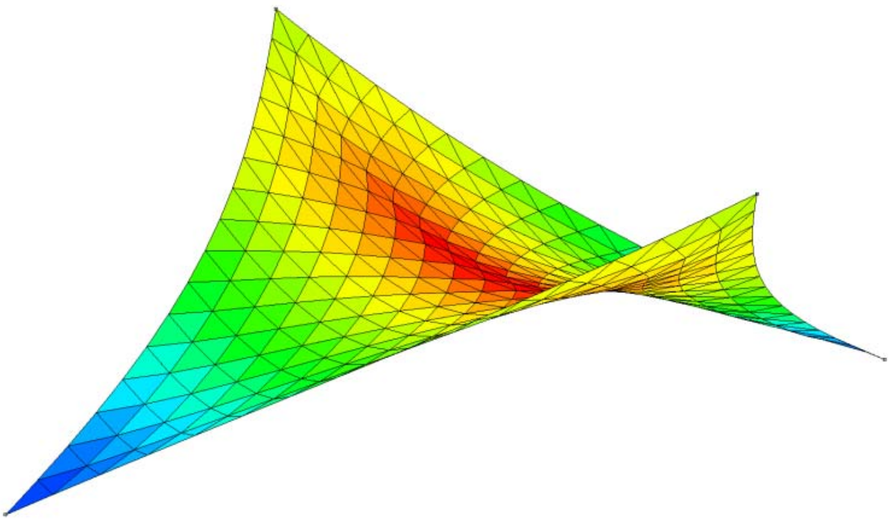
CHALMERS



Conceptual Design and Analysis of Membrane Structures

Master's Thesis in the Architectural Engineering Program

ERICA HENRYSSON

Department of Applied mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2012
Master's thesis 2012:51

MASTER'S THESIS IN ARHITECTURAL ENGINEERING

Conceptual Design and Analysis of Membrane Structures

ERICA HENRYSSON

Department of Applied mechanics
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2012

Conceptual Design and Analysis of Membrane Structures

ERICA HENRYSSON

Master's Thesis 2012:51
ISSN 1652-8557
Department of Applied mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
Load analysis of a hypar, see chapter 6.2

Reproservice Chalmers / Department of Applied Mechanics
Göteborg, Sweden 2012

Conceptual Design and Analysis of Membrane Structures
Master’s Thesis in *Architectural Engineering*
ERICA HENRYSSON
Department of Applied Mechanics
Division of Material and Computational Mechanics
Chalmers University of Technology

ABSTRACT

The light and freeform tension structures are interesting from a structural as well as architectural point of view. With the possibility to span long distances using minimum material they can also be a sustainable option.

The simplicity in the shape of fabric structures is not, however, reflected in the design process which is made complex by the flexibility of the material. Applied loads have a big impact on the final shape of the structure and the unique shapes of tensioned cable net and membrane structures cannot be described by simple mathematical methods. The shape has to be found through a form-finding process either with physical or computer models, a process that is made more complex by the need to include geometrically non-linear behavior and the anisotropic material properties of the fabric.

In this master thesis one approach for formfinding and analyzing tension membrane structures is described. Focus has been on the conceptual stage. For this the computer software SMART Form has been further developed, enabling the possibility to do real-time formfinding and analysis of fabric structures. The software is based on a method where the orthotropic membrane is modeled with a triangular mesh, where the mass is lumped on the nodes. As a computational tool dynamic relaxation is used to find the static equilibrium configuration for the structure. The advantage with this is that there is no need for formulation and manipulation of matrices common in the finite element method.

The results generated by the software is verified by comparing them with hand-calculations as well as with results from analyses done in another software, Tensyl. Results from parametric studies are shown, and finally the tool is used for conceptual designs for a stadium façade. Based on these simple test cases the software seems to be working fine.

Key words: Tensile structures, Fabric Structures, Formfinding, Dynamic Relaxation

Konceptuell design och analys av textila konstruktioner
Examensarbete inom Arkitektur och Teknik
ERICA HENRYSSON
Institutionen för tillämpad mekanik
Avdelningen för Material- och beräknings mekanik
Chalmers tekniska högskola

SAMMANFATTNING

De lätta och eleganta strukturerna som förspännda konstruktioner utgör är intressanta från ett arkitekt- såväl som ett ingenjörsperspektiv. Möjligheten att spänna över långa sträckor, med en minimal materialåtgång gör att de även kan vara ett miljövänligt alternativ.

Enkelheten i formen av textila konstruktioner återspegladas dock inte i designprocessen, som görs komplicerad av flexibiliteten i materialet. Pålagda laster har en stor inverkan på färdiga kabelnätskons- och textila konstruktioner, och dess unika former på kan inte beskrivas genom enkla matematiska metoder. Formen måste hittas, vilket görs antingen med hjälp av fysiska modeller eller med hjälp av datamodeller. Denna process görs mer komplicerad utav behovet att inkludera geometrisk icke-linjäritet och ortotropa materialegenskaper.

I det här examensarbetet beskrivs en process för att hitta formen och en analysmetod för textila konstruktioner. Fokus ligger på den konceptuella fasen av designprocessen. Programvaran SMART Form, har för detta, vidareutvecklats i syfte att möjliggöra att, i realtid, kunna hitta formen för textila konstruktioner och analysera dem. Programmet bygger på en metod där det textila membranet approximeras med ett triangulärt nät och textilens massa samlas i nätets noder. För att lösa ekvationerna används ”Dynamic Relaxation” som inte är beroende av uppförande och manipulering av matriser.

Resultaten från det utvecklade programmet har verifierats genom jämförelser med handberäkningar och med resultat från analyser gjorda i en annan programvara, Tensyl. Resultat från parametriska studier visas, och slutligen används verktyget för att modellera koncept för en arenafasad. Utifrån de utförda analyserna verkar det framtagna programmet fungera bra.

Nyckelord: Textila konstruktioner, Form analys, dragna konstruktioner, konceptuell design

Contents

Preface

In this thesis a tool for conceptual design and analysis of membrane structures has been developed. The majority of the work has been carried out at the engineering company Buro Happold's head office, in Bath (England), in close collaboration with their research and development team, SMART solutions.

Main supervisor has been Dr. Al Fisher, analyst and head of research and development in the SMART Solutions team. At Chalmers Senior Lecturer Dr. Mats Ander has assisted with supervision as well as being the examiner for this thesis.

Acknowledgments

Big thanks are due to the people at Buro Happold, and the SMART Team. I would especially like to thank Dr Al Fisher for his support, feedback and enthusiasm during the project, and for introducing me to the topic. I would also like to thank Jo Renold-Smith and especially Paul Romain for their feedback and for patiently answering all of my many questions about fabric structures.

Furthermore I would like to give thanks to Marcus Stark, for helping me understand the coding for SMART Form and Jens Olsson for his feedback and the many interesting discussions, as well as for helping me out with SMART Form. Finally I want to thank Mats Ander for his support and his positive attitude.

Notations

Roman upper case letters

A	cross section area
C	damping constant
E_x	Young's modulus for the warp
E_y	Young's modulus for the weft
$\bar{F}$	force vector
F_n	component, of force, normal to the surface
G	shear modulus
K_{ix}	total stiffness, in the x-direction, for node i
Ke	kinetic energy
M_i	nodal mass (node i)
P	pressure load
P_{ix}	applied force, in the x-direction, for node i
R	radius
S_i	total stiffness for node i
T	tension
T_i	tension in link i

Roman lower case letters

a	acceleration
c	constant
d	dip
g	constant
h	height of triangular mesh element
h^i	initial height of triangular mesh element
l	length
l_0	initial length
l_1	length of link in triangle coinciding with the warp direction
l_1^i	initial length of link in triangle coinciding with the warp direction
l_i	length of side i in triangular mesh element
m	mass
s	link stiffness

s^i	stiffness for link i
t	time
Δt	time step
u	distance/displacement
u_0	starting position
u_{ix}^t	displacement in the x-direction for node i, at time t
$\ddot{u}_{ix}^t$	acceleration in the x-direction for node i, at time t
v	velocity
v_0	Remaining velocity (from previous iteration)
$\overline{v_n}$	surface normal vector
v_{yx}, v_{xy}	Poisson's ratio
w	span width

Greek lower case letters

α_i	angle for corner i
α_j^i	initial angle for corner j
γ	shear strain
ε_x	warp strain (coinciding with the x-direction)
ε_y	weft strain (coinciding with the y-direction)
σ_x	warp stress (coinciding with the x-direction)
σ_y	weft stress (coinciding with the y-direction)
τ	shear stress

1 Introduction

With the ability to span large distances in a structurally efficient way, tension membrane structures offers a lot of interesting possibilities; from a sustainable, engineering and an architectural perspective. These elegant structures' complex designs require an understanding of shape and form, and the behaviour of the materials and the forces acting on it (and in it). The flexibility of the structure means that applied loads have a big impact on the shape.

The design process is made more complex by the fact that the shape of tensioned cable net and membrane structures cannot be described by simple mathematical methods. They have to be found through a form-finding process either using physical or computer models. The final shape then has to be translated from a three-dimensional undeveloped surface into two-dimensional cutting patterns. This is a complicated procedure since textiles have anisotropic properties (warp and weft). However the increasing capabilities of computers make it possible to more efficiently perform calculations for these structures.

Figure 1-1 Millennium dome (O2 Arena) has become a landmark for London. The textile dome has a diameter of over 100m and was engineered by Buro Happold. [1]

1.1 Background

On the market today there are several computer softwares that can form-find tension structures. A lot of the softwares are however relatively difficult to use and it takes a lot of time and effort to set up a model. As a result of this engineers are switching back and forth between a set of different design and analysis programs. When several concepts are to be evaluated this process becomes very time consuming. A lot of work is put into setting up a model that will generate more detailed results than what is necessary in the early stages of a project, and the design might not even be chosen to be developed further. A conceptual tool would therefore be useful, in which it is possible to quick and easy create the geometry, set up different properties and analyse the shape and the effect of changing the shape and materials in different ways.

The SMART Solutions team at Buro Happold develops new technologies to deliver efficient solutions for the built environment. SMART uses a combination of existing tools and customized in-house software developed to find solutions to a wide variety of problems. Examples of software are "SMART Form" a digital prototyping software for real-time generation of forms (described further in section 1.1.2) and "Tensyl" a nonlinear form finding and analysis software.

1.1.1 Formfinding

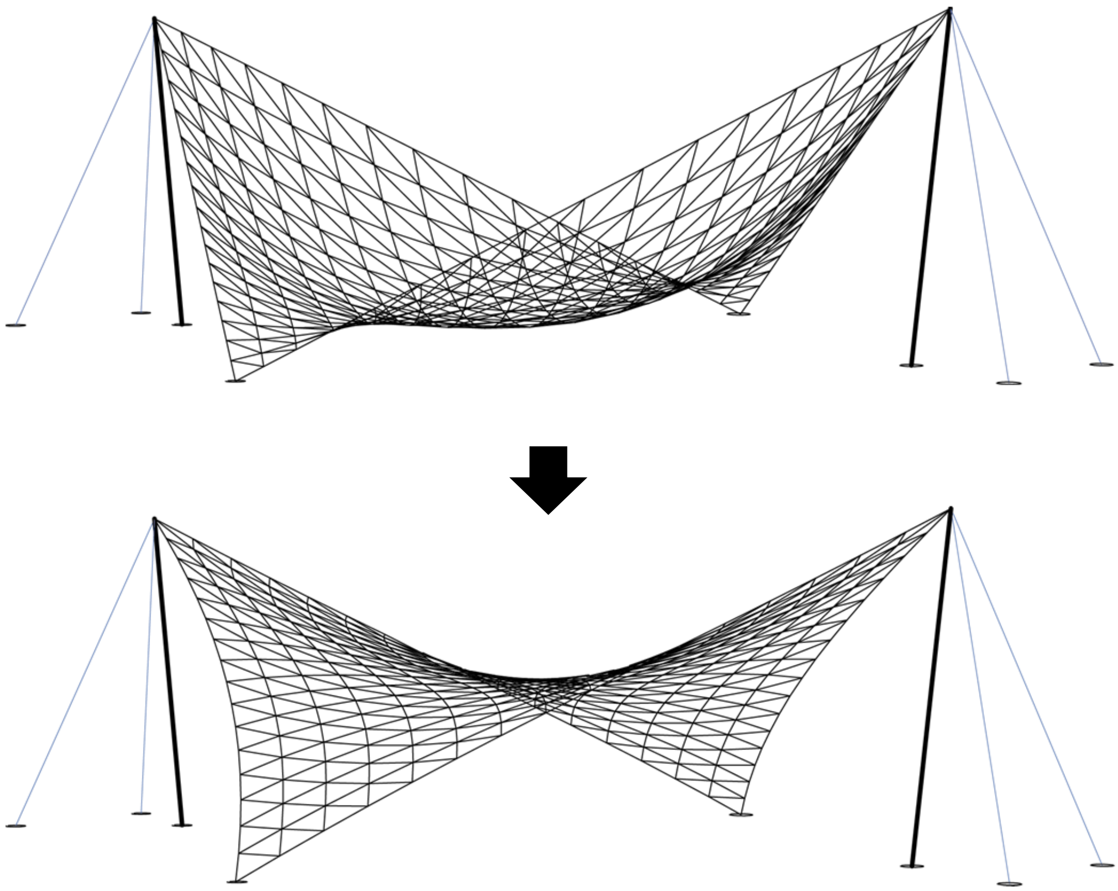


Figure 1-2 Formfinding of a tensioned membrane structure. Starting with the geometry at the top and ending up with the formfound hypar, at the bottom, where all nodes are in equilibrium

Formfinding is a term used for a lot of different methods to find the shape of structures of varying kind. Usually it is the structurally optimal shape that is the goal. When the term is used in this report it referring to the methods for finding the minimal surface, or an optimal shape, for tensioned membrane structures. This is done by controlling the stresses in the surface and finding the geometry where all points (nodes) are in equilibrium. An example of this is shown in Figure 1-2. To explain how the technique works Figure 1-3 and Figure 1-4 are illustrating the principle behind formfinding with a simplified 2D example. Note that this 2D example is an example of formfinding in general, and is not stress controlled. A gravitational force is acting on the nodes which are connected to each other by springs. The sum of the forces acting on a node will cause it to move until it has found the position where the resulting force is equal to zero.

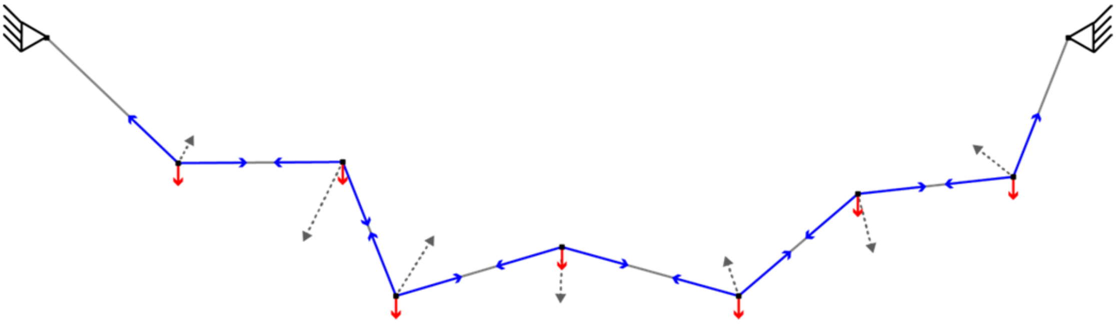


Figure 1-3 An unbalanced system. The dotted lines are representing the moving directions for the nodes.

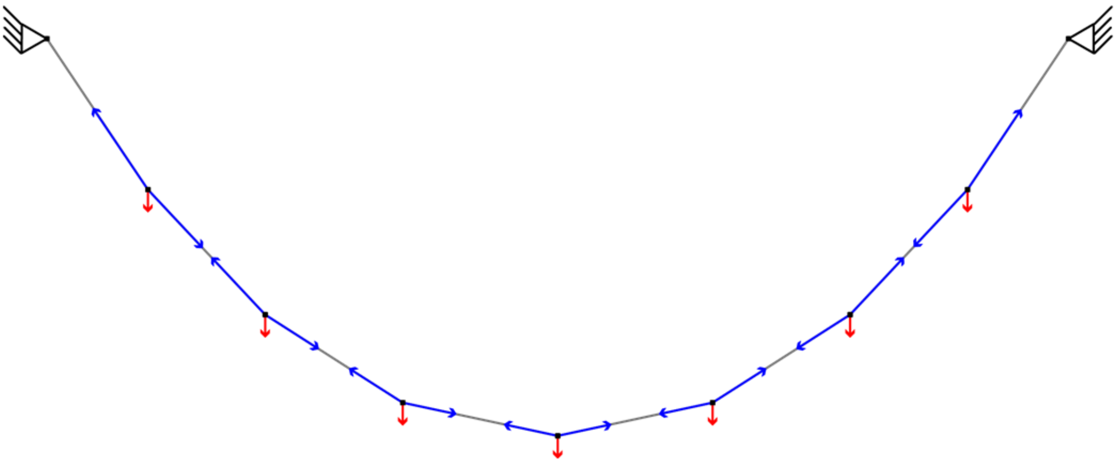


Figure 1-4 The same system as in Figure 1-3, but at a stage where all nodes are in static equilibrium.

1.1.2 SMART Form

SMART Form is Buro Happlod's in house formfinding software for sculptural/conceptual structural design. The software uses dynamic relaxation as the calculation engine (described further in chapter 3) and generates an optimized shape based on the defined gravity and the input geometry. However the simulations made are simplified and only works with systems of bars/springs, which lengths and stiffness can be increased or decreased. There is also no way to apply actual loads or properties to the model, or to output any measurable properties like forces.

As the tool is developed as a plug-in, for the 3D modelling software Rhinoceros, it can make use of its modelling and visualisation capabilities.

A previous version of the tool is free to download on SMART solution's webpage (http://www.smart-solutions-network.com/group/smartform). The tool has however now been developed further, allowing the user to choose between different mesh geometries. It can also visualise different geometric properties on the original and deformed structure, such as panel area, slope, planarity etc. The tool developed in this work is adding on to this platform, using the existing methods and structure of the code.

SMART Form is written in C#, an object oriented programming language and uses Rhinoceros' SDK. With the software structured in isolated classes that make use of objects, with different properties assigned to them, further development of the tool is relatively easy. The existing classes can be expanded with subclasses, thereby making it possible to further develop the different objects in the code without interfering too much with the existing functionalities.

It should be mentioned that the software is developed by structural engineers and not by professional programmers, for this reason the code is most likely not always structured in an optimal way.

1.2 Aim

The aim of this thesis is to develop a conceptual tool for designing tensioned fabric structures, to be used for formfinding and initial quick analyses of the structures. With this tool open up for the possibility to quick and easy sketch and try out several ideas and from these being able to make informed decisions about the design.

1.3 Scope

The focus of this thesis has been on the conceptual stage of the design process for a structure, and how concepts for complex tensile structures could be drawn up and evaluated. For this purpose a tool has been developed to fill the gap that exists between the sketching of ideas to the establishment of a calculation model. The tool is not supposed to replace exciting analysing tools and not proposed to be used as a software for the final calculations.

The intension at this stage is not to deliver a finished product, but rather to develop an analysis tool that works for certain cases. Time has not been spent on developing a code that in every aspect is as efficient as possible.

Furthermore the design of supports and details are outside the scope of this thesis, as focus has been on conceptual design of the fabric structures themselves.

2 Tension Structures

Figure 2-1 Germany's pavilion at the Expo 67 world's fair, in Montreal, designed by Frei Otto and Rolf Gutbrod. [2]

Using fabric as protection against rain, wind and sun is nothing new, tents in various shapes can be found in all parts of the world, and has been around more or less since humans started weaving textiles. They are easy to build and light to transport. In Arabia and North Africa they have long used fabric stretched over alleys to create shade and in a smaller scale the umbrella could also count as a fabric structure, shading the sun and protecting from the rain. The big tensioned membrane structures we see in buildings today gained popularity at the world Expo in Montreal, 1967, with Germany's contribution, designed by Frei Otto and Rolf Gutbrod, Figure 2-1. The design for the pavilion, "a tent-like roof made of a net of cables", won partly because the lightweight construction would be very economical to transport overseas (Klaus-Michael Koch, Karl J Habermann, 2004). Although the original design was a tensile membrane structure, it was finally built as a cable net structure, using the membrane as a secondary cover. Before this point the most common kind of membrane structure had been air supported i.e. either supported by air pressure or with inflated ribs (Bechthold, 2008).

Another mile stone in the history of tension structures is perhaps the Olympia roof in Munich, 1972, designed by Behnisch & Partner and Frei Otto (Klaus-Michael Koch, Karl J Habermann, 2004). With this cable net structure, the first ever done in this scale, research in both computational formfinding methods and computational structural analysis were taken to a new level (Bechthold, 2008).

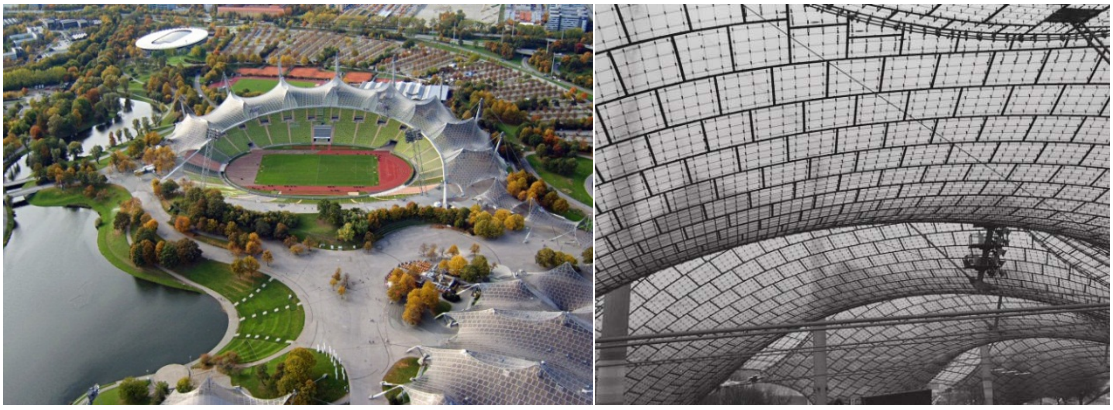


Figure 2-2 & Figure 2-3 Olympic stadium in Munich from 1972. [3], [4]

For sport arenas of all kinds membrane structures have become more or less standard, today. This because fabric structures are efficient solutions when it comes to cover large areas without intermediate supports. The roof at London Olympic stadium (2012) being one of the most recent examples of this, Figure 2-4. Here they needed a light structure that could be dismounted after the games. In this case the whole arena is not covered; the fabric is there to create shelter from the wind for the athletes.

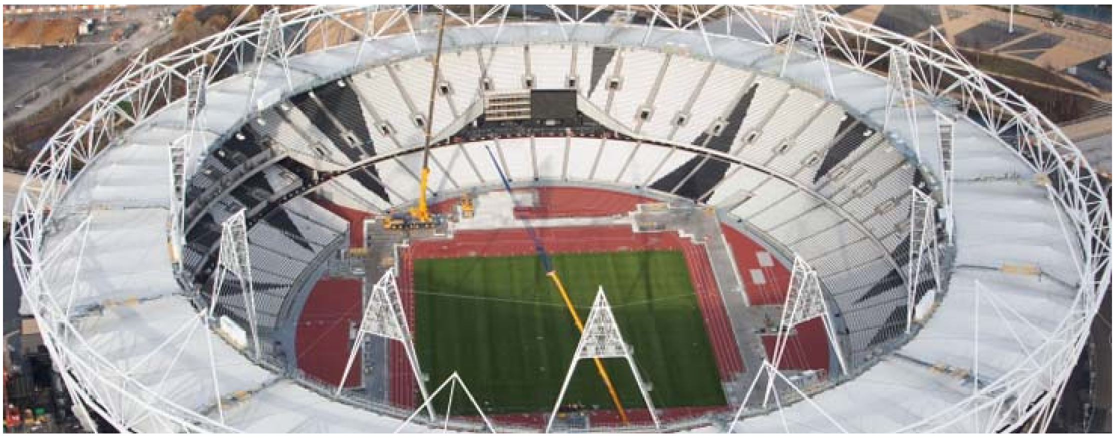


Figure 2-4 London Olympic stadium. Architects: Populous. Engineers: Buro Happold [5]

As tensioned fabric structures start to become more intricate parts of different kinds of buildings the demands on the material gets more and more complex, in case of energy, sound absorption, translucency, etc.. Bangkok international airport, Suvarnabhumi, (finished 2006) being one example of this. Here they used a multiple layer system to be able to absorb the sound, both from the interior and the airplanes on the outside (Dixon, 2007), see Figure 2-5.

Figure 2-5 Bangkok international airport, designed by Murphy/Jahn and Werner Sobek. [6]

2.1 Shapes

Tensioned fabric structures can, as the name implies, only carry load in tension, therefore most of the structures are anticlastic. This way at least one direction will be working in tension, even under heavy loads. Examples of anticlastic shapes can be seen in Figure 2-6 , Figure 2-7 and Figure 2-8.

To further ensure that the structure always is in tension, and ideally never leaving the membrane slack anywhere, a prestress is applied to the fabric.

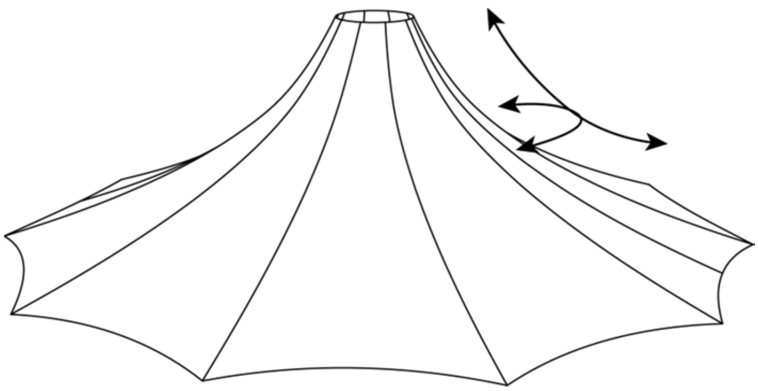


Figure 2-6 illustrating a conic and its anticlastic shape

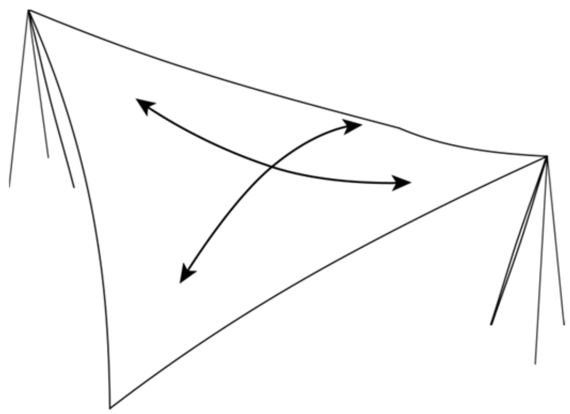


Figure 2-7 Another anticlastic shape is the hypar

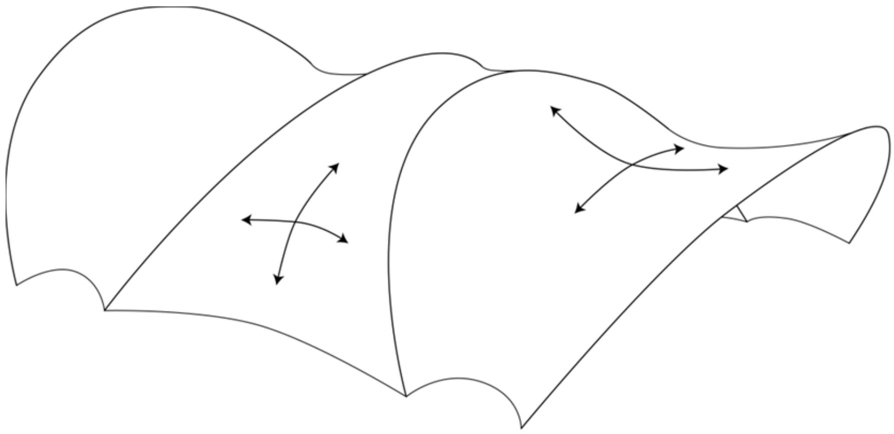


Figure 2-8 Anticlastic structure with arch supports

2.1.1 Soap film models

A classic way for finding the shape of membrane structures is the soap film model. Within a frame a soap film always contacts to smallest surface possible, i.e. the minimal surface, which is what membrane structures, in general, strive towards. With endless possibilities of shapes for the boundaries there is also an endless number of shapes for soapfilms. Frei Otto did a lot of experiments with this kind of models, which led to a new quality in tent-building architecture (Frei Otto, Bodo Rasch, 1995). Figure 2-9 is showing a soap film model of the Dance fountain, designed by him.

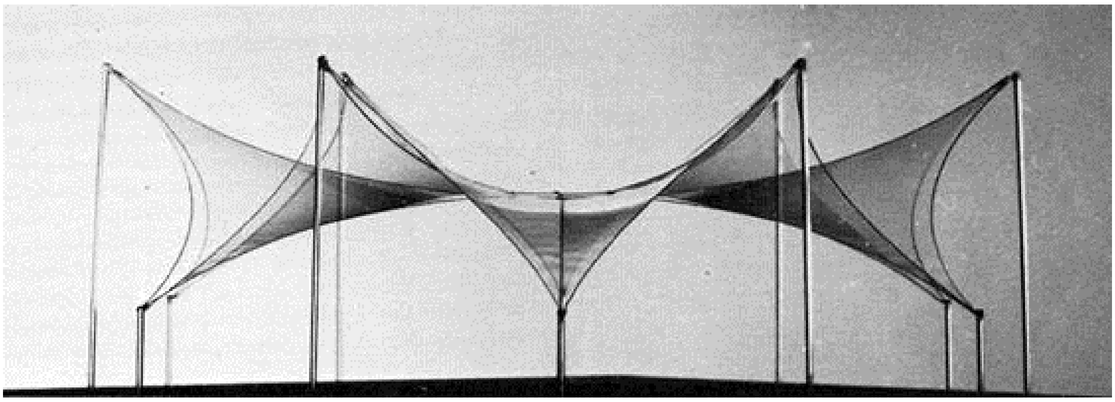


Figure 2-9 Soap film model of the Dance Fountain in Cologne, designed by Frei Otto. [7]

The soap film can be seen as the optimal membrane, when it comes to geometry. It is a good material for representing a large structure in a small scale model, something that is otherwise difficult to do with different kinds of textiles, since they have a different thickness and weight. The behaviour of a soap film can, however, be modelled mathematically in a computer. In this thesis this is accomplished with stress controlled formfinding, using uniform stress (explained further in chapter 4.3).

2.2 Textiles

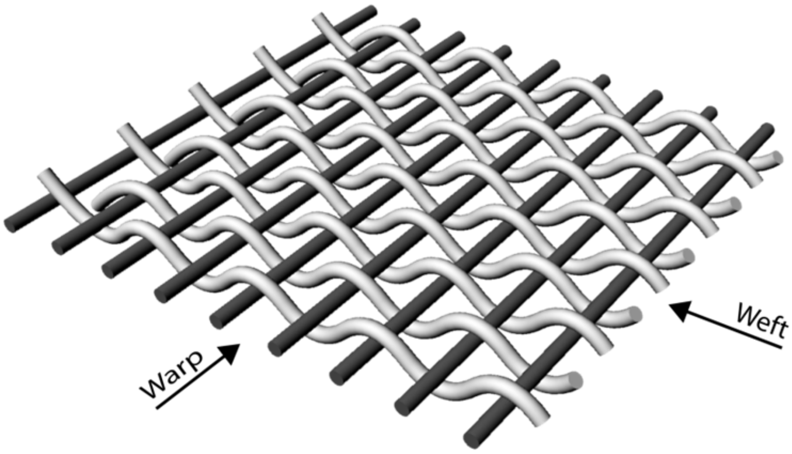


Figure 2-10 A textile weave with warp and weft directions indicated.

A fabric weave is clearly an orthotropic material. The stiffness in the warp direction is different from the stiffness in the weft direction and the fabric is a lot less stiff if pulled in a direction different from these principal ones. This is evident when you pull in any woven textile; it will deform a lot more when pulled in a direction diagonally to the weave, than when pulled along it. The reason for this is the way the weave is built up, with warp stretched in the production, and the weft (also called fill) going over and under the warp threads. Looking at Figure 2-10 it is evident that the effect of pulling in either the warp or weft direction should have consequences one the other, Figure 2-11 and Figure 2-12 is illustrating this. The threads in the weft are forcing the warp to move up and down.

The textile is more or less a quadrangular mesh and therefore, for understandable reasons, the shear stiffness is comparably low.

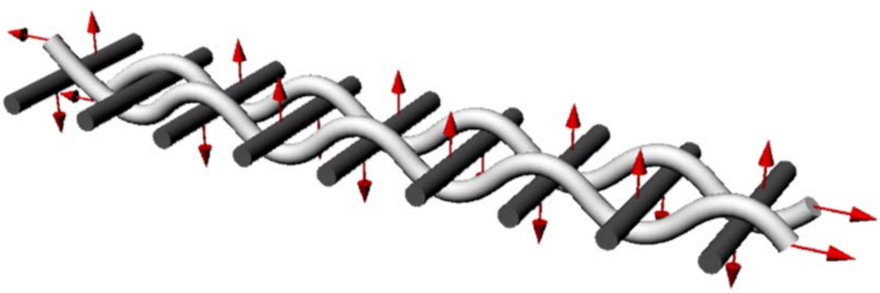


Figure 2-11 illustration of the effect of pulling in the weft direction. The red arrows indicate the movement.

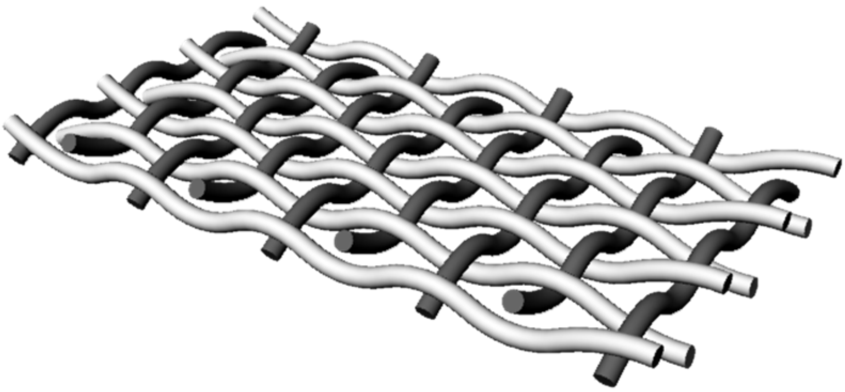


Figure 2-12 Result of tensioning the weft. The threads in the weft (white) are shortened to establish the pretension, and as a result the threads in the warp (grey) get stretched out.

2.3 Design Process for Tension Membrane Structures

In general the design of tensioned membrane structures is done in three steps:

1. Formfinding
2. Static Analysis
3. Patterning

In most cases the shape definition of the structure is not given by an obvious mathematical equation, therefore the formfinding procedure, i.e. finding the basic static shape of the structure, is an important part of the design process. This can be done using physical or computer models. In this work an iterative computational computer model is set up. When the shape is found (computed) the structure is analysed with load cases applied to it, using the formfound shape as initial geometry. Finally the prestressed 3D membrane (the formfound structure) is translated into a 2D pattern for the structure *(Lewis, 2003)*. At this stage the generated shape has prestress applied and the cutting pattern therefore has to be smaller than the final shape, as illustrated in

Figure 2-13.

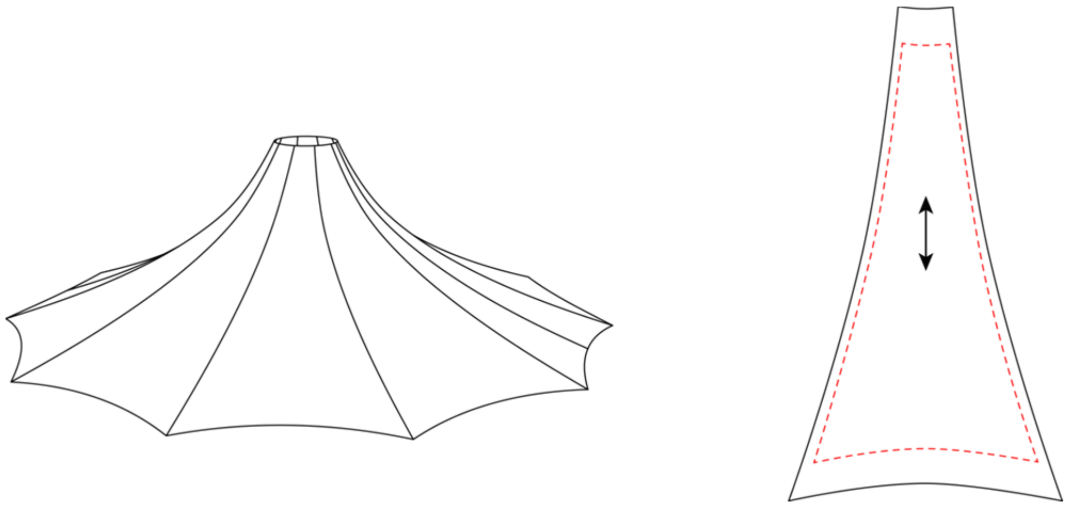


Figure 2-13 Patterning and prestressing principles. The black lines are illustrating the prestressed shape, and the dashed line is representing the cutting pattern (the arrow is showing the warp direction). By making the patterned pieces smaller than what they will be on the finished structure pretension in the membrane is achieved.

Designing fabric structures is not an exact science as a lot factors are imprecise. The strength of one weave is different from the next, even if they are of the same kind. It is also difficult to measure Poisson's ratio accurately (see chapter 4.5.1) and the calculation models are not exact representations of the membrane. By the industry this is dealt with by the use of high safety factors. (Paul Romain, Joanne Renold-Smith, 2012)

2.4 Cable Nets

In a lot of ways a cable net structure behaves the same as membrane structure. The biggest difference between the two when it comes to mechanical behaviour is that a structural textile has a more direct interaction between main directions in the fabric (warp and weft). There is also shear stiffness in a textile that does not exist in a cable net. It follows that the calculations for the a cable net structure is more straightforward, but the shape of the structure still has to be found through formfinding procedures, same as for membranes and, at least for larger structures, it is unpractical to do these calculations by hand.

3 Dynamic Relaxation

Originating from an analogy for computations for tidal flow, drawn by Day, Dynamic Relaxation (DR) has been developed as an explicit solution method for the static analysis of structures (Barnes, 1999). Non-linear material effects were firstly introduced to the method by Holland (Holland, 1967). Later Day and Bunce applied DR to the analysis of cable networks (Bunce J W and Day A S, 1970), and finally Brew and Brotton developed the method to the form most widely used today; a vector form which does not entail a formulation of an overall stiffness matrix (Brew J. S., Brotton D. M., 1971). DR is especially suitable for highly non-linear problems, such as the focus of this work; membrane structures.

3.1 Base Equations

The method is based on a model were the mass of, in this case, a continuum is concentrated to a set of points (nodes) on the surface or in the "joints" of a cable net. By specifying the relationship between the nodes (how they are connecting to each other) the system will oscillate around its equilibrium, under the influence of the out of balance forces. By damping the movement of the nodes the system will, with time, come to rest when static equilibrium is achieved.

The process is based on Newton's second law of motion:

$$Force = Mass * Acceleration$$

From which, for one particle, you get:

$$a = \frac{F}{m} \tag{3-1}$$

$$v = v_0 + a * \Delta t \tag{3-2}$$

$$u = u_0 + v * \Delta t \tag{3-3}$$

Where v_0 and s_0 are the velocity and the position from the previous iteration and Δt is the time step used.

Rewritten for the displacement of any node i in the x-direction at time t:

$$P_{ix} - K_{ix} u_{ix}^t = M_i \ddot{u}_{ix}^t \tag{3-4}$$

Where P_{ix} is the x-component of the applied loads, K_{ix} is the x-component of the total stiffness for the links connecting to the node with mass M_i, and u_{ix}^t and $\ddot{u}_{ix}^t$ are the displacement and accelerations in the x direction.

The same calculations are then carried out for the y- and z-direction.

3.2 Damping

For the system to come to rest a damping term is introduced. Depending on the purpose of the simulation the accuracy of this term is of varying importance.

The ways of damping the system could be divided into two categories, where either or both could be chosen depending on the system.

3.2.1 Viscous Damping

Using viscous damping the movement of the system will for each iteration be decreased with a damping constant. In the case of this work it is the velocity that is decreased, (3-2) then becomes:

$$v = C * v_0 + a * \Delta t \qquad (3\text{-}5)$$

Where C is the damping constant and has a value between one and zero.

When the aim is to simulate the dynamic behaviour of a structure, the viscous damping term is essential. The value for the constant will determine with what speed the structure is moving, and how realistic that movement is. For computer simulation of cloth, in games or animated movies etc. a correct damping factor is therefore indispensable. However when the goal only is to find static equilibrium, as it is for this project, the accuracy of this value is of less importance, since the dynamic behaviour is not what is sought. Other than that, the damping will only change the time it takes to reach the equilibrium for the structure. It should be noted that a higher speed, does not necessarily lead to a faster convergence, since the nodes, with a high speed, can end up moving past the equilibrium position at each iteration.

3.2.2 Kinetic Damping

With the kinetic damping the system is brought to rest by setting the initial velocity, v_0, in (3-2) to zero every time the kinetic energy is at a peak. The process is then restarted from the current geometry.

A kinetic peak is found when the current kinetic energy (Ke) is less than the kinetic energy from the previous iteration ($Ke(t) > Ke(t + \Delta t)$). The true peak occurs sometime between t and $t + \Delta t$ and to minimize the iterations necessary this time could be calculated or it could be assumed that the peak occurs at $t + {}^{\Delta t}/_{2}$, see Figure 3-1. However for the purpose of this thesis it is sufficient and convenient to simplify this even more by just resetting the velocities at time $t + \Delta t$. Since the time step used is relatively small, the kinetic energy at time t and at $t + \Delta t$ should be fairly similar. This simplification means that a few more iterations are needed, but for this work (at this stage), speed was not the main focus. A graph with the kinetic energy plotted for the formfinding of a hypar, can be seen in Figure 3-2.

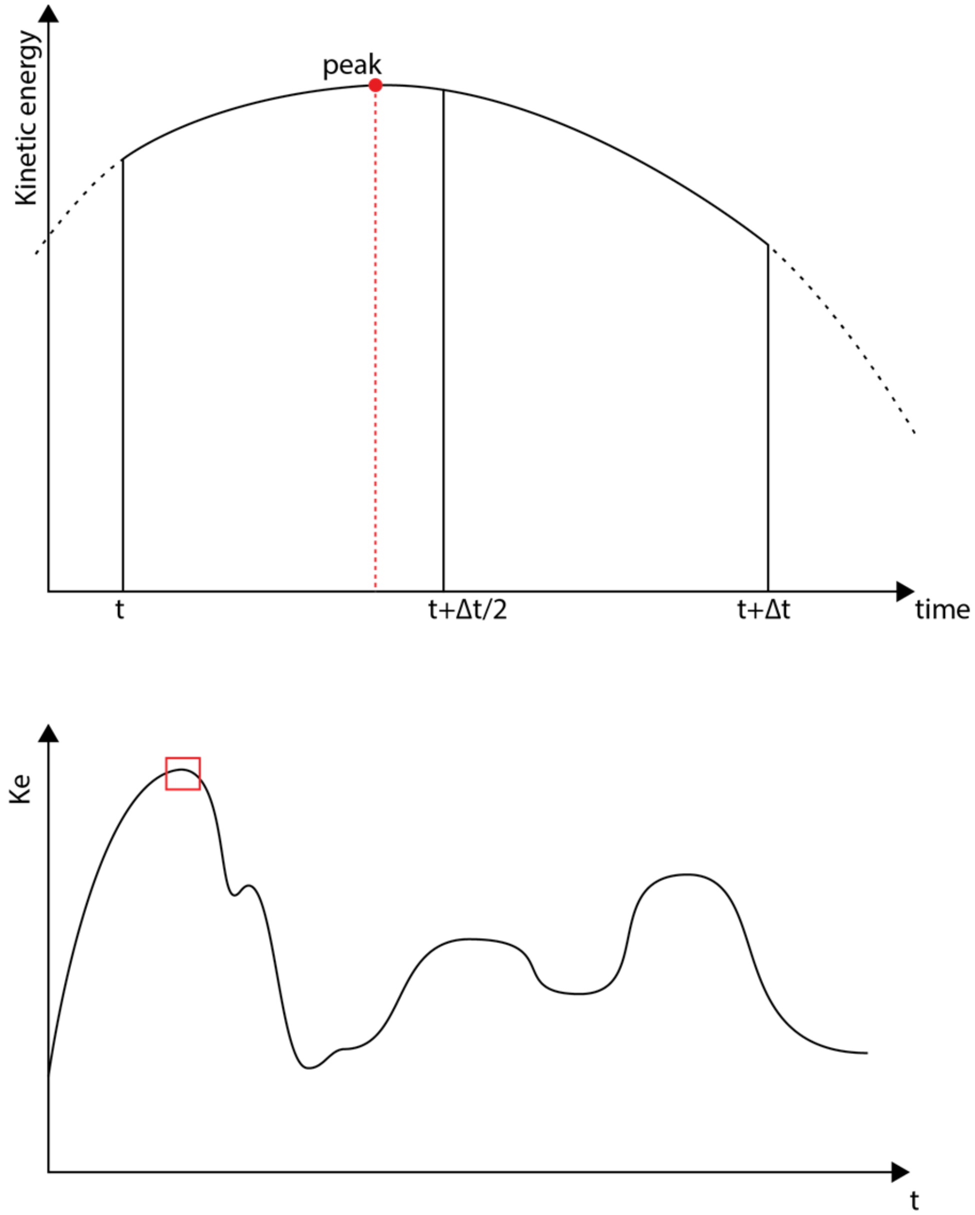


Figure 3-1 The principals for kinetic damping. The bottom graph is showing an example of a plot of kinetic energy for an un-damped structure and the top graph is showing the section marked with the red square in the bottom graph.

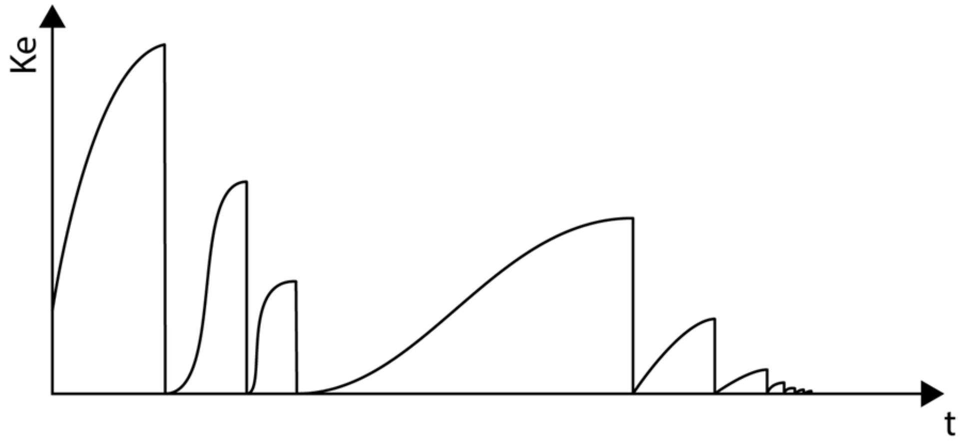


Figure 3-2 A typical graph for the kinetic energy, using kinetic damping.

4 Modelling

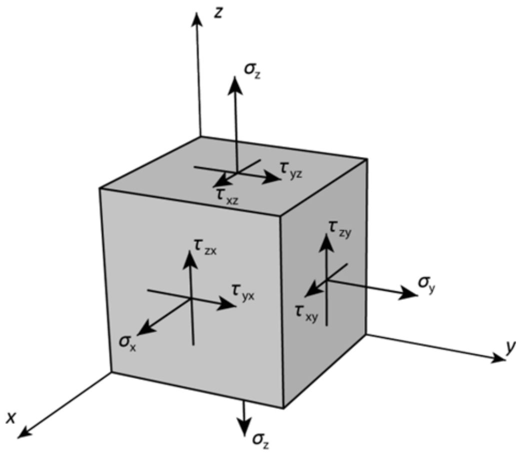


Figure 4-1 Stresses in an idealised 3D material

In structural computation models the material propreties are usually simplified, Figure 4-1 is illustrating one way of representing this. This model could be used to illustrate the stresses, on a material level, in the different threads in a structural fabric, like the one illustrated in Figure 4-2. It is however convinient to look to represent the fabric as an ortotropic 2D material/surface, as shown in Figure 4-3.

In general tensioned membrane structures behave geometrically non-linear, but the material in itself can be assumed to have a linear stress strain relationship.

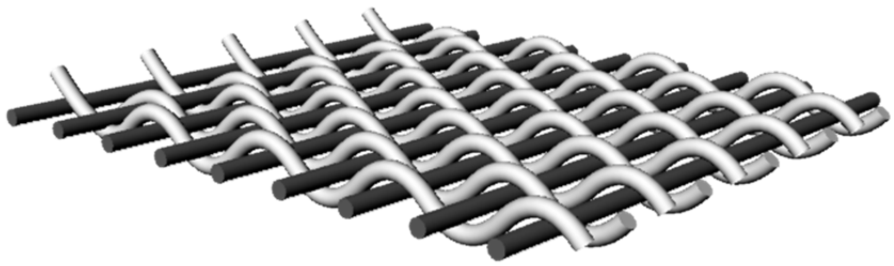


Figure 4-2 The weave of a textile

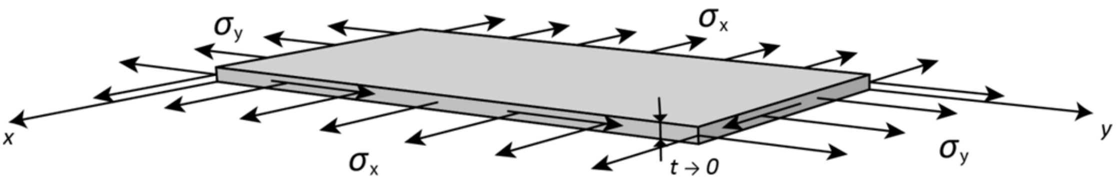


Figure 4-3 Stresses in an idealised 2D membrane

In order to do calculations on a tensile fabric structure the membrane is, in this thesis, simplified even further in a computation model, by representing the membrane with a triangulated mesh, Figure 4-4. This allows the system to represent the different properties for warp and weft, as well as the shear properties for the textile.

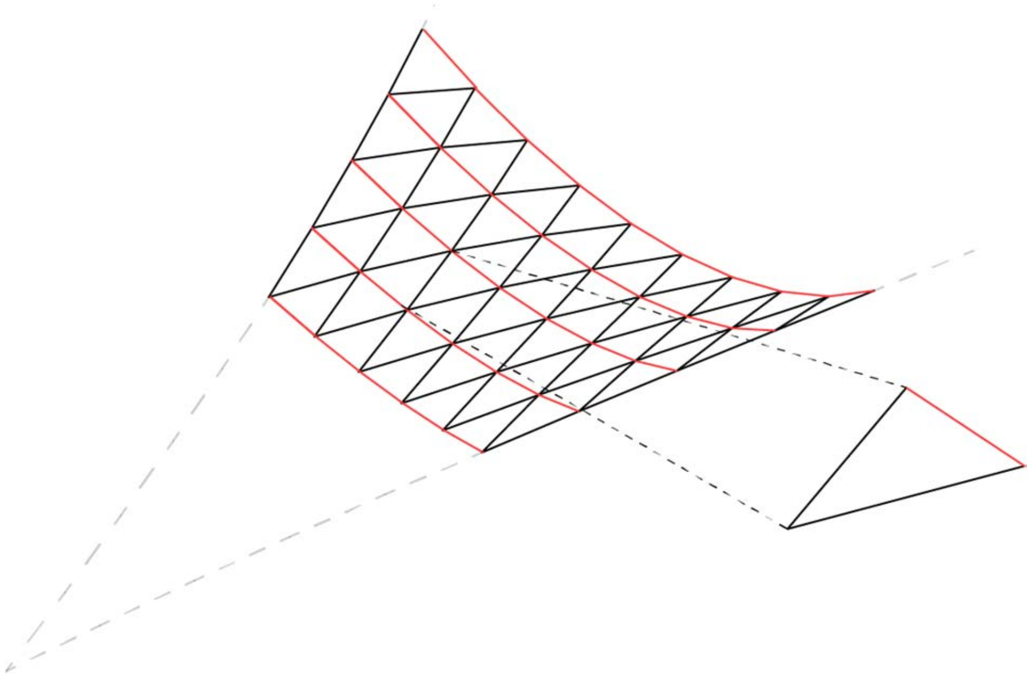


Figure 4-4 Illustration of the mesh and triangles, where the red lines are representing the warp direction. Calculations are carried out, separately, for each triangle in the mesh.

The density of the used mesh depends on the purpose of the simulation, if it is a model to be used for patterning, the distance between the strings in the warp direction is usually set to one width of a fabric patch or half a width.

4.1 Simulating textile behaviour

A number of approaches are available for calculating and simulating different kinds of textile shapes such as matrix methods, continuum mechanics and particle systems (Volino, P & Magnenat-Thalmann, N, 2001). This work will focus on the latter, since it is shown to be efficient when dealing with tensile structures (see section 4.1.1).

Guidance when choosing a method for the simulation of membrane structures can be found looking at the methods for cloth simulation in computer animations, since this is a field in rapid development and where it becomes very important to find a method giving the optimal compromise between accuracy and speed. Volino and Magnenat-Thalmann are talking about various considerations that have to be taken into account, when choosing integration method for the problem to be simulated. Among those are:

- The size of the system/structure i.e. number of particles used to describe the mechanical model.
- The time it takes to carry out the calculations for each iteration
- The type or purpose of the simulation. Either extensive computation of the motion, requiring accurate evaluation of the dynamic behaviour along time, or, as is the case in this thesis, a relaxation process where the simulation has to converge to a static rest state as quickly as possible.

The formfinding of membrane structures, done in this thesis, could be compared to the simulations of draping cloth, where it is the final static shape that is of interest and an accurate simulation of the dynamic behaviour, leading up to it, is of less importance. Several studies have been carried out to get the optimal method for this computer simulation, the implicit Backward Euler Method is one recommended approach, which is especially efficient for draping simulations (Volino, P & Magnenat-Thalmann, N, 2001). However a major part of simulating draped cloth are "collision handling", i.e. simulating the folds and making sure the surface does not allow it to go through itself. This is not a problem that needs to be considered in this thesis since when working with minimal surfaces of tensioned structures it is assumed that no wrinkling or collision should occur in the final shape (unless the structure fails).

4.1.1 The particle system representation

The mechanical system is represented by a particle system with a set of discrete masses. The membrane or cable net (the surface) is represented by the geometry of the connections (links) between neighbouring particles (nodes). The mechanical behaviour is then simulated by calculating the interaction forces between the particles. These will cause nodes to move. Tracing the velocities and displacements of the particles through time you get a simulation of the behaviour of the fabric.

4.2 Calculation analogies

The behaviour of pretensioned membrane structures is mathematically complex and, same as for different cloth simulations, there are a number of methods available to describe it. In general a particle system representation is used. There are then different methods for representing the interactions between the discrete masses in the system. The approach chosen in this thesis is the three nodal constant strain triangle, see section 4.2.1, this is also the most widely used approach in the industry. Although the method has shown to be deficient in the presence of large strains and in the presence of shear stress in particular (Lei, 2010) it is deemed to be the best option for developing this conceptual tool. This because the simplifications, with the method, enables a quick calculation and generates accurate values as long as the shape is reasonable. It should also be noted that this is the same analogy used in commercial tools such as GSA and in Buro Happold's in house analysis software, Tensyl (At Buro Happold the inaccuracies that may occur is dealt with by the use of high safety factors).

Other analogies are for example different matrix approaches, talking with Professor Peter Gosling he suggest a six nodal linear strain triangle element, for a more accurate result. This means that the triangles can have a curved geometry and will therefore give a smoother transition between the elements, also described in (Lei, 2010).

4.2.1 Constant Strain Triangle

The method with constant strain triangles (CST) originated from a desire to simplify membrane structures to a pseudo cable-net system, for which the calculation method is more straight forward. The strains in the membrane are applied to the links (bars), in an imaginary triangular mesh, each having a constant value of strain. This way the structure can indeed be analysed as a simple truss/cable-net, see section 4.3.1.

As mentioned before, the strain-displacement method used in this thesis assumes small strains. Zhang Lei, is showing that a finite element philosophy, with high order terms included in a continuum framework, could be used with the CST element method, and thereby being able to deal with large stresses (Lei, 2010). However this approach is considered to be unnecessarily complicated for a conceptual design tool.

4.3 Formfinding of Prestressed Membranes

To find the optimal form for a membrane structure (the minimal surface), with given boundary conditions, the shape is controlled by dictating the stresses in the warp and weft direction i.e. setting the pretensions and letting these be the only thing controlling the shape. This is known as stress controlled formfinding, also known as force controlled formfinding. The set stresses are converted into forces in the links (see section 4.3.1), causing the nodes to move until they find their equilibrium position, the links themselves have no stiffness at this point and will just adapt to the found shape. If the warp and weft stress is set equal this will generate a surface with a uniform stress, just as a soap film structure. However, just as soap film model cannot be created without closed boundaries, trying to do stressed controlled formfinding without extra fixity or forces applied to the edge nodes will result in a surface that will shrink until it eventually disappears, see Figure 4-5. Therefore the formfinding is done with either constraints or elastically controlled cables along the edges, see Figure 4-6.

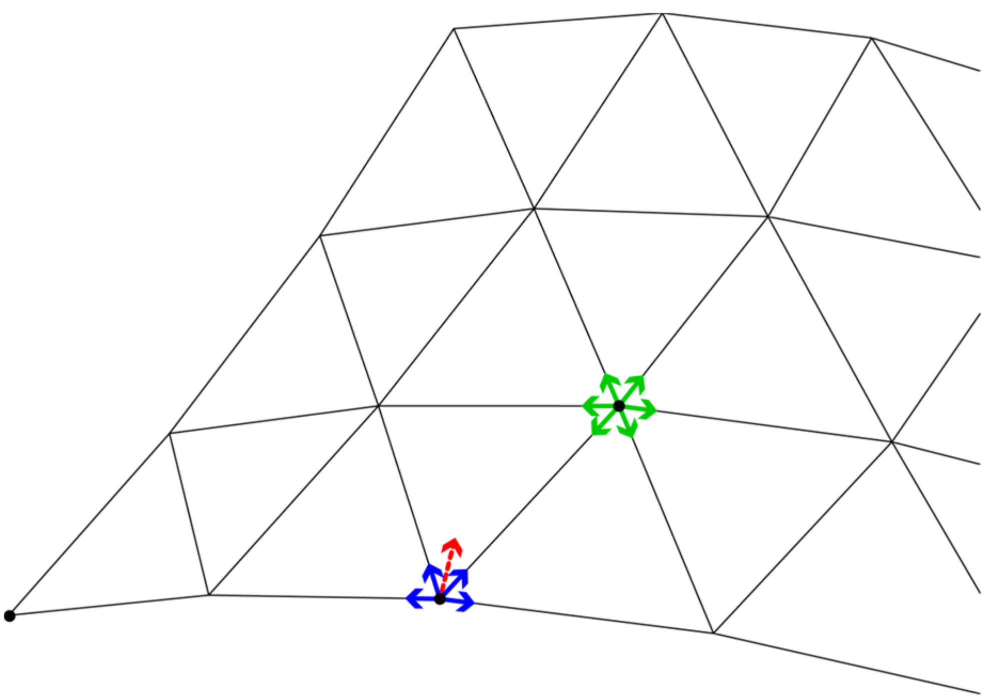


Figure 4-5 The forces acting on a node in a mesh, if the structure was force controlled. The node in the middle of the mesh is in equilibrium and will not move, in this stage. While the node on the edge will continue moving towards the middle of the surface, since there is no force holding it back.

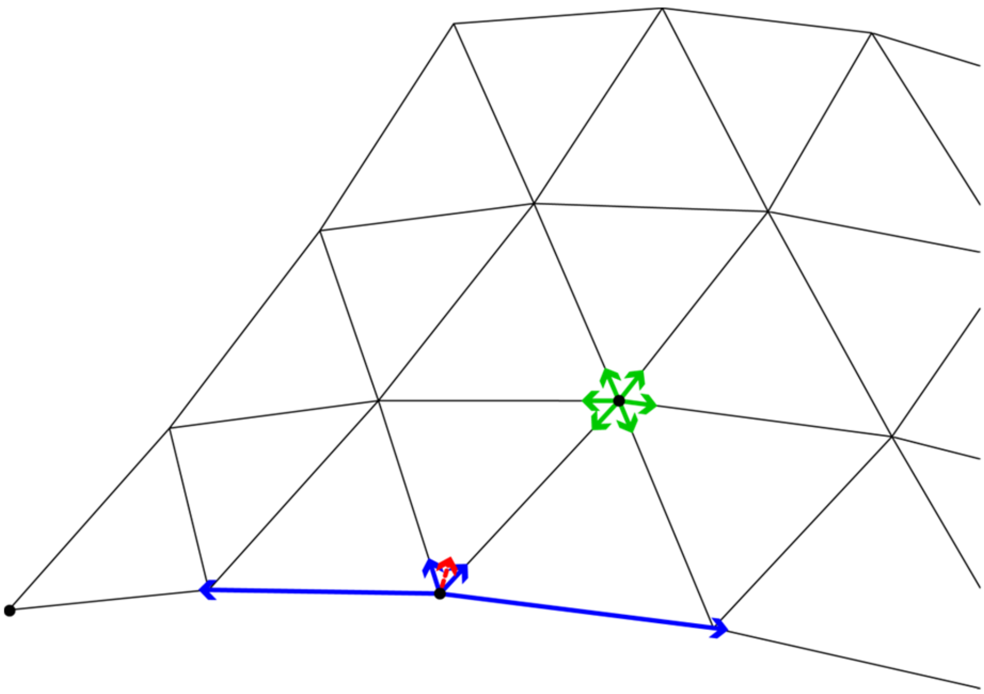


Figure 4-6 The same mesh and nodes as in Figure 4-5, but with an elastically controlled cable running along the edge. Here the node in the middle is in equilibrium, same as before. For the edge node, however, the forces in the links along the edge are a lot higher than the ones from the stress controlled membrane, and are therefore starting to balance out. As the node will move further in towards the centre, the forces in the edge links (the cable) will increase.

4.3.1 Link forces in terms of membrane stresses

In the imaginary mesh, representing the membrane in the structure, it is common to have one side of the triangles aligned with the warp direction of the fabric. This simplifies the equations and makes the model more readable. In the following figures the reference-axis/x-axis are aligned with the warp direction. This means that the reference axis will move with each triangular element, as it deforms, see Figure 4-7. Equations (4-1) are used to convert the warp and weft stresses into link tensions, in the mesh (Barnes, 1999).

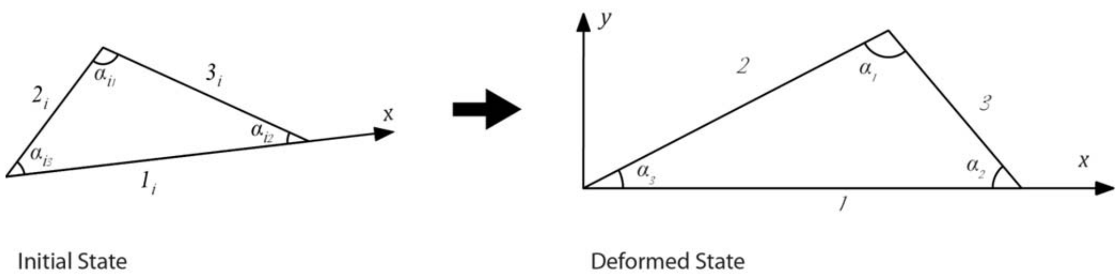


Figure 4-7 Illustrating a triangular element in its original and deformed shape. All sides are assumed to remain straight

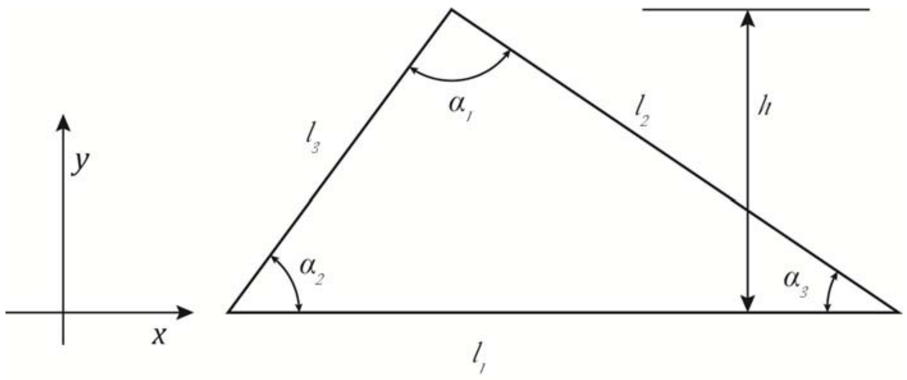


Figure 4-8 Showing the triangular element used in the mesh representing the fabric.

$$T_1 = \frac{h}{2}(\sigma_x - \sigma_y) + \frac{\sigma_y l_1}{2\tan\alpha_1} \quad ;$$

$$T_2 = \frac{\sigma_y l_2}{2\tan\alpha_2} \quad ; \qquad T_3 = \frac{\sigma_y l_3}{2\tan\alpha_3} \qquad (4\text{-}1)$$

Where T_i is the tension associated with the side l_i etc. as seen in Figure 4-8 and σ_x is the stress in the warp direction and σ_y is the stress in the weft direction. (The equations from which (4-1) is derived can be found in Appendix A)

For the case with minimum surface, with uniform stress ($\sigma_x = \sigma_y = \sigma$) equation (4-1) becomes:

$$T_i = \frac{\sigma\, l_i}{2\tan\alpha_i} \qquad (4\text{-}2)$$

For both the cases (uniform and non-uniform stress) this means that if the mesh, consist only of right angled triangles there will be no force (from the membrane) in the diagonals of the mesh, see *Figure 4-9*. This because $\tan 90^o \rightarrow \infty$.

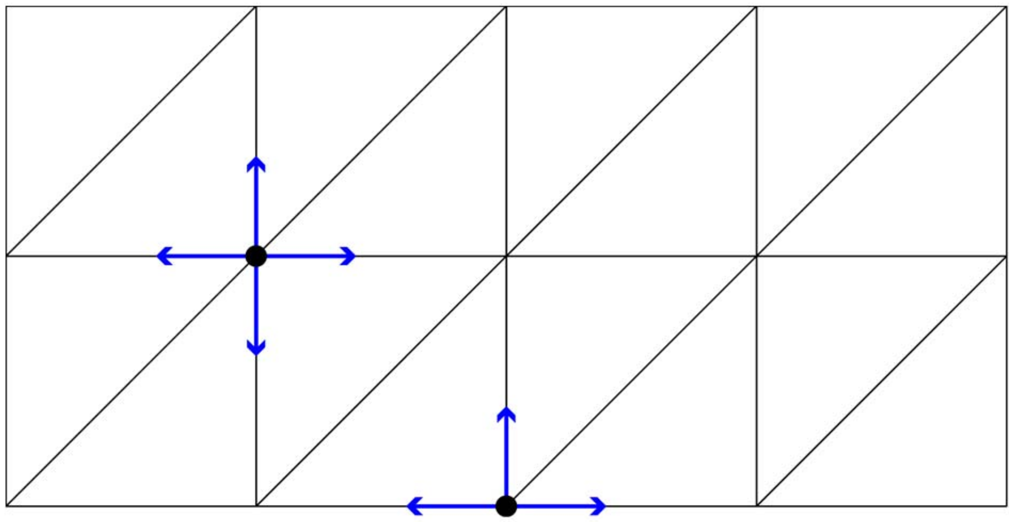


Figure 4-9 In a mesh consisting of right angled triangles, there is no force in the diagonals.

It is relatively easy to calculate the link tensions even if the triangular mesh is not aligned with the reference axis (i.e. the warp direction), see Figure 4-10 and equation (4-3) (Barnes, 1999). This method would also require less precision in the original meshing of the structure. However the problem would then be finding a good way of defining the warp direction for the structure. It was therefore thought that the software would be easier to use and more intuitive if the warp direction is aligned with one of the three sides of the triangular elements in the mesh.

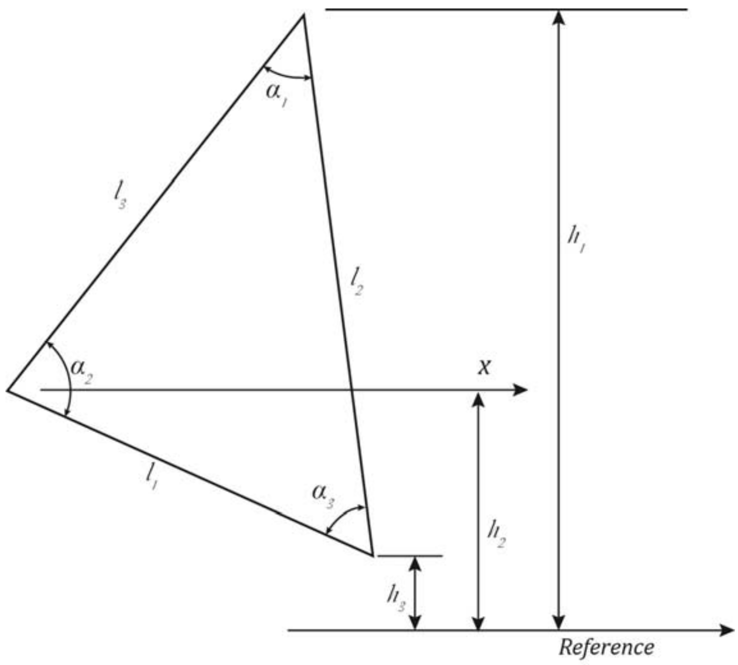


Figure 4-10 Triangular element where no side is aligned with the warp direction of the membrane.

$$T_i = \frac{\sigma_y}{2\tan\alpha_i} + \frac{(\sigma_x - \sigma_y)}{4\,A}(h_i - h_j)(h_i - h_k) \qquad (4\text{-}3)$$

Where variables can be seen in Figure 4-10 and T_i is the tension in the link l_i.

4.3.2 Control Strings

When formfinding a minimal surface, or soap film surface, the nodes can find equilibrium anywhere on the surface, generating a grossly deformed mesh (compare Figure 4-14 with Figure 4-15). This is because the links in the membrane mesh has no stiffness, during the formfinding that would result in a larger force if a node has moved far away from its original position or its neighbouring nodes. With an infinite number of equilibrium positions the nodes are likely to never stop moving. Even if they find equilibrium this scenario is not desirable when continuing with the load analysis, mainly because there are no longer any lines following the warp direction of the fabric. To prevent this, and to get more control over the structure, control strings, sometimes called G-strings or geodesic strings, are introduced. In this thesis they are set to be the links following the warp direction of the fabric, which is usually the case in different analysis methods. An extra force is applied to the links in the control string forcing the links in the control string to follow geodesic paths, see Figure 4-11.

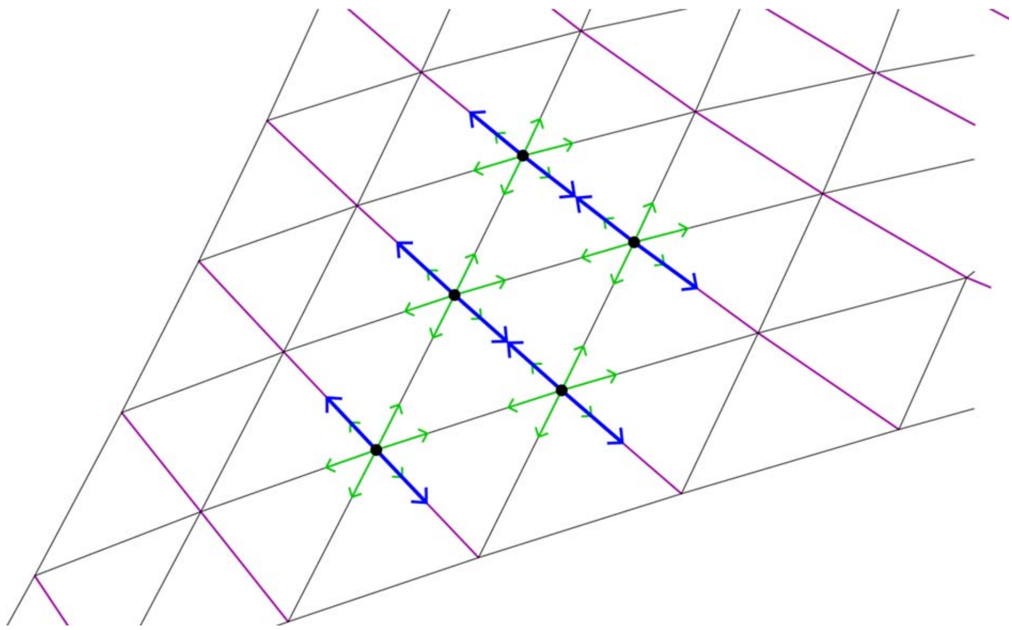


Figure 4-11 illustrating the extra forces added to the control strings.

These extra forces, added to the control strings, are only to act in the plane of the surface and must not affect the global shape or the edge cables. This is done by:

1. For each node on the control strings setting the control force $\bar{F}$ (in this thesis it is set to have a value ten times the maximum stress in the membrane).

2. Setting surface/node normal vectors $\overline{v_n}$ to the mean of adjacent surface normal vectors (a weighting factor is applied proportional to the surface area) see Figure 4-12

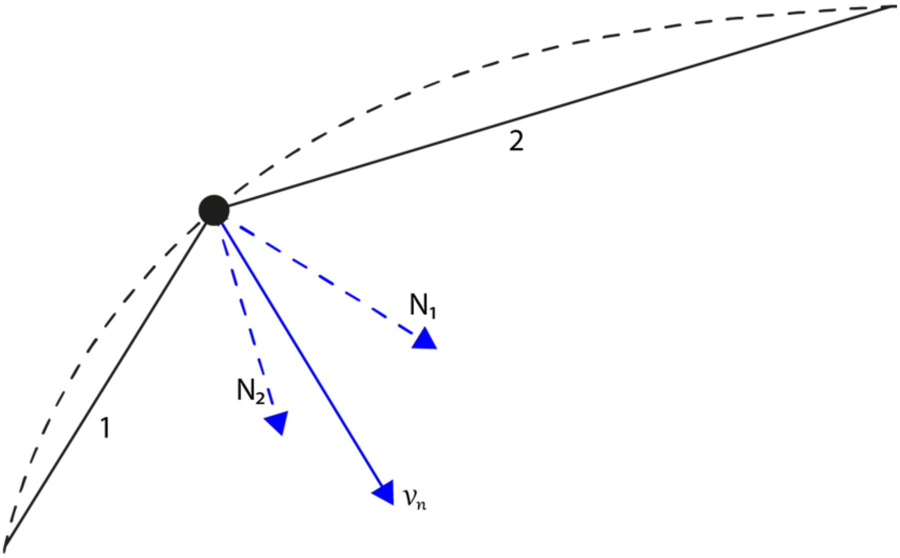


Figure 4-12 Illustrating the surface normal

3. Calculating the component of the control force normal to the surface F_n (Figure 4-13).

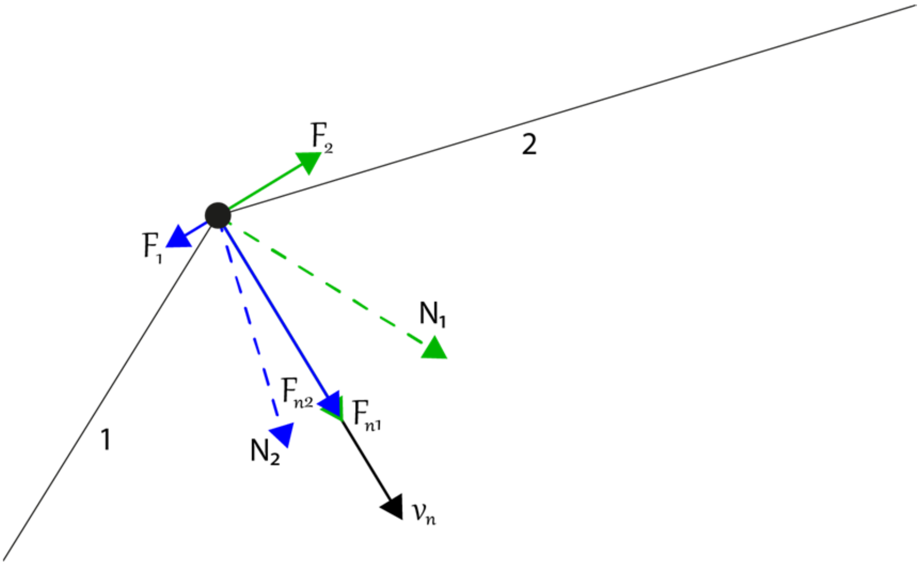


Figure 4-13 Visualising the components of the control force normal to the surface and parallel to it.

4. Subtracting the normal components from the global control forces
$\bar{F} = \bar{F} - F_n \overline{v_n}$.
5. Setting the nodal control forces of all ridge and boundary cables to be zero.
6. Adding these forces to the ("real") forces calculated for the structures, and continue with calculations of accelerations, velocities and new coordinates.

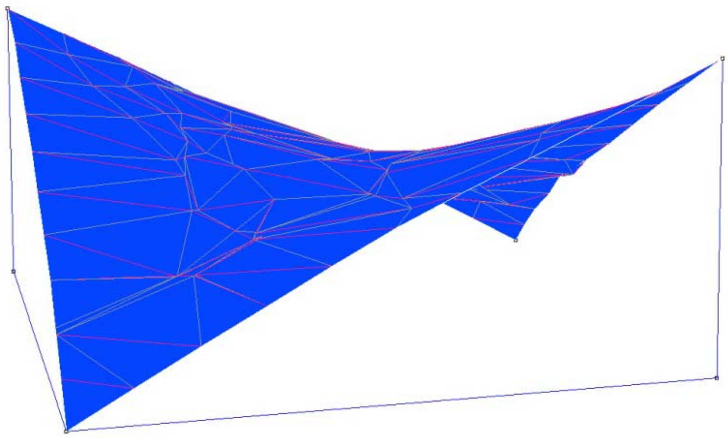


Figure 4-14 Formfinding without any control strings, resulting in a distorted mesh.

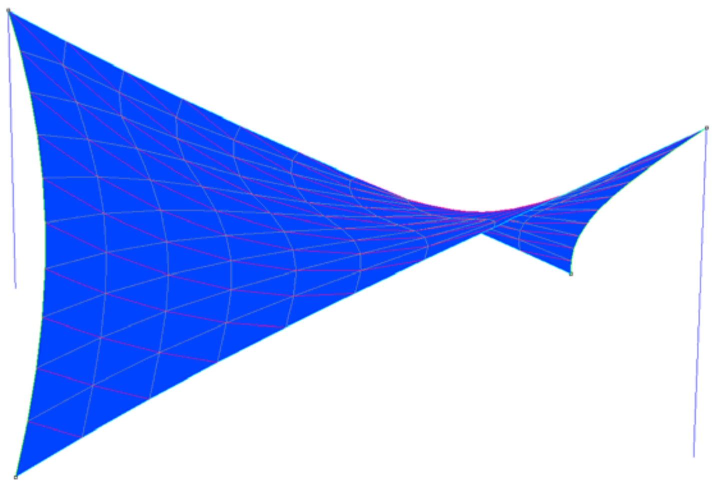


Figure 4-15 Formfinding with control strings

The control forces are only used during the formfinding process, when the membrane is assumed to have no stiffness, and are not active during the analysis

4.4 Load Analysis of Prestressed Membranes

When the optimal or desired shape of the structure has been found it is used as initial geometry for the load analysis. This means that the starting geometry for the analysis has pretension already applied to the membrane, see Figure 4-16. The applied load will then either increase or decrease the tension in the membrane.

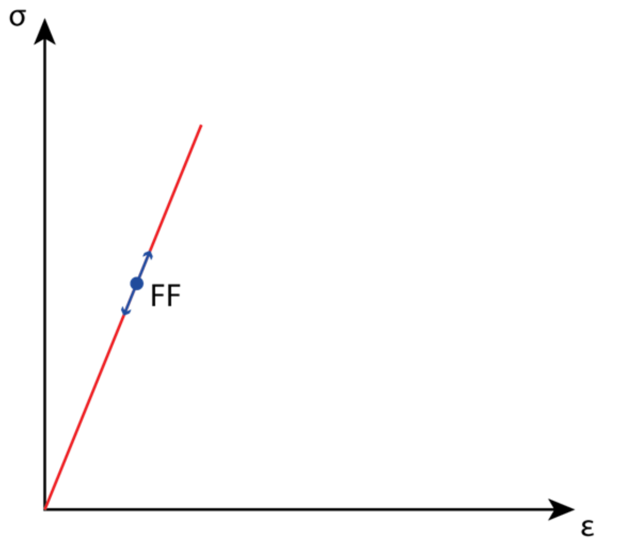


Figure 4-16 A simplified stress-strain relationship, illustrating the starting position for the load analysis. The blue dot is symbolising the initial prestress in the membrane, generating the shape. From this position the stress can increase or decrease because of the applied loads.

Shear stiffness must be introduced during load analysis. Even though principal stress will always be in the weave directions, since the relatively low shear stiffness of fabrics, the shear stiffness will prevent distortion of the mesh.

There are different ways of calculating both direct and shear stresses, in this thesis the link tensions due solely to shear stress, τ, are calculated with equation (4-4) (Barnes, 1999)

$$T_1 = \frac{\tau l_1}{2} - \frac{\tau h_1}{2\tan\alpha_2} \quad ; \qquad T_1 = -\frac{\tau l_2}{2} \quad ; \qquad T_1 = \frac{\tau l_3}{2} \tag{4-4}$$

Both direct stresses and shear stresses are related to the strains from the prestressed state as follows:

$$\varepsilon_x = \frac{l_1}{l_1^i} - 1 \quad ; \qquad \varepsilon_y = \frac{h}{h^i} - 1 \quad ; \qquad \gamma = \frac{1}{\tan\alpha_2} - \frac{1}{\tan\alpha_2^i}\left(\frac{1+\varepsilon_x}{1+\varepsilon_y}\right) \tag{4-5}$$

Where l_1^i ,h^i and α_1^i are length, height and angle in the prestressed state (before loads are applied) for each triangular element. The shear stress is then calculated as:

$$\tau = G * \gamma \tag{4-6}$$

The direct stresses are can be calculated with the following equations:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} \\ d_{21} & d_{22} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \end{Bmatrix} \tag{4-7}$$

For an isotropic 2D material:

$$d_{11} = d_{22} = \frac{E}{(1 - v^2)} \quad ; \quad d_{12} = d_{21} = v \frac{E}{(1 - v^2)} \tag{4-8}$$

For a membrane, an orthotropic material, this becomes:

$$d_{11} = \frac{E_x E_y}{E_y - (E_x {v_{yx}}^2)} \quad ; \quad d_{12} = d_{21} = \frac{E_x E_y v_{xy}}{E_y - \left(E_x {v_{xy}}^2\right)} \quad ;$$

$$d_{11} = \frac{E_y^2}{E_y - (E_x {v_{yx}}^2)} \tag{4-9}$$

Where E_x and E_y are Young's modulus in the warp and weft direction and v_{xy} and v_{yx} are Poisson's ratio in the different directions. (Lei, 2010)

In Tensyl and GSA (analysis softwares) a different equation to calculate the stress in the membrane is used: (4-10).

$$\sigma_x = \frac{E_x \varepsilon_x + E_y v_{yx} \varepsilon_y}{1 - (v_{xy} v_{yx})} \quad ; \quad \sigma_y = \frac{E_y \varepsilon_y + E_x v_{xy} \varepsilon_x}{1 - (v_{xy} v_{yx})} \tag{4-10}$$

If poisons ratio is set to zero (see 4.5.1), these two equations ((4-9) & (4-10)) will be the same (4-11), which is the case for most of the case studies in this thesis. Since this is a conservative way of calculating the stresses

$$\sigma_x = E_x \varepsilon_x \quad ; \quad \sigma_y = E_y \varepsilon_y \tag{4-11}$$

The stresses are then converted into link tensions with the same equations used for the formfinding part, equation (4-1). Since membranes does not work in compression the forces in the links, T_i, are set to zero if the calculated link tensions are negative ($T_i < 0$), indicating that the fabric is slack, instead of being stretched out.

4.5 Material Properties

For practical reasons it is common to use EA_x and EA_y instead of E_x and E_y , when doing calculations for structural fabrics this is also done in this thesis. EA will thus replace E in the equations above ((4-7),(4-10) and (4-11)). This means that stresses will be calculated and given as generalised stresses in the unit "kN/m". An example of such a stress-strain plot can be seen in Figure 4-17 (showing one loading and unloading cycle). To get the EA-values for a fabric a line is fitted through plotted graphs.

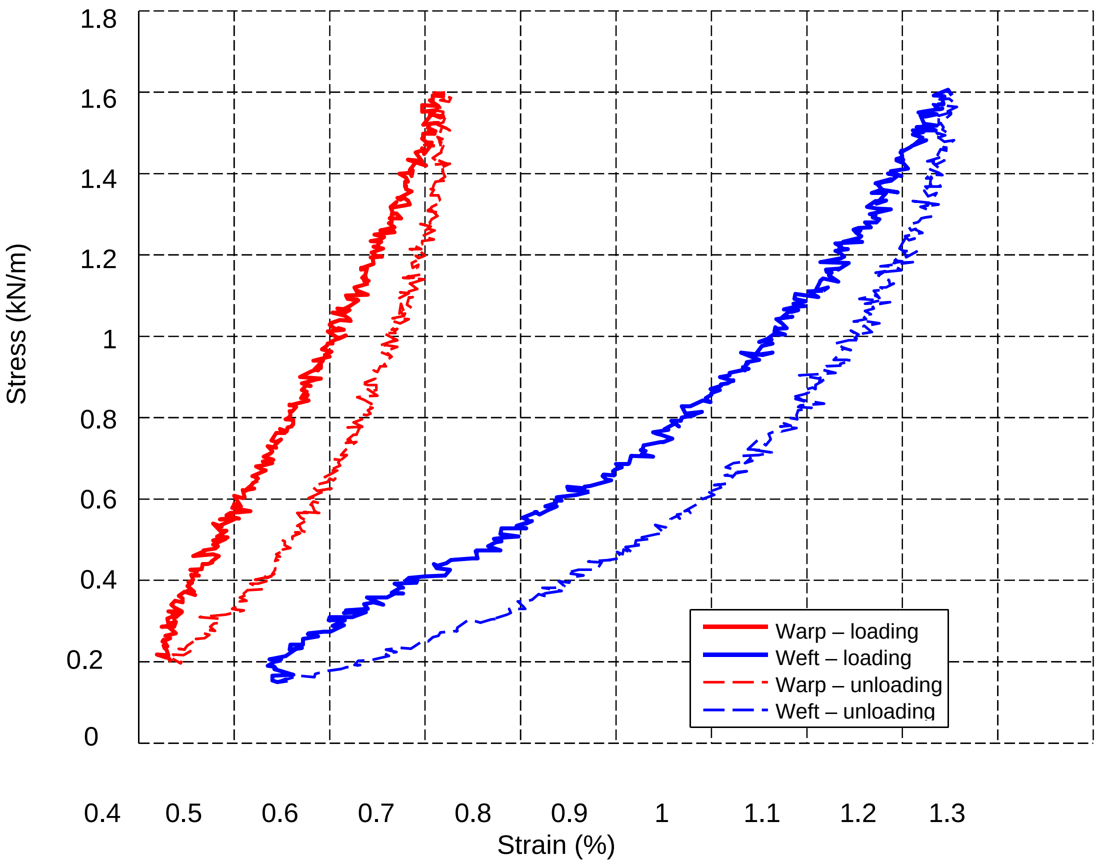


Figure 4-17 An example of a stress-strain plot for a material test of a silicone coated glass fiber fabric, done at Newcastle University (showing loading and unloading cycle 4). [8]

4.5.1 Poisson's Ratio

It is difficult to find a value for Poisson's ratio, when doing calculations for fabric structures. The way fabric is woven it is given that if the warp is stretched it is going to have an effect in the weft direction and vice versa. It is conservative, for a doubly curved surface, to put a low value for Poisson's ratio (for a flat structure it is conservative to assume a high value for Poison's ratio). Since no value has been found for it, for this thesis, and every piece of fabric is different it is set to the conservative value of zero for the carried out tests. This will also result in a faster convergence for the structure, which for a conceptual tool, is very desirable. (Paul Romain, Joanne Renold-Smith, 2012)

4.6 Convergence

When the system has reached a point where it has got low enough kinetic energy or when the residuals for the individual nodes are small enough the system is considered to have converged, even if it has not truly settled. Below a certain threshold very little will change, there will be no noticeable change in the shape nor the values. Additionally, since the model is an approximation in the first place, running the iteration for a longer time, and thereby getting the system to converge, does not necessarily mean that more accurate values are generated. Examples of plotted graphs for the residuals and energy can be found in Appendix D.

4.6.1 Speeding up Convergence

There are several ways to speed up convergence for a system, one that has been mentioned before is the damping. Choosing between viscous or kinetic damping will affect the speed, and if viscous damping is chosen, the amount of viscous damping used, on its own or in a combination with kinetic damping will also affect the speed. This will decide how far the nodes will move in one iteration, and thereby if it is moving an appropriate distance or too far, and passing its equilibrium position, or if it is taking a too small step. It goes without saying that the closer the nodes get to their equilibrium positions in one moving step the faster the convergence.

This is easy enough if it is only one node moving in between two fixed nodes, but as the system grows and each node depends on more and more moving nodes, the problem quickly becomes very complex. It is therefore not possible to find a value for the damping that is the ultimate value for all systems.

4.6.1.1 Adjusting the Time Step

By changing the time step used, for each iteration, the moving distance for the nodes can be adjusted. A good time step could be found by simply testing different values, or be calculated based on the stiffness of the links and the masses lumped to the nodes (4-12). By looping around the nodes the smallest allowable time step, for the system, can be found.

$$\Delta t = \sqrt{\frac{2m}{s}} \qquad (4\text{-}12)$$

Where s is the largest total stiffness, in any direction, for a node with mass, m.

4.6.1.2 Dynamic Masses

In structures with multiple element types that have different stiffness, the calculated time step can be suitable for some nodes but extremely small for others. This is the case for a structure containing both cables and membrane. Here the cables are relatively stiff compare to the membrane, while the masses are of the same magnitude.

Therefore the cable elements will move relatively quickly while the membrane seems to not move at all. This problem is solved by adjusting the masses instead of calculating the time step i.e. using fictitious masses (Barnes, 1999), based on the equation

$$m = \frac{\Delta t^2}{2} * s * c \qquad (4\text{-}13)$$

Where c is a constant used to modify the calculated masses.

For the cable links the linear stiffness, s, is calculated with:

$$s = EA/l_0 \qquad (4\text{-}14)$$

For the links in the membrane a geometric stiffness:

$$s = g\frac{T}{l} \qquad (4\text{-}15)$$

is used, where T and l are the tension and the length of the link, and g is a constant, used to "tune" in the stiffness value.

The stiffness for each node is then:

$$S_i = \sum\left(\frac{EA}{l_0} + g\frac{T}{l}\right) \qquad (4\text{-}16)$$

Where S_i is the total stiffness for node, i.

Since the stiffness and the fictional mass do not affect the analysis values these can be modified, by the constants c and g, to get the optimal speed for the nodes.

Equation (4-16) is used in both the formfinding part and the analysis part. However during the analysis part an additional stiffness is added on top of this, to account for the elastic stiffness of the membrane that is included in the stress calculations (equation (4-7). In analogy with transforming the membrane stresses into link forces (see (4-1)) the extra link stiffness is calculated by converting the membrane stiffness as follows:

$$s^1 = \frac{h}{2}\left(EA_x - EA_y\right) + \frac{EA_y l_1}{2\tan\alpha_1} \;;$$

$$s^2 = \frac{EA_y l_2}{2\tan\alpha_2} \;; \qquad s^3 = \frac{EA_y l_3}{2\tan\alpha_3} \qquad (4\text{-}17)$$

By adding this extra stiffness a faster convergence was achieved in the tests of the program.

5 Solution procedure

The model of the structure, in the software, consists of a system of nodes, connected to each other by a set of links (see Figure 5-1). These links can be of different kind, for the part of the tool developed in this study they are either bars, or cables. A bar does not have any structural properties of its own, but can store information about force and stiffness as well as geometrical properties. Cables have all the properties that a bar has, but they have a stiffness of their own which is depending on what material they are, when the nodes move the stiffness will result in a force in the cable.

In between the links are membrane panels, this class of object inherits all the information contained in the parent class “face”, which has all necessary geometrical information. On top of this the membrane object has information about the stresses and properties, such as stiffness, of the membrane. It is also containing methods for calculations specific for membranes.

In Figure 5-1 the membranes are represented by the grey triangles, both the dashed blue lines and the green lines are representing links. In this case all links are of type bar i.e. no cable elements are drawn in this figure. The green lines are bars in the control strings which also mean that they are in the warp direction.

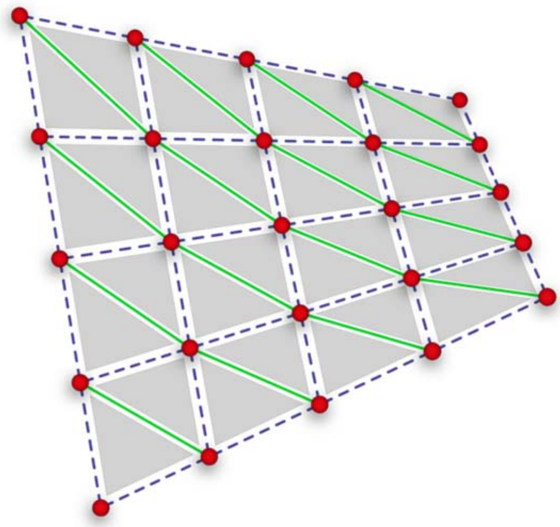


Figure 5-1 illustrating how the structure is built up in the analysis tool.

The algorithm is based on the principle of transferring all information to the nodes (see Figure 5-2). This is done by first calculating the stresses in the membrane, or in the case of formfinding dictating the stresses, and converting them to tensions in the edges of each panel (4-1). The calculated force is applied on the links of the system. If a link is a cable the forces in this element is calculated and added to the forces from the membrane. Then the forces are transferred on to the nodes. From here the acceleration, velocities and finally displacements, of the nodes, can be calculated, as explained in chapter 3. In the next iteration the deformations in the geometry of the membrane panels and the cables will generate new forces.

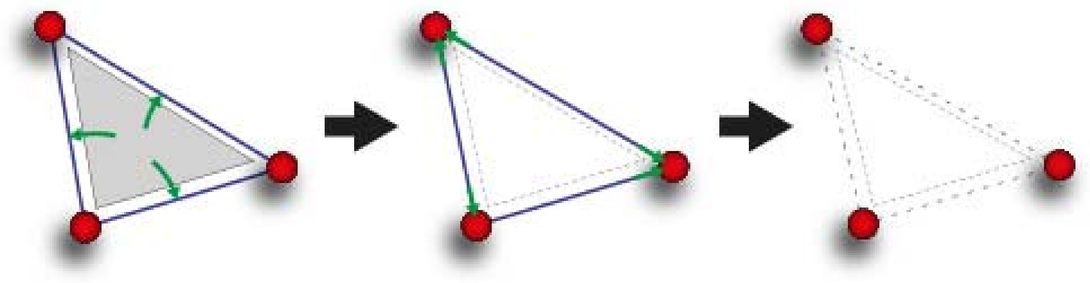


Figure 5-2 Illustrating the algorithm. Step 1: the tension from the membrane is added on to the links, step 2: the forces in the links is added to the nodes. Then the nodes have all the information needed to calculate their new positions.

5.1 Solution algorithm

The following algorithm is devised for the formfinding part of the tool:

Initialisation of analysis

- Draw the base geometry in Rhinoceros.
  - surfaces representing membranes
  - curves representing cables or constraints
  - points representing pinned constraints
- Mesh and sort geometry.
- Set material properties.

Run loop

- Set all nodal and link forces to zero
- Calculate the stiffness for each link (for the stress controlled membrane this value will change as the geometry evolves, see (4-16)).
- Calculate forces in control strings (links that coincide with the warp direction) (see section 4.3.2).
- Calculate dynamic masses (4-13).
- Transfer the assigned warp and weft prestress to the links (4-1).
- Calculate forces in the cables (elastic).
- Transfer the forces from the links onto the nodes.
- Calculate the resulting accelerations, velocities and displacements (3-3).
- Update the locations for the nodes

For the analysis part the algorithm is:

Initialisation of analysis

- Change all links to be elastically controlled (with the formfound geometry as the initial one)

Run loop

- Set all nodal and link forces to zero
- Apply loads (gravity and any other load given by the user).
- Calculate the stiffness for each link (For membrane both elastic and geometric stiffness, see (4-17)).
- Calculate dynamic masses (4-13).
- Calculate the total membrane stresses (pretension plus direct and shear stresses) (4-7) (4-6)
- Convert membrane stresses to link forces (4-1).
- Calculate forces in the cables (elastic).
- Transfer the forces from the links onto the nodes.
- Calculate the resulting accelerations, velocities and displacements (3-3).
- Update the locations for the nodes

Pictures of the user interface, for the tool, can be found in in the user story in Appendix G

5.2 Structure of the Software

The tables below are meant to give an overview for how the software is structured and how it works. In general it is the last subclass in the hierarchies that has been developed in this thesis, and not much changes have been made in the classes handling the geometrical properties of the structure. An exception from this is in the classes handling the information flow (see Table 5-4), where all classes had to be extended.

ELEMENTS	
Vertex	Geometrical information (position, index, neighbours...)
└ Node	abstract class
└ NodeRelax	Adds structural properties (forces, acceleration, velocities...)
Link	Geometrical information (length, index, end points, neighbours...)
└ Bar	abstract class
└ BarRelax	Adds structural properties (forces, stiffness...)
└ CableRelax	Adds properties specific for cables (material, no compression strength...)
Face	Geometrical information (position, index, area, neighbours...)
└ Panel	abstract class
└ PanelRelax	
└ MembraneRelax	Adds properties specific for membranes (warp and weft stresses, equations to translate a continuum onto the triangular mesh)

Table 5-1 Explaining the structure for the geometrical and structural elements in the program. The different layers are illustrating inheritance.

CALCULATIONS	
SmartMesh	contains and handles geometrical information and objects
└ SmartStructure	abstract class
└ StructureRelax	contains and handles structural information and objects

Table 5-2 Explaining the structure for the calculation objects in the software. The different layers are illustrating inheritance.

MESHING	
Grid	
└GridBi	creates a bidirectional grid
└GridMembrane	Creates a triangular grid, where the geometries are organized/sorted to work with the analysis of membranes

Table 5-3 Explaining the structure for the gridding in the software. The different layers are illustrating inheritance.

Information flow / Calling Stack	
dlgRelax	Gets information from the user, which is saved in an object, O1, of "RelaxParameters"
└ Relax	Acts as a middle hand, gets the information from dlgRelax, stored in O1, and, at an appropriate time, passes the information to the "RelaxParameter" object, O2, used in the calculations
└ AnalysisEngine	Is the hub of the code; this is where the main run loop is placed. It is using information received from Relax and O2, and passing it to equations in StructureRelax (called in the run loop).
└ StructureRelax	Contains methods calling the equations for the different structural objects.

Table 5-4 Explaining the information flow, from the user to the analysis. The different layers are illustrating how the information and calls are passed on between the classes.

6 Bench marks and Case Studies

6.1 Planar rectangle with fixed edges

To verify that the program is generating reasonable result a simple planar rectangle membrane, with applied loads, was tested, see Figure 6-1 and Figure 6-2. The result could on this shape be verified with simple hand calculations.

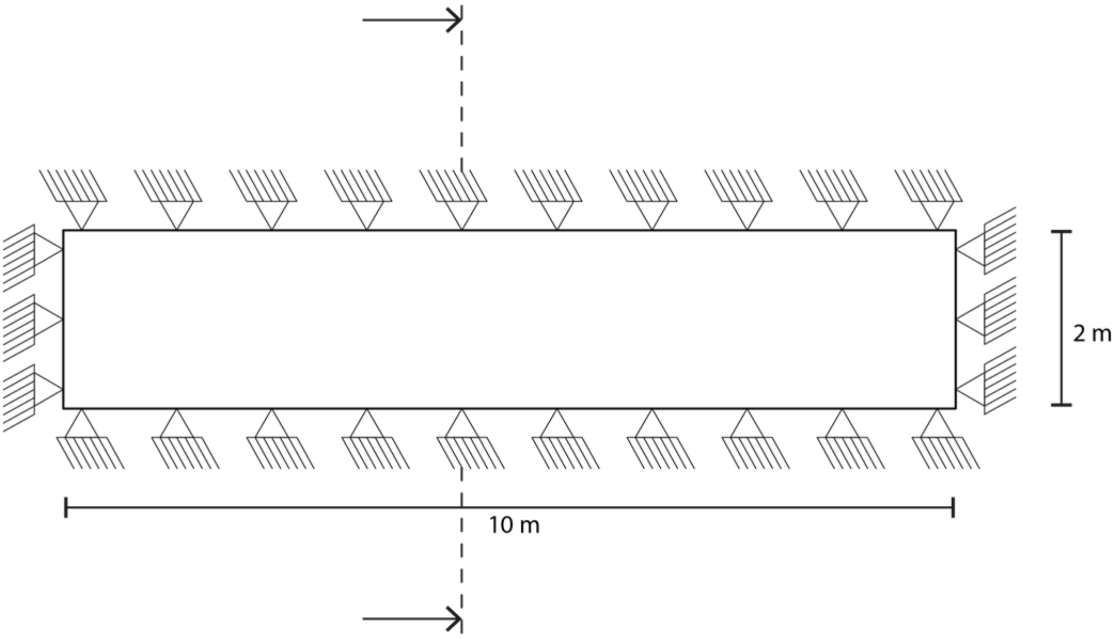


Figure 6-1 The analysed membrane patch.

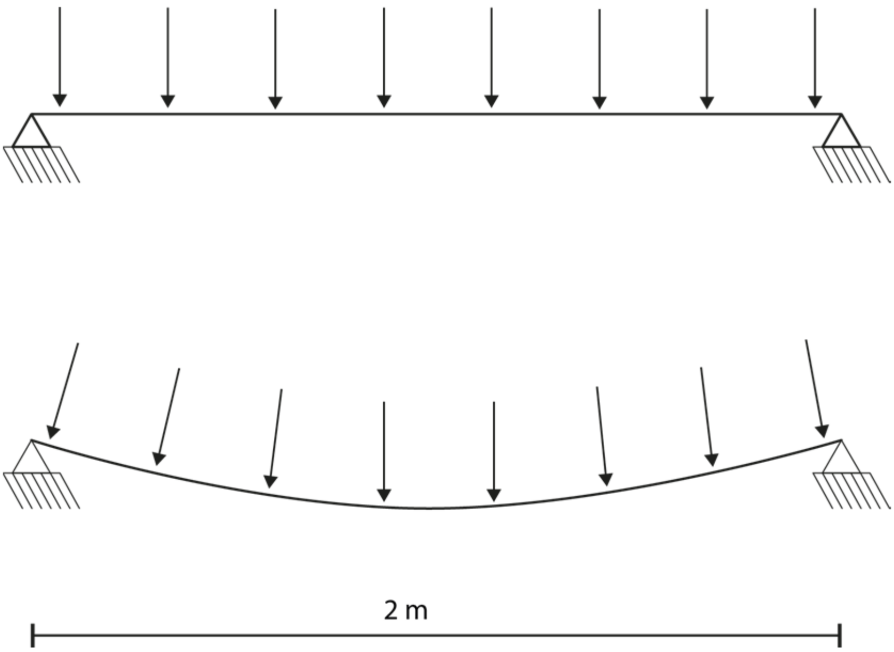


Figure 6-2 A cross section of the analysed membrane, in its original state and in the deformed stage

It could be assumed that the middle of the structure is not affected by the rear boundaries, and it would thereby behave as cable with applied pressure (6-1), see Figure 6-3.

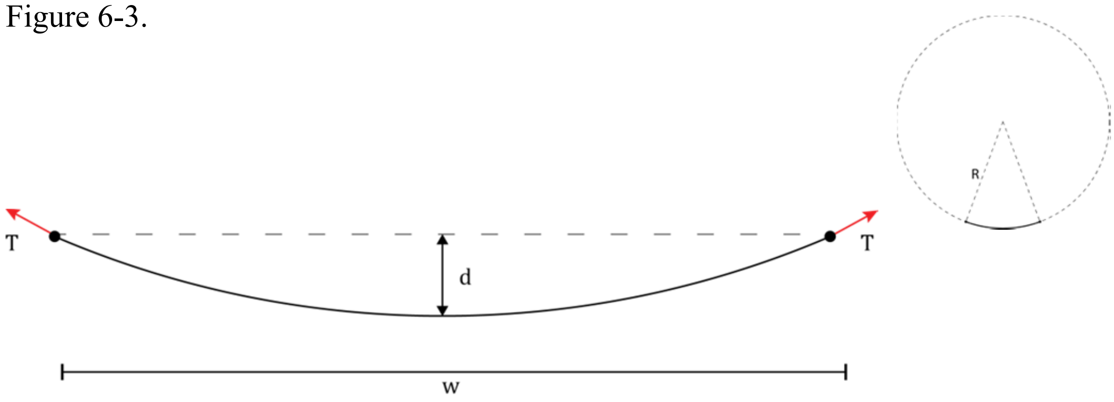


Figure 6-3 Hanging cable with applied pressure load.

$$T = PR$$

$$R = \frac{d}{2} + \frac{w^2}{8d} \qquad (6\text{-}1)$$

Where T is the tension in the cable, P is the pressure applied to the cable (or in this case the membrane) and R is the radius for the circle segment.

For the analysis, in SMART Form, the patch was meshed with a 10x30 grid, see Figure 6-4. The stress pattern in the figure confirms this assumption that the middle part is not affected by the rear boundaries.

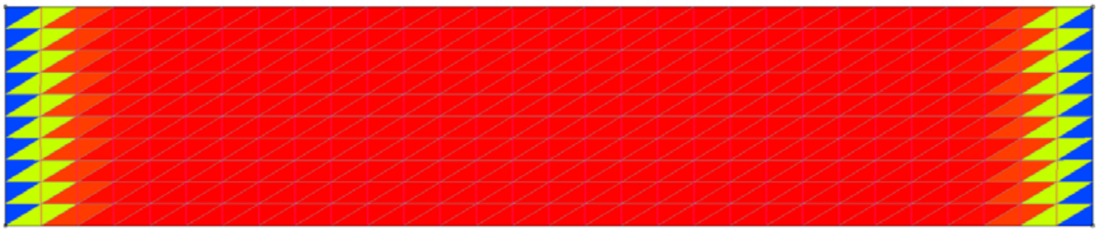


Figure 6-4 Top view of the rectangular structure used to test the calculations of the program, against hand calculations (with the colours representing the warp stress of the membrane).

6.1.1 Warp Stress

The dip in the middle of the structure is 0.104 m, which results in a radius of 4.86 m., the pretension is set to 0.1 kN/m^2 and the applied pressure is 1 kN/m2. Using equation (6-1) the resulting tension in the structure is calculated to roughly:

$T = 1 * 4.86 + 0.1 = 4.96$ kN/m^2

This is a simplification of the problem, since the length of the membrane/”cable” is the length with prestress applied. Therefore the calculated stress should be slightly higher than what it is in reality. But since the prestress is low in comparison with the applied pressure (ten times smaller), the result can be assumed to be fairly accurate.

The program calculates the stress in the membrane to be 4.91 kN/m^2 (using a 10x30 mesh, as shown in Figure 6-4) which seem to be a reasonable result (for more values see Appendix B).

6.1.2 Weft Stress

Analysing the same structure but with the warp direction rotated 90 degrees, making the weft span 2 m, results in a dip of 0.124 m. Calculating this by hand gives an expected stress of roughly 4.20 kN/m^2. SMART Form generates a value of 4.15 kN/m^2 which seem to be an ok value. In Appendix B further results are validated.

6.2 Hypar

An often seen shape in fabric structures is the hypar (hyperbolically shaped membrane). The geometry of this structure is relatively simple, but it is still difficult to analyse it without a computer, which is the reason for why it was chosen for this case study. The results from the analysis are compared with results from Tensyl, and a convergence test has been made, trying different mesh densities. Additionally the structure has been analysed with a different warp direction.

The geometry for the analysed hypar can be seen in Figure 6-5. For the first part a 16x16 mesh is used. The chosen fabric is a PVC of type 2 (see Table 6-1) and galvanised spiral strand steel cables, with a diameter of 10mm (see Table 6-2), are used as edge cables, for more material data see Appendix C. A prestress of 2 kN/m^2 is applied to the membrane, for both warp and weft, the edge cables are not prestressed.

Weight	EA		G	Unfactored Tensile Strength	
	Warp	Weft		Warp	Weft
1,05 kg/m^2	670 kN/m	400 kN/m	10 kN/m	84 kN/m	80 kN/m

Table 6-1 Material properties for the membrane used.

Nominal Strand Diameter	Characteristic Breaking Load	Limit Tension	Metallic Cross Section	Weight
10,1 mm	93 kN	56 kN	60 mm^2	0,5 kg/m

Table 6-2 Material properties for the cable used.

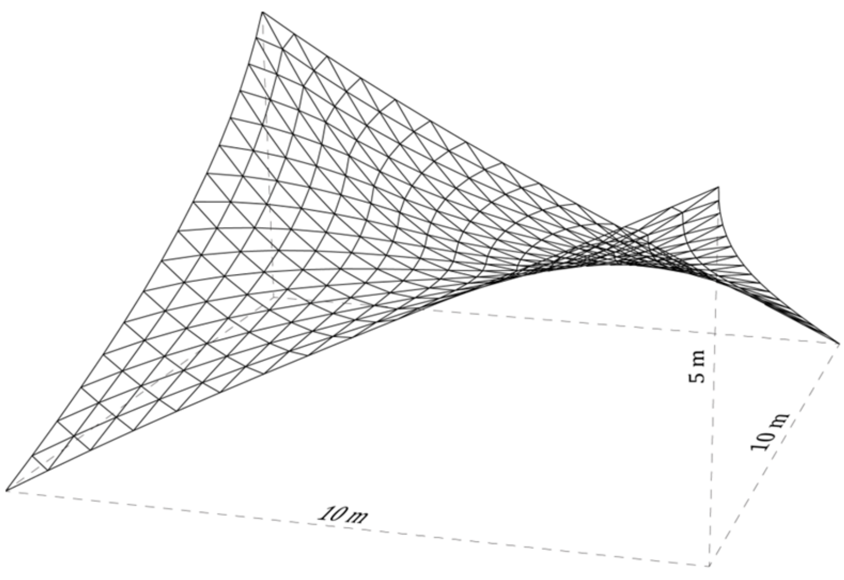


Figure 6-5 Geometry of the analysed hypar, showing the form found shape.

In Figure 6-6 the three main steps of the analysis can be seen, from the starting geometry to the final stage, where the form found shape is analysed with applied loads. For the form finding part, of the analysis, the generated shape is believed to be accurate, by judging the shape apperance, and there is no need for comparing results with with any other software. The load anlysis will also validate the formfound shape in the final stage of the analysis.

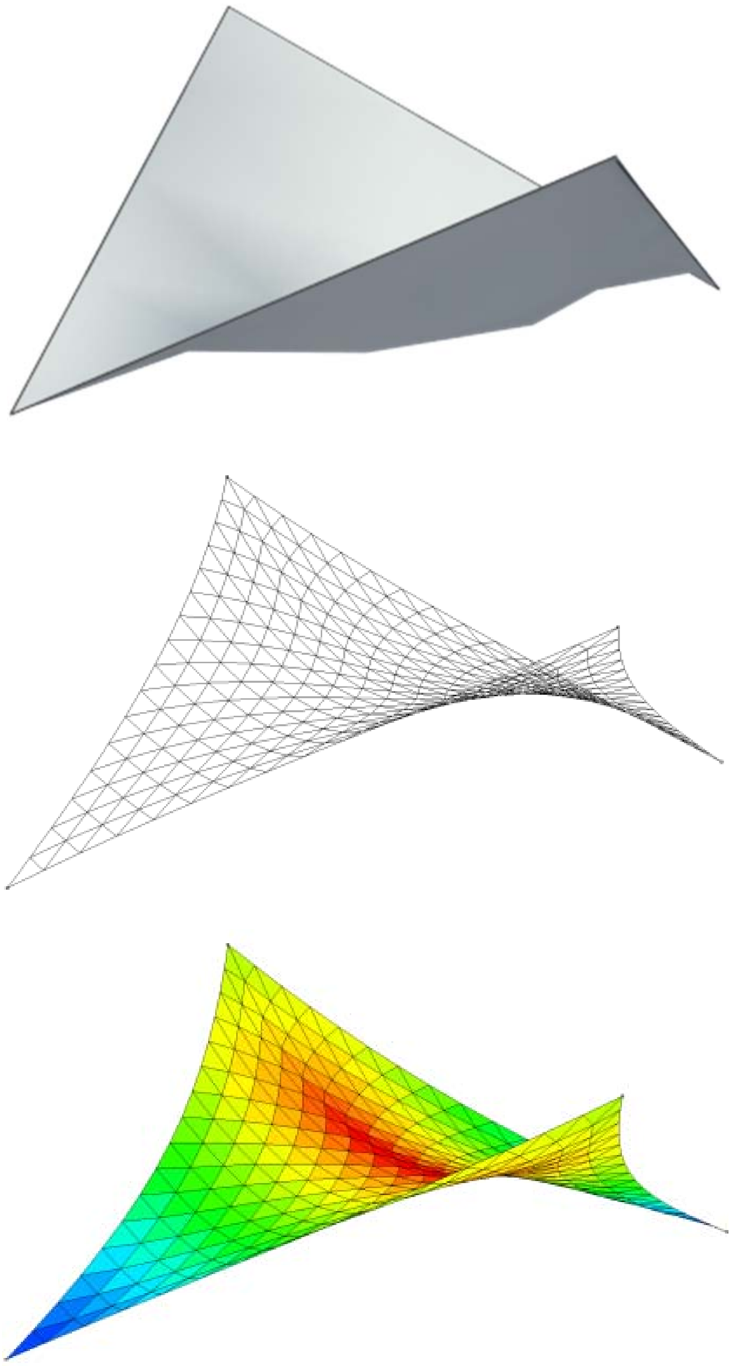


Figure 6-6. From the top: starting geometry, form found shape, and the structure analysed with 1 kN pressure load applied (the colours are visualizing the warp stress)

6.2.1 Comparing results with analysis done in Tensyl

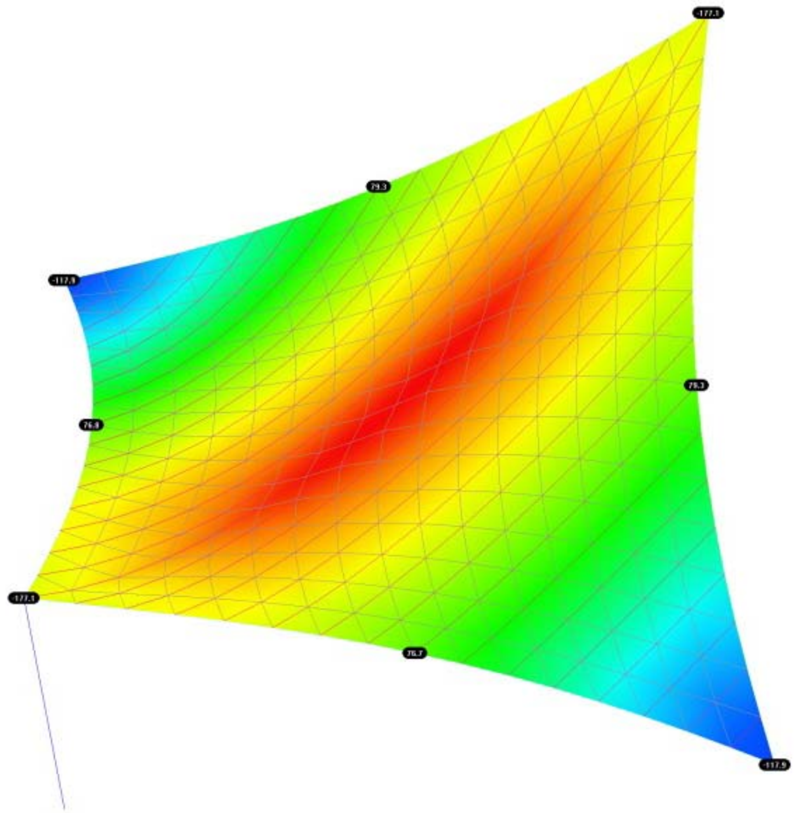


Figure 6-7. A 3D view of the hypar. The warp stress, resulting from an applied pressure of 1 kN, is visualized, with a smooth transition between the coloured values.

As stated above the analysis is made for the hypar in Figure 6-5, having a pretension of 2 kN/m^2, for the membrane. A pressure load, of 1 kN/m2, is applied to the structure, from above.

The results of this analysis can be seen in Figure 6-7 to Figure 6-13. The stress patterns generated in the developed tool and in Tensyl are very similar, looking at both the values and the stress distribution.

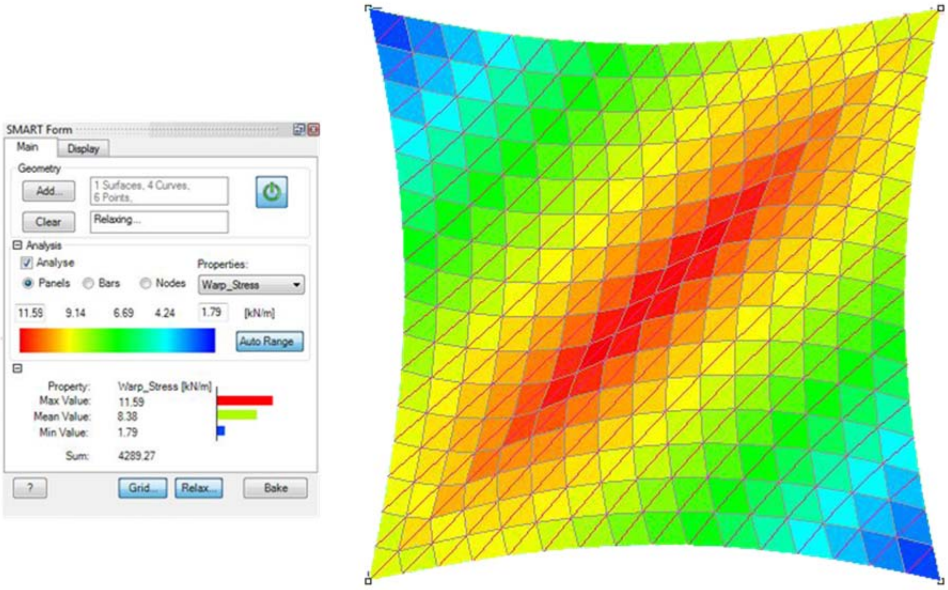


Figure 6-8 A top view of the analysed hypar, visualising the warp stress (also showing the user interface of SMART Form). The range of the values for the warp stress can be seen in the colour bar (ranging from 11.59 kN/m^2, in the middle to 1.75 kN/m^2 in the blue corners)

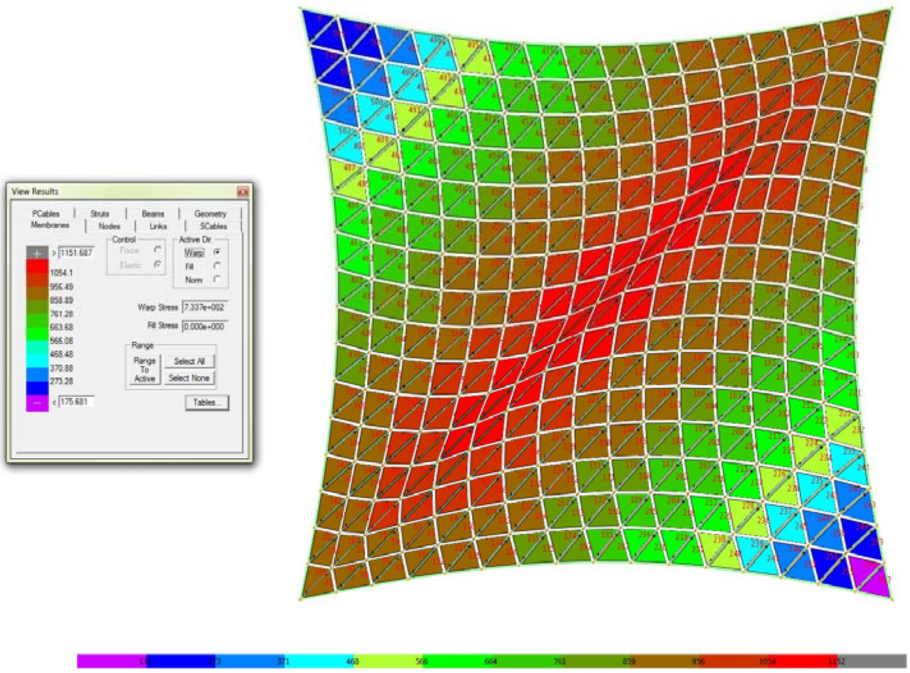


Figure 6-9 A top view of the hypar analysed in Tensyl, visualising the warp stress. Values are ranging from 11.52 kN/m^2 in the middle to 1.76 kN/m^2 for the purple section (The values in Tensyl is given in kg/m^2).

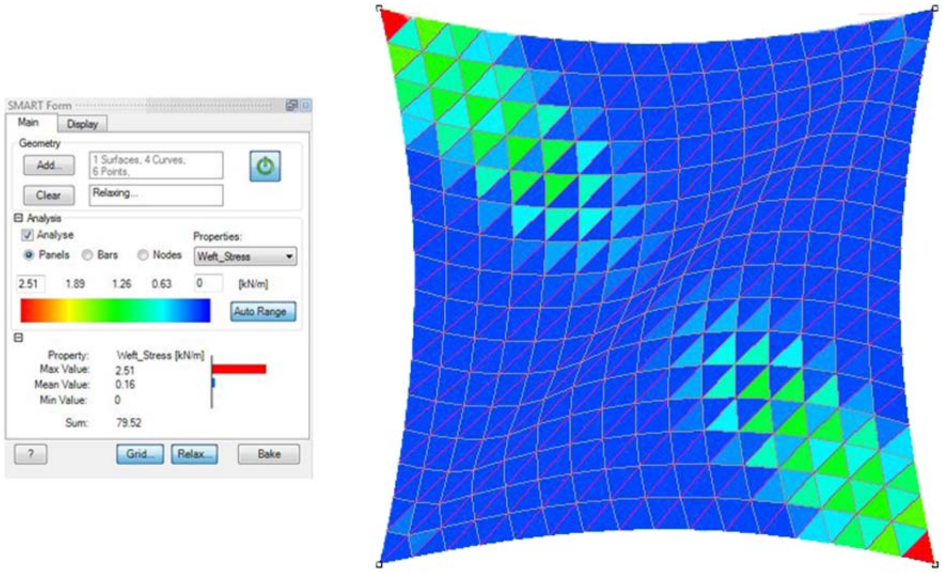


Figure 6-10 A top view of the analysed hypar, visualising the weft stress (also showing the user interface of SMART Form). The range of the values for the stress can be seen in the colour bar (with 2.51 kN/m^2 in the red corners, and the dark blue area being slack)

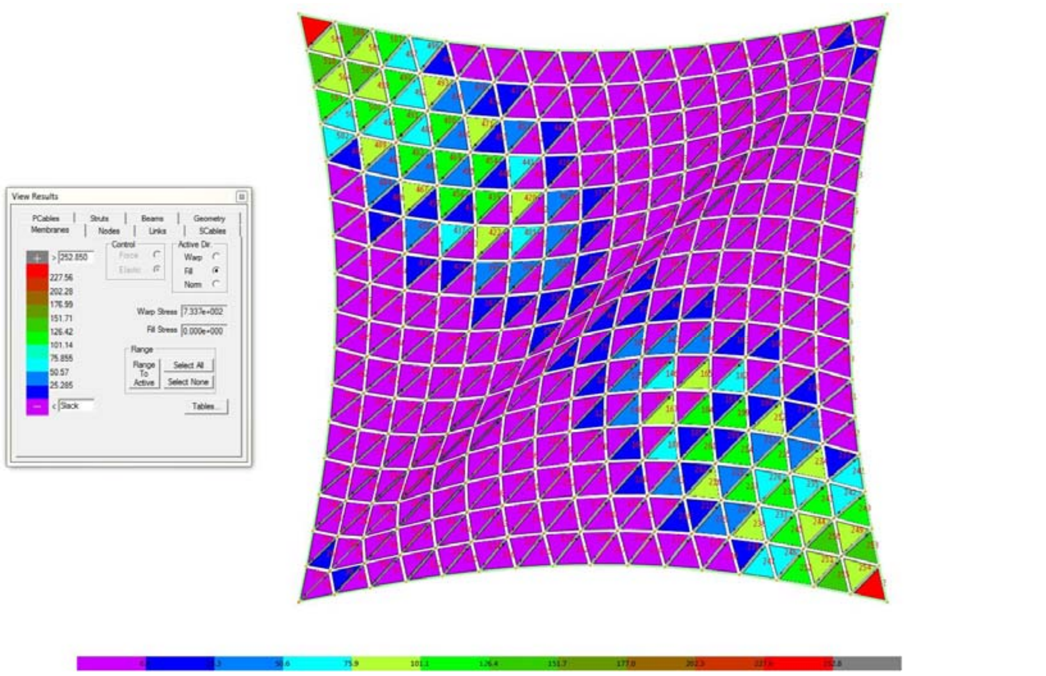


Figure 6-11 A top view of the hypar analysed in Tensyl, visualising the weft stress. Values are ranging from 2.52 kN/m^2 in the red corners to the purple section being slack (The values in Tensyl is given in kg/m^2).

The check board like pattern, for the warp stress, is a result of how the structure is gridded, and the method used to represent the membrane (constant strain triangles) this of course disappears when a smooth colour pattern is used (Figure 6-12).

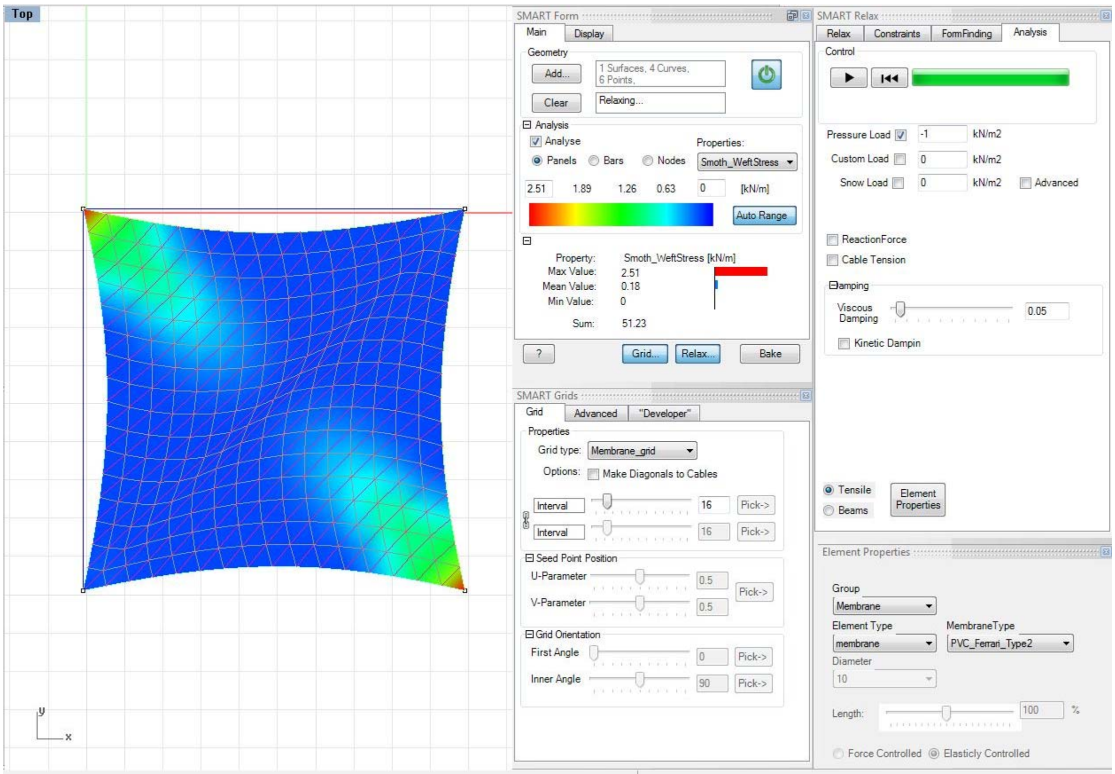


Figure 6-12 Visualising the same weft stress as Figure 6-10, but with smooth colours.

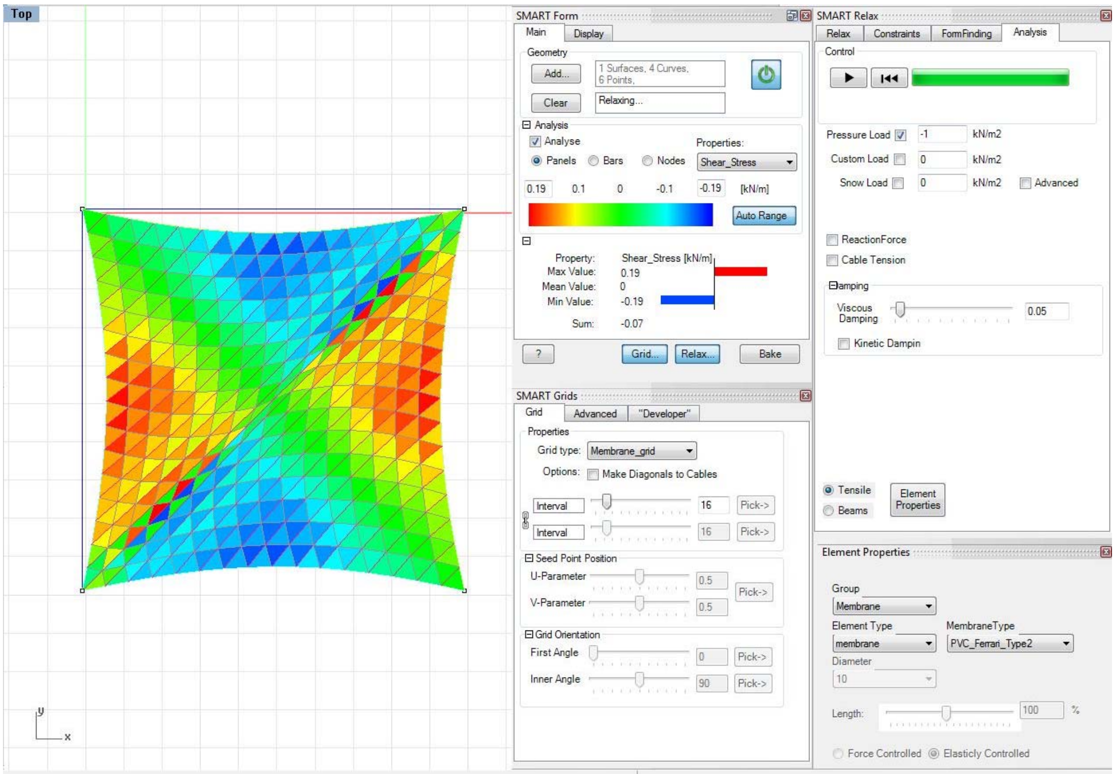


Figure 6-13 Visualising the shear stress for the analysed hypar.

It can be noted that, as expected, the shear stress for the membrane, Figure 6-13, is comparably low, and does therefore not affect the stress levels greatly, but it does still keep the mesh in place.

6.2.2 Convergence test

To gain confidence in the calculated results of the developed tool a convergence test was carried out. The same hypar as above (section 0) was used but using a varying number of intervals for the mesh. In the test the displacement of the mid node was observed, as well as the stress in the warp and weft directions. From the results in Figure 6-14 and Figure 6-15 it can be seen good values are obtained already at a 16x16 mesh and that a 10x10 mesh in most cases would be sufficient.

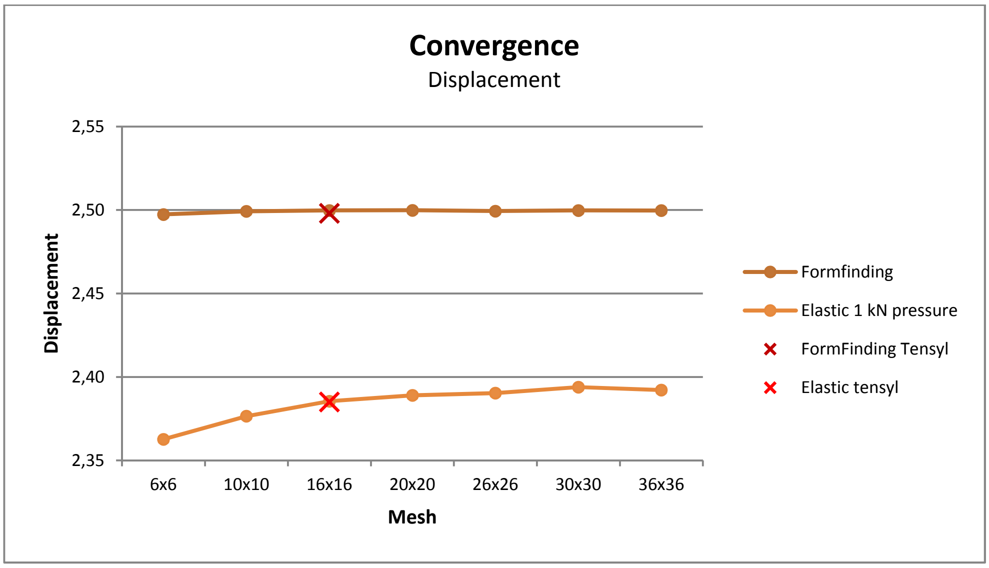


Figure 6-14 The displacement for the mid node during both the formfinding and the load analysis, for the same hypar, but with different mesh densities. The "X:es" marks the results for the same analysis in Tensyl.

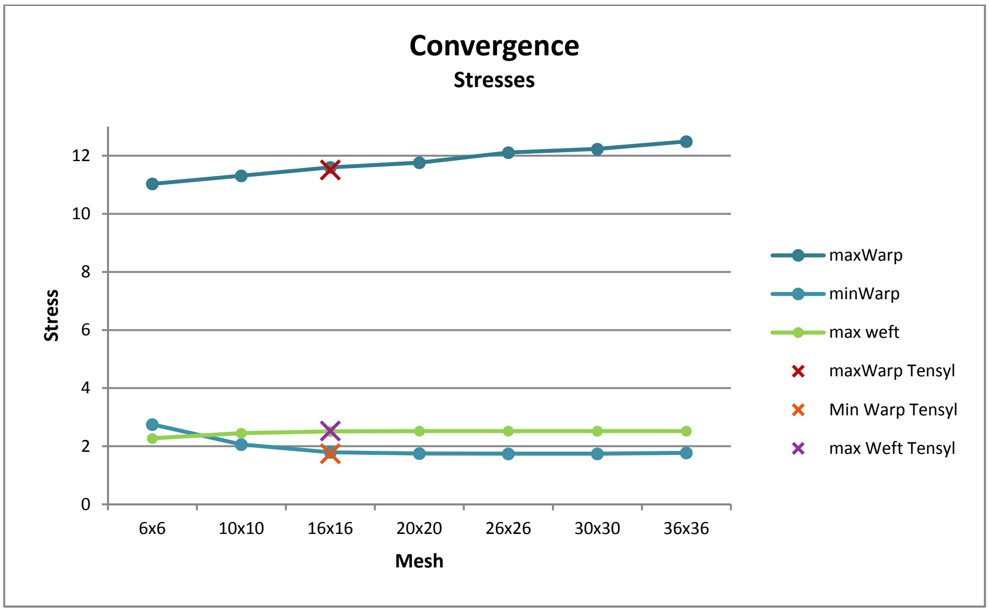


Figure 6-15 The maximum and minimum warp stress and maximum weft stress (the minimum weft stress is equal to zero for all cases) for different mesh densities for the same hypar. The "X:es" marks the results for the same analysis in Tensyl.

6.2.3 Changing the weft direction

The warp of a fabric is usually aligned with the principal stress direction, which for a hypar is diagonally across. If the weave, for the analysed hypar, instead is rotated to be aligned with the edges, the formfound shape would be the same but the ability for the structure to carry load, without deforming, would be drastically reduced, as can be seen in Figure 6-16 and Figure 6-17.

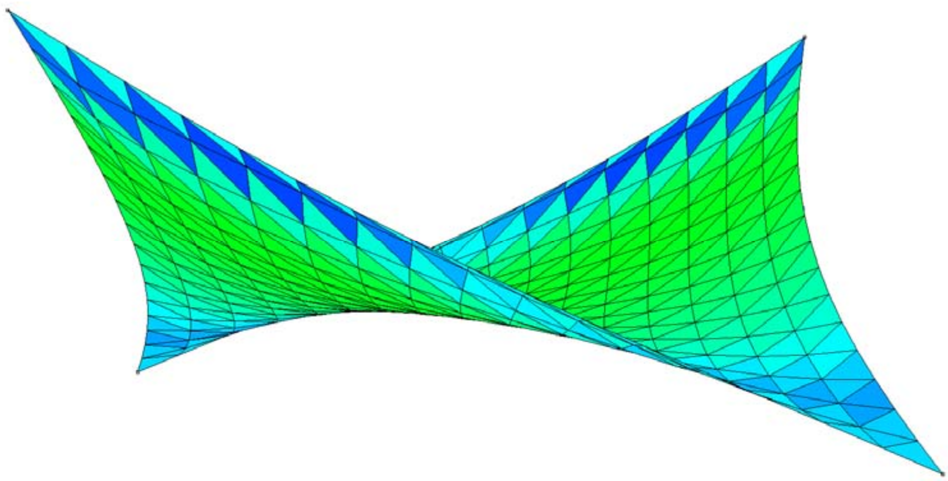


Figure 6-16 Showing deflection for the analysed hypar, with the warp direction aligned with the edges.

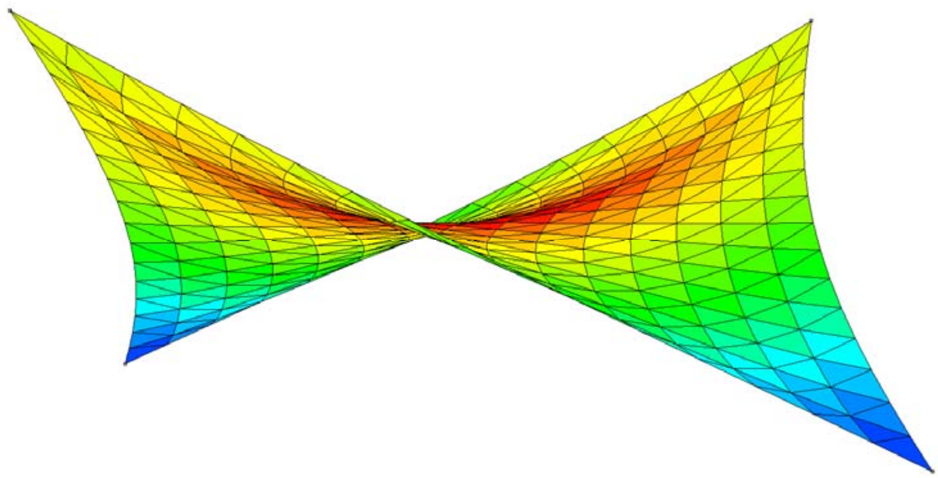


Figure 6-17 Showing the deflection for the analysed hypar, with the warp direction going diagonally across.

In Figure 6-18 and Figure 6-19 the stress patterns for the structure is visualised. In the warp direction the stress is about half of what it was for the hypar analysed in section 0, while the weft stress is higher and more evenly distributed across the surface. To illustrate the difference in the stress distribution, compared to the hypar with the weave following the principal stress, Figure 6-20 and Figure 6-21 are displaying the stresses in the colour scale used in section 0.

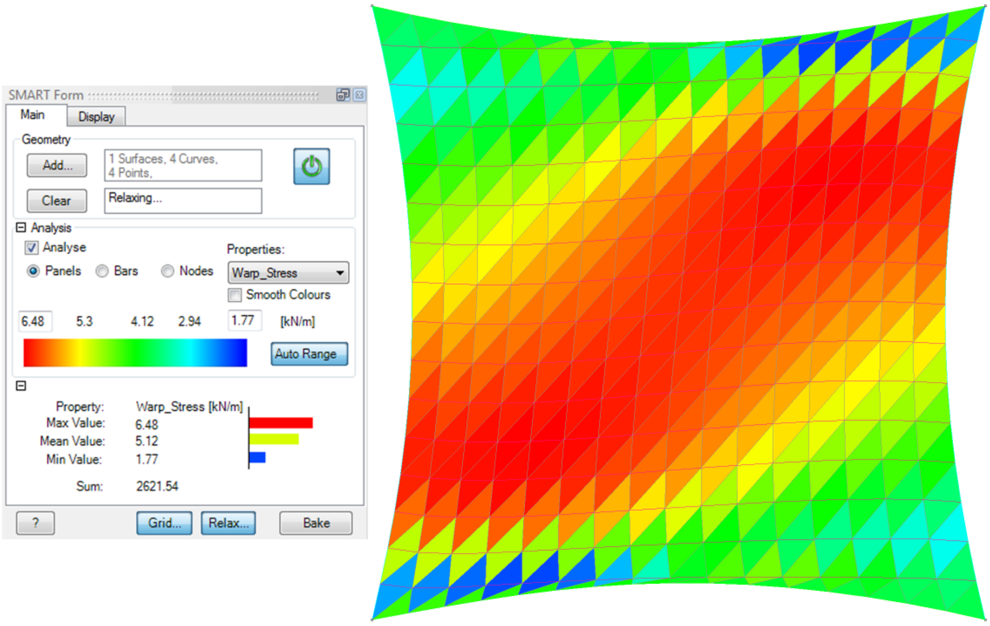


Figure 6-18 Warp stress for a hypar with the warp direction running vertically in the figure.

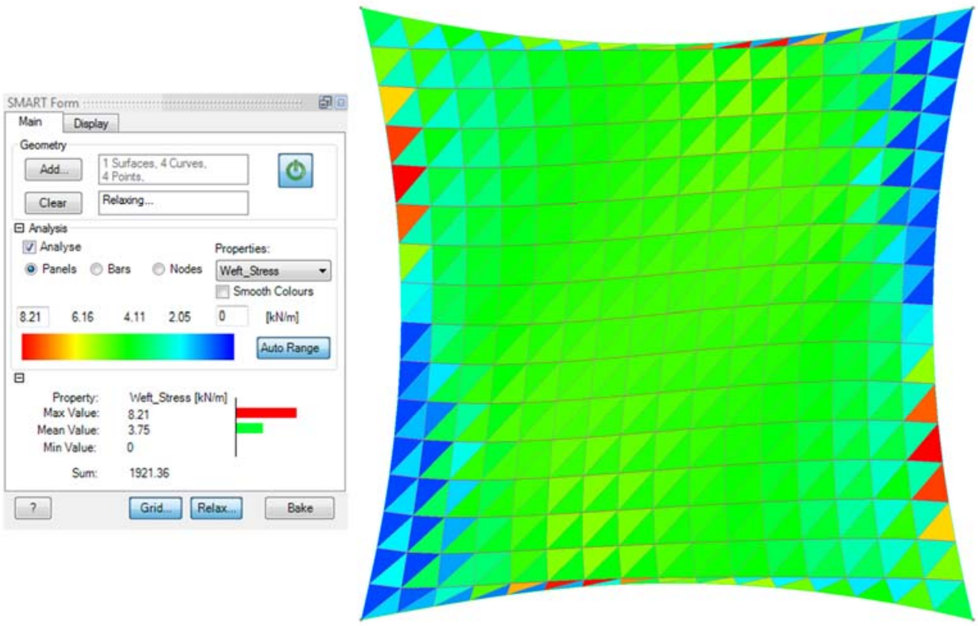


Figure 6-19 Weft stress the same hypar as in Figure 6-18

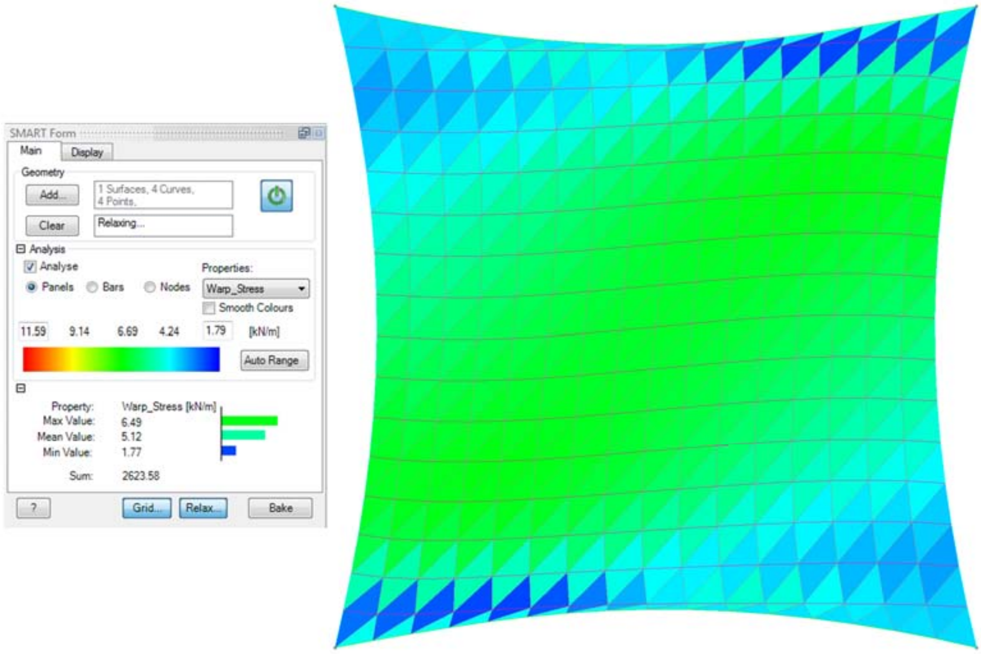


Figure 6-20 Showing the warp stress for the hypar with the same colour scale as in Figure 6-8

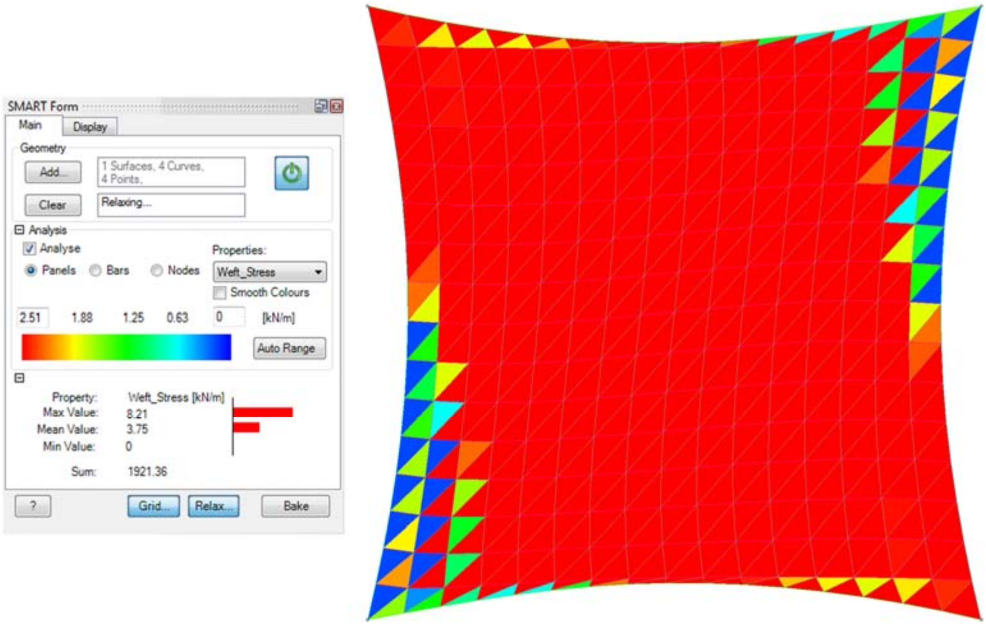


Figure 6-21 Showing the weft stress for the hypar with the same colour scale as in Figure 6-10

In the figures it is clear that the principal stress direction does not change, with changing the weave direction. Especially for the warp stress it is clear that the highest stress occurs in the diagonal between the high points of the hypar.

Looking at the displacement it is also clear why the fabric is not usually oriented like this. The structure is a lot more deformed with this orientation of the weave, even though both examples are using the same materials, have the same prestress (2 kN/m) applied to the membrane and the same applied pressure (1 kN/m^2)

6.2.4 Experimenting with the shape

In this section the shape of the hypar is modified in different ways. The warp is running diagonally across, between the high points of the structure in all examples.

The first example, Figure 6-22, is showing how the shape is affected if the edge cables are changed.

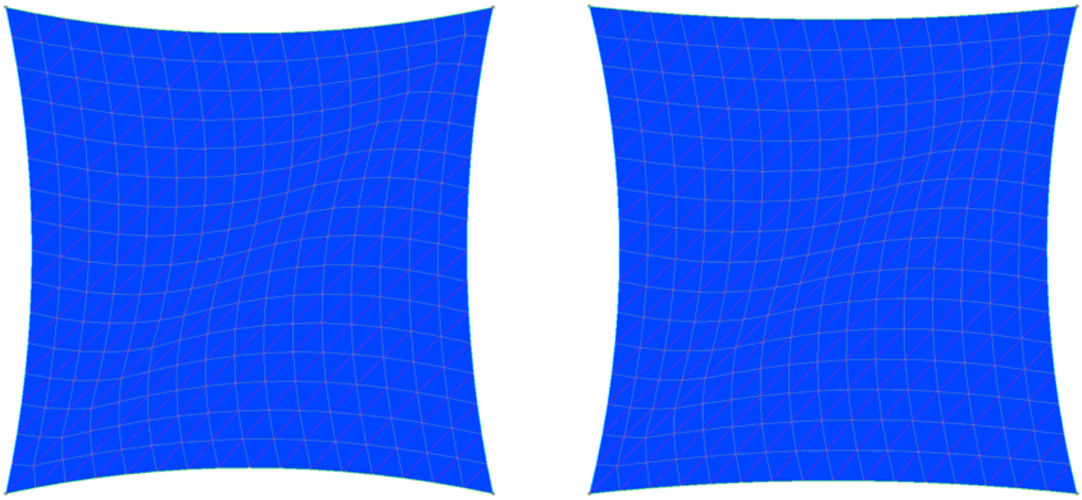


Figure 6-22 Showing a hypar, from the top. Both of the hypars are formfound with edge cables of galvanised steel. The left one has got cables with a diameter of 10 mm (all around), and the one on the right is using cables of different diameters, 8 mm and 28 mm.

For the second part the prestress ratio is changed starting with a uniform prestress (2 kN/m in both warp and weft), as shown in Figure 6-23.

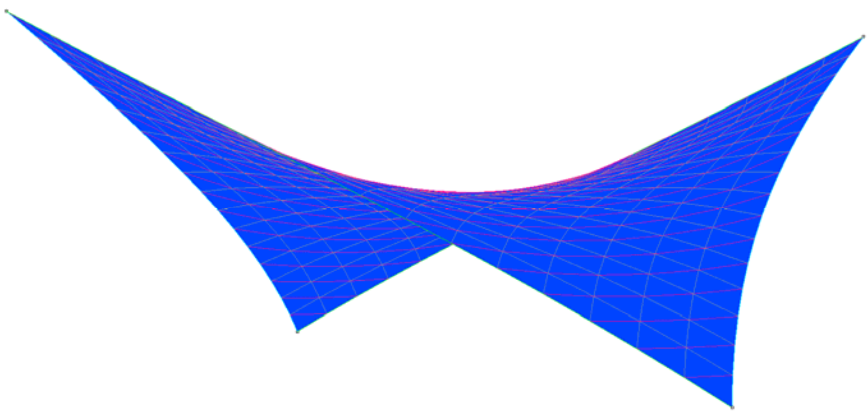


Figure 6-23 Hypar with uniform prestress.

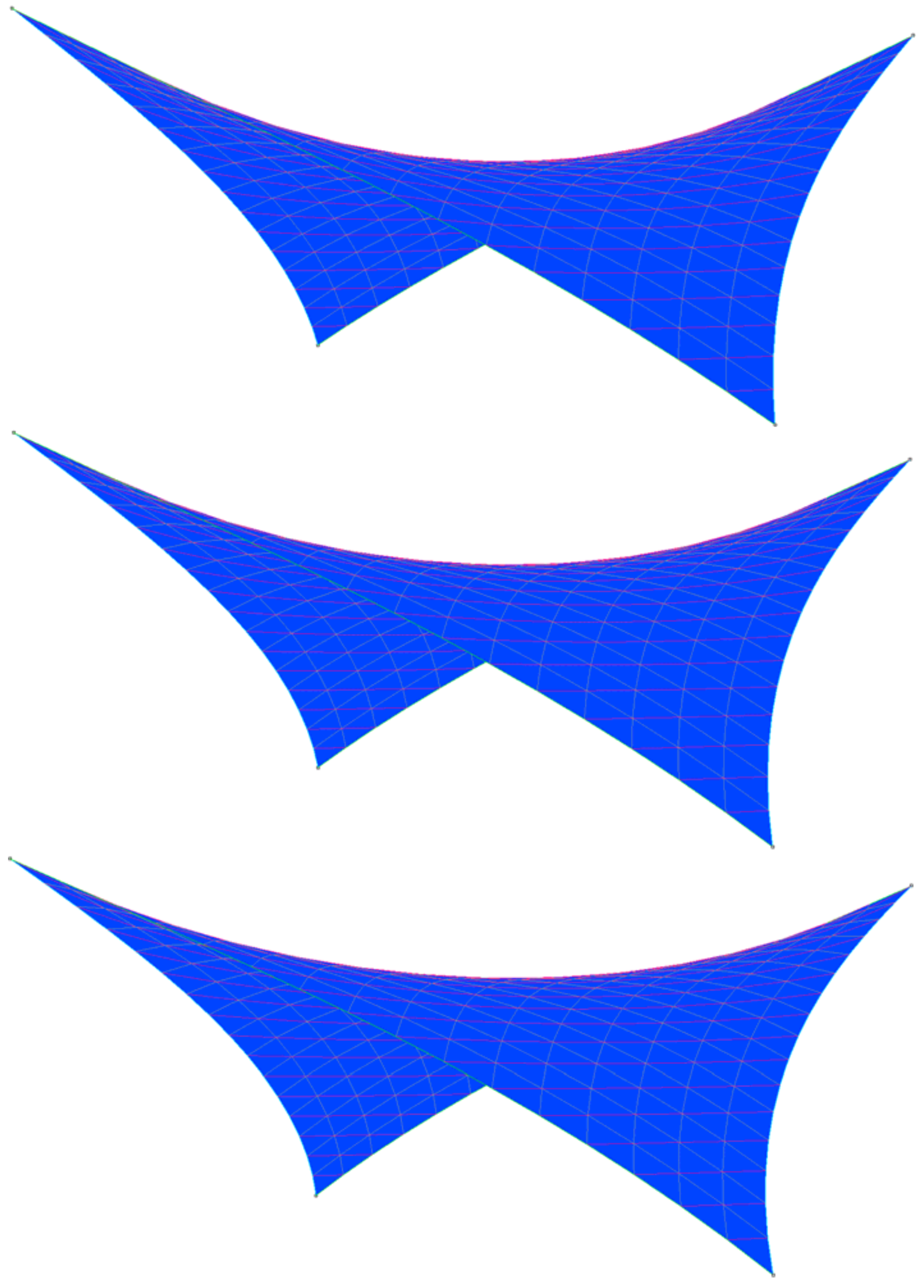


Figure 6-24 Showing the same hypar with increasing warp stress. The top picture has got a ratio, between the warp and weft stress, of 2:1, the one in the middle 3:1, and the bottom one 4:1

In Figure 6-24 it can be seen how the mid part of the membrane is moved further up with an increasing warp stress. Equally in Figure 6-25 it can be seen that by decreasing the warp stress (or increasing the weft stress) the mid part of the structure is gradually “falling” down.

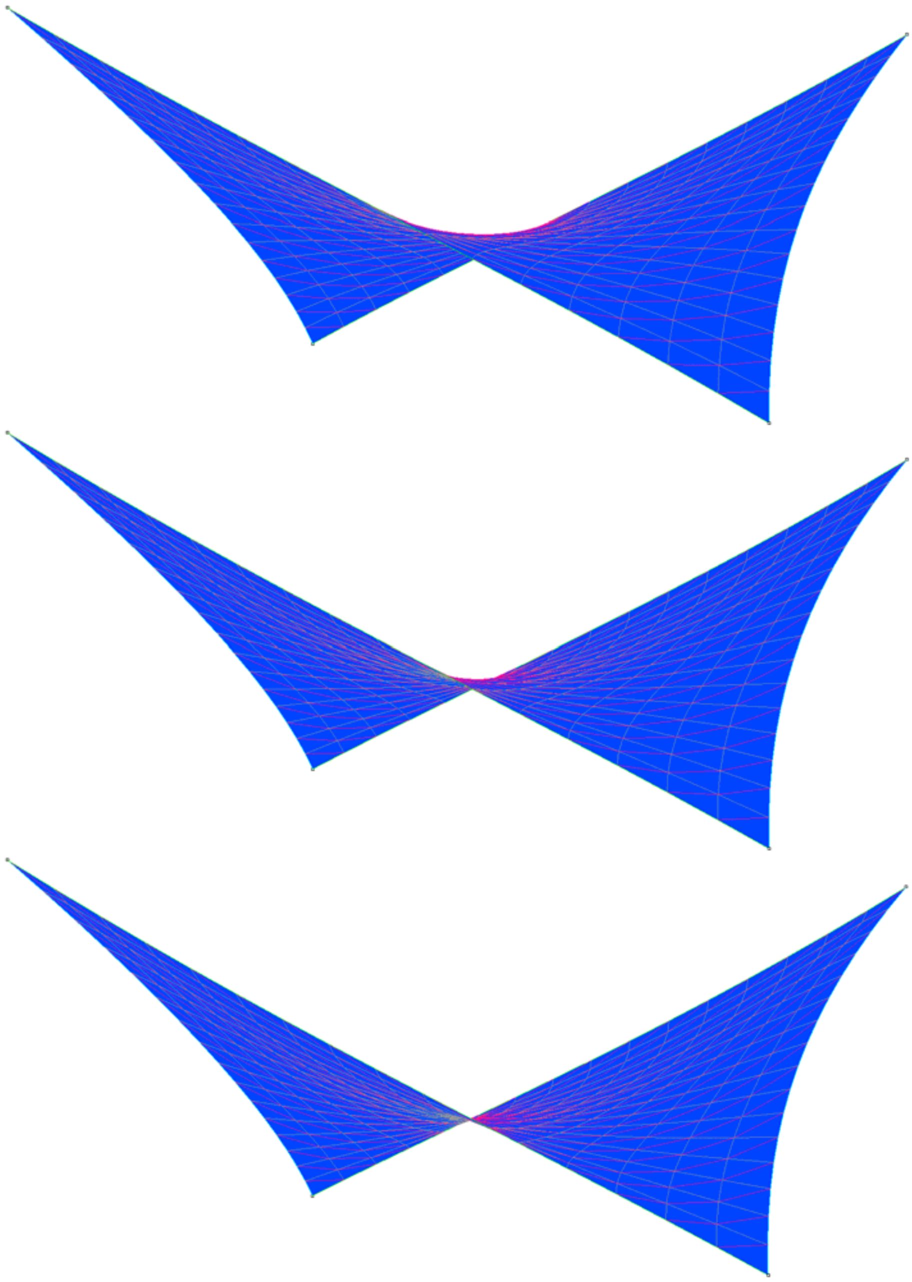


Figure 6-25 Showing the same hypar with decreasing warp stress. The top picture has got a ratio, between the warp and weft stress, of 1:2, the one in the middle 1:3, and the bottom one 1:4.

6.3 Trying out membranes for a structure

To give an understanding of what the calculated stresses mean, and the impact they have on the membrane, the possibility to visualise how much of the material capacity that is utilized, for different cases, was implemented in the program. Figure 6-26 is illustrating this; the same colour scale was used for all cases (0-20% of the capacity). It is also possible to instead visualise it as safety factors, i.e. the stress divided by the capacity.

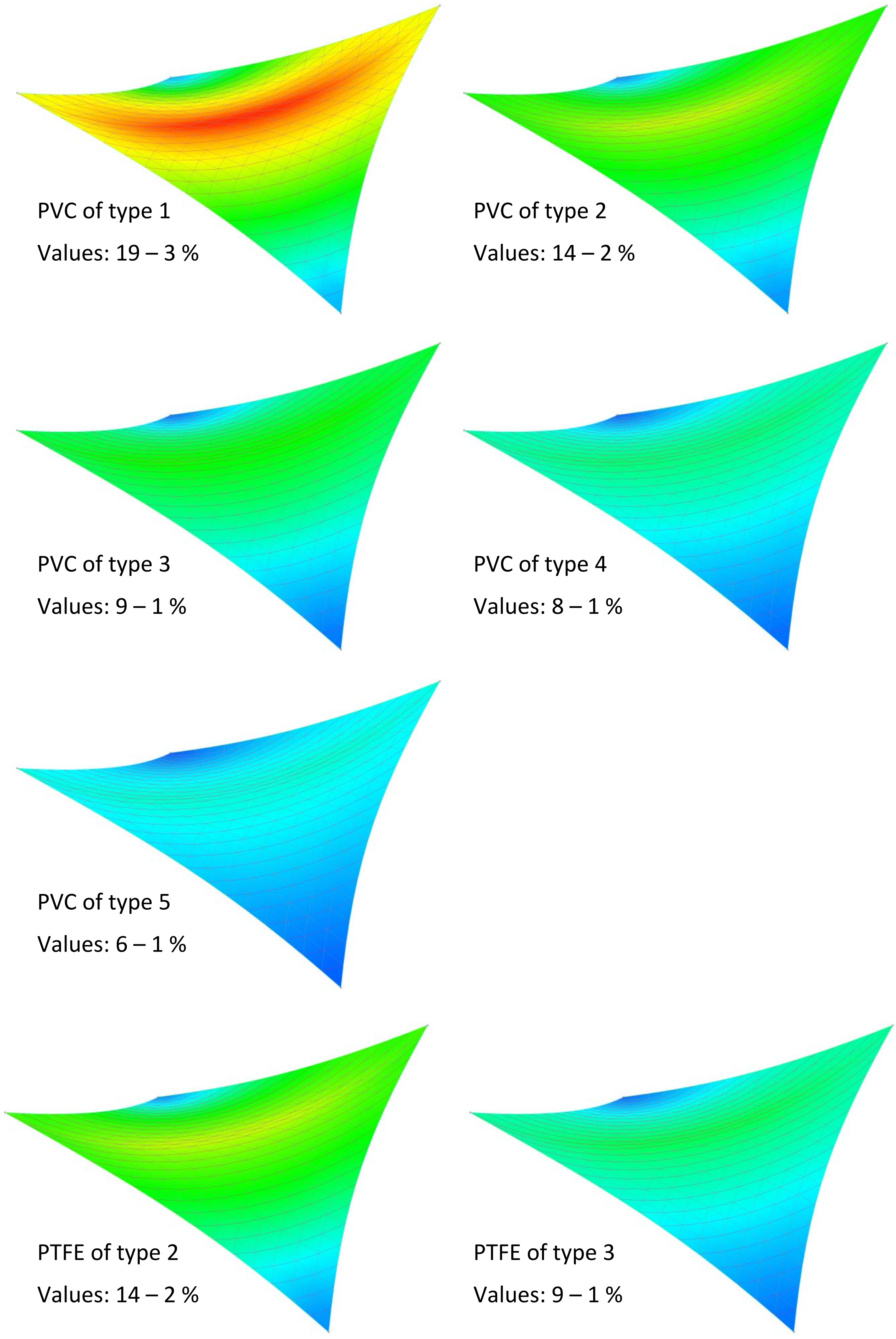


Figure 6-26 Showing the utilized strength of the different membranes, for the warp stress. The colour scale is ranging from blue, 0%, to red, 20% of the capacity.

6.4 Façade of the transformed London Olympic Stadium

The London Olympic stadium was designed to be dismountable, so that when it is used, after the Olympic Games, it will be of a size appropriate for sports events of a smaller scale. The engineers at Buro Happold, that were working on the London Olympic stadium, are now working on the transformation of it, the plan is to reduce the capacity of the stadium from the 80 000 spectators to 25 000. For this to be possible the stadium was designed with a dismountable super structure having a light weight cable supported roof.

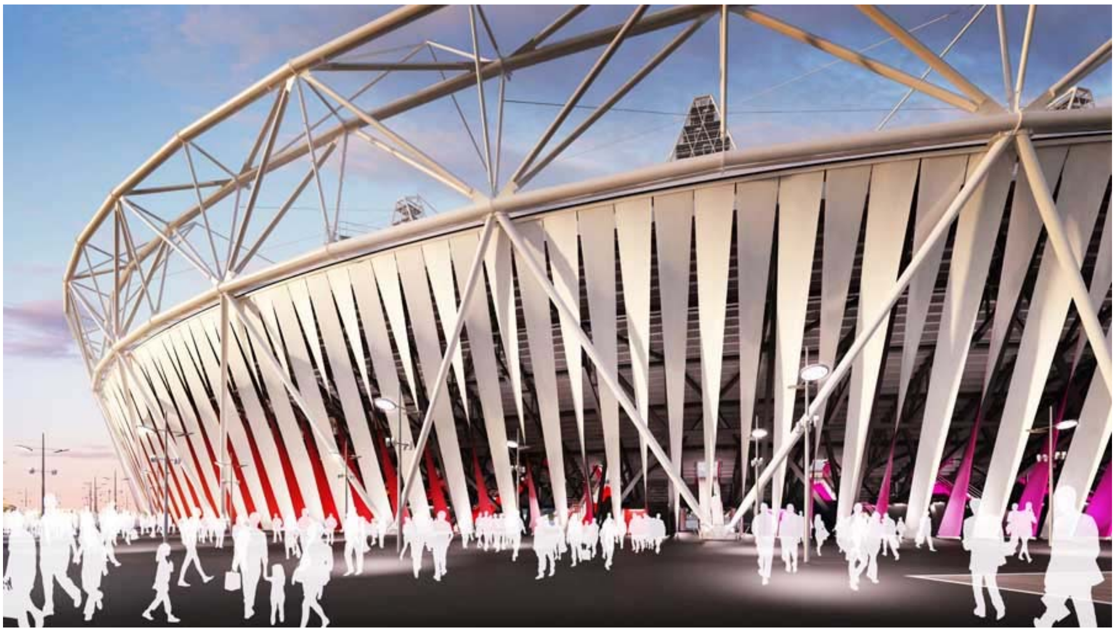


Figure 6-27 Showing the façade of the London Olympic stadium. [9]

For the new façade of the transformed London 2012 Olympic stadium, that Buro Happold are working on, different concepts were discussed and evaluated (the original stadium façade can be seen in Figure 6-27). In this case it was mainly the shape that was of interest, at the initial stage. Two of these concepts have been redone in the tool developed in this thesis (Figure 6-28 to Figure 6-31). The first example, shown in Figure 6-29 and Figure 6-30, was originally done in SMART Form. With the new tool this structure can be more accurately modelled with both membranes and cables.

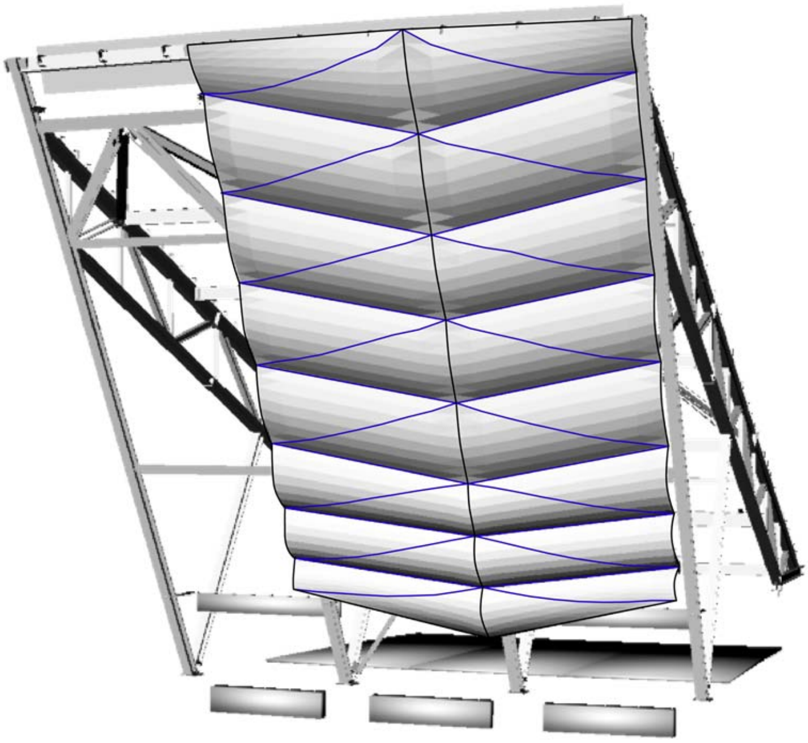


Figure 6-28 Start geometry for a concept for the façade of the transformed London 2012 Olympic stadium. The blue lines are representing cables and the black edges are "fixed" (pinned) constraints.

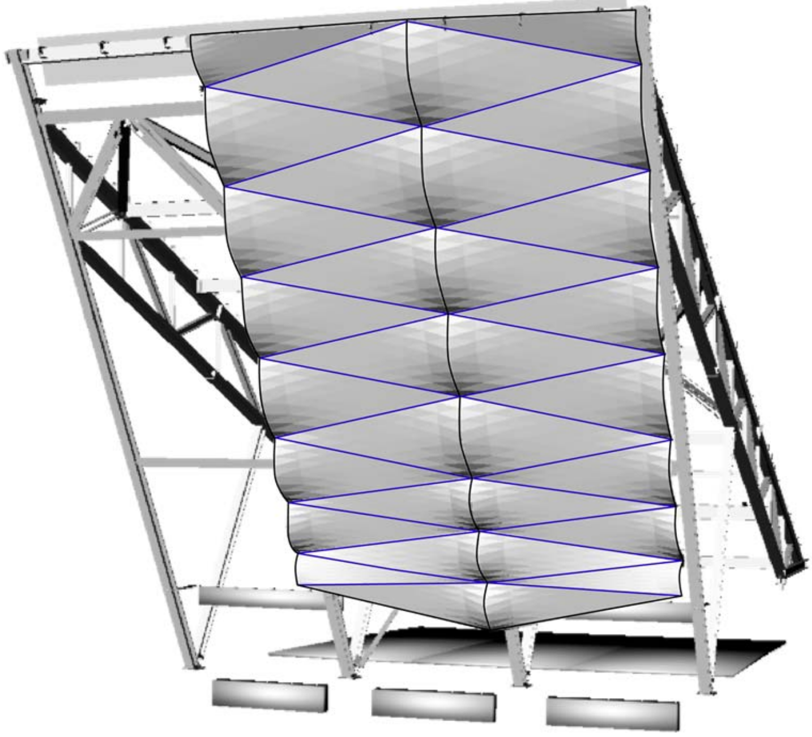


Figure 6-29 Formfound geometry for the façade above (Figure 6-28). The cables have been stretched and are thereby pulling in the membrane

For the design above the warp direction is set to be vertically and cables are going between the fixed arches, on the side of the "panels". Cable elements going diagonally across each surface patch needed to be implemented for this model (these also had to be shortened).

The second example was originally modelled in Tensyl. Using the new tool, the geometry is a lot quicker to set up, and the tool is generating a good visualisation (a 3D surface model) of the structure. Here the model is made up out of 25 surface patches (the "longer" patches are split into two). Four point constraints are put in the middle and curve constraints along the edges of the structure.

The reason for using multiple surface patches is the way the program is structured; cables can only be put in along an edge of a surface, or diagonally across it, additionally, by using 25 patches the overall mesh of the structure gets relatively even.

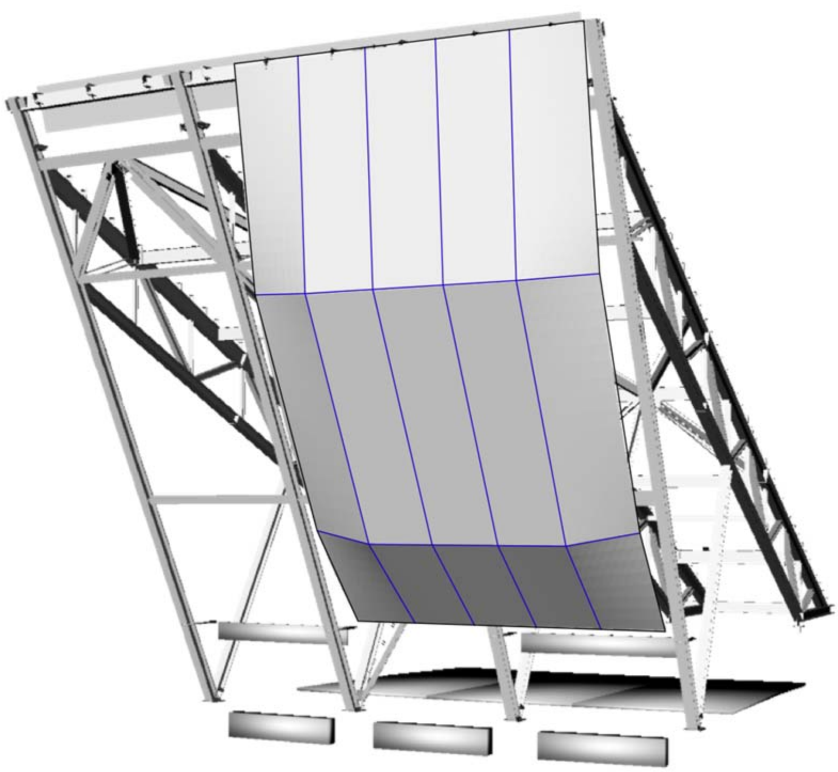


Figure 6-30 Start geometry for a conceptual design for the stadium facade

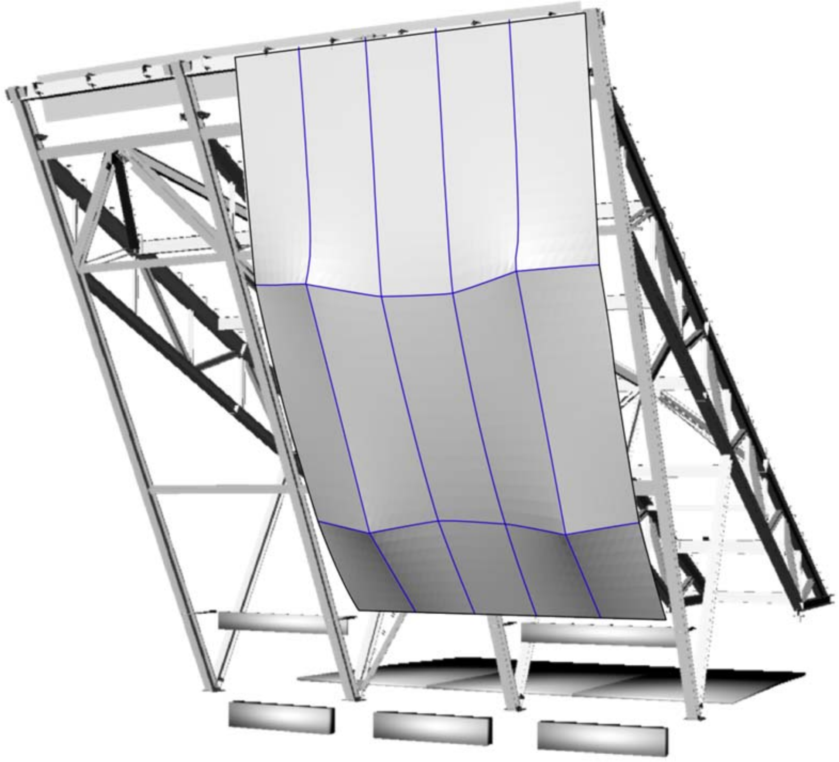


Figure 6-31 Formfound structure for the geometry above (Figure 6-30)

Working with the façade models, it is clear that there is still a lot of work that needs to be done on the program to make it really good. This can be seen especially in Figure 6-31, where the different mesh densities in the top part and the middle part is causing the cable to be pulled down. This could, of course be an effect that is sought, but it is demonstrating that there is a need for a method to mesh different surface patches differently, which is not possible in the present version of the tool.

7 Conclusion

In this work a tool for conceptual design of tensioned fabric structures has been developed and, even though it is far from a finished product, it is a useful tool with a lot of potential.

The simulations and analysis of tension structures, in general, seem to be a science under development, and it is difficult to know how accurate the analogies, used today, are. Talking about different methods for analysing membrane structures, with Professor Peter Gosling, who is working with the Eurocodes for membrane structures, he mentioned a new round robin survey that he was working on. The survey had been sent out to different companies working with these kinds of structures and the participants were asked to analyse a few different models. The given results varied considerably for all of them, which show that even analyses made by professionals could have substantial errors. This might not be a very big problem, since the safety factors used, makes today's structures both safe and long lived. However it shows that it might be possible to use even less material, and thereby saving energy and the environment.

An Achilles' heel for the tool is speed. If the tool is to be really useful during a design process further work needs to be put in to speeding up the program. This could be done by better calculating the fictitious masses, or even implement different fictitious mass in different directions, i.e. for each node having one mass in the x-direction, one in the y-direction and one in the z-direction (m_{ix} , m_{iy} , m_{iz}).

Other things that might speed up the process could be to try out different iteration methods, such as Runge-Kutta or the backward Euler method.

It might also be possible to speed up the analysis by using more threads in the code. The tool is already using multiple threading, one for the analysis and one for the drawing, but it might be possible to divide it in to more threads.

Comparing the developed tool with Tensyl (Buro Happold's analysis tool for tensile structures) the analysis is a lot slower, but when it comes to how much time that is needed for actual work, SMART Form comes out as the winner. Working with Tensyl a lot of time is needed for setting up the analysis model, the properties and the loads. This is really easy and quick to do in SMART Form and the time it takes to run the analysis can be used to work on other things. So in terms of developing a simple conceptual tool the work has been successful, even though there is a lot to do still before the program is completely stable and can be fully relied on.

The fact that Tensyl does the analysis so much faster does also imply that it should be possible to work up the speed for SMART Form's analysis, since Tensyl are using the same analysis solver.

Another big advantage with SMART Form is that the shape can be modified, in real time. Being able to see how the membrane moves by, for example, increasing the prestress gives the user a better understanding of the shape and the structure. Other examples of effects that can be useful to evaluate in real time is how the shape is affected by a change of cable diameters, by shortening/lengthening the cables or even by moving or changing the constraints.

The tool developed in this thesis has great potentials. Speaking with the engineers at Buro Happold, it seems like a tool like this would be very helpful, and a big advantage especially when the engineers are involved in the early stages of the design process. However there is still a long way to go before the tool could be seen as a finished product, several functionalities needs to be implemented and a lot more testing needs to be done to gain more confidence in the tool.

7.1 Suggested further development

There is a lot of interesting theory around tensile structures and a lot to read about different ways of modelling the membrane and how to best simulate the structure, and a lot of functionalities that would be interesting to implement in the developed tool. Unfortunately it is not possible to include it all in the time frame for this work. In this section a few ideas and thoughts that has come up during the work is presented as suggestions for further development.

Conic structures are, a lot of the time, very complex to generate. Depending on the geometry there might be a need of having the prestress change gradually throughout the structure to prevent it from “necking in”, see the mid shape in Figure 7-1. A gradually changing prestress could also be necessary in order to generate an asymmetric shape as the one shown to the right in Figure 7-1.

Another thing that would be fun to investigate and implement into the tool would be the possibilities of doing more hands on sculpturing of the shape. It would be interesting to see if it is possible to, instead of dictating the prestress to shape the structure, in some way push and pull in the model and let the software calculate the prestress needed to generate this shape. This is not a way that people seem to have worked before, but it could open up for new ways of designing tension structures. A feature like this does however require an intelligent way of interpreting the directions given by the user.

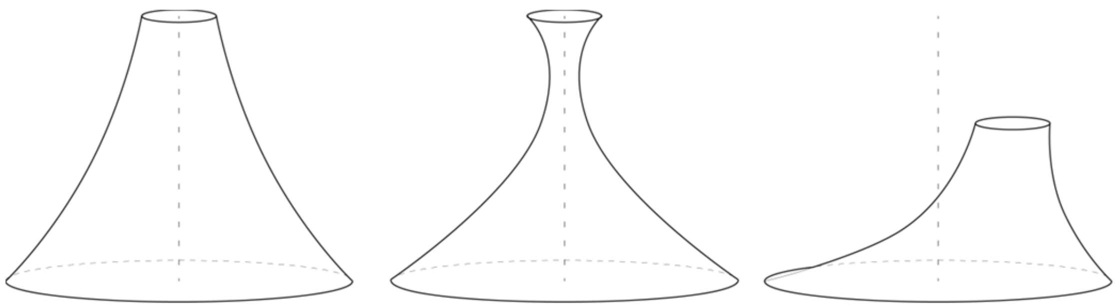


Figure 7-1 Conic shapes that can be created by modifying the prestress, or slightly changing the starting geometry.

Working with the tool it is very clear that, even if the model is really easy to set up, not being able to save the analysis model is a great disadvantage. As the tool is structured today, the input geometry and the final results can be saved, in rhino, but not the model with the chosen materials and loads. It should also be easy to export the model into other analysis tools, to make a more detailed analysis.

8 Bibliography

Adriaenssen, S. M. L., 2000. *Stressed Spline Structures,* University of Bath: PhD Thesis , pp. 13-47.

Bak, A., 2011. *Interactive Formfinding for Optimised Fabric-Cast Concrete,* Department of Architecture and Civil Engineering, University of Bath: Master Thesis.

Barnes, M. R., 1999. Form Finding and Analysis of Tension Structures by Dynamic Relaxation. *International Jornal of Space Structures,* 14(2), pp. 89-104.

Bechthold, M., 2008. *Innovative Surface Structures_ Technologies and Applications.* Abingdon: Taylor & Francis.

Brew J. S., Brotton D. M., 1971. Non-linerar structural analysis by dynamic relaxation. *International Journal for Numerical Methods in Engineering,* Volume 3, pp. 463-483.

Bunce J W and Day A S, 1970. Analysis of Cable Networks by Dynamic Relaxation. *Civil Engineering Public Works Review,* Volume 7, pp. 383-386.

Buro Happold, in-house material data

Buro Happold, Tensyl old user manual

Buro Happold, 2012, Tensyl code (C++)

Dixon, J. M., 2007. Suvarnabhumi Airport, Bangkok. *architectural record,* Issue 8, pp. 108-117.

Frei Otto, Bodo Rasch, 1995. *Finding Form.* Stuttgart: Axel Menges.

Gosling P D, Discussions 07-2012

Gosling P D , Zhang L, 2012. A Linear Strain, Curvature-Driven Trinagular Element for the Analysis of Membrane Strucures. *CMES,* 2007(1), pp. 1-45.

Gründig, L., 1988. Minimal surfaces for finding forms of structural membrane. *Computers & Structures,* 30(3), pp. 679-683.

Holland, J. A., 1967. *Dynamic relaxation applied to local effects, Conf. on Presstressed Concreate Pressure Vessels.* London, ICE.

Klaus-Michael Koch, Karl J Habermann, 2004. *Membrane Structures: innovative building with film and fabric.* Munich: Prestel.

Lei, Z., 2010. *Reliability Analysis of Fabric Structures,* School of Civil Engineering and Geosciences, Newcastle University: PhD. Thesis.

Lewis, W. J., 2003. *Tension Structures Form and Behavior.* London: Thomas Telford Publishing.

Lidell, I. *The Engineering of Surface Stressed Structures*, pp 8-16.

Oasys, 1985–2011. *GSA Help Manual*.

Paul Romain, Joanne Renold-Smith, 2012. *Meetings.* Bath, s.n.

Pauletti, R. M. O., 2008. The natural force density method for the shape finding of taut structures, Computer Methods in Applied Mechanics and Engineering. September 15, 197(49-50), pp. 4419-4428.

Robert D Cook, David S Malkus, Michael E Pleasha, Robert J Witt, 2002. *Concepts and applications of finite element analysis.* Fourth ed. s.l.:John Wiley & sons.

Volino, P & Magnenat-Thalmann, N, 2001. *Comparing Efficiency of Integration Methods for Cloth Simulation. Proceedings of the international Conference on Computer Graphics.* s.l., IEEE Computer Society.

Volino, P, Magnenat-Thalmann, N & Faure, F, 2009. A simple approach to nonlinear tensile stiffness for accurate cloth simulation. *ACM Trans. Graph,* Volume 28, pp. 1-16.

Wakefield, D. S., 1999. Engineering analysis of tension structures: theory and practice. *Enineering Structures,* Issue 21, pp. 680-690.

8.1 Figures

[1] Buro Happold, O2 Arena and Millennium Dome, http://www.burohappold.com/projects/project/o2-arena-and-millennium-dome-65/ (2012-10-30)

[2] Library and Archives Canada/ PA-173152, The West German Pavilion of the 1967 Montréal Exposition, http://www.architectureweek.com/cgi-bin/awimage?dir=2010/0519&article=news_5-2.html&image=13239_image_4.jpg (2012-10-22)

[3] Wikimedia Commons, ArchInform, AD Classics: Munich Olympic Stadium / Frei Otto & Gunther Behnisch, http://www.archdaily.com/109136/ad-classics-munich-olympic-stadium-frei-otto-gunther-behnisch/ (2012-10-12)

[4] krss (2004) Olympiastadion München/Munich, Dach, Frei Otto http://www.flickr.com/photos/krss/6603916903/in/photostream/ (2012-10-12)

[5] Buro Happold, London 2012 Olympic Stadium, http://www.burohappold.com/projects/project/london-2012-olympic-stadium-132/?goback=.gmp_4586196.gde_4586196_member_152534165 (2012-10-22)

[6] Rainer Viertlböck, Gauting, Werner Sobek, Bangkok, http://www.wernersobek.de/index.php?page=74&modaction=detail&modid=285 (2012-10-22)

[7] Dr. Josep Llorens, Report: Textile Roofs (2002) http://www.textile-roofs.com/Reports/TR_report-02.htm (30-10-2012)

[8] MAS Measurement & Analysis Systems, School of Civil Engineering and Geosciences, Newcastle University, 2011, Nuvola Fabric Testing.

[9] Populous, London Olympic Stadium Wrap image, http://www.e-architect.co.uk/london/london_olympic_stadium.htm (2012-10-27)

Appendix A **Equations for link tensions in terms of membrane stresses**

The equations and explanations in this appendix are all taken from (Barnes, 1999).

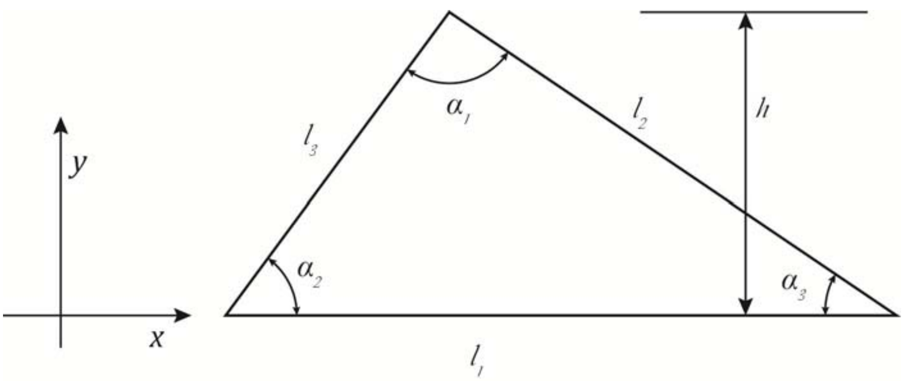


Figure 8-1 Showing the triangular element used in the mesh representing the fabric.

If side 1 is parallel to the x-axis (and the warp directions) the strains related to the x and y axes can be expressed as link extensions as follows

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \dfrac{1}{l_1} & 0 & 0 \\ \dfrac{a_3c_2 - a_2c_3}{Ql_1} & \dfrac{c_3}{Ql_2} & \dfrac{-c_2}{Ql_3} \\ \dfrac{a_2b_3 - a_3b_2}{Ql_1} & \dfrac{-b_3}{Ql_2} & \dfrac{b_2}{Ql_3} \end{bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \\ \Delta_3 \end{Bmatrix} \tag{A-1}$$

$$\text{or: } \{\varepsilon_x\} = [G]\{\Delta\}$$

where $a_i = cos^2\theta_i$, $b_i = sin^2\theta_i$, $c_i = sin^2\theta_i cos^2\theta_i$ and $Q = b_2c_3 - b_3c_2$ with θ_i in the deformed state.

The equivalent link forces can then be expressed in terms of the stresses ($\varepsilon_x, \ \varepsilon_y, \tau_{xy}$ per unit width) by applying the principle of virtual work:

$$\{\varepsilon^*\} = [G']\{\Delta^*\}$$

where $\{\Delta^*\}$ is an infinite small virtual deformation and $[G']$ are identical to $[G]$ except that the side lengths are those for the deformed state.

The virtual work is then:

$$\{\Delta^*\}^T\{T\} = \{\varepsilon^*\}^T\{\sigma\}A$$

where A is the deformed area of the element.

Hence the side tensions equivalent to the stresses are:

$$\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = A[G']^T \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} \qquad (A\text{-}2)$$

It is convenient to calculate the tensions due to shear separately. The tensions due to direct stresses den becomes:

$$T_1 = \frac{\sigma_x A}{l_1} + \frac{\sigma_y A}{Q l_1}(a_3 c_2 - a_2 c_3) \quad ;$$
$$T_2 = \frac{\sigma_y A c_3}{Q l_2} \quad ; \quad T_3 = -\frac{\sigma_y A c_2}{Q l_2}$$

With trigonometric manipulation this becomes:

$$T_1 = \frac{h}{2}\left(\sigma_x - \sigma_y\right) + \frac{\sigma_y l_1}{2\tan\alpha_1} \quad ;$$
$$T_2 = \frac{\sigma_y l_2}{2\tan\alpha_2} \quad ; \qquad T_3 = \frac{\sigma_y l_3}{2\tan\alpha_3} \qquad (4\text{-}1)$$

Appendix B Results from simple test case

Warp				
Smart Form				
mesh	**prestress [kN]**	**applied pressure [kN/m^2]**	**result [kN]**	**hand calc. [kN]**
10x30	0,1	1	4,91	4,96
6X20	0,1	1	4,88	4,96
10X30	0,05	1	4,86	4,89

Tensyl				
6x20	0,1	1	4,83	4,96

Weft				
Smart Form				
mesh	**prestress**	**applied pressure**	**result**	**hand calc.**
10x30	0,1	1	4,15	4,2
10x30	0,05	1	4,14	4,13

Tensyl				
6x20	0,1	1	4,07	4,11

Appendix C **Material Data**

Material	Grade	Weight [kg/m2]	EA [kN/m]		G [kN/m]	Unfactored Tensile Strength [kN/m]	
			Warp	Weft		Warp	Weft
PVC	Type 1	0,8	670	400	10	60	60
	Type 2	1,05	670	400	10	84	80
	Type 3	1,05	1300	900	15	112	112
	Type 4	1,3	1300	900	15	149	128
	Type 5	1,45	1600	1200	20	196	166
PTFE	Type 2	0,8	900	600	10	84	80
	Type 3	1,15	1000	650	10	140	120

Table C-8-1 showing the material properties used for the membranes in the program.

C.1 Code for materials

Cable List

```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class CableList
    {
        protected List<MaterialCable> _cables;
        protected CableGalvanisedSteelSS _galvanisedSteelSS;
        protected CableStainlessSteelSS _stainlessSteelSS;
        protected Seam _seam;
        protected NoCable _noCable;

        protected bool _active;



        public CableList()
        {
            _cables = new List<MaterialCable>();

            _galvanisedSteelSS = new
CableGalvanisedSteelSS(8);//smallest diameter and the first one in
the list
            _stainlessSteelSS = new CableStainlessSteelSS(6);
            _seam = new Seam();
            _noCable = new NoCable();

            _cables.Add(_galvanisedSteelSS);//0
            _cables.Add(_stainlessSteelSS);//1
            _cables.Add(_seam);//2
            _cables.Add(_noCable);//3


            _active = false;
        }

        //*******************************************
        // Properties
        //*******************************************

        public List<MaterialCable> List
        {
            get { return _cables; }
        }

        public bool Active
        {
            get { return _active; }
            set { _active = value; }
        }

    }
}
```

Material Cable

```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class MaterialCable:Material
    {
        //Atributes
        protected double _diameter;
        protected int _section;
        protected double _metalicCrosSection;
        protected double _weight;
        protected double _breakingLoad;
        protected List<int> _cableDiametrs;
        protected double _limitTension;


        public MaterialCable()
            : base()
        {
            _cableDiametrs = new List<int>();
        }

        public double Diameter
        {
            get { return _diameter; }
            set { _diameter = value; }
        }

        //section is the aproximate diameter (the integer shown in
the list of diamters for the cable)
        public int Section
        {
            get { return _section; }
            set { _section = value; }
        }

        public double MetalicCrosSection
        {
            get { return _metalicCrosSection; }
            set { _metalicCrosSection = value; }
        }

        public double Weight
        {
            get { return _weight; }
            set { _weight = value; }
        }

        public List<int> CableDiameters
        {
            get { return _cableDiametrs; }

        }



        public virtual void SetSection(int d)
        {
```

```
        }

        public void SetSectionByIndex(int index)
        {
            if (index > _cableDiametrs.Count)
                RMA.Rhino.RhUtil.RhinoApp().Print("\nCable diameter
out of range");

            else
            {
                int section = _cableDiametrs[index];
                SetSection(section);
            }
        }


        public double BreakingLoad
        {
            get { return _breakingLoad; }
            set { _breakingLoad = value; }
        }

        public double LimitTension
        {
            get { return _limitTension; }
            set { _limitTension = value; }
        }



        /** copies values from one cable to another, without making
both instances pointing to the same object
       *
       */
        public void CopyCableProperties( MaterialCable Original)
        {
            _breakingLoad = Original._breakingLoad;
            _density = Original._density;
            _diameter = Original._diameter;
            _E = Original._E;
            _G = Original._G;
            _limitTension = Original._limitTension;
            _metalicCrosSection = Original._metalicCrosSection;
            _section = Original._section;
            _weight = Original._weight;
        }

    }
}
```

Cable Galvanised Steel Spiral Strand

```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class CableGalvanisedSteelSS:MaterialCable
    {
        //This is a cal=ble class containing information about the
properties of a specific cable
        //To add another section/diameter/size remember to add the
diameter in the "_cableDiameters"
        //in "intitialize", and to add it in the "setSection(int d)".

        public CableGalvanisedSteelSS()
        {
            initialize();
        }

        public CableGalvanisedSteelSS(int d)
        {
            initialize();

            SetSection(d);

        }

        protected void initialize()
        {

            _E = 160000000000; //N/m
            _breakingLoad = 260000; //N

            _diameter = 17e-3; //m
            _section = 17;
            _metalicCrosSection = 168e-6; //m
            _weight = 1.3; //kg/m
            _limitTension = 158000; // N


            _cableDiametrs.Add(8);
            _cableDiametrs.Add(10);
            _cableDiametrs.Add(12);
            _cableDiametrs.Add(14);
            _cableDiametrs.Add(17);
            _cableDiametrs.Add(20);
            _cableDiametrs.Add(24);
            _cableDiametrs.Add(28);
            _cableDiametrs.Add(32);
            _cableDiametrs.Add(36);
        }



        public override void SetSection(int d)
        {
            d = d - (d % 2);

            switch (d)
```

```
{

    case 8:
        _diameter = 8.1e-3; //m
        _section = 8;
        _metalicCrosSection = 39e-6; //m
        _weight = 0.3; //kg/m
        _breakingLoad = 59000; // N
        _limitTension = 36000; // N
        break;

    case 10:
        _diameter = 10.1e-3; //m
        _section = 10;
        _metalicCrosSection = 60e-6; //m
        _weight = 0.5; //kg/m
        _breakingLoad = 93000; // N
        _limitTension = 56000; // N
        break;

    case 12:
        _diameter = 12.2e-3; //m
        _section = 12;
        _metalicCrosSection = 87e-6; //m
        _weight = 0.7; //kg/m
        _breakingLoad = 134000; // N
        _limitTension = 81000; // N
        break;

    case 14:
        _diameter = 14.1e-3; //m
        _section = 14;
        _metalicCrosSection = 117e-6; //m
        _weight = 0.9; //kg/m
        _breakingLoad = 181000; // N
        _limitTension = 110000; // N
        break;

    case 17:
        _diameter = 17e-3; //m
        _section = 17;
        _metalicCrosSection = 168e-6; //m
        _weight = 1.3; //kg/m
        _breakingLoad = 260000; //N
        _limitTension = 158000; // N
        break;

    case 18:

    case 20:
        _diameter = 20.1e-3; //m
        _section = 20;
        _metalicCrosSection = 237e-6; //m
        _weight = 1.9; //kg/m
        _breakingLoad = 367000; // N
        _limitTension = 222000; // N
        break;

    case 22:
```

```
                case 24:
                    _diameter = 24.4e-3; //m
                    _section = 24;
                    _metalicCrosSection = 347e-6; //m
                    _weight = 2.7; //kg/m
                    _breakingLoad = 537000; // N
                    _limitTension = 325000; // N
                    break;

                case 26:


                case 28:
                    _diameter = 28.3e-3; //m
                    _section = 28;
                    _metalicCrosSection = 467e-6; //m
                    _weight = 3.7; //kg/m
                    _breakingLoad = 722000; // N
                    _limitTension = 438000; // N
                    break;

                case 32:
                    _diameter = 31.3e-3; //m
                    _section = 32;
                    _metalicCrosSection = 572e-6; //m
                    _weight = 4.5; //kg/m
                    _breakingLoad = 884000; // N
                    _limitTension = 536000; // N
                    break;

                case 36:
                    _diameter = 36.3e-3; //m
                    _section = 36;
                    _metalicCrosSection = 769e-6; //m
                    _weight = 6.1; //kg/m
                    _breakingLoad = 1189000; // N
                    _limitTension = 721000; // N
                    break;

                default: //16mm cable
                    _diameter = 17e-3; //m
                    _section = 17;
                    _metalicCrosSection = 168e-6; //m
                    _weight = 1.3; //kg/m
                    _breakingLoad = 260000; //N
                    _limitTension = 158000; // N
                    break;
            }
        }




    }
}
```

Membrane List

```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class MembraneList
    {
        List<MaterialMembrane> _membranes;
        MaterialPVCPolyesterFerrariType2 _ferrariType2;
        MaterialPVCPolyesterFerrariType3 _ferrariType3;
        MaterialPVCPolyesterVerseidagType1 _verseidagType1;
        MaterialPVCPolyesterVerseidagType4 _verseidagType4;
        MaterialPVCPolyesterVerseidagType5 _verseidagType5;
        MaterialPTFEVerseidagType2 _PTFEVerseidag2;
        MaterialPTFEVerseidagType3 _PTFEVerseidag3;

        public MembraneList()
        {
            _membranes = new List<MaterialMembrane>();

            _ferrariType2 = new MaterialPVCPolyesterFerrariType2();
            _ferrariType3 = new MaterialPVCPolyesterFerrariType3();
            _verseidagType1 = new
MaterialPVCPolyesterVerseidagType1();
            _verseidagType4 = new
MaterialPVCPolyesterVerseidagType4();
            _verseidagType5 = new
MaterialPVCPolyesterVerseidagType5();
            _PTFEVerseidag2 = new MaterialPTFEVerseidagType2();
            _PTFEVerseidag3 = new MaterialPTFEVerseidagType3();

            _membranes.Add(_ferrariType2);
            _membranes.Add(_ferrariType3);
            _membranes.Add(_verseidagType1);
            _membranes.Add(_verseidagType4);
            _membranes.Add(_verseidagType5);
            _membranes.Add(_PTFEVerseidag2);
            _membranes.Add(_PTFEVerseidag3);

        }

        //*******************************************
        // Properties
        //*******************************************

        public List<MaterialMembrane> List
        {
            get { return _membranes; }
        }


    }
}
```

Membrane Material

```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class MaterialMembrane:Material
    {

        //Attributes
        protected double _EAx;//warp
        protected double _EAy;//weft

        protected double _weight;

        protected double _tensileStrX;
        protected double _tensileStrY;

        protected double _tearResX;
        protected double _tearResY;

        protected double _poissonX;
        protected double _poissonY;



        public MaterialMembrane()
            : base()
        {
            _poissonX = 0;//0.3;
            _poissonY = 0;// 0.3;
        }



        //---- Properties -------------------------------------------

        public double EAwarp
        {
            get { return _EAx; }
            set { _EAx = value; }
        }

        public double EAweft
        {
            get { return _EAy; }
            set { _EAy = value; }
        }

        public double Weight
        {
            get { return _weight; }
            set { _weight = value; }
        }

        public double TensileStrengthWarp
        {
            get { return _tensileStrX; }
            set { _tensileStrX = value; }
```

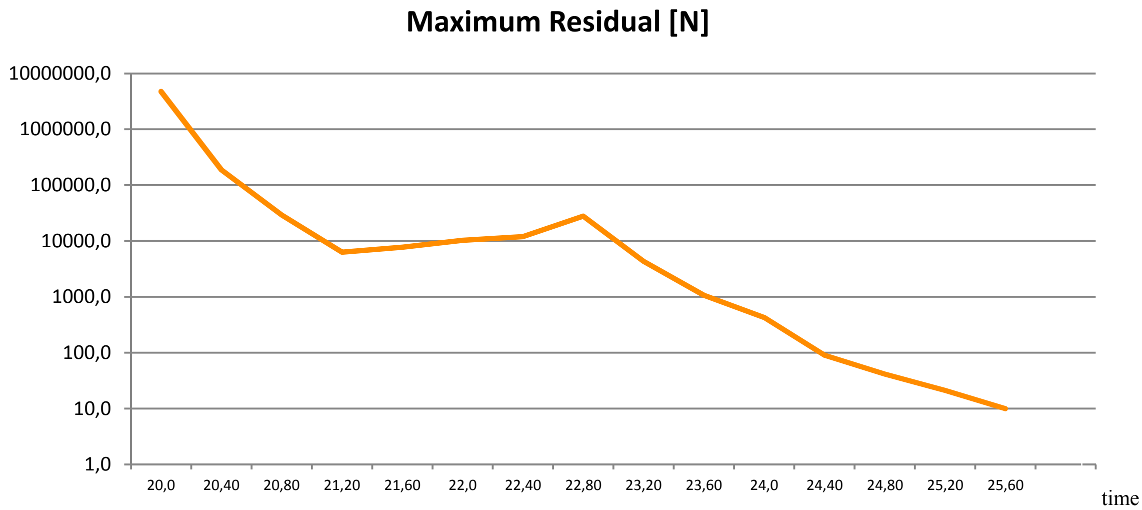
```
        }

        public double TensileStrengthWeft
        {
            get { return _tensileStrY; }
            set { _tensileStrY = value; }
        }

        public double TearResistanceWarp
        {
            get { return _tearResX; }
            set { _tearResX = value; }
        }


        public double TearResistanceWeft
        {
            get { return _tearResY; }
            set { _tearResY = value; }
        }

        public double PoissonsRatioWarp
        {
            get { return _poissonX; }
            set { _poissonX = value; }
        }

        public double PoissonsRatioWeft
        {
            get { return _poissonY; }
            set { _poissonY = value; }
        }


        public void CopyMembraneProperties(MaterialMembrane original)
        {
            _density = original._density;
            _EAx = original._EAx;
            _EAy = original._EAy;
            _G = original._G;
            _tearResX = original._tearResX;
            _tearResY = original._tearResY;
            _tensileStrX = original._tensileStrX;
            _tensileStrY = original._tensileStrY;
            _weight = original._weight;
        }

    }
}
```

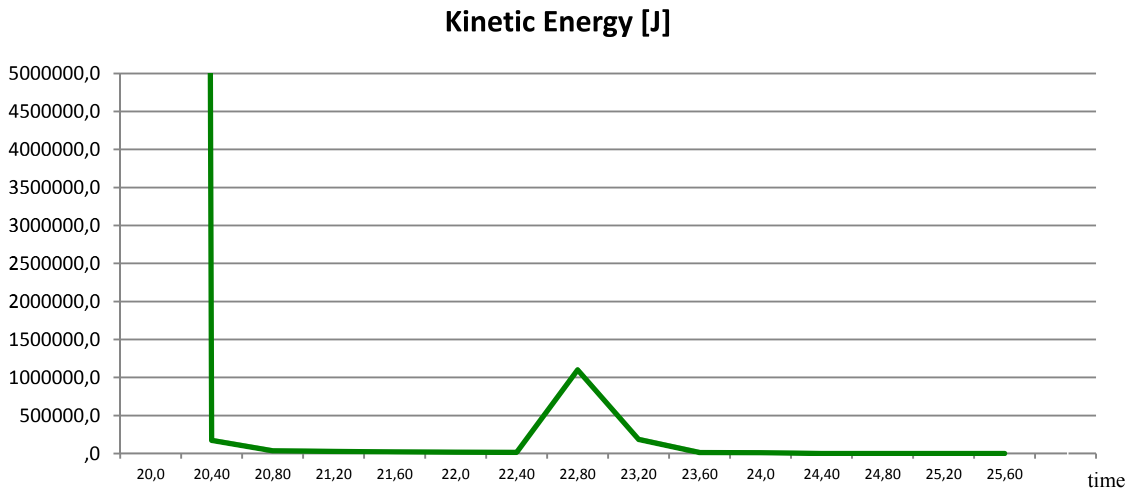
Material PVC Polyester Ferrari Type 2

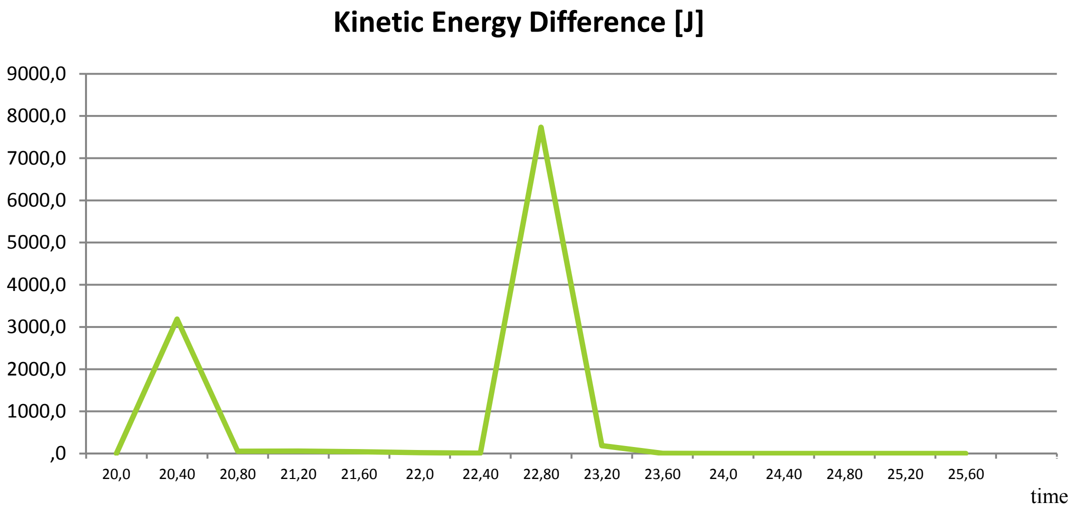
```
using System;
using System.Collections.Generic;
using System.Text;

namespace SMARTForm
{
    public class MaterialPVCPolyesterFerrariType2: MaterialMembrane
    {

        public MaterialPVCPolyesterFerrariType2()
            : base()
        {

            _G = 10000; // N/m
            _EAx = 670000; // N/m
            _EAy = 400000;// N/m

            _weight = 1.05; // kg/m2

            _tensileStrX = 84000; // N/m
            _tensileStrY = 80000; // N/m
            _tearResX = 550; // N
            _tearResY = 500; // N


        }



    }
}
```

Appendix D Convergence Graphs

The graphs are plotted for formfinding of a hypar, using both kinetic damping and a viscous damping of 0.05.







The maximum residual, shown in the top graph, is the maximum of out of balance force for any node.

The reason for why the graphs does not look like the examples of typical kinetic energy graphs are that the values are plotted at a set time step. Thereby the curves are not smooth and the exact peak values might be missed in the plots.

Appendix E Benchmark Hypar with Other warp direction

The warp of a fabric is usually aligned with the principal stress direction, which for a hypar is diagonally across. If the warp direction instead is rotated to be aligned with the edges, the stress in the warp direction decrease to about half of what it was before, while the weft stress is higher and more evenly distributed across the surface.

The materials and loads are the same as for the hypar analysed in chapter 6. A prestress of 2 kN/m are applied to the membrane, and a pressure load of 1 kN/m^2 is added to the structure. The membrane is PVC Polyester type 2, with galvanised steel, spiral strand, edge cables.

Figure D-8-2 and Figure D-8-4 are showing analysis results from SMART Form, these results are compared with results from Tensyl shown in Figure D-8-3 and Figure D-8-5 .

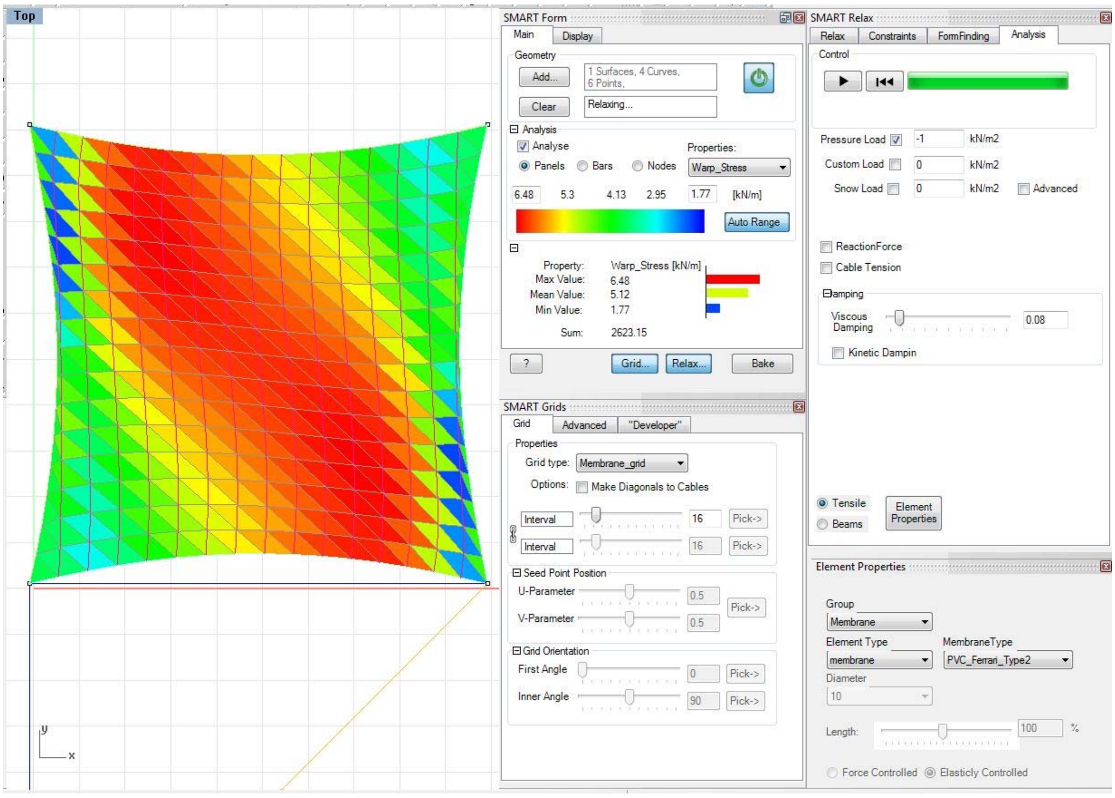


Figure D-8-2 Results from SMART Form. Showing the warp stress pattern for with the warp running in the vertical direction (values are ranging from 6.5 kN/m to 1.8 kN/m)

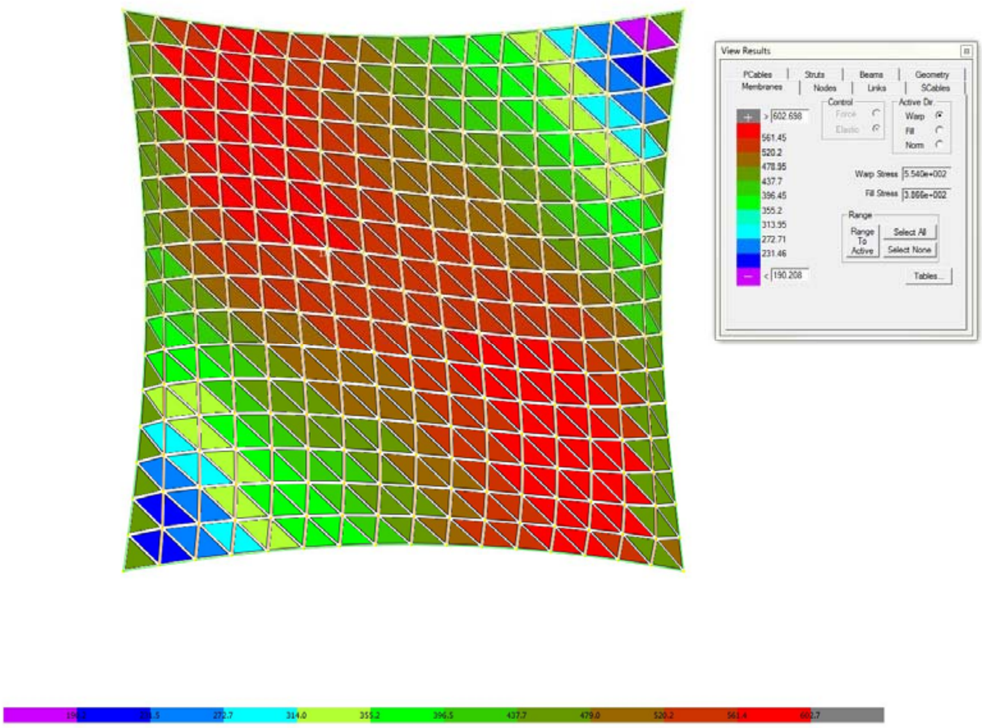


Figure D-8-3 Showing the warp stress pattern from an analysis done in Tensyl, with the warp running in the vertical direction (values are ranging from 6.0 kN/m to 1.9 kN/m)

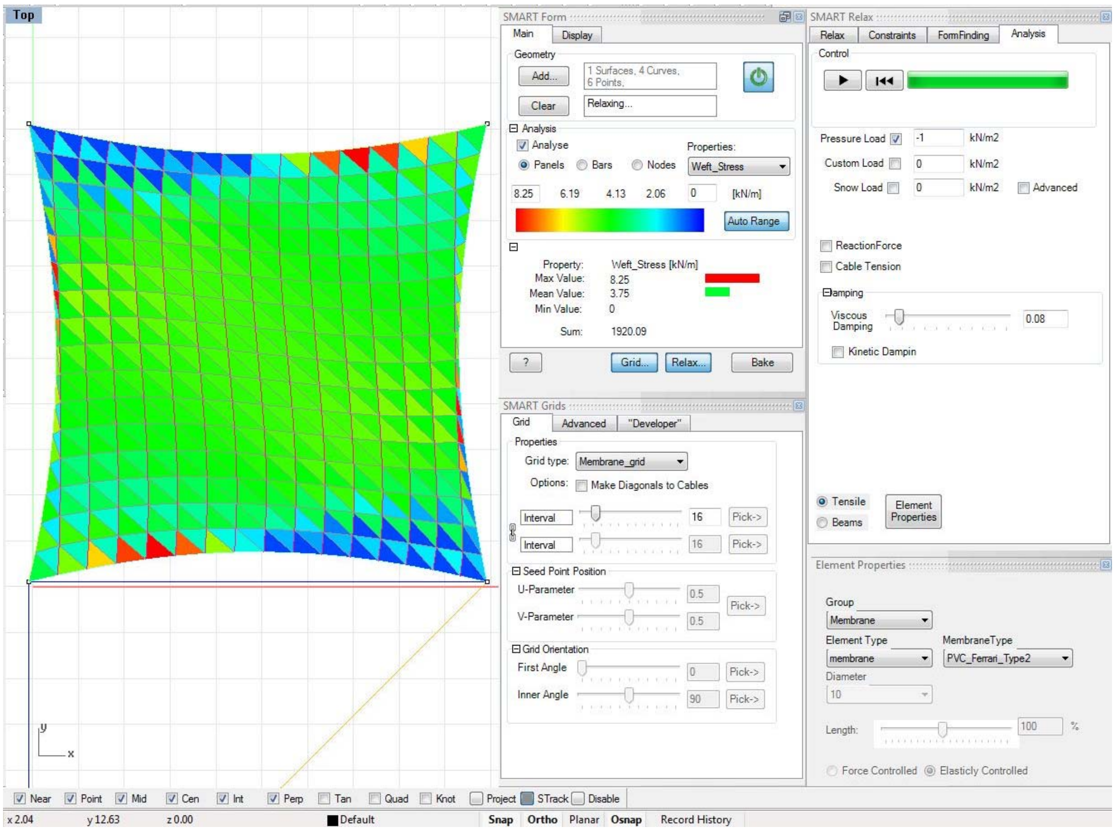


Figure D-8-4 Showing the weft stress calculated in SMART Form (values are ranging from 8.3 kN/m to 0)

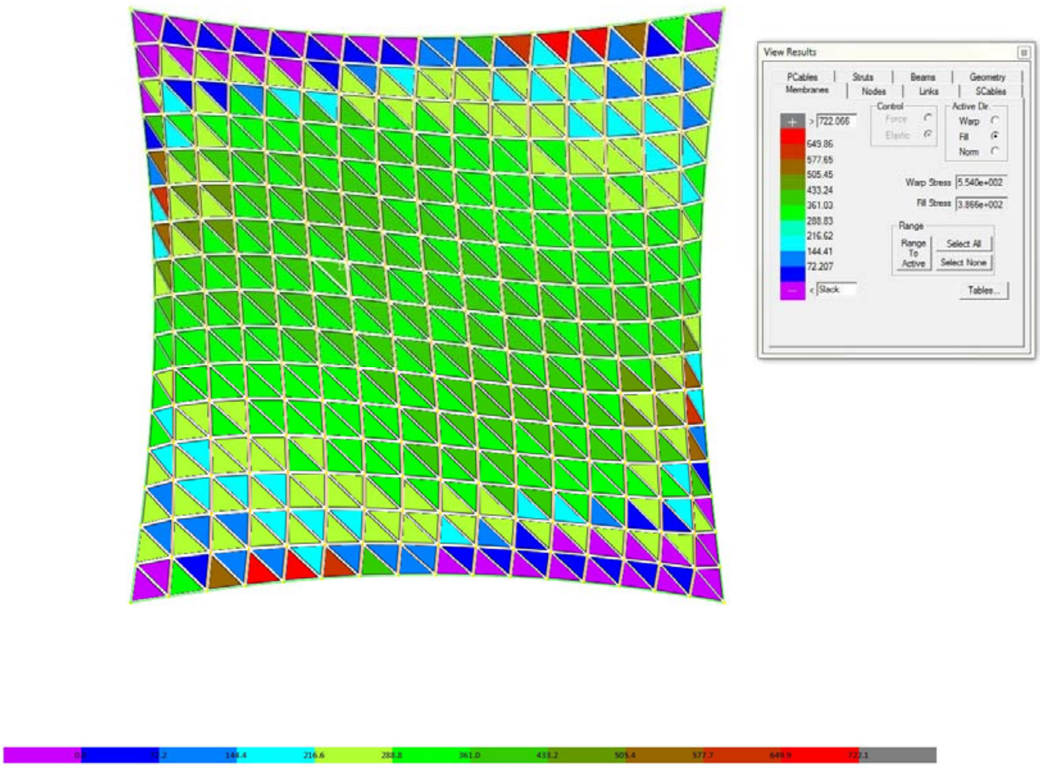


Figure D-8-5 Showing the weft stress calculated in Tensyl (values are ranging from 7.2 kN/m to 0)

Appendix F Fictitious mass tests

To analyse the speed of the software a test with different factious masses for the nodes was carried out. The result from this study can be seen in the table below and has been summarized in Figure E-1.

Mesh	Formfinding		Result	Analysis		Result	Test Number	Stress values		
	Fictitious mass			Fictitious mass				Warp stress		
	Edge node	Mid Node		Edge node	Mid Node			min	max	mean
originally (10x10)	47000	6	OBS, mass at the end of formfinding				1	0	3,15	2,3
10x10	4500	50	Converged relatively quickly. (Cables would move during KD)	4500	4000	Converged quicker than before, and at VD=0.05	2	0,77	2,9	2,3
	3500	40	Converged relatively quickly	3500	3500	Converged quicker than before, and at VD=0.05	3	0,77	2,9	2,3
	45000	5	Did not work at all	4000	4000	Converged quicker than before, and at VD=0.05	4	0,77	2,9	2,3
20x20	3500	40	with VD=0.05, bouncing around equilibrium	4000	4000	VD=0.08 takes time but converges. Doesn't for VD=0.05	5	0	3,3	2,3

	7000	80	VD=0.05 Finds form quick	2000	2000	Bounces around	6			
	6000	60	VD=0.05 bounces around equilibrium	8000	8000	VD=0.05: Converges slowly	7	0	3,29	2,34
	8000	100	unbalance	7000	8500	VD=0.05: Almost finds it, but not all the way. VD=0.06 works, but slow!	8	0	3,29	2,33
	8000	150	unbalance	7000	9000	VD=0.05: Slow!! -wasn't worth waiting on	9			
	8500	250	works with VD=0.055 (quite slow though)	6000	8000	VD=0.05 Definitely not an instant correct result, but this gives a fairly good speed ("tuning" of values seems to go on for a while)	10	0	3,27	2,36
	9000	300	works with VD=0.055 (quite slow though)	5500	7000	VD=0.05 : Does not find equilibrium	11			
	10000	400		6000	7500	VD=0.05: converges (slowly)	12	0	3,31	2,33

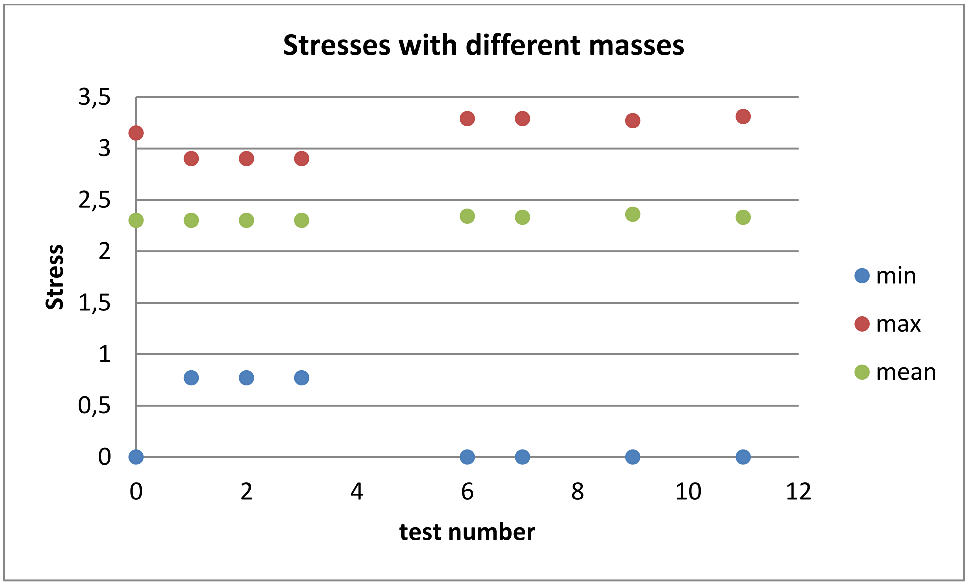
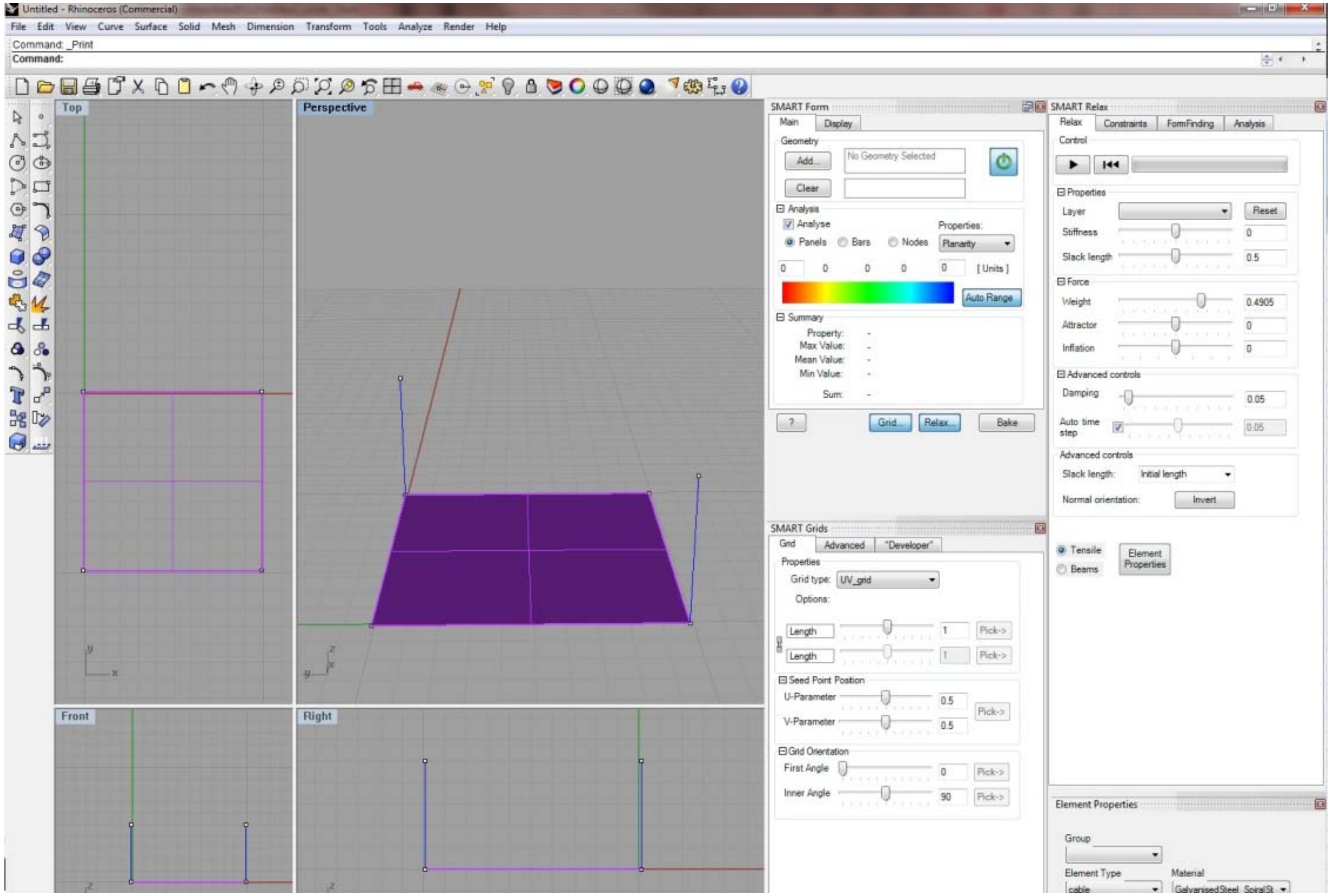


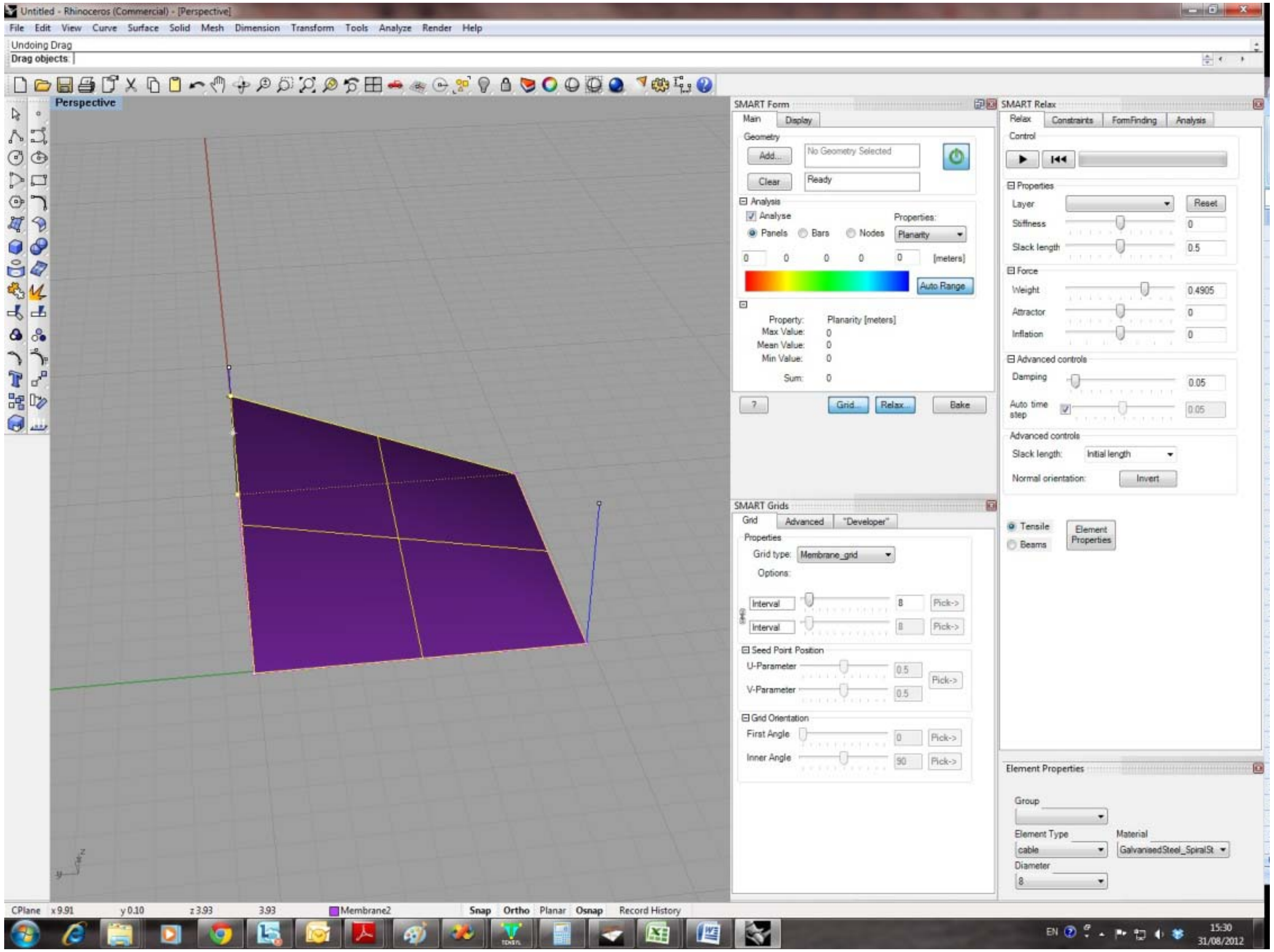
Figure E-1 Plot of the stress values for different fictitious masses

Appendix G User story: Analysing a hypar

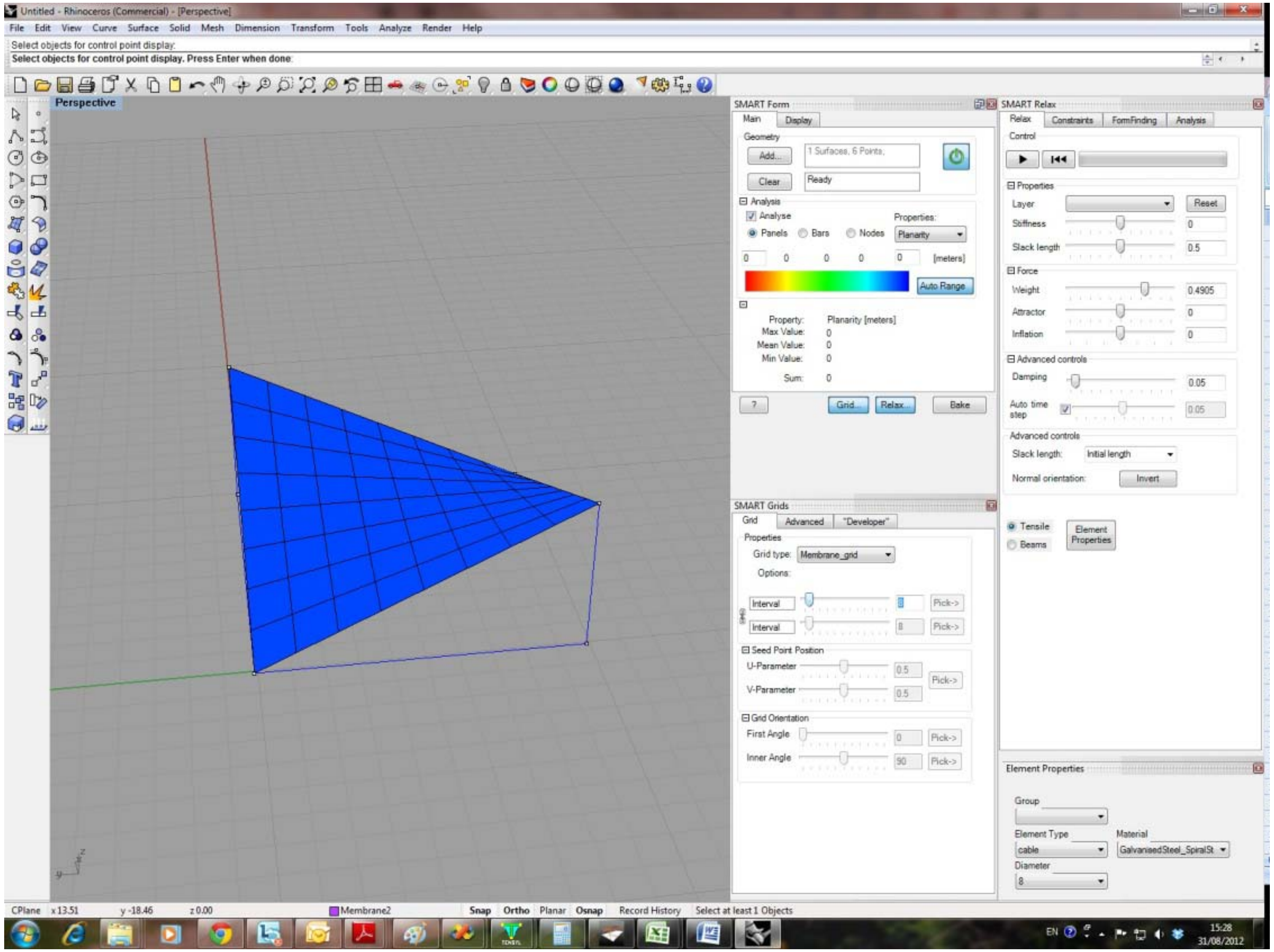
Starting Geometry



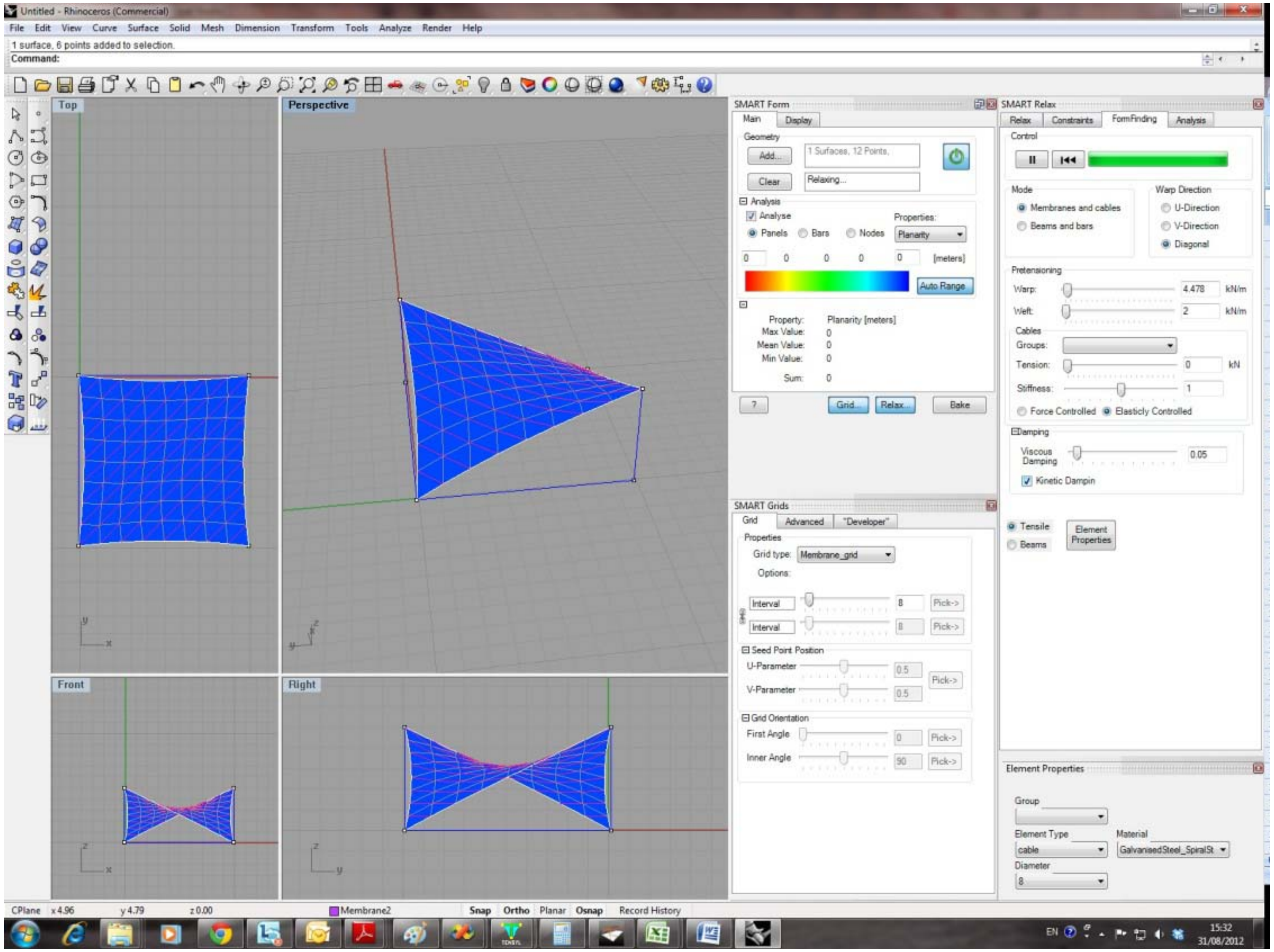
Starting off with a flat surface in Rhino which is then modified, to have the corners in the position wanted for the final geometry.



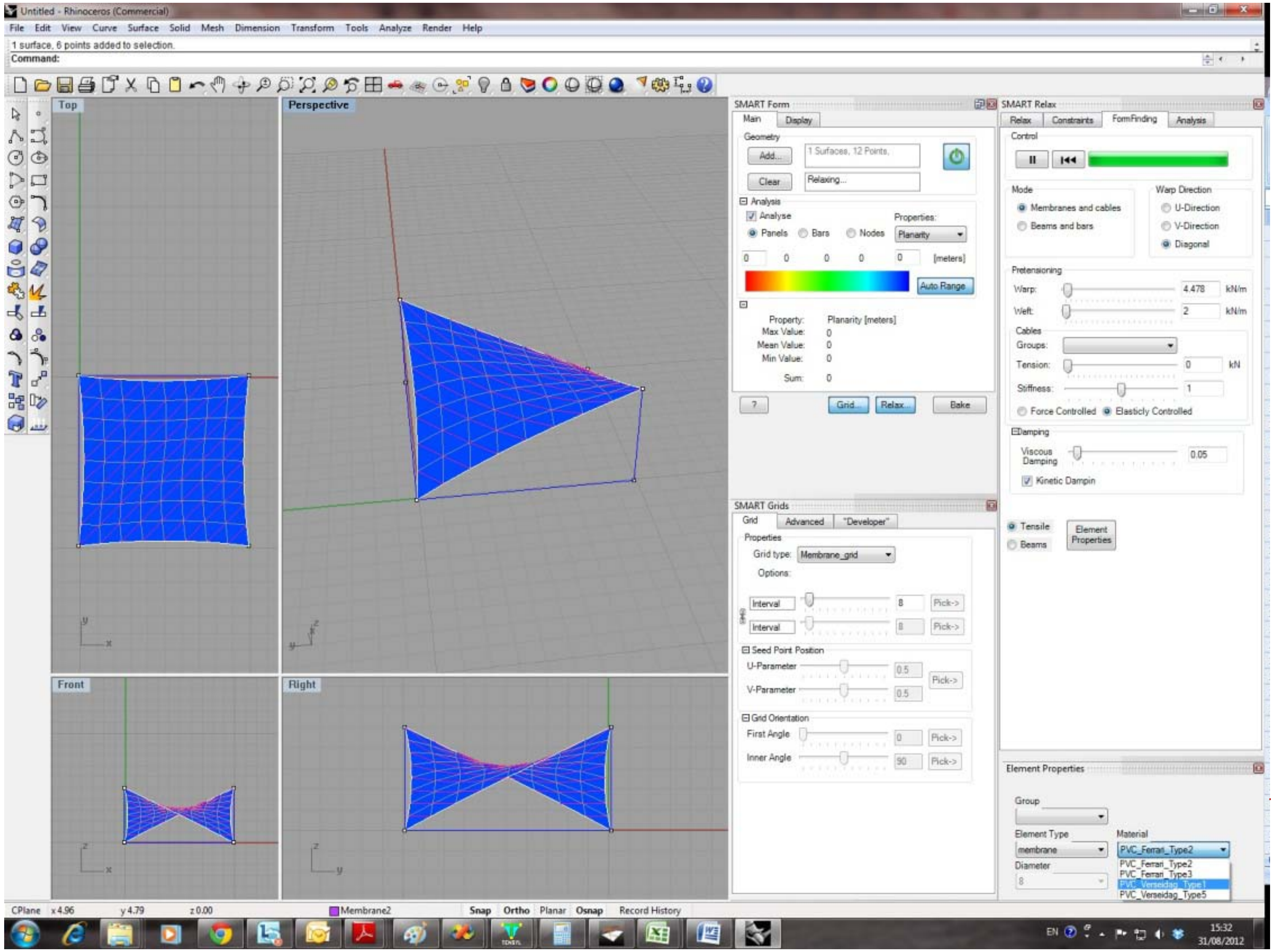
SMART Form

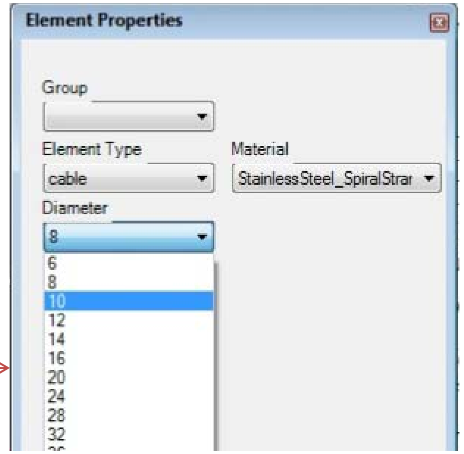


The geometry is added into SMART Form and gridded with a "membrane grid", number of intervals can be changed depending on how exact result one is after.



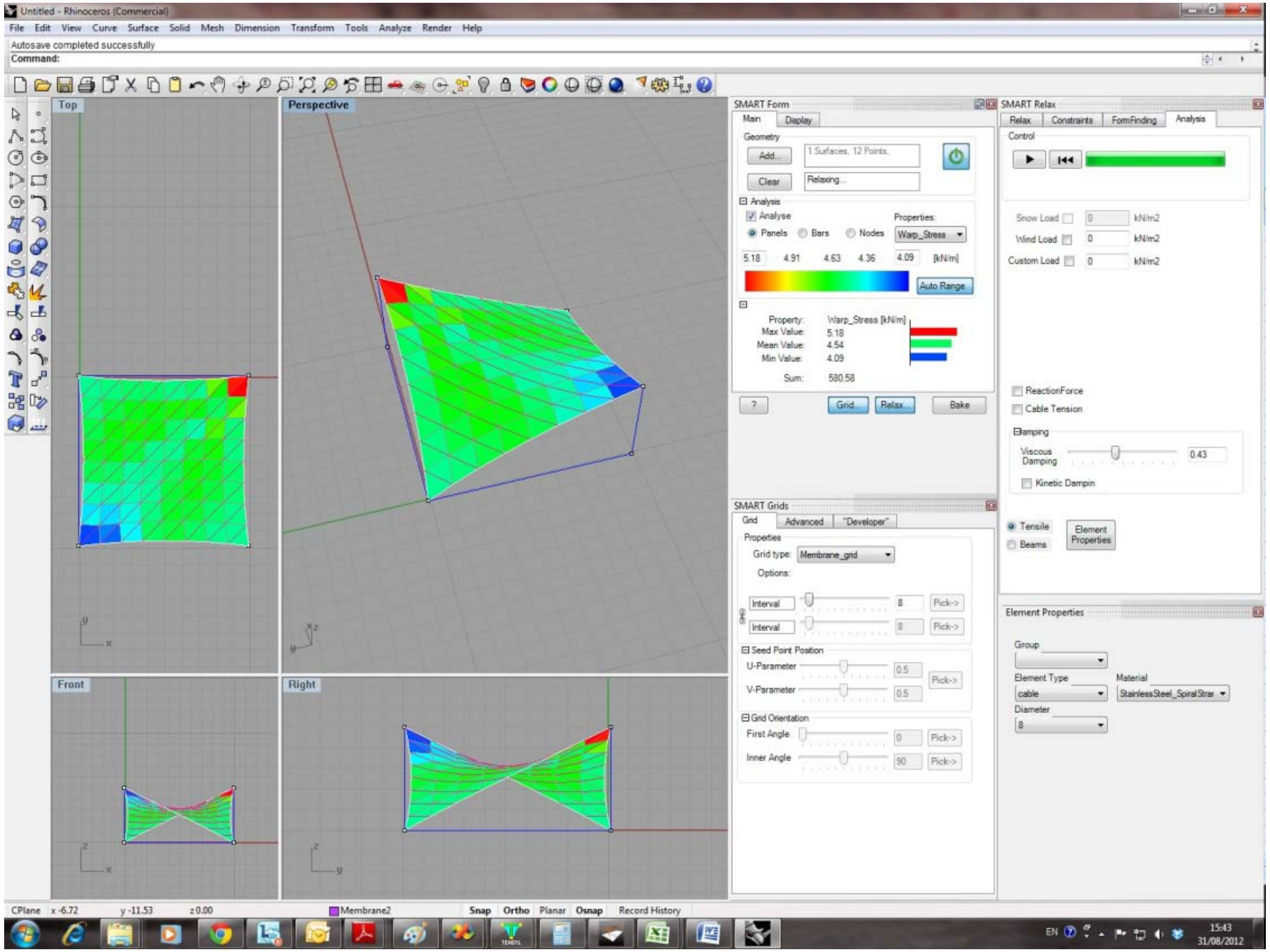
Choosing the tab “Formfinding” and pressing the play button, the formfinding analysis will start. The warp direction can be changed to be any of the directions of the triangular grid. Prestress in warp and weft direction can be changed to get closer to the sought final shape. Prestress in cables can also be added.





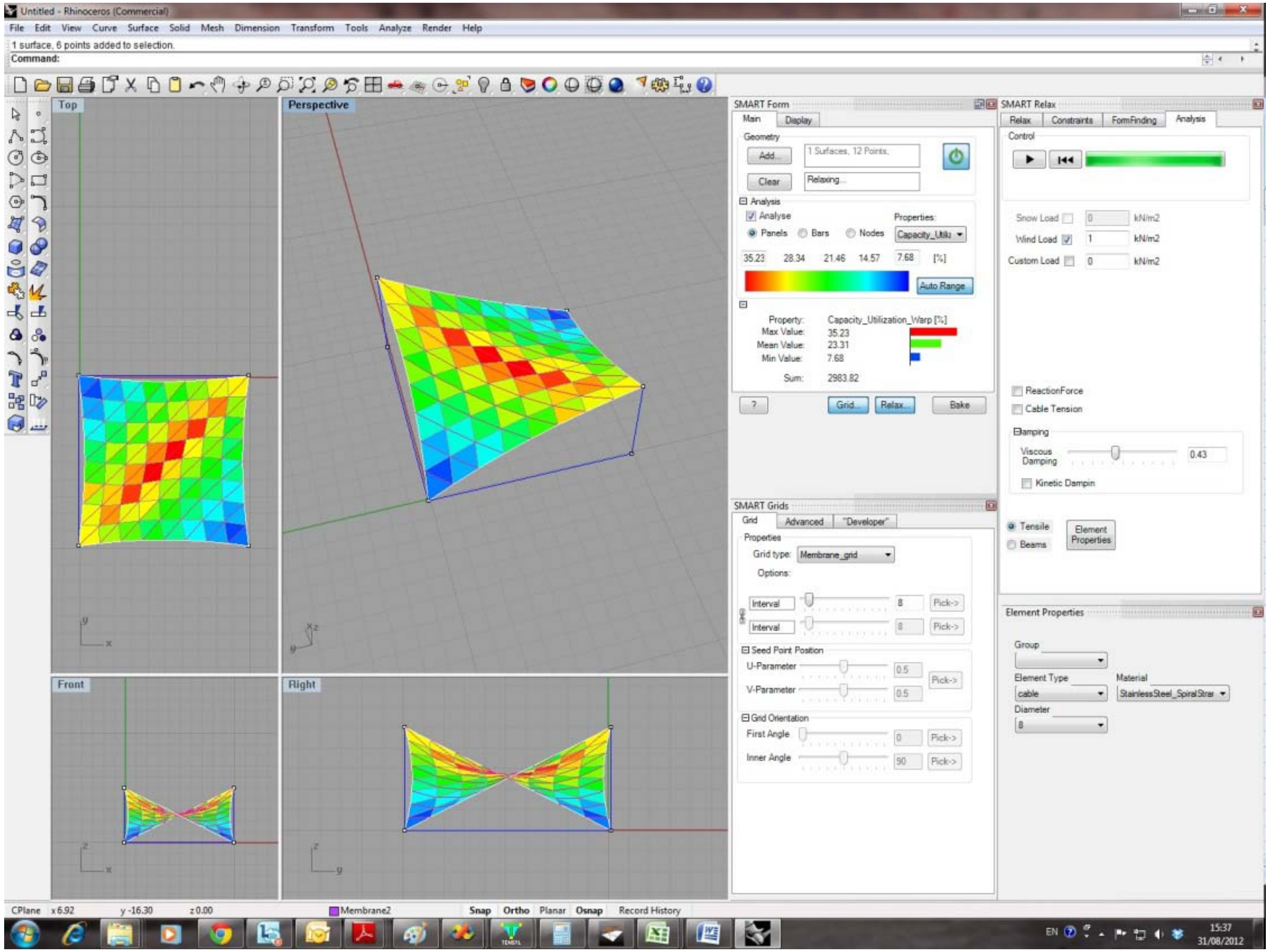
The materials used in the model can be changed at any point during the formfinding or the analysis

Analysis

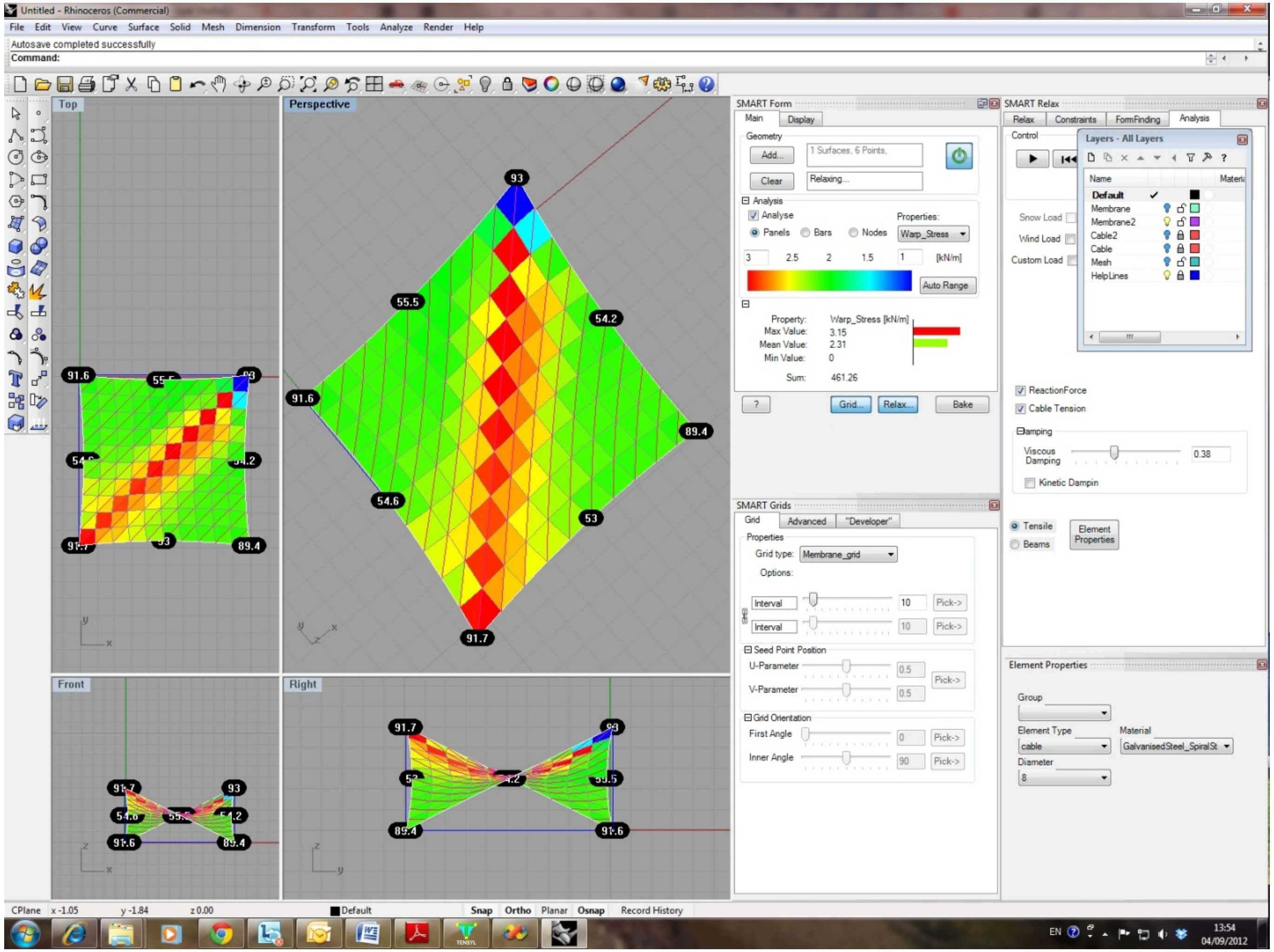


When switching to the "Analysis" tab, gravity is turned on and all elements are analysed with elastic control. The membrane will have the formfound shape as initial geometry, as a result the reaction forces will be larger as the shape moves further away from the minimal surface.

This is an analysis of a hypar with PVC type 2 and edge cables of stainless steel, spiral strand, with diameter ~10mm. The visualisation is set to show membrane stresses in the warp direction.



Adding a wind load of 1 kN/m^2, and changing the visualisation to “utilisation of the tensile capacity” in the warp direction (without safety factors).



The reaction forces at the support can also be displayed if that is of interest.