last load case **LC** as the primary load case **PLC** for the new iteration. The idea of a primary load case is to add up the total amount of load and following deformation created by the incremental process. Therefore, deformation of the beam is performed through 49 iterations. Instabilities appear after the 49th iteration, as visible in the following illustration, where iterations were run up to load case no.74 (Figure 27).

---

**Figure 26**: Beam bending iterative process through shortening of a virtual cable

**Figure 27**: Beam instabilities with iterations ran up to load case no.74
GRP 40, GRP 60, 70 refer to the implemented membrane and edge cable geometry, which are deactivated before the linking step. The beam is then form-found without presence of any other perturbing structural element.

```
+PROG TEMPLATE urs:14.1
HEAD 'Global'
sto$1 1
sto$lc 0
sto$fx 800
END

+PROG ASE urs:14.2
HEAD 'Solve'

let$fiter 0
let$fplc 0
let$f$n 1

loop$1 #x
let$f$fiter $fiter+1
let$f$lc $f$lc+1

CTRL cabl 0
CTRL BEAM 2
CTRL WARN 197 $ turns off warning for large rotations & displacements
syst prob th3 iter 200 tol .001 fmax 0.2 plc #plc
GRP all
  GRP 30 facs 0.001 prex 0.06*#n $ Cable
  GRP 40 off $ Membrane and membrane support cables deactivated
  GRP 60,70 OFF

lc $lc dlz 0 titl 'iter $fiter'
end
let$f$plc $lc
let$f$n $n*1.1 $ Speed up iterations by increasing prestress
endloop
END
```

Figure 28: Programming for the beam-bending step
3.2.4. Stresses output and geometry update

Those two stages are important steps of the simulation, but not representing any physical one. These are performed after each geometry change, as indicated in the flow-chart (Figure 22).

*Implementation in SOFiSTIK®*

Stresses output are performed by a dedicated module, which converts calculated forces in the beam into stresses. A basic visualisation of stresses in the beam is available in the graphical interface (Figure 29). A more reliable output is given in the interactive result plotting and visualisation module WINGRAF.

![Beam stresses graphical evaluation](image)

*Figure 29: Beam stresses graphical evaluation*

Geometry update consists in adapting local coordinates to the new deformed geometry (Figure 30 and Figure 31). Saving of this new geometry is then performed, to set it as the new system on which is based the next modelling stage.
Local coordinates systems at iteration 1  
Detail of the beam local coordinate system (iter. 1)  
Not updated local coordinates systems at iteration 49

Figure 30: Not updated coordinate systems after form-finding

Local coordinate system after updating, iter. 49  
Detail of the updated coordinate system

Figure 31: Updated local coordinate system after form-finding

**Programming**

The code for stress analysis is shown on Figure 32. The SOFiSTiK® module AQB is dedicated to design of beam elements. Calculated forces are converted into a stress state. `PAGE UNII 0` sets the system of units for input, while `STRE` select a stresses output. `LC 49` defines the load case to be calculated, here the last valid iteration of the beam form-finding stage.

Module ASE is used to update the local coordinates system to the new geometry, and to save this geometry as the new system, through the command `STOR YES`. 
Figure 32: Programming for the stresses output and geometry update
3.2.5. Membrane-beam links implementation

Physical realisation

“One commonly used flexible edge detail is the rope edge, where a rope running in a sleeve supports the edge forces” [6]. This method creates a pocket, a “hollow hem” (see Figure 33). In order to avoid movement between rope and membrane, an additional webbing can be mounted to absorb tangential forces (see Figure 34). The rope (here the GFRP beam) is pushed through this hem during erection of the structure, before fixing it with the webbing.

Figure 33: Hem sleeve with rope as edge element [8]

Figure 34: Rope edge with sewn-on webbing [8]
**Implementation in SOFiSTIK®**

Fifty structural nodes have been set on the beam before bending it, and fifty other corresponding nodes on the membrane edge (see geometry input, paragraph 4.2.2).

In order to model the continuous and fixed linking between the bent beam and the membrane, those nodes are fixed, one to each other, thanks to a defined SOFiSTiK® command.

The choice of number of nodes was led by this linking process. Indeed, with a four cm gap between each node, and a mesh density of 0.07 m (see chapter 2.3.2. about SOFiSTiK® parameters), QUAD elements of the membrane are not smaller than the distance between two nodes. Thus, once every membrane node is linked to the beam nodes, a continuous and fixed linking is performed between the membrane edge and the bent beam. If QUAD elements edges are shorter than the distance between two nodes, a more precise calculation is carried, and the membrane gets curves between each node after pre-stressing.

An illustration of those links visible on the back of the membrane is given in the next picture (*Figure 35*). Membrane nodes are not represented because they don’t have any support condition. The links appear in yellow.

*Figure 35: Detail of links between beam and membrane*
Programming

The code is shown on the next screen shot (Error! Reference source not found.). Module SOFIMSHA is used each time geometry is changed.

Connection between beam and membrane nodes is realised thanks to the command “NODE NO _ _ _ _ FIX NR1 _ _ _ _”, with NO calling the number of the “slave” node, and NR1 the number of the “master” node. Indeed, the definition must be set with the “master” node being owned by the solid element (here, the beam), and the “slave” node by the softer element (here the membrane). If this condition is not respected, the link is not anymore stiff, and elongates under loading, as on the illustration on the right (Figure 36).

A loop is implemented to link each couple of nodes one after the other. In the same task, the end node of the beam is fixed (NODE NO 1001 FIX PP), while support of the virtual cable is deleted (NODE 1104 FIX FREE).

The commands SYST REST and CTRL REST 2 are compulsory to keep geometry and stresses of the previous load case (LC) previously saved, as basis for geometry changes. If not, the system is set back to its initial situation, stresses deleted, and the changes occur on the initial structure.

+PROG SOFIMSHA urs:22.1
HEAD Links
page unii 5
SYST REST
CTRL REST 2
NODE NO 1001 FIX PP
$ ^______________________________
LOOP#m 51
  NODE NO 1155-#m FIX KF NR1 1001+#m
ENDLOOP
node 1104 fix free $ Fixed point of the virtual cable
END

Figure 36: Bad link implementation consequence

Figure 37: Programming for the linking step
3.2.6. Membrane pre-stress

On most of membrane structures, pre-stressing of the membrane is an absolutely necessary step. This process increases the stiffness of the membrane, in order to increase its stiffness.

Physical realisation

Various tensioning devices can be used to apply a tension force in the membrane by pulling it, or to pre-tension its edge cables by shortening them. Usually electrical, hydraulic or mechanical devices are used.

Implementation in SOFiSTIK®

Membrane geometry is implemented with its approximate final shape. A curve radius is chosen for membrane edges and boundary cables.

Membrane pre-stress normally leading to this curvature radius is calculated with the following relation:

\[ S_s = S_m \cdot R \]

with \( S_s \) being the edge cable pre-stress, \( S_m \) the membrane pre-stress, and \( R \) the radius defining curvature of the membrane at its edges (Figure 38).

To be able to pre-stress the membrane without occasioning wrinkles due to compression, its stiffness is drastically reduced. A defined pre-stress is then applied in both direction X and Y.

With the defined membrane pre-stress and edge radius, related cable forces can be calculated. Because they already have the correct length, the pre-stress simulating this force is applied on the edge cables with full stiffness.

Because the GFRP rod has a very low diameter, it cannot resist to the membrane pre-stressing without starting to bend further. This further bending unfortunately leads to crushing of edges QUAD elements (Figure 39, support nodes representation has been deactivated to lighten images). SOFiSTIK® does not accept those QUAD elements with too sharp angles, which bring to instabilities during calculation (Figure 41).
Therefore, a support point is implemented in A, to hold the beam end without any displacement during membrane pre-stress.

Another consequence of the pre-stress force applied on the beam, is that with a very high value, additional curvatures start to appear close to beam supports in A and C (Figure 40).

For this reason, the magnitude of the pre-stress force is limited to 0.07 kN/m as a maximum.
Programming

The code is visible on the next screen shot (Figure 42). Module ASE is called to apply the pre-stressing load. Virtual cable group GRP 30 remains deactivated.

GRP 40 refers to membrane QUAD elements. Its stiffness factor FACS has a value of 1.0, and pre-stresses in both directions PREX and PREY are set to the defined membrane pre-stress. Those values are in kN/m.

GRP 60, 70 refers to the two groups where the edges cables were defined. Their stiffness factor is kept to 1.0, while the calculated pre-stress PREX is applied. Pre-stress values for cable elements are in kN.

LC 100 is the name of the new load case just defined. The pre-stressed system is saved under number 100. The self-weight DLZ is set to a very low number because it cannot be null.

+PROG ASE urs:13
HEAD Membrane Prestress
syst prob th3 iter 300 PLC 49 fmax 0.7
CTRL CABL 0

GRP 30 OFF $ Deactivate virtual cables
GRP 40 FACS 1.E-10 PREX 0.01 PREY 0.01
GRP 60,70 FACS 1 PREX 0.0134

LC 100 DLZ 0.00001 titl 'Prestress'
END

Figure 42: Programming for the pre-stressing step

3.2.7. Relaxation

This last construction step consists in releasing supports avoiding interaction between beam and membrane, to let them find a new equilibrium.

Physical realisation

Relaxation means here the release of support point A, which role is to maintain the beam during pre-stressing to avoid too big deformations.

This step is generally not needed in a real structure, since the beams are strong enough, and/or the geometrical configuration does not lead to this beam instability.

Implementation in SOFISTIK®

Point A support conditions are deleted. Another important step realised during this task is the membrane return to full stiffness properties, after its artificial reducing (Figure 43).
Membrane state after pre-stress | Membrane state after relaxing (node A support conditions changed, and full stiffness back)

Figure 43: Relaxing step effect

Programming

The code is visible on the next screen shot (Figure 44). Module SOFIMSHA is used to modify the geometry by putting back point A (node no. 1001) support conditions to out-of-plane restrictions. All support conditions are first suppressed before implementing again the original ones.

Module ASE calculates the new equilibrium of the structure, with all stiffness back to full values. The command `NMAT YES` assures failure of QUAD elements if they are submitted to compression.
3.2.8. Loading

In this last step, an axial punctual load will be first defined as a load case, then applied on the realised structure (Figure 45).

Figure 44: Programming for the relaxing step

Figure 45: Position of punctual axial load
**Physical realisation**

The load simulated in this case study is a virtual one, which aim is to demonstrate the stabilising role of the restraining membrane. As it only works for the simulated load direction it would have to be combined with additional elements to build a structural system capable to support real environmental loads. In reality, active-bending hybrid structures are submitted to same loads as other more usual structures. A wind load could for example horizontally stimulate such a structure.

**Implementation in SOFiSTIK®**

The nodal axial load has first to be defined, with an application point, a direction and a magnitude. The defined load case can then be applied on the structure (Figure 46).

![Membrane state after relaxing](image1)

<table>
<thead>
<tr>
<th>Membrane state after relaxing</th>
<th>Membrane state after loading (1.0 kN)</th>
</tr>
</thead>
</table>

*Figure 46: Loading step effect*

**Programming**

The code is visible on the next screen shot (Figure 47).

The load is defined in module SOFILOAD. *NODE* gives the application point, *TYPE* its direction (here global X direction with *PXX*), and *P1* its magnitude.
Three parameters of the membrane have been chosen to be the variables of the study: stiffness of the membrane (E-modulus), orientation of warp and weft fibres, isotropic or anisotropic membrane properties, and pre-stress.

Assessment criteria of the simulations are displacement of the node A (in the global XY plane), maximum displacement in the structure, maximal membrane force, bending moment $M_y$ of the beam, and beam maximal Von Mises stress.

3.4. Results

3.4.1. Output visualisation and analyse

The module WINGRAF is used to output in a graphical interface the simulation results. Various representations are available, from tables to vectors. Following illustrations give an overview of outputs used in WINGRAF for this study (Figures 47 to 50). Those sample results are given or an isotropic membrane, with diagonally oriented fibres and E-modulus of 1000 MPa, and the membrane was pre-stressed by 0.01 kN/m and loaded by 5 kN.
3.4.2. Influence of membrane pre-stress

**Tested variable**

Influence of membrane pre-stress is analysed in this paragraph. Systems with membrane pre-stress values of 0.07 kN/m, 0.04 kN/m and 0.01 kN/m were tested (see Figure 52). A system without any added pre-stress and another without any stabilising membrane were also tested.
System parameters
The system was tested for a longitudinal orientation of warp and weft fibres, and anisotropic membrane properties (refer to paragraph 3.1.2. for more details about defined membrane properties). Tolerance was set to 0.1 (read paragraph 2.3.2. for explanations about tolerance).

In order to test the bent beam load-bearing capacity without any membrane, the following system (Figure 53) was also submitted to axial load. All properties except pre-stress are the same as in the other systems. Suppression of the membrane and its horizontal edge cable is the only modification. The vertical edge cable is kept to avoid the beam to return to its initial straight position.

Results
Systems with no pre-stress and without membrane do not converge for a load higher than respectively 0.05 kN/m and 0.01 kN/m. Therefore, they are not represented in charts.

In our specific case, pre-stressing the membrane increases equivalent stresses in the bent beam, and its bending moment $M_y$, without effect of the load magnitude. This behaviour can be seen on the two following charts (Figure 54 and Figure 55). The constant difference between curves under increasing loading is explained by the fact that beam stress and bending moment are already generated by the...
bending and pre-stressing process, before any loading. Structures with distinct pre-stresses are thus already differentiated before loading.

Figure 54: Beam maximal Von Mises stress during increasing axial loading for different pre-stresses

Figure 55: Beam maximal bending moment My under increasing axial loading for different pre-stresses
Node A displacement and maximum membrane force are also higher for pre-stress values of 0.07 and 0.04 kN/m than for 0.01 kN/m (Figure 56 and Figure 57).

![Node A displacement](image1)

**Figure 56: Node A displacement under increasing axial loading for different pre-stresses**

![Max membrane force](image2)

**Figure 57: Maximal membrane force under increasing axial loading for different pre-stresses**
The following table compares results under the first load case, for all tested systems (Table 5).

Table 5: Results for first load case for different pre-stresses

<table>
<thead>
<tr>
<th>Pre-stress [kN/m]</th>
<th>Load [kN]</th>
<th>Node A displacement [mm]</th>
<th>Max. membrane force [kN/m]</th>
<th>Max. bending moment My [kN.m]</th>
<th>Max. Von Mises beam stress [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.07</td>
<td>0.05</td>
<td>1.54</td>
<td>0.529</td>
<td>0.0157</td>
<td>161.3</td>
</tr>
<tr>
<td>0.04</td>
<td>0.05</td>
<td>0.808</td>
<td>0.367</td>
<td>0.0138</td>
<td>141.4</td>
</tr>
<tr>
<td>0.01</td>
<td>0.05</td>
<td>0.776</td>
<td>0.241</td>
<td>0.0130</td>
<td>133.4</td>
</tr>
<tr>
<td>0</td>
<td>0.05</td>
<td>0.788</td>
<td>0.448</td>
<td>0.0128</td>
<td>131.8</td>
</tr>
<tr>
<td>No membrane</td>
<td>0.01</td>
<td>78.3</td>
<td>-</td>
<td>0.0113</td>
<td>114.8</td>
</tr>
</tbody>
</table>

The maximum bending moment My and the Von Mises stress in the beam are decreasing with a lower pre-stress, and the lowest values are for the system without membrane. The maximum membrane force is higher for the system without pre-stress than for the others. And the remarkable result is the huge displacement measured for the beam without stabilising membrane, an increase by 100%.

Next table (Table 6) shows how much the pre-stressing process generates stresses in the bent beam. As a logical effect, a higher pre-stress significantly increases stresses in the bent beam.

Table 6: Beam max. Von Mises stress due to the pre-stressing process

<table>
<thead>
<tr>
<th>Pre-stress [kN/m]</th>
<th>Max. Von Mises beam stress after bending, without pre-stress [MPa]</th>
<th>Max. Von Mises beam stress after pre-stressing, without loading [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.07</td>
<td>131.0</td>
<td>162.9</td>
</tr>
<tr>
<td>0.04</td>
<td>131.0</td>
<td>141.4</td>
</tr>
<tr>
<td>0.01</td>
<td>131.0</td>
<td>133.4</td>
</tr>
</tbody>
</table>

Conclusions

The first remarkable conclusion is the justification of the membrane as a stabilising element. The bent beam without membrane shows a node A displacement a hundred percent higher than for all other structures with a membrane.

Secondly, it has been shown that more pre-stress leads to higher stresses and deflections in the system. Pre-stress has normally a positive effect on the load-bearing capacity of a structure. Indeed, by postponing the apparition of wrinkles due to compression forces in the membrane, it allows the membrane to play its stabilising role further. The negative collateral effect of pre-stress is the increase of stresses and bending moment My in the beam, because of the tension force it applies on it. In the
case of this structure, because the beam is not held at its end, the negative pre-stress effect is more important than its positive effect.

3.4.3. Influence of membrane stiffness

Tested variable
Influence of membrane stiffness is evaluated in this paragraph. Systems with anisotropic membranes were tested, with a stiffness of 1000 MPa, and 500 MPa (all other parameters remaining the same). A system with the previously defined (paragraph 3.1.2.) anisotropic membrane properties was also tested. Once with the stiffer fibres in X direction, and once with the stiffer fibres in Y direction (Figure 58).

![Figure 58: Orientation of anisotropic membrane](image)

System parameters
The system was tested for a longitudinal orientation of warp and weft fibres, and a pre-stress of 0.01 kN/m. Tolerance was set to 0.1.

Results
Regarding the beam maximum Von Mises stress and maximum bending moment My, not any remarkable difference is visible between the isotropic systems with a stiffness of 1000 Mpa or 500 MPa. Stress and moment is slightly lower with a higher stiffness. The anisotropic system with stiffness of 500 MPa in the Y direction behave as the isotropic system with E=500 MPa. The structure with a stiffness of 500 MPa in the Y direction, and stiffer in the X direction shows lower stresses and moments than the others (see Figure 59 and Figure 60).
Figure 59: Beam maximal Von Mises stress during increasing axial loading for different degrees of membrane stiffness.

Concerning displacement of node A and maximum membrane forces, not any particular behaviour can be extracted. Except the lower displacements and higher membrane forces for a stiffer membrane (see Figure 61 and Figure 62). That is explained by the fact that the stiffer membrane have a more efficient stabilising effect, but therefore significantly higher forces going through it.

Structure with the anisotropic membrane having stiffer fibres in X direction shows lower membrane forces, and slightly lower displacement than the others (excepting the isotropic 1000 MPa membrane).
Conclusions

As a logical conclusion, this test shows that a stiffer membrane reduce displacements magnitudes, but at the same time increase forces going through it.
Moreover, the beam stresses and moments are higher when the membrane applies more tension to further bend the beam, i.e. in Y direction. Therefore, a system with a stiffer membrane in the Y direction leads to more displacements of the beam, and thus more Von Mises stress and My bending moment. That explains the lower stress and moment values for the structure with an anisotropic membrane soft in Y direction and stiff in X direction.

However, one remarkable property is that concerning membrane forces, beam stresses and bending moment, an anisotropic membrane seems better than an isotropic membrane. And because the lower displacements are owned by the membrane which produces the bigger membrane forces, the anisotropic membrane with softer fibres in Y direction and stiffer fibres in X direction seems to be for this system the best choice.

3.4.4. Influence of membrane fibres orientation

Tested variable
Influence of membrane fibres orientation is analysed in this paragraph. Structures with diagonally oriented warp and weft fibres were compared to structures with longitudinally oriented fibres.

System parameters
The system was tested with an isotropic membrane, presenting an E modulus of 1000 MPa, and a pre-stress of 0.01 kN/m. Tolerance was set to 0.06.

Results
First result is that a system with a membrane made of longitudinal fibres is more instable than a system with diagonal fibres. Calculation stops converging for a load of 3 kN for longitudinal fibres, against a load of 8 kN for diagonal fibres. The difference of convergence with the previous test series (paragraph 3.4.3.) is due to a lower tolerance.

Then, regarding all test parameters, structures with diagonally oriented fibres clearly show better behaviours: less displacement, lower maximal membrane force, lower beam maximal Von Mises stress and maximal bending moment My (see Figure 63, Figure 64, Figure 65 and Figure 66).
Figure 63: Node A and maximum deflections under increasing axial loading for diagonally and longitudinally oriented fibres

Figure 64: Maximal membrane force under increasing axial loading for diagonally and longitudinally oriented fibres
Figure 65: Beam maximal Von Mises stress during increasing axial loading for diagonally and longitudinally oriented fibres

Figure 66: Maximal bending moment $M_y$ in the beam under increasing axial loading for diagonally and longitudinally oriented fibres

Principal membrane forces orientation for diagonally and longitudinally oriented fibres is represented under a load of 1 kN in the following illustration (Figure 67). Vectors scale is the same for both orientations. It is clearly visible that forces paths are less organised for longitudinally oriented fibres than for diagonally oriented fibres. The better distribution of membrane forces are an explanation for the improved behaviour of the latter system.
Conclusion

This test clearly shows that a membrane with diagonally oriented fibres is more advantageous for the global structure under loading than a membrane with longitudinal fibres.

Furthermore, the huge load-bearing capacity of this hybrid system, up to 8 kN, must be highlighted.
Conclusion
The scope of this master-thesis was to investigate the influence of the restraining membrane parameters on the load-bearing capacity of an active-bending hybrid structure. A planar simple active-bending hybrid structure has been analysed for varying parameters under loading, using the finite element method.

Three parameters of the membrane have been chosen to be the variables of the study: stiffness of the membrane (E-modulus) in terms of isotropic or anisotropic membrane properties, orientation of warp and weft fibres, and pre-stress.

Assessment criteria of the simulations were the displacement of the node A (in the global XY plane), maximum displacement in the structure, maximal membrane force, bending moment $M_y$ of the beam, and beam maximal Von Mises stress.

It has been demonstrated that a membrane with diagonally oriented fibres is clearly more advantageous for the global structure under loading, than a membrane with longitudinal fibres.

The anisotropic membrane with softer fibres in Y direction and stiffer fibres in X direction seems to be the best choice for this system, regarding all assessment criteria.

In the case of this structure, the negative effect of the pre-stress is more important than its positive effect. The structure is thus more stable with a very low pre-stress of 0.01 kN/m. This underlines how active-bending structures are the result of a very delicate balance.

And finally, the huge load-bearing capacity of active-bending hybrid systems with correct parameters has been experimented, and highlight the potential of active-bending hybrid systems.

Development of model simulation in the finite element environment SOFiSTiK® was the challenge in the period of this master-thesis. Experience of the software SOFiSTiK® parameters, tricks and theoretical background had to be built in order to be able to correctly manipulate it. In order to carry this project, theoretical knowledge in the innovative field of active-bending hybrid structure had to be ensured through the absorption of a significant number of research papers and thesis.

Finally, the deep immersion in this digital design construction field was a particularly rich and intense experience. The need to quickly develop knowledge and skills in this innovative structural environment, as much concerning theoretical aspects as computational skills, was a real exciting challenge. The personal outputs of this experience are numerous, regarding methodology, organisation and knowledge development, applied in the field of innovative architectural and engineering computational structural design.
Bibliography


