

Note:
To change the product logo for your own print manual or PDF, click "Tools > Manual Designer" and modify the print manual template.

RhinoMembrane

© 2009 Enter your company name

Title page 1

Use this page to introduce the product

by Enter your company name

This is "Title Page 1" - you may use this page to introduce your product, show title, author, copyright, company logos, etc.

This page intentionally starts on an odd page, so that it is on the right half of an open book from the readers point of view. This is the reason why the previous page was blank (the previous page is the back side of the cover)

RhinoMembrane

© 2009 Enter your company name

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Printed: marzo 2009 in (whereever you are located)

Publisher

...enter name...

Managing Editor

...enter name...

Technical Editors

...enter name...
...enter name...

Cover Designer

...enter name...

Team Coordinator

...enter name...

Production

...enter name...

Special thanks to:

All the people who contributed to this document, to mum and dad and grandpa, to my sisters and brothers and mothers in law, to our secretary Kathrin, to the graphic artist who created this great product logo on the cover page (sorry, don't remember your name at the moment but you did a great work), to the pizza service down the street (your daily Capricciosas saved our lives), to the copy shop where this document will be duplicated, and and and...

Last not least, we want to thank EC Software who wrote this great help tool called HELP & MANUAL which printed this document.

Table of Contents

Foreword 5

Part I Introduction 7

1 Welcome 7

2 Form finding of membrane structures by the Updated Reference Strategy 7

Part II Command List 13

1 Node Properties 13

2 Mesh Node Properties 13

3 Mesh Properties 14

4 Curve Properties 15

5 Mesh face properties 16

6 Edge Properties 17

7 Boundary detection 17

8 Form Find 17

9 Clear Solution 17

10 Options 18

11 About 19

12 Stress Plot 20

Part III Tutorials 23

1 Creating a minimal surface 23

2 Membrane structure with edge cables 27

3 Pneumatic structure 33

4 Conical shape with anisotropic linear prestress distribution 37

5 Tensegrity 42

Index 0

© 2009 Enter your company name

Foreword

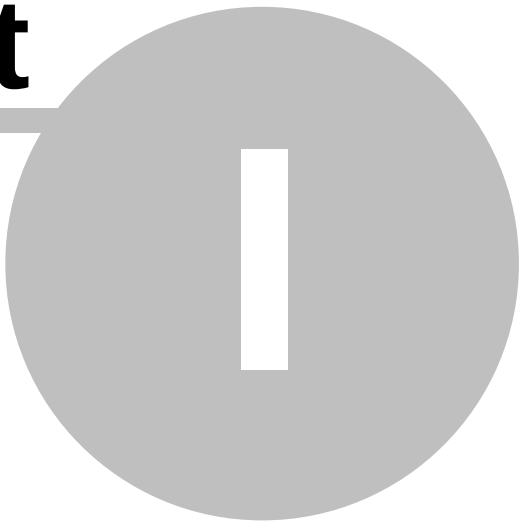
This is just another title page
placed between table of contents
and topics

© 2009 Enter your company name

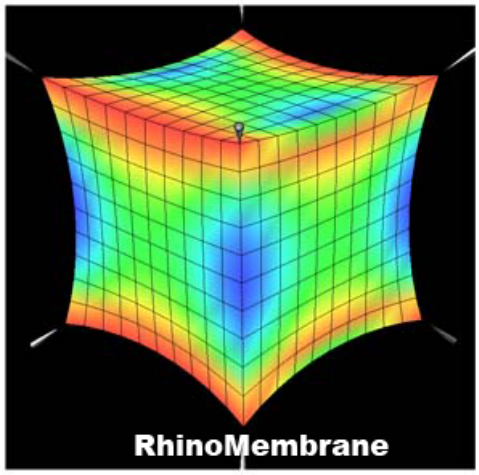
Top Level Intro

This page is printed before a
new top-level chapter starts

Part



1 Introduction



1.1 Welcome

RhinoMembrane

is a application with one of the most powerful form-finding strategies available today (see URS details)
Using a Fea (Finite Element Analysis) model it is able to relax any mesh into a stressed surface in equilibrium with positive
stress fields only

Using sparse matrix strategies it is fast and reliable and any shape from conventional hypar , minimal surfaces
and pneumatic cushions can be modeled.

What else can it do it up to you !

Best Wishes
Kai-Uwe Bletzinger
Gerry D'Anza , Johannes Linhard

1.2 Form finding of membrane structures by the Updated Reference Strategy

General

© 2009 Enter your company name

Membrane structures require due to their extreme slenderness a different design process than conventional structures in engineering: Since membranes only work in tension (compression will cause wrinkling in the structure), the structure has to be shaped in such a way that it can bear the main load cases only by tensile forces. To guarantee this essential requirement, the shape of the structure has to be determined by a form finding procedure: Instead of prescribing a material field $\mathbf{X}$ (i.e. the amount and position of material in space which composes the structure), an admissible stress field σ is prescribed for which the equilibrium shape $\mathbf{x}$ is then determined.
In detail, the user has to specify the prestress distribution in the membrane and the edge cables (*Pre-Tension*) as input for the form finding process. For pneumatic structures such as air-inflated cushions it is even possible to the pressure p in the form finding process. Additionally, sufficient boundary conditions such as fixed points or fixed edges have to be specified to obtain a unique solution for the equilibrium shape.

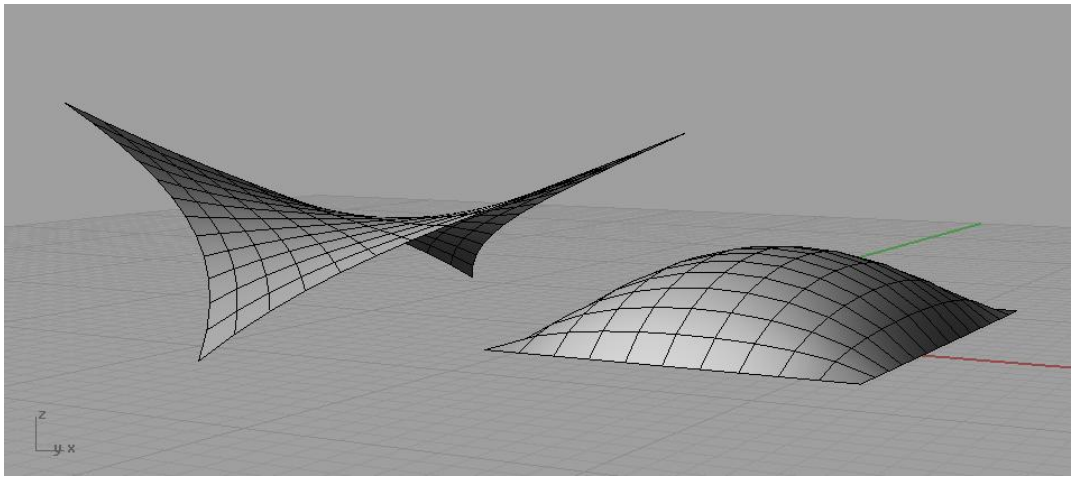


Fig. 1: Membrane with edge cables and inflated cushion

Updated Reference Strategy

The governing equation for form finding is given by the principle of virtual work

$$\delta w = t\int_a \sigma : \frac{\partial(\delta \boldsymbol{u})}{\partial \boldsymbol{x}} da = t\int_a \sigma\, \delta \boldsymbol{u}_{,x}\, da = 0 \qquad (1)$$

which states that the virtual work of a stress field in equilibrium vanishes. σ is the prescribed *Cauchy stress tensor* which acts on the surface in equilibrium, $\delta \boldsymbol{u}_{,\boldsymbol{x}}$ is the derivative of the virtual displacement with respect to the geometry of the actual

© 2009 Enter your company name

surface. The thickness of the membrane is denoted by t.
However, a straightforward numerical solution of the equilibrium equation (1) is impossible since the obtained system matrix for the evaluation of the unknown discretization parameters would be singular. The answer to this otherwise irritating property lies in the inverse nature of the given problem: We are looking for the discretized equilibrium surface with a certain topology of the discretization parameters, which reveals in a finite element context as the mesh, particularly the nodal positions. It turns out, that these nodes can be moved around on the corresponding equilibrium surface while still maintaining a valid discretization of the sought-after surface ("floating meshes"), i.e. the equilibrium equation is always fulfilled. In other words, no unique discretization of the surface exists, which leads to a singular system matrix and prohibits the direct solution. To overcome this problem, a regularization method for this inverse problem has to be applied.

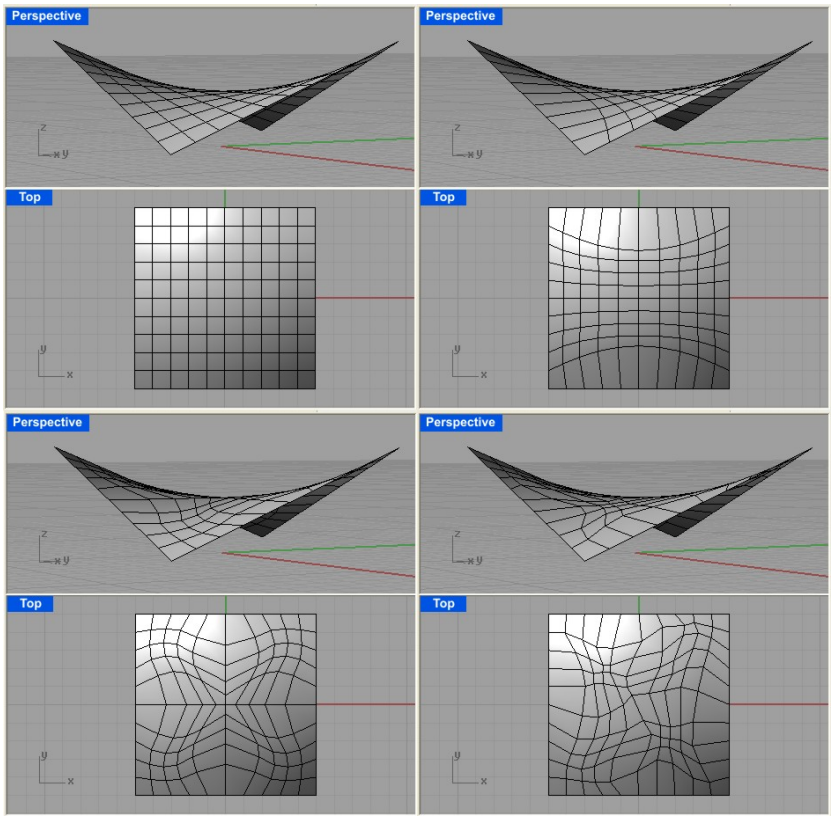


Fig. 2: "Floating meshes" for hypar-like membrane structure

The implemented regularization technique "Updated Reference Strategy" is a powerful method which is consistently derived from nonlinear continuum mechanics: Its main idea is to replace the original singular problem with a related, yet well-posed problem: Instead of assuming the Cauchy stress tensor σ of the unknown equilibrium surface to be given, we prescribe the 2nd Piola-Kirchhoff stress tensor $\mathbf{S}$ referring to the arbitrary starting geometry. These two different stress measures are linked by the

© 2009 Enter your company name

deformation gradient **F**, which again is a measure for the occurring displacements between the reference and actual geometry:

$$\boldsymbol{S} = \det \boldsymbol{F}\ \boldsymbol{F}^{-1} \cdot \boldsymbol{\sigma} \cdot \boldsymbol{F}^{-\mathrm{T}}$$

(2)

It is important to note that only the Cauchy stress tensor represents real physical stresses, as they act on the final configuration. The 2nd Piola-Kirchhoff stress tensor (PK2) refers to the undeformed geometry and is a quantity, which is merely used for calculation purposes. If the actual configuration is identical with the reference, the two stress measures are yet identical.
The derivation starts from Eq. (1), in which the "pulled-back" Cauchy stress tensor σ is expressed by the 2nd Piola-Kirchhoff stress tensor **S**.

$$\delta w = t \int_A \left(\boldsymbol{F} \cdot \left(\det \boldsymbol{F}\ \boldsymbol{F}^{-1} \cdot \boldsymbol{\sigma} \cdot \boldsymbol{F}^{-\mathrm{T}} \right) \right) : \delta \boldsymbol{F}\, dA = t \int_A \left(\boldsymbol{F} \cdot \boldsymbol{S} \right) : \delta \boldsymbol{F}\, dA$$

(3)

By using this equation, we are thus able to calculate a unique equilibrium surface for the given PK2 stress state. The resulting Cauchy stresses, however, will differ from the targeted stress state. This necessitates an extension of the algorithm by another iteration loop: The newly obtained actual configuration, which is closer to the final equilibrium shape, is set to be the reference configuration for the next shape finding step. This constitutes an essential part of the proposed procedure which is therefore called "Updated Reference Strategy" (URS). After this "reference update", the 2nd Piola-Kirchhoff stress tensor is again specified according to the target stress field in order to obtain a well-posed problem and to be able to determine a new actual configuration. And as the corresponding displacements between actual and reference configuration are becoming smaller, the difference between the PK2 and Cauchy stresses is consequently reduced and the solution converges to the one of the unmodified problem. Hence, the modification of the original problem by the stabilization fades out.
This extremely powerful method is capable of handling arbitrary stress states with or without external loading. Furthermore, it can be used with any discretization and can be applied in the same manner.

Special care has to be taken if warp and weft shall be pretensioned with different magnitude, and therefore an anisotropic prestress distribution is prescribed: It can be mathematically proven, that the only surfaces which have a constant anisotropic prestress field are structures with zero Gaussian curvature (as a plane or cylinder). For all doubly curved structures, which is the normal case for membrane structures, slight deviations from the target stresses have to be accepted as no mathematically

© 2009 Enter your company name

exact solution exists for this problem. As each solution at every form finding step represents a physically stable solution itself, the user simply has to adjust the number of executed form finding steps so that the solution matches his needs of geometrical shape and accuracy in the arising stresses.

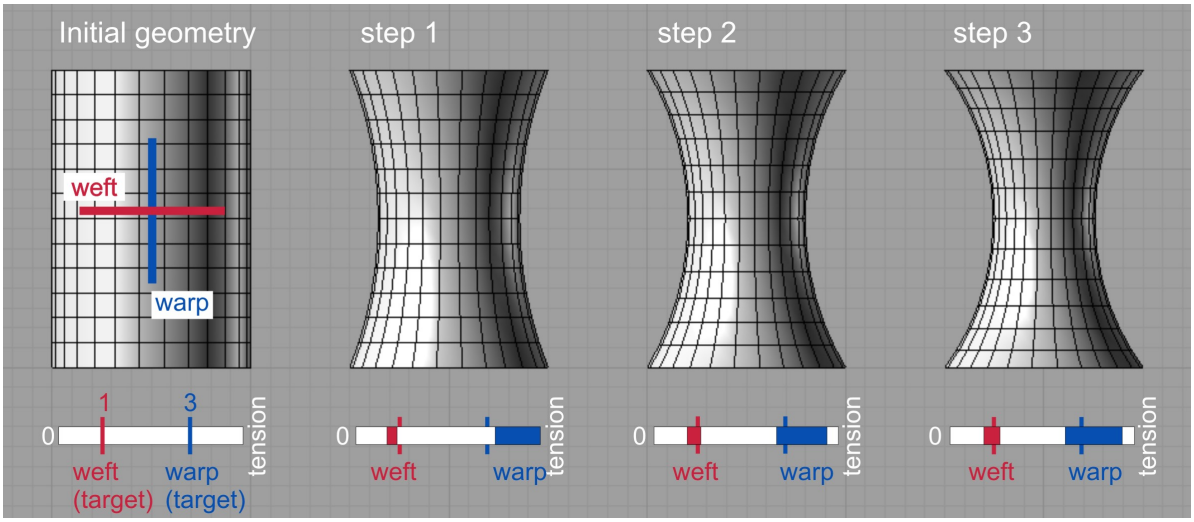


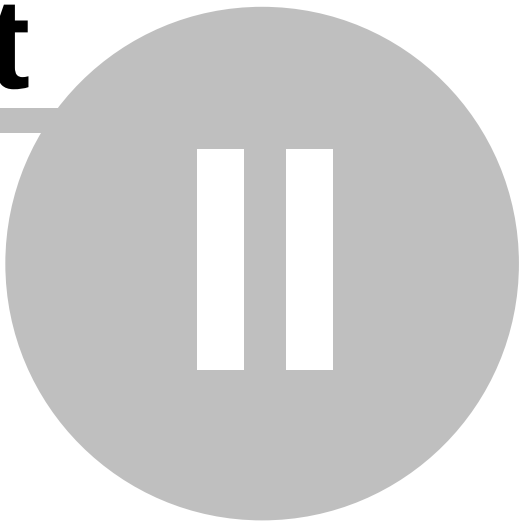
Fig. 3: Stress deviation at a catenoid with anistropic pretension

© 2009 Enter your company name

Top Level Intro

This page is printed before a
new top-level chapter starts

Part



2 Command List

Enter topic text here.

2.1 Node Properties

_fnNodeProps



Create some Rhino Point objects before using this command.
The command will prompt to select points and the asks for a restraint.



dx = fix movement in x axis direction
dy = fix movement in y axis direction
dz = fix movement in z axis direction

This command is used to restrain space positions generally connected to cable elements
for mesh node restraints look at the similar command Mesh Node Properties

2.2 Mesh Node Properties

_fnMeshNodeProps



Same as Node properties but works for mesh nodes

After calling the command select in the restrain nodes dialog
the degrees of freedom to keep fixed.

© 2009 Enter your company name

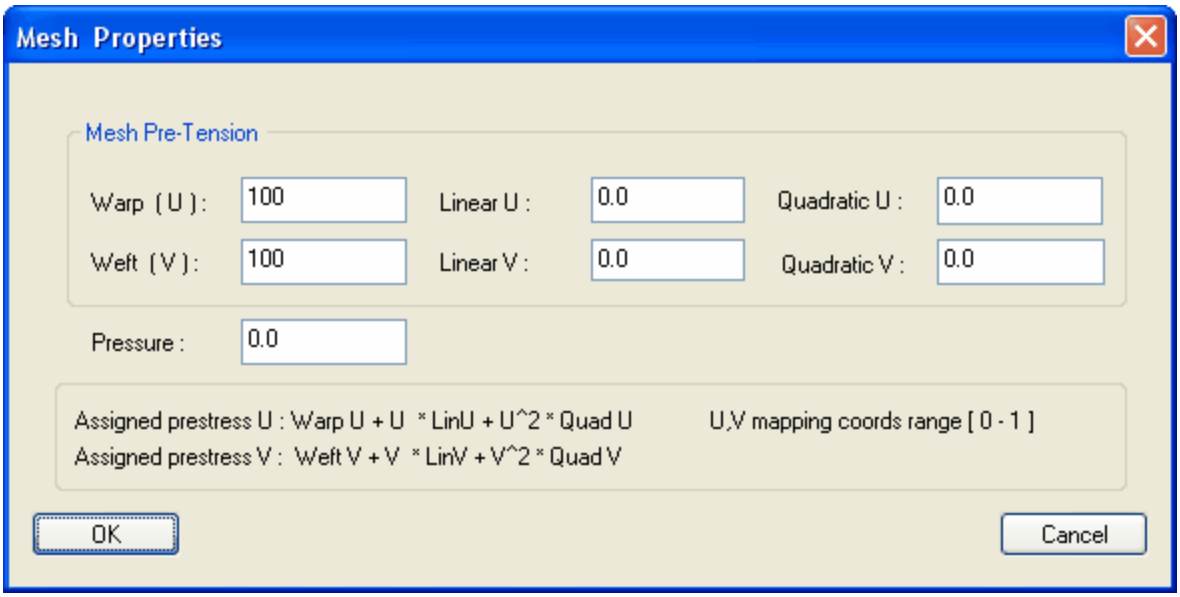


2.3 Mesh Properties

_fnMeshProps



Edit or assign a Warp-Weft pretension to mesh objects



The assigned prestress will be computed with the formula :

Warp = Wu + U x Lin U + U^2 x Quad U (same for weft)
where U.V are the texture coordinates of the mesh .

This means basically that if we want to assign a linear or quadratic prestress function it will be necessary to apply U-V mapping coordinates to the mesh. The bitmap U-V-MAP.JPG or U-V-MAP2.JPG found in the RhinoMembrane install folder can help

© 2009 Enter your company name

in visualizing the U-V mapping coordinates.

The help function **fnPlotPrestress** is a helper display function that will plot on video the computed prestress values for each mesh face.

Look at the Conical shape tutorial for a example on how to use linear prestress function.

Pressure value other than zero will inflate the mesh in the direction of mesh normal's

Note:
Only meshes to which a warp-weft pretension has been assigned will be processed by the form finder

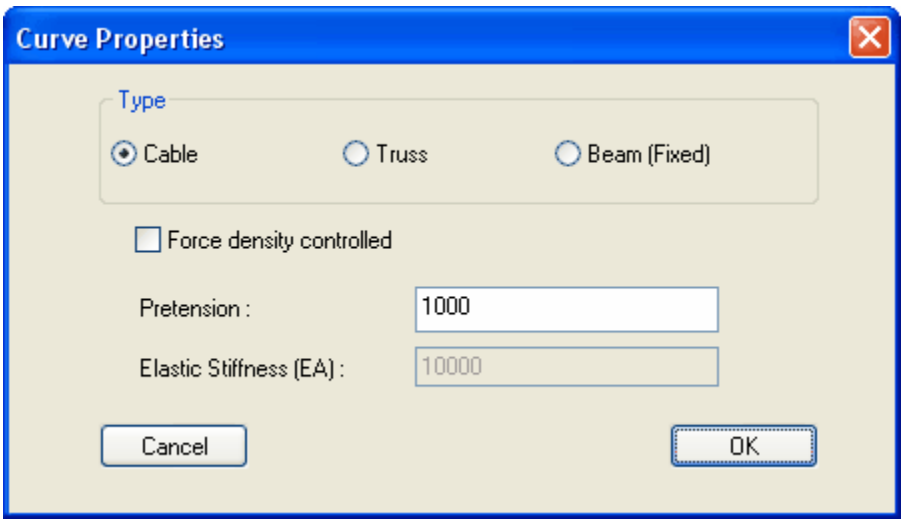
2.4 Curve Properties

_fnCurveProps



Edit/assign a pretension value to a line/poly line.
With this command we can add cable-truss elements to our model.

The dialog will prompt for :



© 2009 Enter your company name

Pretension : force value
Elastic stiffness (EA) : stiffness for truss type elements
Force Density : Enable force density control (Only for cable elements)

Type

- cable = behaves as a cable with pretension
- truss = not implemented yet
- beam = fixes both end points

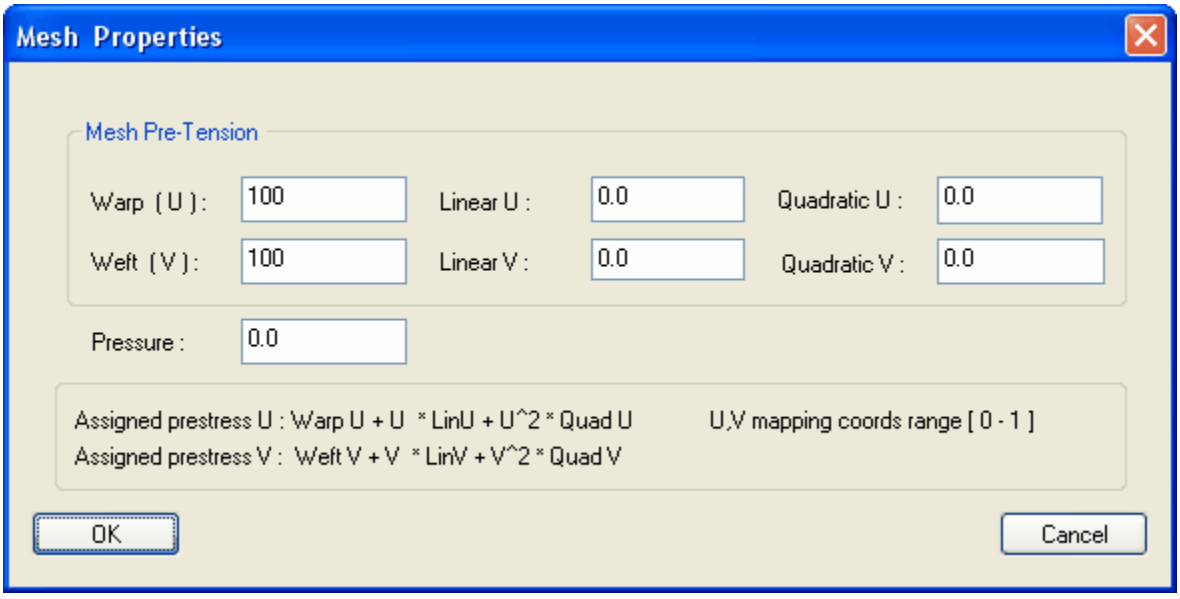
2.5 Mesh face properties



fnMeshFaceProps

With this command we can assign to local faces a Warp-Weft prestress

The command will prompt to select mesh faces and afterwards
the mesh warp-weft prestress dialog compares



Note that only Warp (U) and Weft (V) properties are valid
for local mesh property assignment.
The values applied locally with this function will override other values applied using Mesh properties.

© 2009 Enter your company name

2.6 Edge Properties

_fnEdgeProps



Same as Curve properties except that it works
over mesh edges.

Note:
When we use this command a new (cable-truss-beam) element is added to the model
so it can be used to simulate cable's running in pockets or fixed boundary edge's

2.7 Boundary detection

fnBoundMesh



This command will detect all free edges of a mesh object
and add a cable(truss,beam) element automatically

Useful for creating cable edges or fixing edges (beam option)

2.8 Form Find

fnFormFind

Here we launch the form finder

The solution will be a number of meshes/curves cloned from the original objects
and placed on the current layer.
We suggest to create and activate a new layer before invoking the command

After form finding the rendered view with plot stresses over the surface (only warp direction at the moment)

2.9 Clear Solution

_fnClear



© 2009 Enter your company name

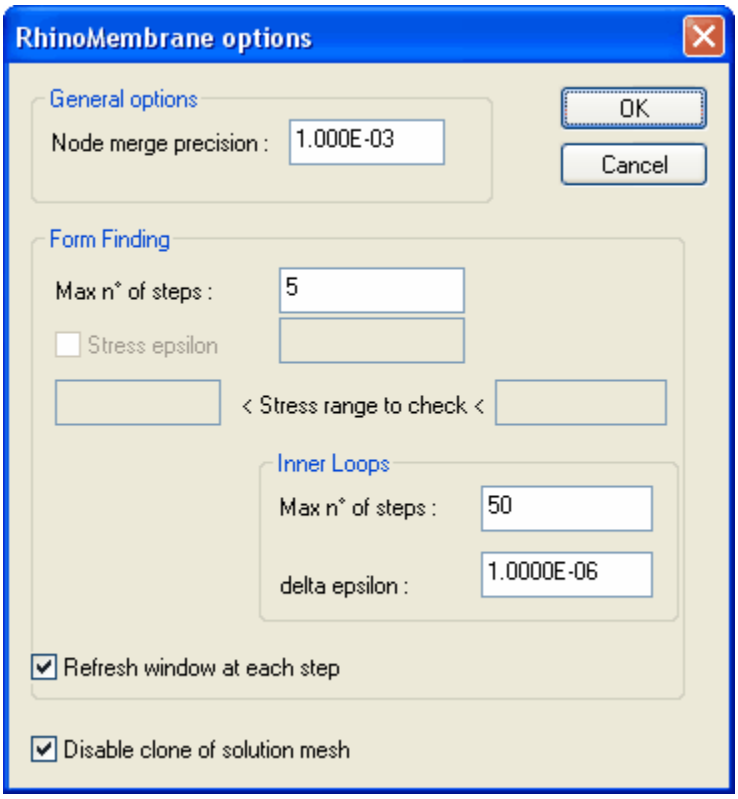
This command will clear all solution data and clear any data linked to meshes,nodes and lines

2.10 Options

fnOptions



Rhino Membrane options dialog
Here we set all options used to relax the mesh into a stressed surface



General options :

Node merge precision :
More than one mesh can be relaxed and this option will merge in a single node position all mesh
vertex's that are in a small sphere with radius = merge precision

Form Finding :

© 2009 Enter your company name

Max n° of steps
Form finding is a typically non-linear problem
so here we set the number of iterations to find the solution

Inner loops :
Max n° of steps
for each outer step a n° of inner solutions are computed to converge
the shape with assigned pretensions
the inner loops will stop when 2 consecutive solution are within the epsilon tolerance.
This means briefly that all distances between nodes at solution i and solution i-1 are smaller that the epsilon value

Refresh Window at each step :
This option will redraw the current geometry at each outer loop

Disable clone of solution mesh (not implemented)

2.11 About

RhinoMembrane (tm) is a Rhinoceros application
developed by :

Gerry D'Anza & Johannes Linhard
With URS theory support from prof. Bletzinger

© 2009 Enter your company name

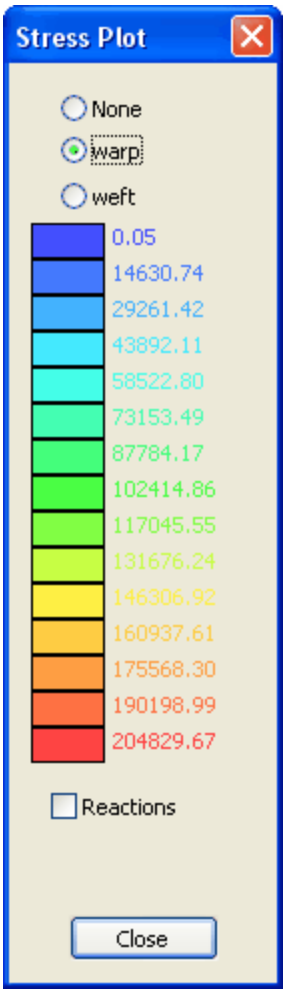
2.12 Stress Plot

!_fnAnalysis



After Form-Finding we can ask to plot stresses in both warp and weft direction using the stress plot command.

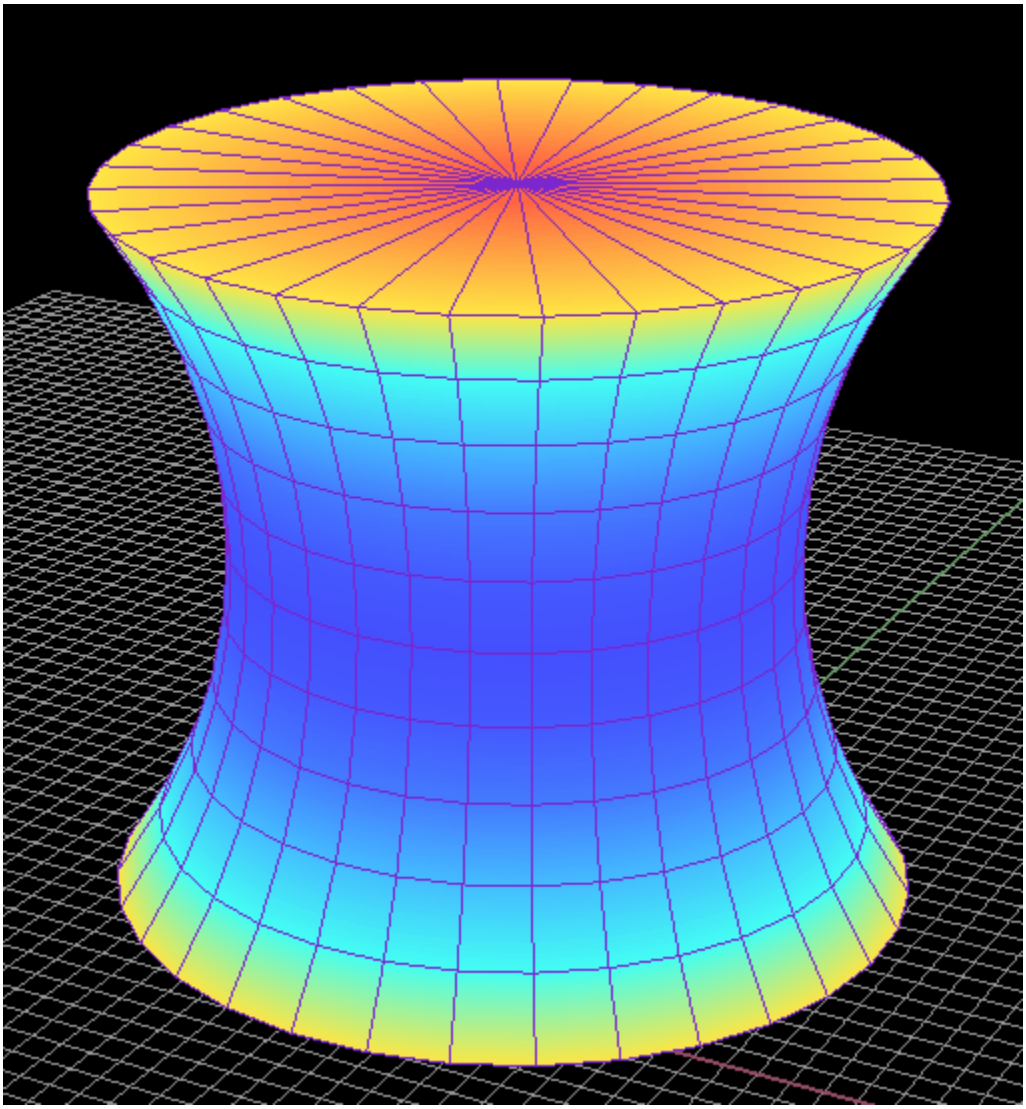
Be aware that the current Rhino Viewport should be set in Render mode to view stresses



Select from the dialog a stress option
to see them plotted on the final form found mesh

The Reactions check box enables plot of nodal reaction forces
The maximum reaction force will have a length of 1 unit and other forces scaled as necessary

© 2009 Enter your company name

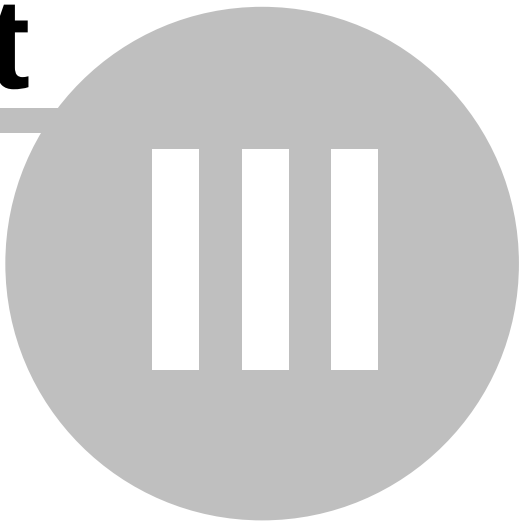


© 2009 Enter your company name

Top Level Intro

This page is printed before a
new top-level chapter starts

Part



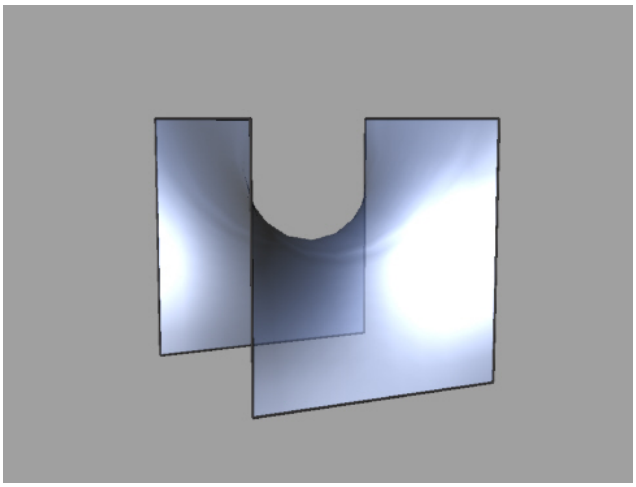
3 Tutorials

Enter topic text here.

3.1 Creating a minimal surface

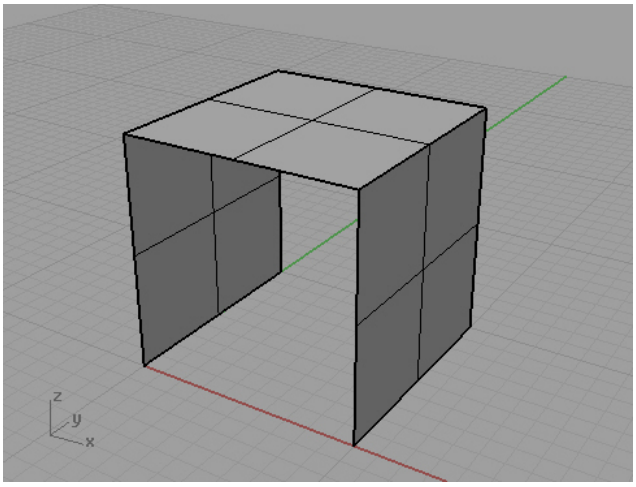
In this tutorial you will learn how to create a **minimal surface within fixed boundaries**. As an example we will start with the well-known

Scherk's surface.

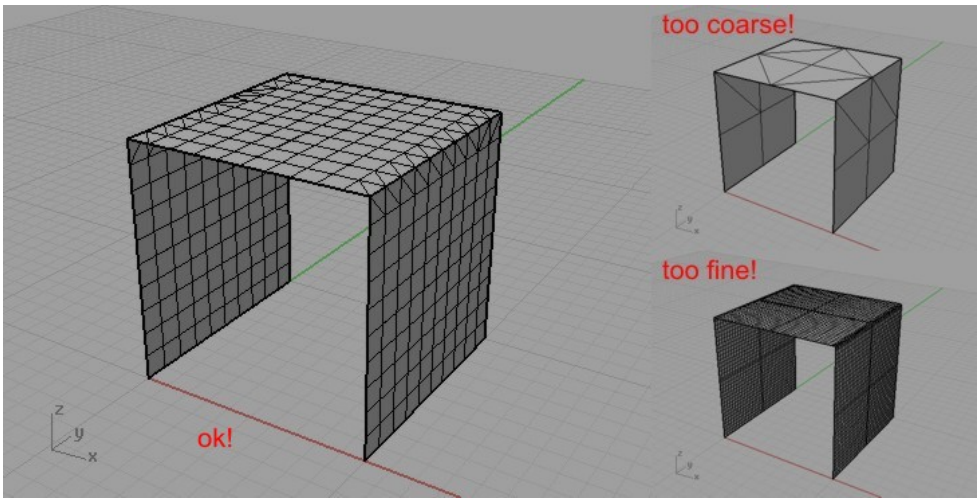


At first, we have to generate a NURBS surface, which serves as initial geometry for the form finding iteration. The boundaries of the initial surface have to be identical with the ones from the intended Scherk's surface, as they are fixed throughout the iteration. The interior of the surface can be shaped arbitrarily – in order to facilitate convergence, it is yet advantageous to shape it as close as possible to the final surface.
For the initial geometry generate three adjacent rectangles (e.g. with the **SrfPt** command) and **Join** them together so that they will form one entity.

© 2009 Enter your company name



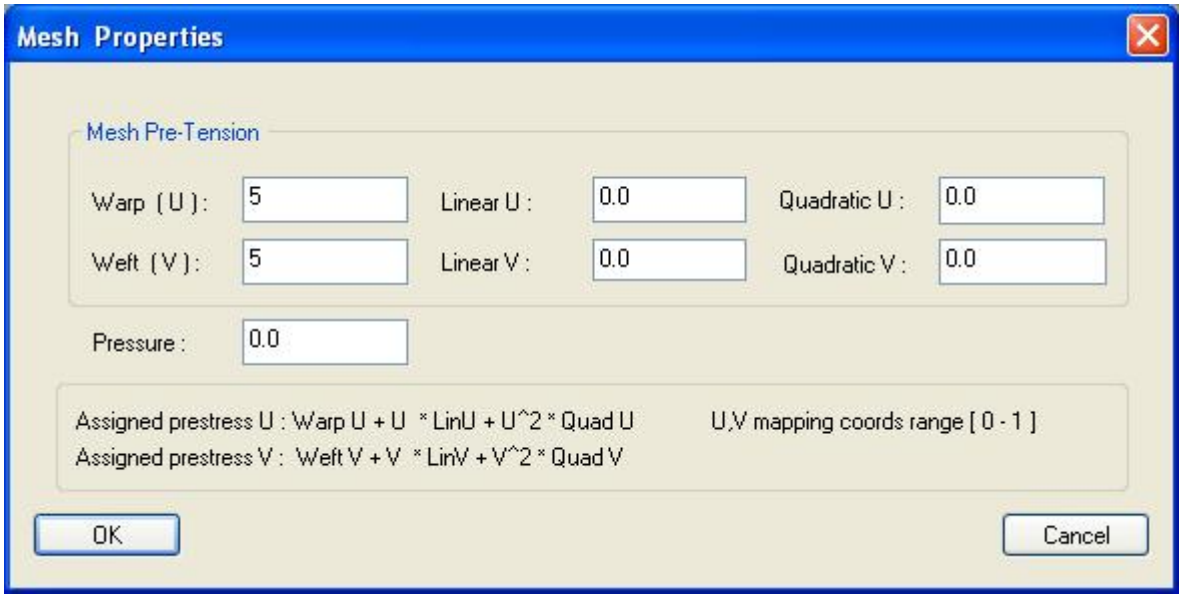
Now you have to convert the NURBS patch into a polygon **Mesh**. As only linear/bilinear interpolation is used for meshes, pay attention to discretize your geometry with sufficient elements.
If you are using too coarse mesh, the solution will not be accurate, if you are using way too many elements, your computer may be running out of memory.



In order to avoid gaps in the mesh, we have to weld all vertices together. This is necessary, because Rhino generates individual vertices for each individual surface – even if joined together.
To do so, enter **WeldVertices** and select all vertices of our mesh.
In the next step, we have to assign the problem parameters. We start by assigning a prestress distribution to the mesh: As we want to generate a minimal surface, the pretension in warp and weft direction *must* be equal (soap film analogy).
Set the mesh properties by clicking on

© 2009 Enter your company name

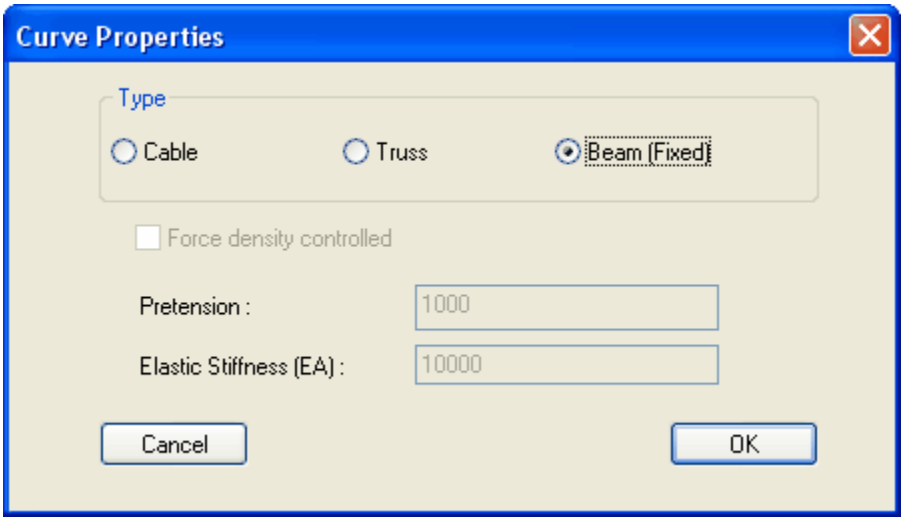
 or entering the command **fnMeshProps**. Select the mesh and enter in the popup window the pre-tension for warp and weft (e.g. set both to the value of 5).



The boundaries for Scherck s surface are fixed. We could consequently restrain each node at the boundary

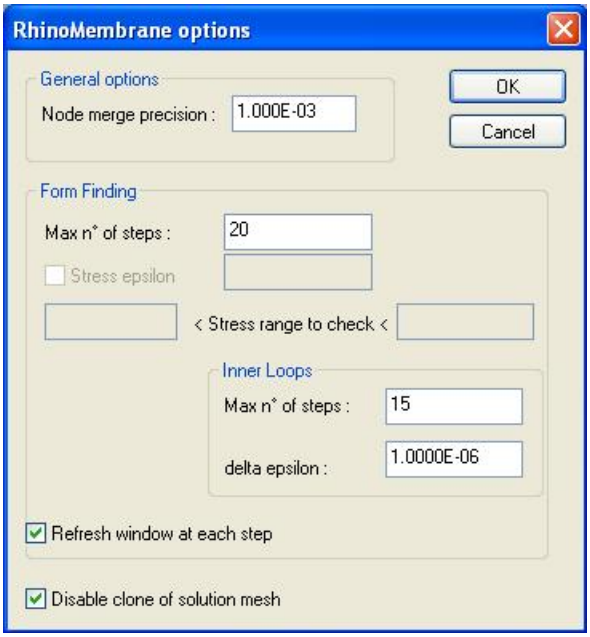
( or **fnMeshNodeProps**), but it can be easier done by creating a fixed boundary mesh. Click on

( (**fnBoundMesh**), select the mesh and choose the option **Beam (fixed).**



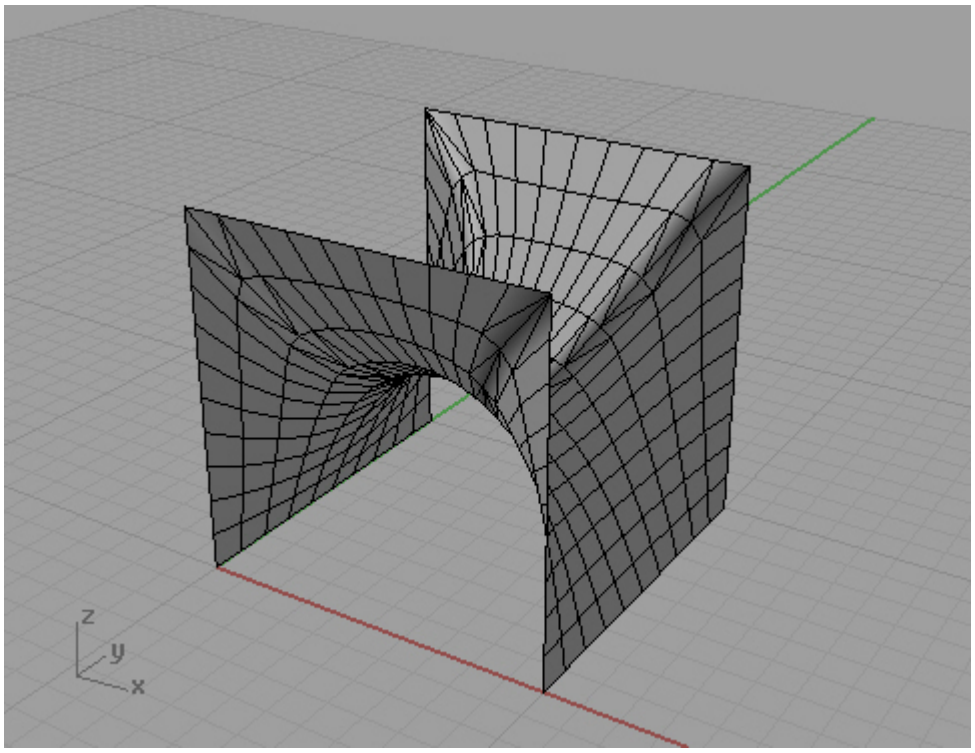
© 2009 Enter your company name

This will generate a poly line, whose vertices are all fixed.
Next, we have to set the number of form finding steps we want to perform: Click on  or enter **fnOptions**: Since we are starting from a very rough approximation of our final geometry, we have to perform a slightly higher number of form finding steps. Therefore, set the maximum number of form finding steps to 20. If your approximation of the final surface is better, fewer steps will be enough.



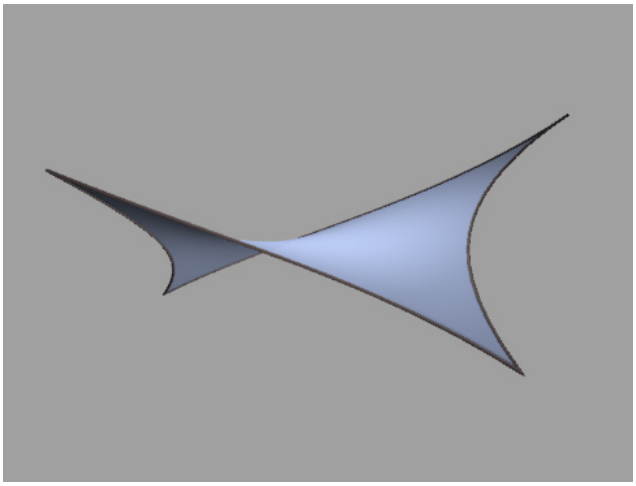
To start the form finding procedure, click on  (**fnFormFind**). You can see how the structure converges to the final minimal surface. At the end of the form finding procedure, a new mesh object will created on the active layer. The geometry of this surface represents the equilibrium geometry for the prescribed pre-tension (in our case Scherk s minimal surface).

© 2009 Enter your company name



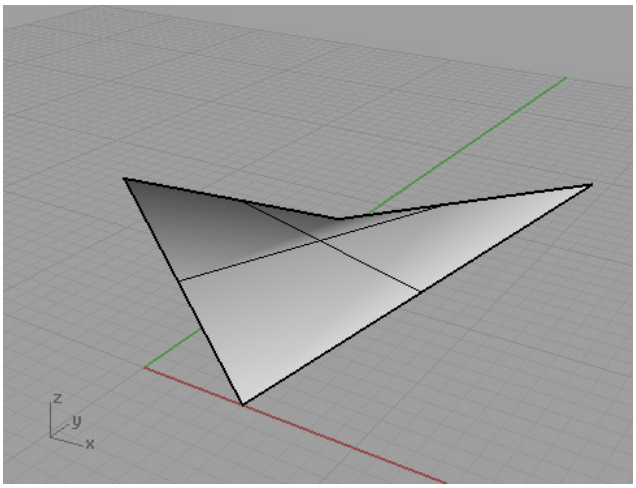
3.2 Membrane structure with edge cables

In this example, we will create a classic four point sail: The boundary of the prestressed membrane is supported by four edge cables,
which meet at four fixed corner points.

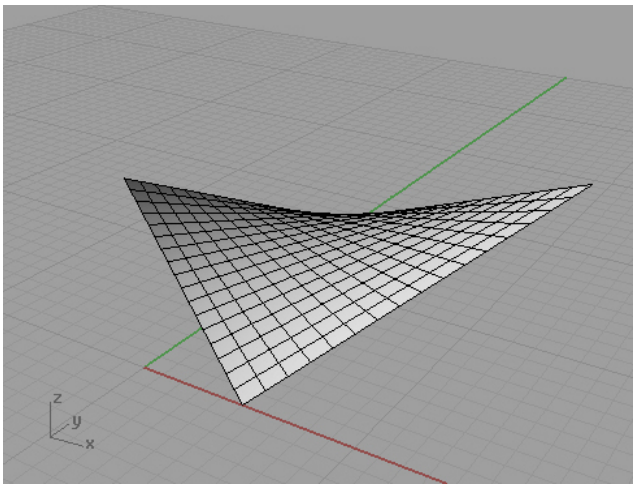


As in Tutorial 1, create an initial geometry for your four point sail using NURBS. The four corner points have to have their final position,
as they will stay fixed throughout the form finding.

© 2009 Enter your company name



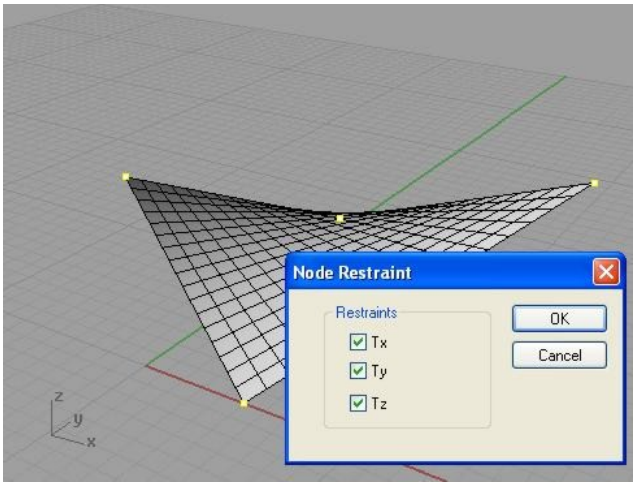
Convert the NURBS patch to a **Mesh** object.



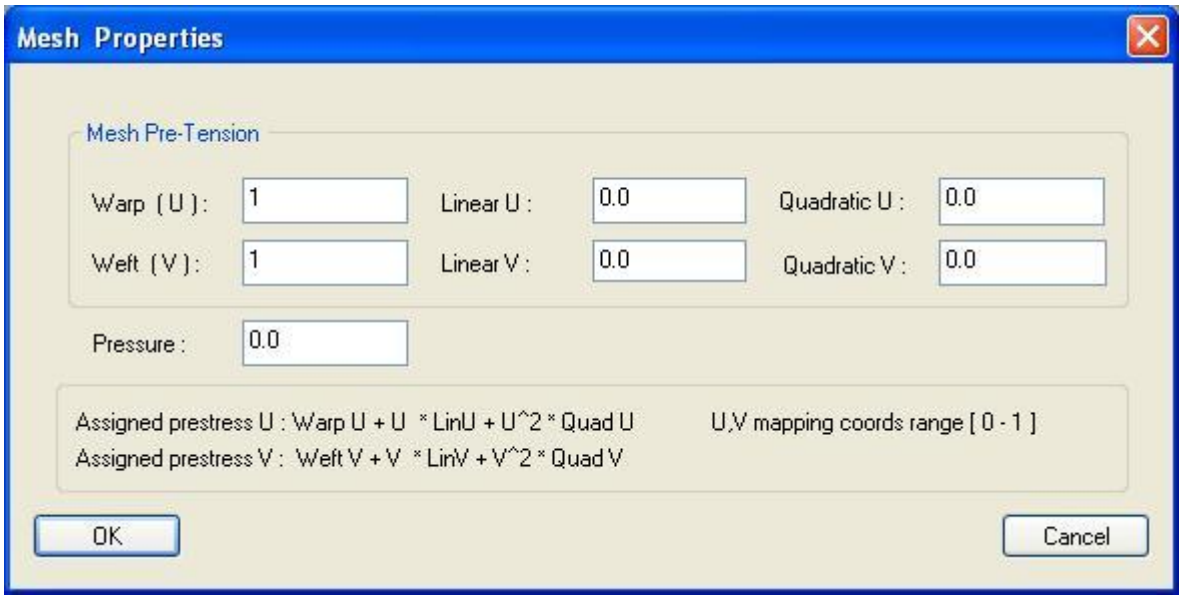
Next, assign the parameters for our form finding problem. Let's start with the boundary conditions:

Fix all three spatial degrees of freedom of the four corner nodes ( or **fnMeshNodeProps**):

© 2009 Enter your company name



Set the pre-tension for the membrane ( or **fnMeshProps**).
Choose an isotropic prestress distribution of 1 in both warp and weft direction.



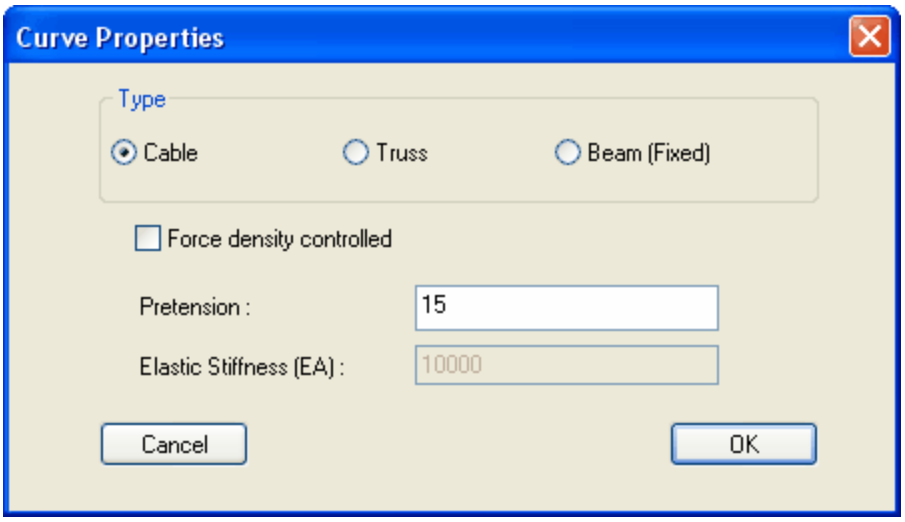
Now we will create the edge cables: We could do this step by step by assigning a pre-tension

( or **fnEdgeProps**) to each individual mesh edge.
But as this is a pretty time consuming task, it is easier to create a *cable boundary mesh* using

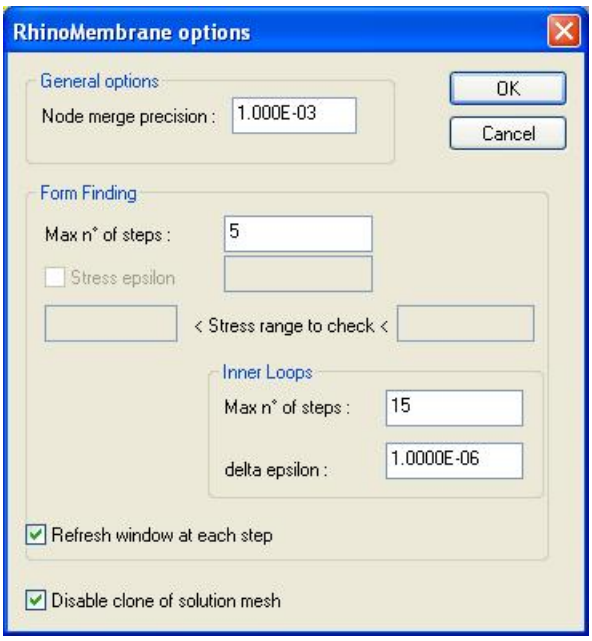
 or **fnBoundMesh**. The necessary pre-tension in the edge cables can be determined by the pre-tension in the membrane multiplied with the desired radius of the edge cables (Boiler formula). Let 's assume we want to have a radius R = 15.

© 2009 Enter your company name

Consequently, the pre-tension of the edge cables has the values 15 x 1 = 15.



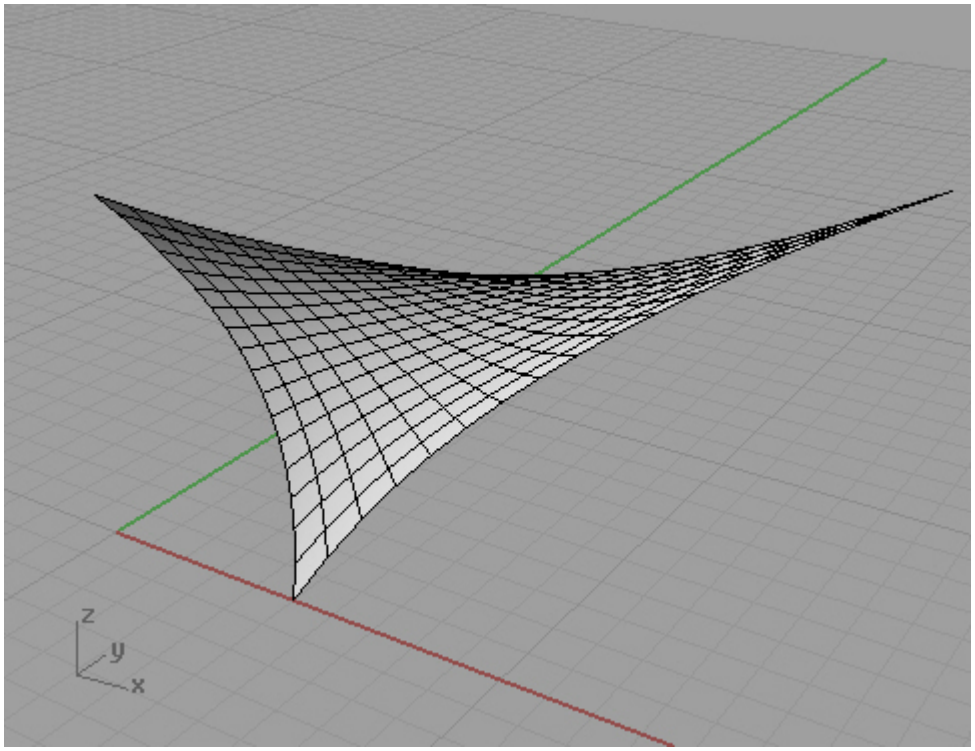
As our initial geometry is a rather good approximation of the final membrane surface, it is enough to execute 5 form finding steps (you can try a higher number, but there will be only minor changes). Set the maximum number of form finding steps using  or **fnOptions**.



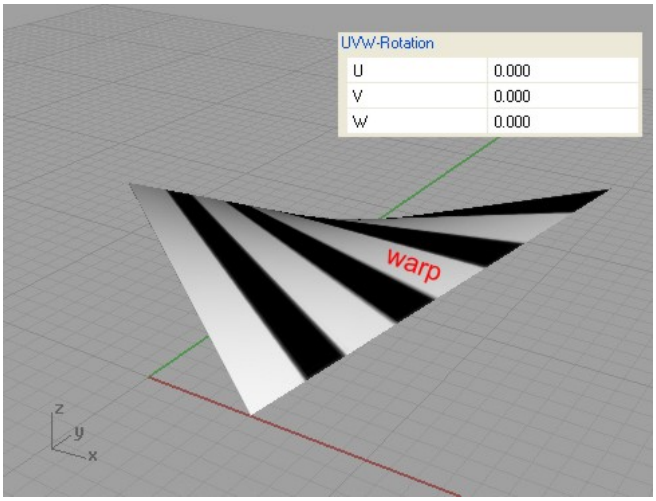
Start the form finding procedure ( or **fnFormFind**). At the end of the calculation, a new mesh object with the final

© 2009 Enter your company name

shape of the four point sail is created on the active layer. It is the equilibrium shape for the prescribed membrane and edge cable pre-tension.

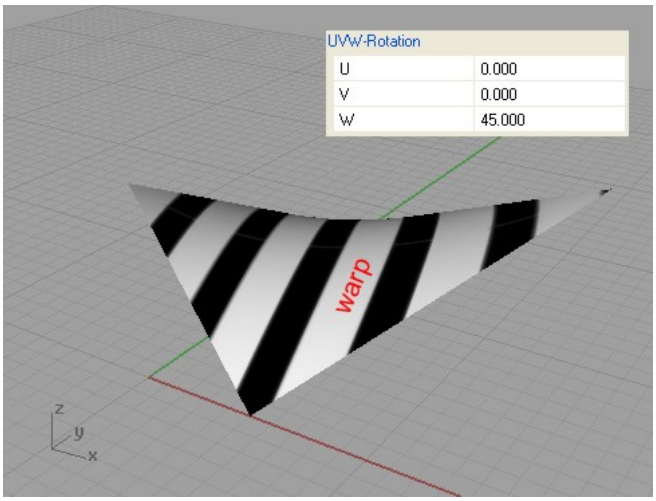


Of course, the pre-tension in warp and weft direction doesn't have to be equal – the membrane can be prestressed anisotropically. The definition of the fiber directions is done via the texture mapping in Rhino. To visualize the warp direction, apply a texture with horizontal lines to the initial mesh and switch to render mode. The lines on the mesh will be the warp direction for the calculation (weft is orthogonal to warp).

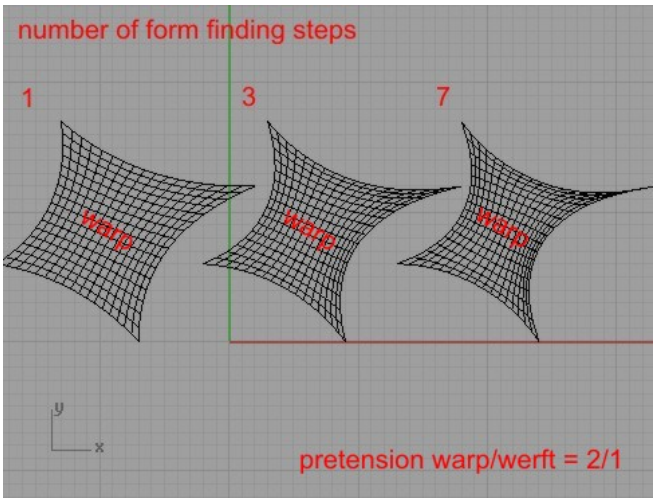


© 2009 Enter your company name

By adjusting the texture mapping (use the w angle in UVW rotation), you can modify the warp direction.



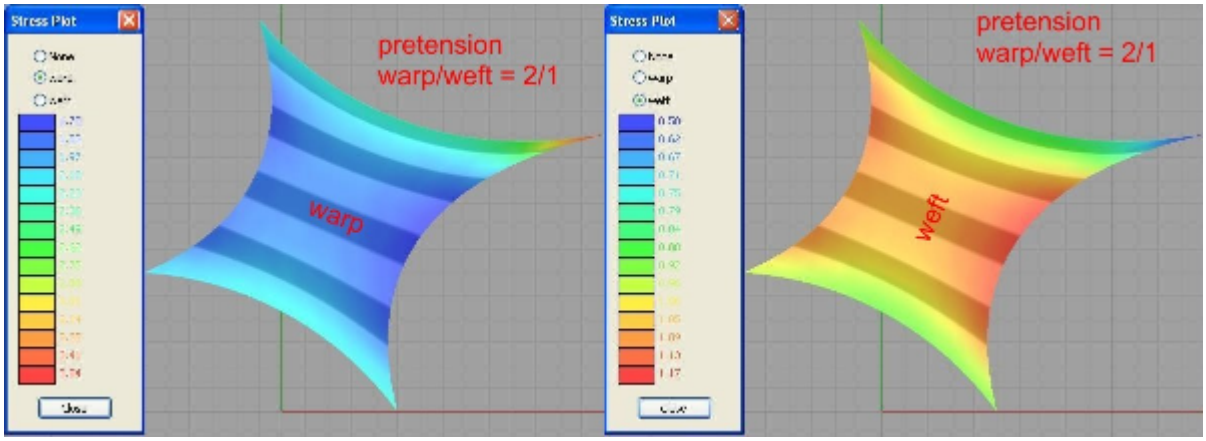
As there is no exact theoretical equilibrium shape for an anisotropically prestressed membrane, the form finding algorithm cannot find one either (what doesn't exist, can't be found): As a consequence, the mesh will deform with the number of form finding steps. Yet as all intermediate solutions are stable equilibrium shapes for a prestress distribution close to the target state (not exact the target state, but getting closer with every iteration), we advise you to do only a short number of form finding steps (1-5), until you've reached a good agreement of form and prestress (see the help section for a further discussion of the problem).



In order to control the resulting stress distribution in the membrane, you can plot the stresses in the membrane: Switch to render or shaded mode and click on  (or type **fnAnalysis**).

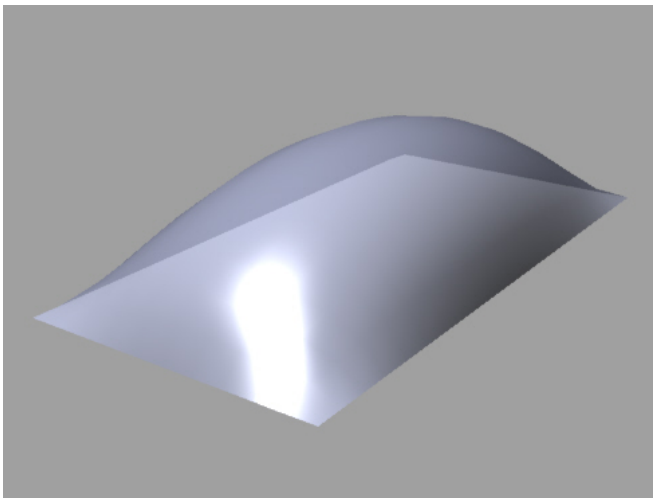
© 2009 Enter your company name

A new toolbar will pop up allowing you to select the stresses you want to plot. The stress directions correspond with the warp and weft direction which was previously defined by the texture mapping.



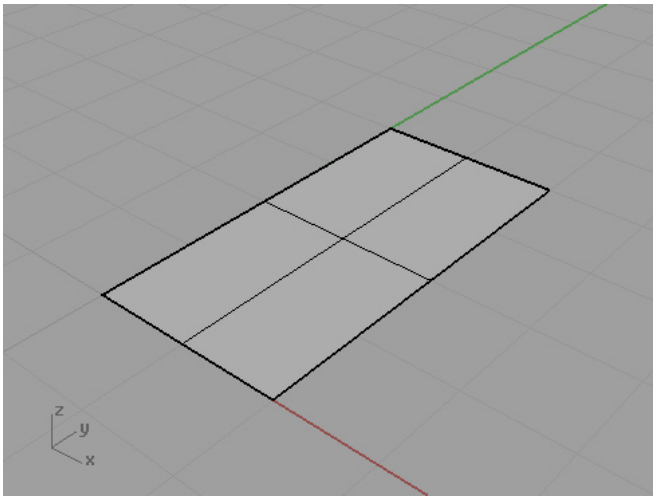
3.3 Pneumatic structure

In the third tutorial, we will do the form finding of a pneumatic structure. Here, we prescribe not only the pre-tension of the membrane,
but the pressure acting on the membrane as well.
By these means, we can generate air-supported cushions.

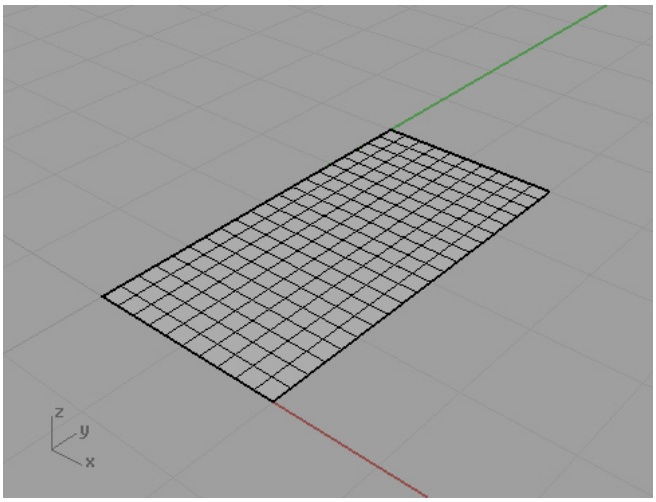


We start by creating a NURBS patch serving as initial geometry for form finding.
Let's generate a rectangular pneumatic structure measuring e.g. 2 by 4.

© 2009 Enter your company name



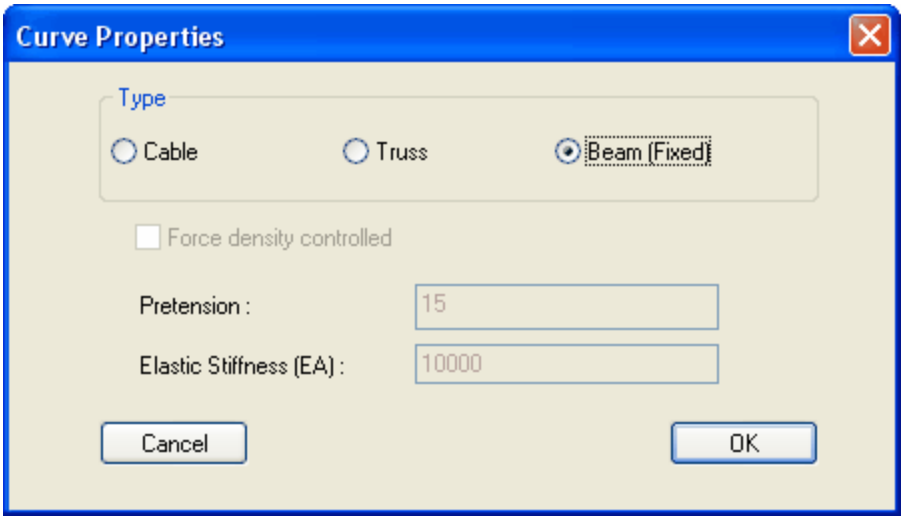
Convert the NURBS patch into a **Mesh**.



The cushion shall be fixed at the boundary. So we create a boundary mesh

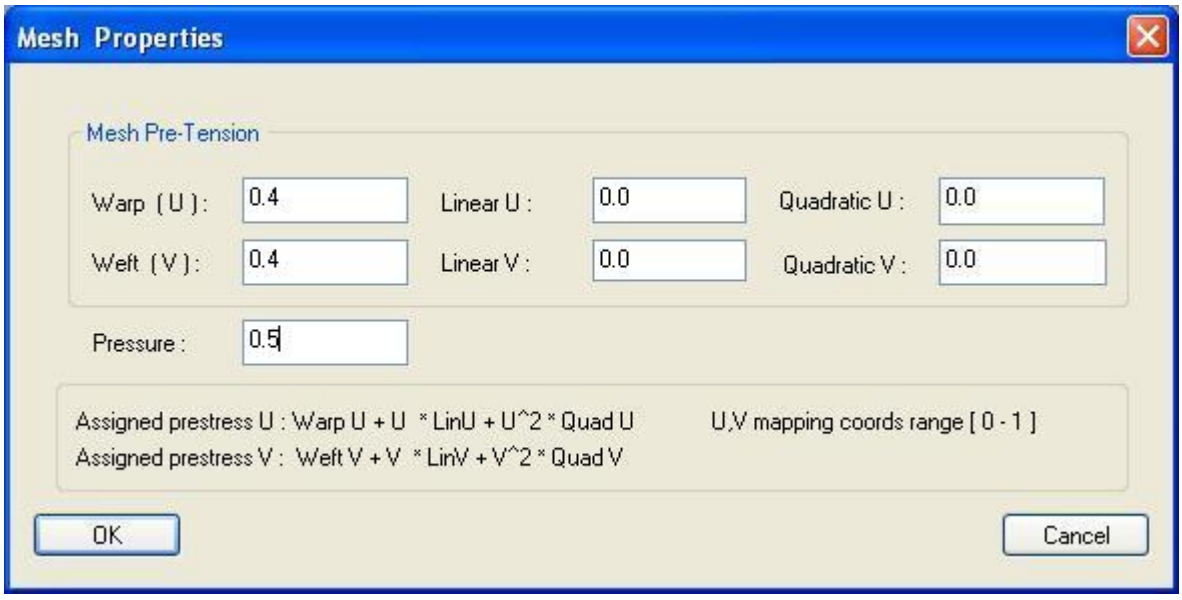
( or **fnBoundMesh**) of fixed beams.

© 2009 Enter your company name



Now we can assign the properties to the membrane

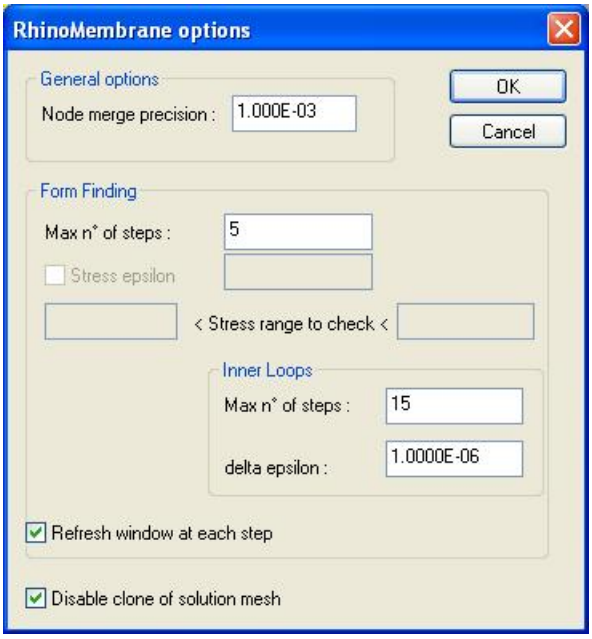
(or **fnMeshProps**). We set the pre-tension in both warp and weft direction to 0.4. The pressure is set to 0.5. Please pay attention to the different units of the respective entities: Membrane pre-tension has the unit [force/length], pressure the unit [force/(length·length)]. Similar to edge cables, pressure and pre-tension in the membrane are linked via Boiler formula.



Set the number of form finding steps in options menu

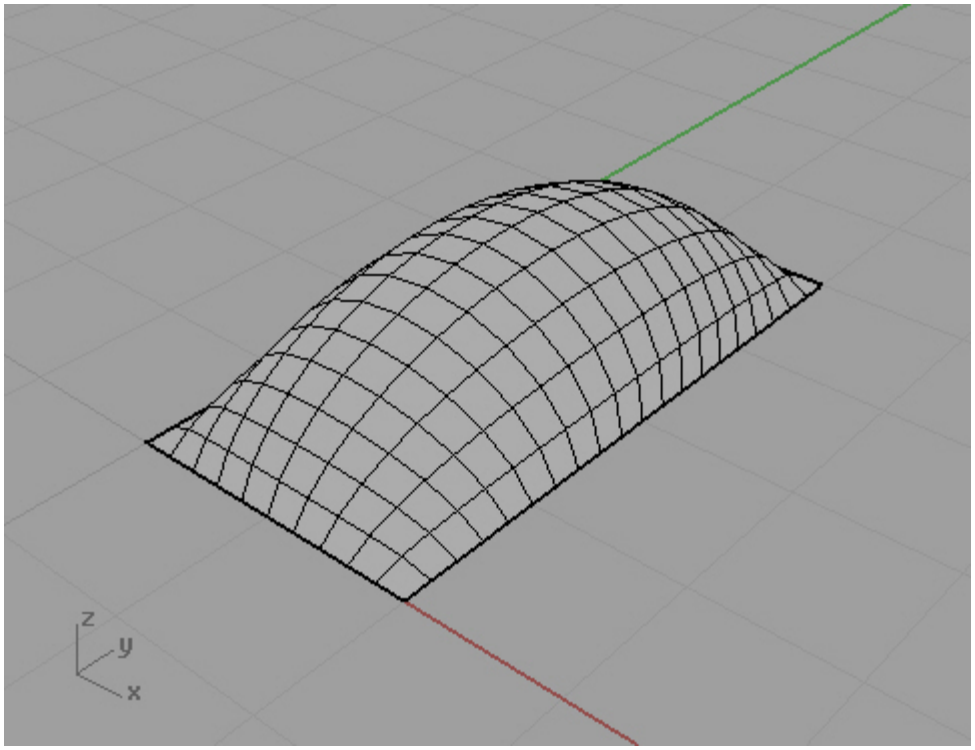
(or **fnOptions**). For this simple example, 5 steps should do the trick.

© 2009 Enter your company name



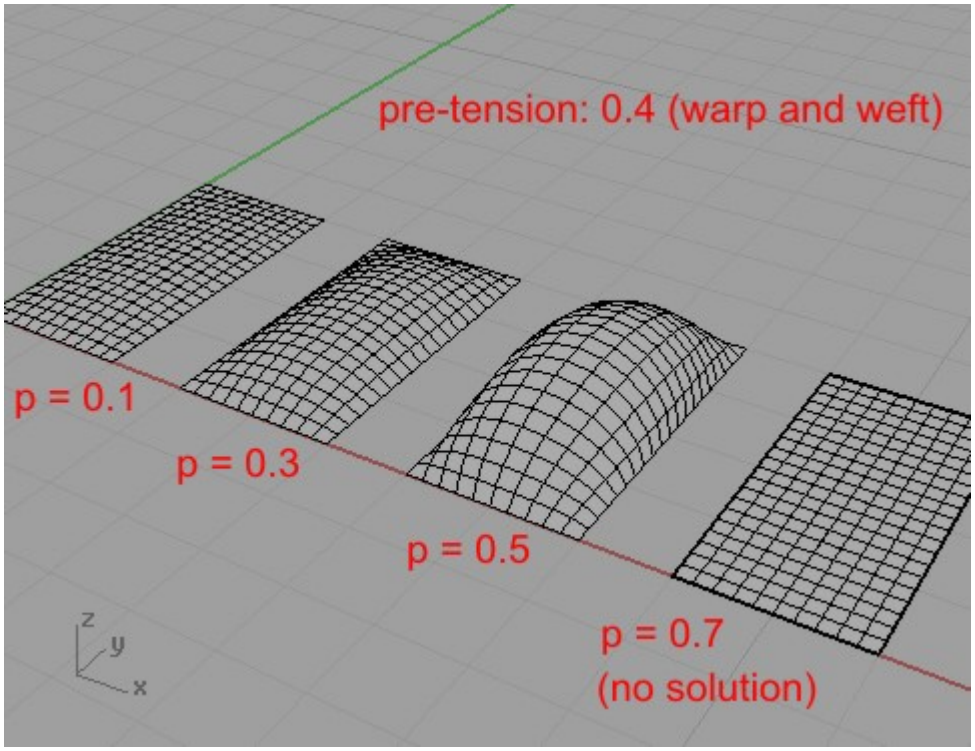
Start the form finding procedure

( or **fnFormFind**). At the end of the calculation, a new mesh object with the final shape of the pneumatic cushion is created on the active layer. It represents the equilibrium shape for the prescribed membrane pre-tension and pressure distribution.



© 2009 Enter your company name

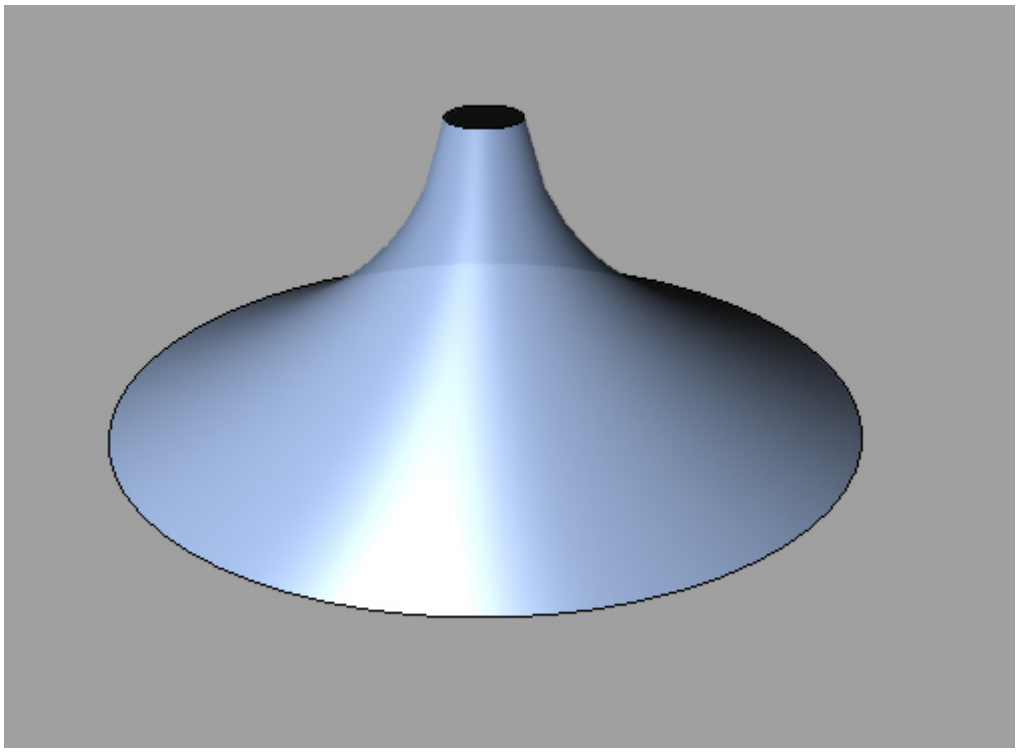
We advise you to play around with the pressure value a bit: If you decrease the pressure while keeping the membrane prestress constant, the structure will be flatter. If you increase the pressure, the structure will be more curved. But pay attention: If the pressure exceeds a certain limit value (depending on the geometry and membrane-pretension), no solution for the equilibrium shape exists any more and consequently the form finding procedure will stop without result.



3.4 Conical shape with anisotropic linear prestress distribution

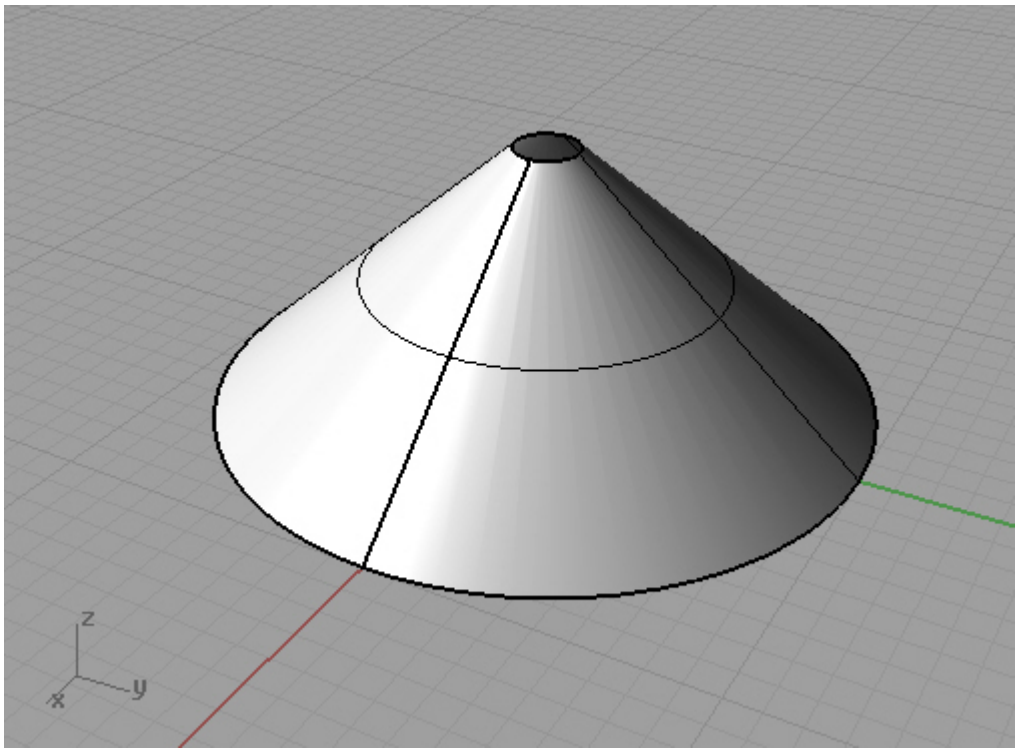
This tutorial demonstrates how to create a conical shaped membrane between fixed hoops. You will learn how to use the texture mapping function to define warp and weft direction, which is necessary for the definition of the anisotropic pre-tension.

© 2009 Enter your company name

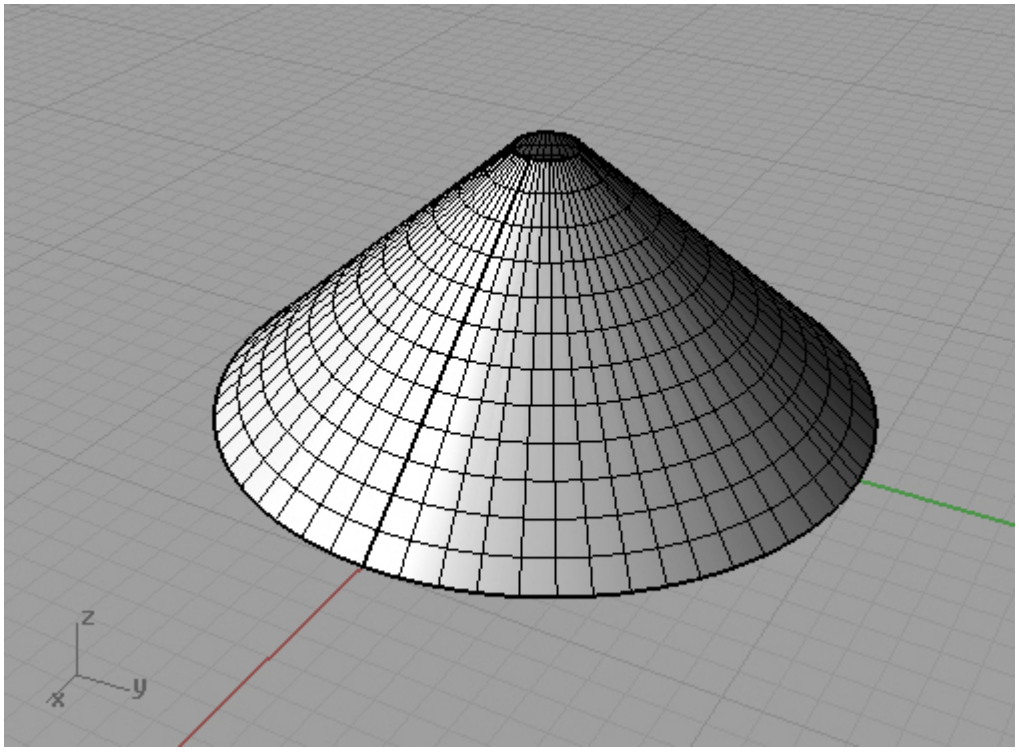


The first step is to generate the initial geometry for form finding: As the structure exhibits a rotational symmetry, we can create a NURBS surface by rotating a straight line around a rotation axis. Hereby, a conical shape is generated. In this example, the rotation axis is the global z-axis. The radius of the upper hoop measures 1 unit, the radius of the lower hoop 10 units. The height is set to 8 units.

© 2009 Enter your company name



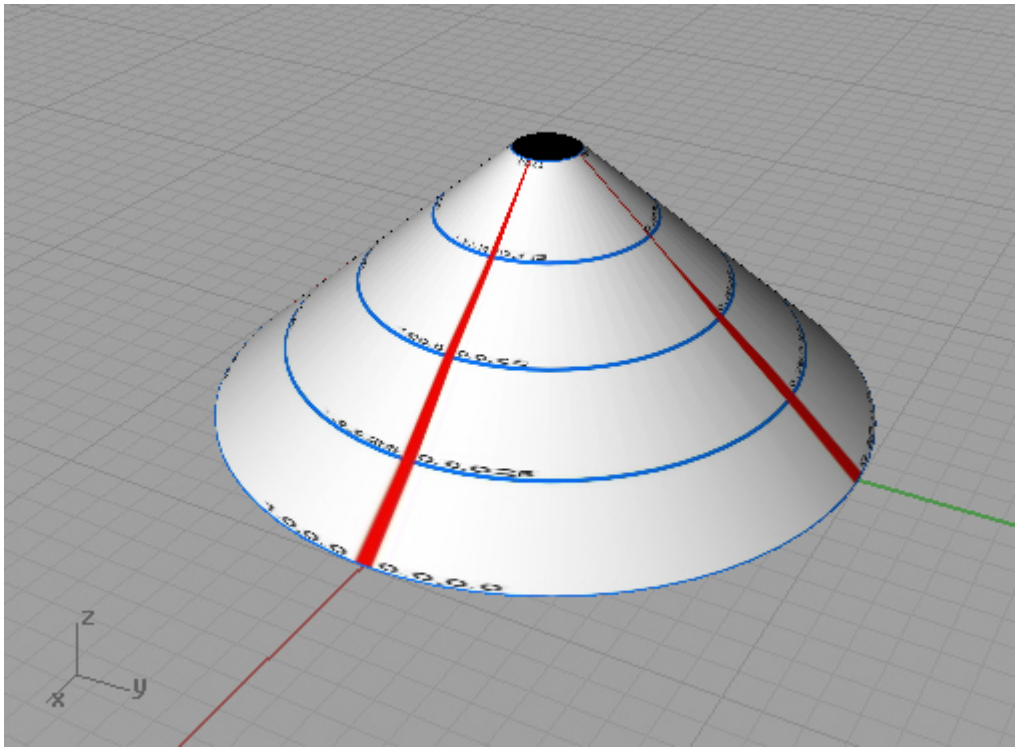
As usual, convert the NURBS patch to a **Mesh** object.



Now we apply a texture mapping to the structure. Use the texture file ***U-V-MAP.jpg***

© 2009 Enter your company name

provided in the installation folder of Rhino Membrane. In this example, a cylindrical mapping gives the best results. With the help of the texture mapping, texture parameters U and V are mapped onto the surface. We will use these parameters U and V to define the weft and warp direction of a fabric. To visualize the texture, go to render mode.

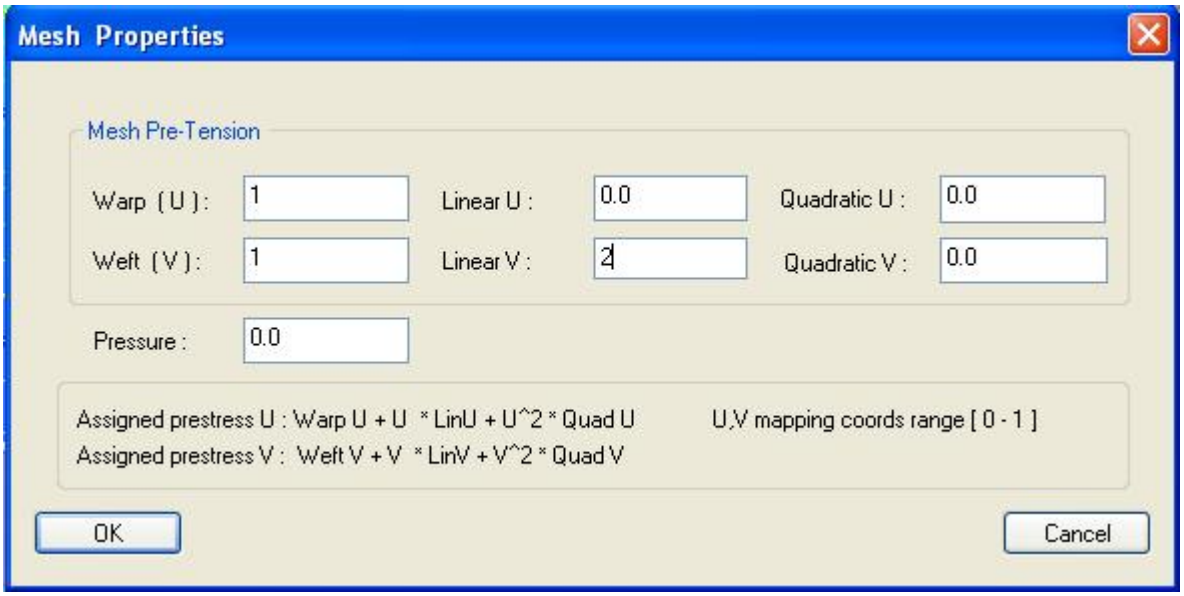


The blue lines indicate the direction of the first coordinate U, the red lines the direction of the second coordinate V. If you zoom in, you will notice, that the V coordinate has a value of zero at the lower hoop, while its value is 1 at the upper one.

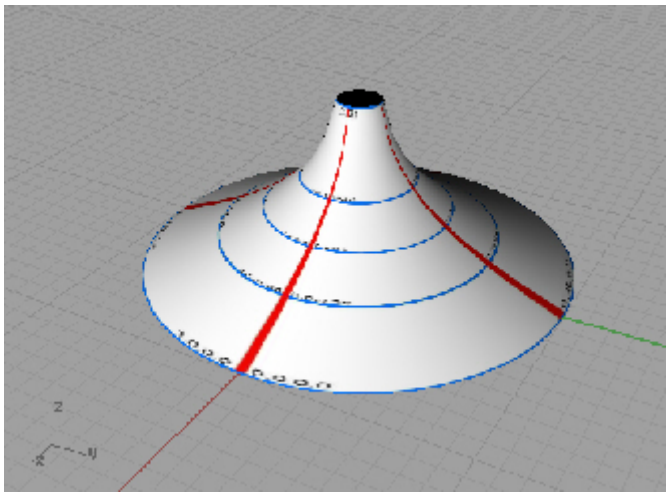
Now, we have to start with the input of our problem data: To define the two fixed hoops, we either create a boundary mesh ( or **fnBoundMesh**) of fixed beams or fixed all nodes at the hoops individually ( or **fnMeshNodeProps**).

We specify an anisotropic prestress distribution: In hoop direction (blue line or U coordinate), a constant stress of 1 is prescribed. In meridian direction, we want to have a linear stress distribution: At the upper hoop (V = 1) the meridian pre-tension should be 3, at the lower hoop (V=0), the pre-tension 1. Therefore we have a constant pre-stress of 1 in meridian direction (Warp (U)) and a linear varying component of 2 (Linear U). The stress function can be entered in the Mesh properties window ( or **fnMeshProps**) as follows:

© 2009 Enter your company name

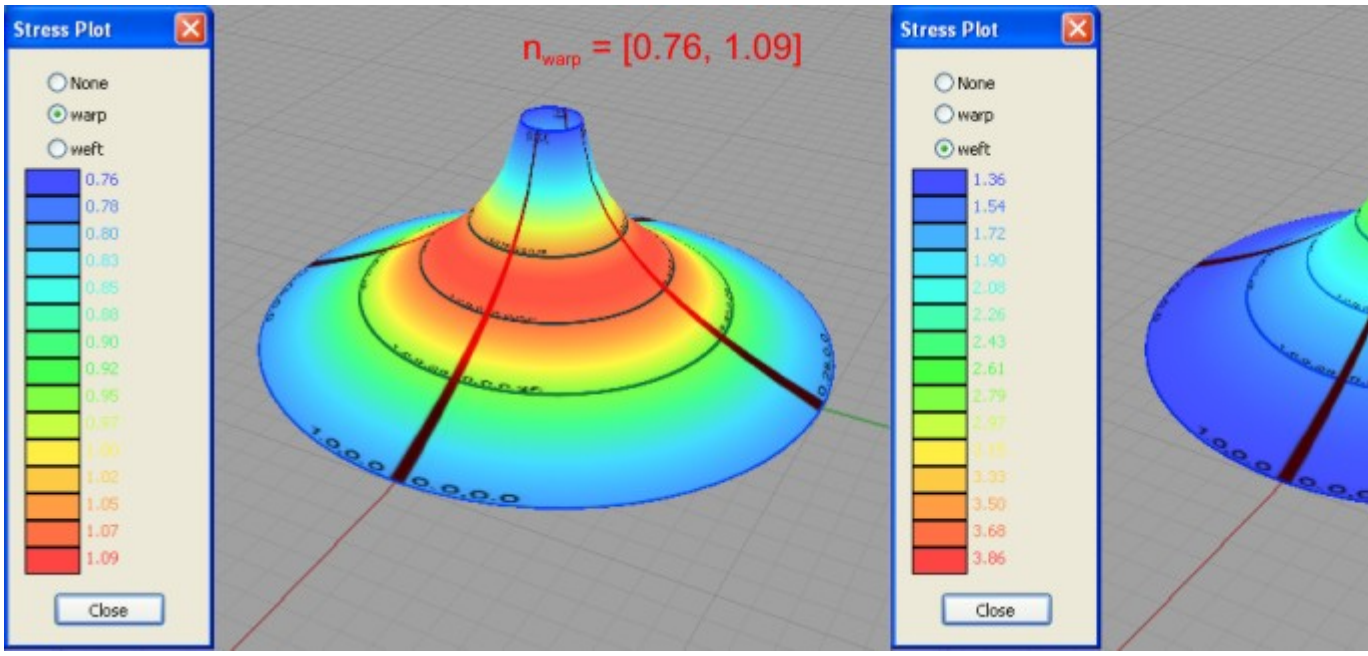


Adjust – if necessary - thenumber of form finding iterations (e.g. 2) by clicking on  or typing **fnOptions**) and start the form finding process ( or **fnFormFind**). The result should look something like this:



If we take a look at the final stress distribution ( or **fnAnalysis**), we can see that the values for the warp stress (in hoop direction) vary around 1, the weft stresses range from about 1 to about 3. As already mentioned in tutorial 2, an exact solution for most of the prescribed stress distributions is physically not possible and therefore only an approximation can be achieved. If you increase the number of form finding steps, the stresses should come closer to the desired stress distribution, yet the geometry could be deformed very much. We advise you to play around with the number of form finding steps, until you are satisfied with your solution.

© 2009 Enter your company name

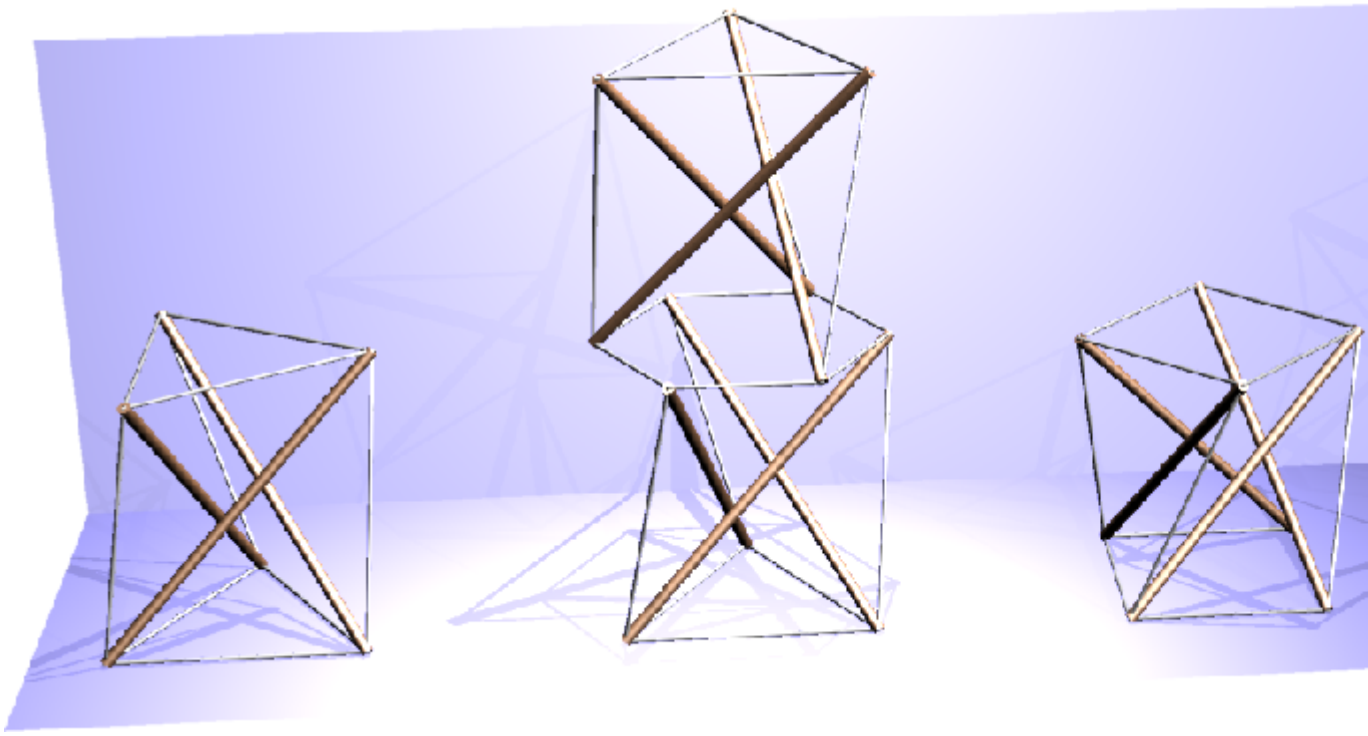


3.5 Tensegrity

By combining cable elements for form finding with elastic cable elements we are able to do the form finding of **tensegrity structures**. The proposed form finding method is based on a simulation of the actual assembling process of tensegrity systems. This means that “almost” the whole structure is modeled with elastic cable elements, which have a certain tension/compression stiffness. The required pretension of the system is obtained by modeling the remaining structural elements with form finding cables with a prescribed prestress. This causes elastic deformation in the main structure and leads to a stable equilibrium shape.

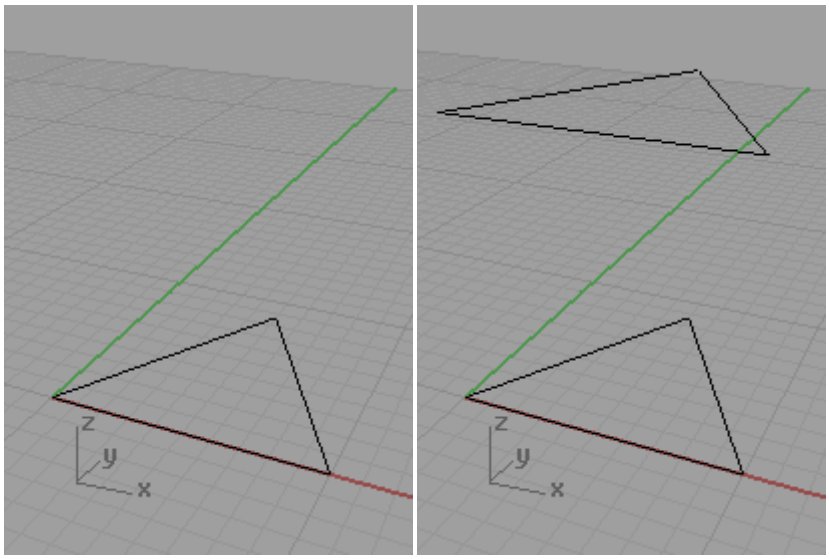
For an easy start, we will begin with simple **twist elements** (also called cylindrical tensegrity).

© 2009 Enter your company name

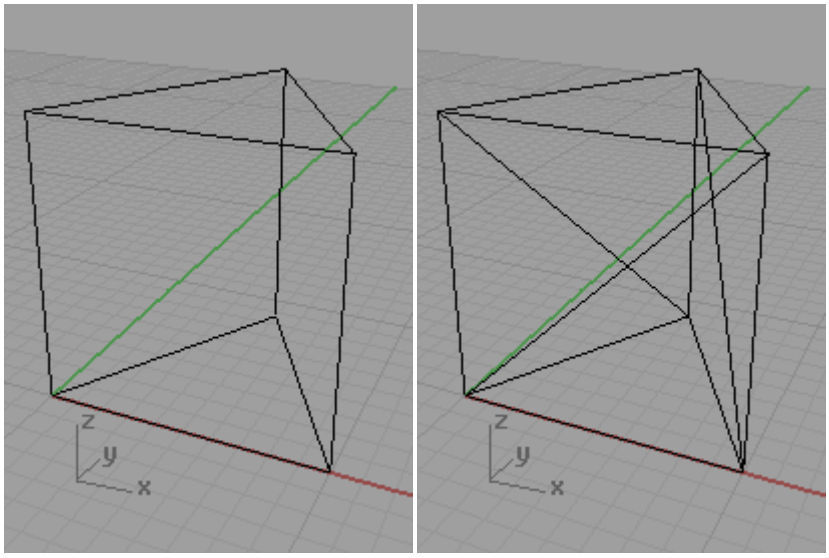


1) As usual, a first approximation of the final geometry has to be made in order to start with the numerical form finding procedure. This means that we have to know the topology of the structure in advance. For the simplest tensegrity structures – the tripod – you have to create a bottom and a top triangle consisting of **one line element for each edge**. These horizontal elements will become tensioned cables later on.

© 2009 Enter your company name



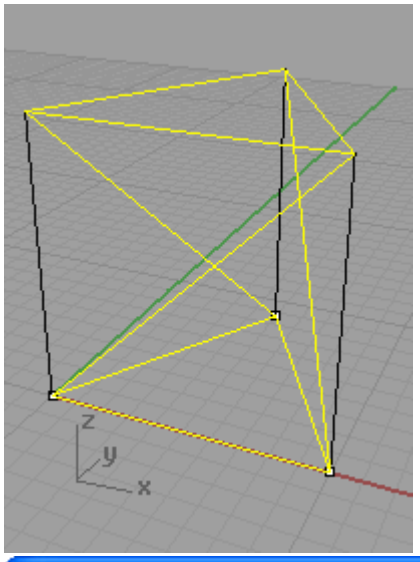
2) In the next step, the two triangles are connected by three vertical and three diagonal line elements. The first ones will become tensioned cables, the latter ones compressive elements.

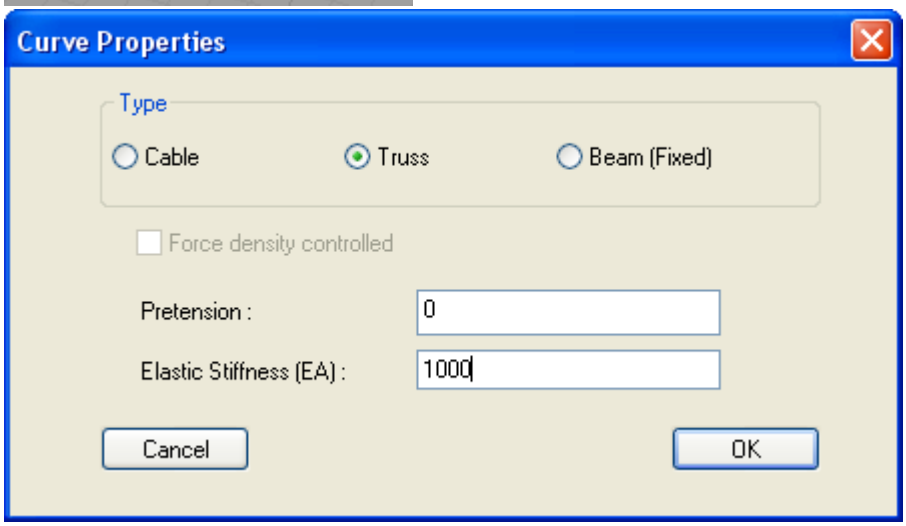


3) By clicking on  (**_fnCurveProps**) we set an elastic stiffness for all linear elements except for the three vertical cables. As the elastic elements, which belong to the "**Truss**" element type, should be initially stress-free, the pretension is

© 2009 Enter your company name

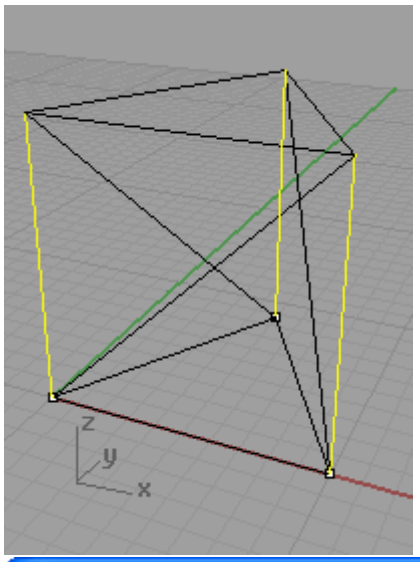
set to 0.

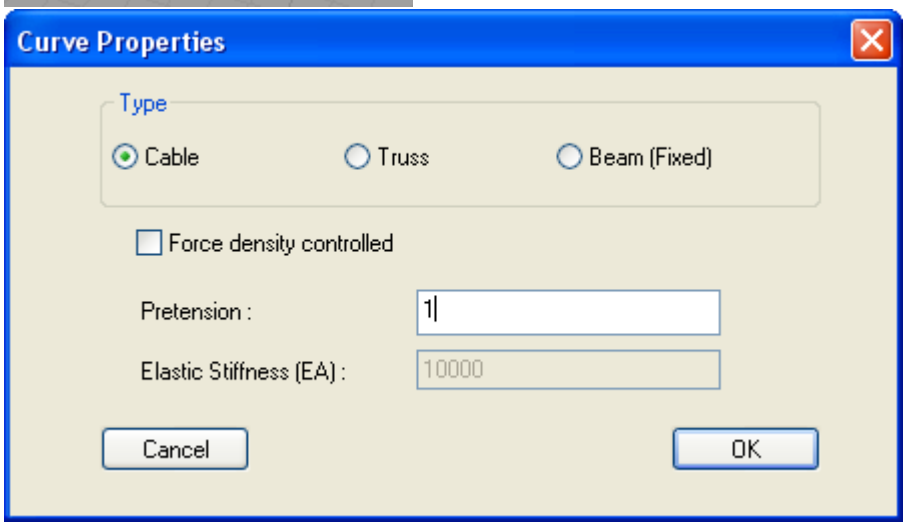




4) The three vertical cables are modeled with "**Cable**" elements. By doing so, we can assign them a pretension, which is independent of the deformation process. To set these properties, click again on  (**_fnCurveProps**), choose the "Cable" element type and set the pretension to 1.

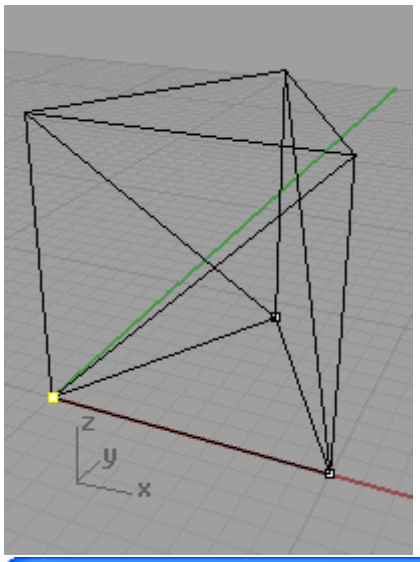
© 2009 Enter your company name

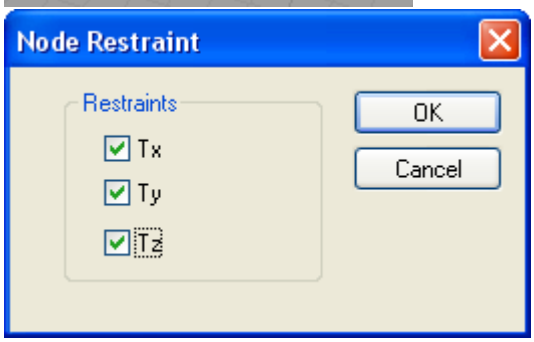


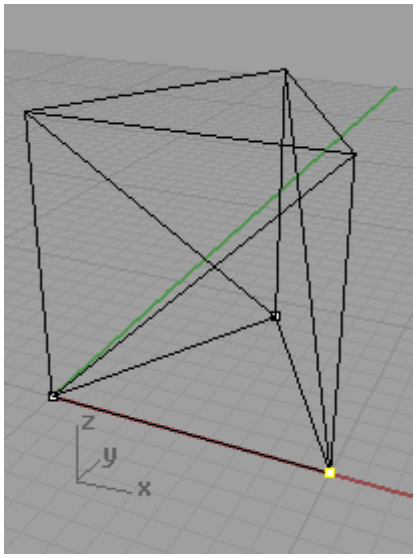


5) For a maximum degree of freedom in the form finding process, statically determined boundary conditions have to be applied. To do so, create three nodes at the vertices of the lower triangle and fix only six degrees of freedom ( or **_fnNodeProps**). Have a look at the following pictures to see how it is done!

© 2009 Enter your company name

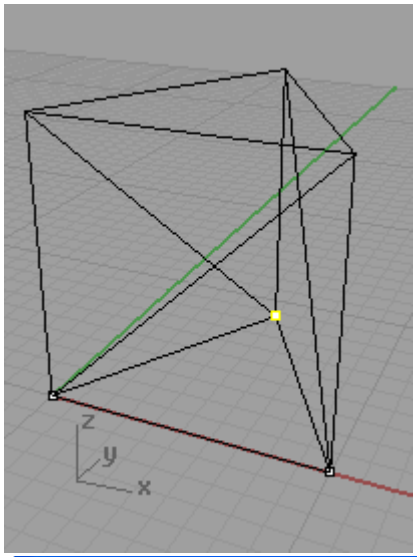






© 2009 Enter your company name

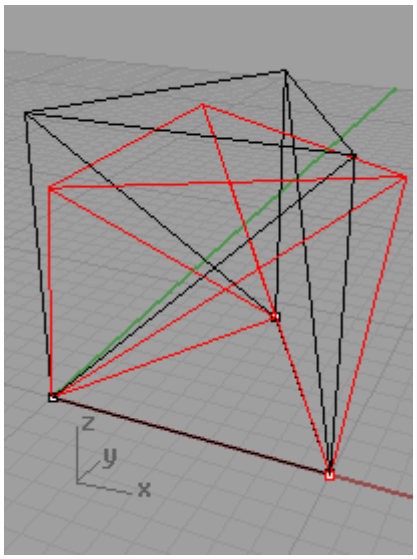




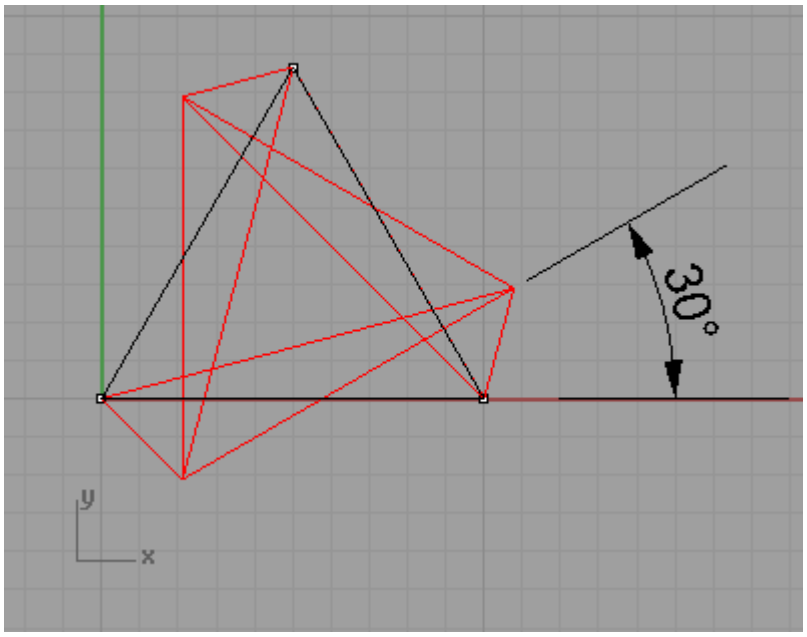


6) Now the form finding process can be started. The form finding cables pull on the remaining structure and move it into an equilibrium shape. Try it out by clicking on  (**_fnFormFind**). You can use the standard options. But before you start with the form finding, be sure to set an extra layer active, so that the resulting geometry will be stored on this new layer.

© 2009 Enter your company name



7) If you have a closer look on the equilibrium shape, you will notice that the upper triangle is "twisted" by a certain angle with respect to the lower triangle. And interestingly, this twist angle measures always 30 degrees independent of the initial height of the structure, the pretension in the vertical cables, etc. It's even possible to prove this fact mathematically!



8) If you have mastered this first tensegrity structure, you can

© 2009 Enter your company name

try out the following:

- Instead of prescribing the pretension in the vertical cables, try to prescribe the tension forces in the horizontal cables or alternatively the compressive forces in the diagonal trusses

- Create a **quadpod**, a **pentapod**, a **hexapod**, ⋯ and find out their respective unique twisting angles.

- Try to create a two-layered tensegrity structure by setting a tripod on top of another one

© 2009 Enter your company name

Endnotes 2... (after index)

© 2009 Enter your company name

Back Cover