

Multimedia Engineering Pte Ltd

# Workflow for Design of Stressed Membrane

Multimedia Engineering Pte Ltd

April 2021

## TABLE OF CONTENTS

---

1	Introduction to Stressed Membrane .....	2
1.1	Stressed Membrane Mesh Model by Windas.....	3
1.2	Equilibrium Form –Finding.....	4
1.2.1	Material Properties .....	4
1.2.2	Applied Pre-Stress and Pre-Tension.....	4
2	Mesh Model by Windas .....	6
2.1	Simple Conical Polygon .....	6
2.1.1	Initial Geometry .....	6
2.1.2	Form-Finding.....	8
2.2	Simple Hypar .....	13
2.2.1	Initial Geometry .....	13
2.2.2	Form-Finding .....	15
3	Mesh Model from External Tool (Rhino) .....	20
3.1	Initial Geometry .....	20
3.2	Form Finding .....	21
4	Mesh Refinement.....	25
4.1	Mesh Refinement in Windas.....	25
4.2	Mesh Refinement in Rhino/Grasshopper .....	26
4.3	Mesh Creation in Rhino / Grasshopper .....	29
4.3.1	Meshing Surfaces .....	29
4.3.2	Meshing with Partition Cable.....	31
5	Structural Analysis.....	32
5.1	Simple Conical Polygon .....	32
5.2	Simple Hypar .....	34
5.3	Triple Hypar.....	36

# 1 INTRODUCTION TO STRESSED MEMBRANE

---

Stressed membrane is an approach to create tensile membrane form that is in equilibrium under initial prestress. This method utilises the 3D triangular constant strain finite element to idealise the membrane surface. It can be considered as a more realistic representation of the physical membrane structure compare to that model by the force density method.

The procedures on the preparation of a stressed membrane model for design and analysis are:-

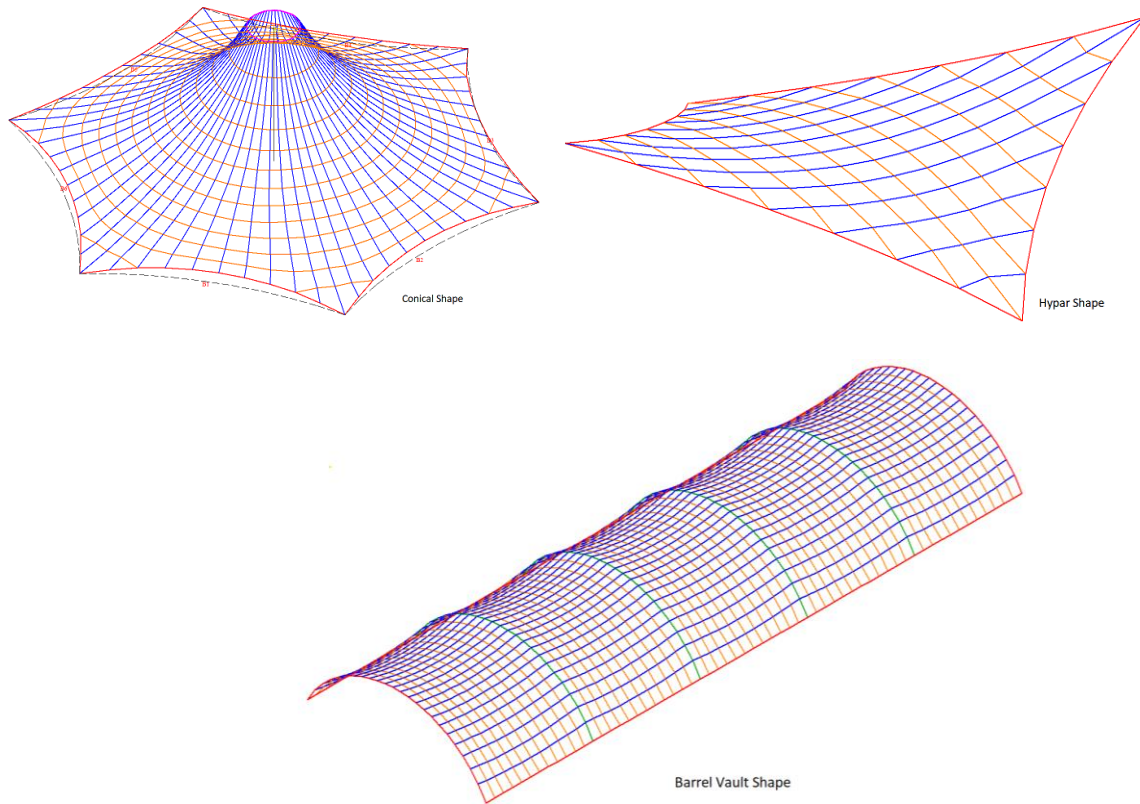
1. Using Windas and the force density method to create the initial equilibrium model. Convert the force density model to stressed membrane form and use the Updated Reference Strategy Method to find the equilibrium model and updated prestress.
2. Using any CAD software like Rhino or other meshing tools to create a 3D triangular mesh as closely as to the design intent. Import the 3D model into Windas. Apply prestress and use the Update Reference Strategy Method to seek an equilibrium form and updated prestress.

This workflow will be dissected into 4 sections as follows :

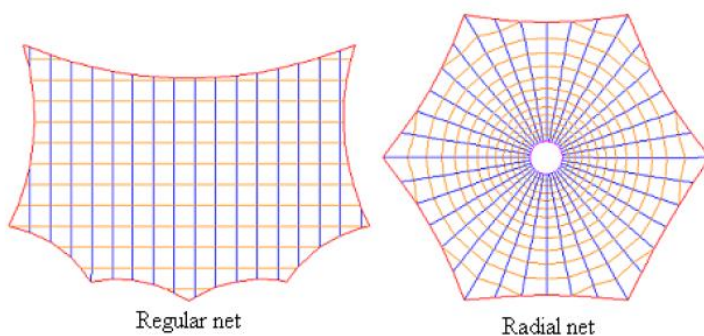
1. First section presents the workflow on the creation of stressed membrane mesh model using Windas and the force density method.
2. Second section presents the work flow on the creation of stressed membrane mesh model using external tool (i.e. Rhino in this example).
3. Third section presents the refinement technique for stressed membrane model using Windas and Rhino/Grasshopper. The stressed membrane are refined to achieve more uniformly distributed triangles. This technique is necessary for model with partition for internal cables.
4. Fourth section presents the design and analysis of various stressed membrane forms. Several items to look out for will be discussed as well.

## 1.1 STRESSED MEMBRANE MESH MODEL BY WINDAS

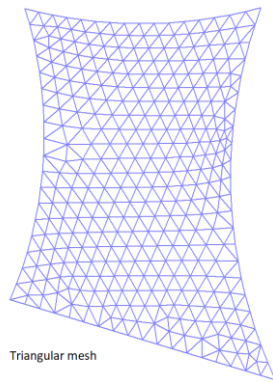
Windas has automatic function to produce initial mesh for generic tensile membrane forms. These forms are typical anticlastic surface that are commonly used for membrane including, but not limited to, conical shape, hyper shape, and barrel vault.



Windas allows form finding with pre-defined surface boundaries. The general net forms are regular net (or xy-net) and radial net.



Windas can also produce triangular type of mesh under 'Region' command.



Generic mesh created by Windas requires alteration before it is suitable for Stressed Membrane analysis. The alterations are based on the rules below :

- **no force density** values should be imposed on the mesh,
- **no warp-weft truss elements (t11)** should remain inside the model,
- membrane mesh must be **fully triangulated**.

In general, Stressed Membrane method should be able to handle any typical membrane mesh so long the requirements above mentioned are fulfilled.

## 1.2 EQUILIBRIUM FORM—FINDING

### 1.2.1 Material Properties

In tensile membrane form finding and structural analysis, the elastic modulus of material section properties needs to be specified. Different fabric stiffness will interact differently with applied pre-stress and loading. As of now, the tensile membrane material will be assumed isotropic. Value of Young Modulus (E) also assumed to be the same for warp and weft direction. For every mesh model produced for analysis, tensile membrane and cable properties must be specified.

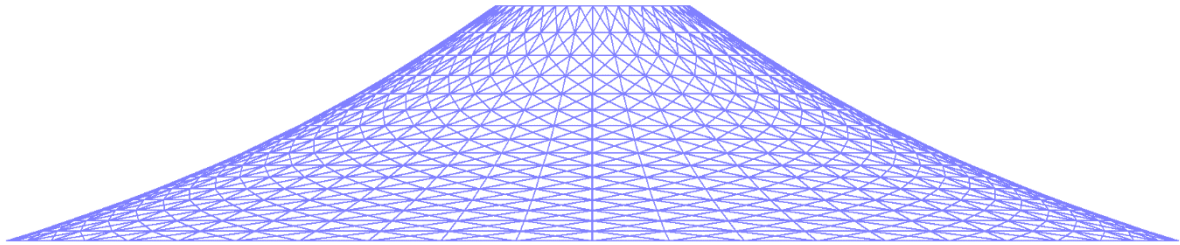
### 1.2.2 Applied Pre-Stress and Pre-Tension

Pre-stress contributes significantly to membrane's stiffness and its equilibrium shape. Stressed membrane is a 2D analysis, therefore pre-stress value shall be applied to each individual 2D elements constructing the global mesh uniformly. The Updated Reference Strategy method will redistribute the pre-stress values according to its shape and geometry.

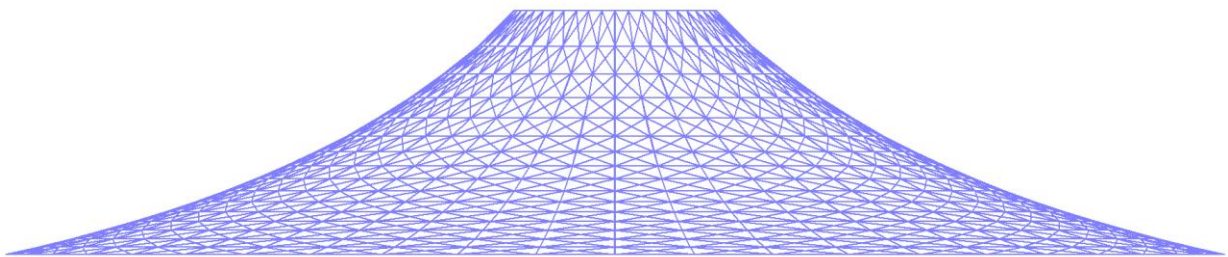
For PTFE/PVC, pre-stress value may vary around 2 – 10 N/mm<sup>2</sup>.

For ETFE, around 1-5 N/mm<sup>2</sup>.

Let's take these two conical shape as an example. Pre-stress value of 2 N/mm<sup>2</sup> and 10 N/mm<sup>2</sup> are being imposed respectively for minimal surface form finding. The result will vary as shown below.



Form-finding result under initial pre-stress of 2N/mm<sup>2</sup>



Form-finding result under initial pre-stress of 10N/mm<sup>2</sup>

This example shows visible change of slope between the 2 mesh. Both mesh are perfectly identical except for its pre-stress value. It is a clear indicator that the final shape of the same initial mesh will vary according to the imposed pre-stress.

When utilised, border cable pre-tension will also strongly influence form-finding result. Suitable pre-tension value has to be gauged based on the model sagging criteria, membrane material, and membrane area. Recommended value can be seen from the attached force density defined by the initial mesh. Normally pre-tension value varies from 1-200kN.

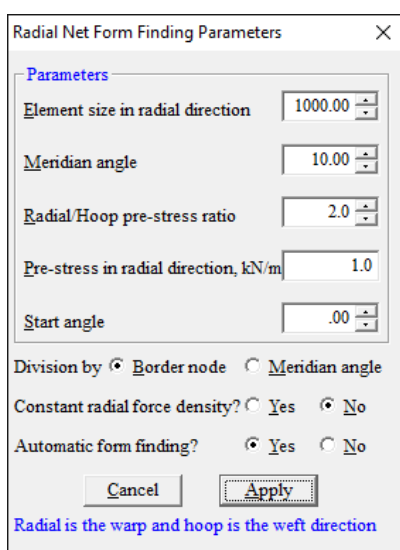
## 2 MESH MODEL BY WINDAS

### 2.1 SIMPLE CONICAL POLYGON

This tutorial shows essential steps to form find and analyse a simple conical polygon tensile membrane using stressed membrane method in Windas.

#### 2.1.1 Initial Geometry

Generate the initial conical polygon mesh using Windas. Click **Generate | Membrane Forms | Conical Tent | Polygon | Apply**. Evaluate appearing command box below. Note that changing parameters in this box will change the resulting mesh.



Radial Net Form Finding Parameters

Parameters

Element size in radial direction: 1000.00

Meridian angle: 10.00

Radial/Hoop pre-stress ratio: 2.0

Pre-stress in radial direction, kN/m: 1.0

Start angle: .00

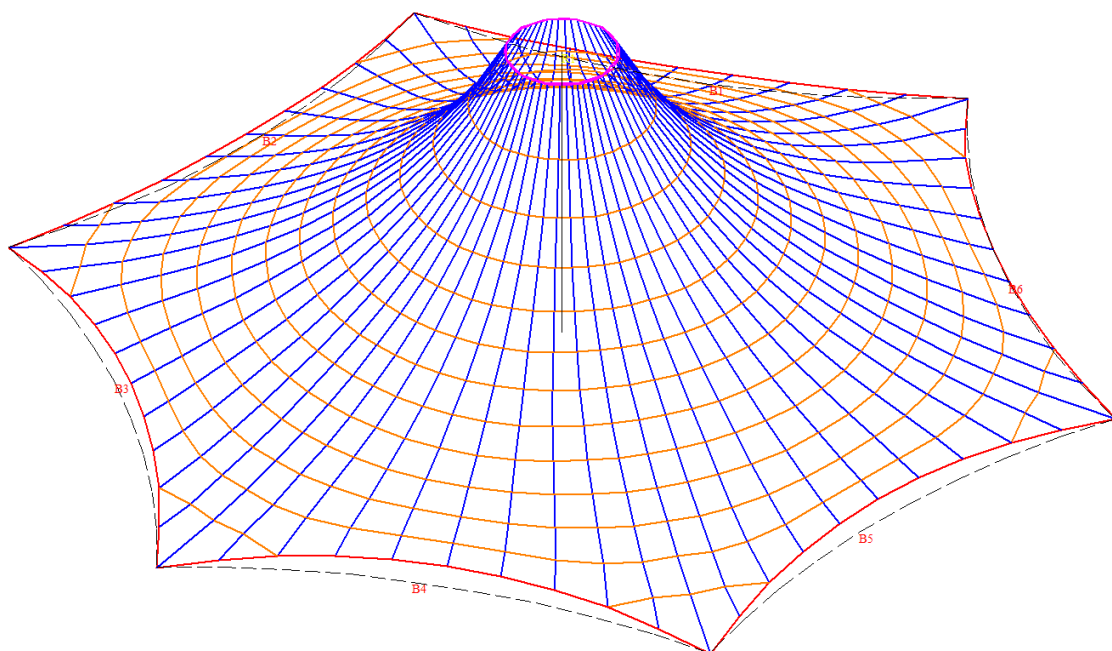
Division by: ☒ Border node ☐ Meridian angle

Constant radial force density? ☐ Yes ☒ No

Automatic form finding? ☒ Yes ☐ No

Cancel Apply

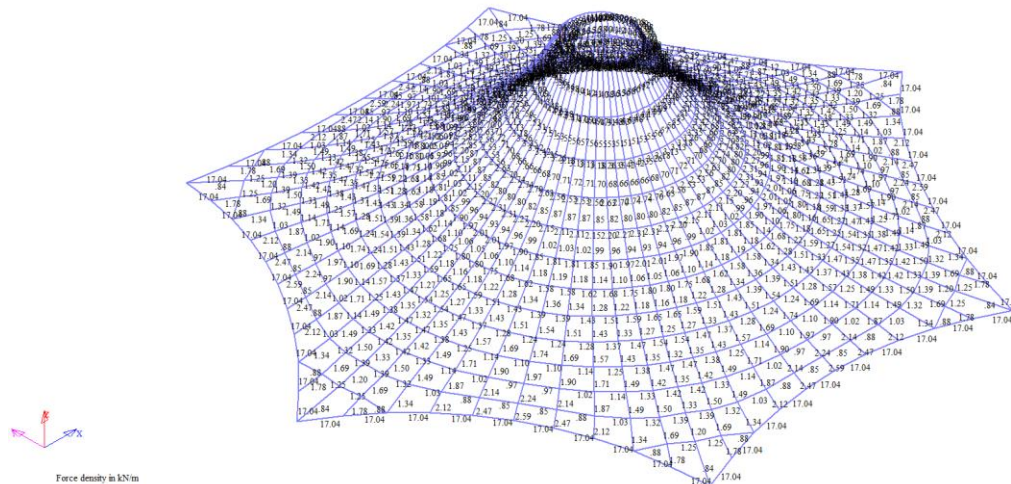
Radial is the warp and hoop is the weft direction





For stressed membrane analysis, force density load will not be utilised. However, it is a good practice to take down the border cable's force density value as it is useful to gauge input pre-tension for stressed membrane form-finding later on. This is to maintain the mesh shape as intended.

Use **Display | Load value | Force Density**. Focusing on the border cable elements, pre-tension value can be estimated to be 17kN.



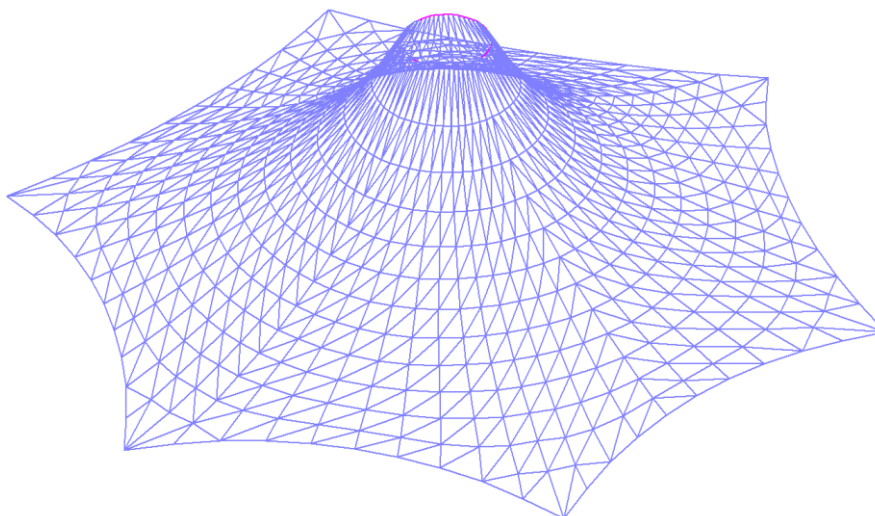
Alteration work need to be done for Stressed Membrane analysis.

Simply use **Delete | Load | Force Density | All** to remove any force density values attached to this model.

Use **Delete | Element | type t11** to remove all warp-weft truss elements.

As the top ring cable element is not used, we can also **Delete | Element | C7** (colour code for the top ring cable).

Then use **Winfabric | Triangulation**.

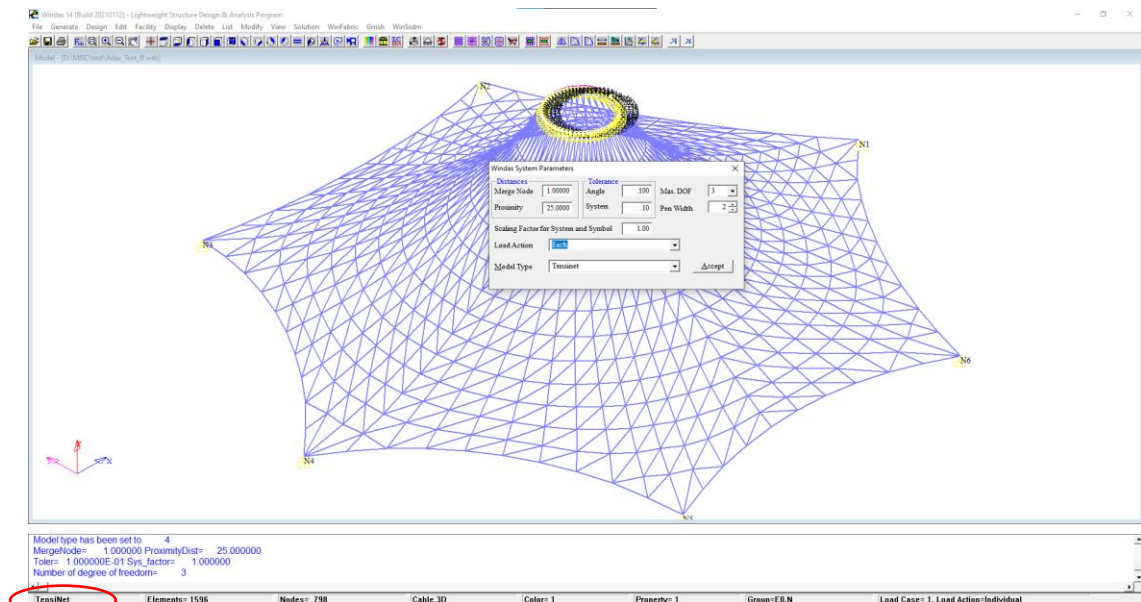




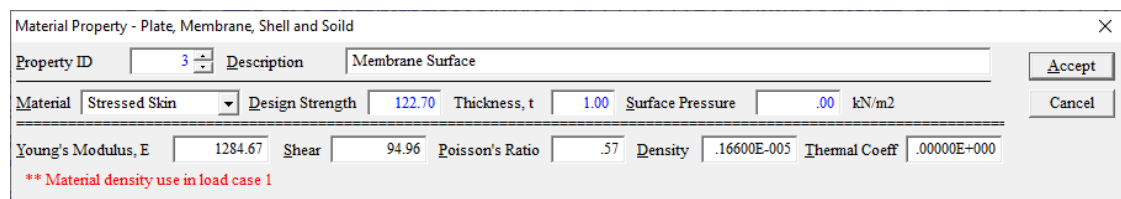
### 2.1.2 Form-Finding

This section lays out step by step procedure of form-finding with stressed membrane method in Windas. Checking the final mesh equilibrium is also demonstrated.

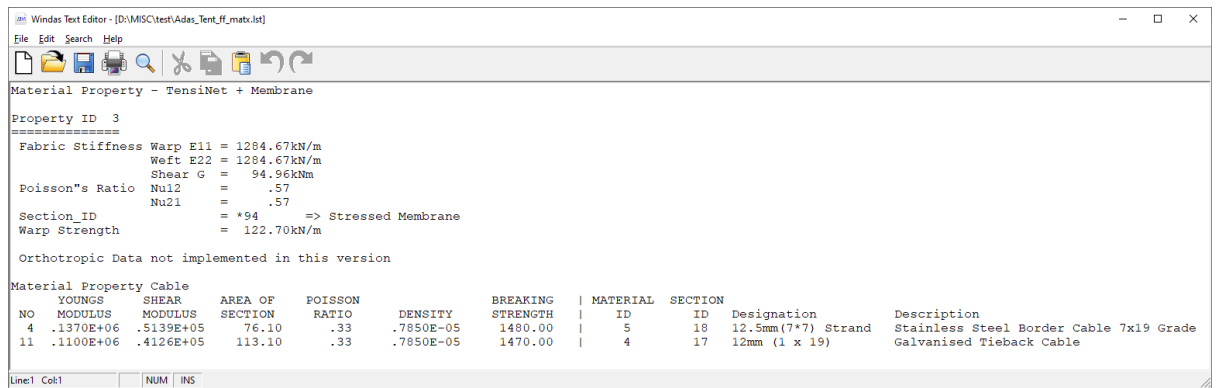
1. Set Windas system to **Tensinet** using **Facility | Set | System | TensiNet**. Check the system setting on the bottom left corner of Windas.



2. To minimise frontwidth for analysis we use **Facility | Reorder | Element Reorder | Fast**.
3. Assign membrane properties using **Generate | Material & properties | Plate-Shell-Solid-Surface**. Windas automatically assigns membrane element to Property 3. Therefore we can go to Property ID 3, set material as Stressed Skin. The default material given is PTFE FGT-600. For this exercise, default values are used. Should the material differs, Young's Modulus (E), Shear, Poisson's Ratio and Density value need to be adjusted accordingly.

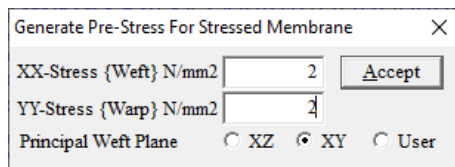


Click **List | Material & Section Properties** to see that the elements has been set properly.

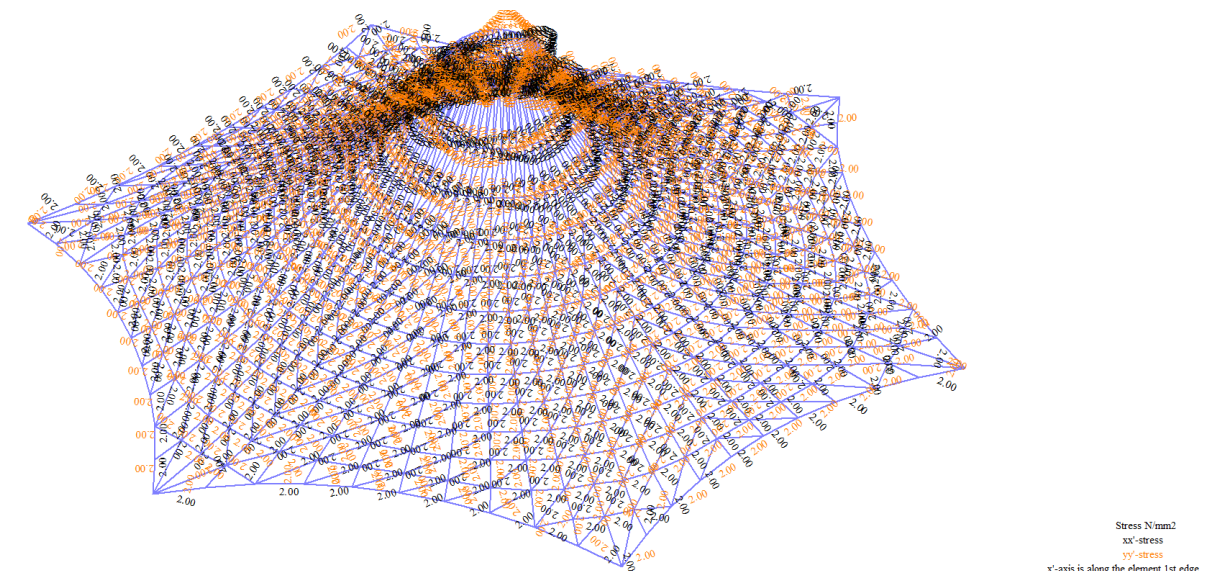


- Input pre-stress using **Generate | Load | Pre-Stress | C2** (colour code for membrane element). Assume value of  $2\text{N/mm}^2$  for both directions.

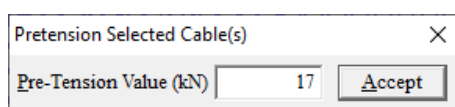
Principal Weft Plane is the perpendicular plane of surface where pre-stress is being applied. Conical shape stretches on XZ plane (mounting upwards), therefore the principal weft plane is XY plane.



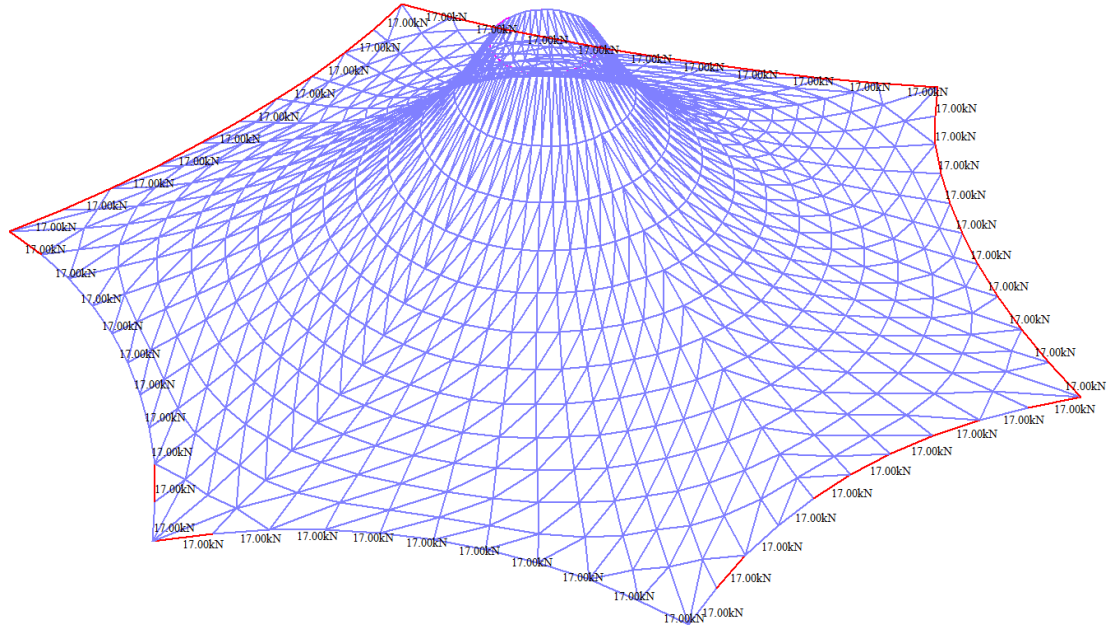
Use **Display | Load value | pre-stress** to check.



- Input pre-tension using **Generate | Load | Pre-Tension | C14** (colour code for border cable element).



Use **Display | Load value | pre-tension** to check.

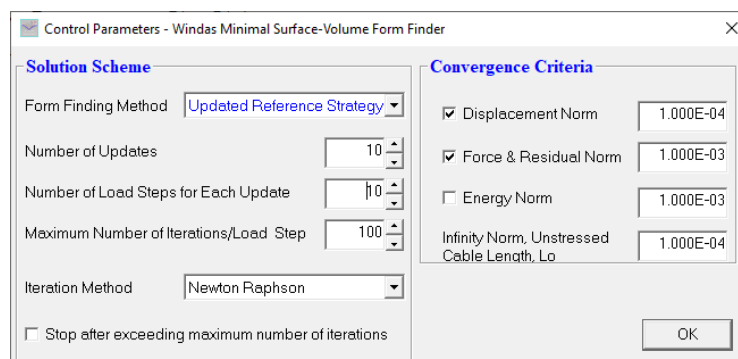


6. For analysis purpose, it is recommended to input loading before form-finding. In this exercise, we will impose typical membrane loading that are normally used for our project. Self-weight has been automatically applied by Windas.

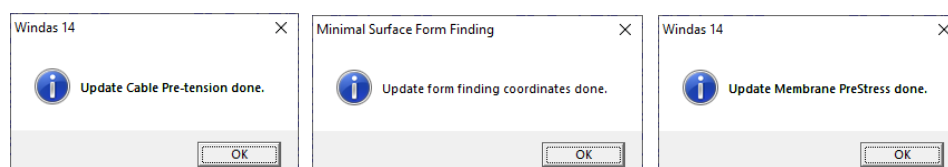
Use **Generate | Load** command and apply the load accordingly.

Loadcase		Value (kN/m <sup>2</sup> )	Input Load Type
1	SW + PS + PT	-	-
2	Rain Load	- 0.1	Point Load (z-equivalent)
3	Wind Load (Pressure)	- 0.5	Pressure   Wind Loads
4	Wind Load (Suction)	0.5	Pressure   Wind Loads

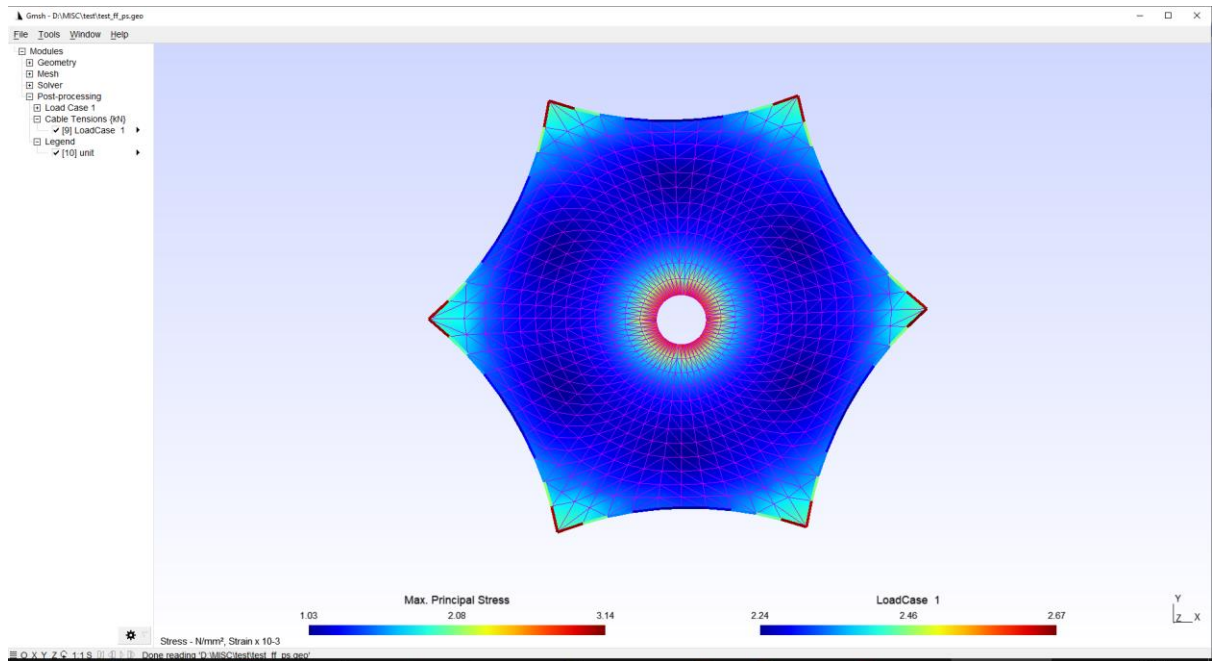
7. Run equilibrium form-finding under **Winfabric | Minimal surface form-finder** choose **Updated Reference Strategy** as the method.



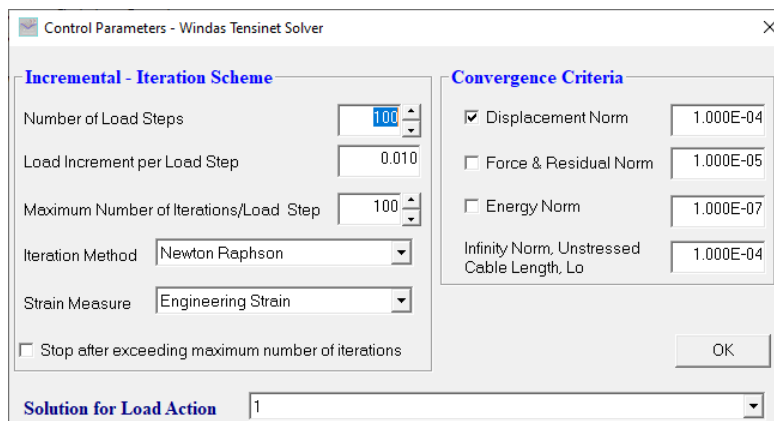
If this process is successful these pop-up messages will appear.



This gmsh plot will pop-up to allow observation on the updated pre-stress and pre-tension.



8. To check the equilibrium condition, run Tensegrity solver on Load case 1 only. The structure deformation has to be close to 0 to be confirmed equilibrium. Use **Solution | Tensegrity**.



**Control Parameters - Windas Tensinet Solver**

**Incremental - Iteration Scheme**

- Number of Load Steps: 100
- Load Increment per Load Step: 0.010
- Maximum Number of Iterations/Load Step: 100
- Iteration Method: Newton Raphson
- Strain Measure: Engineering Strain
- ☐ Stop after exceeding maximum number of iterations

**Convergence Criteria**

- ☒ Displacement Norm: 1.000E-04
- ☐ Force & Residual Norm: 1.000E-05
- ☐ Energy Norm: 1.000E-07
- Infinity Norm, Unstressed Cable Length, Lo: 1.000E-04

**Solution for Load Action**: 1

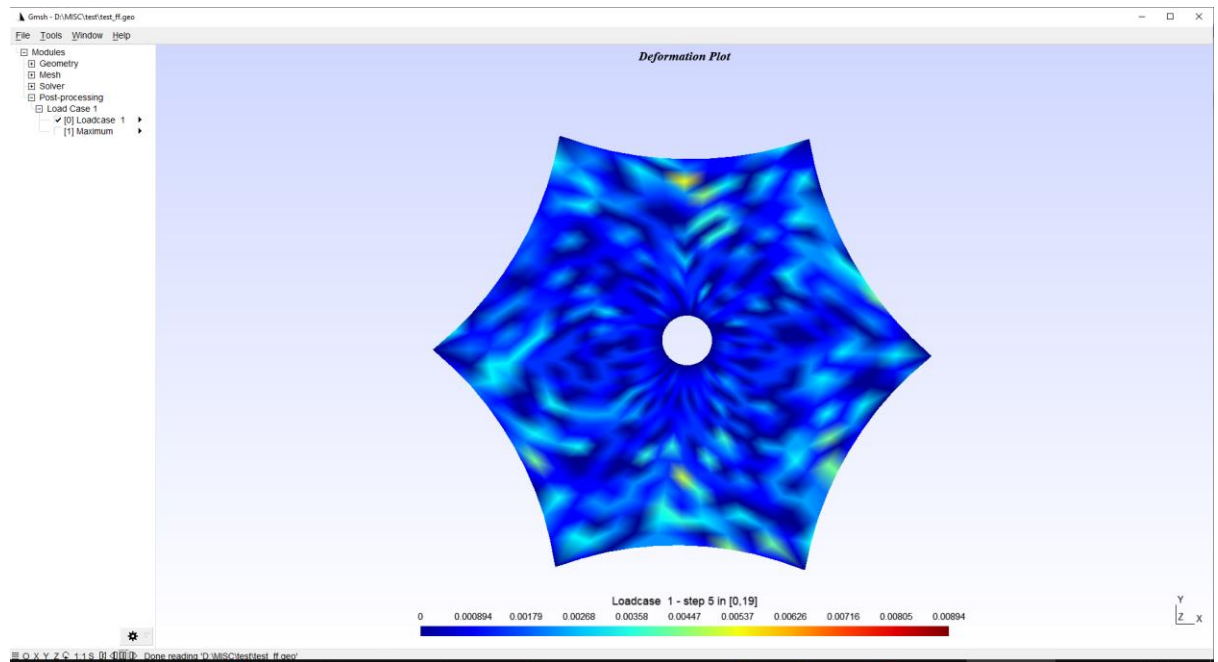
OK

Either use **List | Results | Movement | Maximum** or **Gmsh | Animate | Deform Shape | Loadcase 1** to observe displacement results.

```
project : Windas- A Lightweight Structure Analysis Program

Maximum Displacement List
=====

Loadcase 1 Loadcase Name = Loadcase 1
Maximum x-displacement of      .003 at node 385
Maximum y-displacement of     -.004 at node 622
Maximum z-displacement of      .008 at node 256
```



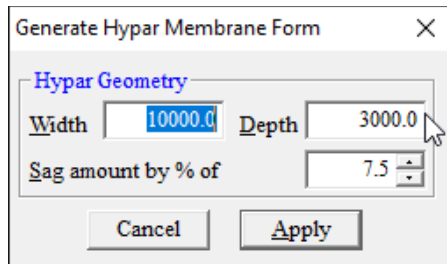
Evidently, the deformations are close to 0. Hence this form-found structural model is in equilibrium.

## 2.2 SIMPLE HYPAR

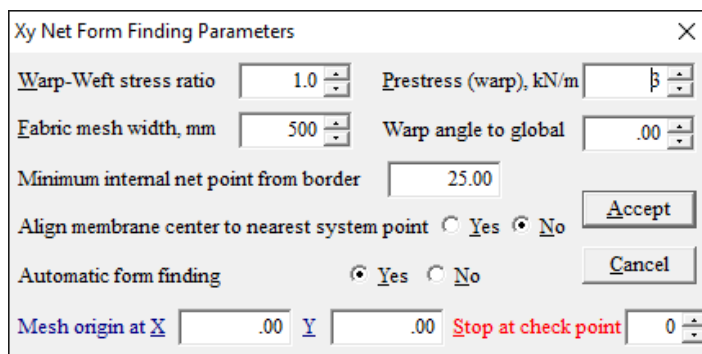
This tutorial shows essential steps to form find and analyse a simple hypar tensile membrane using stressed membrane method in Windas.

### 2.2.1 Initial Geometry

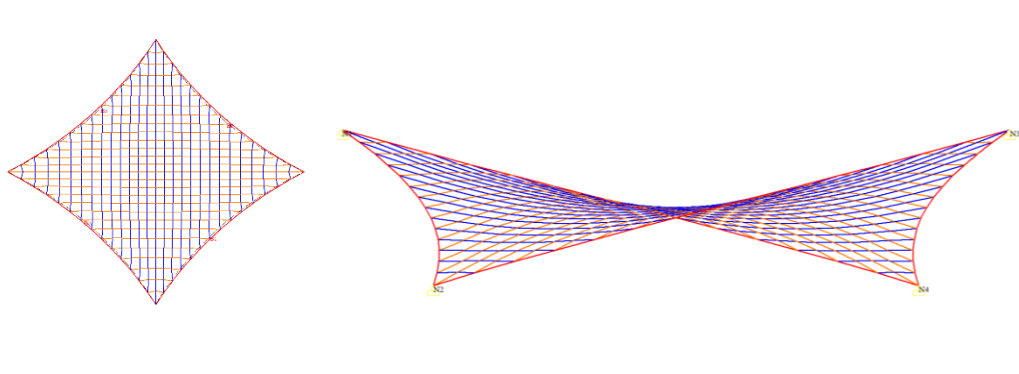
Use Windas to generate Hypar form automatically. Click **Generate | Membrane Forms | Hypar** and the hypar dimensional dialog will appear.



Click **Apply** and a hypar of 10m width and 3m depth with border curvature of 7.5% sag will generate. The XY-net form-finding dialog will appear subsequently. For this exercise, change the fabric width of 500mm and the pre-stress in warp direction to 3.0kN/m.



Obtaining this hypar geometry.




This shape was form found under force density rules. The force density values attached to the cable can be used as a benchmark to gauge the cable pre-tension value that will be applied for stressed membrane analysis.



## Use List | Load | Force Density | C14.

Windas Text Editor - [D:\MISC\test\Adas\_Hypar\_ff\_fd.lst]

File Edit Search Help



Line	Element	FD in kN/m	TYPE	COLOR	LENGTH
1	2	71.989	13	14	788.09
2	3	71.989	13	14	788.09
3	8	72.988	13	14	770.60
4	13	72.988	13	14	770.60
5	22	73.771	13	14	757.57
6	31	73.771	13	14	757.57
7	44	74.372	13	14	747.64
8	57	74.372	13	14	747.64
9	74	74.810	13	14	740.45
10	91	74.810	13	14	740.45
11	112	75.097	13	14	735.77
12	133	75.097	13	14	735.77
13	158	75.238	13	14	733.46
14	183	75.238	13	14	733.46
15	212	75.238	13	14	733.46
16	241	75.238	13	14	733.46
17	274	75.097	13	14	735.77
18	307	75.097	13	14	735.77
19	344	74.810	13	14	740.45
20	381	74.810	13	14	740.45
21	422	74.372	13	14	747.65
22	463	74.372	13	14	747.64
23	508	73.771	13	14	757.57
24	553	73.771	13	14	757.57
25	602	72.988	13	14	770.60
26	651	72.988	13	14	770.60
27	704	71.989	13	14	788.09
28	730	71.989	13	14	788.08
29	734	72.988	13	14	770.60
30	737	73.771	13	14	757.57
31	740	74.372	13	14	747.64
32	743	74.810	13	14	740.45
33	746	75.097	13	14	735.77
34	749	75.238	13	14	733.46
35	752	75.238	13	14	733.46
36	755	75.097	13	14	735.77
37	758	74.810	13	14	740.45
38	761	74.372	13	14	747.65
39	764	73.771	13	14	757.57
40	768	72.988	13	14	770.60
41	770	71.989	13	14	788.09
42	798	71.989	13	14	788.09
43	800	71.989	13	14	788.09
44	804	72.988	13	14	770.60
45	807	73.771	13	14	757.57
46	810	74.372	13	14	747.65
47	813	74.810	13	14	740.45
48	816	75.097	13	14	735.77
49	819	75.238	13	14	733.46
50	822	75.238	13	14	733.46
51	825	75.097	13	14	735.77
52	828	74.810	13	14	740.45

Line:23 Col:59

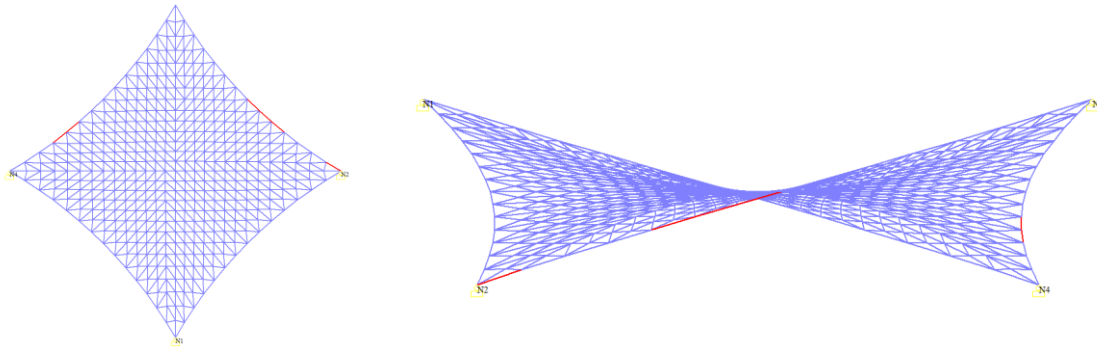
NUM INS

Average number of pre-tension values on border cable elements can be estimated to be 73kN. Once this value is obtained, all force density load need to be erased. Use **Delete | Load | Force Density | All** to do so.

The truss elements are meant to represent the mesh as 1D elements in force density method. Stressed membrane method is a 2D method, hence only membrane and cable element are needed. Thus, use **Delete | Element | type t11** to remove all warp-weft truss elements.

Lastly, to triangulate the mesh, use **Winfabric | Triangulation**.



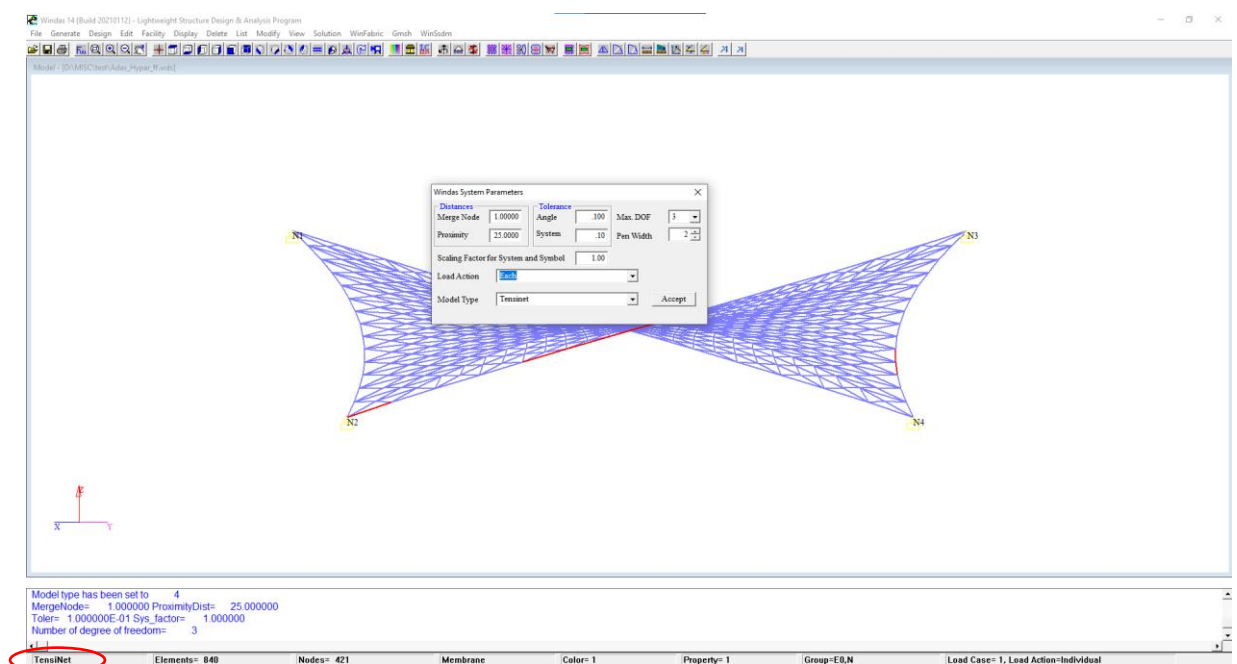


The mesh has been cleared from truss elements and the attached force density value. This model can be used for stressed membrane form finding.

## 2.2.2 Form-Finding

This section will lay out the step by step process of form-finding with stressed membrane method in Windas. Checking equilibrium of final shape will also be demonstrated.

1. Set Windas system to **Tensinet** using **Facility | Set | System | TensiNet**. Check the system setting on the bottom left corner of Windas.



2. To minimise frontwidth for analysis we use **Facility | Reorder | Element Reorder | Fast**.
3. Assign membrane properties using **Generate | Material & properties | Plate-Shell-Solid-Surface**. Windas automatically assigns the membrane element to Property 3. Therefore we can go to Property ID 3, set material as Stressed Skin. The default material given is PTFE FGT-600. Should the material differs, Young's Modulus (E), Shear Modulus, Poisson's Ratio and Density value need to be adjusted accordingly.

Material Property - Plate, Membrane, Shell and Soild

Property ID: 3 Description: Membrane Surface

Material: Stressed Skin Design Strength: 122.70 Thickness, t: 1.00 Surface Pressure: .00 kN/m<sup>2</sup>

Young's Modulus, E: 1284.67 Shear: 94.96 Poisson's Ratio: .57 Density: .16600E-005 Thermal Coeff: .00000E+000

**\*\* Material density use in load case 1**

Click **List | Material & Section Properties** to see that the elements has been set properly.

Wintas Text Editor - [D:\MSC\test\Adas\_Tent\_H\_mats.txt]

Material Property - TensiNet + Membrane

Property ID: 3

=====

Fabric Stiffness Warp E11 = 1284.67kN/m  
Weft E22 = 1284.67kN/m  
Shear G = 94.96kNm

Poisson's Ratio Nu12 = .57  
Nu21 = .57

Section\_ID = \*94 => Stressed Membrane  
Warp Strength = 122.70kN/m

Orthotropic Data not implemented in this version

Material Property Cable

NO	YOUNGS MODULUS	SHEAR MODULUS	AREA OF SECTION	POISSON RATIO	DENSITY	BREAKING STRENGTH	MATERIAL ID	SECTION ID	Designation	Description
4	.1370E+06	.5139E+05	76.10	.33	.7850E-05	1480.00	5	18	12.5mm(7*7) Strand	Stainless Steel Border Cable 7x19 Grade
11	.1100E+06	.4126E+05	113.10	.33	.7850E-05	1470.00	4	17	12mm (1 x 19)	Galvanised Tieback Cable

Line1 Col1 NUM INS

4. Input pre-stress using **Generate | Load | Pre-Stress | C2** (colour code for membrane element). Assume value of  $2\text{N/mm}^2$  for both directions.  
Principal Weft Plane is the perpendicular plane of surface where pre-stress is being applied.  
Hypar shape overall stretches on XY plane. The principal weft plane is XZ plane.

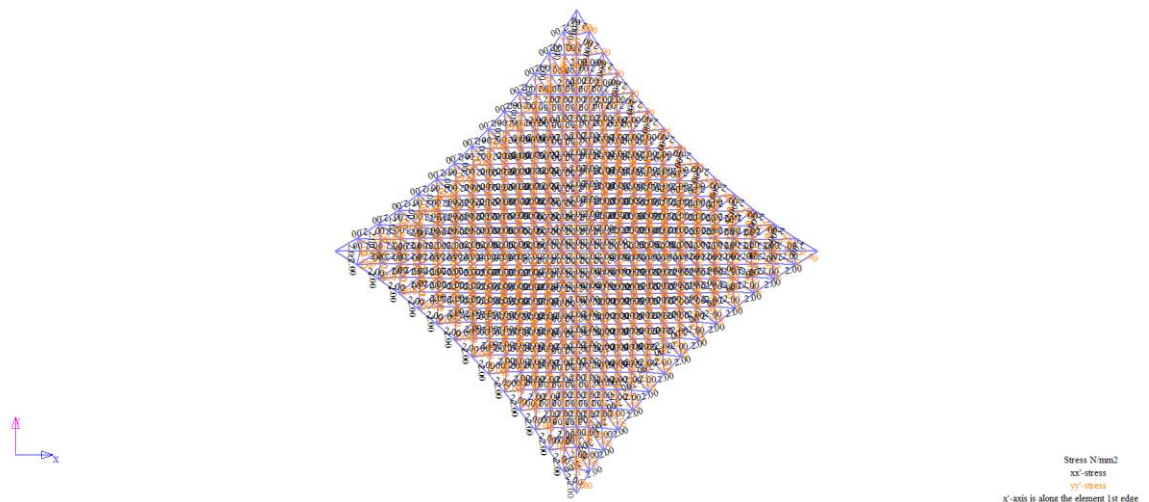
Generate Pre-Stress For Stressed Membrane

XX-Stress (Weft) N/mm<sup>2</sup>: 2

YY-Stress (Warp) N/mm<sup>2</sup>: 2

Principal Weft Plane: ☒ XZ ☐ XY ☐ User

Use **Display | Load value | pre-stress** to check.

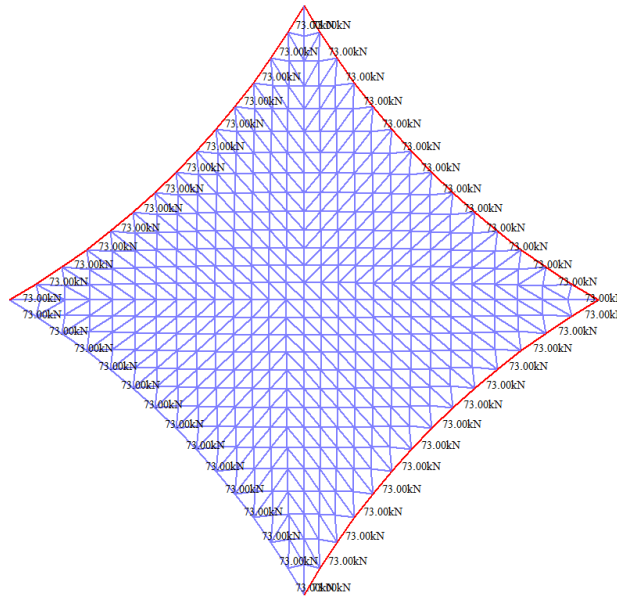


5. Input pre-tension using **Generate | Load | Pre-Tension | C14** (colour code for border cable element).

Pretension Selected Cable(s)

Pre-Tension Value (kN): 73

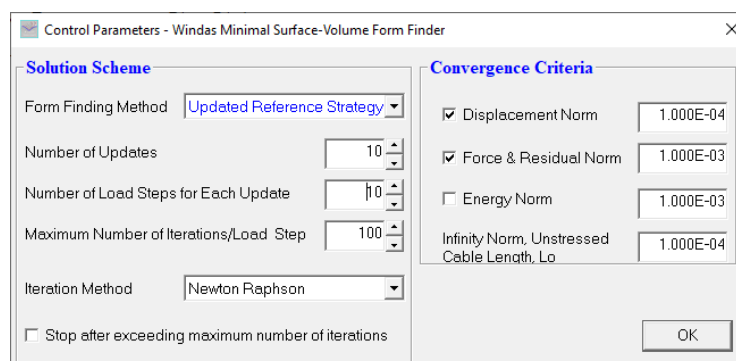
Use **Display | Load value | pre-tension** to check.



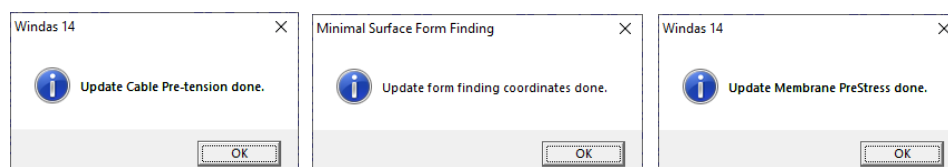
6. It is recommended to apply loading before form-finding.  
Use **Generate | Load** command and apply the load accordingly. Below is shown the loading normally used for our projects.

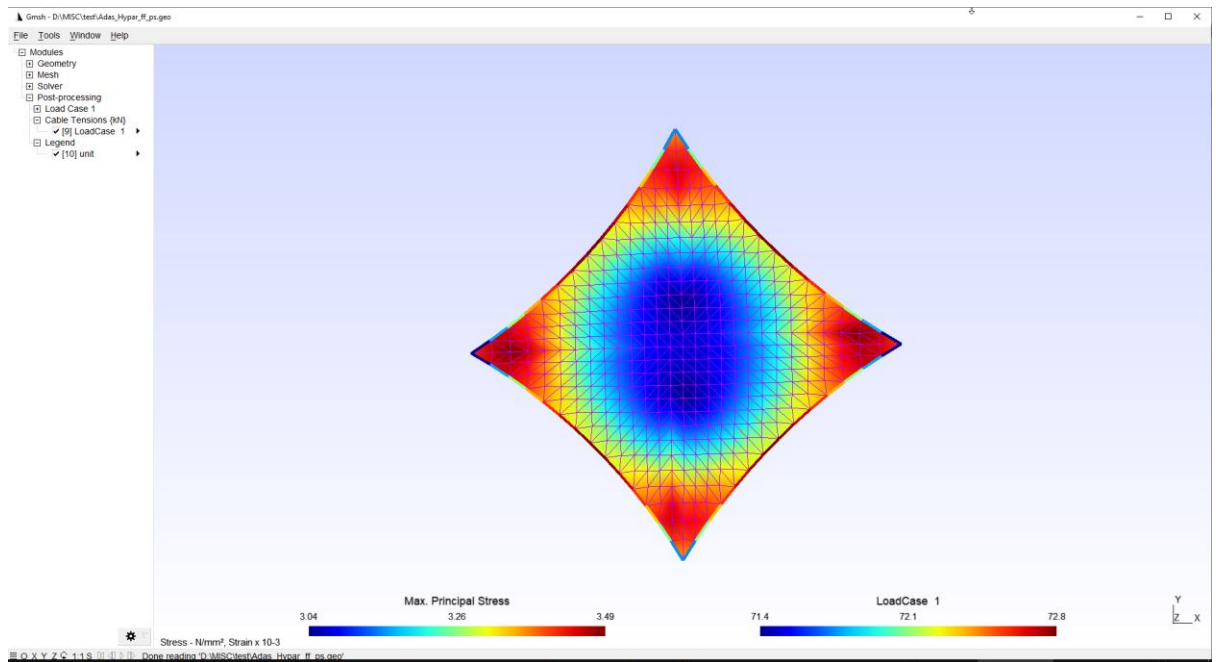
Loadcase		Value (kN/m <sup>2</sup> )	Input Load Type
1	SW + PS + PT	-	-
2	Rain Load	- 0.1	Point Load (z-equivalent)
3	Wind Load (Pressure)	- 0.5	Pressure   Wind Loads
4	Wind Load (Suction)	0.5	Pressure   Wind Loads

7. Run equilibrium form-finding under **Winfabric | Minimal surface form-finder** choose **Updated Reference Strategy** as the method.

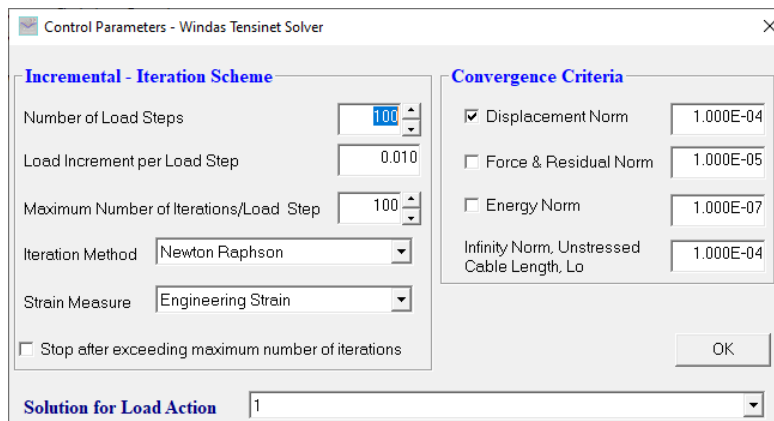


If this process is successful these pop-up messages will appear.





8. To check the equilibrium condition, run Tensegrity solver on Load case 1 only. The structure deformation has to be close to 0 to be confirmed equilibrium. Use **Solution | Tensegrity**.

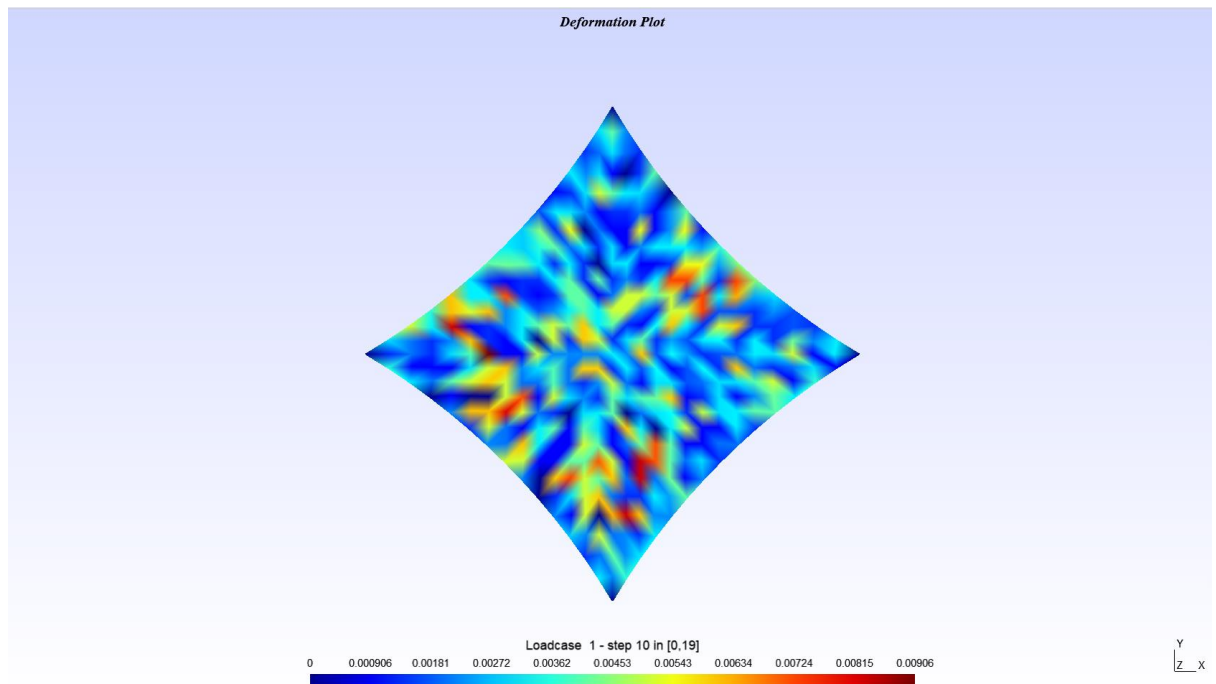


Either use **List | Results | Movement | Maximum** or **Gmsh | Animate | Deform Shape | Loadcase 1** to observe displacement results.

```

project : Windas- A Lightweight Structure Analysis Program
Maximum Displacement List
=====

Loadcase 1 Loadcase Name = Loadcase 1
Maximum x-displacement of      -.002 at node 224
Maximum y-displacement of      .002 at node 385
Maximum z-displacement of      -.009 at node 202
  
```



Evidently, the deformations are close to 0. Hence this form-found structural model is in equilibrium.

### 3 MESH MODEL FROM EXTERNAL TOOL (RHINO)

Tensile membrane project allows a wide variety of shapes and forms. For irregular shapes with unique boundary conditions, creating initial geometry from other meshing tool might be necessary. In this exercise, a triangulated mesh created by Rhino 7 will be used.

The shape is essentially made of 3 uniform hypar attached to each other. Some unique steps to accommodate the mesh in Windas will be demonstrated.

#### 3.1 INITIAL GEOMETRY

This model is obtained from Rhino 7.

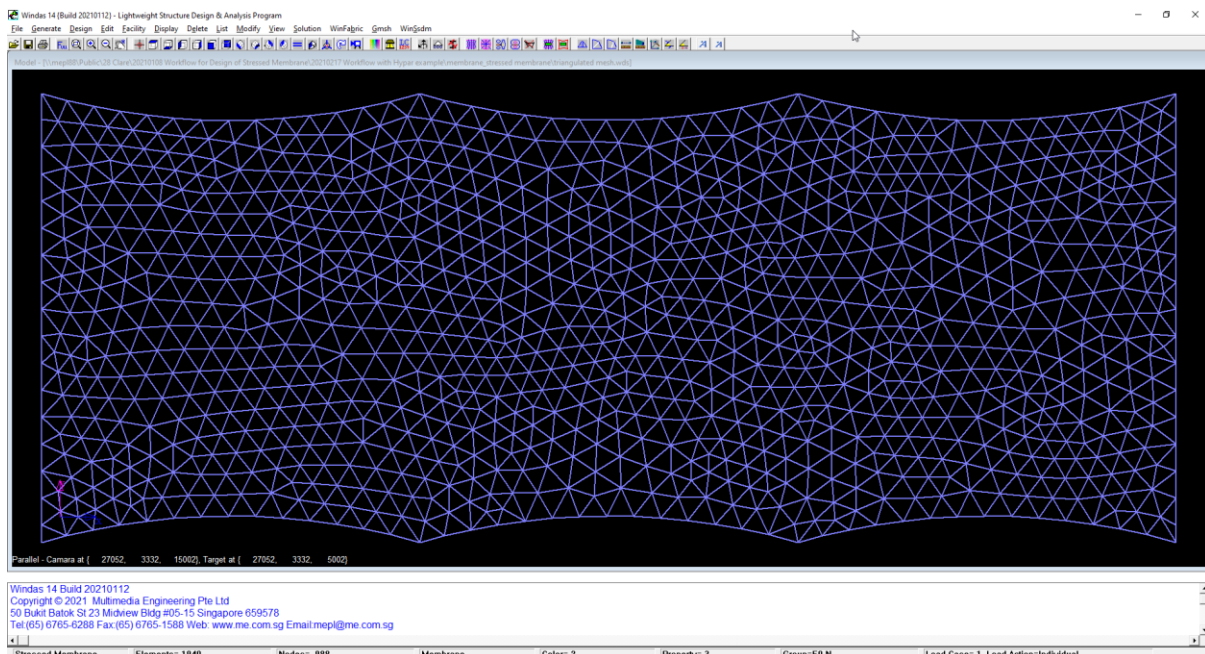
To successfully transfer this model from Rhino 7 to Windas, only mesh and line elements are allowed. The mesh needs to be a good joined mesh. The mesh has to connect at all interfacing points and made of uniform sizes mesh.

Border cable elements has to match the mesh edges accordingly. The border cable elements need to share the same node locations as the mesh itself.

Save as .dxf file.

The following steps will explain the alteration work that needs to be done in Windas

1. In Windas interface, use **File | Import | AutoCAD {DXF R12}**



2. Set Windas system to **Tensinet** using **Facility | Set | System | TensiNet**.
3. As this model was obtained from external source, the element ID have not been set according to Windas database. The elements should be addressed as follows :

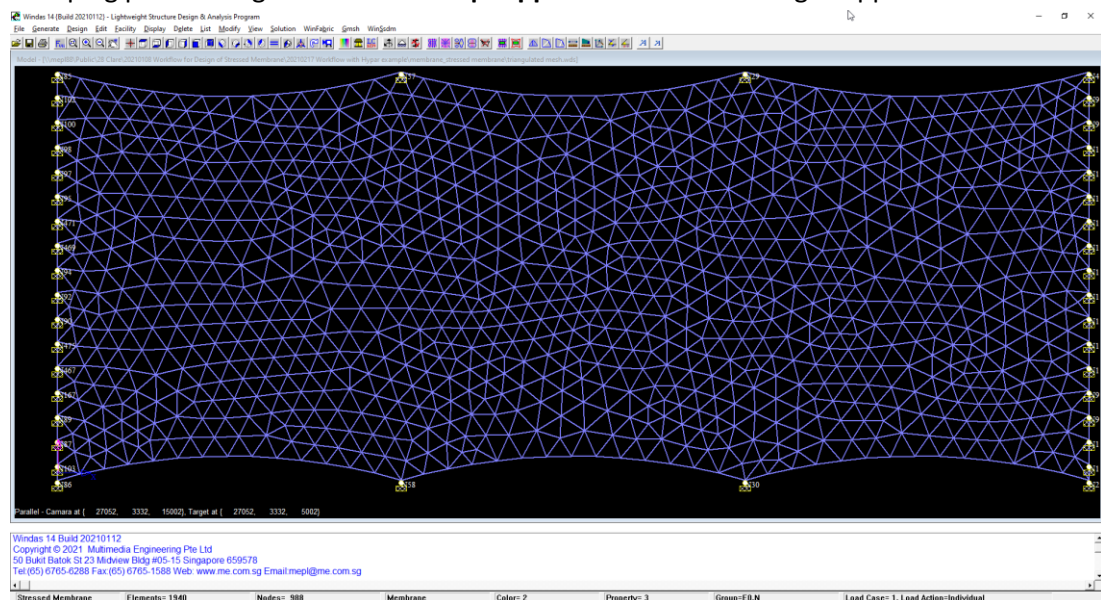


	Color ID	Property ID	Element Type
Mesh	2	3	Membrane
Border Cable	14	4	Cable 3D

Use : **Edit | Element Attributes | C2 | Property 3 | Element type : Membrane.**

**Edit | Element Attributes | C14 (or other colour code that the cables are assigned to) | Property 4 | Element type : Cable 3D.**

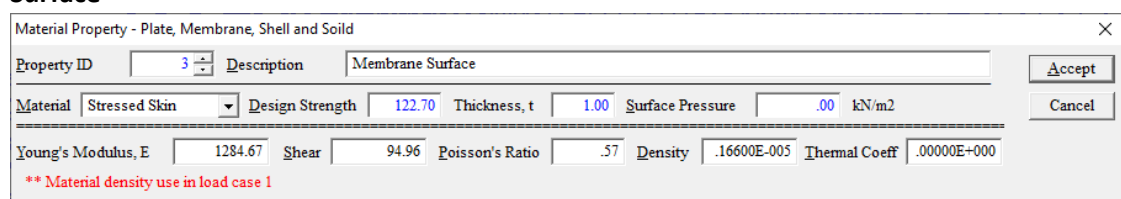
- In this model, we need to define the support points.  
Fixed edges are represented by pinned support along the line.  
Border cables are not supposed to be represented by any support.  
However, it is necessary to define a single support at the end of border cables to define the clamping plates design. Use **Generate | Support** command to assign supports.



Suppose everything has been set accordingly, the same rules of form finding and analysis should apply.

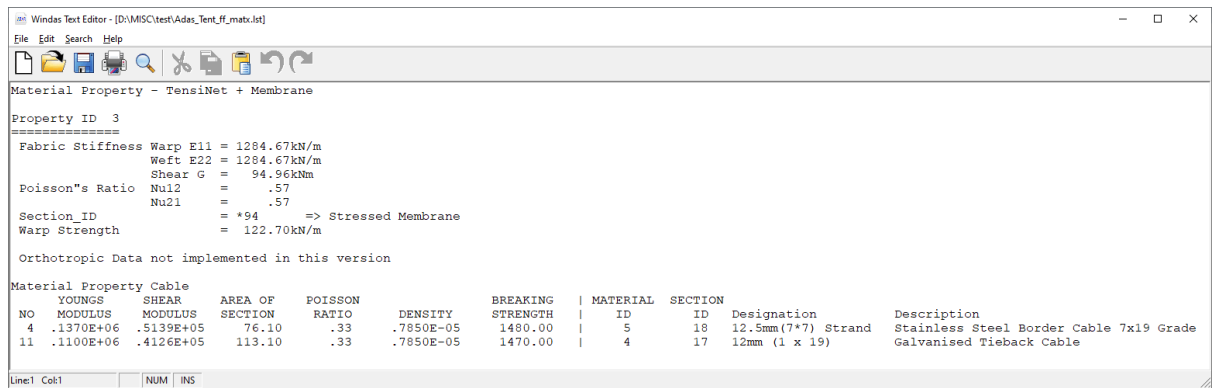
## 3.2 FORM FINDING

- To minimise frontwidth for analysis we use **Facility | Reorder | Element Reorder | Fast.**
- Assign membrane properties using **Generate | Material & properties | Plate-Shell-Solid-Surface**

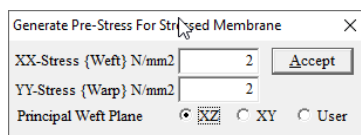


Click **List | Material & Section Properties** to see that the elements has been set properly.

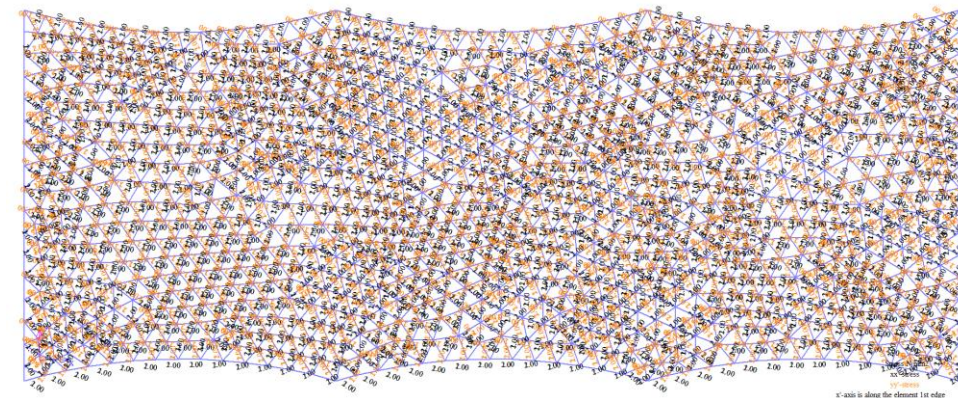




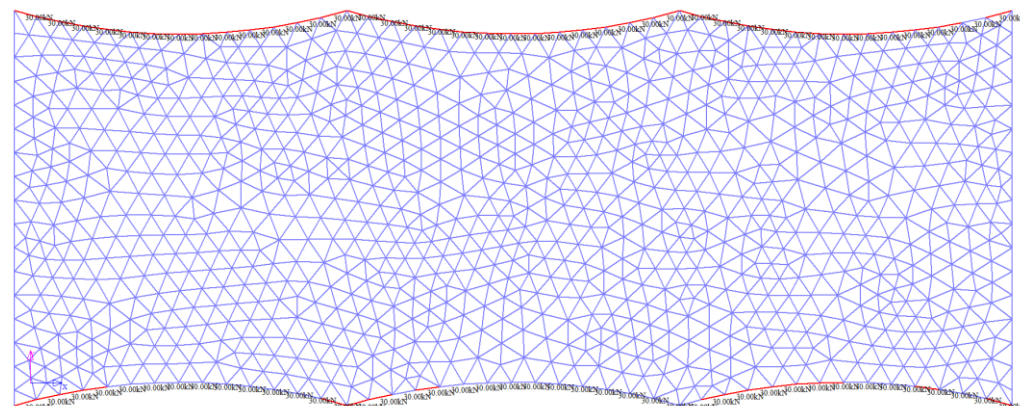
- Input pre-stress using **Generate | Load | Pre-Stress | C2** (colour code for membrane element). Assume value of  $2\text{N/mm}^2$  for both directions. Hypar shape overall stretches on XY plane. The principal weft plane is XZ plane.



Use **Display | Load value | pre-stress** to check.



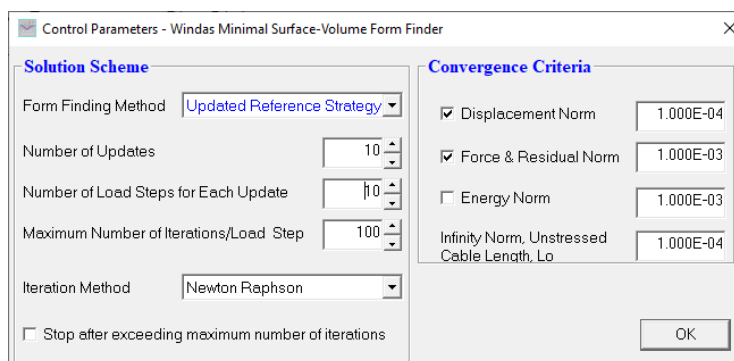
- Input pre-tension using **Generate | Load | Pre-Tension | C14** (colour code for border cable element). Initial force density value is not available in this case. Pre-Tension value shall be gauged according to engineering judgement. In this exercise, pre-tension of 30 kN is used. Use **Display | Load value | pre-tension** to check.



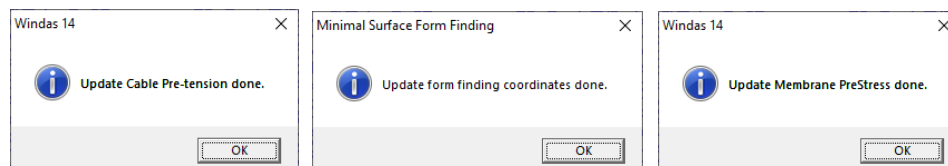
5. It is recommended to apply load before form-finding in Windas.  
Use **Generate | Load** command and apply the load accordingly. Self-weight has been automatically applied by Windas.

Loadcase		Value (kN/m <sup>2</sup> )	Input Load Type
1	SW + PS + PT	-	-
2	Rain Load	- 0.1	Point Load (z-equivalent)
3	Wind Load (Pressure)	- 0.5	Pressure   Wind Loads
4	Wind Load (Suction)	0.5	Pressure   Wind Loads

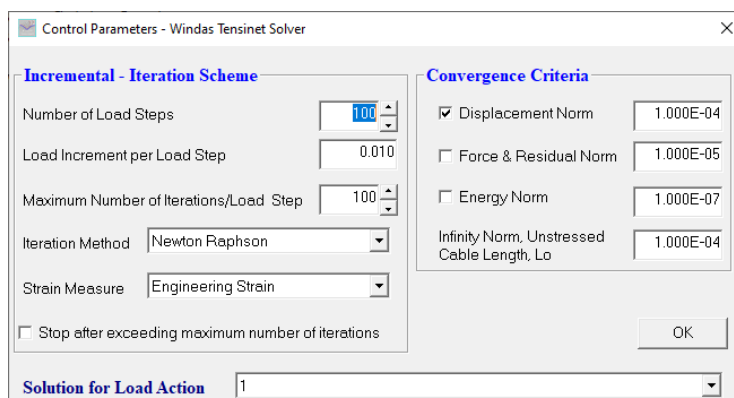
6. Run equilibrium form-finding under **Winfabric | Minimal surface form-finder** choose **Updated Reference Strategy** as the method.



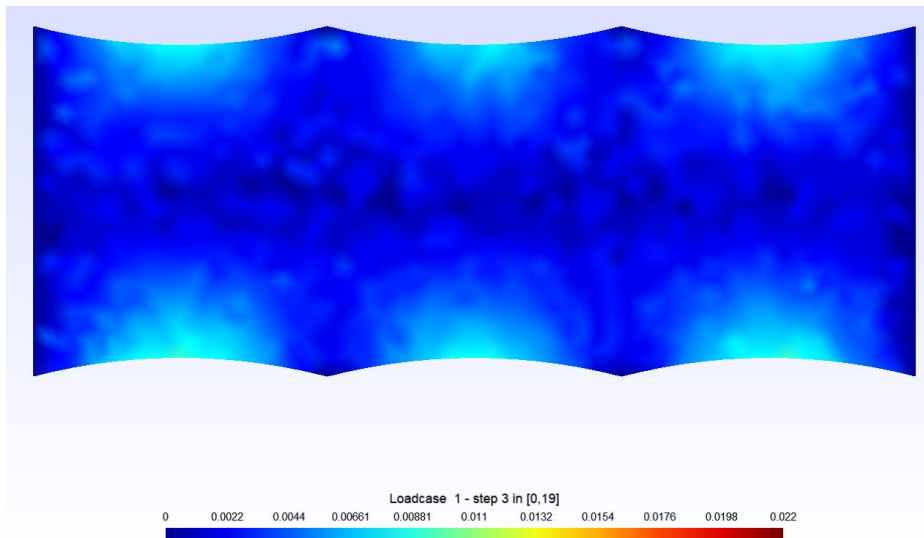
If this process is successful, these pop-up messages appear.



7. To check the equilibrium condition, run Tensegrity solver on Load case 1 only. The structure deformation has to be close to 0 to be confirmed equilibrium. Use **Solution | Tensegrity**.



We can either use **List | Results | Movement | Maximum** or **Gmsh | Animate | Deform Shape | Loadcase 1** to observe displacement results.



Evidently, the deformations are close to 0. Hence this form-found structural model is in equilibrium.

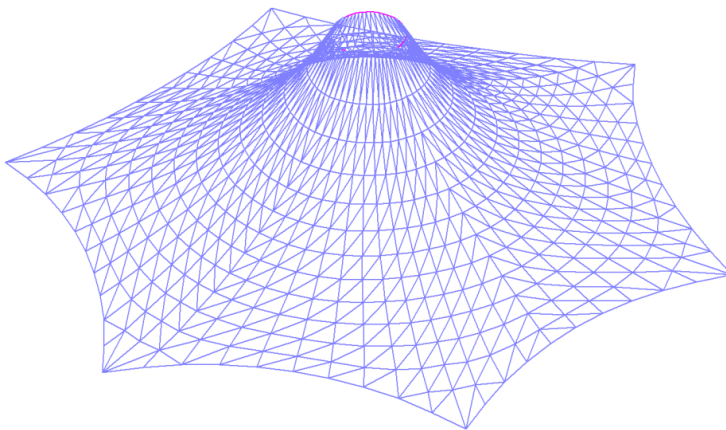
## 4 MESH REFINEMENT

Uniformly distributed triangle mesh is an important condition to achieve for Stressed Membrane analysis. More refined model will yield more realistic analysis result.

Mesh refinement can be done easily in Windas. Alternatively, mesh refinement can be done with more control parameters in Rhino/Grasshopper. Rhino/Grasshopper meshing tool is also capable of converting xy-mesh to fully triangular unstructured mesh. In special cases where cable partition is required, this mesh refinement using Grasshopper will be necessary.

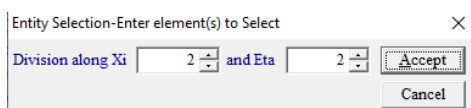
### 4.1 MESH REFINEMENT IN WINDAS

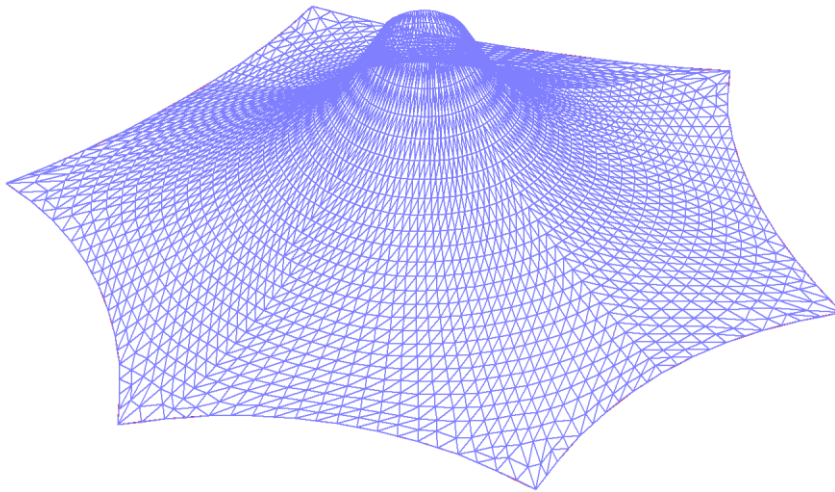
Given this xy-net simple conical polygon mesh model that has been form-found from previous exercise.



Use **WinFabric | Refinement | C2** (indicating the mesh color property) | **Accept**.

This command window indicates the refinement scale of the mesh. Accept default values to get twice finer mesh in both warp-weft directions.

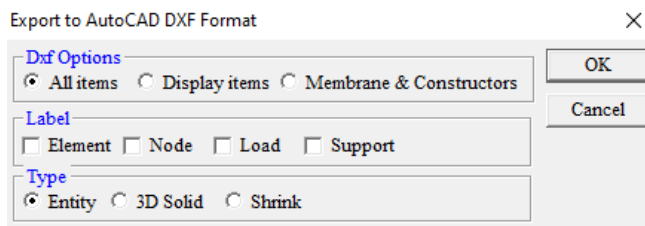




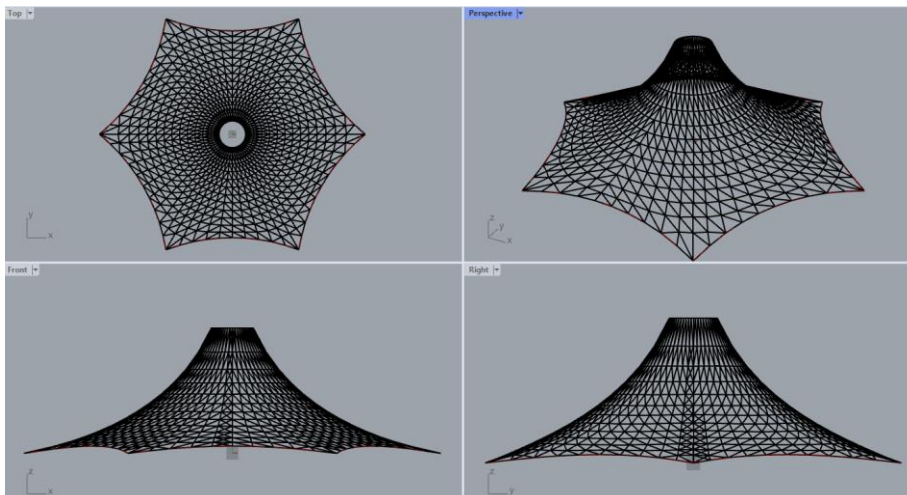
## 4.2 MESH REFINEMENT IN RHINO/GRASSHOPPER

Referring to the same simple conical polygon previously, this section lay out the sequential steps of using Rhino/Grasshopper meshing tool to refine it. In this section, **Kangaroo** Grasshopper plug-in must be installed to user's PC.

1. Working from Windas, export the mesh to .dxf format using **File | Export | AutoCAD {DXF12} | OK**. Accept the default from command window below.

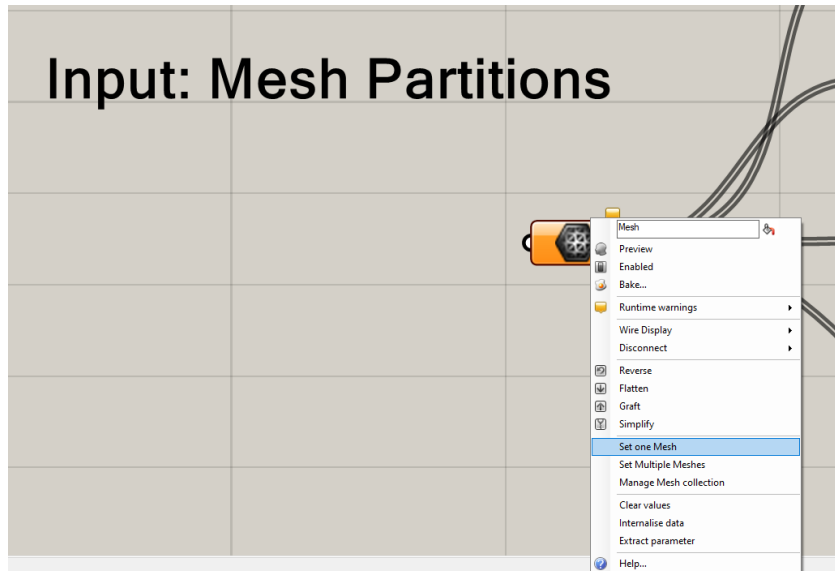


2. Open this .dxf file using Rhino 7 for this example. Mesh elements will be stored in layer **Color\_2** whereas the cable element will be stored in **Color\_14**. Refining the mesh means that the mesh length will change. In this case, the cable element will not be relevant anymore after mesh refinement. It is advisable to delete the cable element in this stage before refinement.

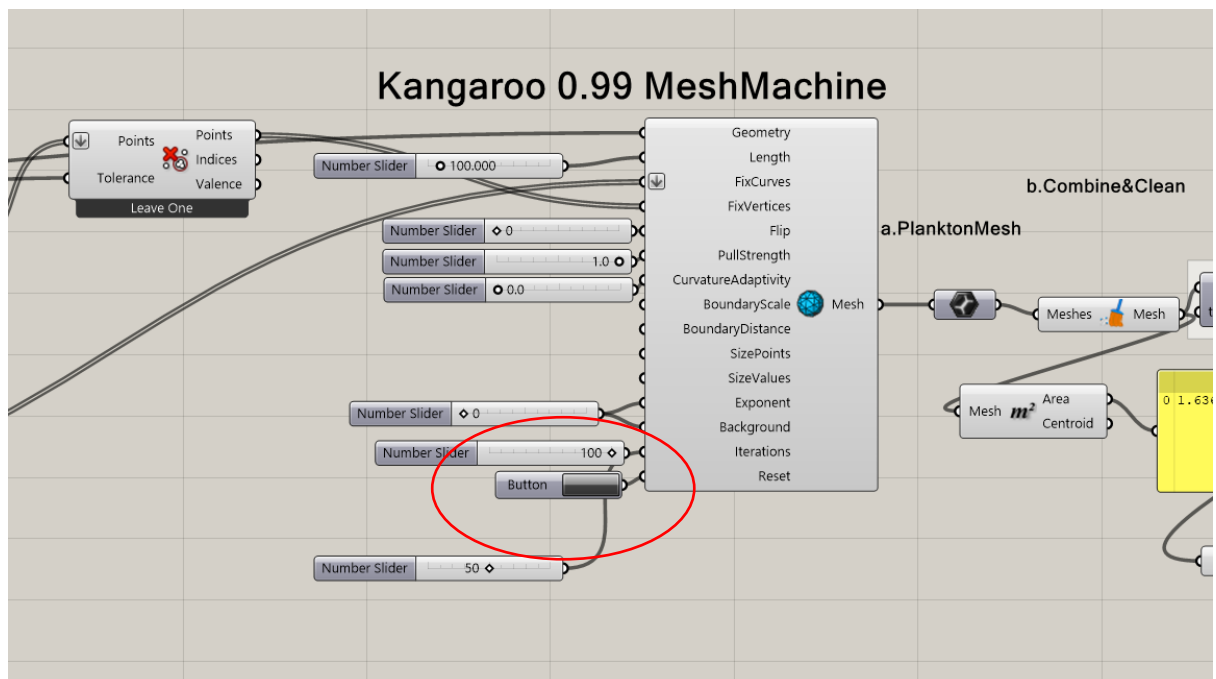




3. Open **Grasshopper** in Rhino 7. **Please copy this script in your PC** and drag this script to the **Grasshopper** interface : \\ithink\Public\00 Staff Orientation Programme\04 Design & Engineering\02 Software Tutorials\06 Meshing with grasshopper\Mesh triangulation with cable split lines\triangulate mesh.gh
4. To assign the mesh to this script, click right on the mesh input component -> set one mesh. The Rhino 7 window will automatically appear, choose on the mesh available.



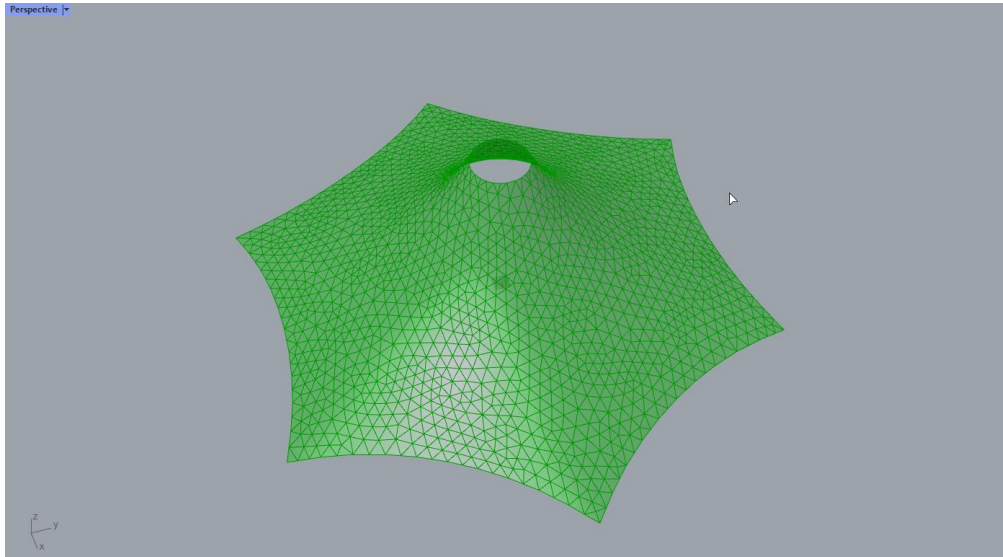
Once the mesh has been recognized, click the button to run the Kangaroo MeshMachine as to recombine the mesh geometry.



Note that the complete tutorial to use this script is available in the same directory :  
\\ithink\Public\00 Staff Orientation Programme\04 Design & Engineering\02 Software  
Tutorials\06 Meshing with grasshopper\Mesh triangulation with cable split lines

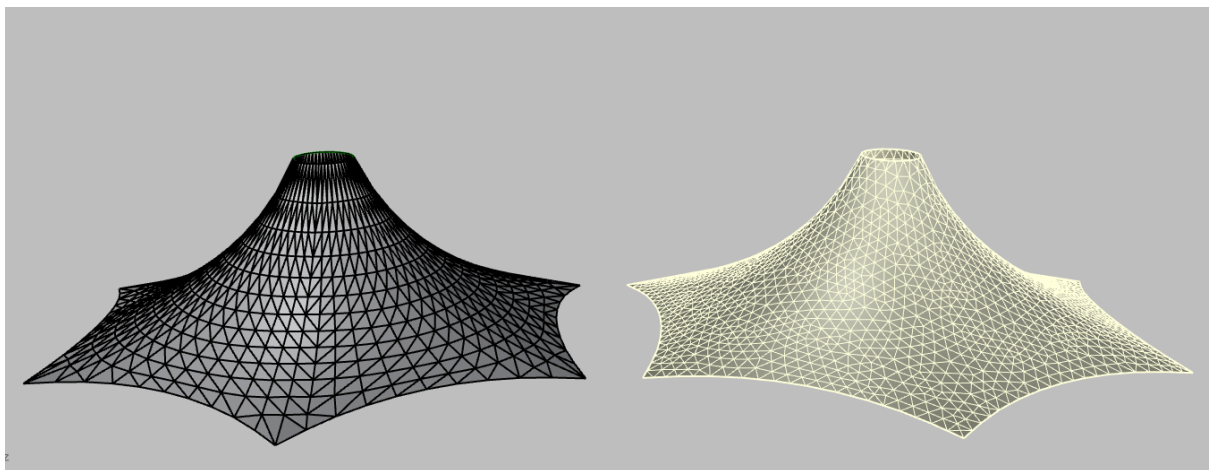
Shall there be any issue while using the script, please refer to the complete tutorial.

5. Click on the “Meshes Mesh” component to preview the updated mesh in Rhino 7 Window.



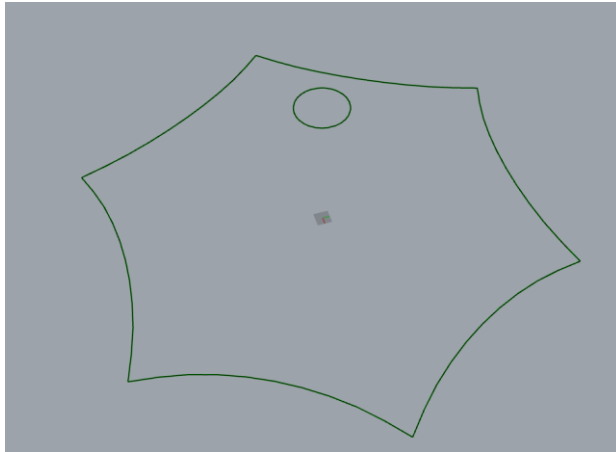
As observed, the triangulated mesh is now evenly distributed and more uniform than the xy-net mesh. Some very skewed mesh near the top ring can be eliminated with this method. To get the mesh out as geometry, click right on the “Meshes” component, choose Bake. Baking mesh does not require a specific Layer, any layer will do.

Figure below shows in comparison how the mesh presents before and after refinement.

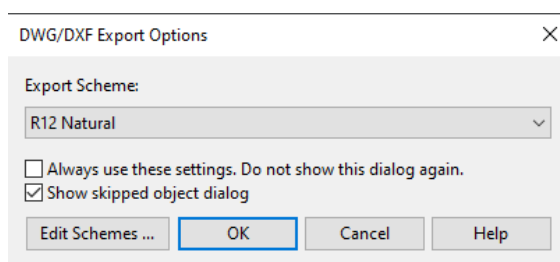
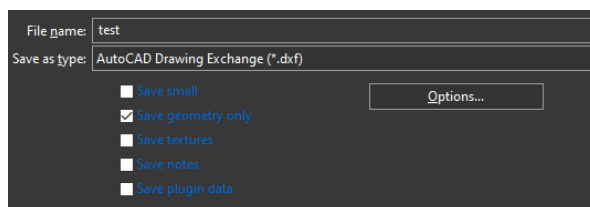


6. To get the updated border cable, click on “Naked Edges”, choose to Bake it on layer : **Color\_14**. These lines are to represent the border cable element according to the new mesh.





7. Export both the mesh and the border cable that was just Baked. Select all the above-mentioned elements, type export. Choose to export to .dxf file and make sure “geometry only” box is ticked. R12 Natural shall work.



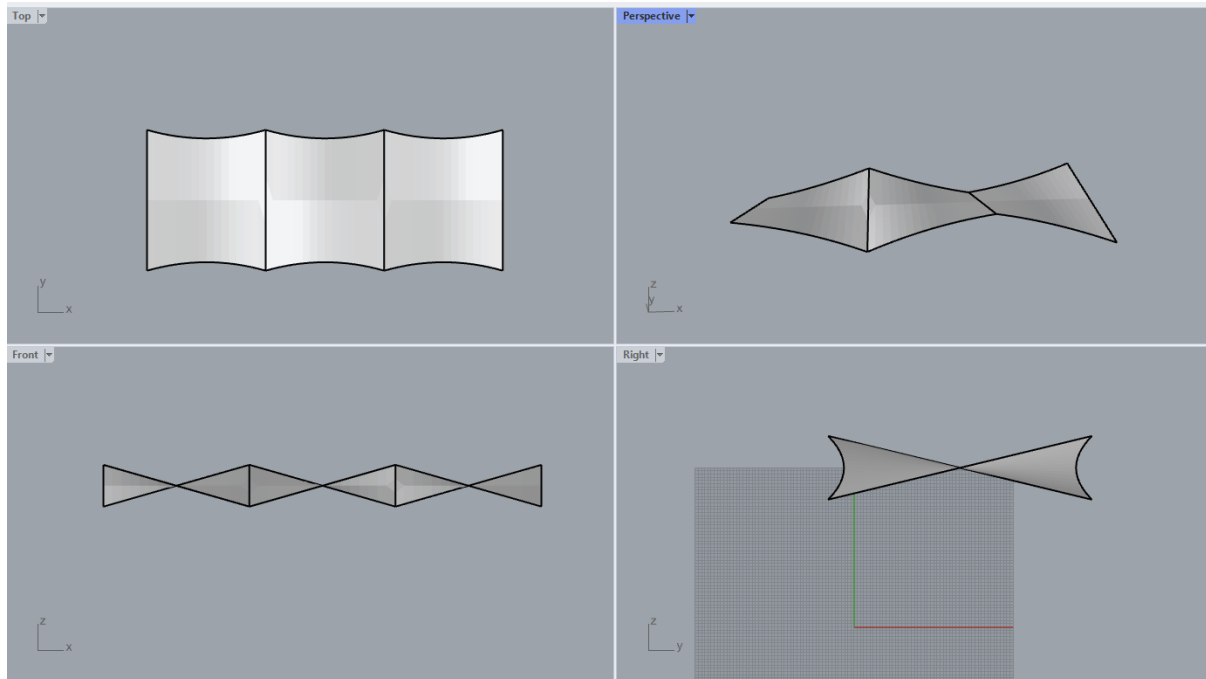
8. Open Windas interface. Click **Import | AutoCAD {dxf R12}** choose the .dxf file. From here on, tutorial Section 3 may be used.

### 4.3 MESH CREATION IN RHINO / GRASSHOPPER

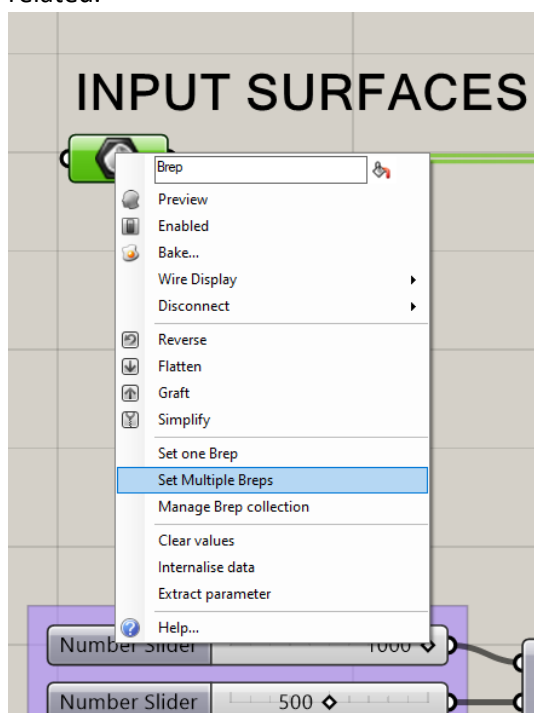
In this section, membrane model will be generated directly from Rhino 7. The reference model is a simple triple hypar that was presented in previous section. The tool allows mesh triangulation directly from multiple surfaces with several separating lines. Installing **Iguana** grasshopper plug-in to your PC is necessary.

#### 4.3.1 Meshing Surfaces

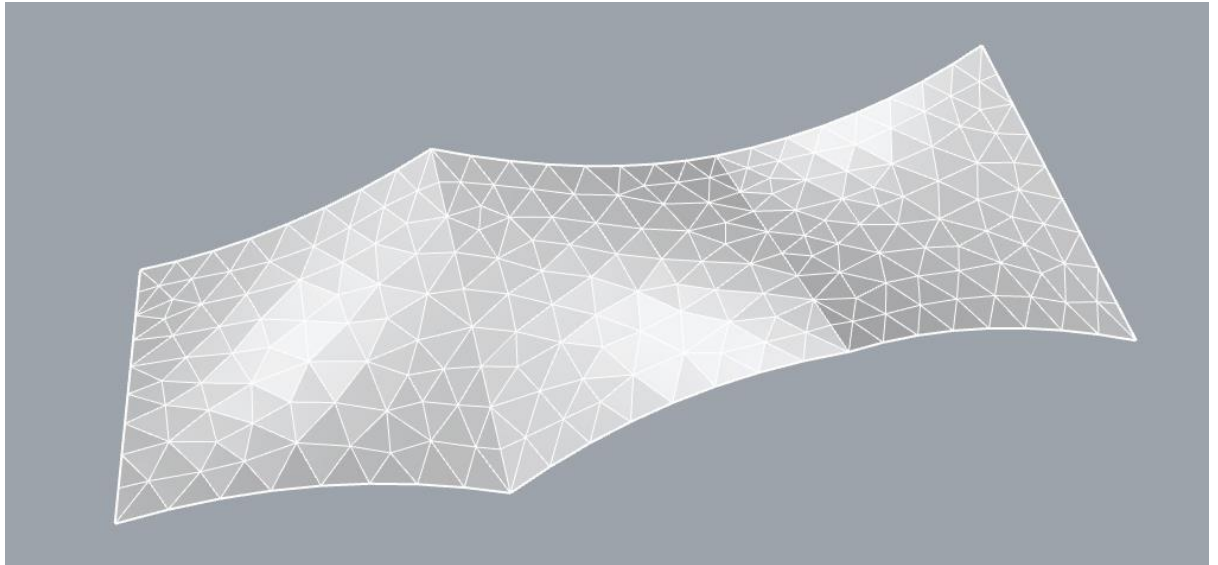
A triple hypar model consists of 3 single hypar surfaces as shown below. The aim of this meshing tool is to produce uniformly distributed triangular single mesh. Instead of having 3 separate meshes, this tool allows the mesh to be fully joined and connected, proper for analysis.



1. Copy to your PC the script from this directory : [\\ithink\Public\00 Staff Orientation Programme\04 Design & Engineering\02 Software Tutorials\06 Meshing with grasshopper\Mesh triangulation of surface\Mesh Triangulation.gh](#)  
Open grasshopper and drag the script in.
2. Pick the brep input button, click right, click multiple brep. Please choose all the surfaces related.



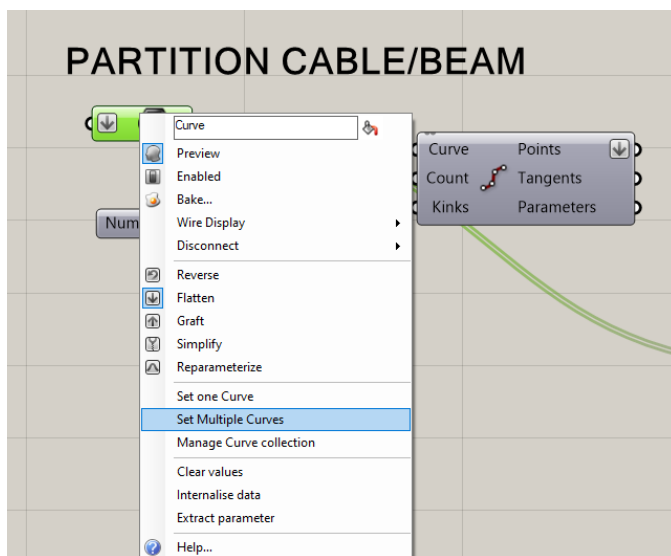
3. Once the input has been set, preview the triangulated mesh right away from the “Meshes button”. Bake in any layer. As observed, the separated meshes now are joined together neatly on each nodes.



4. Bake “Naked Edges” to get suitable perimeter elements.

#### 4.3.2 Meshing with Partition Cable

Suppose the mesh model is designed with reinforcement cable. The mesh itself need to be dissected accordingly. To get appropriate cable elements, use the same script above. Pick the input curve button, right click, click multiple curves, and choose all the partition cables.



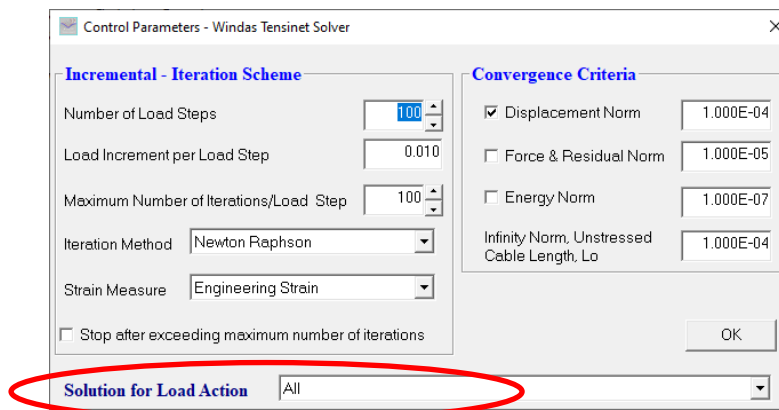
Bake the resulting curves.

## 5 STRUCTURAL ANALYSIS

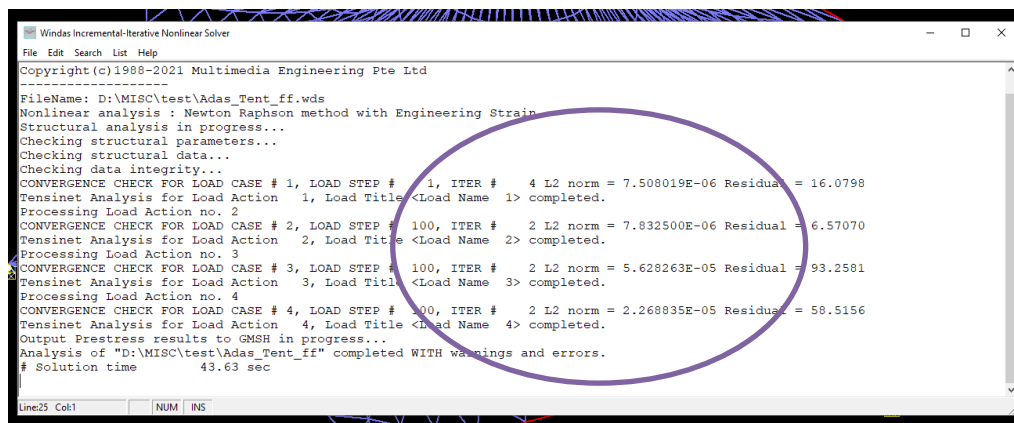
This section is to explain how to do stressed membrane loading analysis in Windas. Mesh models from previous sections will be used as examples, assuming there is no change of model since form-finding. Some components to look out for will also be discussed.

### 5.1 SIMPLE CONICAL POLYGON

1. Open the mesh model in Windas. Run Tensegrity solver again for **All** load cases.



Windas will print out this text window to show briefly how the analysis runs.

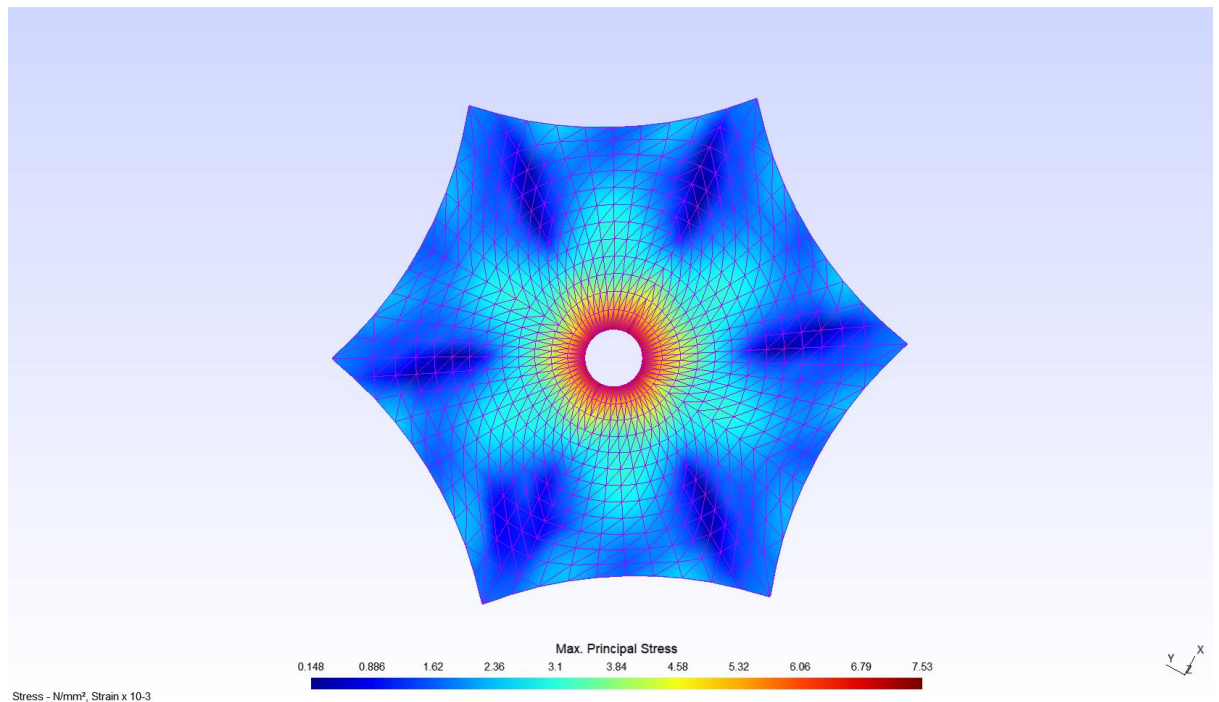


Ideally, all displacement norm should converge to a very small number.

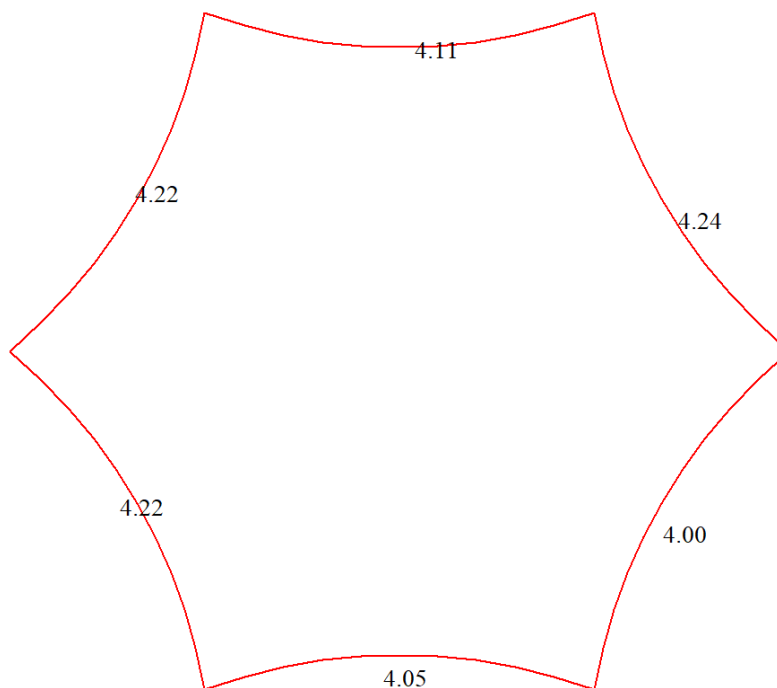
In the event where convergence is not found, the control parameters of Windas should be adjusted before running analysis.

2. View the maximum principal stresses of our membrane using gmsh plot. Use **Gmsh | Stressed membrane stress plot** to view.

The figure below shows the maximum principal stress plot of loadcase 3 (pressure wind load). In practice, we pay attention to areas where the stresses are concentrated. If the safety factor does not meet criteria, we may consider mesh reinforcement in these particular area.

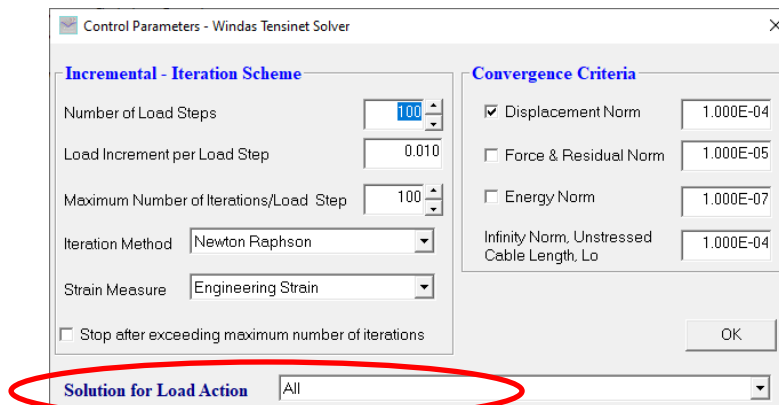


3. Maximum Factored Border Cable Tension can also be obtained. Display using **View | Result | Border cable tension (max)** to show the maximum values in kN.

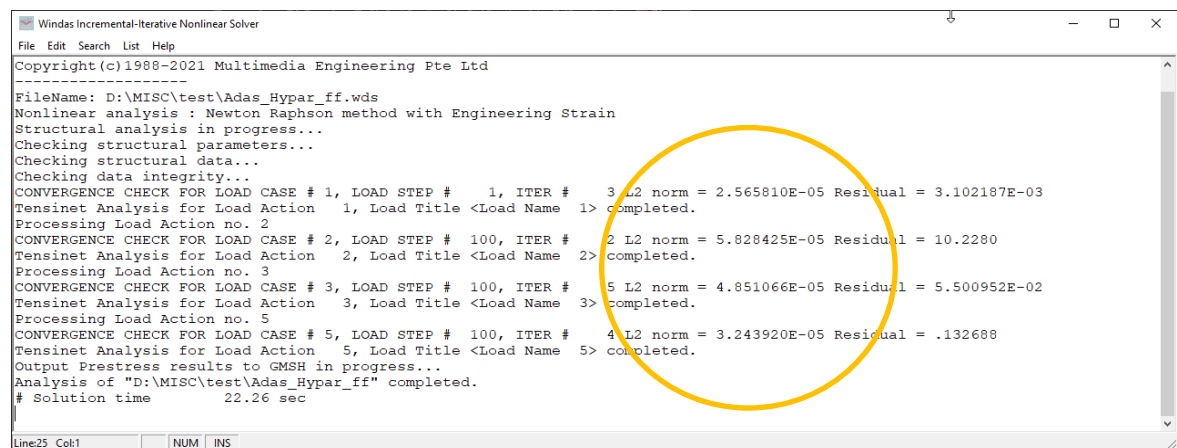


## 5.2 SIMPLE HYPAR

1. Run Tensegrity solver again for **All** load cases.



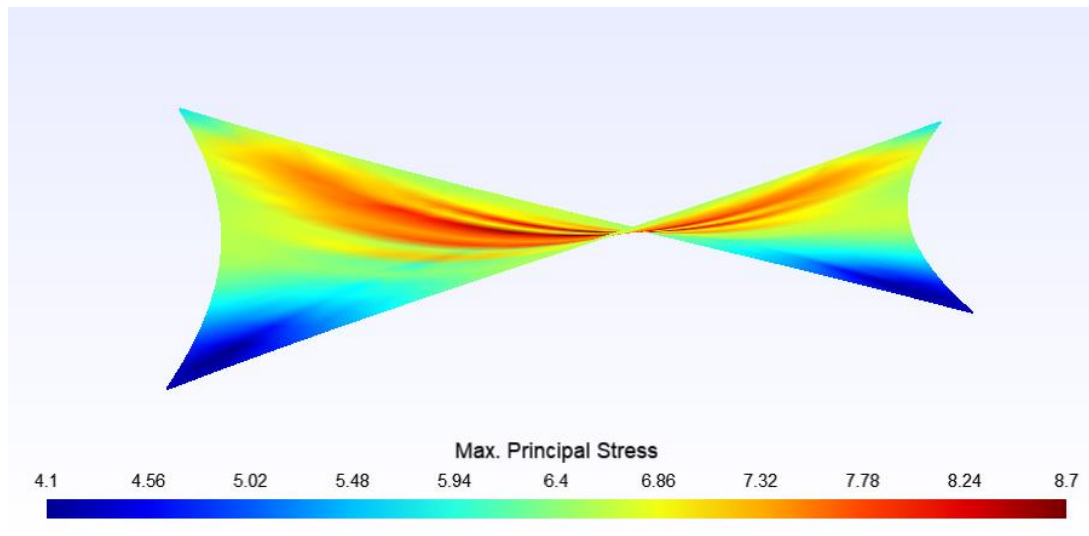
Windas will print out this text window to show briefly how the analysis runs.



Ideally, all displacement norm should converge to a very small number.

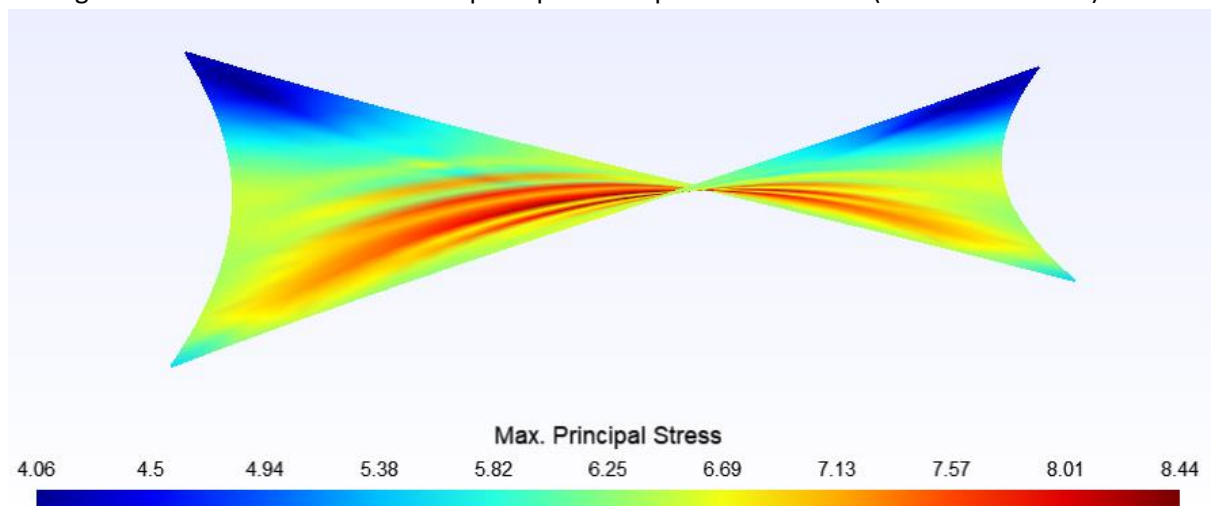
In the event where convergence is not found, the control parameters of Windas should be adjusted before running analysis.

4. View the maximum principal stresses of our membrane using gmsh plot. Use **Gmsh | Stressed membrane stress plot** to view.  
The figure below shows the maximum principal stress plot of loadcase 3 (pressure wind load).



As the hypar membrane is being pressured downwards by the windload, the stresses on the high points of hypar will be greater than stresses on the low points. As observed, high stresses will occur across the two high points.

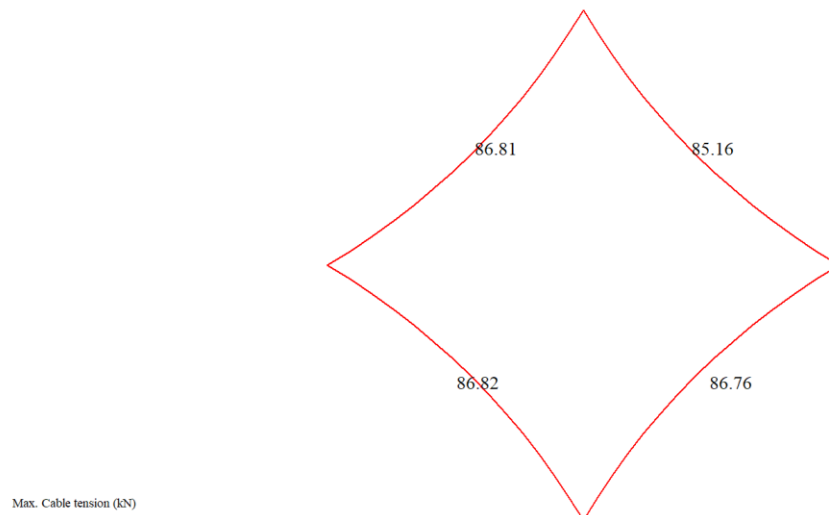
The figure below shows the maximum principal stress plot of loadcase 4 (suction wind load).



Contrasting to loadcase 3, suction wind load results in greater stresses on the lower points as compared to the high points. As expected, the high stresses occur across the two low points as well.

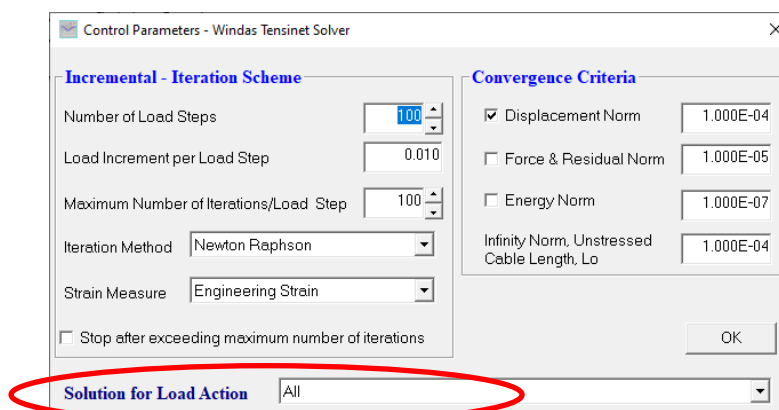
- Maximum Factored Border Cable Tension can also be obtained. Display using **View | Result | Border cable tension (max)** to show the maximum values in kN.



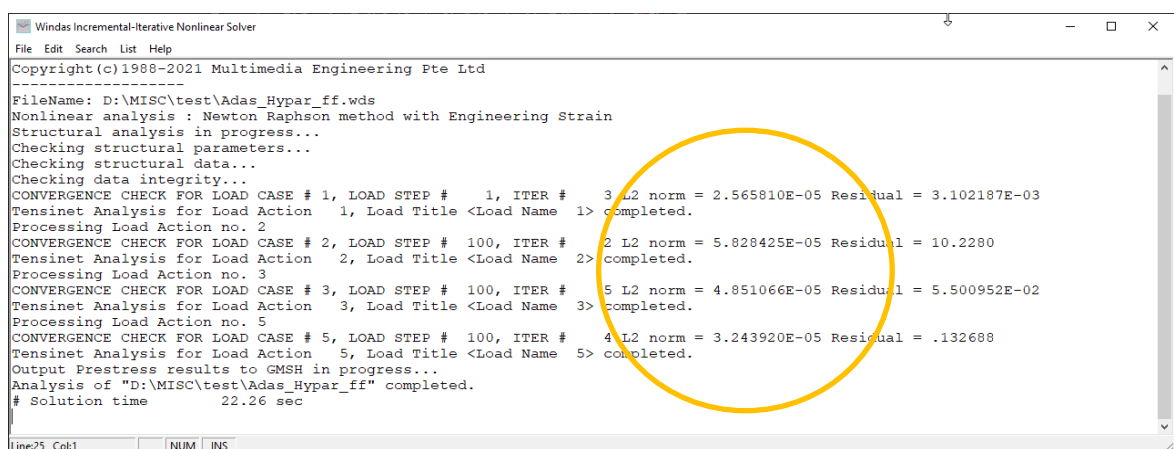


### 5.3 TRIPLE HYPAR

1. Run Tensegrity solver again for **All** load cases.



Windas will print out this text window to show briefly how the analysis runs.

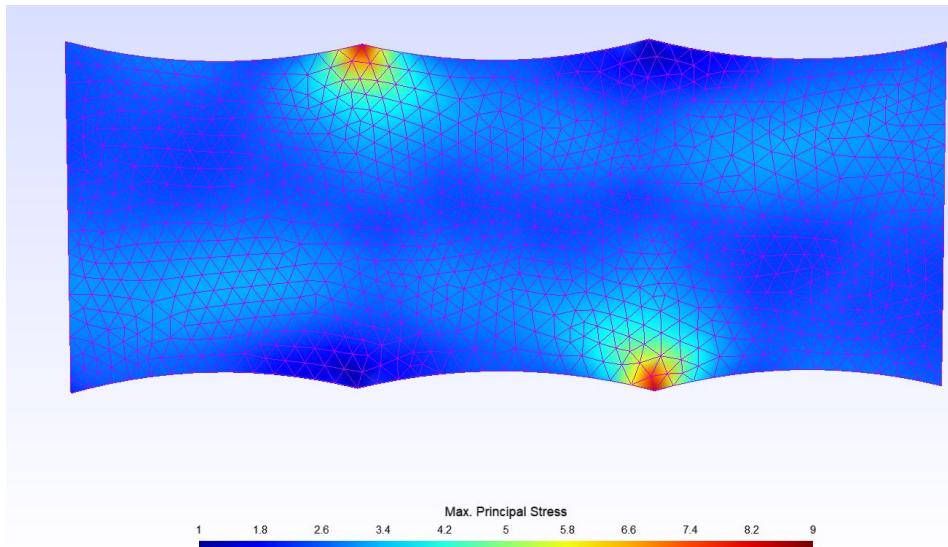


Ideally, all displacement norm should converge to a very small number.

In the event where convergence is not found, the control parameters of Windas should be adjusted before running analysis.

6. View the maximum principal stresses of membrane using gmsh plot. Use **Gmsh | Stressed membrane stress plot** to view.

The figure below shows the maximum principal stress plot of loadcase 2 (rain load).



As the membrane is being pressured downwards by the rain load, the stresses on the high points of hypar will be greater than stresses on the low points. As observed, high stresses will occur across the two high points.

7. Maximum Factored Border Cable Tension can also be obtained. Display using **View | Result | Border cable tension (max)** to show the maximum values in kN.

