Windas 14 Tutorial Manual 2020

August 2020

Table of Contents

1. Introduction to Form Finding

Form finding is a process of finding an equilibrium form of a model under a specific set of internal force and external constraints.

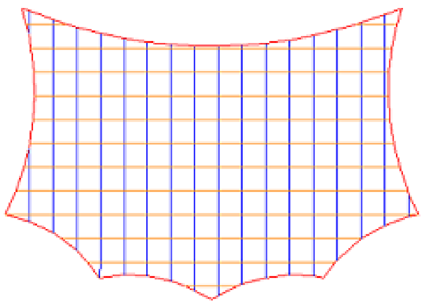
Several methods are available for computer-aided form finding. Some of the most popular methods are the Force Density method, the Dynamic Relaxation, the Surface Stress Density method and the Updated Reference Strategy Method.

1.1. Force Density

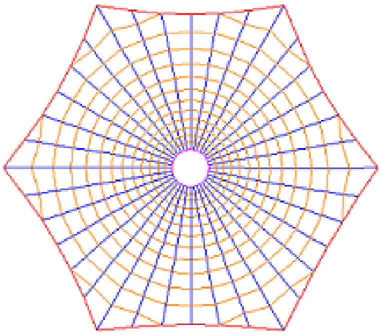
Windas implemented the Force Density Method for form finding. Force Density is defined as the force per unit length. This approach of tensile membrane structure form finding is to determine the force density for each net element that will result in global self-equilibrium. When a tensile membrane is in a state of self-equilibrium, the internal force is in equilibrium too.

1.2. Membrane Forms

Membrane form is either regular (Cartesian) or a radial as shown in the figure below:

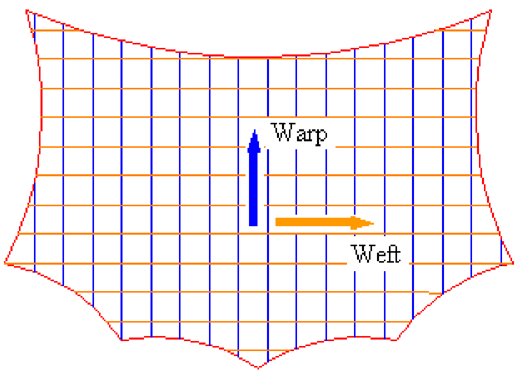


Regular net



Radial net

In the force density method, a membrane form is represented by a system of net. A regular net has its principal axis parallel to the warp and weft directions of the fabric respectively. The warp lines are represented by blue net while the weft lines are represented by orange net respectively.



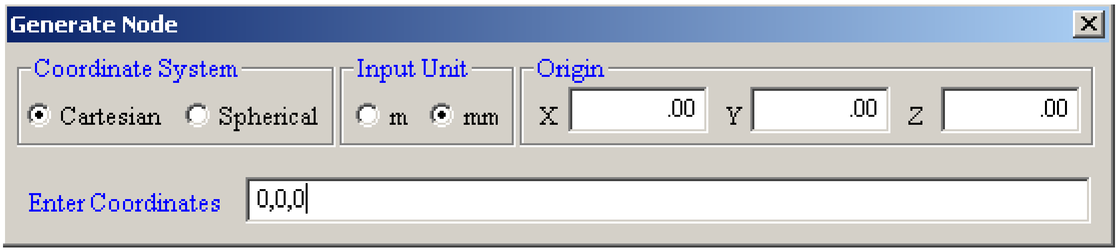
2. A Simple XY Net / Regular Net

This tutorial shows the essential steps of form finding using Windas. Create a simple 10 x 5m flat membrane with 10% sag along the borders. With this model, we are going to perform some editing features in Windas as well.

2.1. System Points

System points are the nodal points used to define the external profile (border) of a membrane. A system point is a reference point on the membrane border and it is used to connect to steel supporting elements. For design and detailing of membrane parts, a system point is used as reference point for the design of masts, tieback cables, clamping plates, etc.

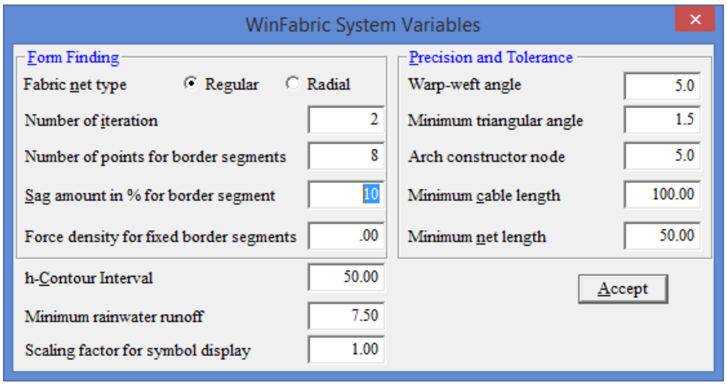
The system points for a simple 10 x 5m membrane are the four corner points. Use the **Generate | Node** command to create the four system points.



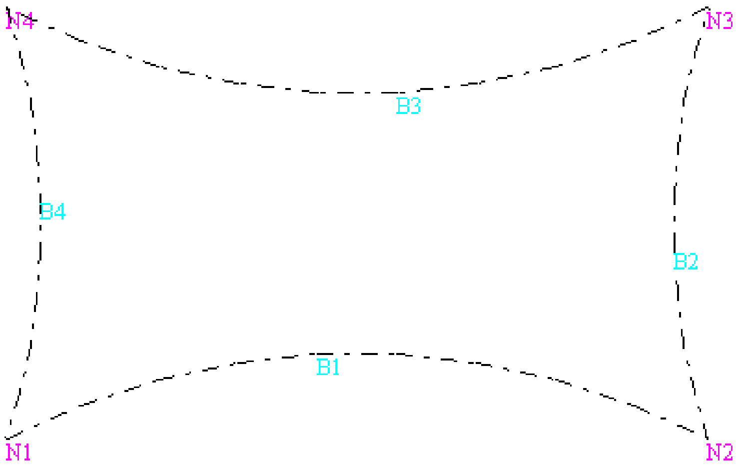
Node ID	X	Y	Z
1	0.0	0.0	0.0
2	10000.0	0.0	0.0
3	10000.0	5000.0	0.0
4	0.0	5000.0	0.0

2.2. External Membrane Border

The external membrane border is formed by connecting the nodal points 1, 2, 3,4 together in anticlockwise order.
The command to create the external border is **Generate | Membrane Constructors | External Border**.
Click **Accept** to accept the default settings as shown below.



The external border defined by segment B1, B2, B3 and B4 is created.



Each external border segment, by default, is defined with 8 points and a curvature of 10% sag toward the membrane centre. The default curvature is towards the membrane centre.

Use the **List | Membrane Constructors | External border** command to get a listing of the border segments' characteristics.

Segment No.	Nodes Start	Nodes End	No. of Point	Segment Fixity	Radius of Curvature Kind	Radius of Curvature Extent	% of Sag
1	1	2	8	0	1	13000.00	10.00
2	2	3	8	0	1	6500.00	10.00
3	3	4	8	0	1	13000.00	10.00
4	4	1	8	0	1	6500.00	10.00

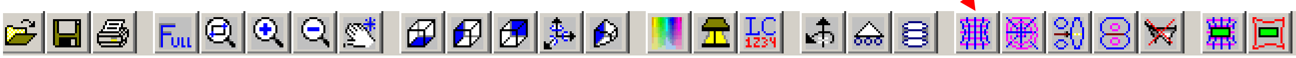
Unrestrained segment

Sag towards membrane centre

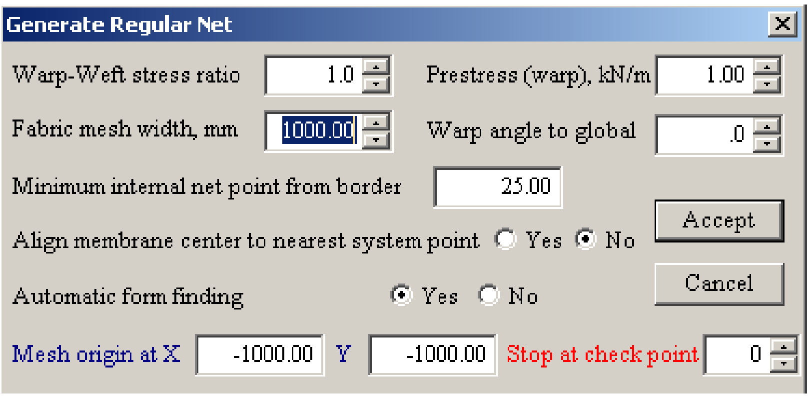
10% of the segement length

2.3. Perform Form Finding

The form-finding command of regular net is **Generate | Membrane Forms | Regular Net**.
As this command is being used often, it is included on the quick menu bar.
Click this icon to select the **Generate | Membrane Forms | Regular Net** command.

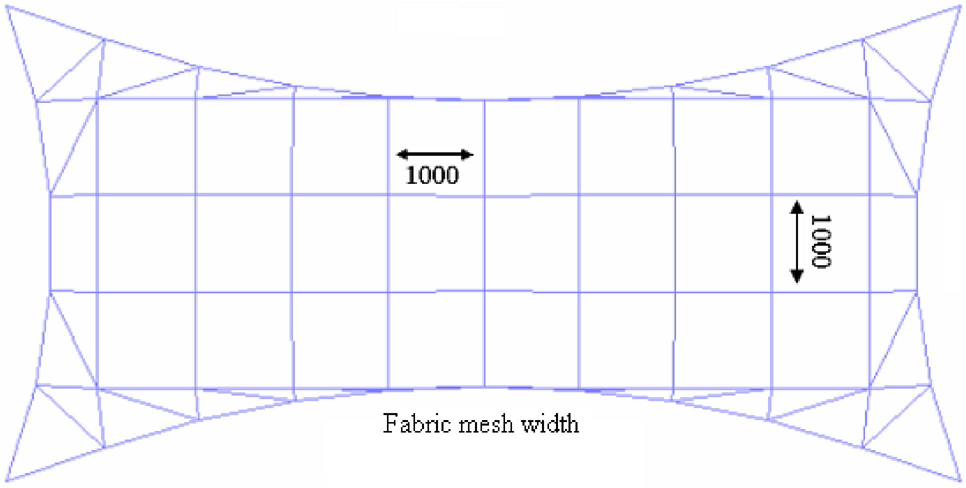


The Generate Regular Net dialog box appears:



One of the primary factors that influence the form finding thus the design of a tensile membrane structure is the Fabric mesh width. Coarse membrane model is formed with larger fabric mesh width while finer model with smaller fabric mesh width. The choice of the fabric mesh width for a given model is a matter of experience. The default fabric mesh width is 1000mm.

Click the **Accept** button to accept default parameters for form finding and your first membrane model is created automatically.



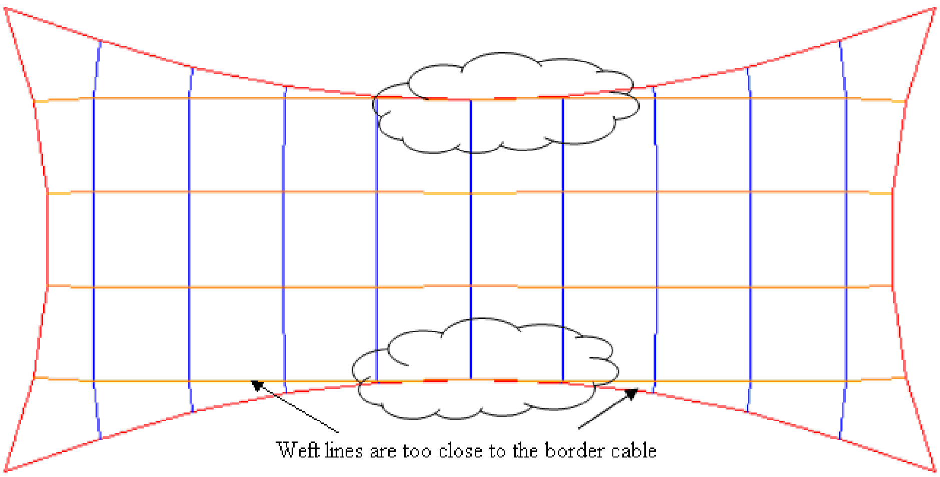
The generated model comprises of membrane surface, warp nets, weft nets and border cables. Save the model using the **File Save** command and name the model as **Simple XY Net**.

2.4. Model Representation and Editable Features

The table below gives the model representation of the membrane net created.

Representation	Color	Color ID	Property ID
Fabric Net (Warp)	Blue	1	1
Fabric Net (Weft)	Cyan	10	2
Border Cable	Red	14	3
Membrane Surface	Light Blue	2	15

To view the net only, turn off the display membrane surface with this command.



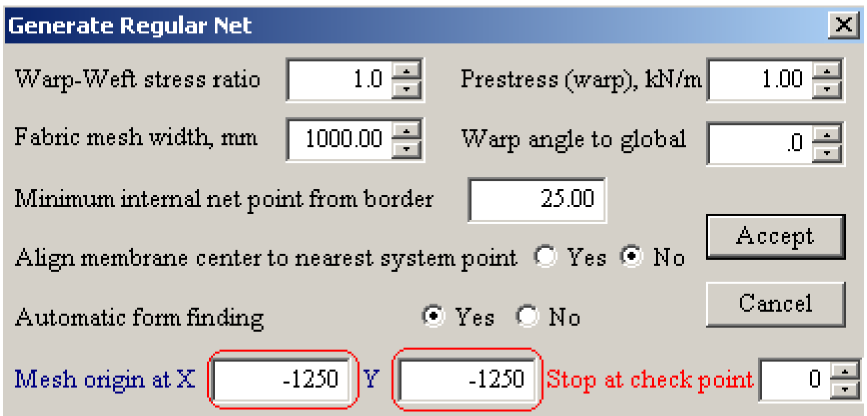
In this example, two of the weft lines are too close to the border cable. There is nothing wrong with the model except the membrane elements are distorted and may affect the accuracy of load analysis. If you are not satisfied with the membrane model created, you may change the model by changing either the Mesh origin, Fabric mesh width or the external border curvature.

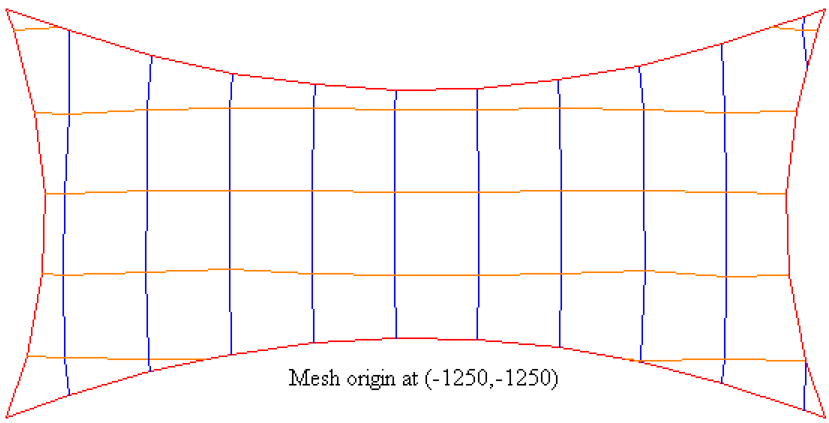
2.4.1. Undo Form Finding

Before conducting any changes, delete the founded form by clicking on the undo form finding toolbar. Then you can try again with different fabric mesh width.

2.4.2. Changing Mesh Origin and Check Point Stop

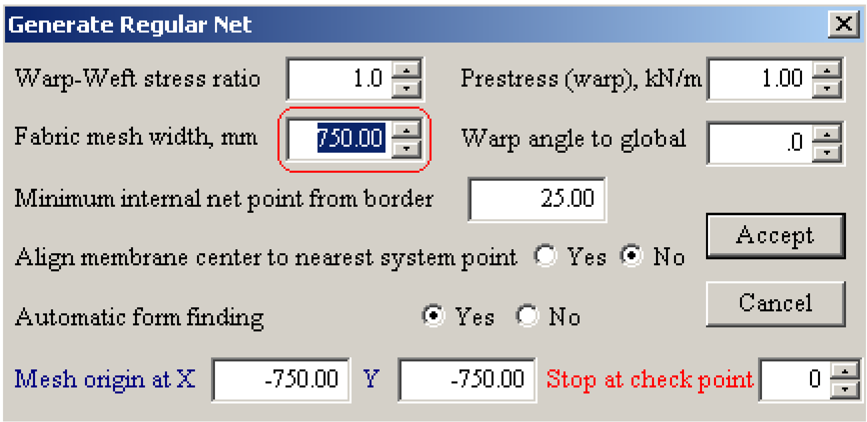
Undo the form finding and start again with the mesh origin at x = -1250.0, y = 1250.0

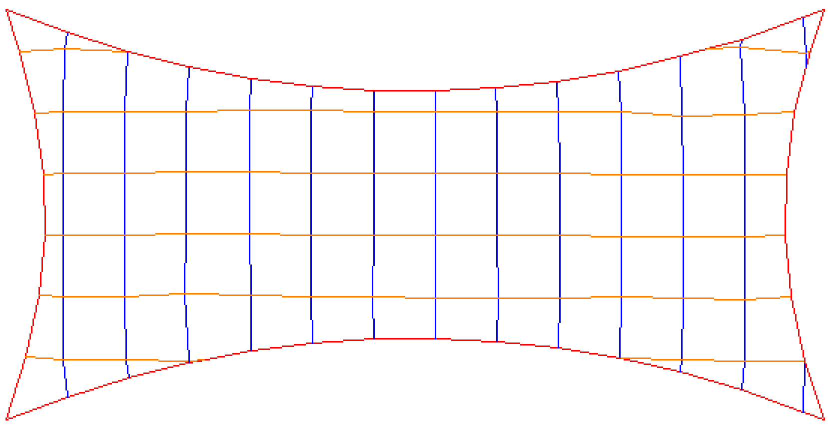




2.4.3. Changing Fabric Mesh Width

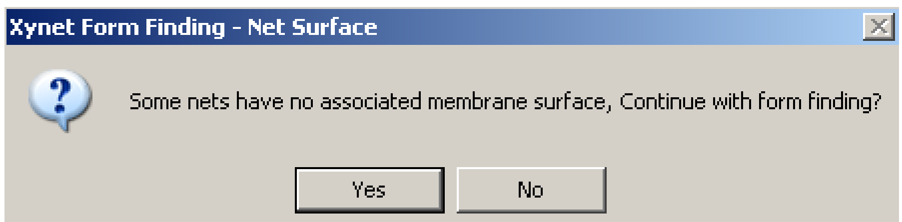
Perform form finding with a fabric mesh width of 750 mm. (Note: always remember to undo the existing form finding)





This model with a fabric mesh width of 750mm is better than the one with a width of 1000mm.

Occasionally, you may receive the following message with a given fabric mesh width.

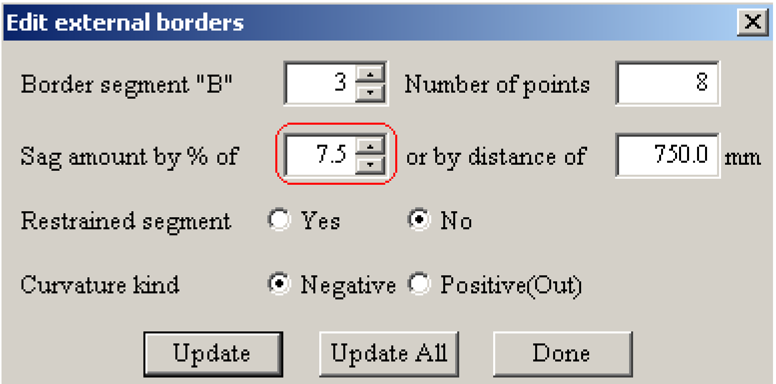


This is due to warp and weft line too close to the border and you may need to manually edit the model if you want to use it for load analysis.

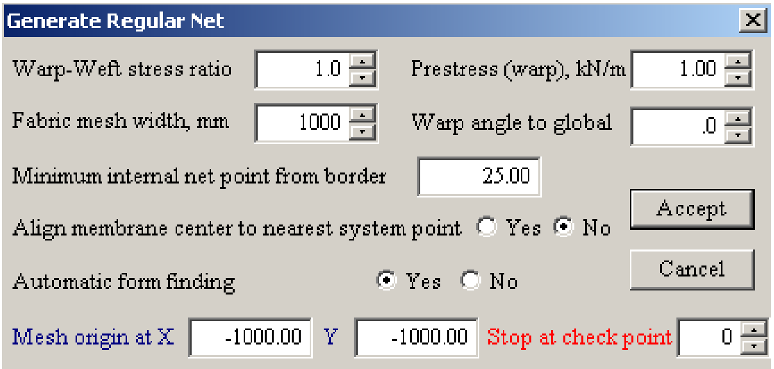
Try fabric width 500mm or other values as pleased.

2.4.4. External Border Curvature

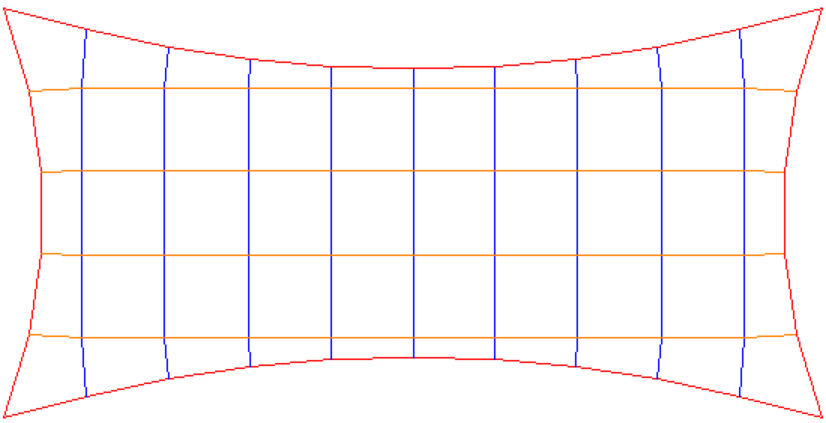
Use the **Edit | Membrane Constructors | External Border | Each** command to modify the border curvature for segment B1 and B3 to 7.5%.
Reducing the sag value also means that the external border will appear to be more straight than curvy.



Perform form finding with the same parameters below,



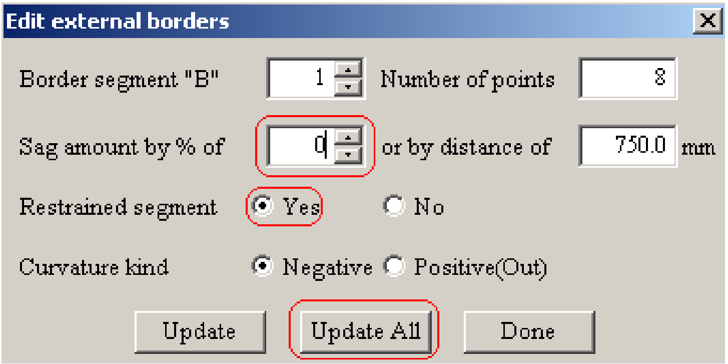
As it is shown below, the warp and the weft lines are now further away from the border cables as the border cables tend to straighten.



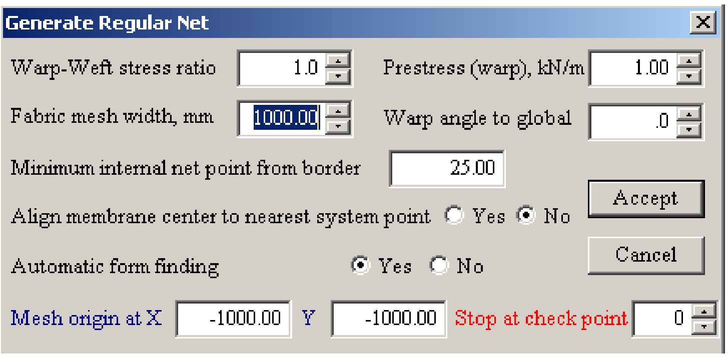
2.4.5. Straight External Border

To create straight border, use the **Edit | Membrane Constructors | External Border | Each** command to change the sag amount by % of to zero and set the condition of restrained to **Yes**. Click on the **Update All** button to edit to all four border segments.

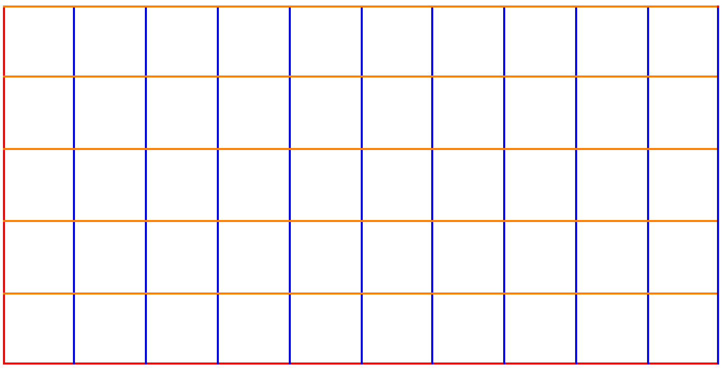
(Note : You must restrain the border segment in order to get a straight border)



Perform form finding of a regular net with the **Generate | Membrane Form | Regular Net** command. Use the form finding parameters as shown in the dialog box.



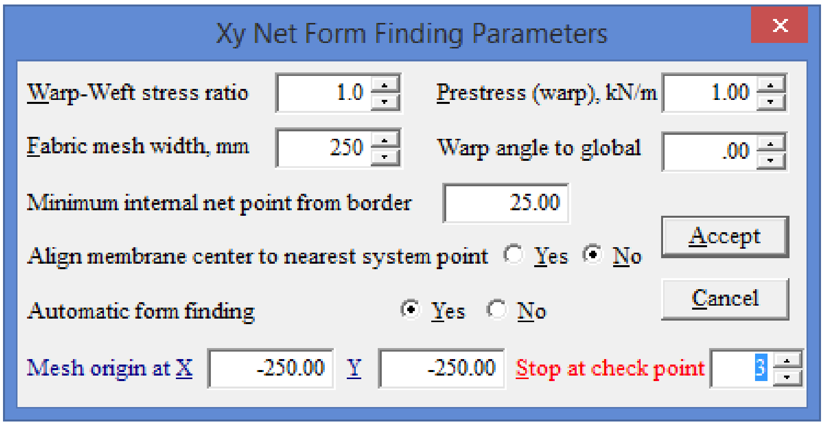
A rectangular membrane model is created!



The surface area of the membrane can be check by using the **WinFabric | Fabric Surface | Area** command. Check that the surface area of the 10 x 5m membrane is indeed 50m^2.

2.4.6. Form Finding Checkpoint

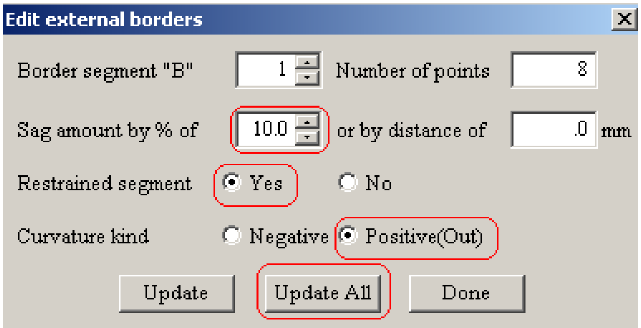
In this command window we can observe the "stop at check point" column. With this facility you may select to stop the form finding process at various check point. These check points are useful for diagnostic purpose. It allows the form finding to stop at different level thus allows manual modification of the mesh before form finding.



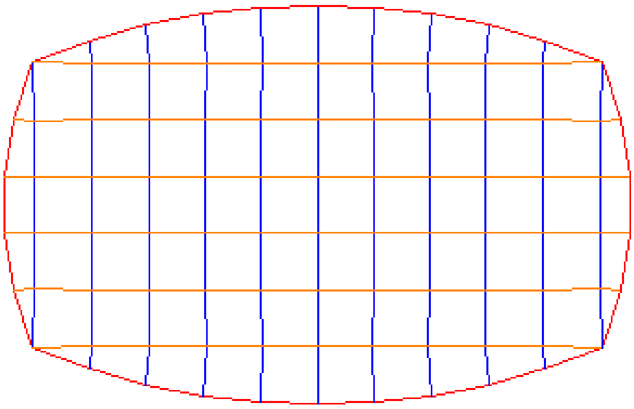
To re-start form finding at stop at check point, use the **WinFabric | Force Density Form Finding** command.

2.4.7. Positive Border Curve

An external border segment is said to have positive curvature when it is bulging out. To test out this function, edit all the border segments to 10% sag and set curvature kind to positive (out) by clicking **Edit | Membrane Constructors | External Border | Each** then input the data as shown below.
Note : border segment with positive curvature must be restrained.



Perform form finding with any parameters.



There is a better way of modeling positive curvature using membrane constructor given in Chapter Four.

2.4.8. Warp Angle to Global

By default, the warp line is parallel to the global x-axis.
There are occasions when the desired warp lines are at an angle to the x-axis.
Use warp angle to global or use the **Modify | Rotate | Model** command to accomplish the task.

3. A Simple Radial Membrane Net

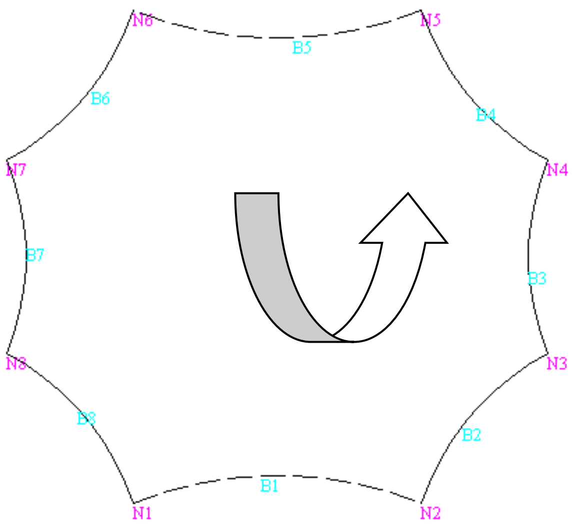
A radial net has its origin inside the membrane and radial outwards to the border. The most commonly seen radial membrane is a tent with a high point ring.

3.1. System Point

The system points defining the eight corners of a radial net with hexagonal base are:

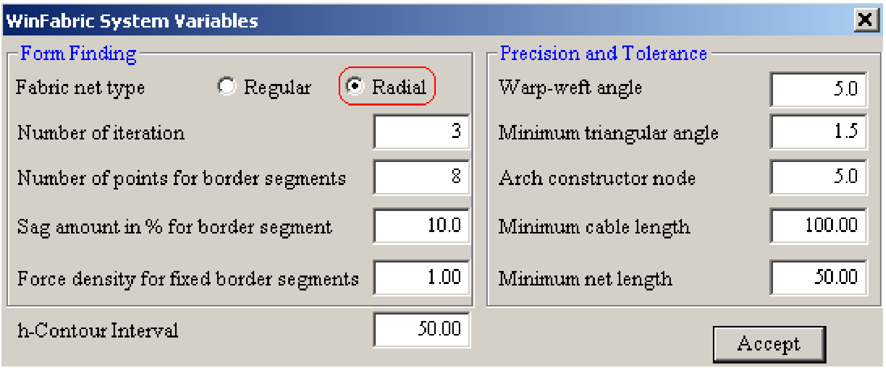
Node ID	X	Y	Z
1	-5000	-8660	0
2	5000	-8660	0
3	-9396	-3420	0
4	9396	-3420	0
5	-9396	3420	0
6	9396	3420	0
7	5000	8660	0
8	-5000	8660	0

Use the **Facility | Reorder | Node | Anticlockwise** command to reorder the node sequence.



3.2. External Membrane Border

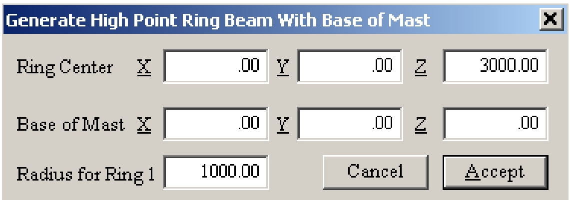
Specify the external border with the **Generate | Membrane Constructors | External Borders** command. Set the fabric net type to radial.

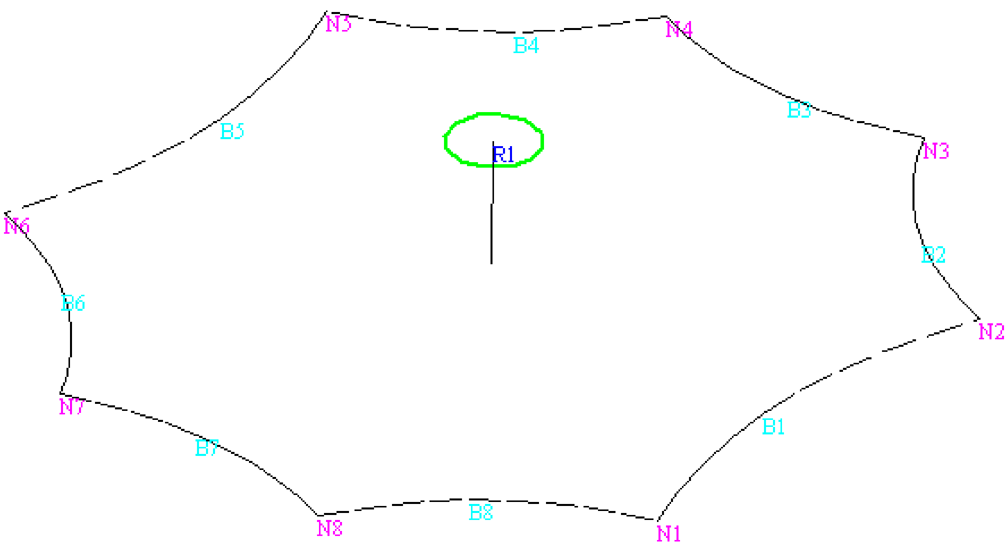


3.3. High Point Ring

High point ring is one of the most commonly used membrane constructor in tensile membrane structures.

Use the **Generate | Membrane Constructors | Circular Ring** command to create a circular ring of radius 1000mm at the centre of the membrane and is 3000mm above ground.



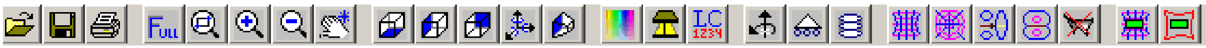


Membrane Constructors with high point ring and a vertical mast in NW View

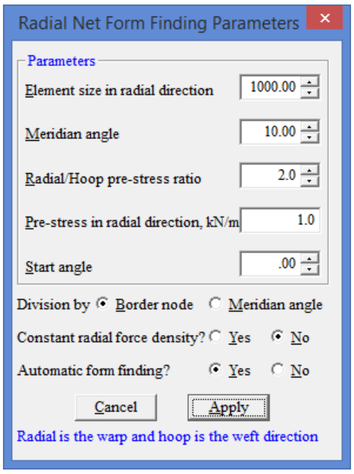
Save the model as **A Simple Radial Net** with the **File | Save as** command.

3.4. Form Finding

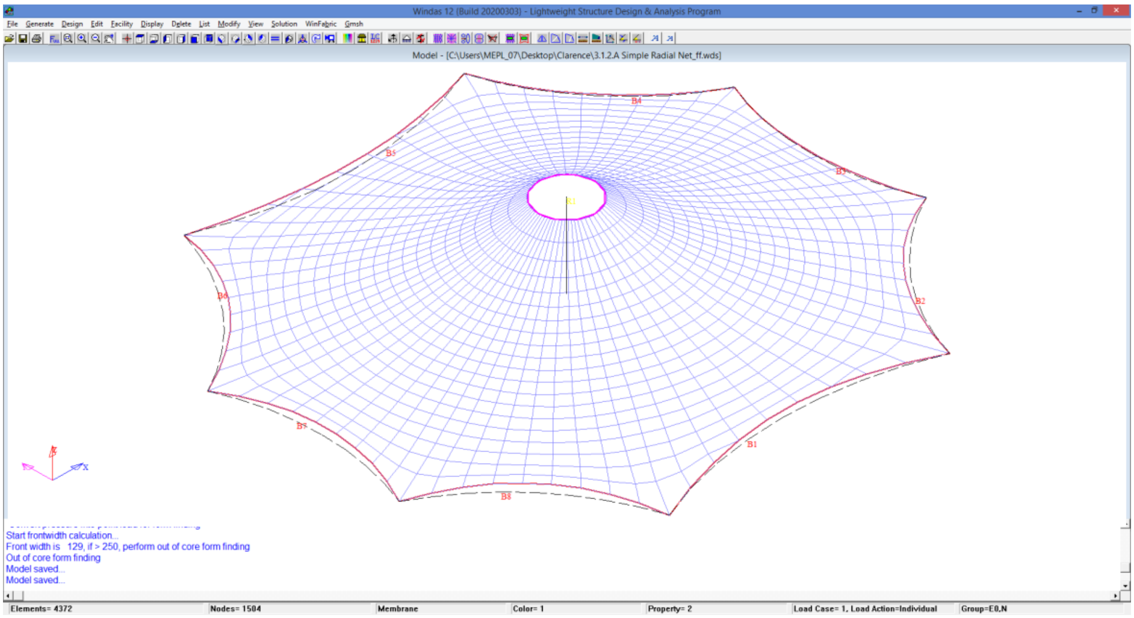
Perform radial net form finding with the **Generate | Membrane Form | Radial Net** command or click on the quick access button.



The radial net form finding parameters are as shown in the dialog box.



The default division of the radial net is from the ring centre to the **Border Node** whereas the default number of points dissection the border segments is 8. Therefore, we will get eight radial seam line from the ring towards each border segments.



Radial Membrane After Form Finding in Perspective View

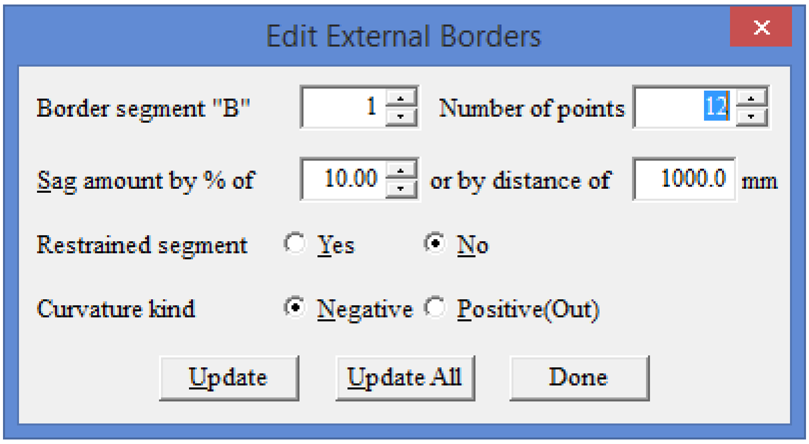
To change the number of points on border segment, **Edit | Membrane Constructors | External Border | Each.**

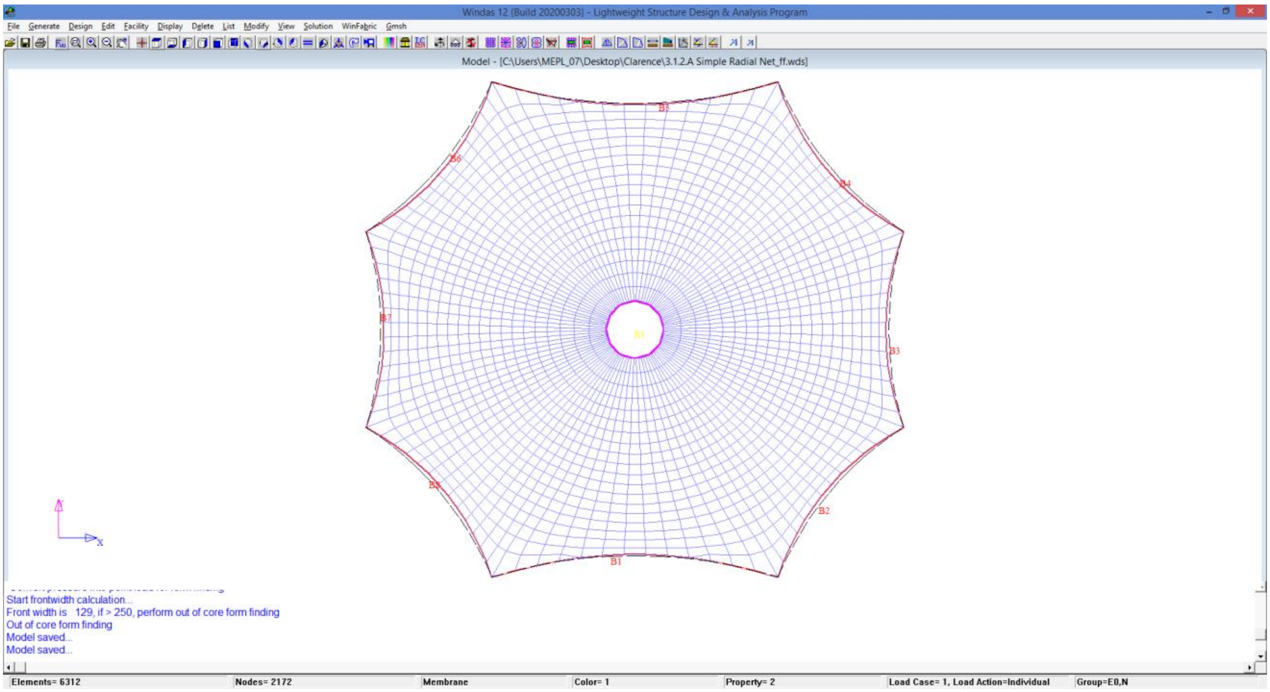
3.5. Radial Net Division Method

There are two methods of division of radial line (waft line) used for form finding of a radial net, Border Node Method and Meridian Angle Method.

3.5.1. Border Node Method

Use **Edit | Membrane Constructors | External Border | Each** command to change the number of point representing the external segment to 12. Click Update All when the number of points has been set.



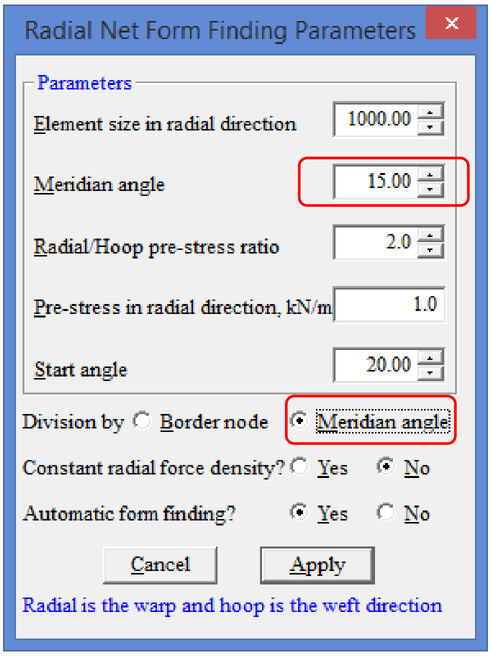


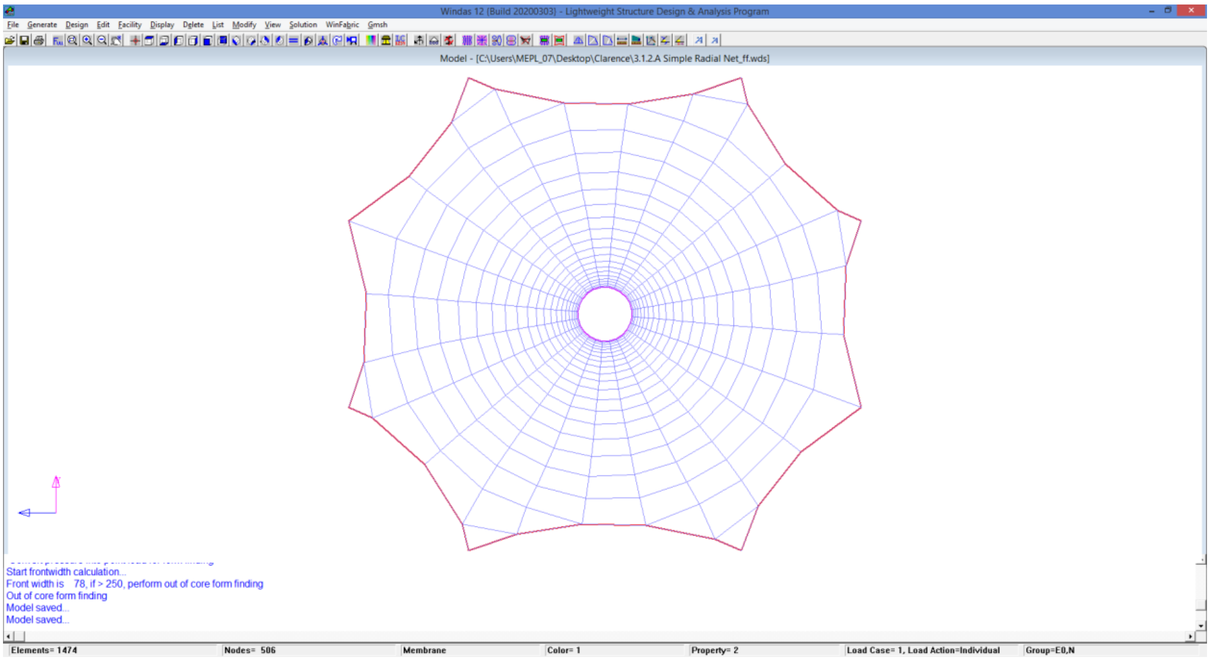
Now there are 12 radial (warp) lines for each border segment in the radial net.

3.5.2. Meridian Angle Method

The division of radial (warp) lines is by the meridian angle. For a meridian angle of 15°, there will be a total of 24 radial (warp) lines on the radial net. Now instead of editing the border node, we can directly edit from the form finding command box. Note that we need to tick on the Meridian Angle instead of border node for this command to work.

Note : always remember to do the undo form-finding before conducting any new form-finding.

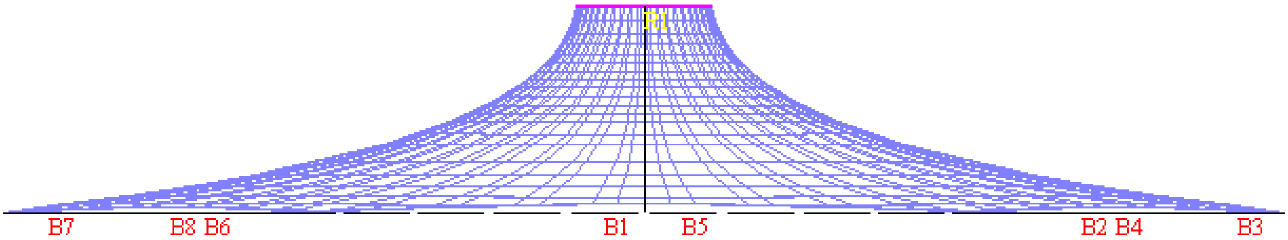




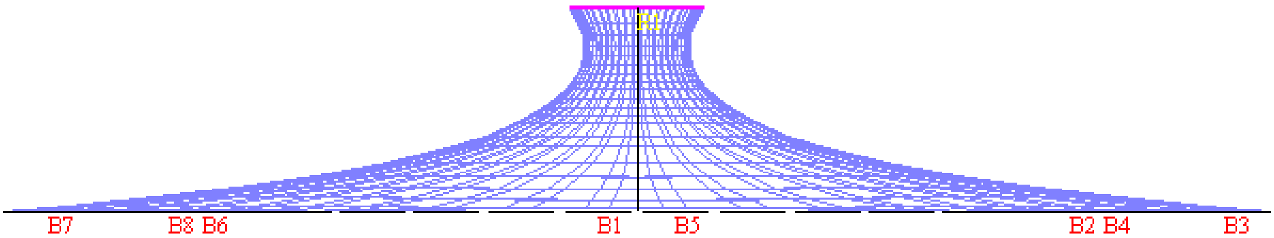
Now there are 24 radial (warp) lines in the whole radial net segment.

3.6. Radial Hoop Pre-Stress Ratio

Radial to hoop pre-stress ratio is the ratio of the pre-stress along the Radial (warp) to the hoop (weft) direction. The default value is 2.0.
Undo form-finding and try with a new radial to hoop ratio of 0.5 and 1.0 respectively.



Radial/Hoop Prestress Ratio = 1.0

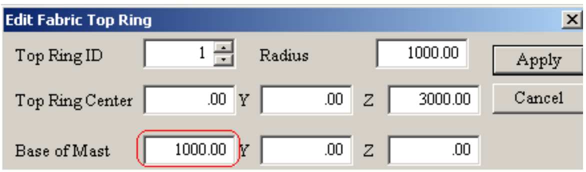


Radial/Hoop Prestress Ratio = 0.5

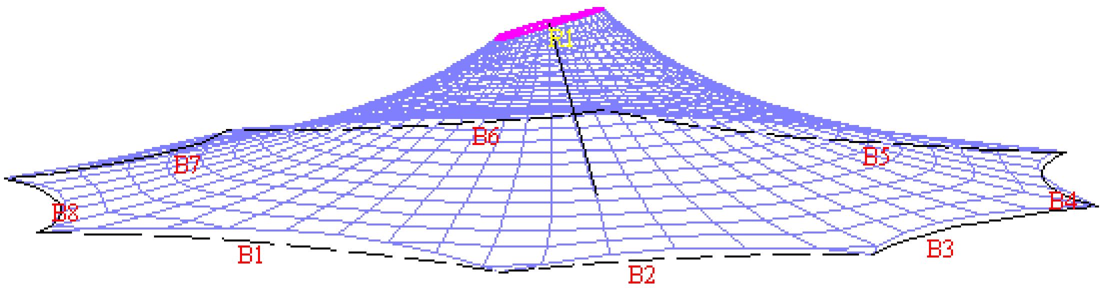
The shape of a radial model is determined by then radial to hoop ratio.

3.7. Tilted Mast

Edit the high point ring using the **Edit | Membrane Constructor | Circular Ring** command. Change the x-axis of base of the mast to 1000.

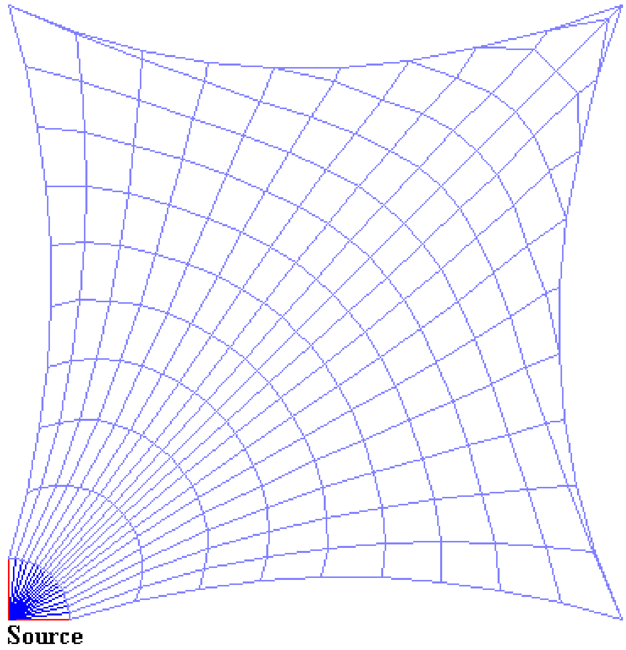


And repeat the form finding.



4. Radiating Net

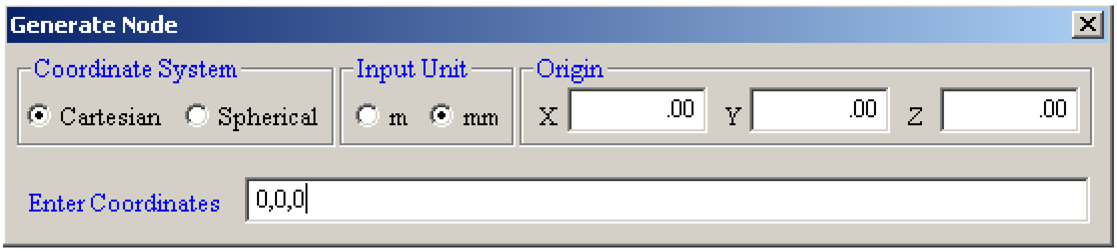
A special form which is a hybrid of regular and radial form is known as radiating or source net. The warp lines are radiating out from Node ID 1.



Radiating form is commonly used in area where a complete radial tent is not possible. For example, a cantilever canopy attached to a wall with a 90° turn.

4.1. System Points

Use the **Generate | Node** command to create the four system points.

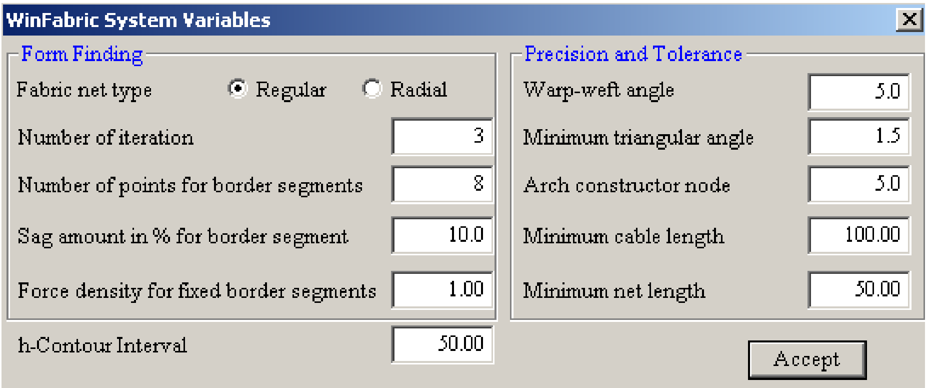


Node ID	X	Y	Z
1	0.0	0.0	0.0
2	10000.0	0.0	0.0
3	10000.0	10000.0	0.0
4	0.0	10000.0	0.0

4.2. External Membrane Border

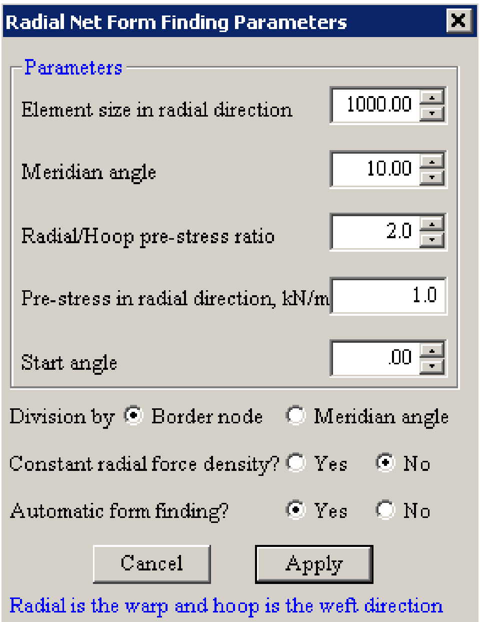
The external is defined by nodal point 1, 2, 3 and 4. The command to create the external border is **Generate | Membrane Constructors | External Border | All | Accept**.

Select **Regular** fabric net type. **Accept**. External membrane border defined by segment B1,B2, B3 and B4 will then be formed. Save the model as **Radiating Net**.



4.3. Form Finding

Use the **Generate | Membrane Forms | Radiating Net** command to perform form finding.

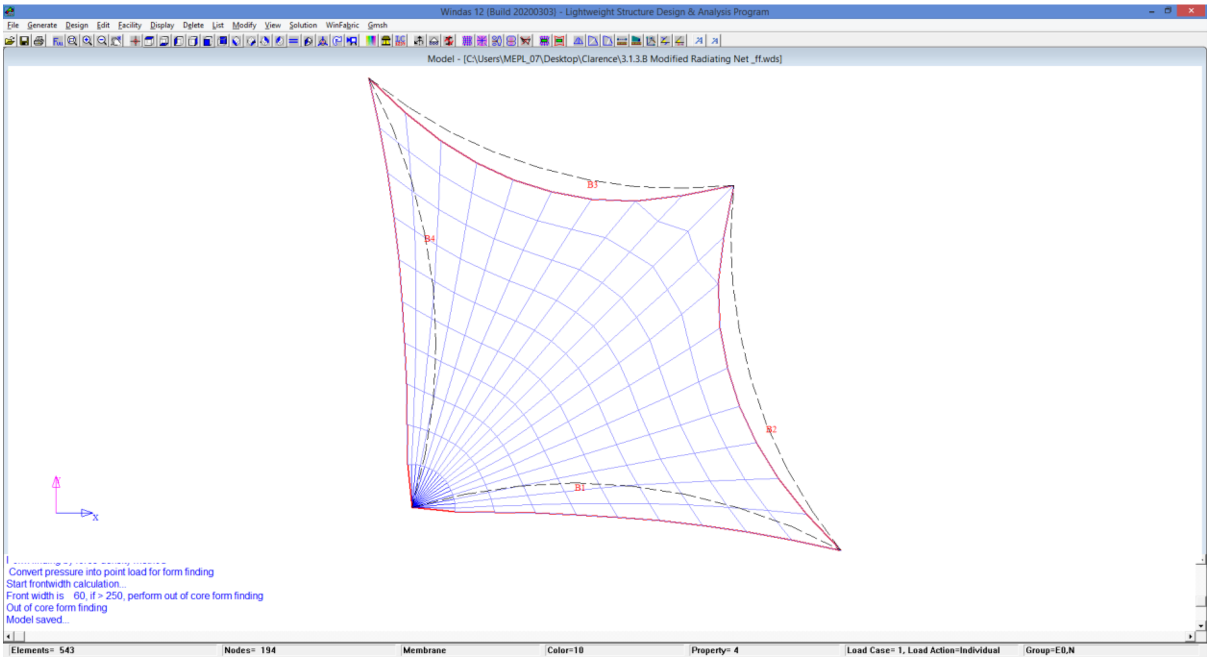


Click the **Apply** button to accept the default form finding parameters. The radiating net will then be formed.
Radiating net is used for regular membrane form with radiating seam lines.

Radiating net is produced using radial net form finding parameter with regular net type.

4.4. Modified Form

Modify the simple radiating net example into the following figure.



Edit the system points by using **Edit | Nodal Coordinates** update according to this table below.

Node ID	X	Y	Z
1	0.0	0.0	2000
2	10000	-1000	0.0
3	7500	7500	0.0
4	-1000	10000	0.0

Follow the same steps to do external membrane border and form finding. The particular modified form will be generated.

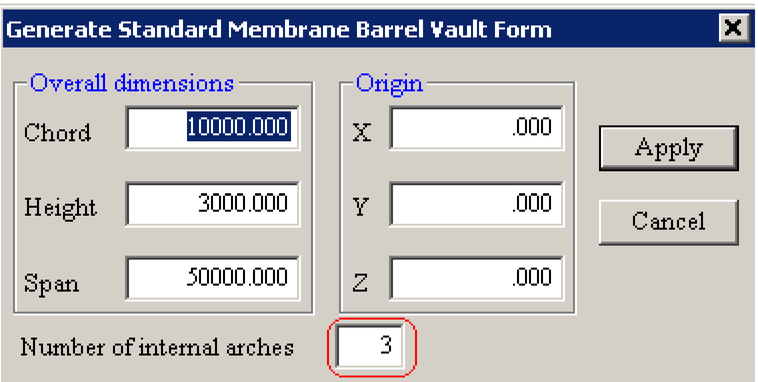
5. Barrel Vault Forms

A barrel vault form membrane structure is a membrane supported by circular arches. This form is commonly used to cover a long sheltered walkway. The edge and the end of the membrane may be either straight or curved.

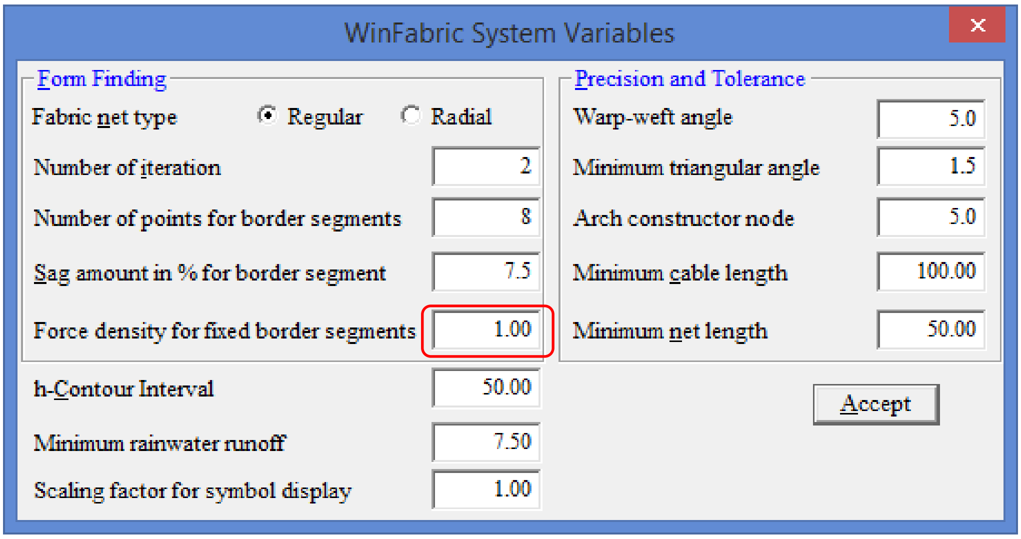
5.1. Standard Barrel Vault Form

The command for automatic generation of standard barrel vault form is **Generate | Membrane Forms | Barrel | Accept.**

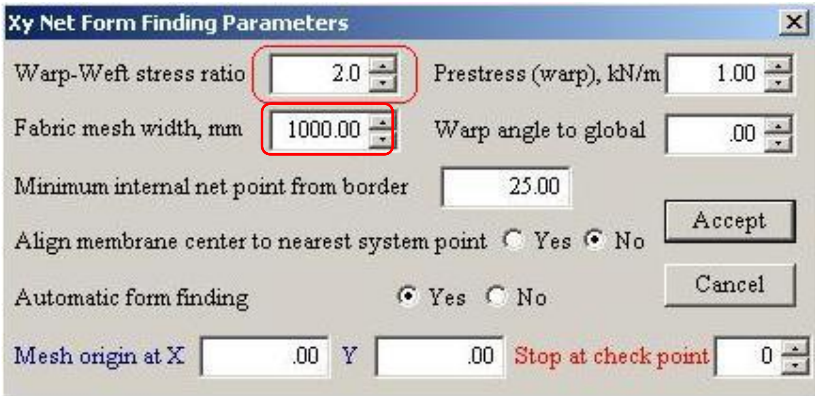
The following example demonstrates form finding of a barrel vault form automatically. The barrel vault has a chord length of 10m and span of 50m. It is supported internally by three circular arches.



Set the net type to **regular** for barrel vault form.

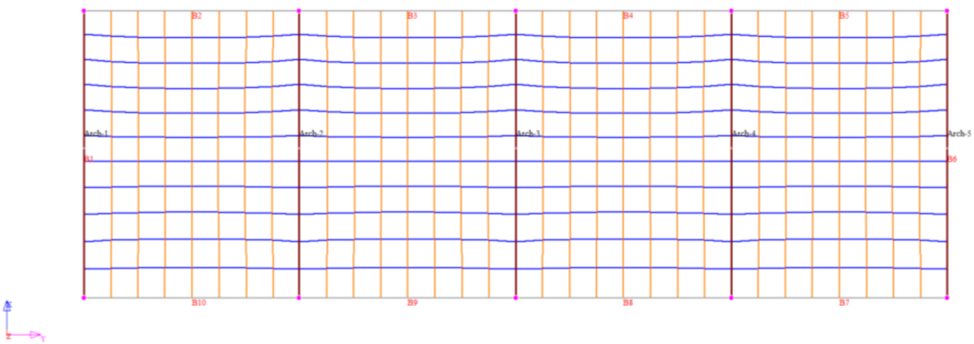


Like the stress ratio in a pressure vessel, the warp-weft stress ratio of barrel vault should be at least 2.0.

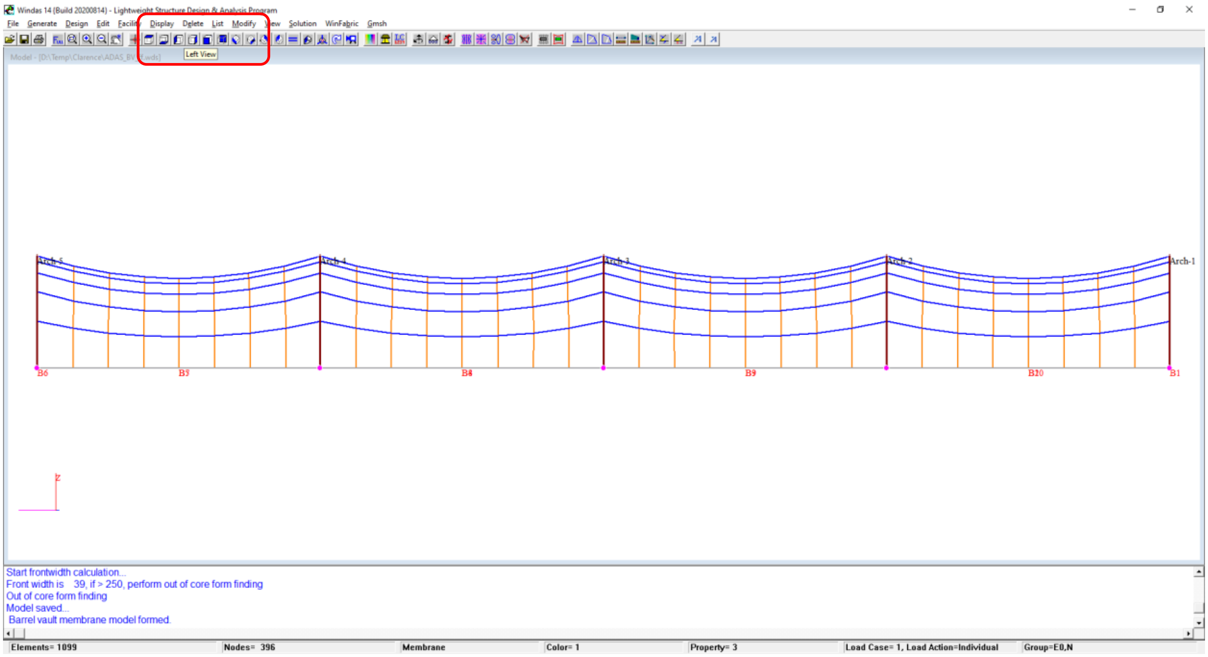


A standard barrel vault is formed. The model is saved automatically as **ADAS_BV.**

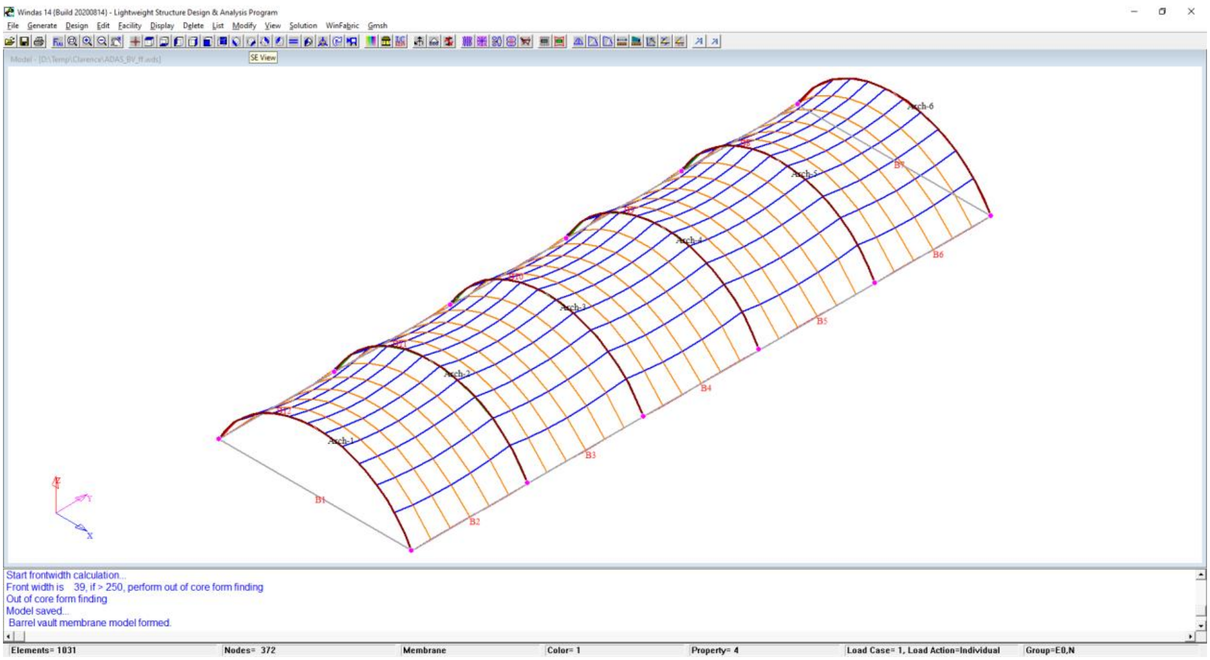
The plan view is as shown below,



Click **left view** to get this curved view.



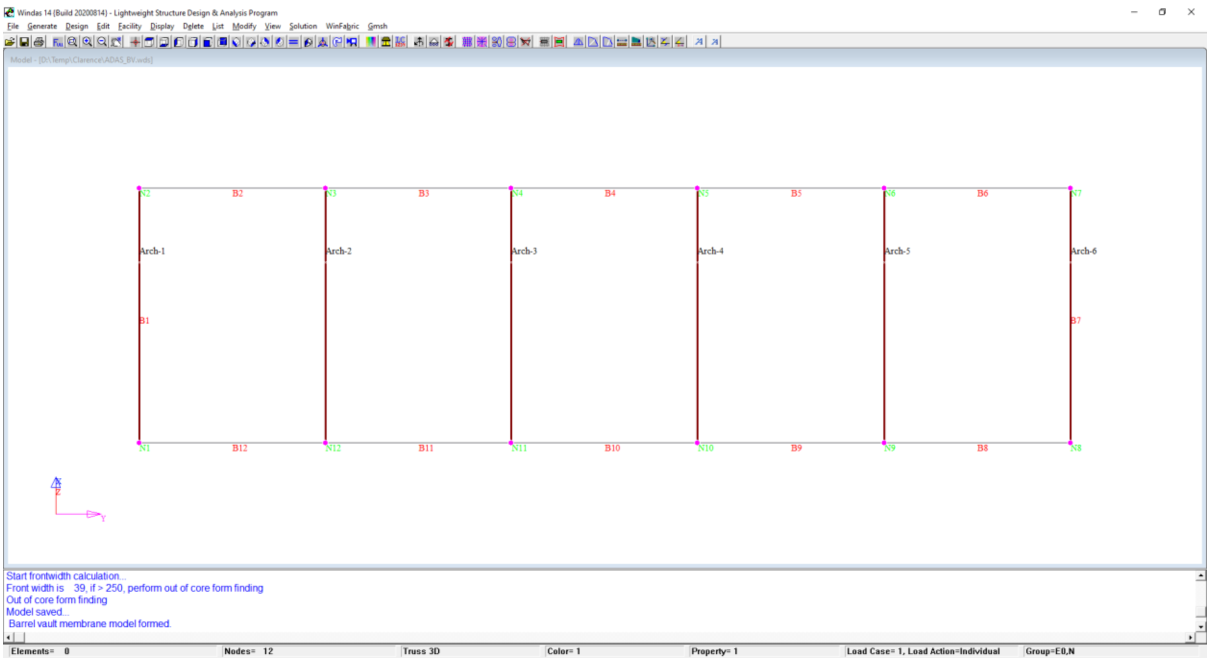
Click **SE View** to get this view. We can see that the side edges are straight whereas at both ends of the barrel vault are curved edges.



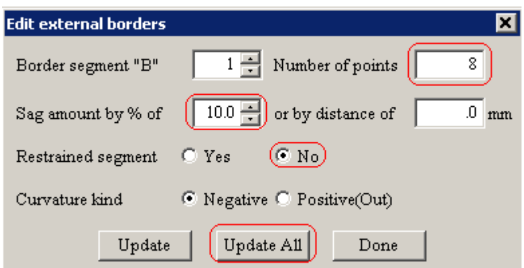
5.2. Standard Barrel Vault with Free Border

For creating all standard membrane forms Windas have automatic system points, external border and membrane constructors.

From previous model created, **undo form finding**. Click the **undo form-finding command** to return the model to state with system points, external border, etc. We can see that all the external borders are still straight, as shown below.

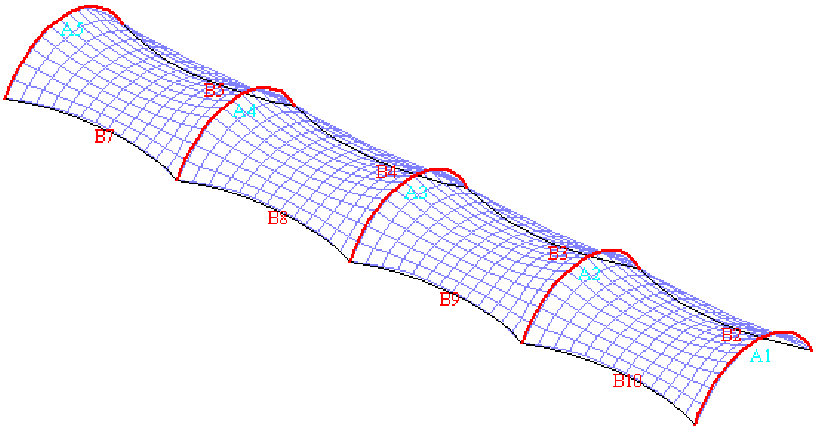


Use the **Edit | Membrane Constructors | External Border | Each** command to modify the external border to unrestrained border.



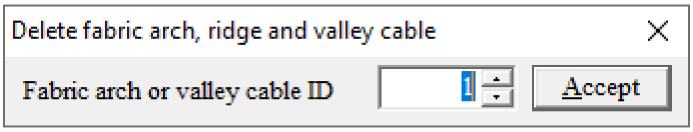
Save the model as **BV**. Perform form finding using the **Generate | Membrane Forms | Arch & Border** command.

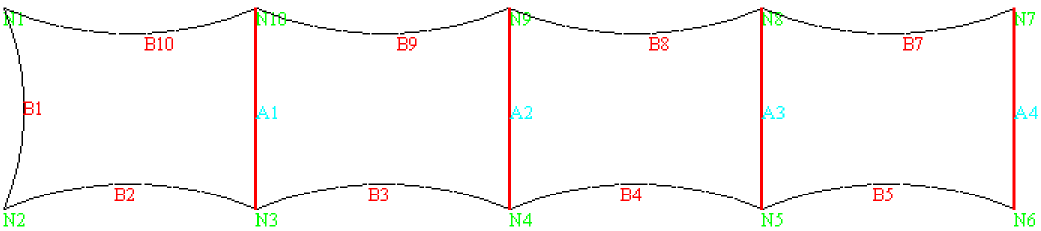
The following structure will then be formed.



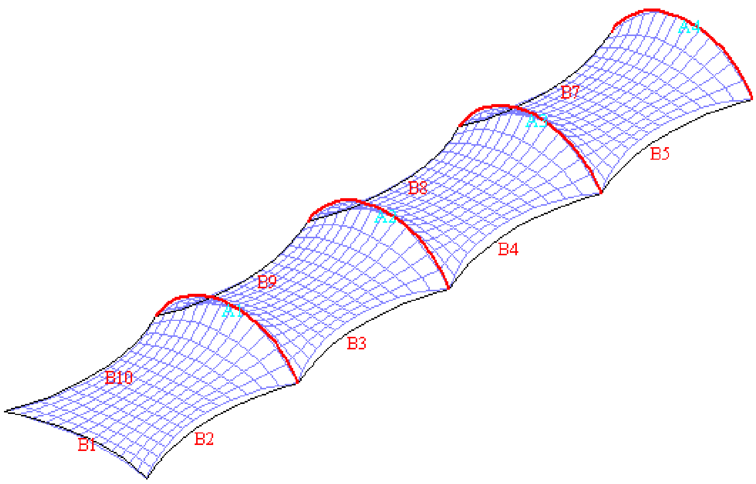
Now undo form finding of the **BV** model again. Save the model as **BV Free End**.

Delete external arch ID 1 using **Delete | Membrane Constructors | Arch, Ridge & Valley Cables** command.

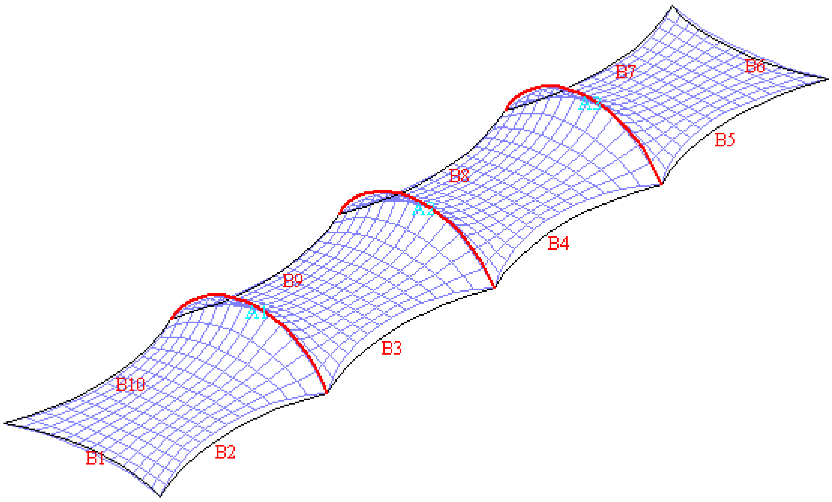




Perform form finding using the **Generate | Membrane Forms | Barrel | Regular membrane**.



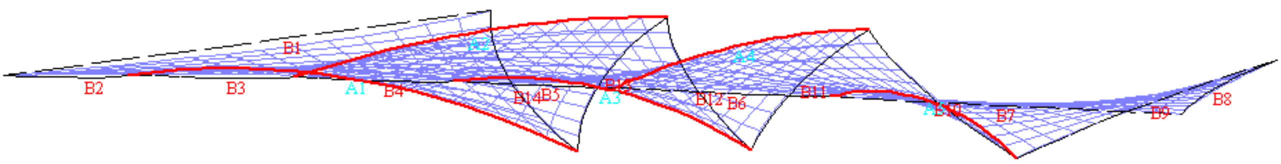
Similarly, delete the last arch and perform form finding gives the following model. Use the command **Delete | Membrane Constructors | Arch, Ridge & Valley Cables** and choose that particular arch. If the arch does not delete properly, choose **Delete | Element |** choose the remaining arc element.



5.3. Modified Barrel Vault With Free Border

This section presents a non-standard **barrel vault with free border** command. The model comprises 14 system points and 5 internal arches. This section will illustrate the procedure of creating this non-standard barrel form from scratch.

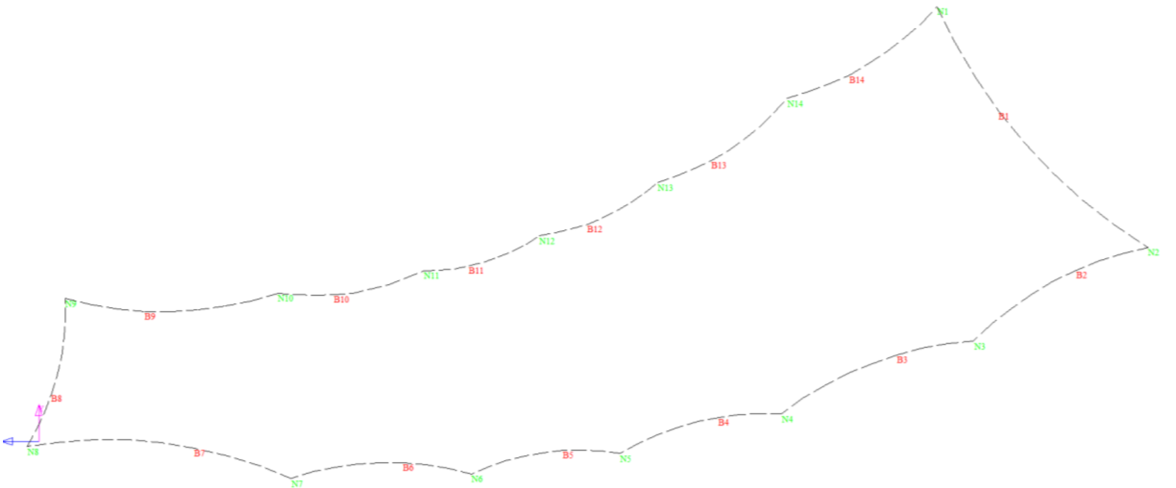
The arches are arranged in such a way that water will flow from the high point to the low point. This type of structure is commonly used as front entrance canopy or along a corridor.



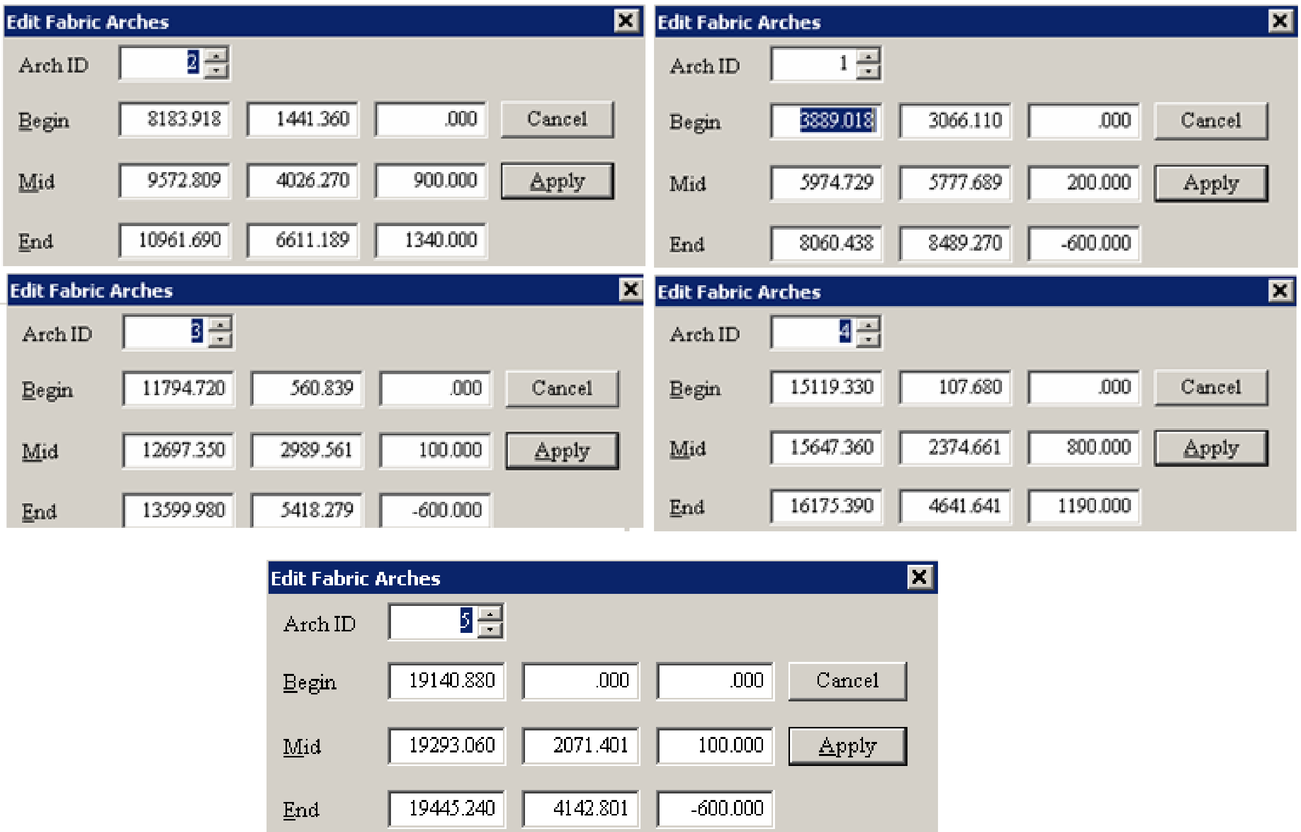
Create the system points using **Generate | Node (more) | Cartesian.** Input these values in the command box accordingly.

Node ID	X	Y	Z
1	4709.50	10538.84	1490.00
2	0.00	5161.65	0.00
3	3889.02	3066.11	0.00
4	8183.92	1441.36	0.00
5	11794.71	560.84	0.00
6	15119.33	107.68	0.00
7	19140.88	0.00	0.00
8	25036.36	709.52	0.00
9	24188.86	4015.95	990.00
10	19445.24	4142.80	-600.00
11	16175.39	4641.65	1190.00
12	13599.98	5418.28	-600.00
13	10961.69	6611.19	1340.00
14	8060.44	8489.27	-600.00

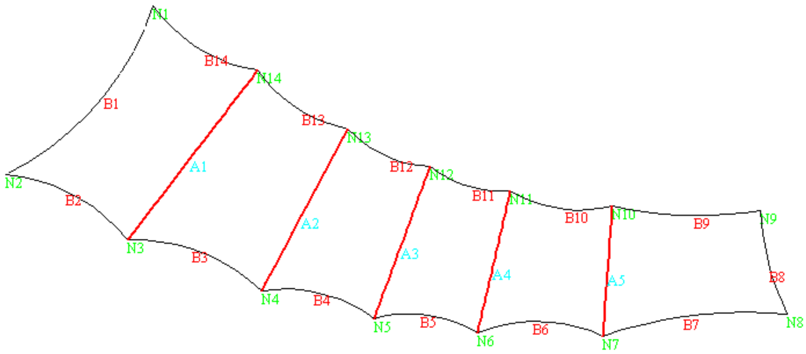
The external membrane border is defined by nodal point 1 to 14 in anticlockwise order. The command uses to create the external membrane border is **Generate | Membrane Constructors | External Border | All | Accept.**



Use the **Generate | Membrane Constructors | Arches | Regular | 3 points** command to create the arches. The coordinates of the five arches are as shown below :



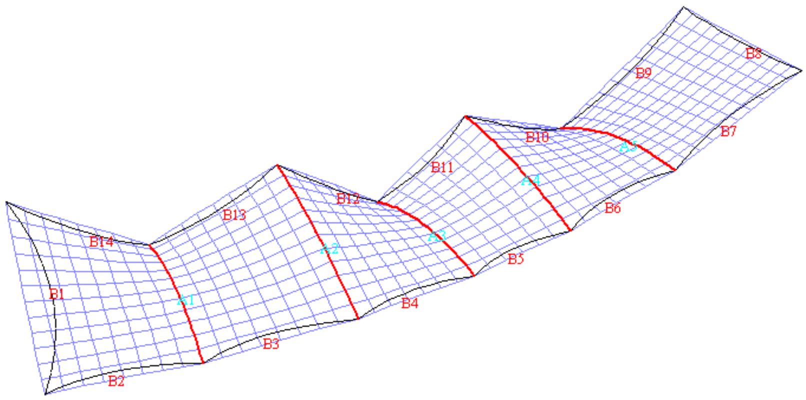
These five arches form a wave form along the membrane border.



Note : the end points of the arches are the system points.

Perform form finding using the **generate regular net** command in the quick access tool box. Otherwise we can find the command in **Generate | Membrane Forms | Regular Net.**

The fabric mesh width is 500mm.

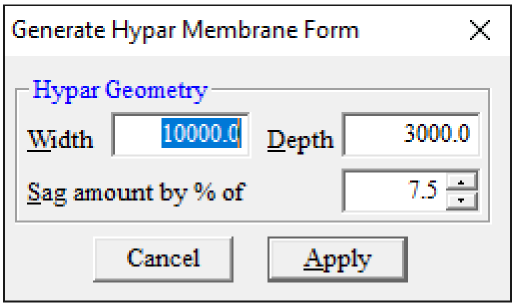


6. Hypar Forms

The expression Hypar is derived from **hy**perbolic **pa**raboloid form. This form is commonly used to refer to saddle shaped surfaces even for surfaces that are not pure hyperbolic paraboloids. It consists of two high points and two low points forming a saddle shape.

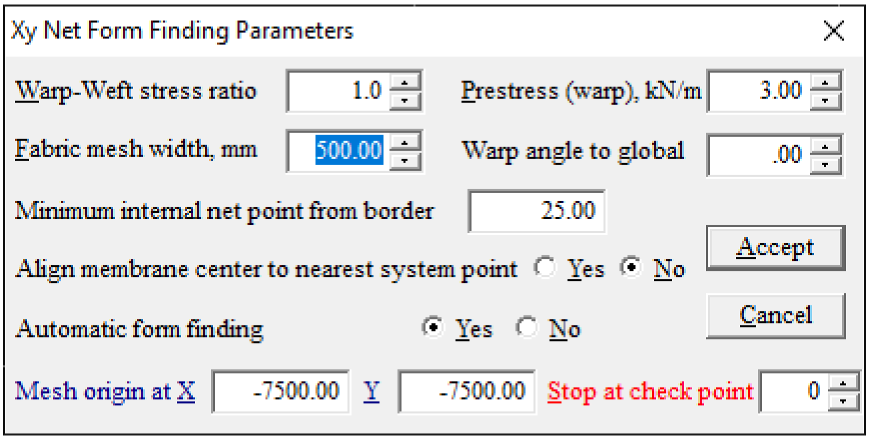
6.1. A simple hypar

The command for automatic hypar membrane form is **Generate | Membrane Forms | Hypar**. Select the command and the hypar dimensional dialog appears.

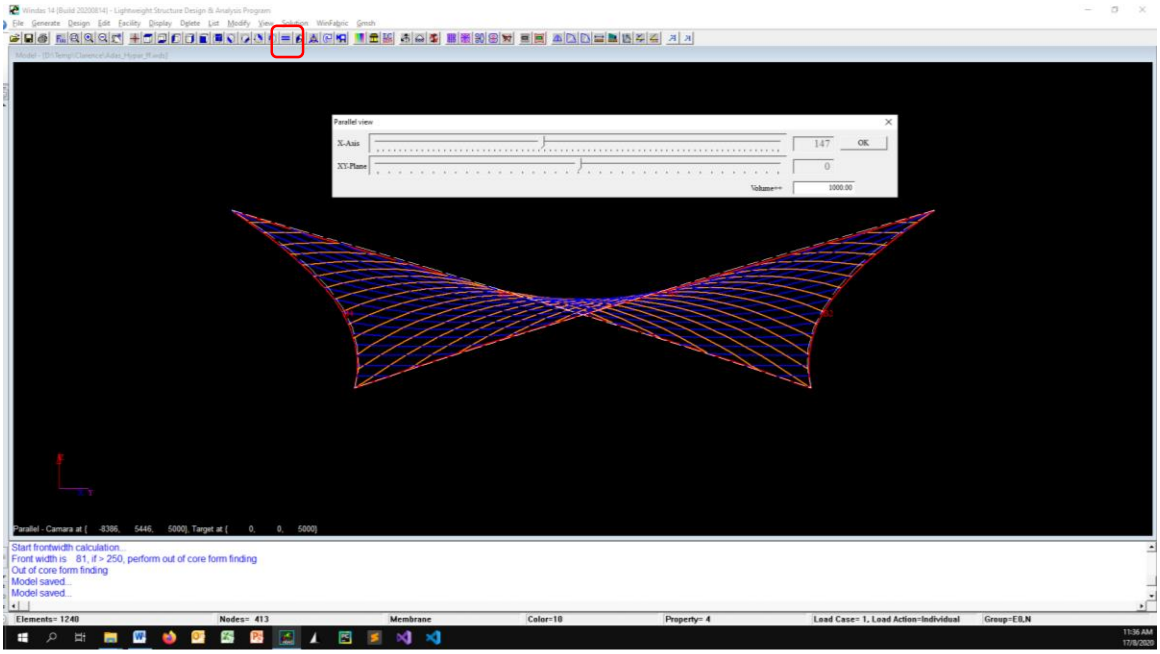


Click **Accept** and a hypar of 10m width and 3m depth with a border curvature of 7.5% sag will be generated.

The XY-net form finding dialog will appear after **Accept** button. Change the fabric width of 500mm and the pre-stress in the warp direction to 3.0kN/m. Click **Accept** to form find a hypar membrane.



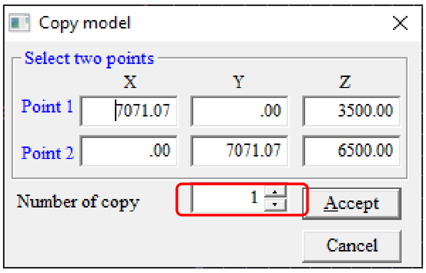
Use the **Parallel View** command in the quick toolbox to observe the hypar from any angle.

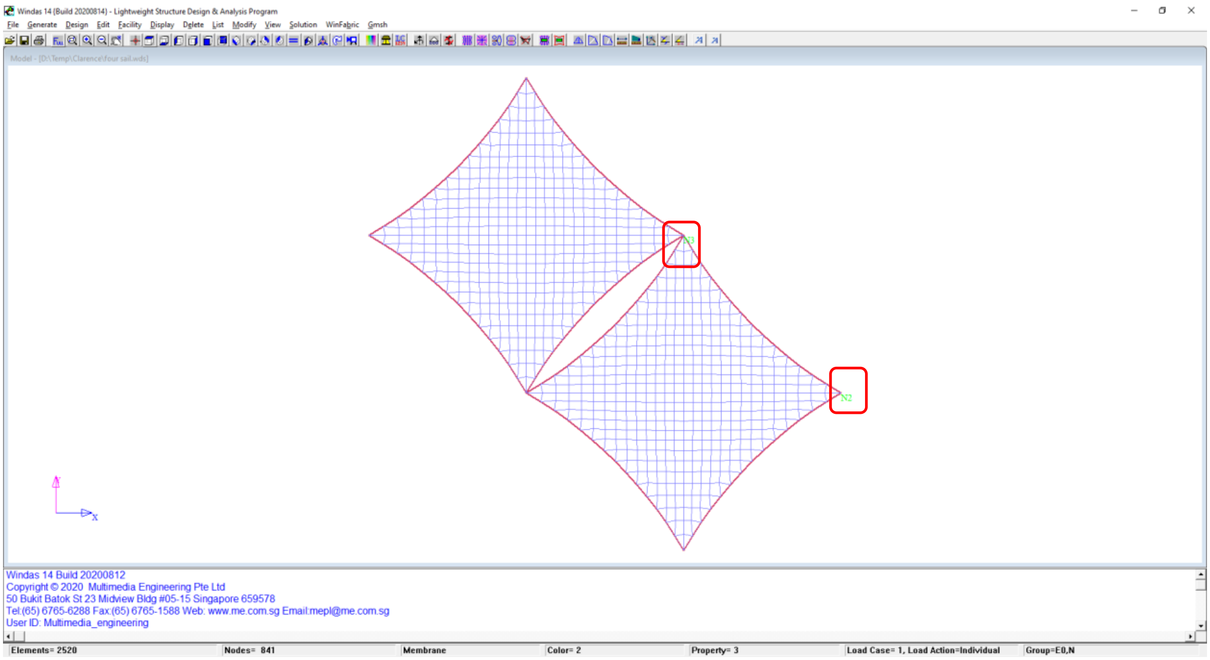


6.2. Four Hypar Sail

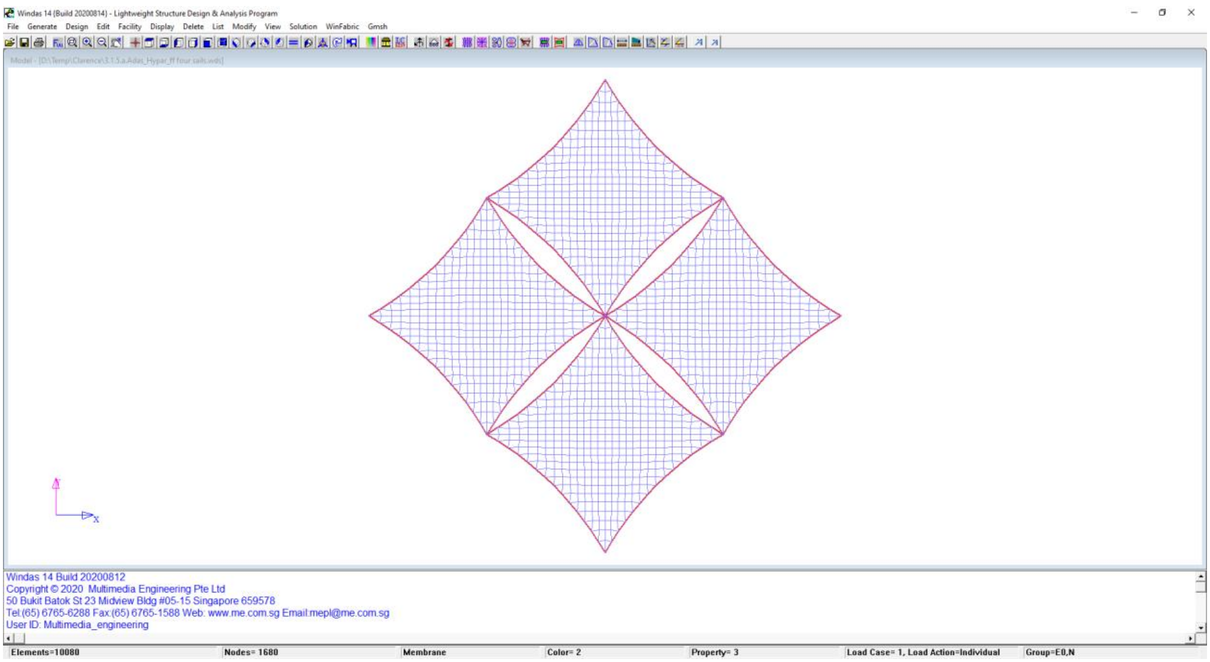
We will create this structure using the previous simple hypar model. First, **Save As** the model with other file name such as **A Four Hypar Sail**.

Copy the model from system point 1 to system point 2 using **Modify | Copy | Model.** Click the Point 1 box and then click one of the edge point of the hypar to automatically fill in the point coordinate. If point 1 coordinate box has been filled, click the point 2 coordinate box. Now, click an adjacent edge point from the previous point that you choose. Change the number of copy to 1. A hypar duplicate in this particular position will be generated.





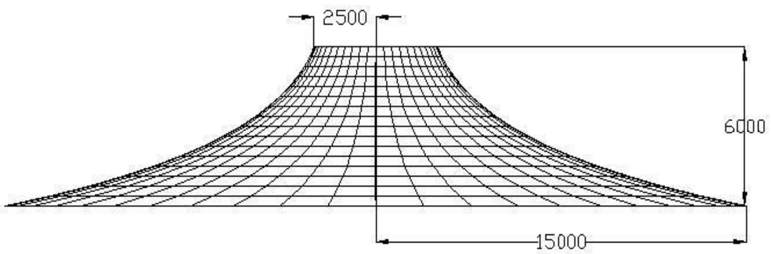
Repeat those steps to the other diagonal sides of the hypar accordingly. We will then obtained the four hypar sails.



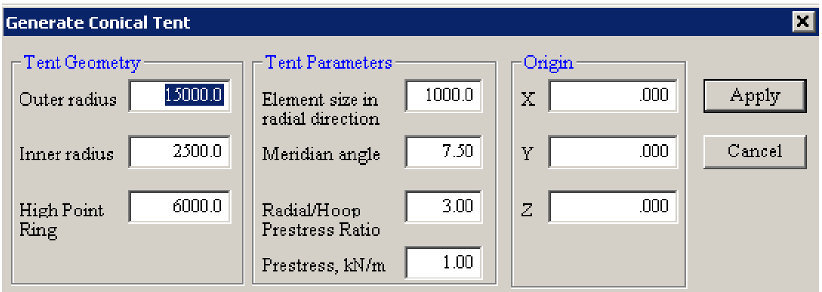
7. Radial Cones

7.1. Circular Base

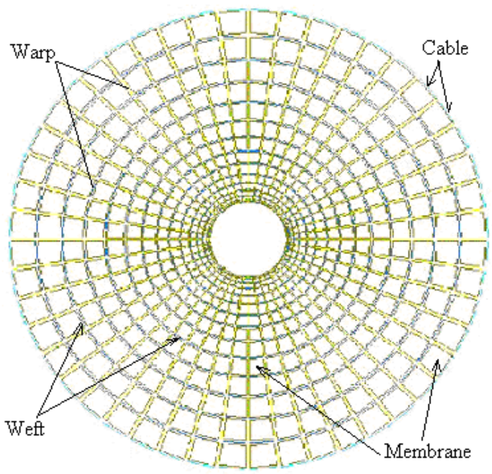
A tent with a circular high point ring and a circular base is used to demonstrate the use of standard membrane form command for fast generation of membrane form. The geometry of the tent is shown on the diagram below.



Select the **Generate | Membrane Forms | Conical Tent | Circular** command. Click on the **Accept** button to perform form finding with the default parameters.



Change the viewing angle to plan and display the model in shrink mode using the **Display | Element | Shrink | All | Accepts** command. You can see the four different types of elements.

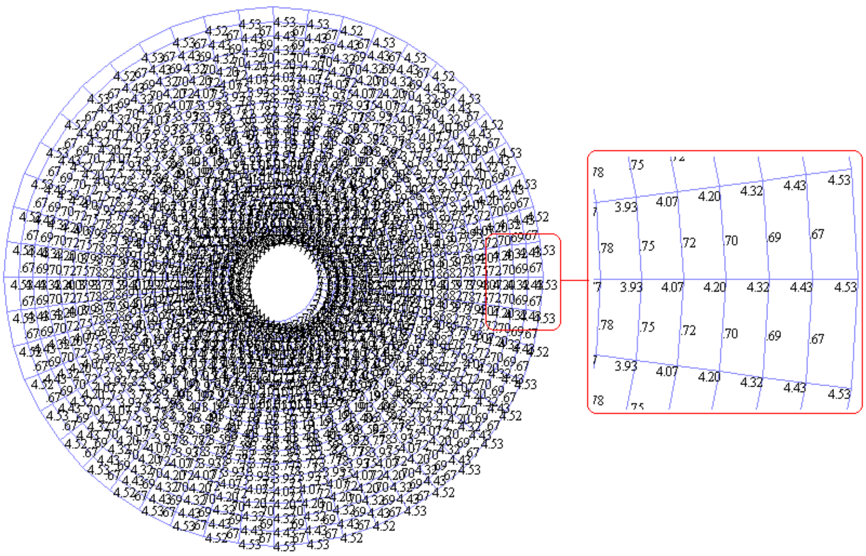


Representation	Color	Color ID	Property ID
Fabric Net (Warp/Radial)	Blue	1	1
Fabric Net (Weft/Ring)	Cyan	10	2
Border Cable	Red	13	3
Membrane Surface	Light Blue	2	15

In radial net model, the warp direction is known as the radial direction and the weft direction is known as the ring direction respectively.

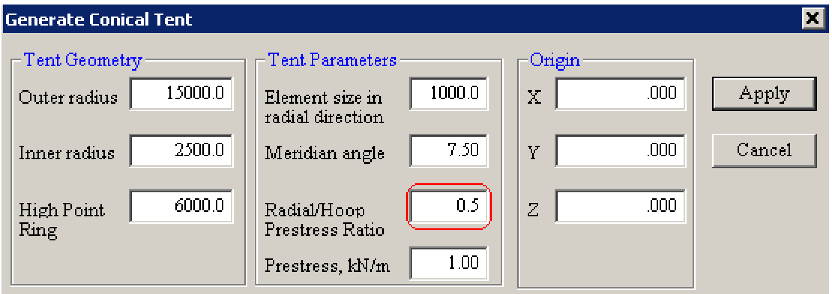
The force density on the model is also generated automatically.

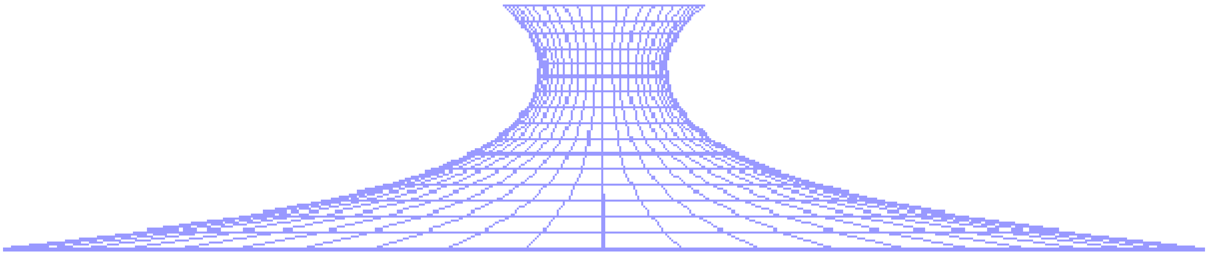
Click **Display | Load Values | Force Density** to show the force density distribution on the circular tent.



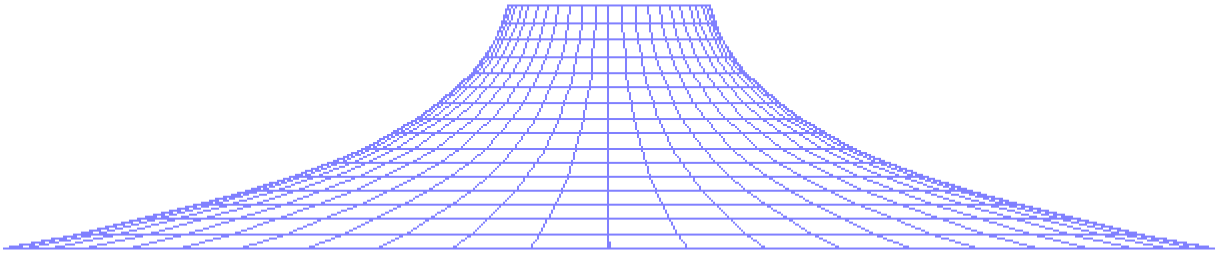
As you can see, the force density along the warp and weft directions are not the same and changes as you move along from the base of the tent to the high point ring. The force density value is a function of the pre-stress and the pre-stress ratio. For radial membrane form, maximum membrane stress always occurs at the high point ring.

Different radial to hoop pre-stress ratios will produce different generated forms. Try to recreate various 'circular base radial cone' membrane forms using pre-stress ratio of 0.5, 1.0, 5.0 and 10.0.





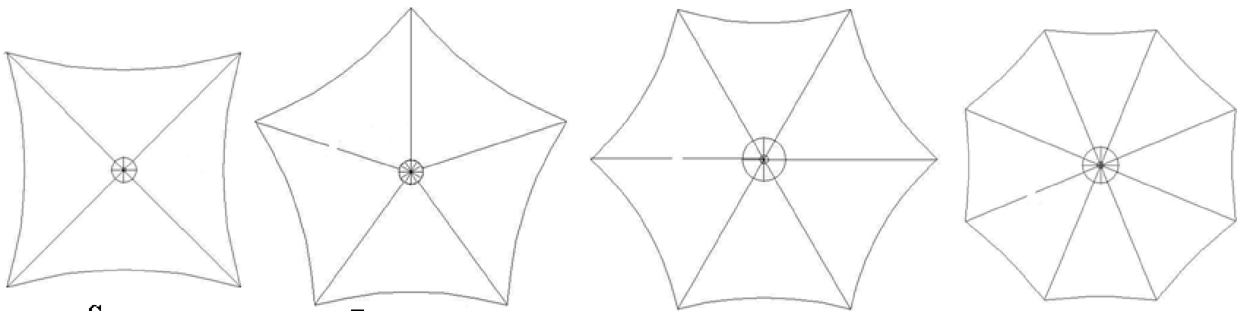
Radial/Hoop Prestress Ratio=0.5



Radial/Hoop Prestress Ratio = 1.0

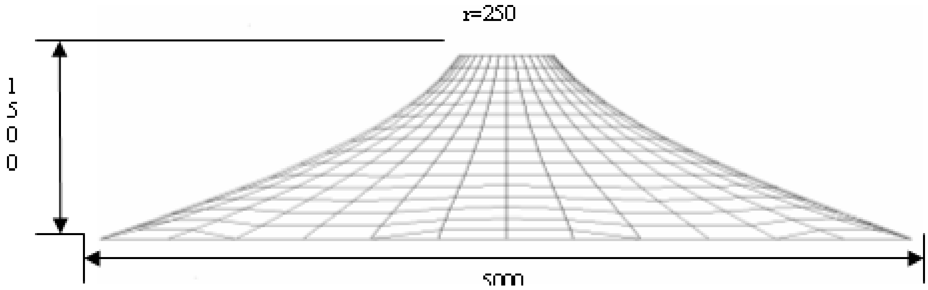
7.2. Polygonal Base

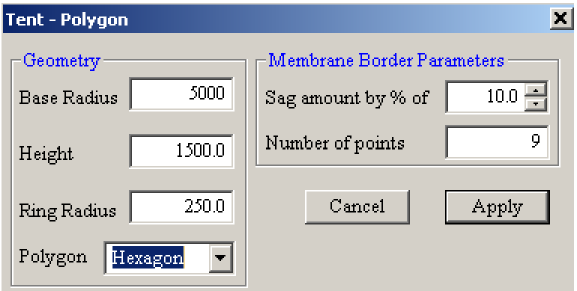
Many other kind of standard tent forms are included in Windas including square, pentagon, hexagon, octagon and decagon tents.



Square Pentagon Hexagon Octagon

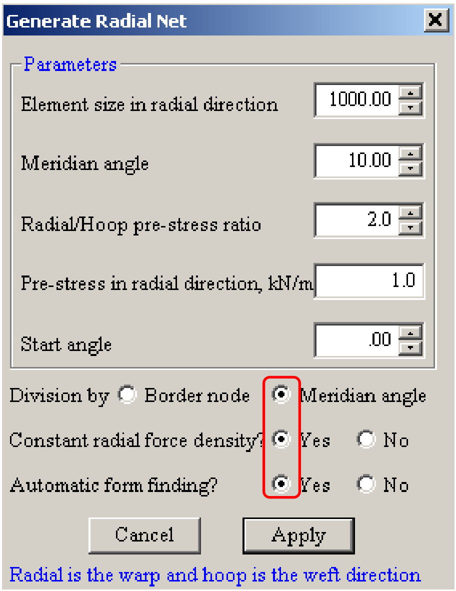
Command to use is **Generate | Membrane Forms | Tent | Polygon.** Select hexagon to create a tent with hexagonal border.

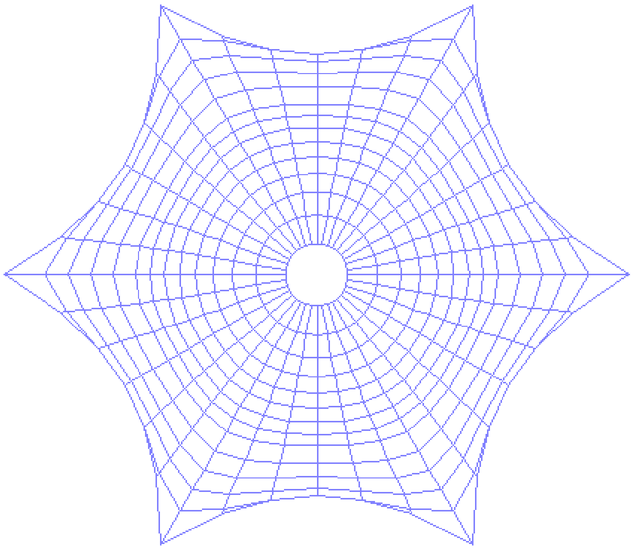




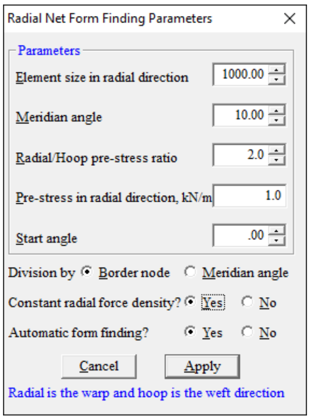
Performs form finding with default parameters to create a hexagonal tent.

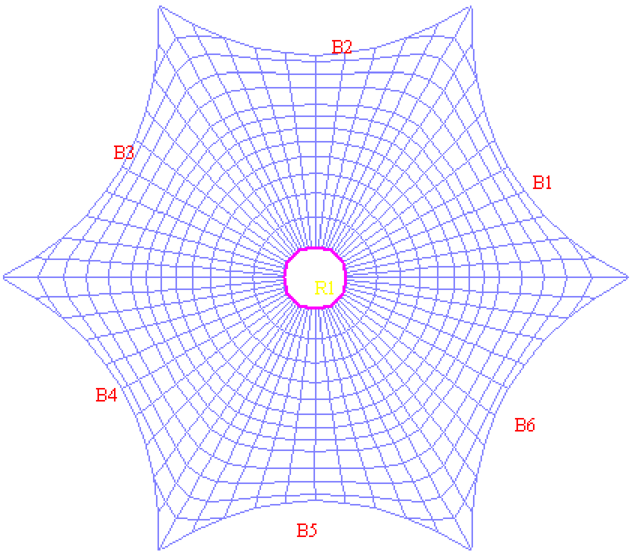
The meridian angle is used to determine the number of radial warp lines to complete a 360° around a complete tent. A meridian angle of 10 will result in 36 radial warp lines.





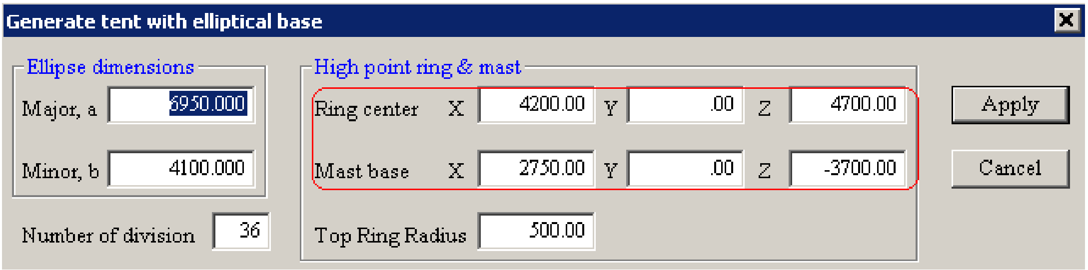
We can repeat the form finding using different method and produce different kind of form. First, cancel the existing form-finding. Click **Generate Radial Net** from the quick command toolbar. Use the division by border node method to form the tent.





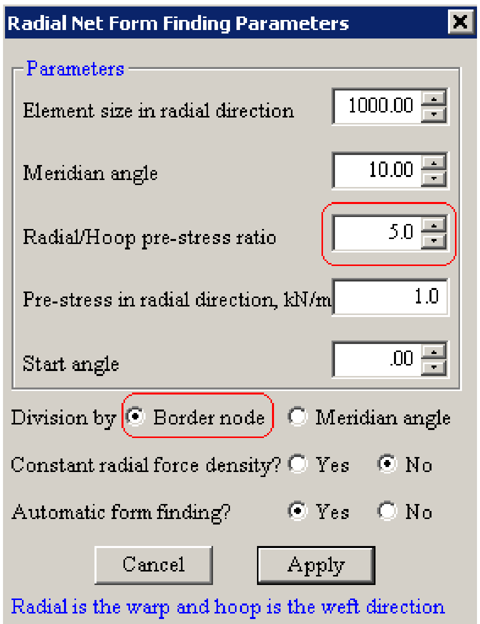
7.3. Elliptical Tent

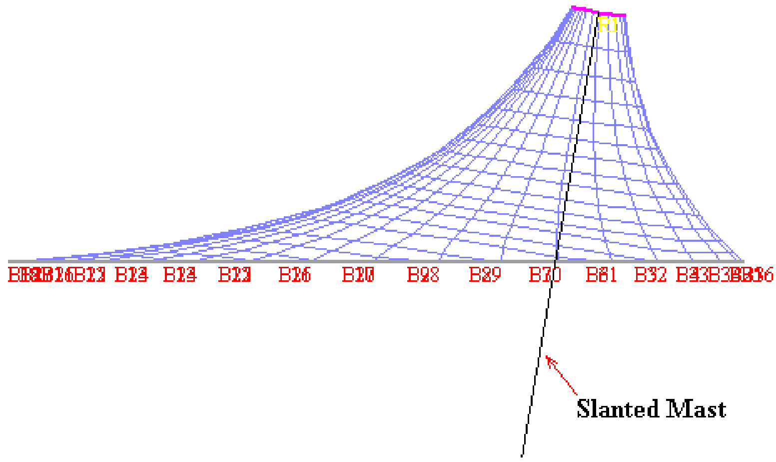
Click **Generate | Membrane Forms | Tents | Elliptical** to generate an elliptical tent. The following dialog box appears:

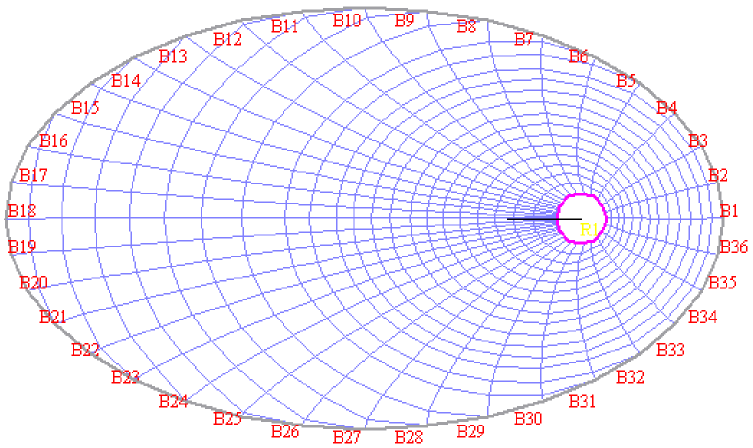


We shall use the default values for this exercise. Please note that the base of the mast and the high point ring centre are not at the same X and Y locations.

Change the Radial/Hoop pre-stress ratio to 5.0 and note that the division method is by border node.





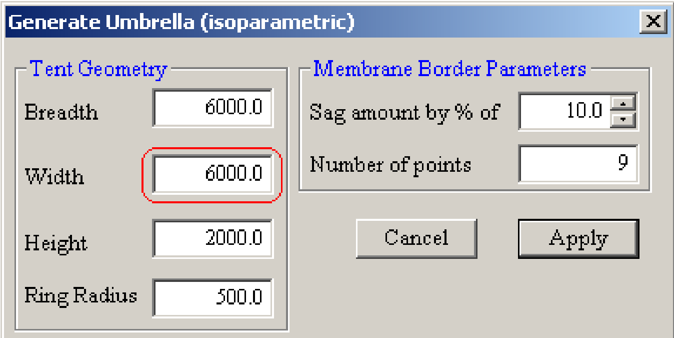


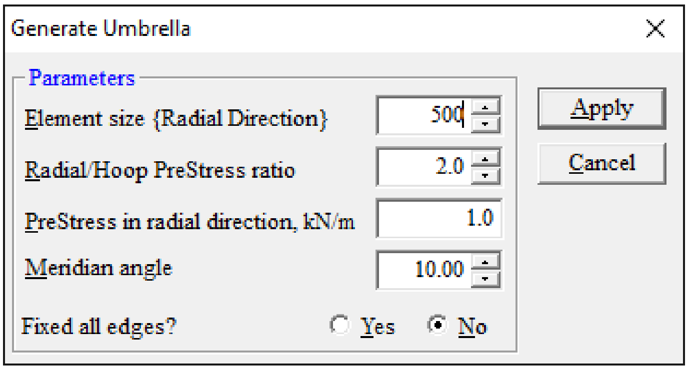
The elliptical tent is formed with the radial warp lines radiating out from the center of the top ring to each border node.

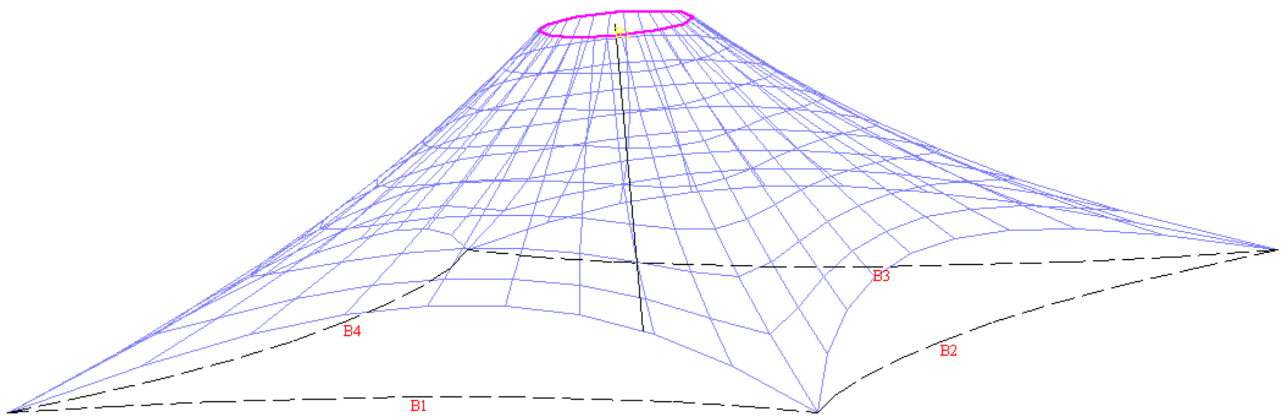
8. Umbrella

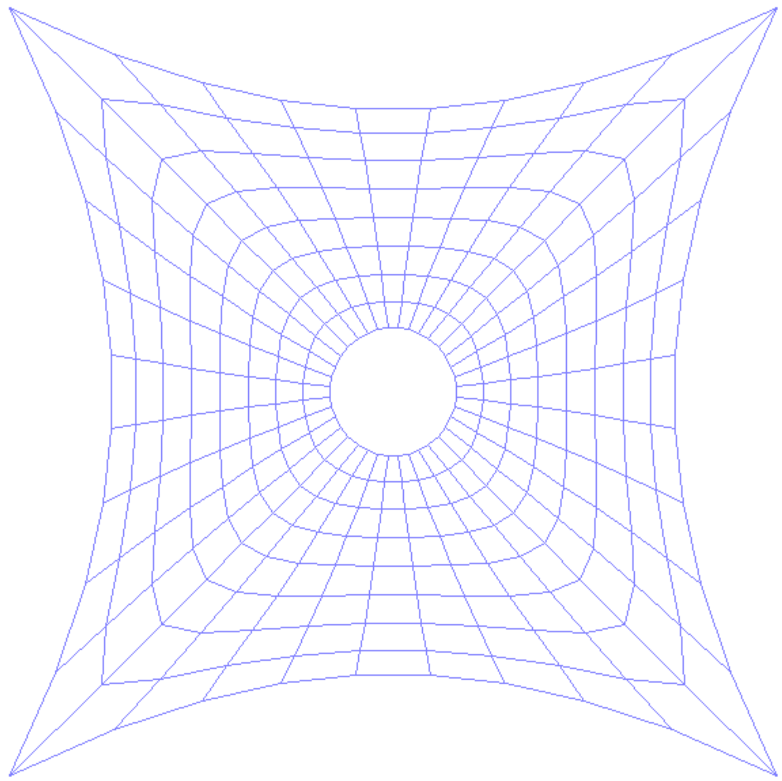
Umbrella form is almost similar to rectangular tent with a few key differences. In umbrella form, the membrane net division always goes straight to the corner; whereas in rectangular tent sometimes do not go straight to the corner, making it difficult for the seaming process. Another unique characteristic of umbrella form is that it does not have curvature around the membrane which disabling the possibility of water ponding on the membrane.

Click **Generate | Membrane Forms | Umbrella** to create the basic form of umbrella. Enter width value of 6000 and click the apply button to create a rectangular umbrella form. Put 500 for element size in radial direction to get a finer mesh.

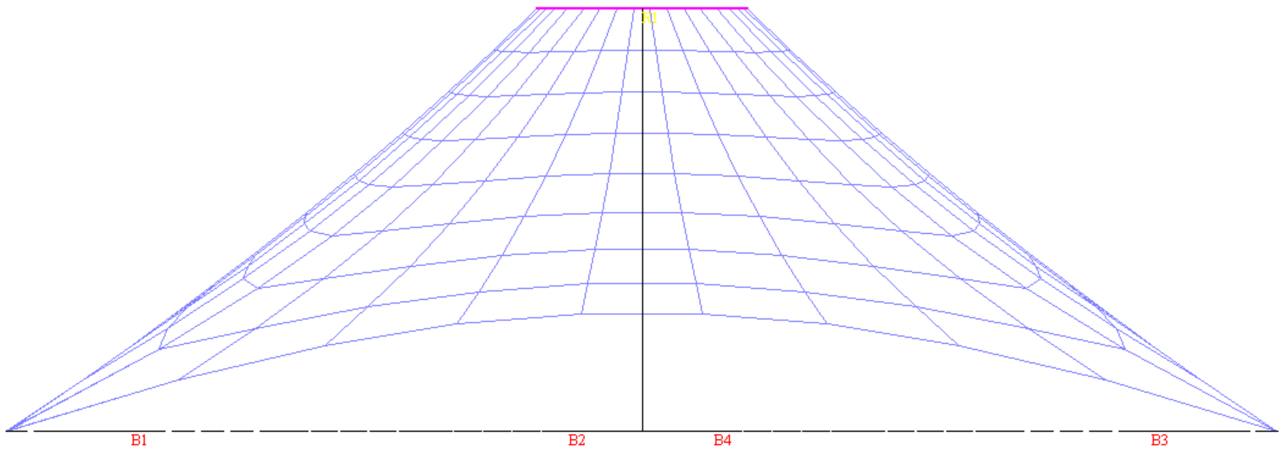








We can see from the top view that the membrane net goes straight from the center to the corner, making it easier for the seaming process.



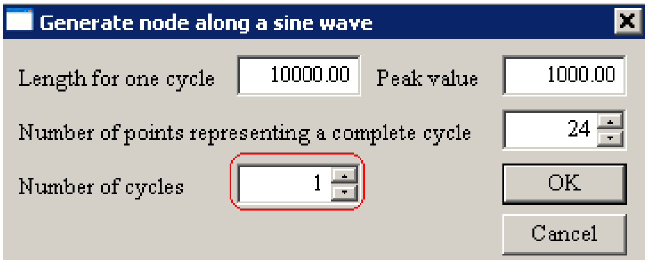
Also we can see a very limited amount of curvature on the membrane from the side view.

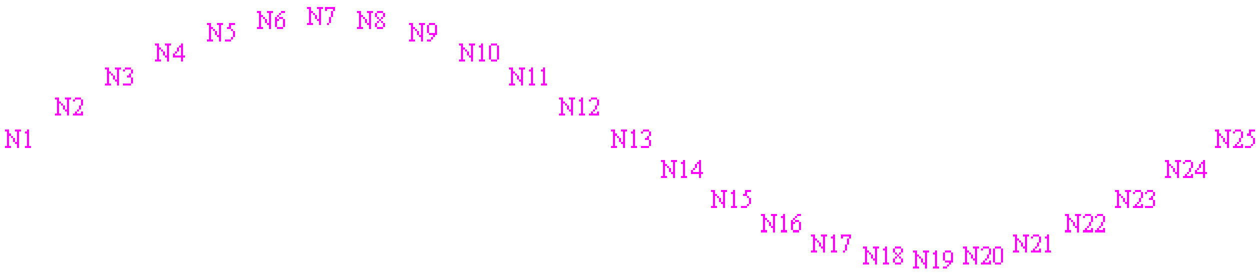
9. Surface

Ruled membrane surfaces are commonly used as overhang canopy at shopping malls, supermarket and public buildings. The main characteristic of ruled membrane surface is its linear, clear and distinct profile.

9.1. Spline Surface

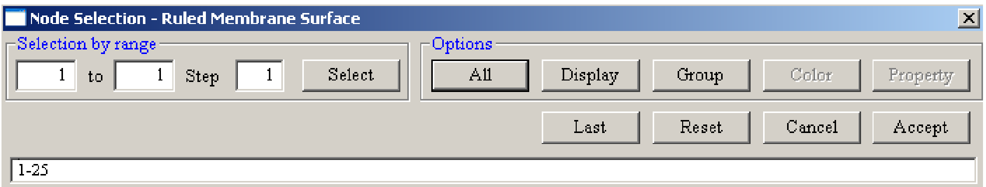
Select the **Generate | Node {more} | Shape | Sine curve** command to generate nodes along a single cycle of sine curve.

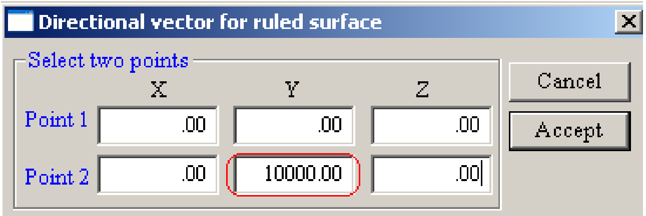




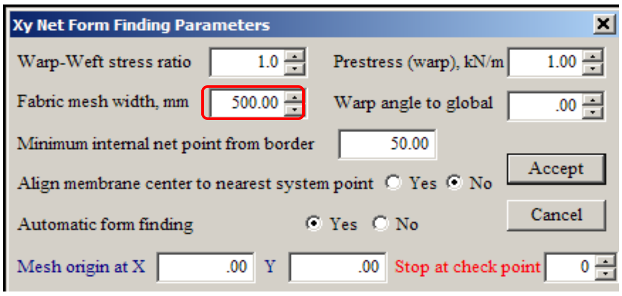
Save the model as **Spline Surface**.

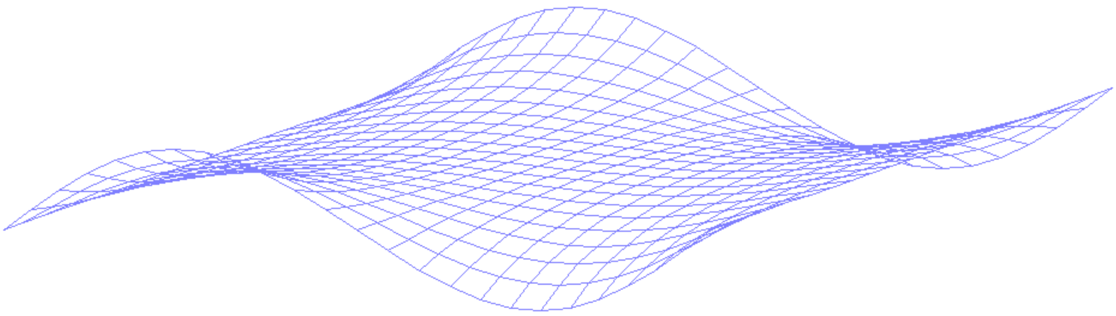
Use the **Modify | Node | Spline (Cubic)** command to modify the sinusoidal shape into an arbitrary shape. Then, select **Generate | Membrane Forms | Surface | Ruled Surface | All | Accept** to create the membrane form. Select all the nodes.





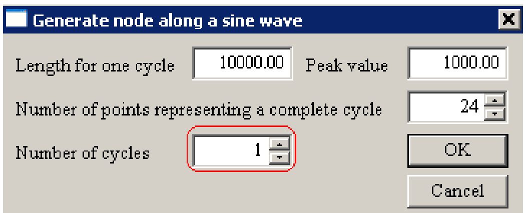
Define the directional vector for the ruled surface.

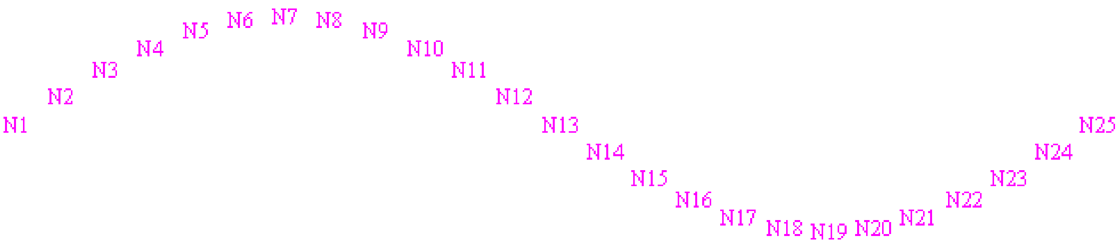




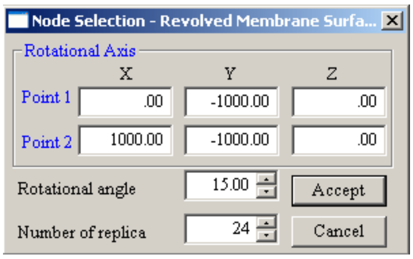
9.2. Revolved Surface

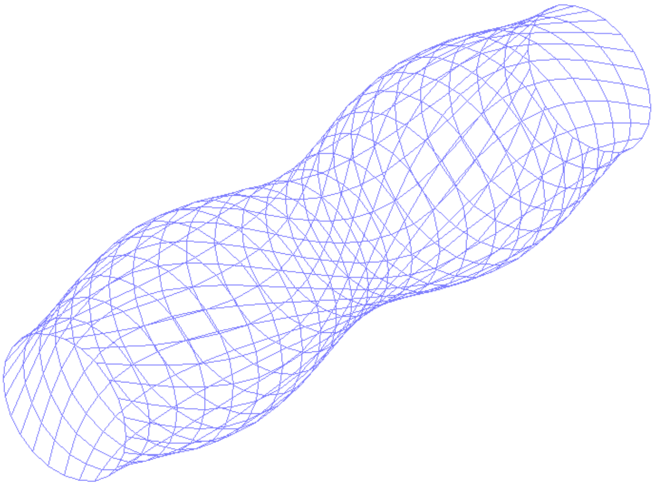
Select the **Generate | Node {more} | Shape | Sine curve** command to generate nodes along a single cycle of sine curve.



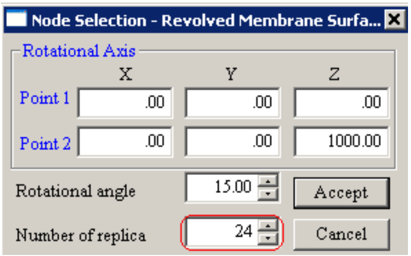


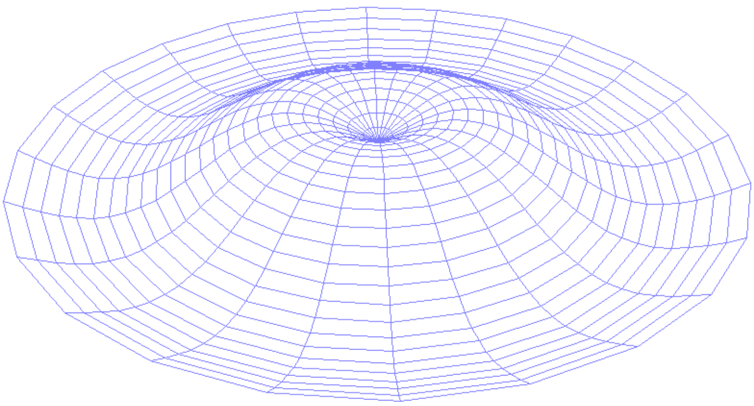
Select **Generate | Membrane Forms | Surface | Revolved Surface | All** to rotate the selected nodes to form a complex surface.





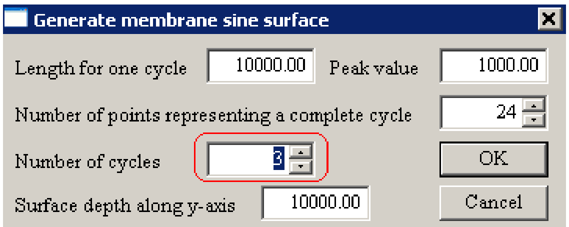
Repeat the same step but change to these parameters. The following parameters will generate another form.

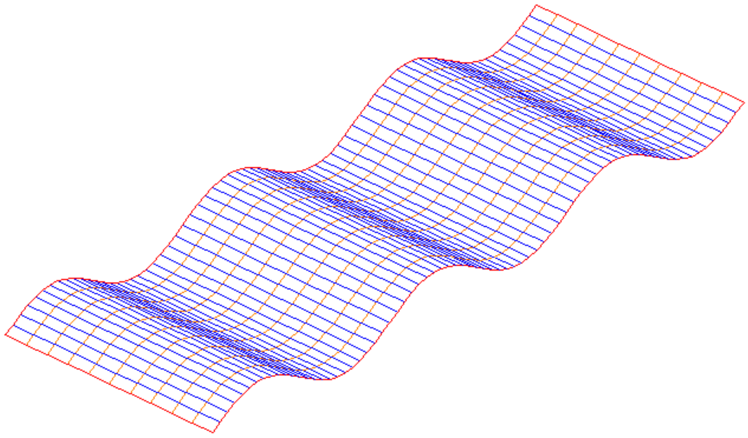




9.3. Sine Curve

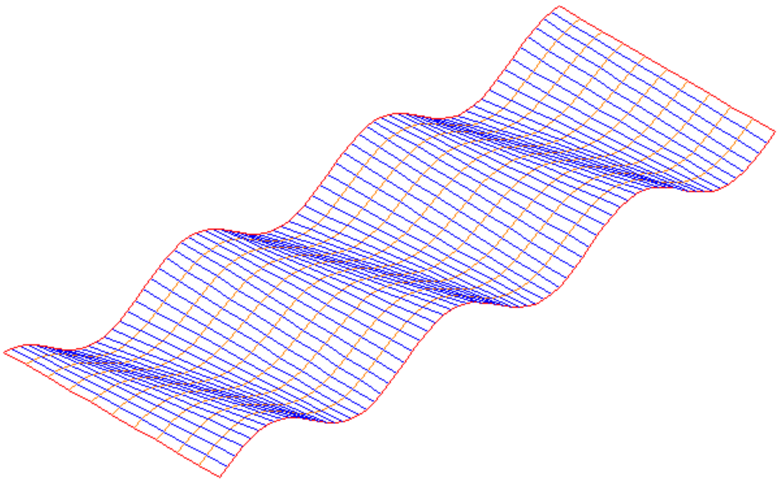
Select the **Generate | Membrane Forms | Surface | Sine Surface** command. Change the number of cycles to 3 and click the OK button and save the model. Click **Accept** to form a ruled surface with a sine curve profile.





9.4. Sine And Cosine Curves

Another common form of ruled surface is a sine curve along one end and a cosine curve along the other end.



10. XY Mesh

XY mesh is a very useful command to generate a membrane with an irregular geometry. It will also generate the membrane net into a constant rectangular shape; whereas in regular net form finding will not give you a constant rectangular shape.

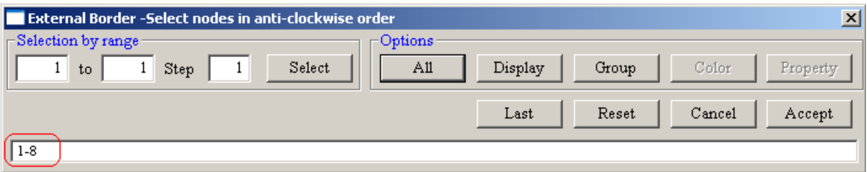
10.1. System Point

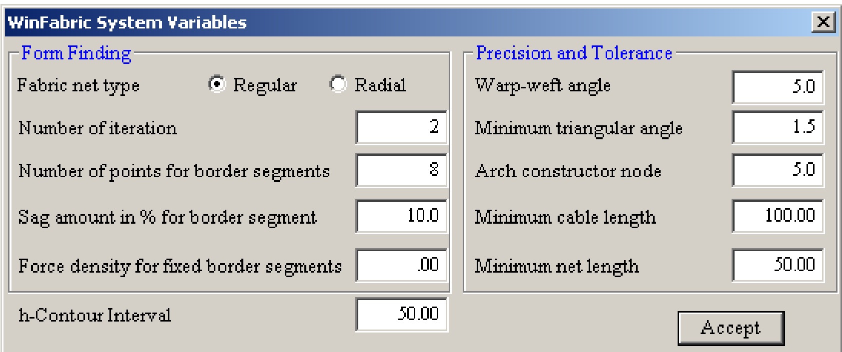
Use the **Generate | Node** command to create this system points.

Node ID	X	Y	Z
1	0	0	0
2	3750	-2500	-1500
3	7500	-3750	1000
4	11250	-2500	-1500
5	15000	0	0
6	11250	2500	-1500
7	7500	3750	1000
8	3750	2500	-1500

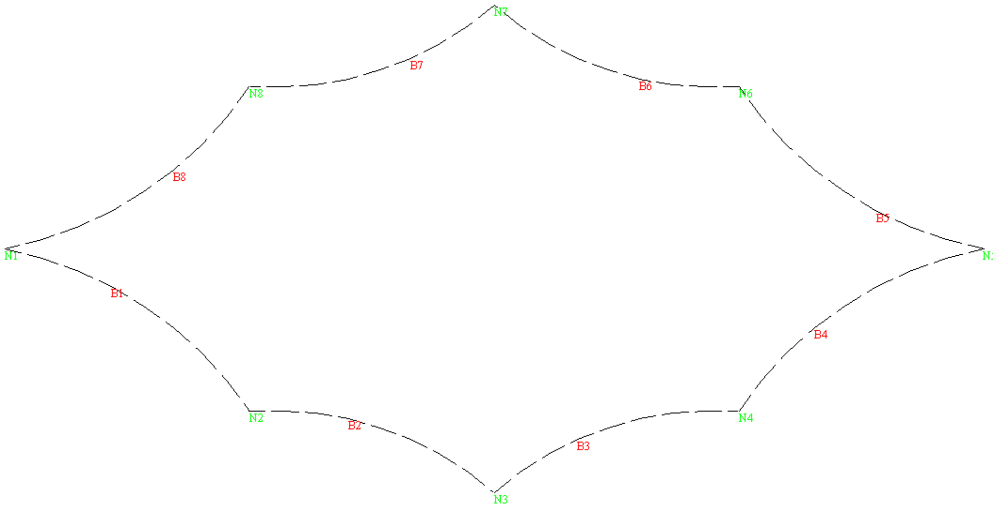
10.2. External Membrane Border

Use **Generate | Membrane Constructors | External Border** to create the external border.



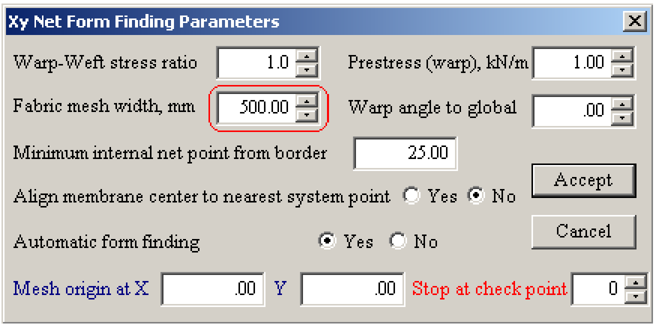


Click the accept button to generate the external borders with the default parameters.

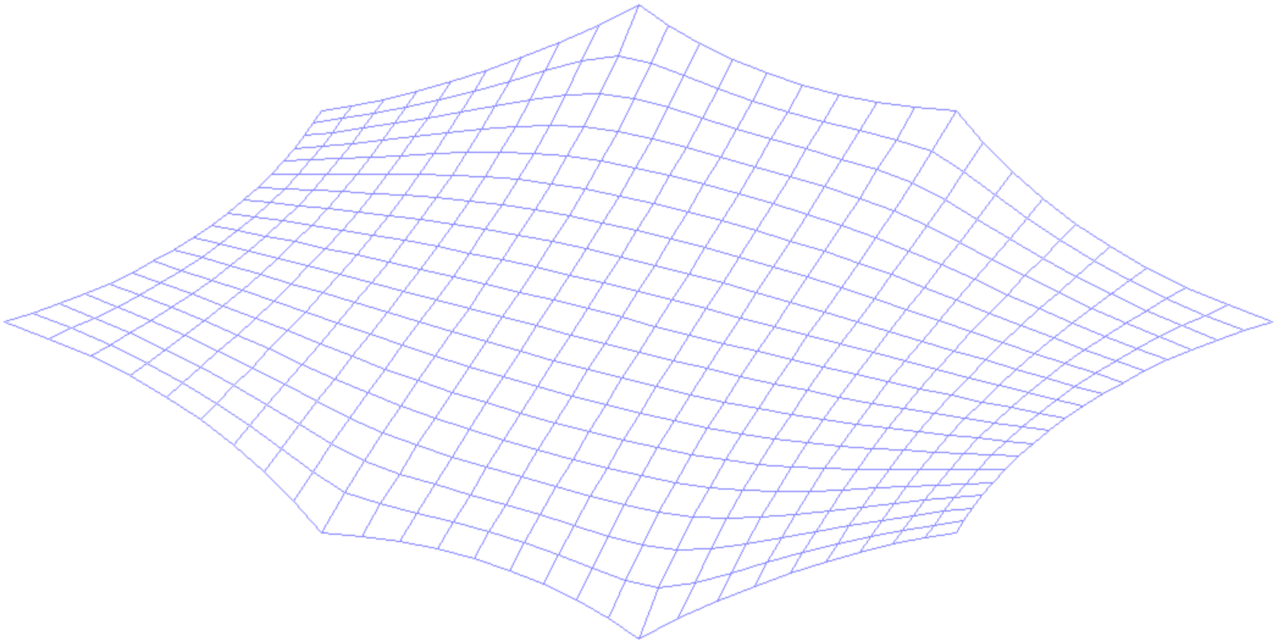


10.3. Form Finding

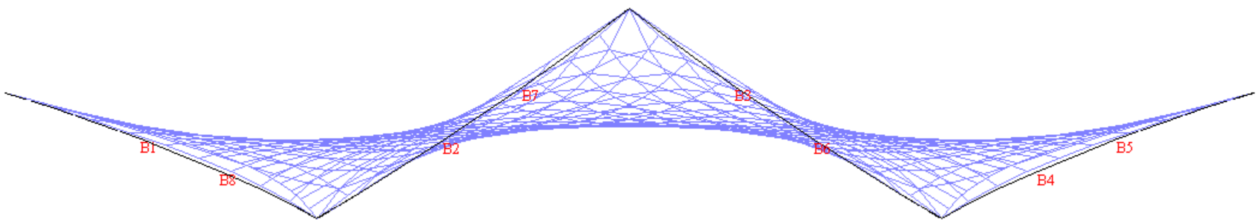
Use the **Generate | Membrane Forms | Xy Mesh** command to generate the form finding under XY mesh.

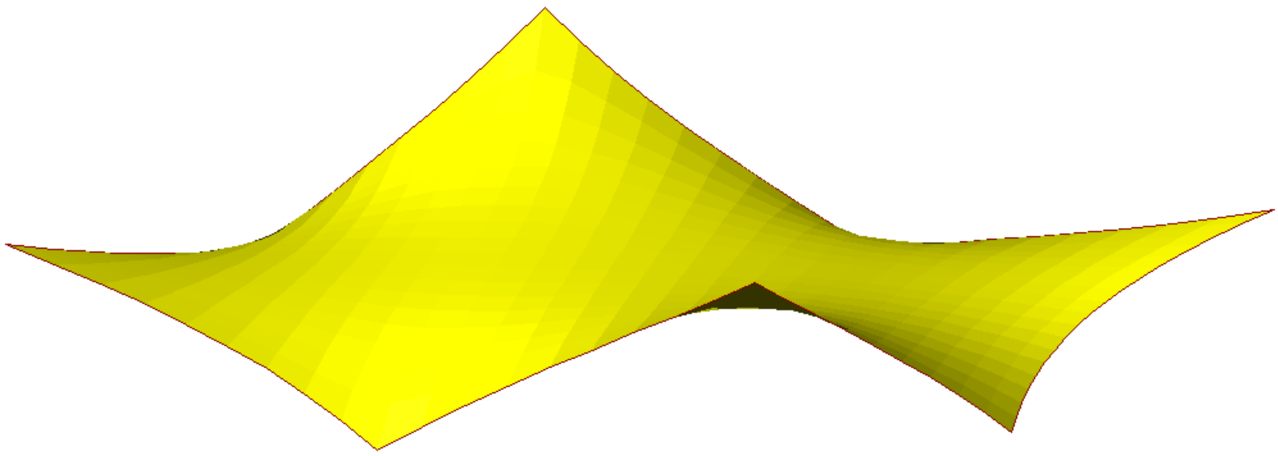


Put 500 mm for the fabric mesh width and click the accept button to generate the form finding.



As you can see that the generated membrane net are all rectangular.

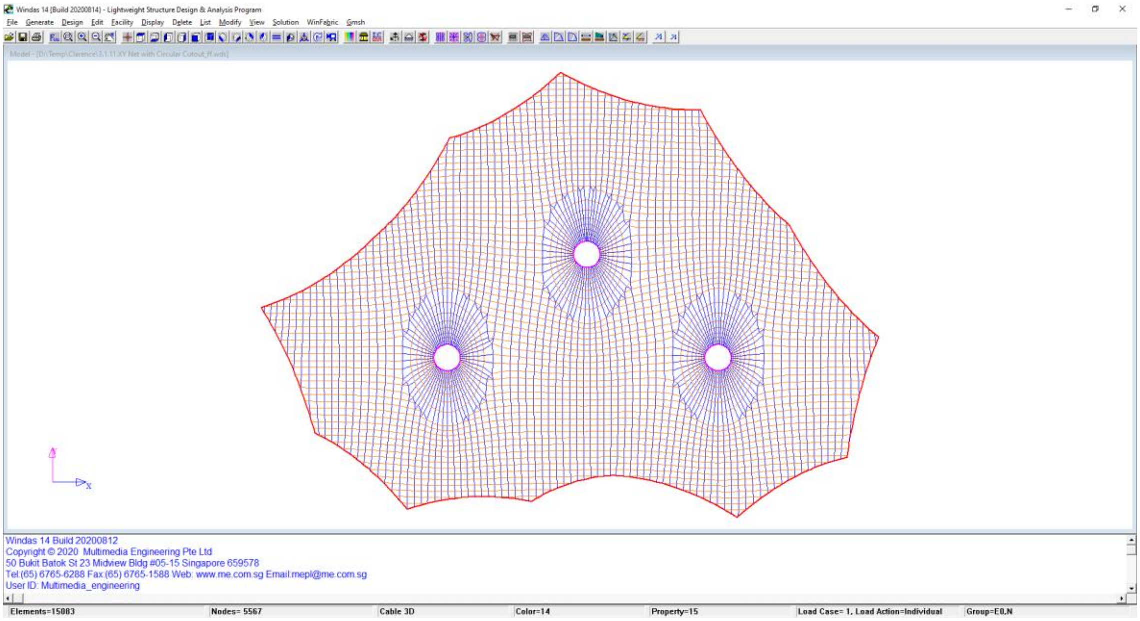




11. XY Net with Circular Cut-out

The maximum width of fabric material available in the market is 3m. For large tent like membrane form, material wastage is extremely high if the cutting pattern is done radially. It is more economical to adopt XY net with circular cut-out membrane form for large tent.

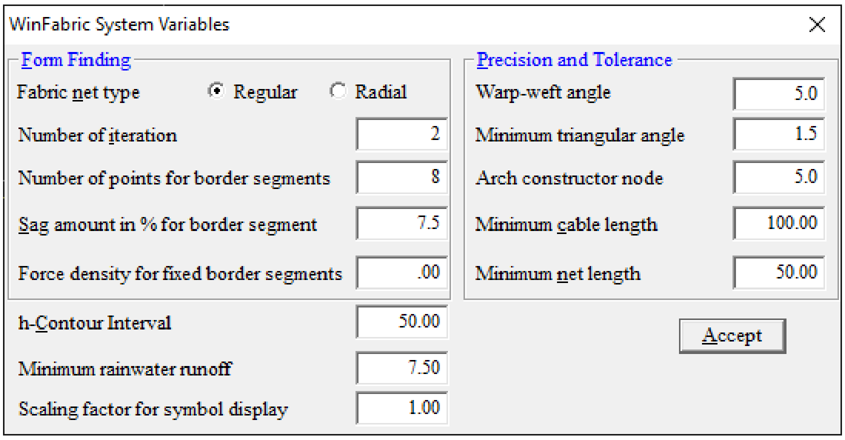
11.1. A Triple Cone



Create this system point using **Generate | Node (more) | Cartesian** and key in these values accordingly.

Node	x-coord	y-coord	z-coord
1	3804.31	-431.08	155
2	11965.13	-7165.67	155
3	23002.56	-6479.32	155
4	30502.45	-4177.43	155
5	41256.21	-7889.81	155
6	51014.21	-2549.6	155
7	53835	8095	155
8	45867.5	18038.74	155
9	37985.16	28249.17	155
10	25597.91	31561.7	155
11	15746.58	25744.14	155
12	9278.85	17135.68	155
13	-990	10712	155

The external membrane border is defined by nodal point 1 to 13 in anticlockwise order. The command for creating this external membrane border is **Generate | Membrane Constructors | External Border | All | Accept.** The fabric net type is **Regular**.



11.1.1. High Point Rings

To create the high point rings, click **Generate | Membrane Constructors | Circular Ring.**

Input the ring coordinates as shown below,

Fabric Top Ring 1
Ring Centre { 15529.01, 6273.31, 4750.00} Radius= 1175.00
Base of Mast { 15529.00, 6273.31, 155.00}

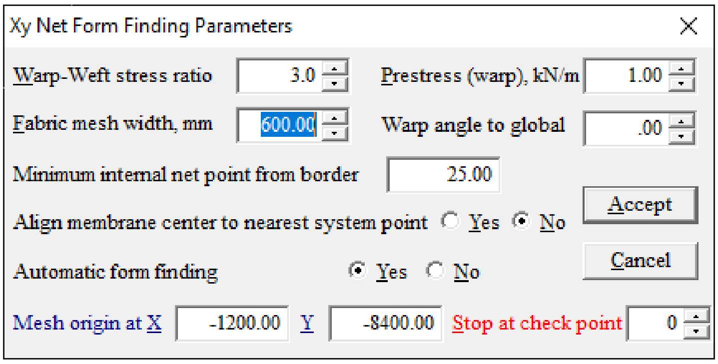
Fabric Top Ring 2
Ring Centre { 27929.48, 15434.25, 5250.00} Radius= 1175.00
Base of Mast { 27929.48, 15434.25, 155.00}

Fabric Top Ring 3
Ring Centre { 39579.82, 6273.37, 4750.00} Radius= 1175.00
Base of Mast { 39579.82, 6273.37, 155.00}

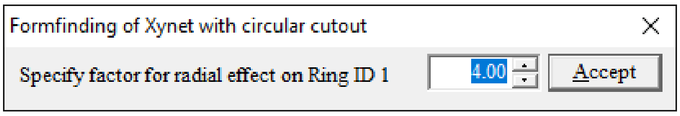
11.1.2. Form Finding

Perform form finding with the regular net command **Generate | Membrane Forms | Regular Net** or simply click on the **regular net form finding button** located at the quick button toolbar.

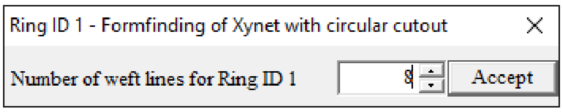
In the XY Net form finding parameter dialog box, change the fabric width to 600mm.



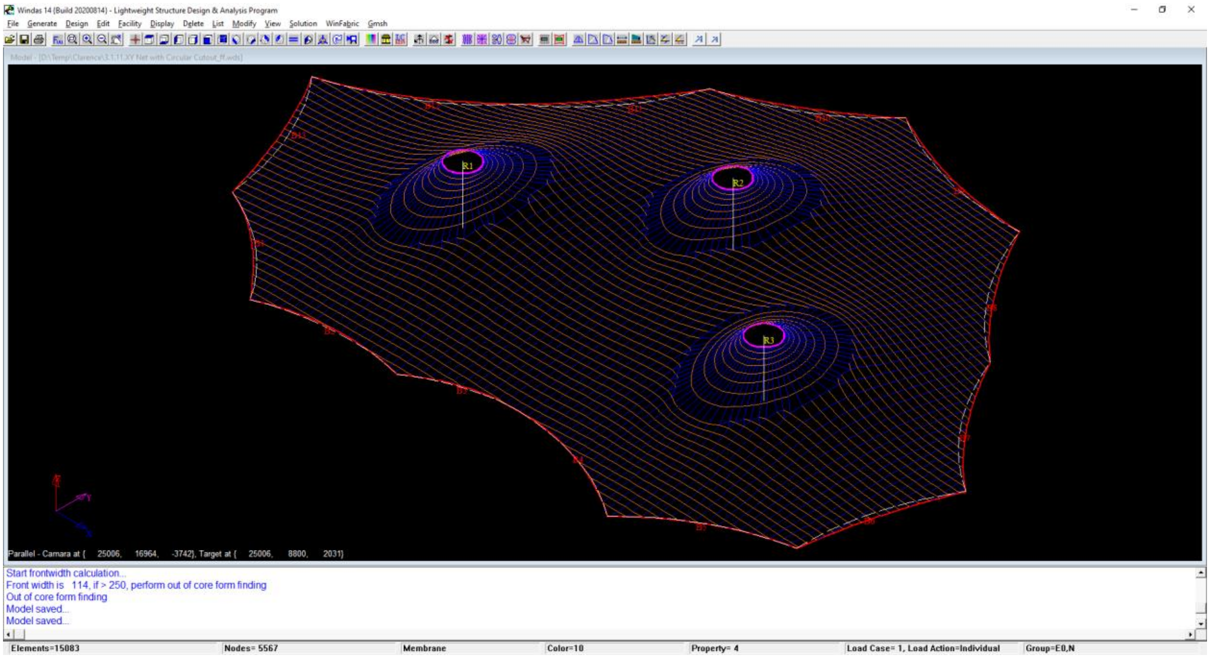
Specify the extent of the radial net from the ring centre in terms of the ring radius. Use the same value for ring 2 and ring 3.



Specify the number of weft line radiating from the ring as 8 for all three rings.



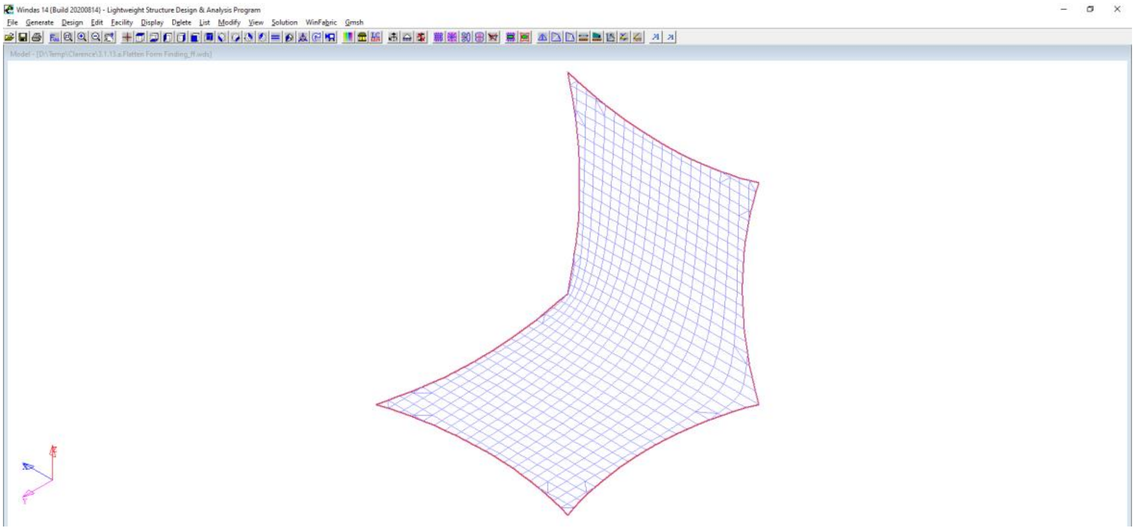
A triple cone membrane will then be formed.


Windas 14 (Build 20200814) - Lightweight Structure Design & Analysis Program
File Generate Design Edit Facility Display Delete List Modify View Solution WinFabric Gmsh
Parallel - Camara at (25006, 16964, -3742), Target at (25006, 8800, 2031)
Start frontwidth calculation...
Front width is 114, if > 250, perform out of core form finding
Out of core form finding
Model saved...
Model saved...
Elements=15083
Nodes= 5567
Membrane
Color=10
Property= 4
Load Case= 1, Load Action=Individual
Group=E0,N


ZOO

12. Bended Membrane Form Finding

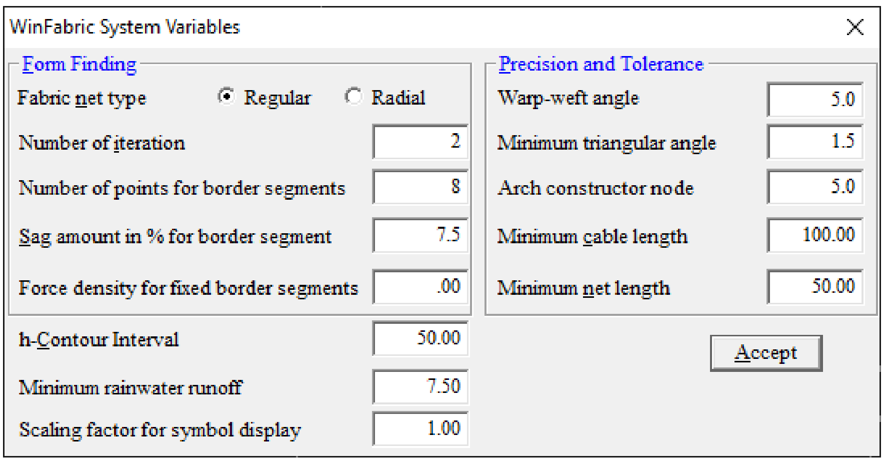
This example will attempt to create an L-shaped membrane form using the tilt command. The final form would look like this figure below.

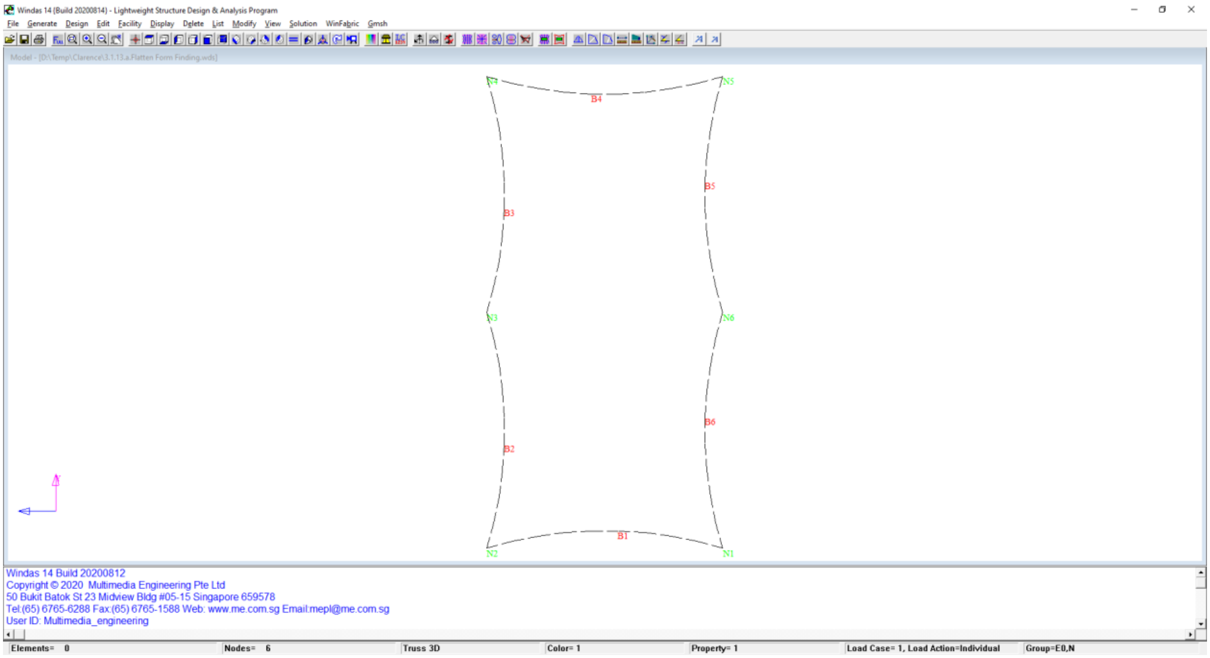


Create the system points below using **Generate | Node (more) | Cartesian.** Key in the values accordingly.

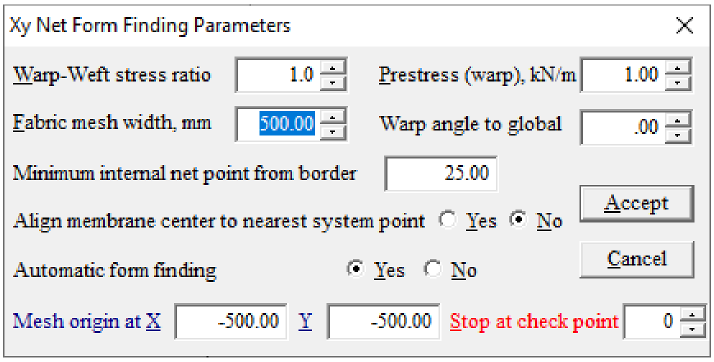
Node	x-coord	y-coord	z-coord
1	0	0	0
2	10000	0	0
3	10000	10000	0
4	10000	20000	0
5	0	20000	0
6	0	10000	0

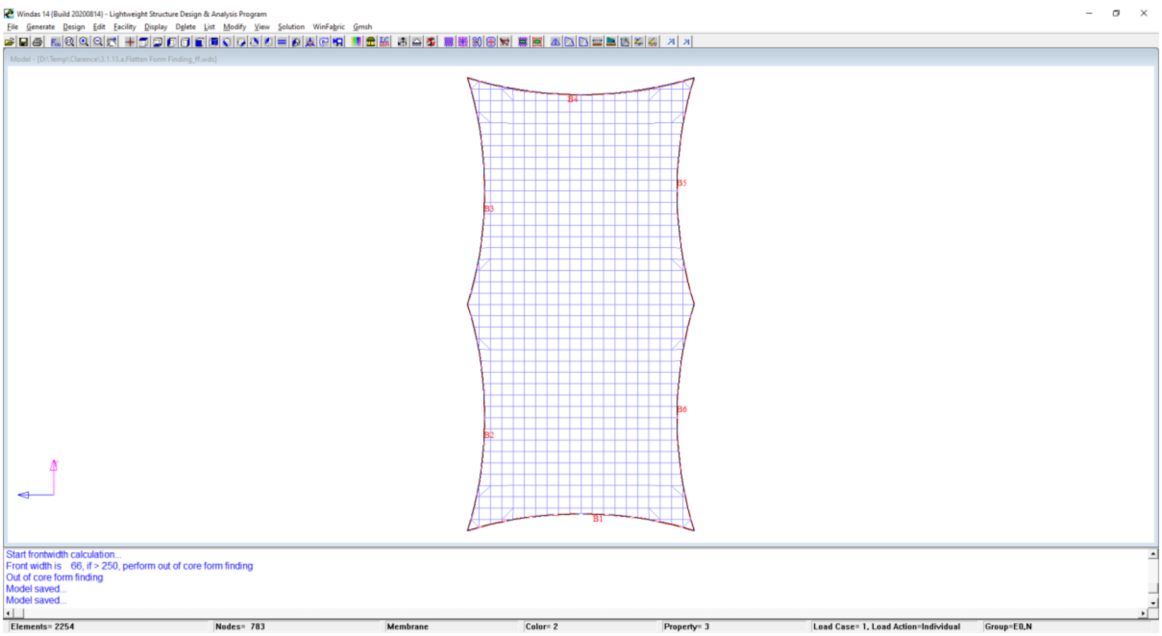
The external membrane border is defined by nodal point 1 to 6. Click **Generate | Membrane Constructors | External Border | All | Accept**. We will then proceed with Windas default setting.





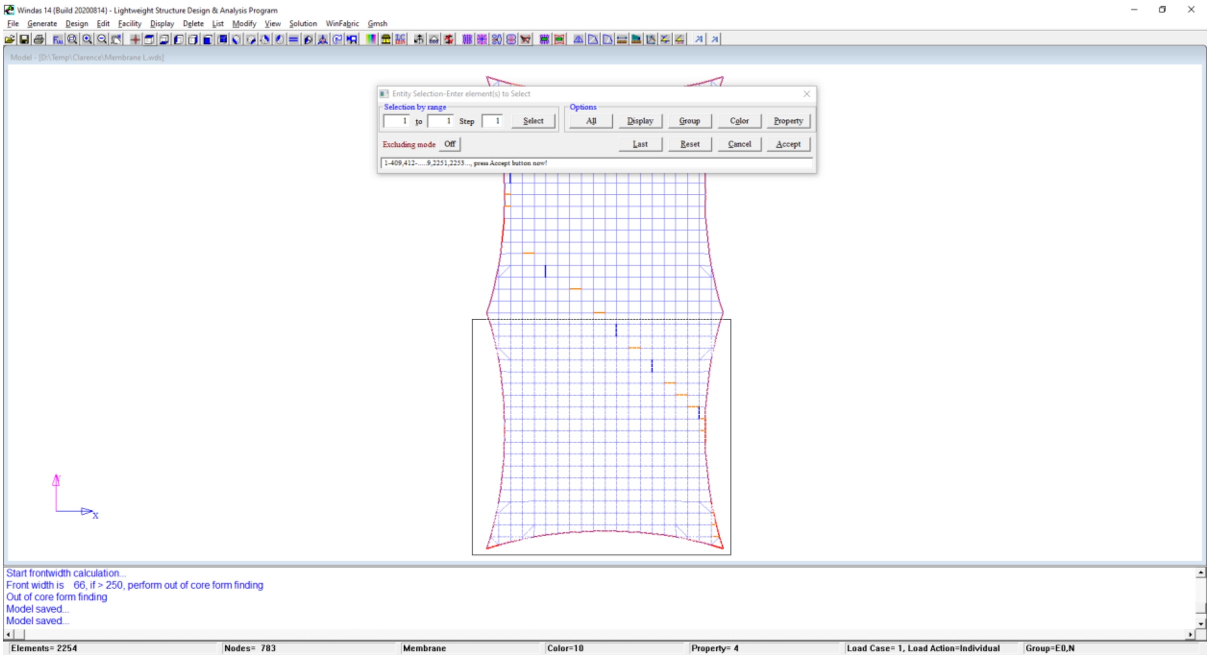
These borders are actually flat. We need to perform form-finding first before bending the structure. Use **Generate | Membrane forms | Regular Net** to perform the form- finding. Accept default.



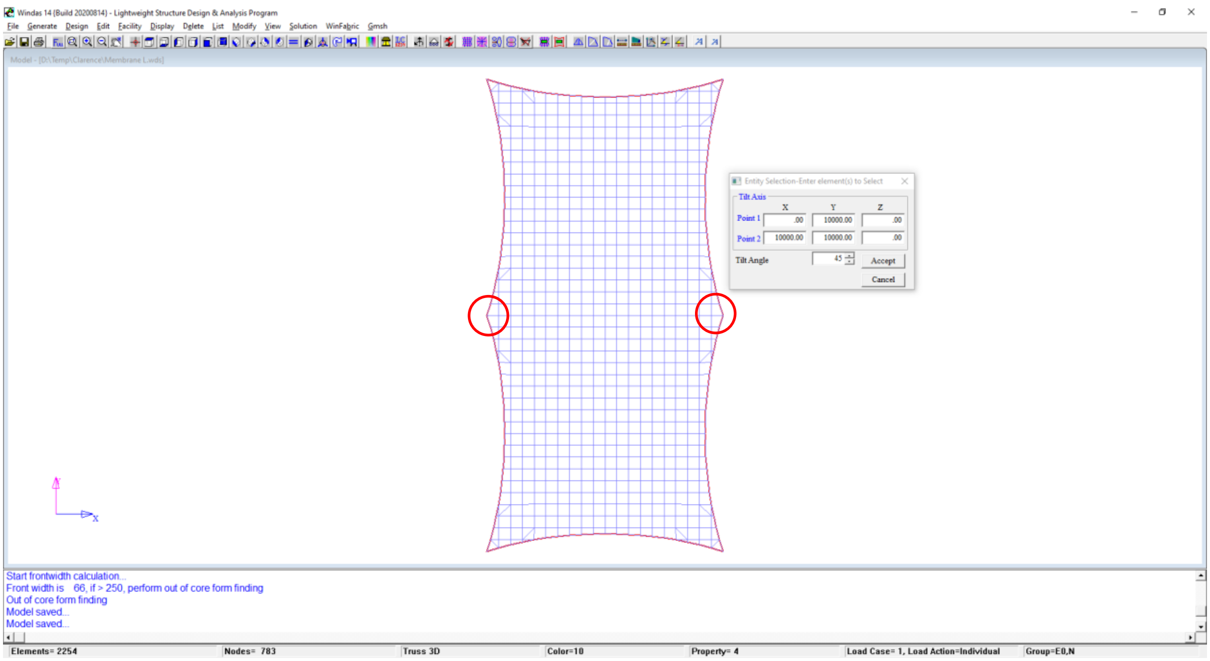


Click **File | Save As** we can name the model as **Membrane L**.

We will now proceed to bending the figure. Click **Modify | Element | Tilt**. Choose the half element of the bottom part of the structure. Click Accept.

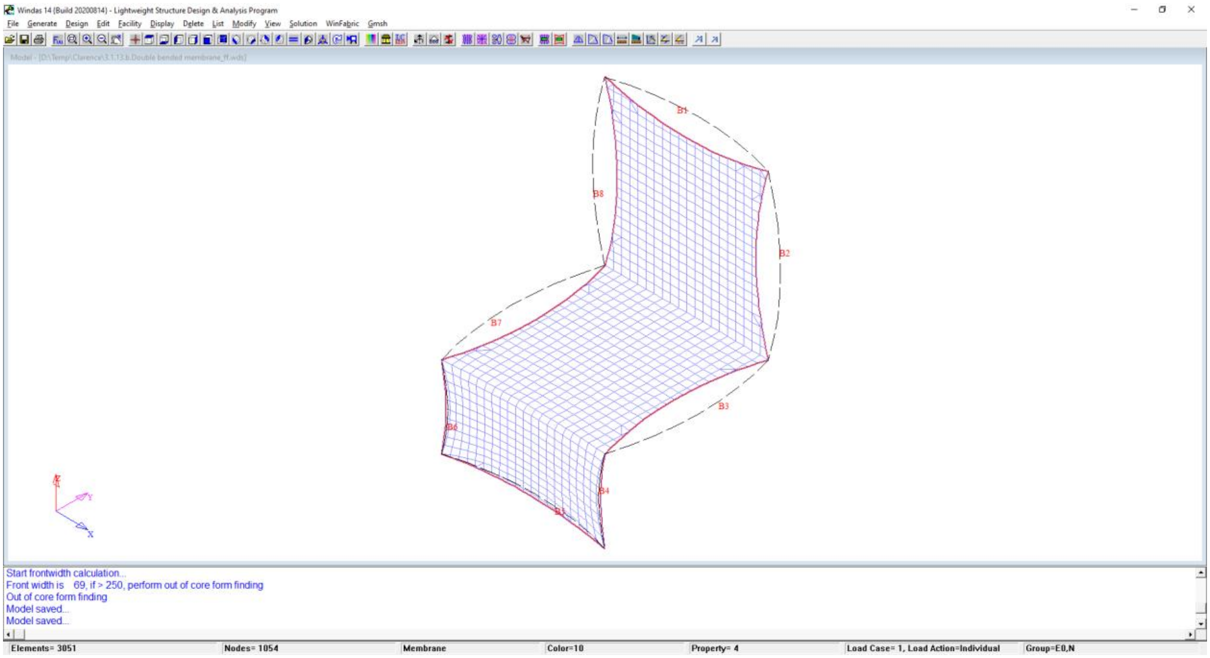


We will now have to choose the bending axis. Choose the middle points as the tilting axis. Insert the tilting degree as 90°.



The proposed model should then be generated.

We can also create double bended Membrane as such repeating the same steps.



You can follow this system coordinate and proceed with the procedure.

Node	x-coord	y-coord	z-coord
1	0	0	0
2	10000	0	0
3	10000	10000	0
4	10000	20000	0
5	10000	25000	0
6	0	25000	0
7	0	20000	0
8	0	10000	0

13. Arch Ridge & Valley

This section will show how to create membrane forms with arches, ridges, and valley cables.

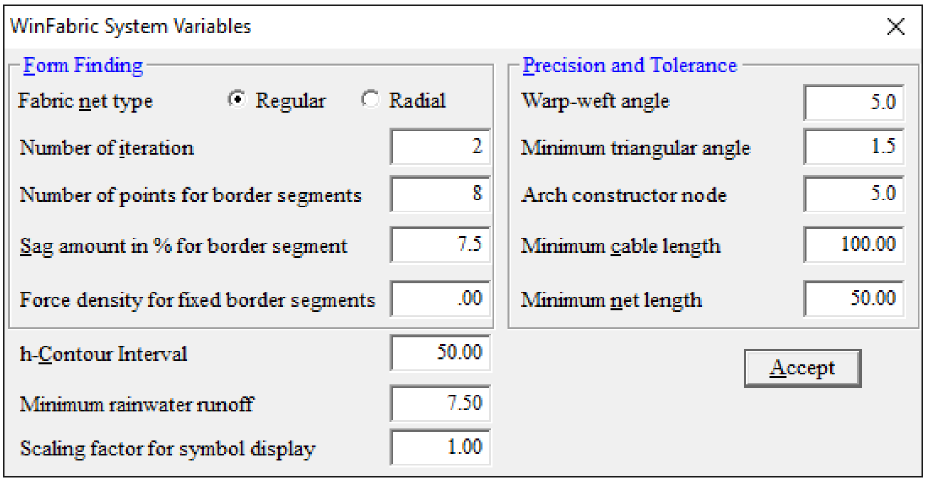
13.1. External Arch

In this example an external steel arch is being introduced to a rectangle membrane.

Create the following system points with **Generate | Node (more) | Cartesian** command.

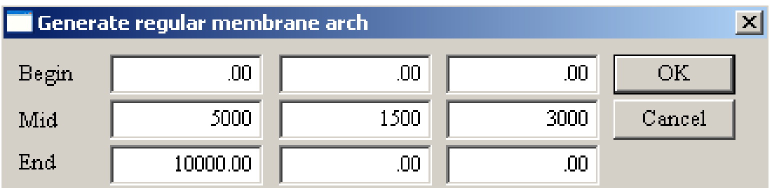
Node ID	X	Y	Z
1	0.0	0.0	0.0
2	10000.0	0.0	0.0
3	10000.0	10000.0	0.0
4	0.0	10000.0	0.0

Create the external membrane border with **Generate | Membrane Constructors | External Border** command. The external membrane border is defined by system points 1, 2, 3, and 4 in anticlockwise order. Click **Accept** to accept the default system from Windas.

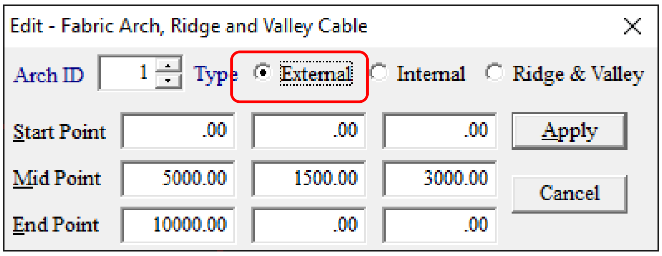


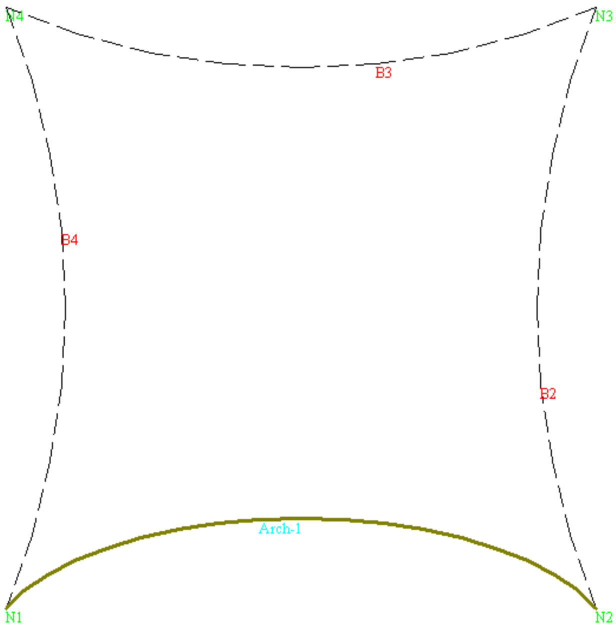
Save the model as **External_Arch**

To create the external arch, use **Generate | Membrane Constructors | Arches | Regular | 3Pt Arch** command. Insert the values shown below.

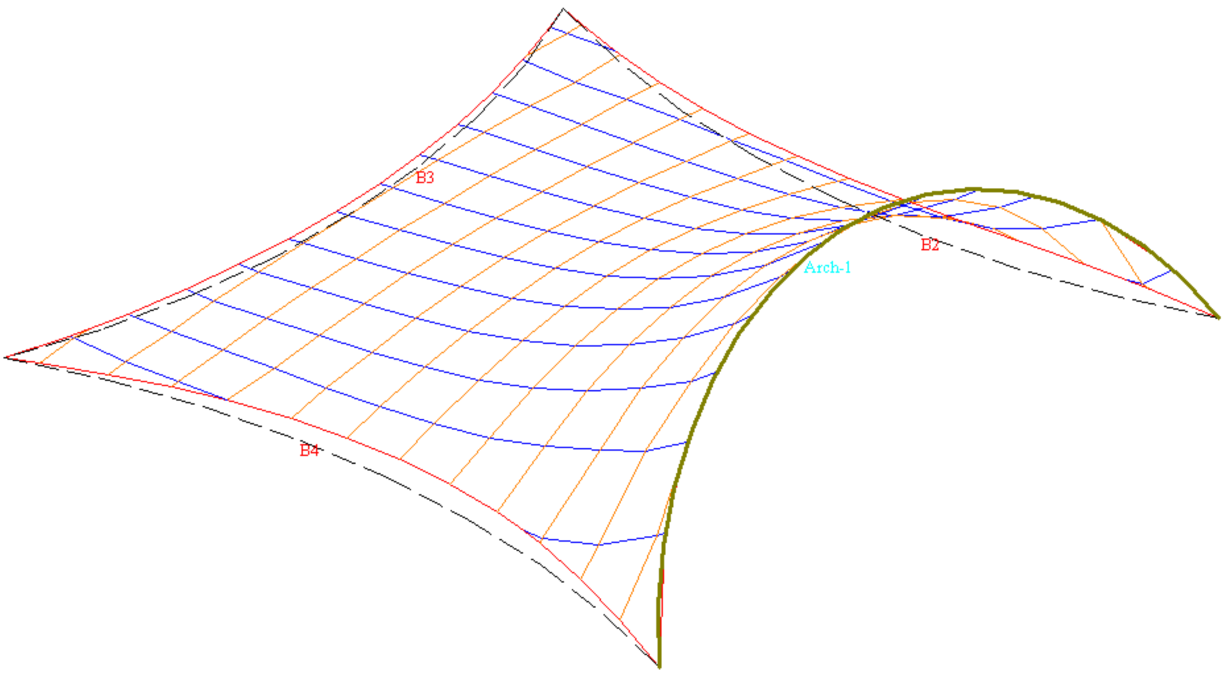


By default, the arch is an internal arch and is shown in red. Use **Edit | Membrane Constructors | Arch Ridge and Valley** command.





Perform form finding with fabric mesh width of 800mm gives

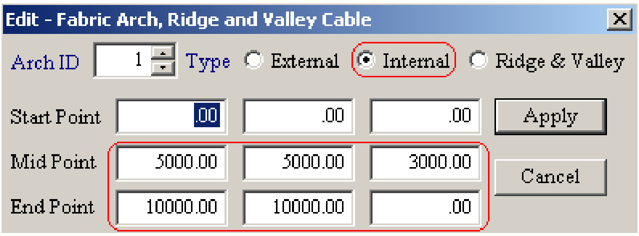


13.2. Internal Arch

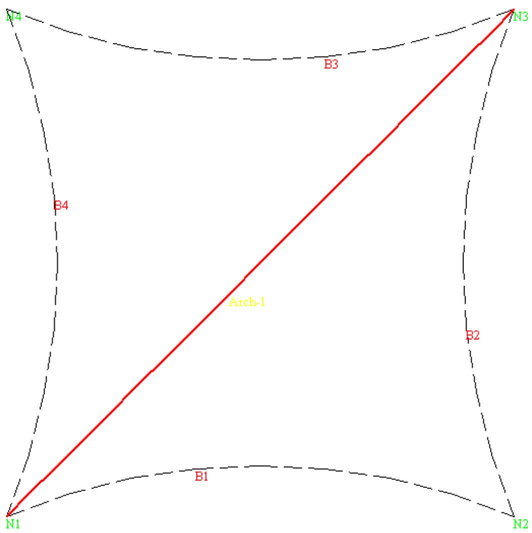
Internal steel arches are often used in the design of tensile membrane structures. The membrane is either run over the arches or fastened intermittently to the steel arch via clamping plate or garland cables.

Singapore Botanic Garden

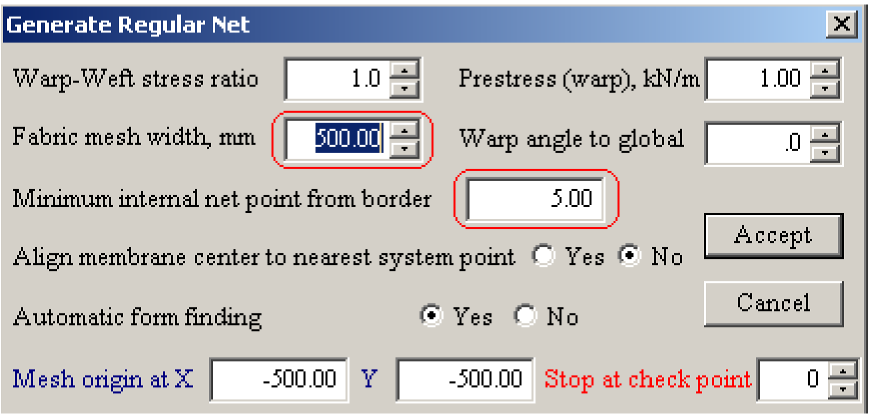
Use the **Edit | Membrane Constructors | Arch, Ridge & Valley** command to modify the external arch of **External_Arch** model and change back to internal arch. Change the position of the mid point and the end points of the arch to the values shown in the dialog box below.



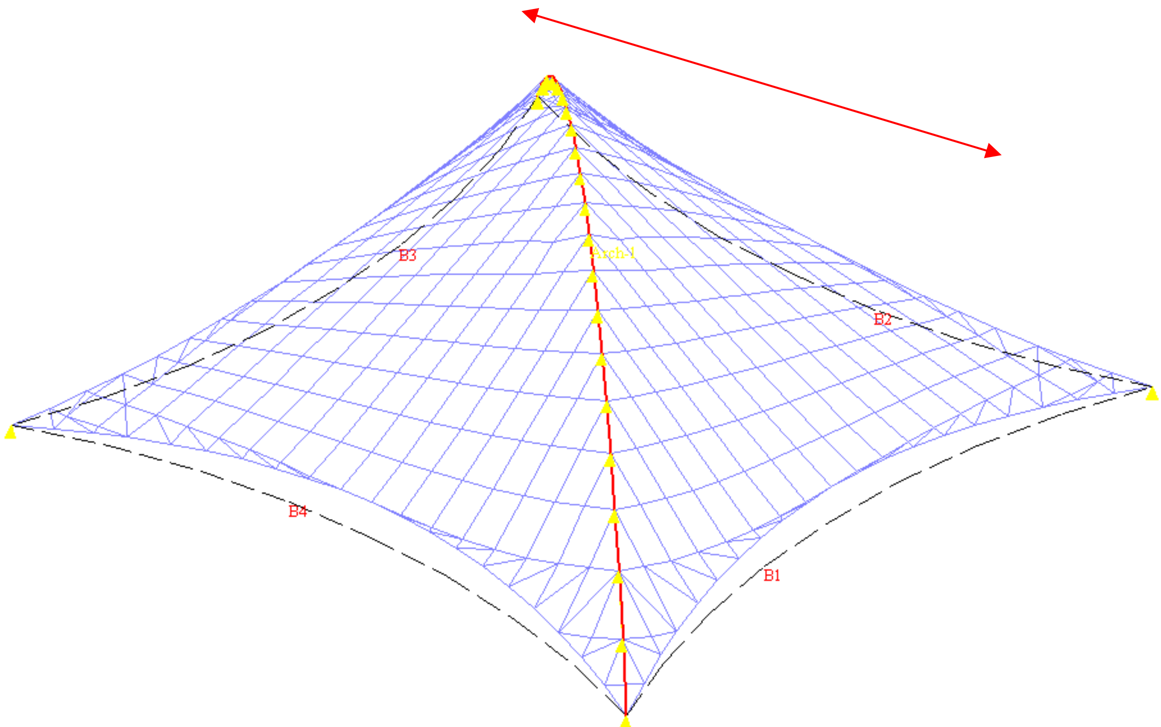
Save the model as **Internal_Arch**.



Perform form finding using Fabric mesh width of 500mm and allows the internal net point to be as close as 5mm to the external border.



Change the view of the model to NW View and display all the supports on the membrane with the **Display | Support** command.



The internal arch is represented by a series of internal supports.

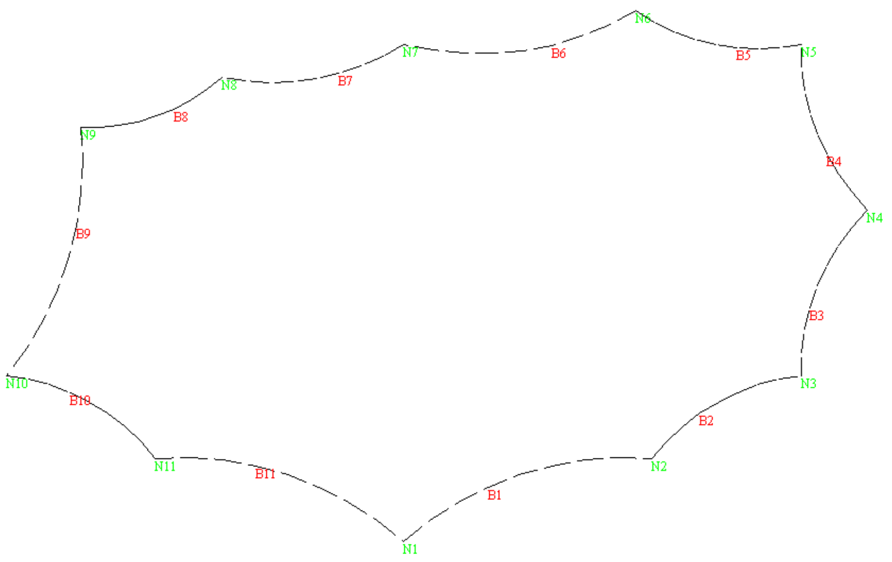
The end points of the arch are normally the system points.

13.3. Ridge and Valley Cable

Ridge and valley cables are used to form controlled membrane shape as well as providing extra stiffness against large deflection. They are very useful in large membrane structures.

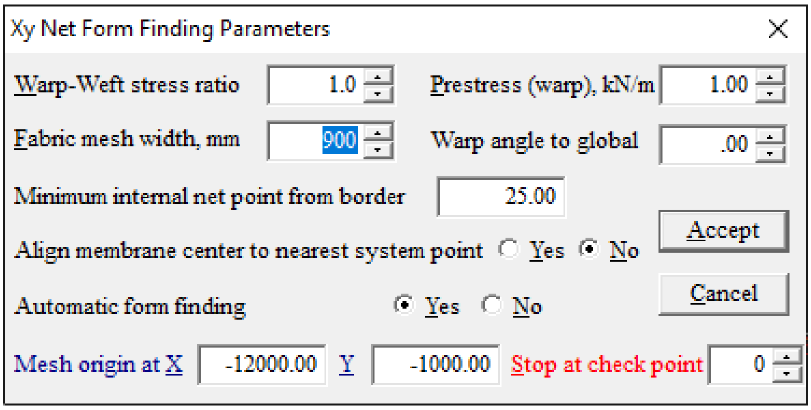
Create the following system points and form the membrane external border, all with default values. You can use **Generate | Node (more) | Cartesian** command, enter these values accordingly, then **Generate | Membrane Constructors | External Border | All | Accept** and accept default system.

Node ID	X	Y	Z
1	0.00	0.00	7500.00
2	7500.00	2500.00	2500.00
3	12000.00	5000.00	6000.00
4	14000.00	10000.00	2500.00
5	12000.00	15000.00	6000.00
6	7000.00	16000.00	2500.00
7	0.00	15000.00	7500.00
8	-5500.00	14000.00	2500.00
9	-9750.00	12500.00	6000.00
10	-12000.00	5000.00	6000.00
11	-7500.00	2500.00	2500.00

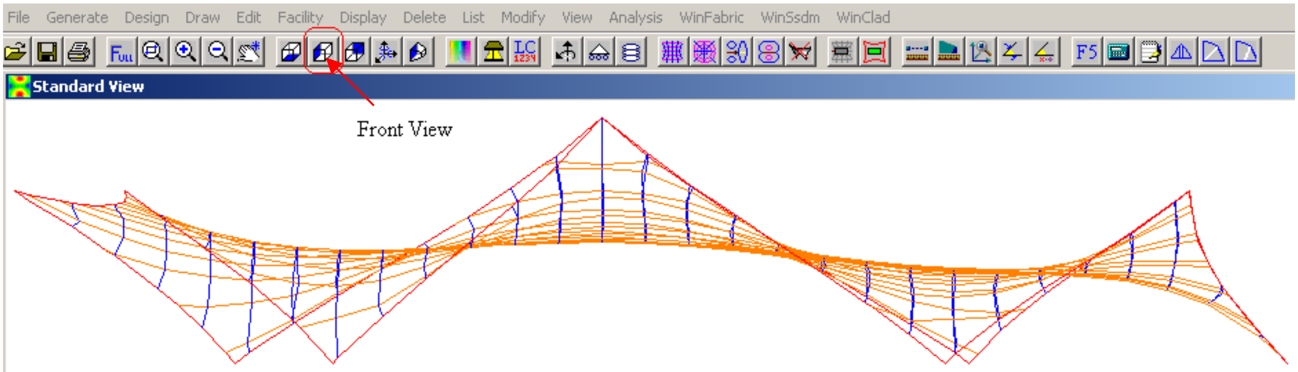


Save the model as **Valley**

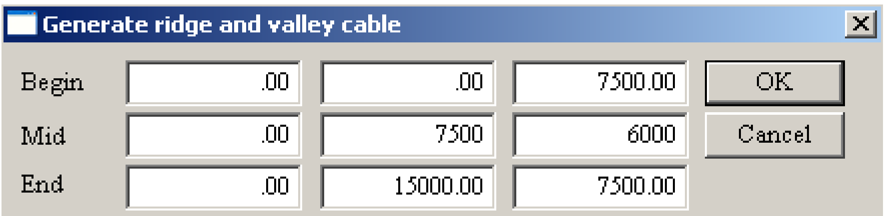
Perform form finding with fabric mesh width of 900mm using XY Net Form Finding in the quick access toolbar.

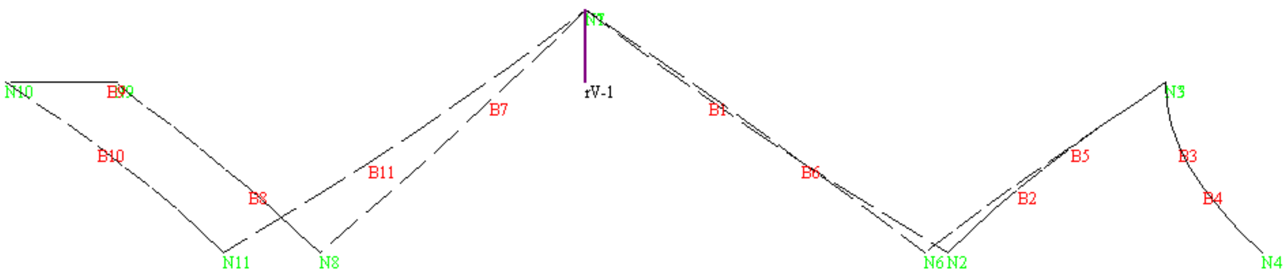


Click front view to see the prespective as shown below:

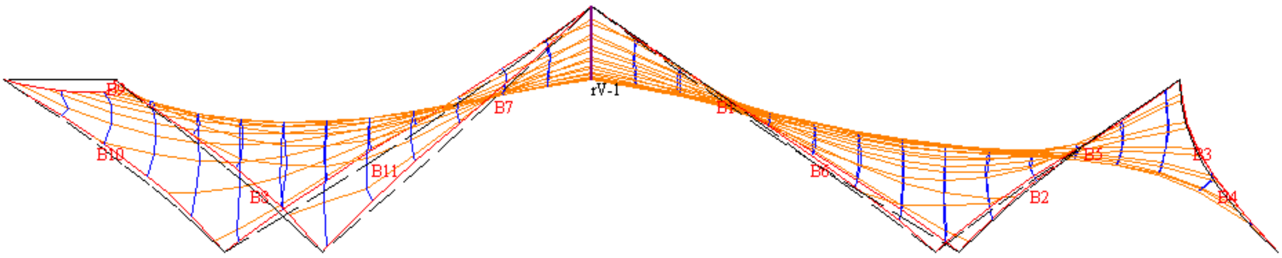


Undo the form finding. Generate a valley cable with the **Generate | Membrane Constructors | Ridge & Valley** Cable command. Input the coordinates below.





Perform form finding with fabric mesh with of 900mm.

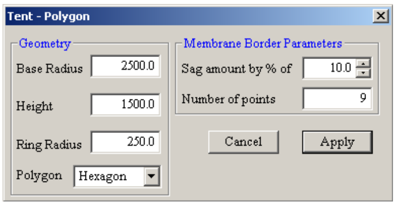


As you can see the central part of the membrane is lifted up by the valley cable.

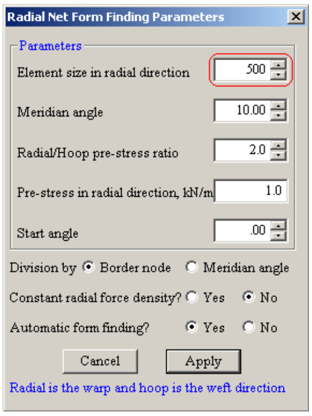
Unlike the external and internal arches, a ridge or valley cable is represented by cable elements and is not restrained.

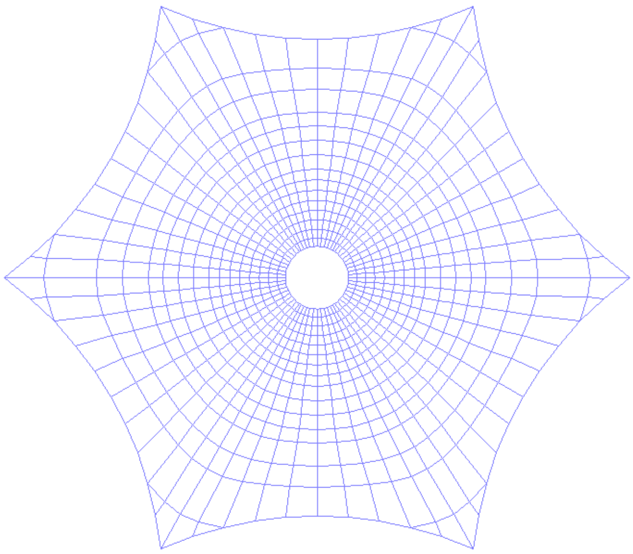
14. Restrained Border in Radial Net

Use the **Generate | Membrane Forms | Conical Tent | Polygon** to create the basic form of a polygon tent.



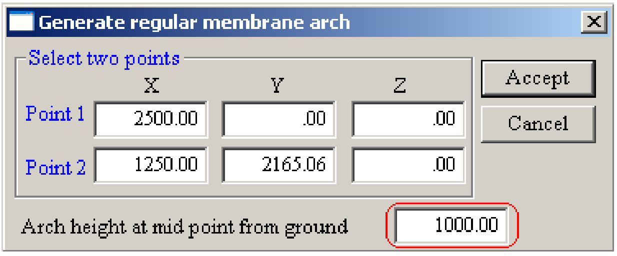
Click the **apply** button to accept the default parameter



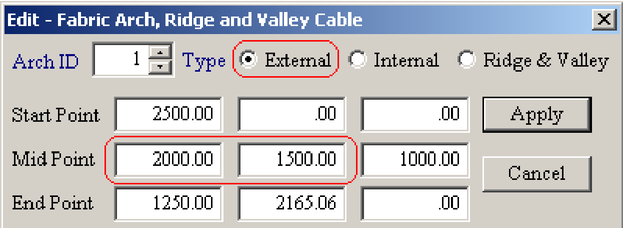


We have created a standard form which we will modify accordingly to have a model that we desired.

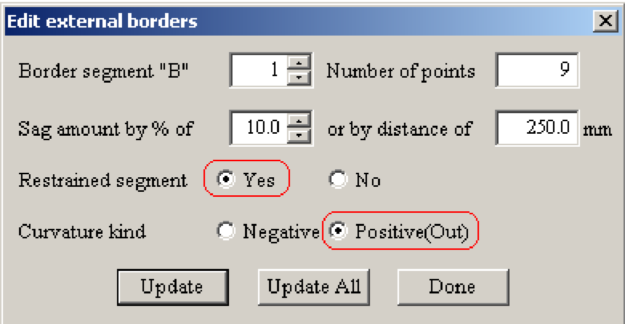
Undo form finding. Use **Generate | Membrane Constructors | Arches | Regular | 2 pts Arch** command to generate an arch at the external border defined by node ID 1.



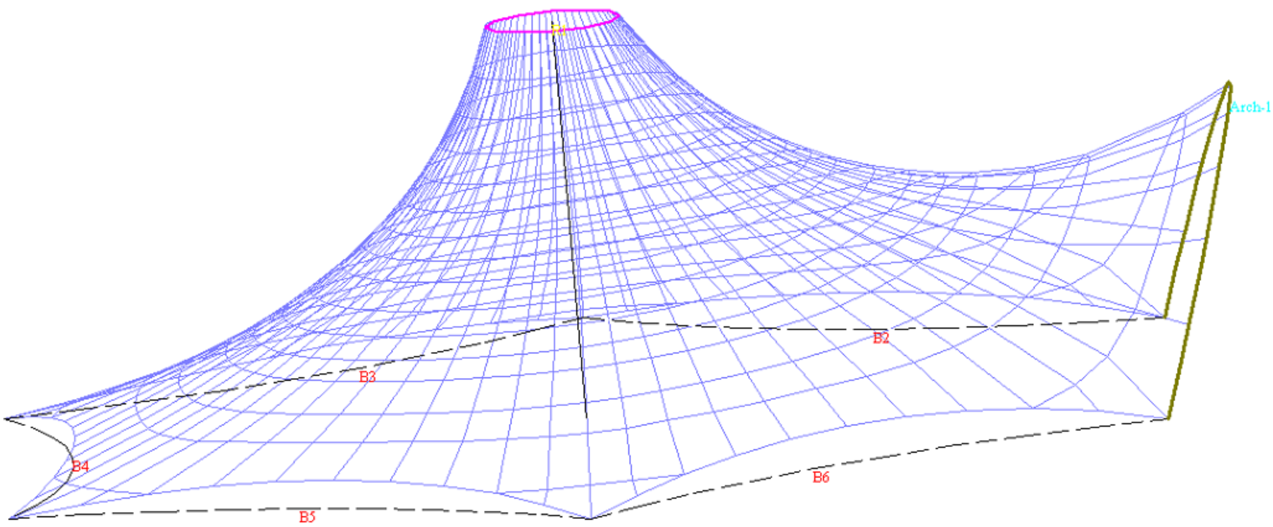
After the arch is formed, edit the arch with the **Edit | Membrane Constructors | Arch, Ridge & Valley** command.



and change the attributes of the external border to the external arch. Windas will give the external arch pretension and the external border in form finding. Use the **Edit | Membrane Constructors | External Border | Each** command.



Perform radial net form finding. Use the **Generate | Membrane Forms | Radial Net** command.

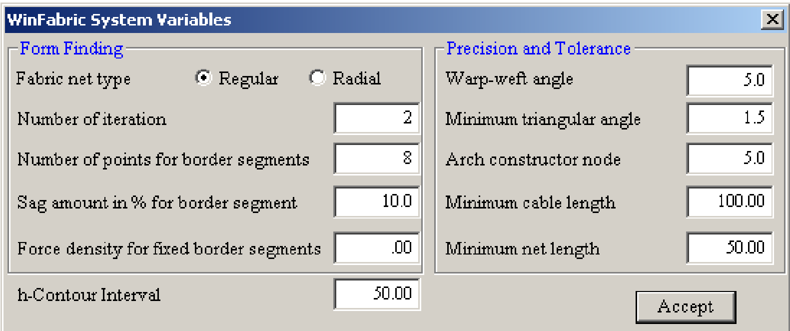


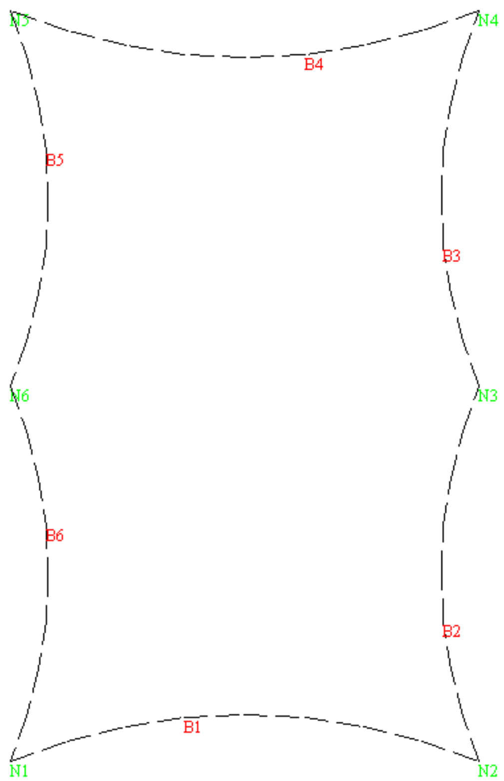
15. Restrained Border in Regular Net

First we need to create the system points. Use **Generate | Node (more) | Cartesian** command to create all the coordinates. Type in these values accordingly.

Node ID	X	Y	Z
1	0	0	0
2	2500	0	0
3	2500	2000	0
4	2500	4000	0
5	0	4000	0
6	0	2000	0

Use the **Generate | Membrane Constructors | External Borders | All | Accept** command to create the external borders.

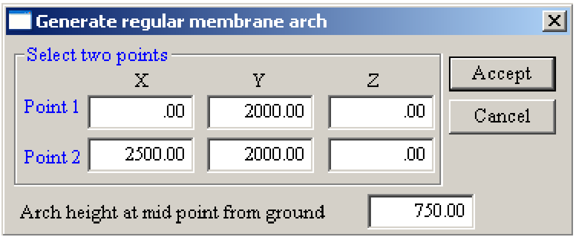


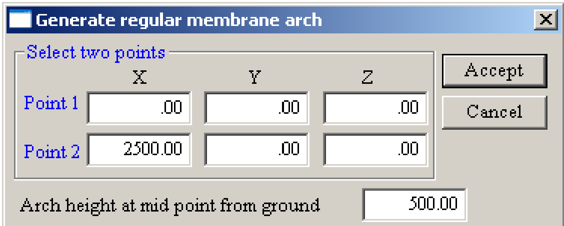


Save the model as **RB_Regular**

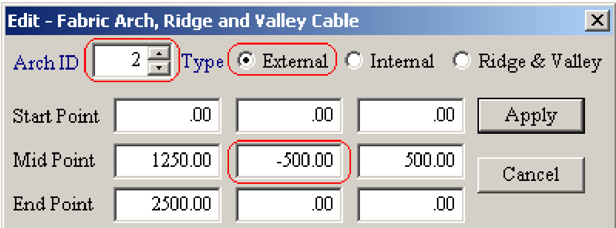
Now we need to generate the internal arch and the external arch as the restrained border. Use **Generate | Membrane Constructors | Arches | Regular | 2 pts Arch** command to generate two arches.

Key in the values below accordingly.

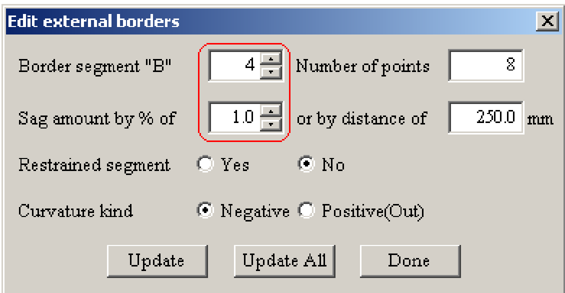




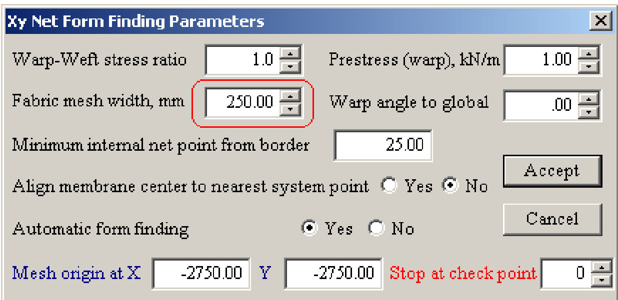
Use **Edit | Membrane Constructors | Arch, Ridge & Valley** to edit the curvature of arch ID 2.

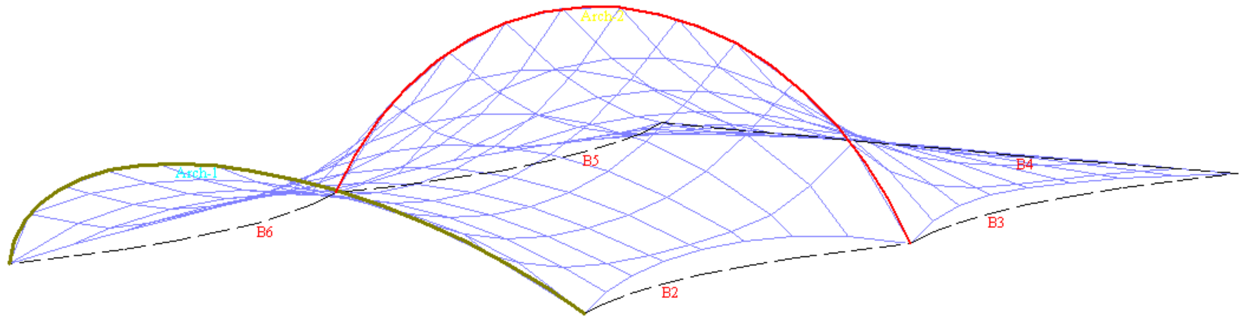


We also want to restrain the other end of the structure to a wall for an example. Use **Edit | Membrane Constructors | External Border | Each** command to edit the sag percentage of External Border 4. Click **Update** instead of Update All so that the only segment edited is Border 4.



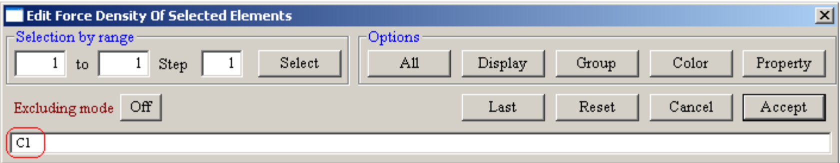
Use **Generate | Membrane Forms | Regular Net** command to establish the form finding.

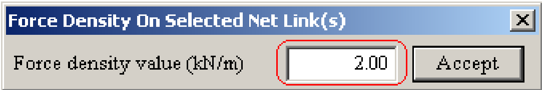




We can look from the model that the curve between the 2 arches is too deep; therefore we need to increase the force density along the warp direction.

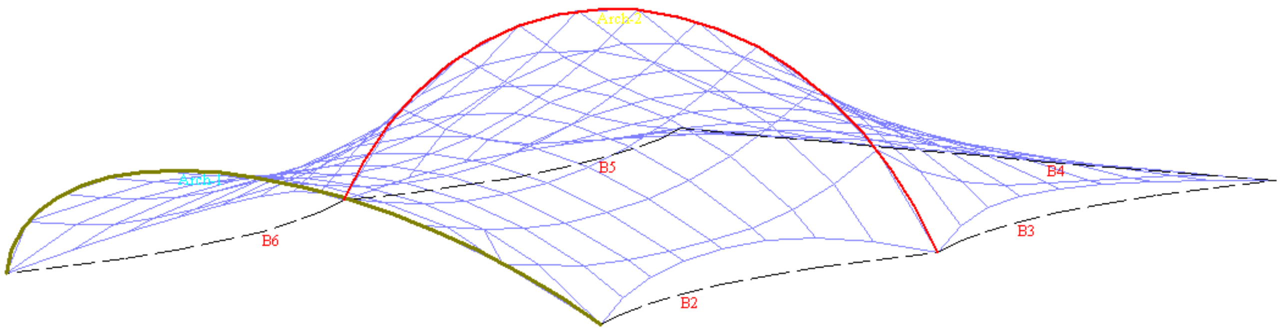
Now click **Edit | Force Density | Each** to edit the force density value along the warp direction. Recall that **warp** in Windas is represented by dark blue colour (Colour ID C1).



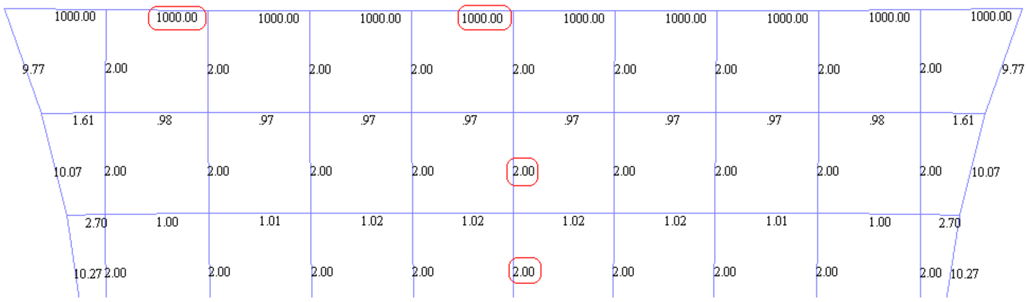


Since you want external border ID 4 butted against a wall, we need to make the membrane border as a straight line. To make the membrane border a straight line, you need to increase the force density to a larger value. Click **Edit | Force Density | Group** enter group 4 and 1000 kN/m.

Use **WinFabric | Force Density Form Finding** command to view the result after changing the value of force density.

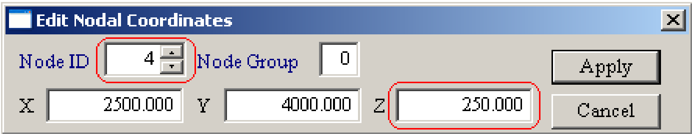


Click **Display | Load Value | Force Density** to view the force density value on the desired area.

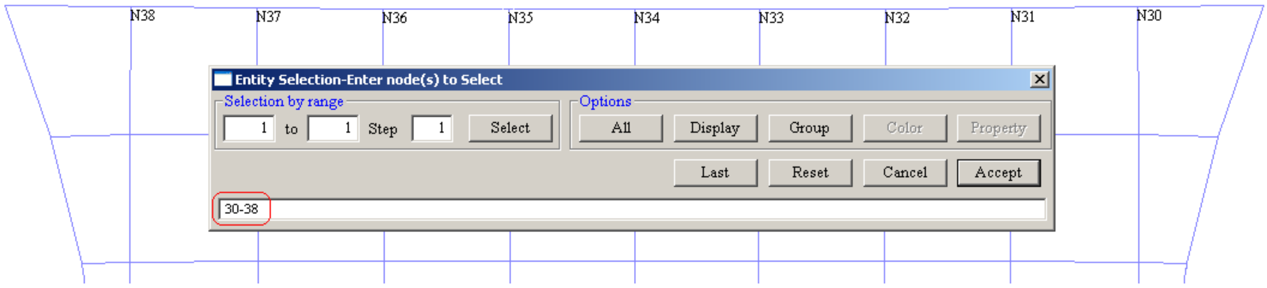


Since it is butted against a wall, therefore we need to put support along the border. We can also modify the shape by changing the nodal coordinates.

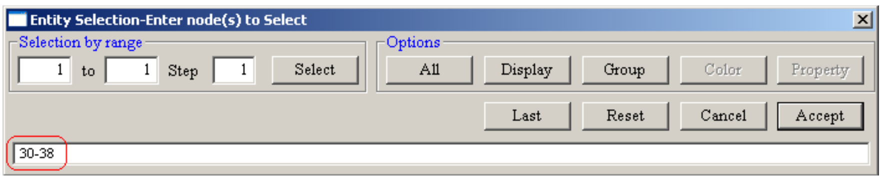
Use **Edit | Nodal Coordinates** command to modify the height of the restrained border.

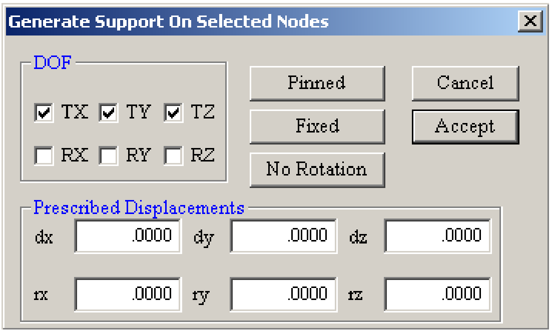


Use **Edit | Nodal XYZ | Z-Slope** to make all the nodes between the two nodal a straight line.

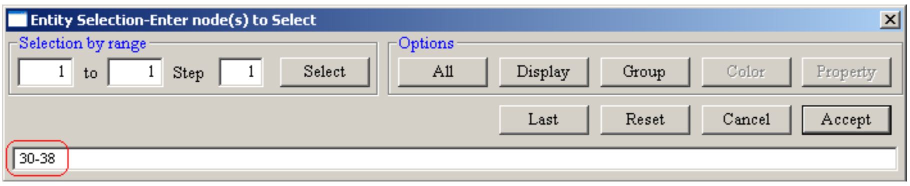


Now we need to generate supports on those nodes. Use **Generate | Support | Nodes30** command to do so. Select **Pinned** support.

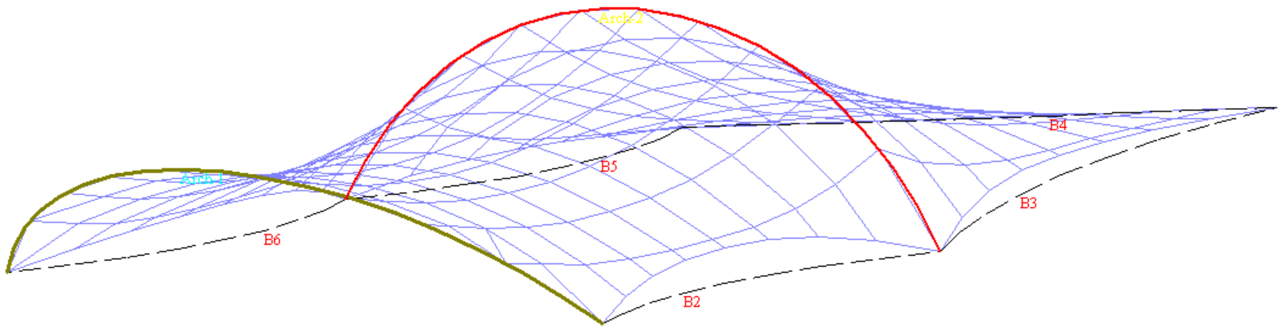


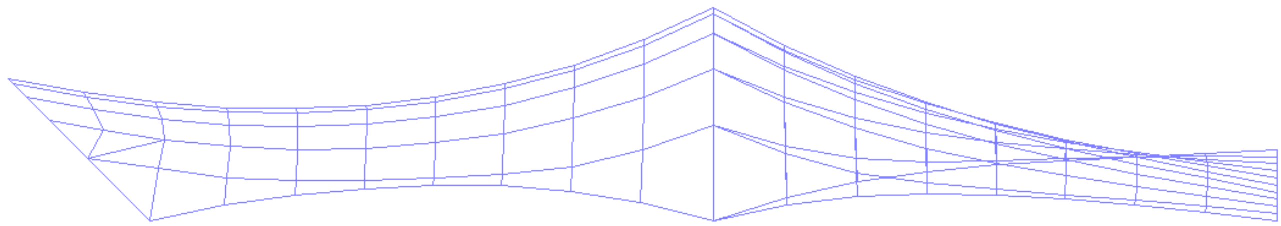


Last step, we need to make the model of the border line embedded to a wall, so we need to use **Edit | Nodal XYZ | Y** command to make the y-coordinates of the border line align to a wall.



Use **WinFabric | Force Density Form Finding** command to achieve the final form.





Practice and try to combine several methods to get various forms of restrained border.

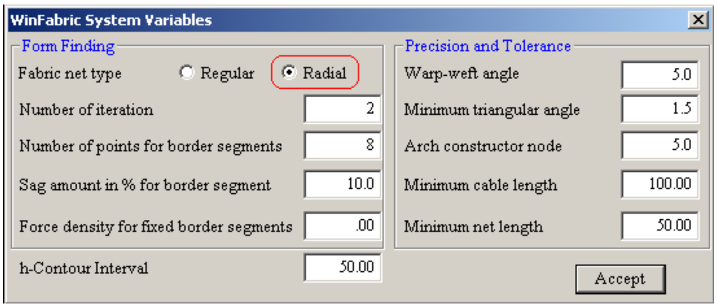
16. Rosetta Cutout

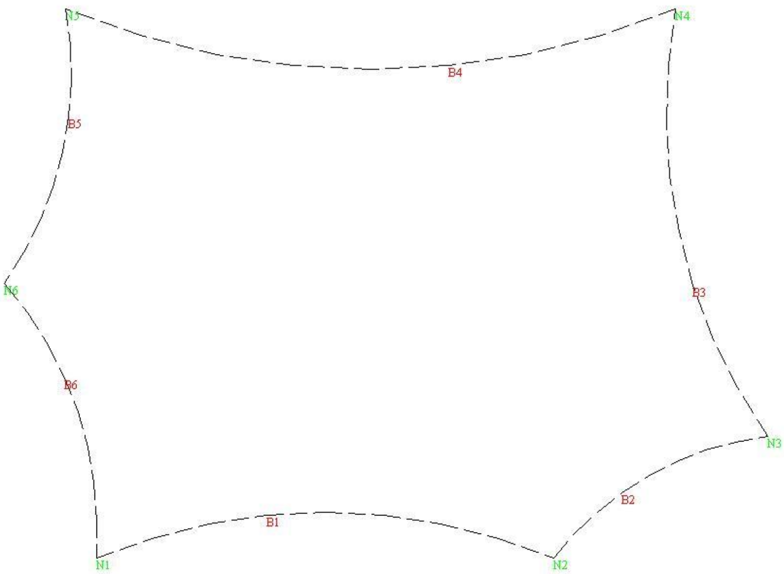
Another interesting form of a cutout is a Rosetta cutout, which its membrane is generated through radiating net. It has a unique characteristic that stress at the high point cutout is minimum.

Use the **Generate | Node (more) |Cartesian** command to create this system points.

Node ID	X	Y	Z
1	0	0	0
2	7500	0	4000
3	11000	2000	0
4	9500	9000	-6000
5	-500	9000	0
6	-1500	4500	4000

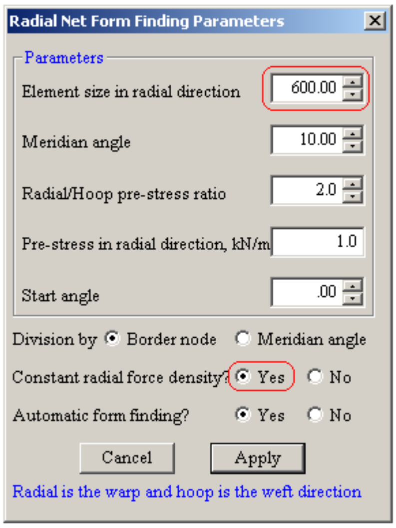
Use the **Generate | Membrane Constructors | External Borders** command; select the system points in anti-clockwise order to define the external border. Simply click **All | Accept**. Choose **Radial** border.

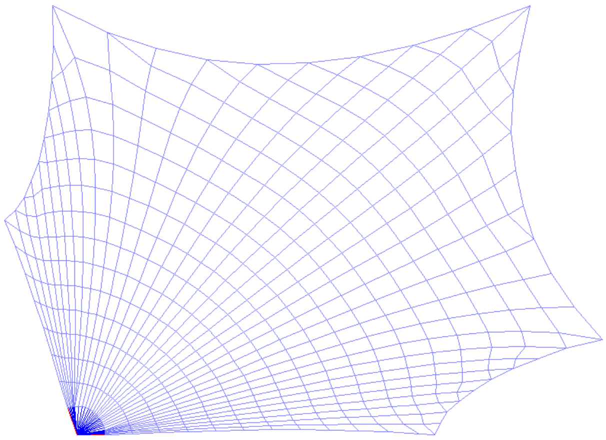


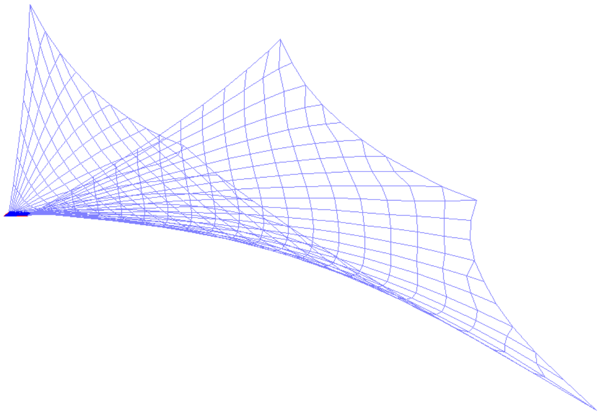


Save the model as **Rosetta.**

Perform form finding using the **Generate | Membrane Forms | Radiating Net** with default values.

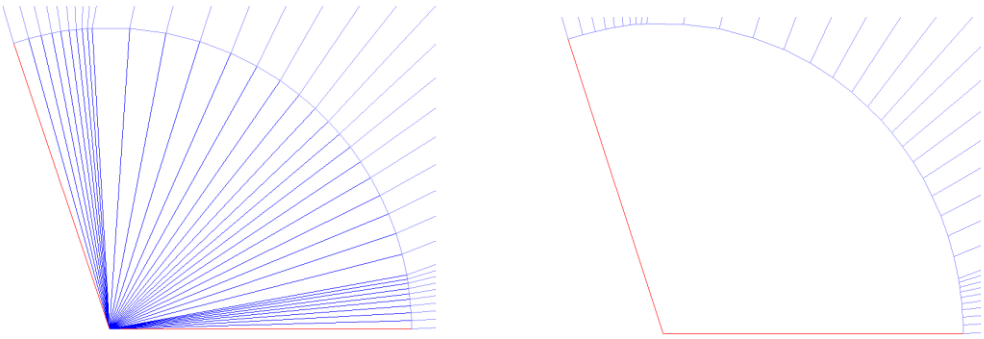






Now, we will proceed to generate the Rosetta cutout.

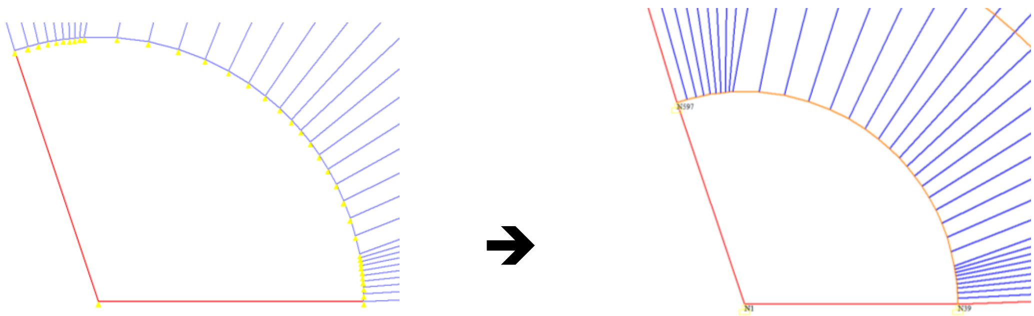
After we are done the form finding, we need to make a hole on the first weft of the radiating net centre. Refer to the picture below to see where the hole is supposed to be. First of all we need to delete the force density on the elements that we are going to remove. Using **Delete | Load | Force Density** command and choose all the elements in dark blue color to remove the force density on those elements. Recall that the dark blue (color code : C1) are the warp of the structure. Use **Delete | Element** command to remove all the elements that are *located inside the designated hole* which are in dark blue color.



After we remove all the elements that we do not need, we also need to remove all the supports around the hole, so the cable could just hang on the end-support. Note that we still need the support at the end of this particular weft. Refer to the picture below.

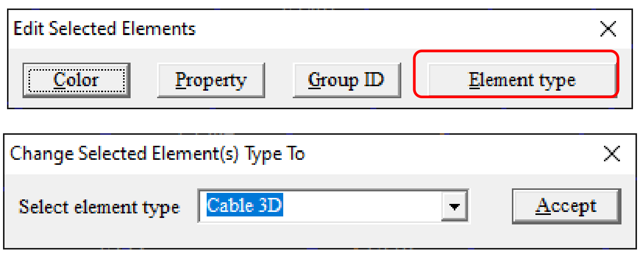
Use **Delete | Support** command and pick all the supports that we are going to delete to remove all the supports around the hole.

Note : to help you choose the correct supports, **right click | label | support**. This label can also be used for elements name, node, etc.

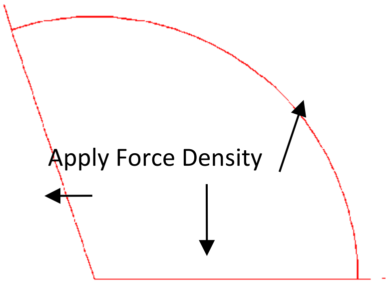


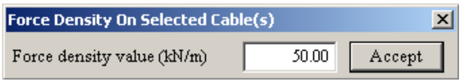
Next step is to change the element type around the curvature of the hole into a cable type element. Use **Edit | Element Attributes** then carefully choose the designated curve elements. You can always turn on the element label command to help you choose the elements.

Choose **Element type | Cable 3D.**

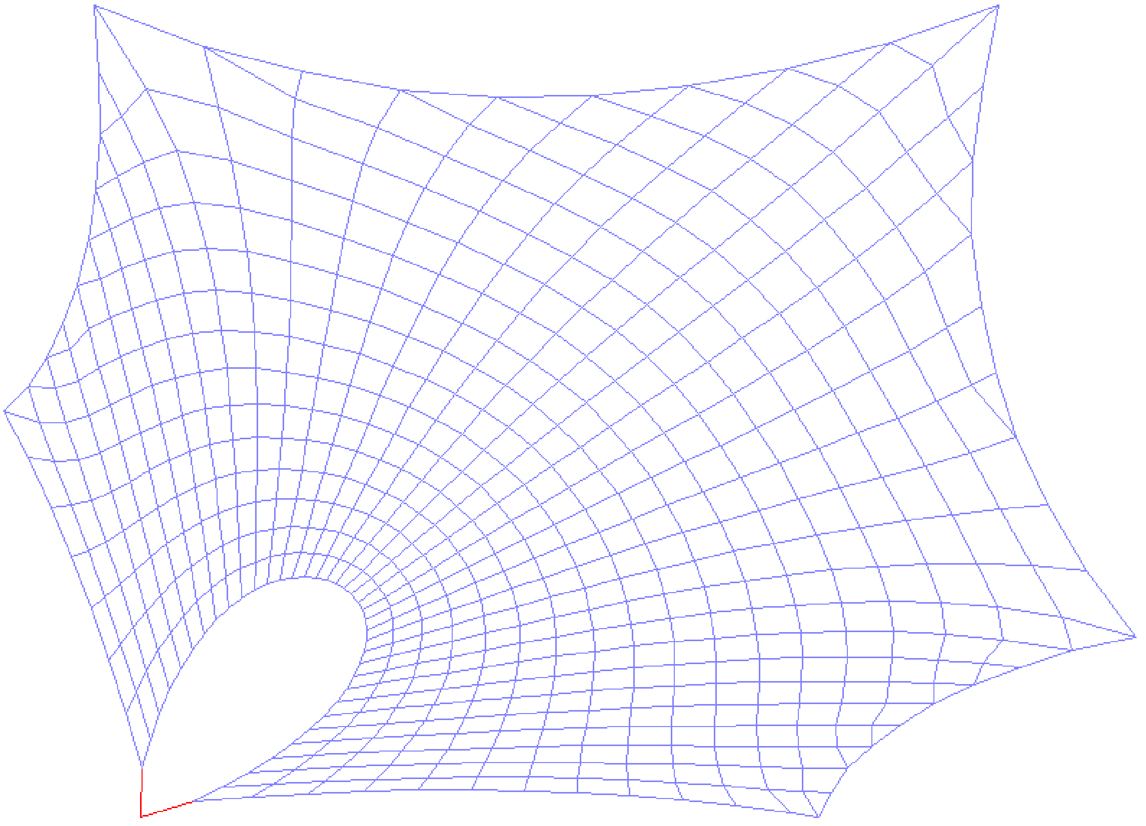


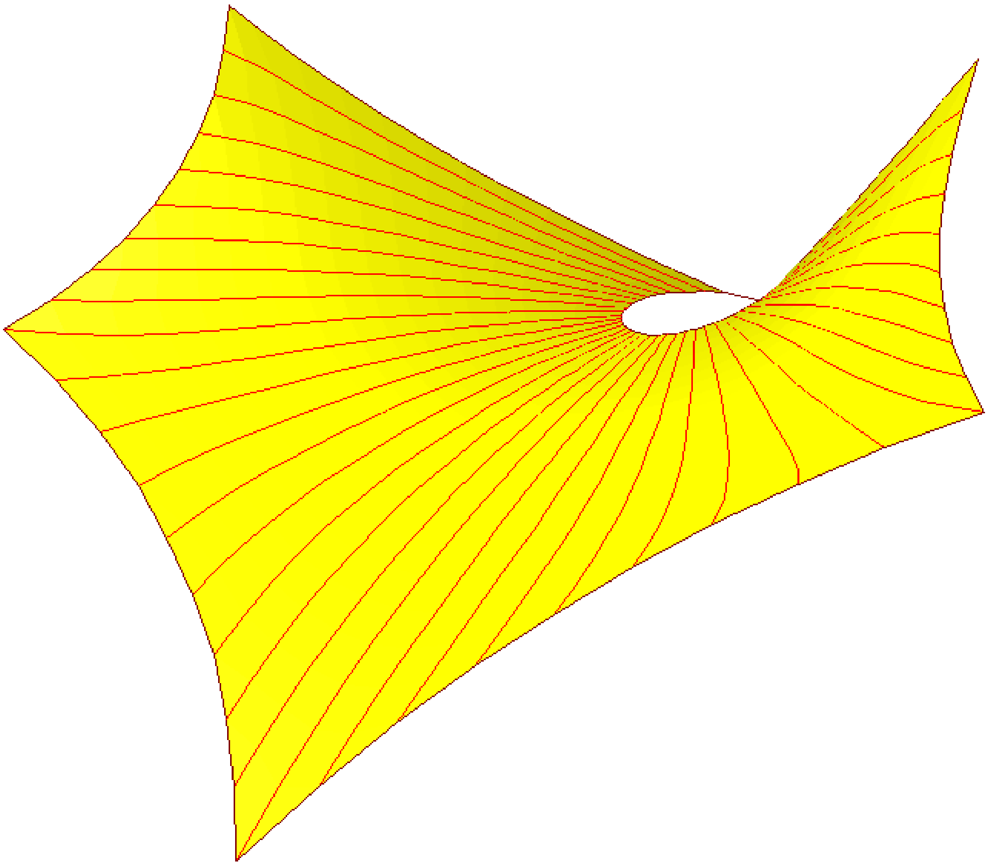
Now after the hole and the cable are formed, we need to apply force density into the cable around the hole. Use **Generate | Load | Force Density | Cable** and choose the curve elements surrounding the hole.

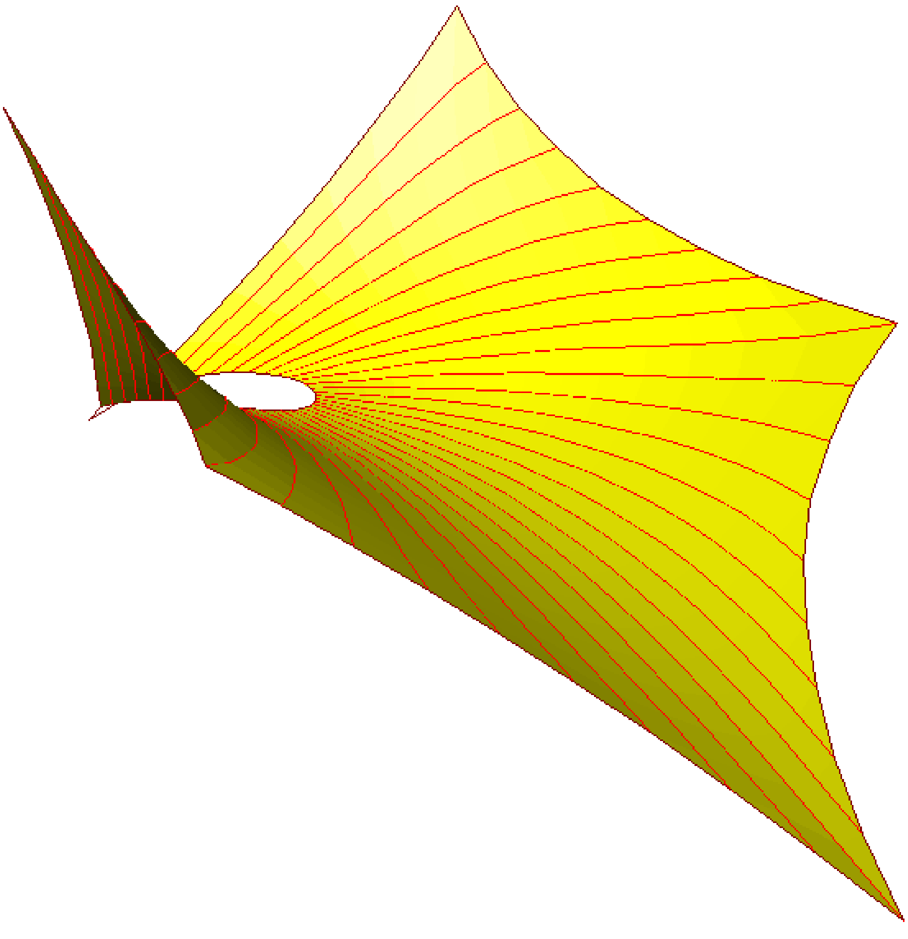




Use **WinFabric | Force Density Form Finding** command to get the final form of Rosetta cutout.



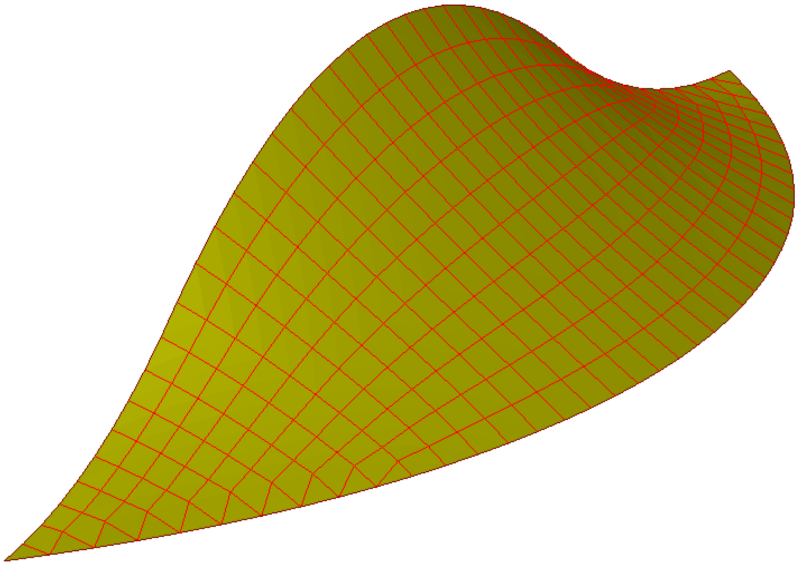




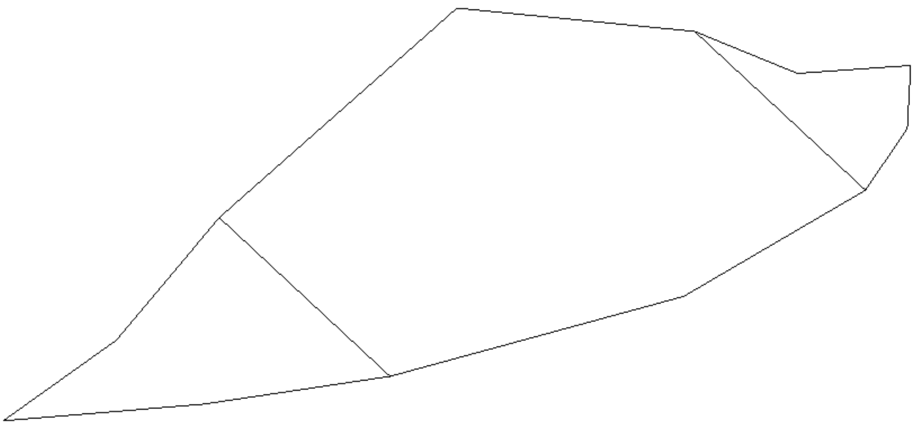
17. Multi-cell Cushions Form Finding

17.1. Standard Regular Forms / Random Shape Form

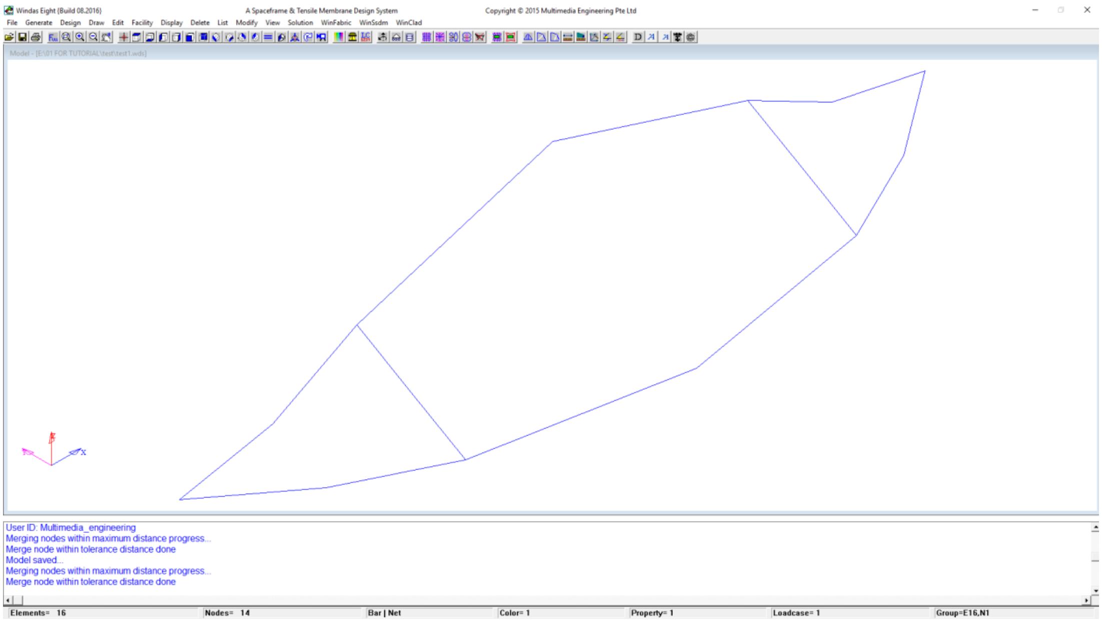
The example of membrane shape we want to achieve is shown below;

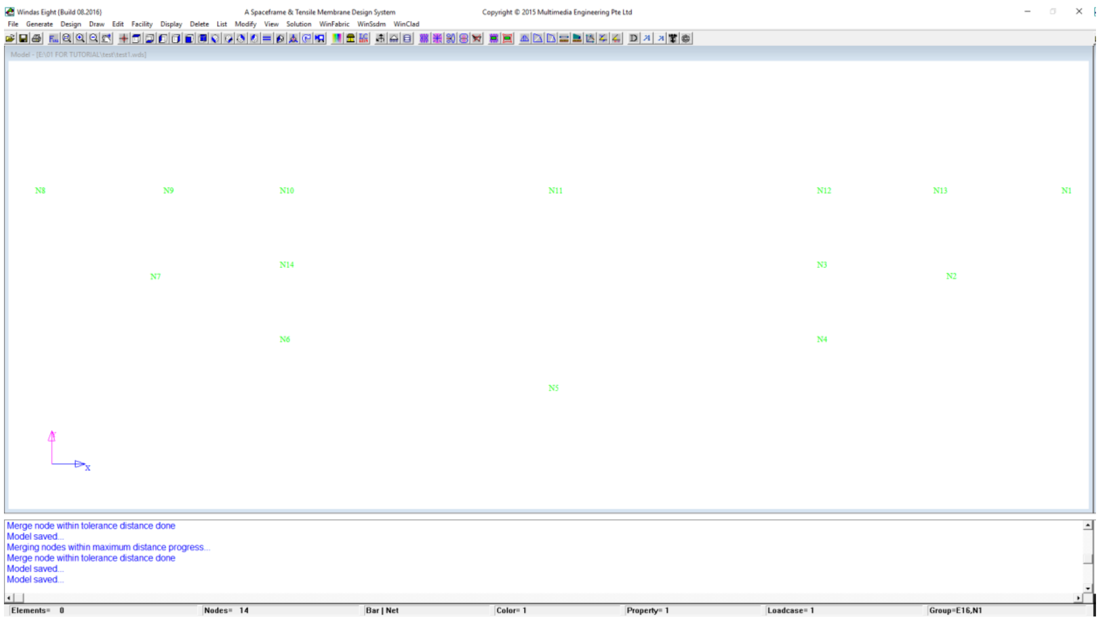


First, we need to take some reference points and draw the line to connect the reference point in AutoCAD or other software and save it as (.dxf) file. We just need to draw the perimeter as a straight line to facilitate us to create this model in Windas. For this kind of shape we can draw the perimeter as shown below;

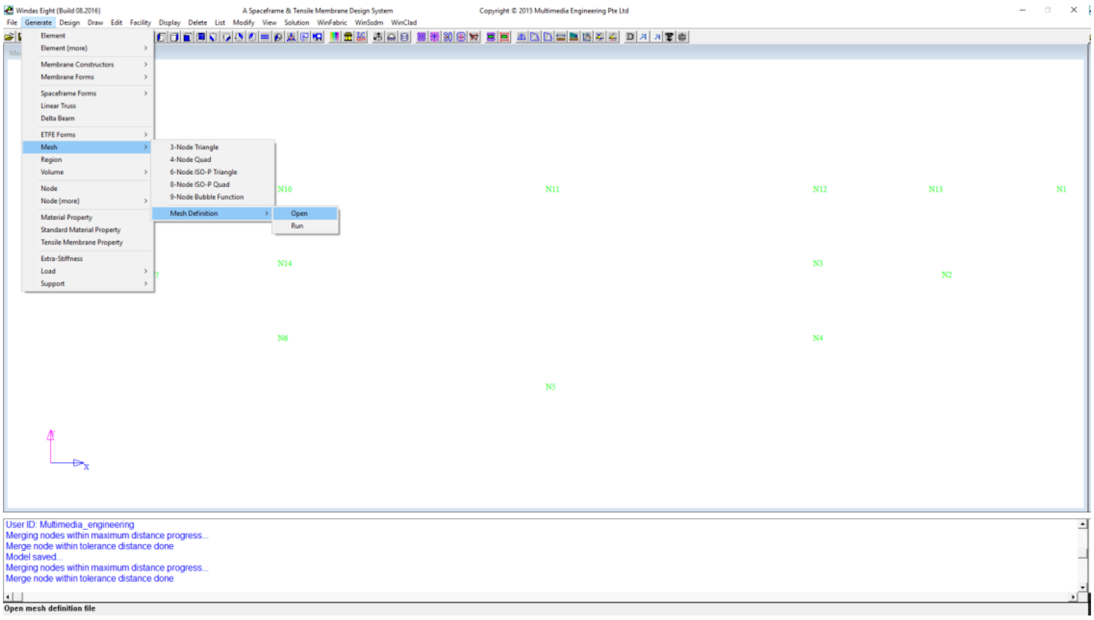


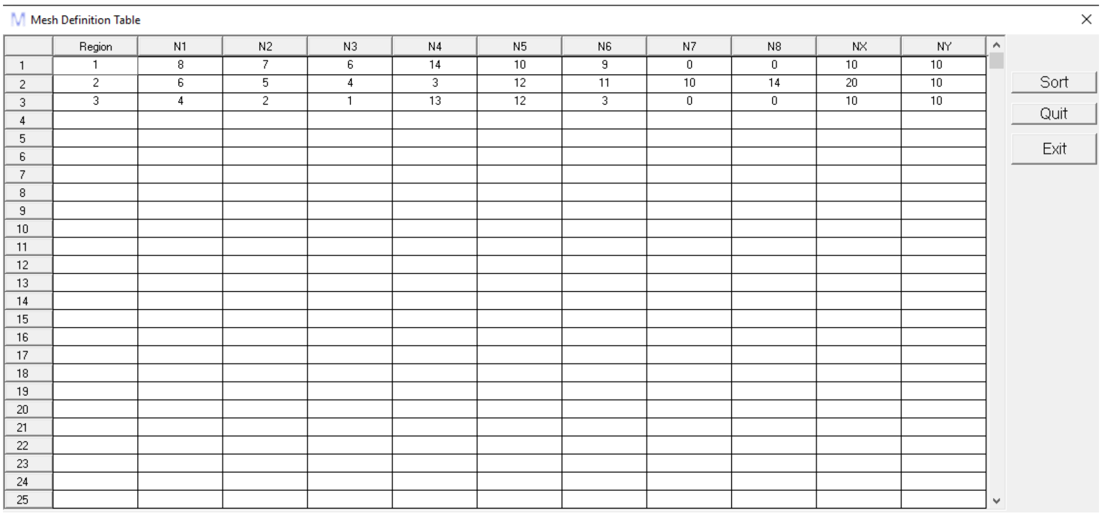
After that we can import the (.dxf) file to Windas. Then we need to delete the line we made before so only the node point remain in our Windas model.





Now we can use the "Mesh" command to create the membrane model. Click on **Generate | Mesh | Mesh Defination | Open.** Then a new window will appear as shown below. In the new window you can see **Region, N1….N8, NX,** and **NY** column. As we see there are only 8 nodes that can be connected in one region so we divided the membrane into several regions. For this case, we divide the membrane into 3 region. Then fill the N1 to N8 column with the node number. (**Note: we need to fill the node number in anticlockwise sequence.** See the example below). The NX and NY column is to define how many divided area in X and Y direction we need for our membrane model.

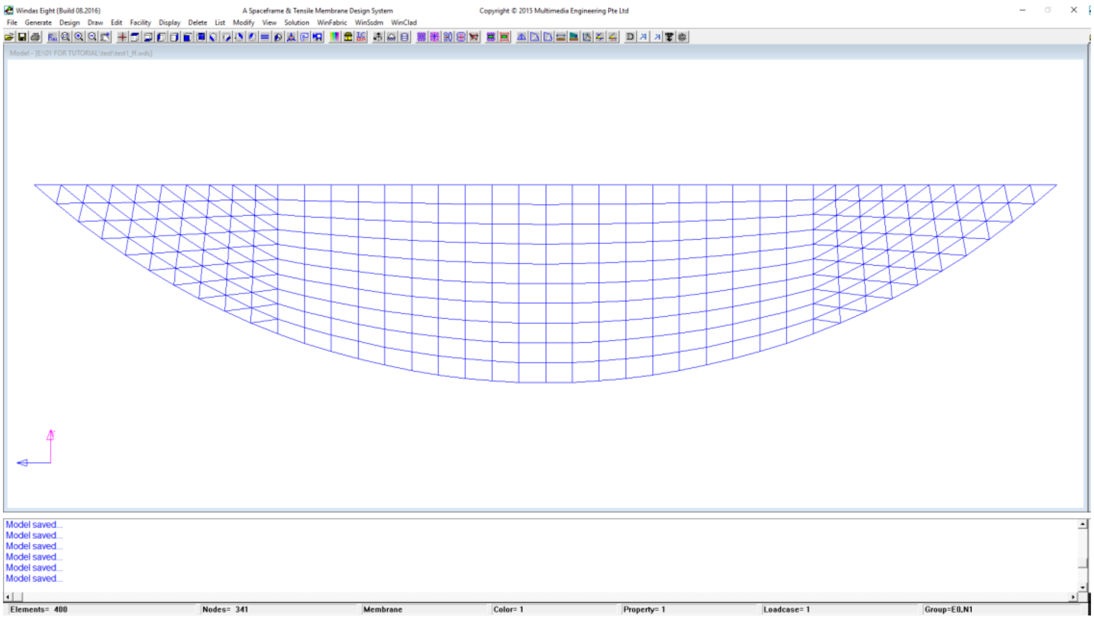



Mesh Definition Table

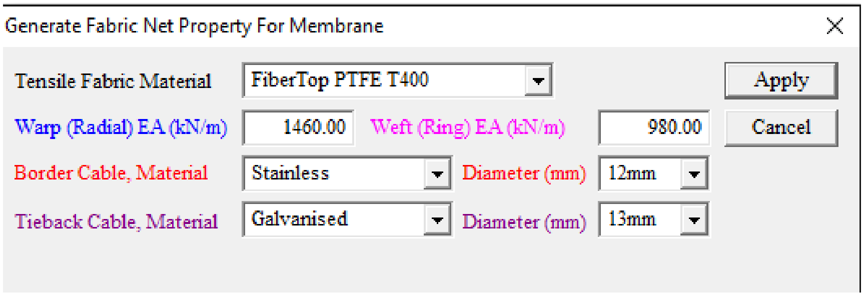
	Region	N1	N2	N3	N4	N5	N6	N7	N8	NX	NY
1	1	8	7	6	14	10	9	0	0	10	10
2	2	6	5	4	3	12	11	10	14	20	10
3	3	4	2	1	13	12	3	0	0	10	10
4											
5											
6											
7											
8											
9											
10											
11											
12											
13											
14											
15											
16											
17											
18											
19											
20											
21											
22											
23											
24											
25											

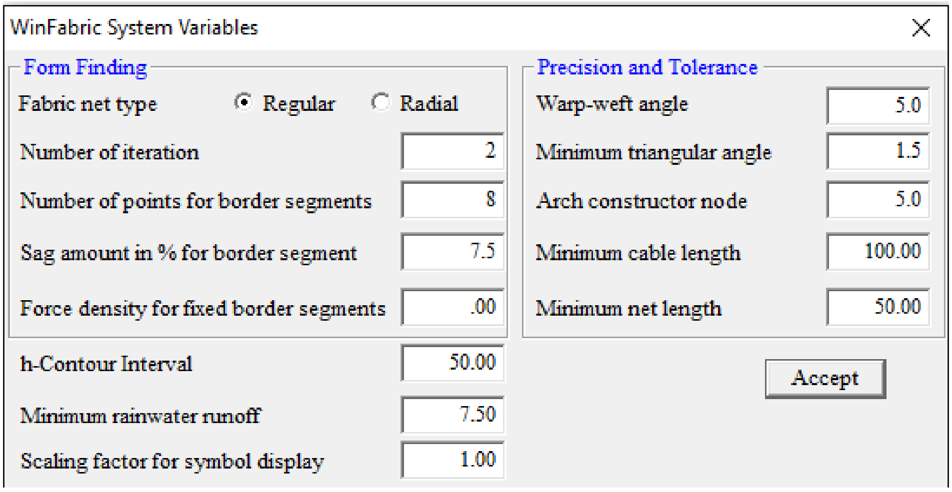
Sort
Quit
Exit

Save the file and also **save as " filename_ff "** so that later we can open the file before we did the form finding. After "save as" you can see that the mesh definition we made before is still there. Now click on the **Generate | Mesh | Mesh Defination | Run.** Windas will automatically create the membrane shape as shown below;

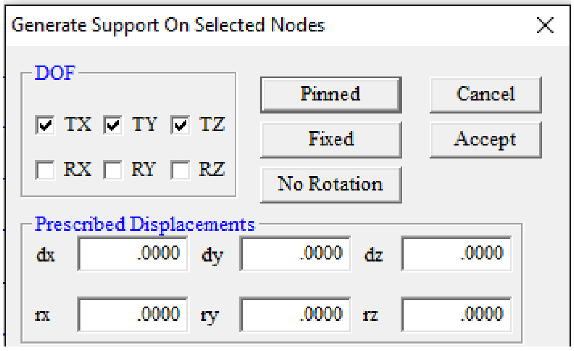


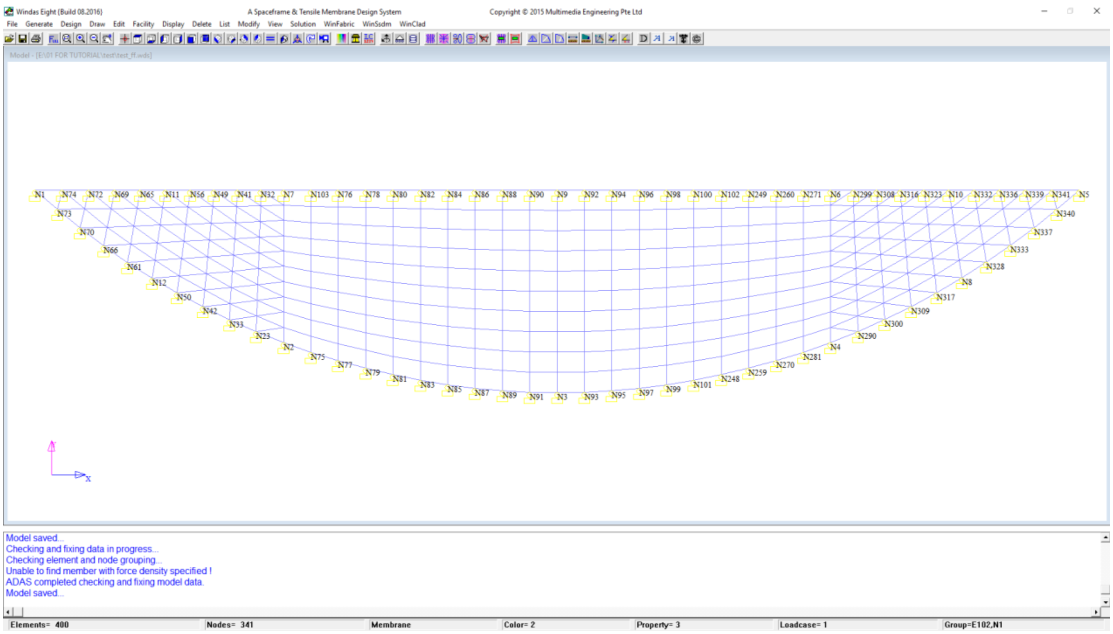
Now we need to change the color of membrane surface to **C2** and the property to **P3**. We can change the color and the property using **Edit | Element Attributes.** Then apply the tensile membrane property and also the support. First, apply the tensile membrane property by click **Generate | Tensile Membrane Property.** For this example we PTFE as the tensile membrane property.



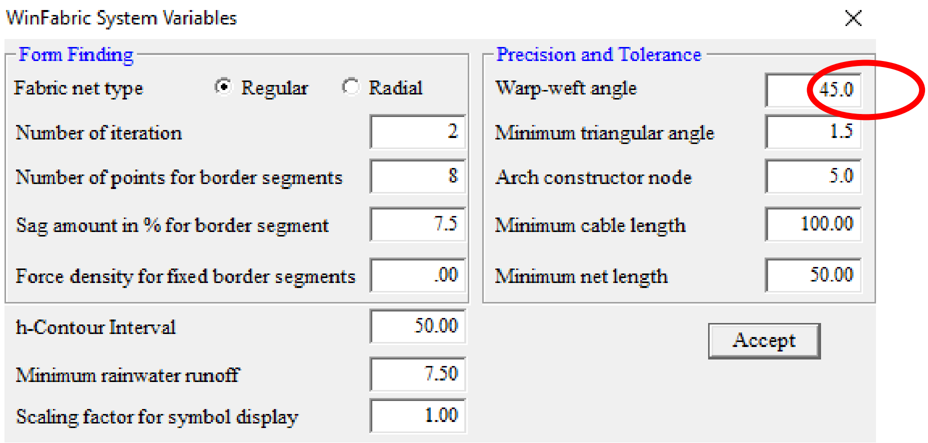


After that apply the support. Since we want the loads from membrane later will be transferred to the steel along the membrane perimeter, therefore we need to apply the support at membrane perimeter. Use **Generate | Support | Perimeter** and choose **Pinned**

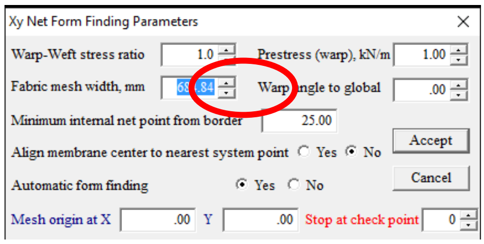




Now, before we apply the surface net, we need to set the fabric first. As we see that there are some parts with triangle mesh so we need to adjust the warp-weft angle to 45°. Use **Facility | Set | Fabric** to adjust the angle as shown below;



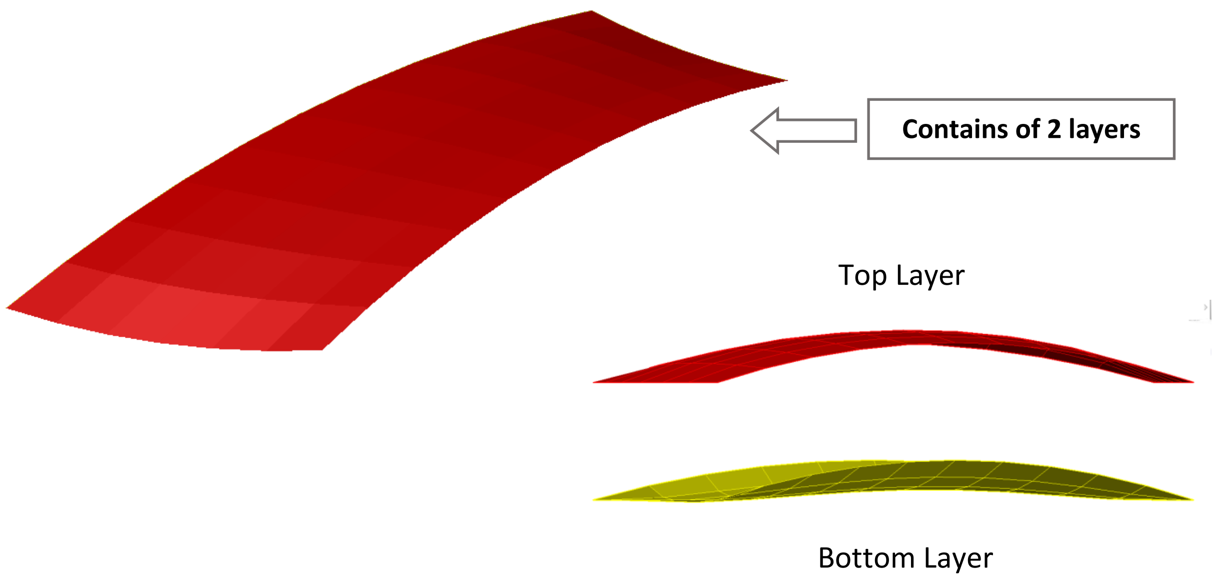
Then use **WinFabric | Mesh→FD Model | Surface→Net** to apply the warp and weft net. Now we need to generate the border cable to our membrane model. Use **WinFabric | Mesh→FD Model | Border Cable** then the red line along the perimeter will appear (click if it doesn't appear) which is indicate as border cable. In addition we need to update the force density using **Edit | Force Density | Update | XY net.** After do all the step mentioned above the tensile membrane model is ready for analysis. However we still can adjust the mesh size using **WinFabric | Mesh→FD Model | Mesh Net.** In the XY Net Form Finding Parameters window change the fabric mesh width as show below.



After finish the modelling we can continue with **Materialization** and **Triangulation for load analysis preparation** as mentioned in other chapter in this tutorial.

17.2. ETFE Single Cushion

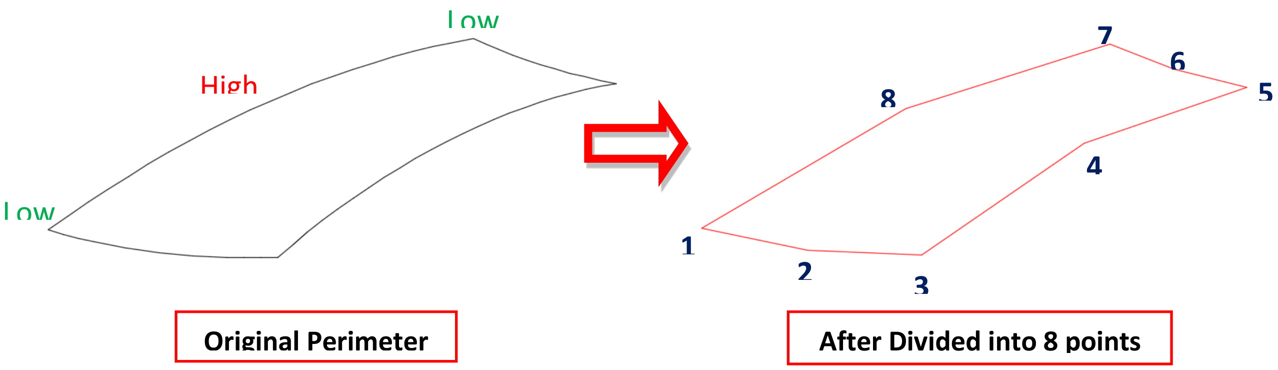
The example of ETFE Single Cushion shape we want to achieve is shown below;



First, we need to take 8 reference points at the perimeter and draw the line to connect all reference points in AutoCAD or other software and save it as (.dxf) file.

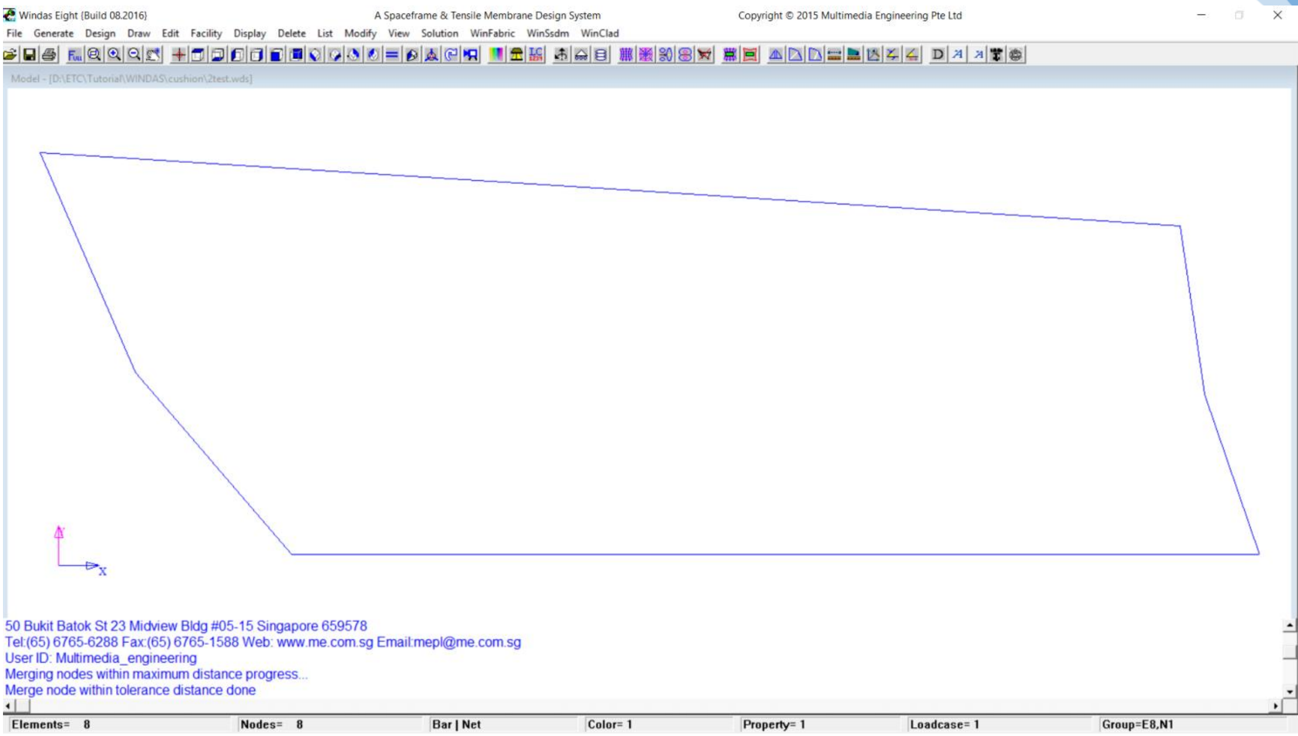
If there is an Arc, we need to convert it into straight line by dividing it into 3 reference points (Low point → High Point → Low Point) and connect these 3 points using 2 lines.

For this kind of shape we can draw the perimeter as shown below;

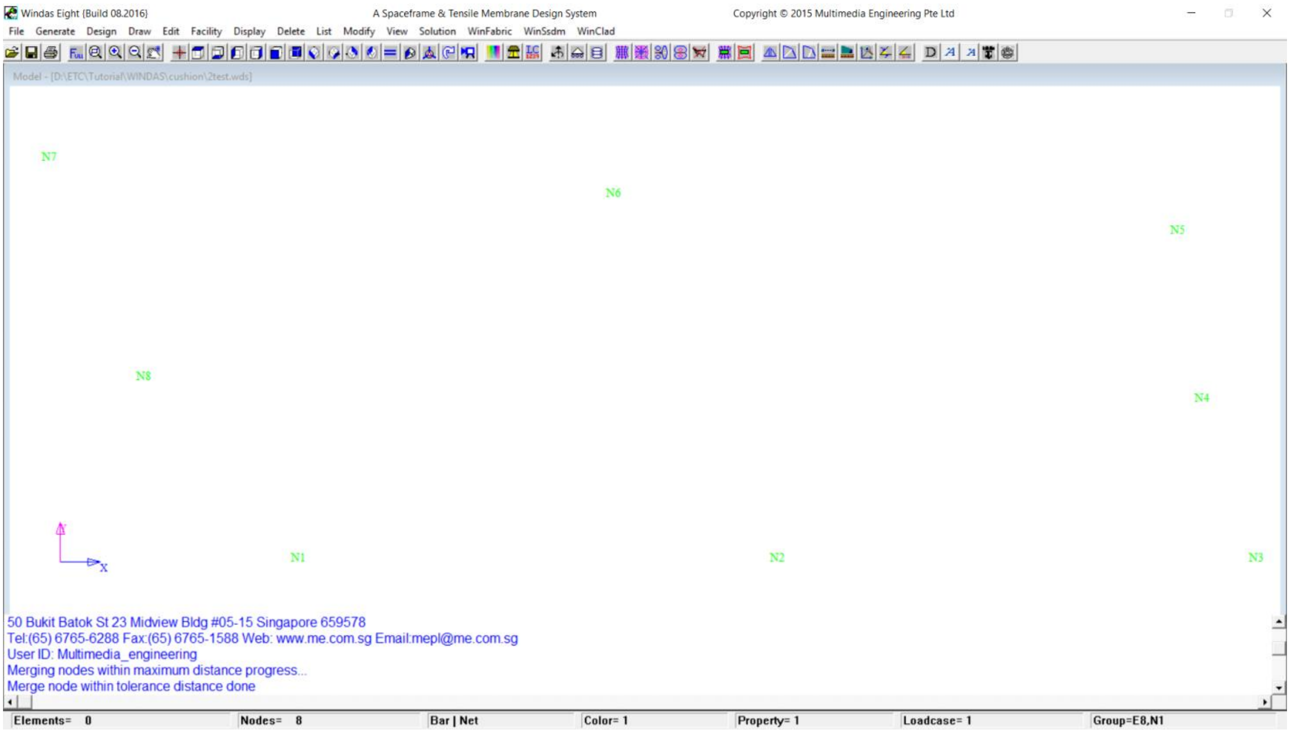


After that we can import the (.dxf) file to Windas. Then we need to delete the line we made before so only the node point remain in our Windas model.

To Delete the line, Click on **Delete | Element | Display .**



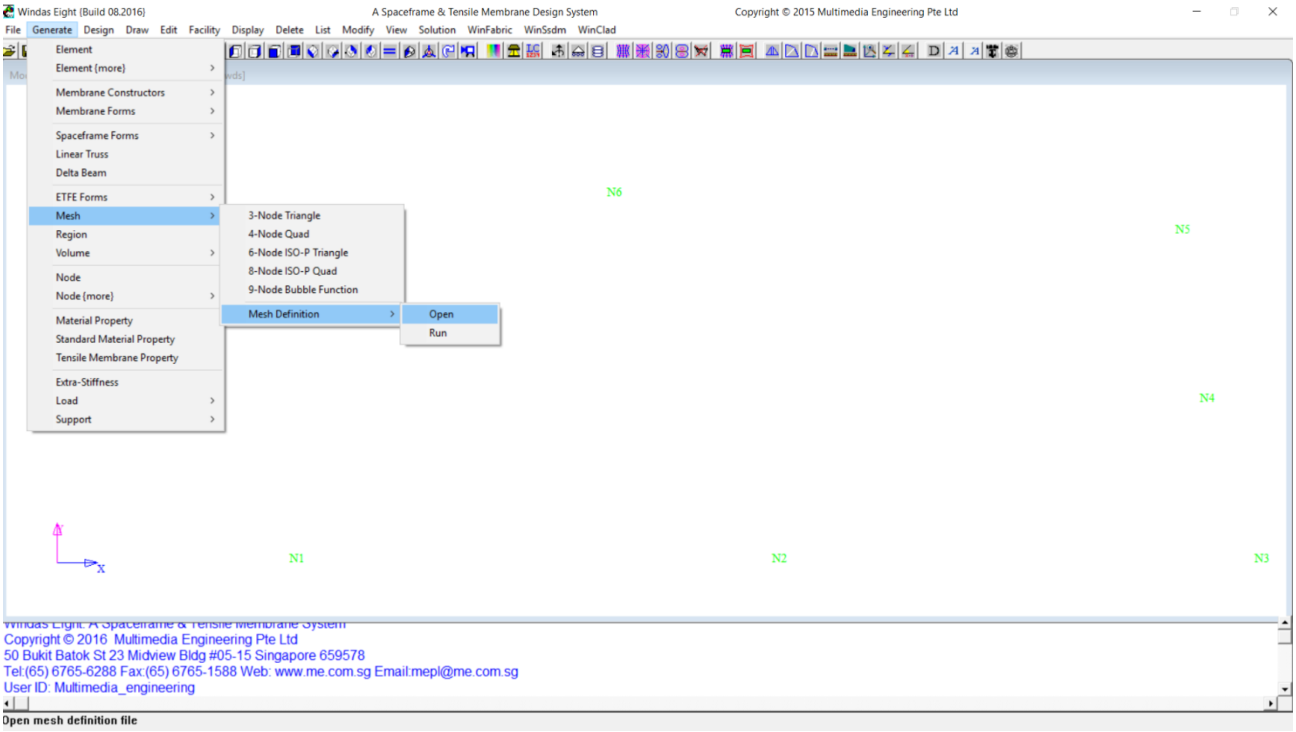
Then our Reference Lines will be deleted and only 8 nodes left in our workspace. We can reorder the nodes' sequence by clicking **Facility | Reorder | Node Reorder | Anticlockwise.**



Save the file and also **save as " filename_ff "** so that later we can open the file before we did the form finding. After "save as" you can see that the mesh definition we made before is still there.

Now we are ready to use the “**Mesh**” command to create the ETFE model.

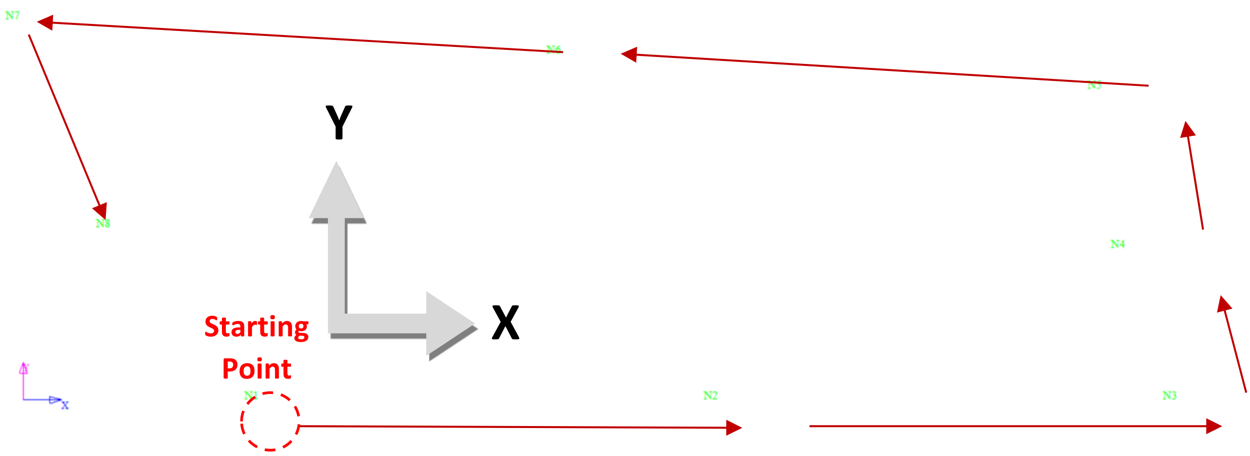
Click on **Generate | Mesh | Mesh Definition | Open.**



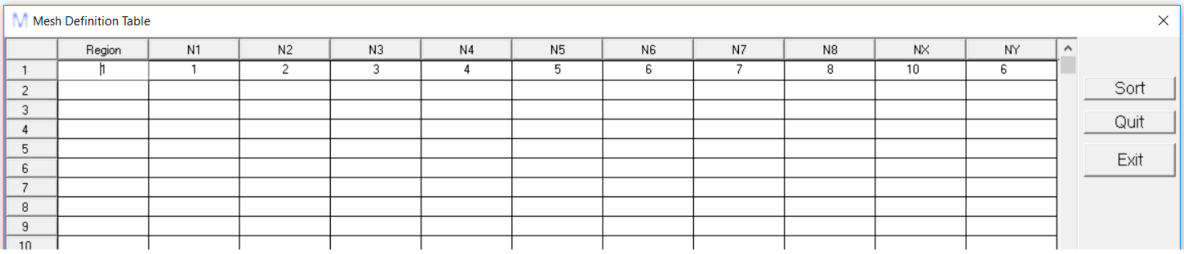
Then a new window will appear as shown below.

In the new window you can see **Region, N1….N8, NX,** and **NY** column. As we see there are only 8 nodes that can be connected in one region. In this case, 1 region is enough for us to make the mesh, but, in more complex shape we may need more than one region.

Then fill the N1 to N8 column with the node number. (**Note: we need to fill the node number in anticlockwise sequence.** See the example below).



Our X and Y Direction depend on our starting point, for this example, our X direction will be at the horizontal direction and our Y direction will be at the vertical direction.

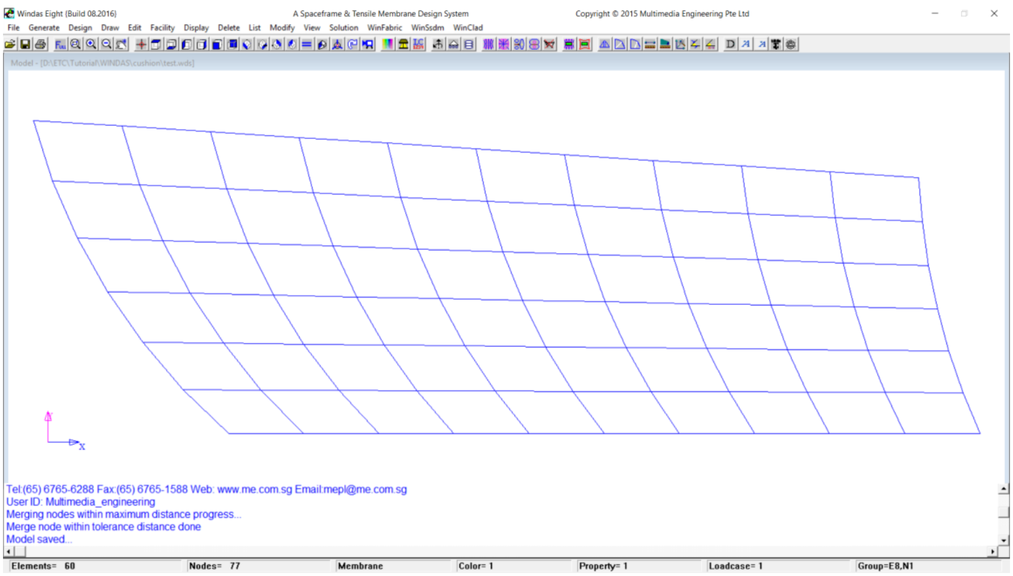

Mesh Definition Table

	Region	N1	N2	N3	N4	N5	N6	N7	N8	NX	NY
1	1	1	2	3	4	5	6	7	8	10	6
2											
3											
4											
5											
6											
7											
8											
9											
10											

The NX and NY column is to define how many divided area in X and Y direction we need for our membrane model. To Close the window, Click **Sort | Exit.**

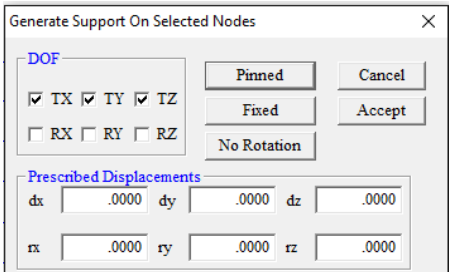
Now click on the **Generate | Mesh | Mesh Defination | Run.**

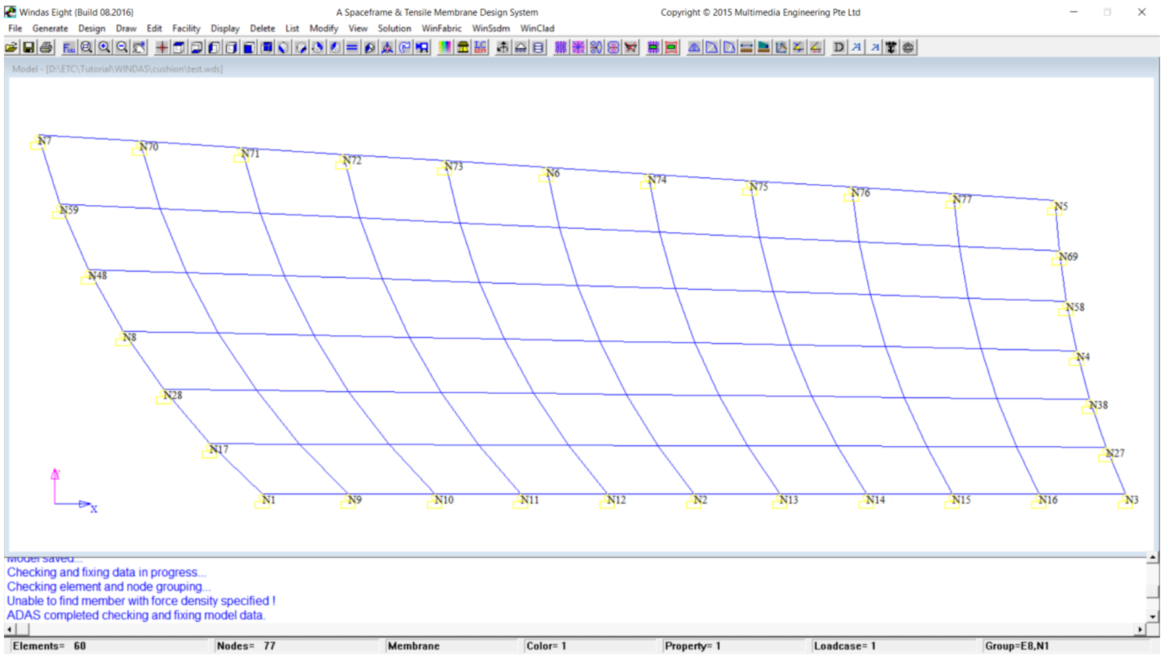
Windas will automatically create the membrane shape as shown below;



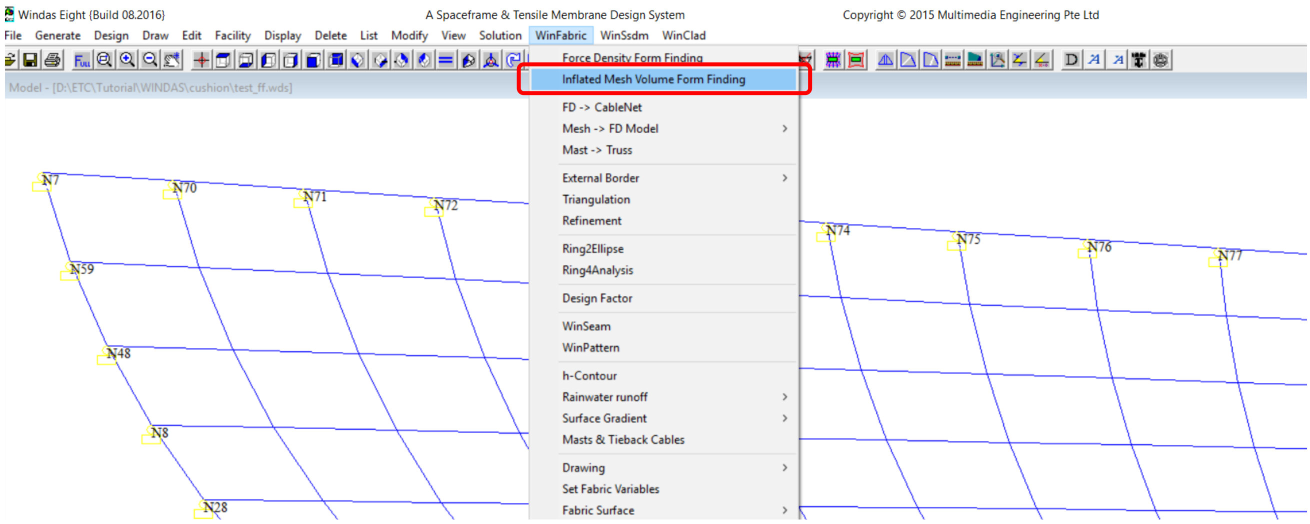
In order to inflate the surface, we need supports along the perimeter of the surface.

To generate support, Click on **Generate | Support | Perimeter** and choose **Pinned**

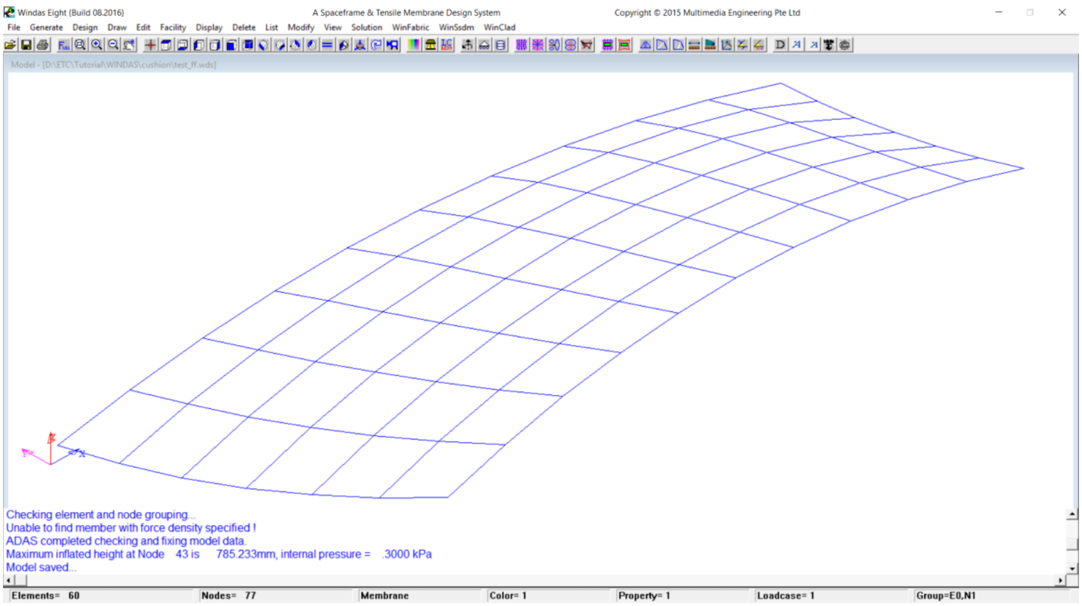




Now we've already generate support along the perimeter of the mesh, and our mesh is ready to be inflated. To Inflate the mesh, use **WinFabric | Inflated Mesh Volume Form Finding**



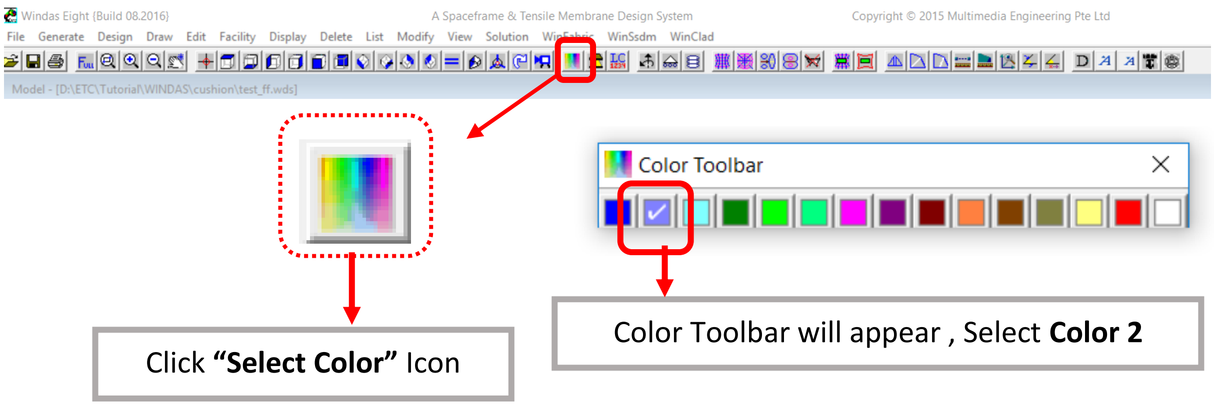
Then, Inflated mesh Volume Form Finding Dialog Box will appear in your worksheet. **Accept** settings below.



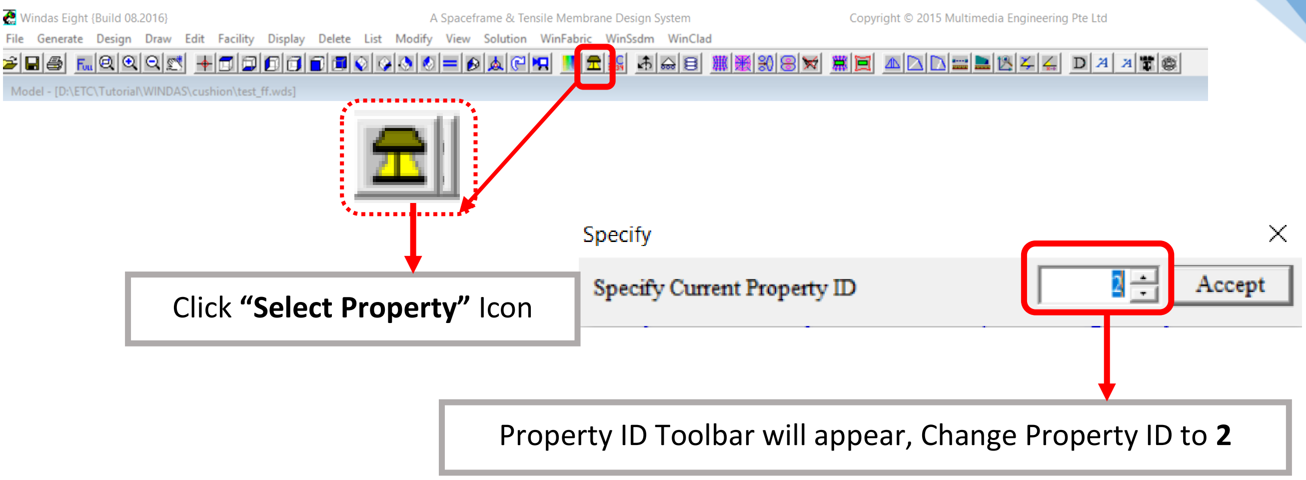
Now, We've already had Top Layer of the cushion,

Each of the ETFE Cushion contains of 2 layer (Top Layer and Bottom Layer). We've already generated the Top Layer, now we need to generate the Bottom Layer.

Before we generate the Bottom layer, we need to change the **Color**

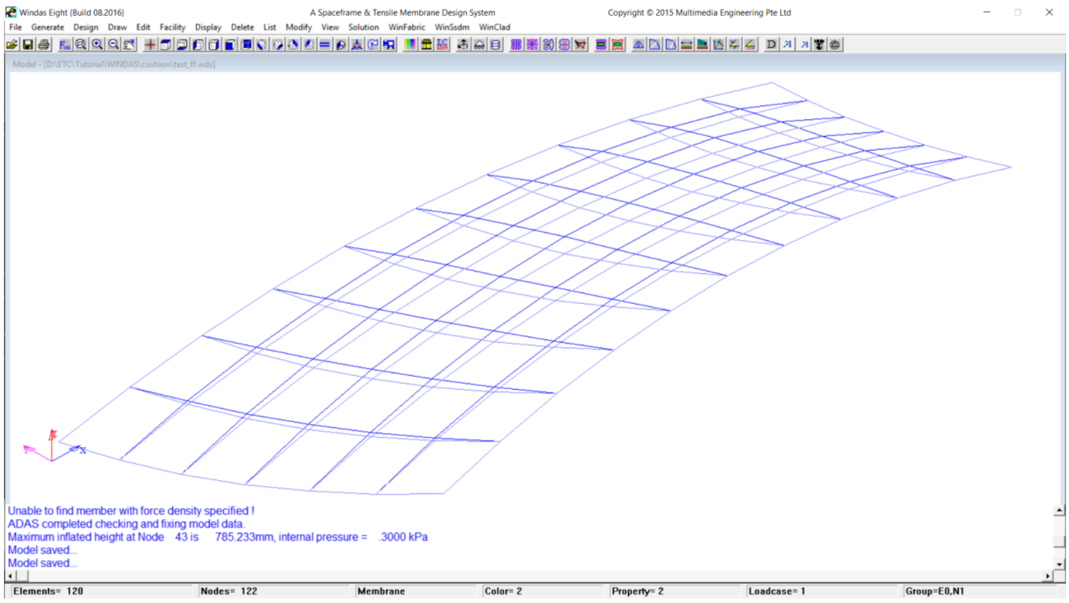


After Changing the Color, now we need to change the **Property ID**

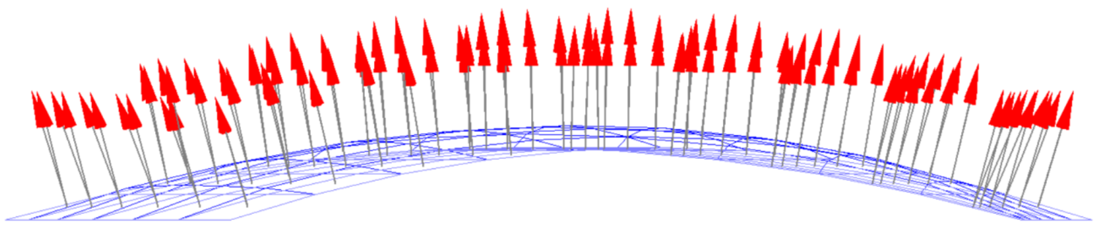


If the Color and Property ID have been changed, Now, We are ready to generate the Bottom Layer using Mesh command.

Click **Generate | Mesh | Mesh Definition | Run.** The Bottom Layer will appear in different color (Color 2 – Light blue color) as shown below;

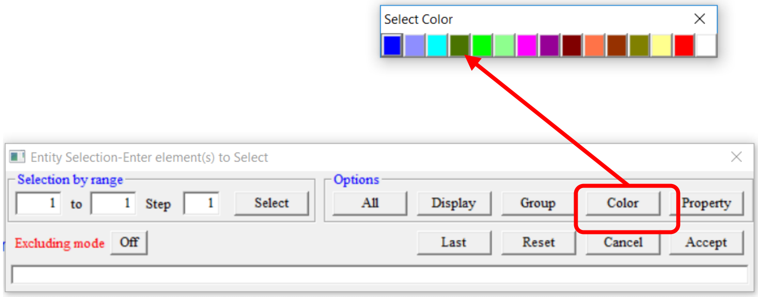


We need to check the Membrane Normal Direction using **Display | Membrane Normals**

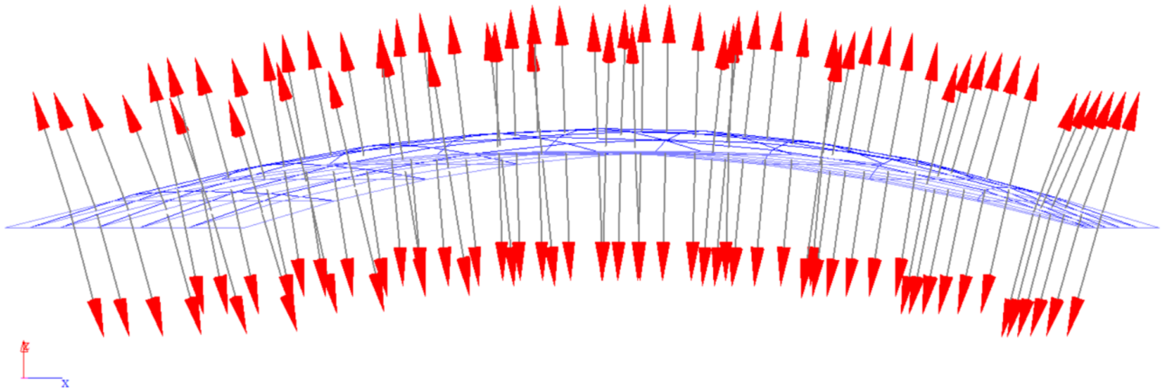


Both layers have the same Z direction

For the Bottom Layer, the Z direction should be in downward direction, so we need to change the Z direction of the Bottom Layer. Click **Edit | Surface Normals | -Z**



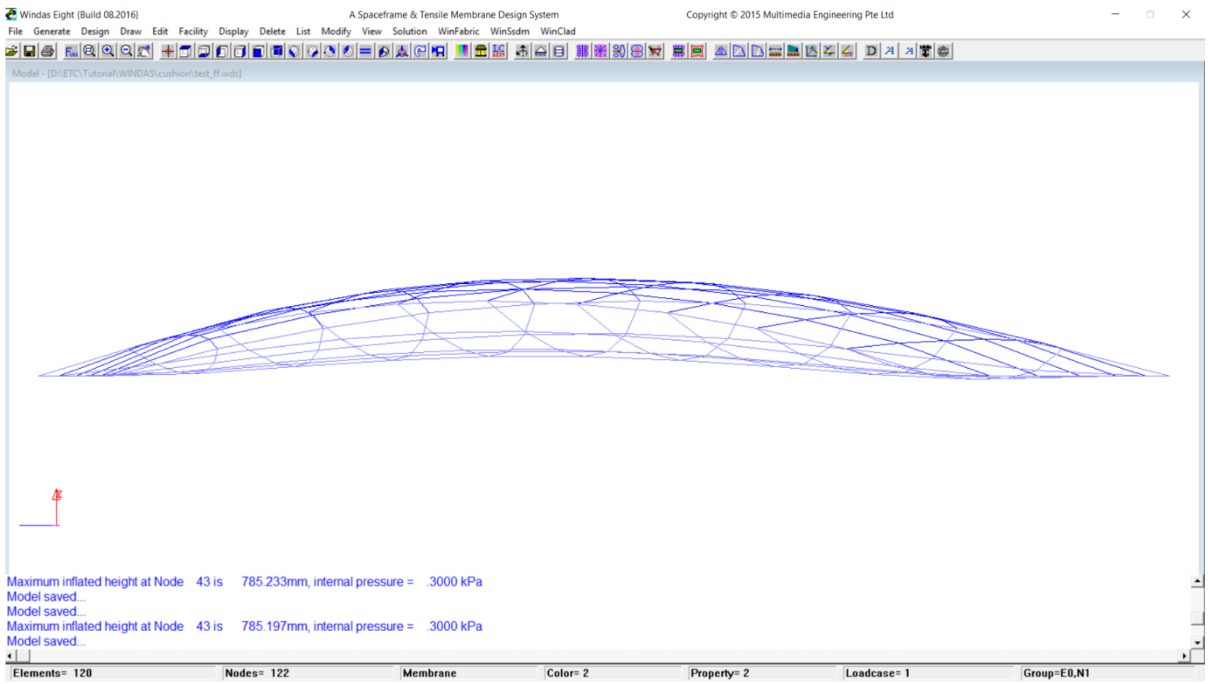
Click **Color** and select **Color 2** then **Accept.**



The Z direction of the bottom layer has been changed.

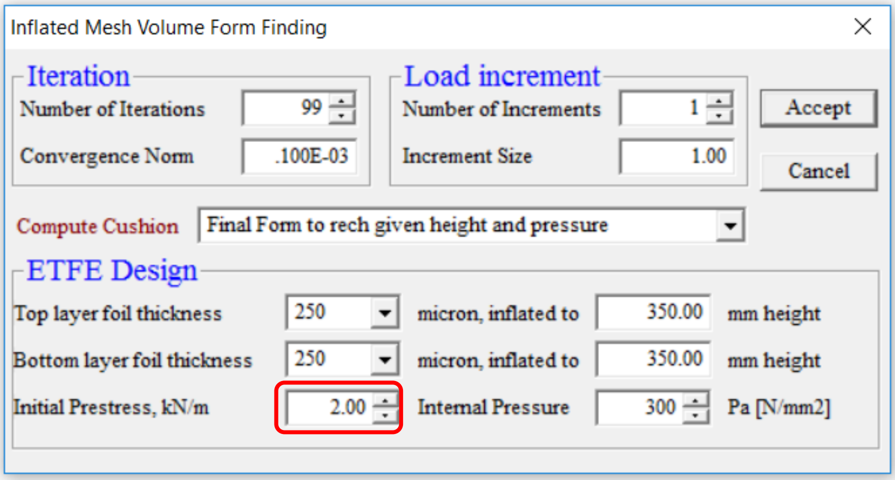
Now, We can reinflate the cushion, using **WinFabric | Inflate Mesh Volume Form Finding | Accept.**

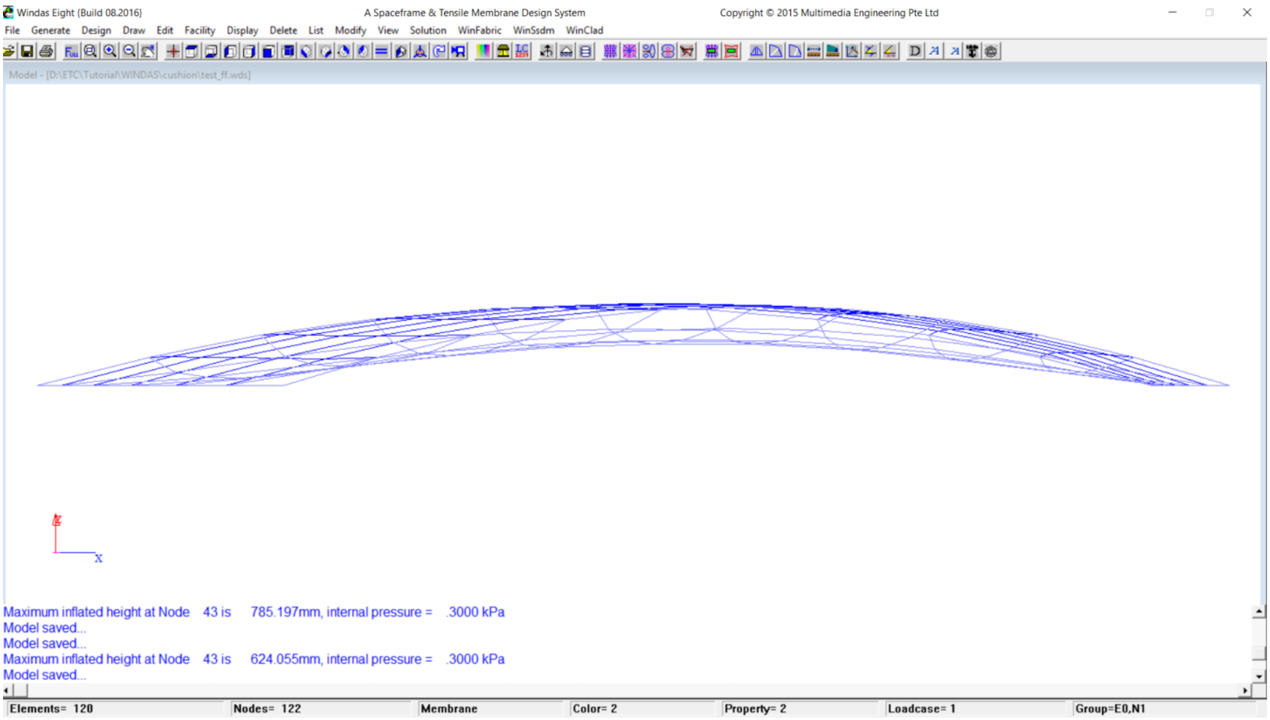
We can see the cushion inflation as shown below;



Previously, we generate the cushion using Initial Prestress 1, Now we can try change the prestress to 2 to see the different.

Click on **WinFabric | Inflated Mesh Volume Form Finding |** Change Initial Prestress to 2

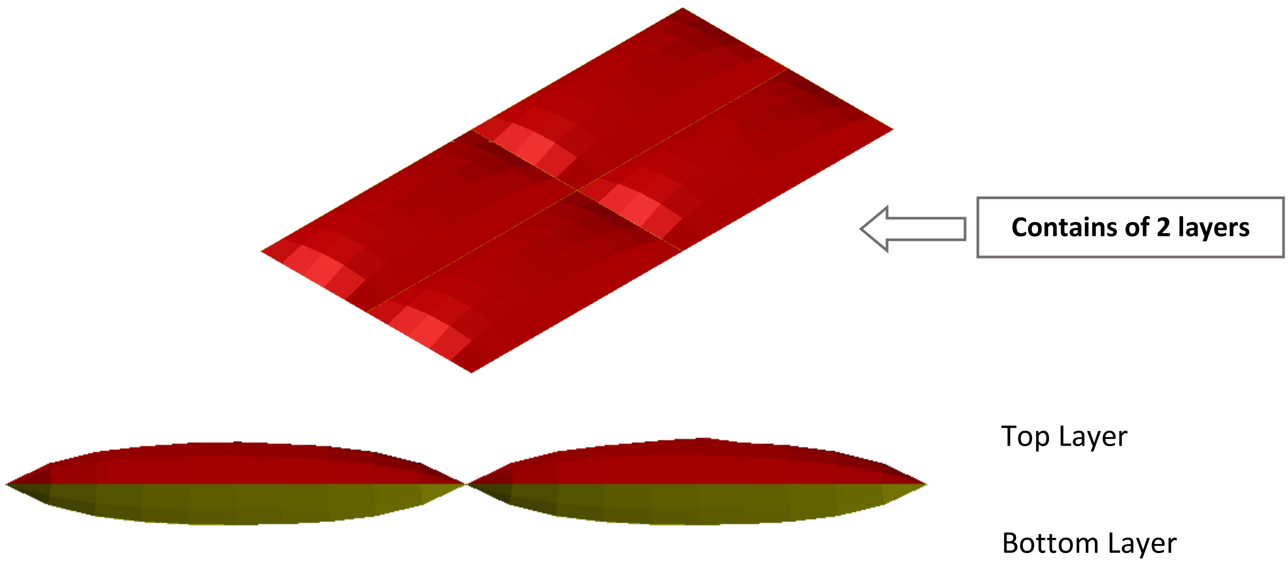




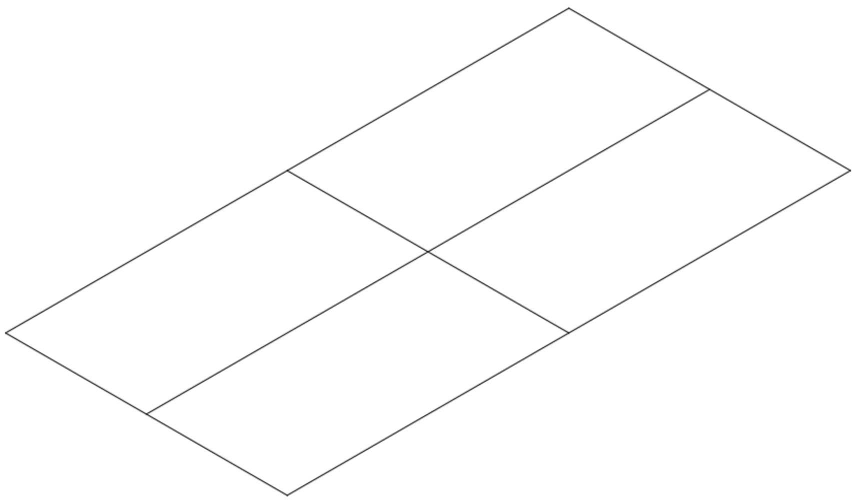
The Cushion's Height is differemt

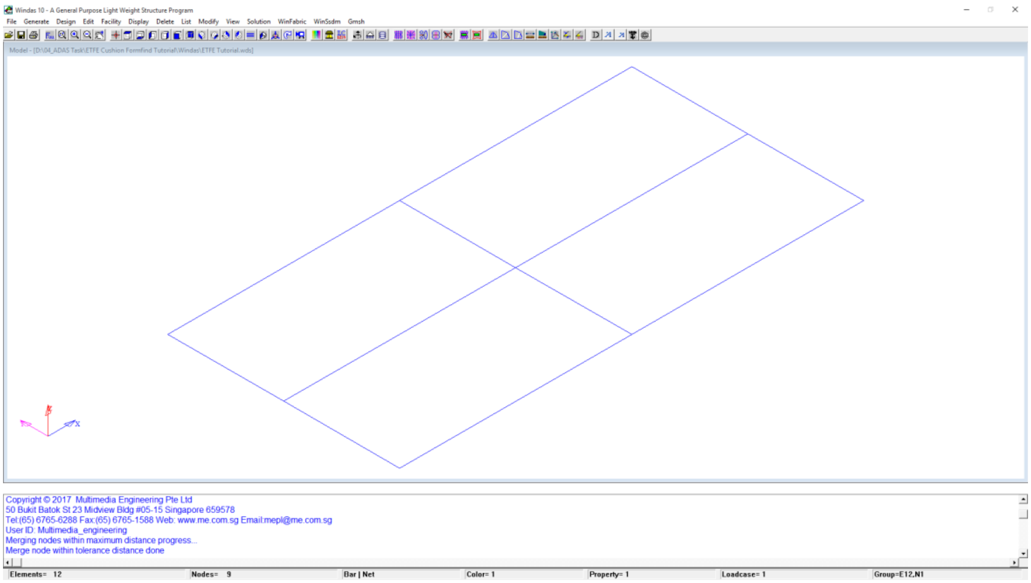
17.3. ETFE MultiCell Cushions

The example of ETFE MultiCell Cushions shape we want to achieve is shown below;

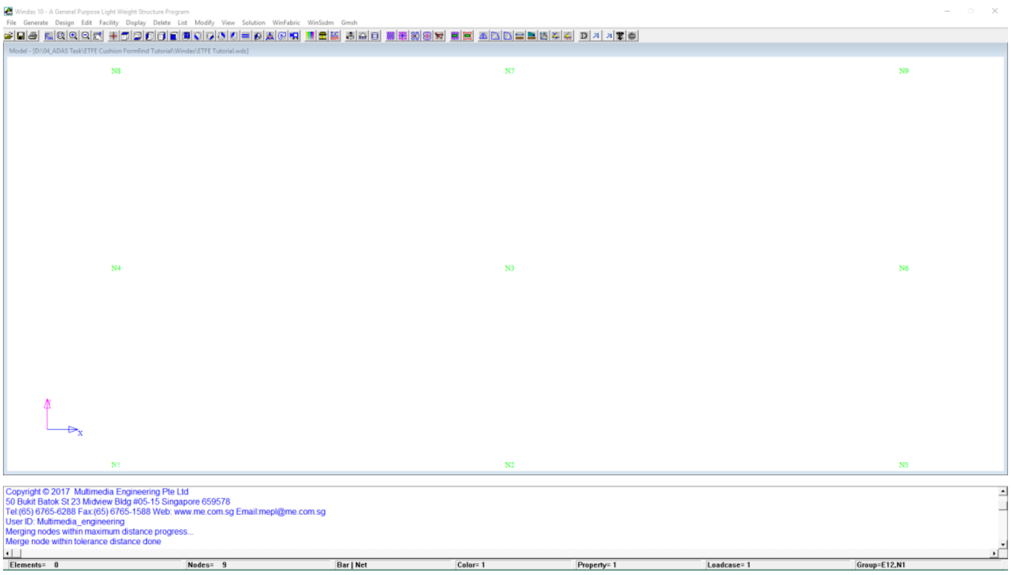


First, we need to take some reference points and draw the line to connect the reference point in AutoCAD or other software and save it as (.dxf) file. We just need to draw the perimeter as a straight line to facilitate us to create this model in Windas. For this kind of shape we can draw the perimeter as shown below;

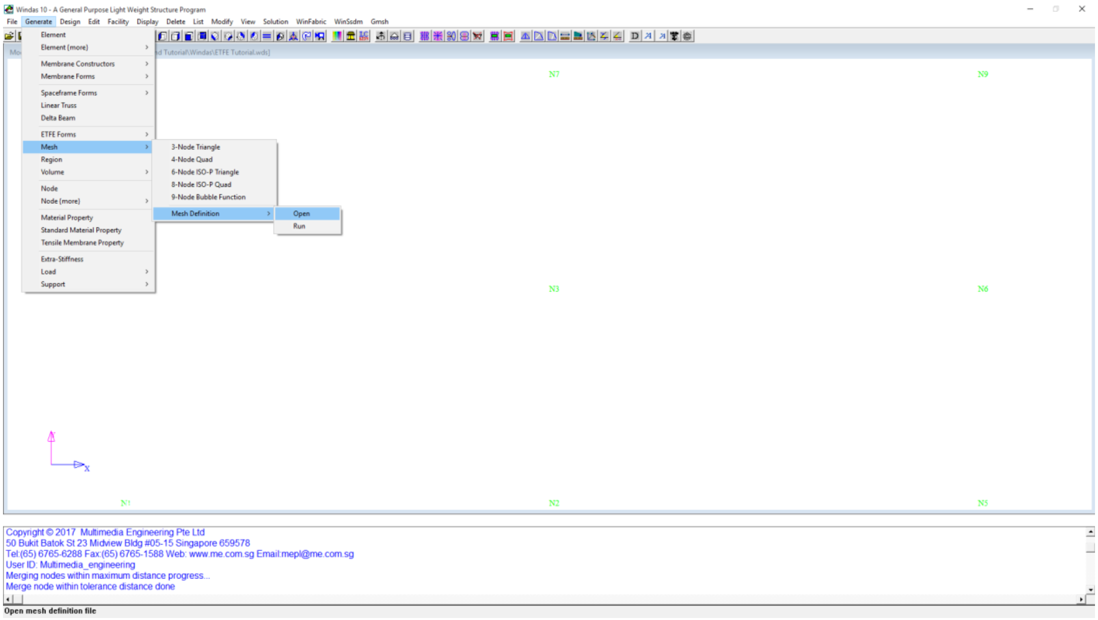


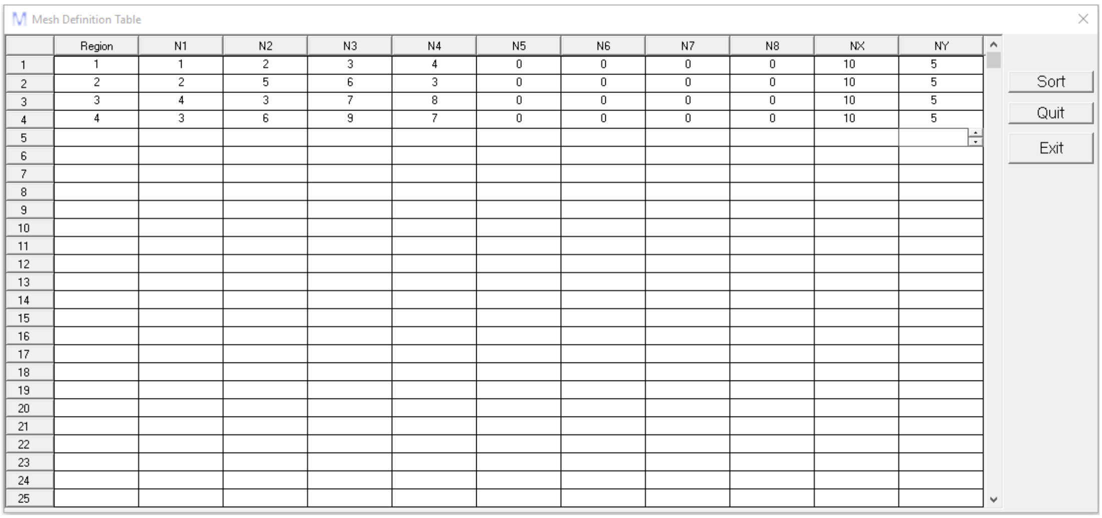


After that we can import the (.dxf) file to Windas with **File | Import | AutoCAD {DXF R12}**. Then we need to delete the line we made before by clicking **Delete | Element | Display** so only the node point remain in our Windas model.



Now we can use the "Mesh" command to create the ETFE cushions model. Click on **Generate | Mesh | Mesh Defination | Open.** Then a new window will appear as shown below. In the new window you can see **Region, N1….N8, NX,** and **NY** column. For this case, we will use 4 nodes each region and divide the mesh into 4 regions as we intend to do 4 ETFE cushions. Then fill the N1 to N4 column with the node number. (**Note: we need to fill the node number in anticlockwise sequence.** See the example below). The NX and NY column is to define how many divided area in X and Y direction we need for our cushions model.

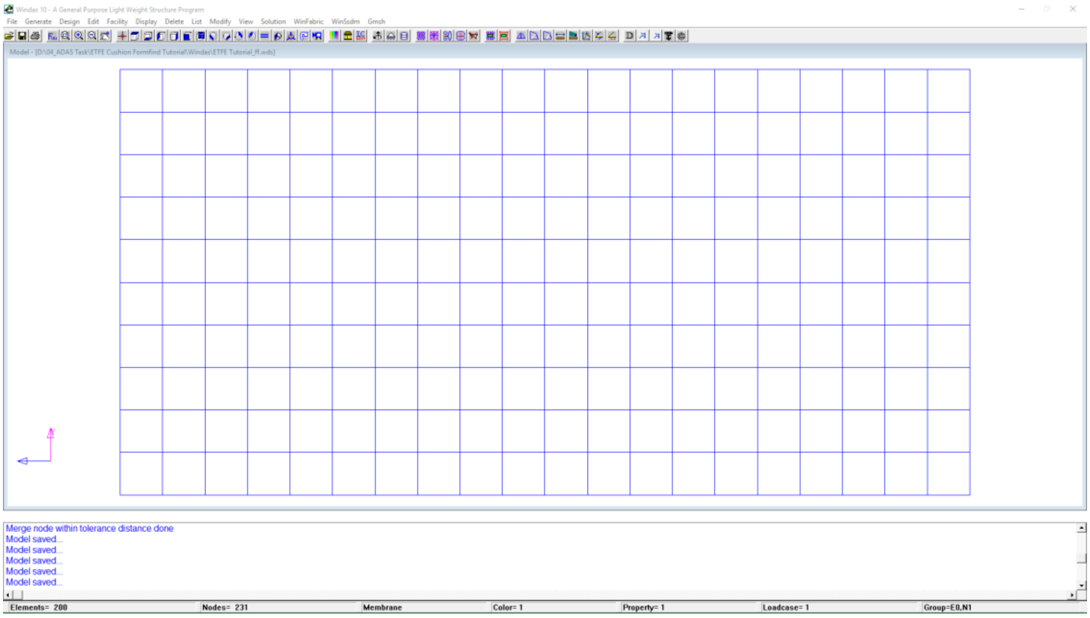




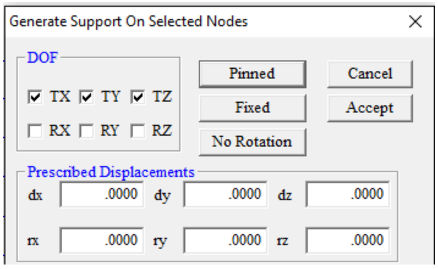
	Region	N1	N2	N3	N4	N5	N6	N7	N8	NX	NY
1	1	1	2	3	4	0	0	0	0	10	5
2	2	2	5	6	3	0	0	0	0	10	5
3	3	4	3	7	8	0	0	0	0	10	5
4	4	3	6	9	7	0	0	0	0	10	5

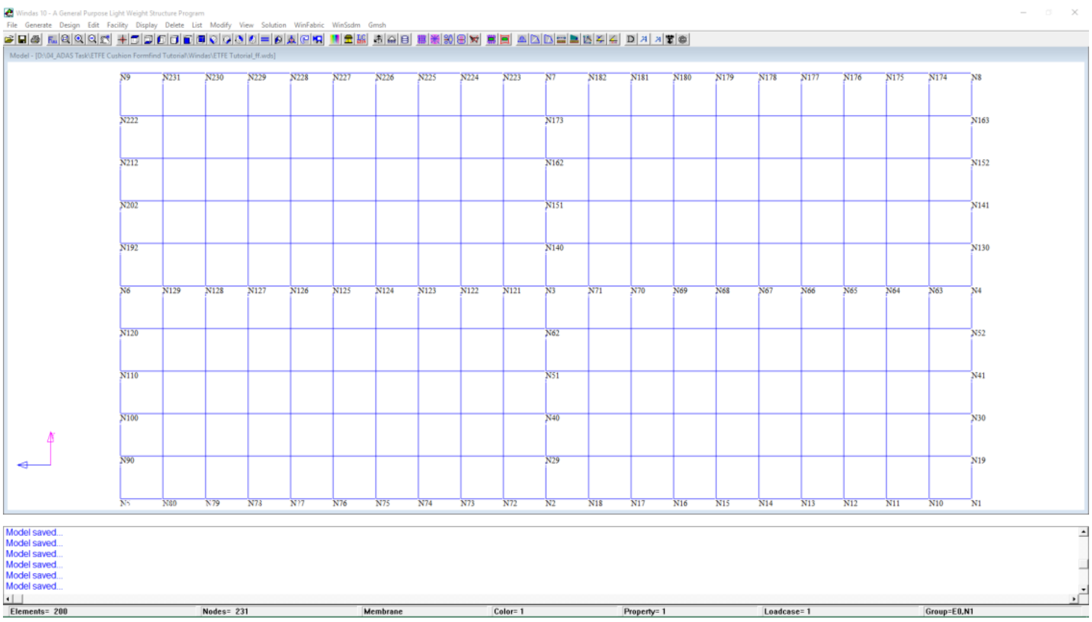
Before closing this window, click **Sort** to make sure that the input data are saved then **Exit**.

Save the file and also **save as " filename_ff "** so that later we can open the file before we did the form finding. After "save as" you can see that the mesh definition we made before is still there. Now click on the **Generate | Mesh | Mesh Defination | Run.** Windas will automatically create the mesh shape as shown below;

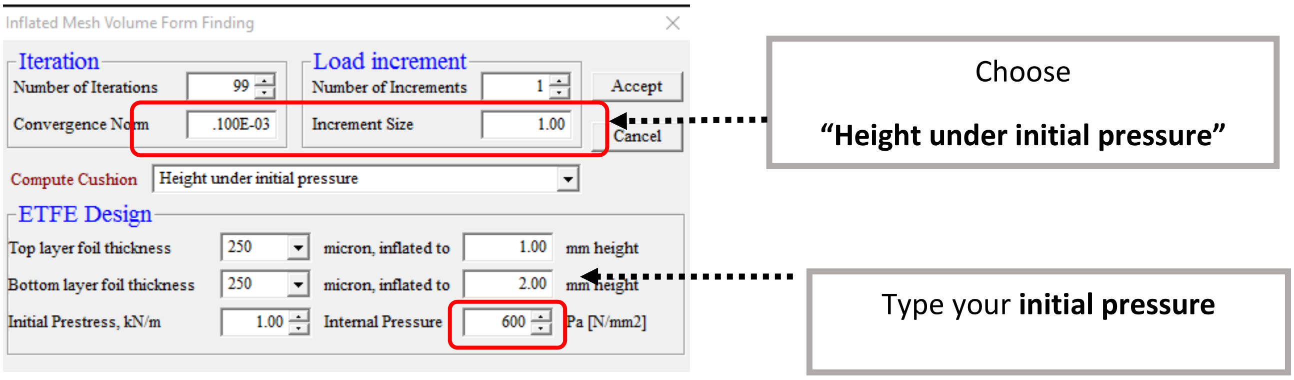


Apply the support. Since we want the loads from ETFE cushions later will be transferred to the steel along the cushions perimeter, therefore we need to apply the support at cushions perimeter. Use **Generate | Support | Nodes** select **Group 1** and choose **Pinned**

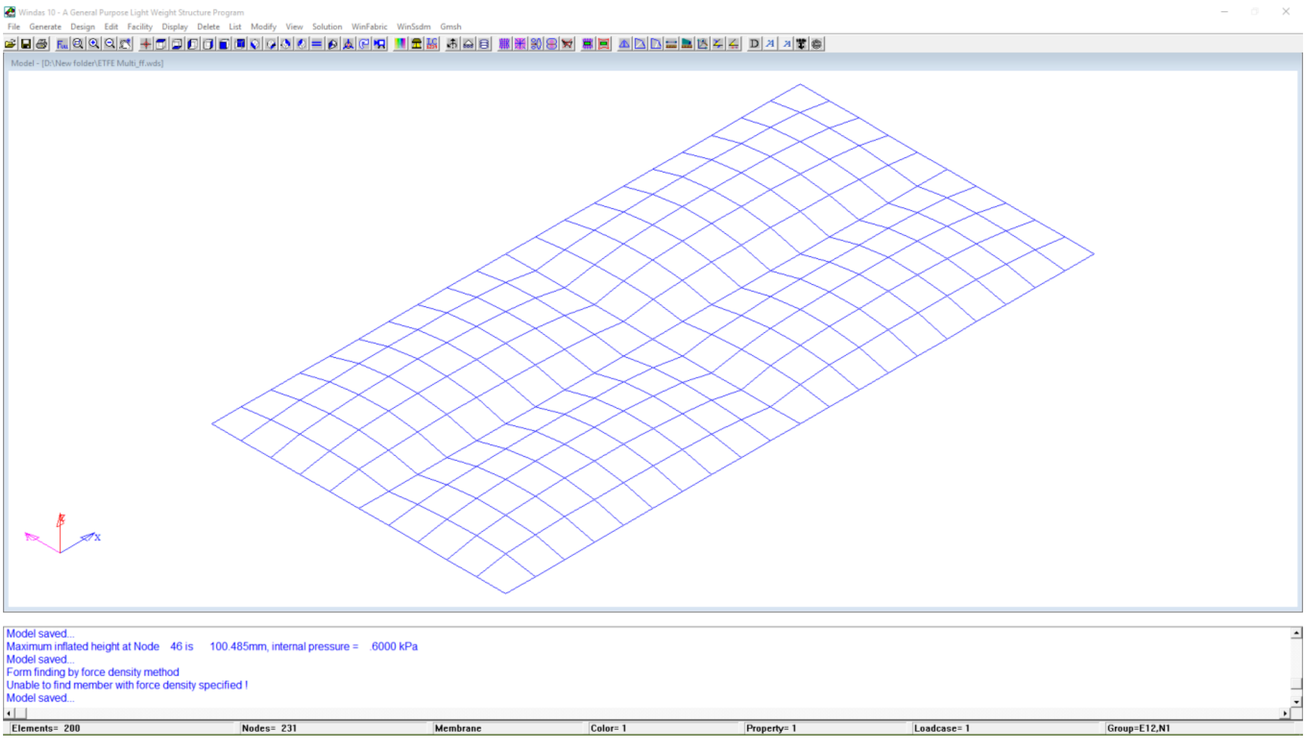




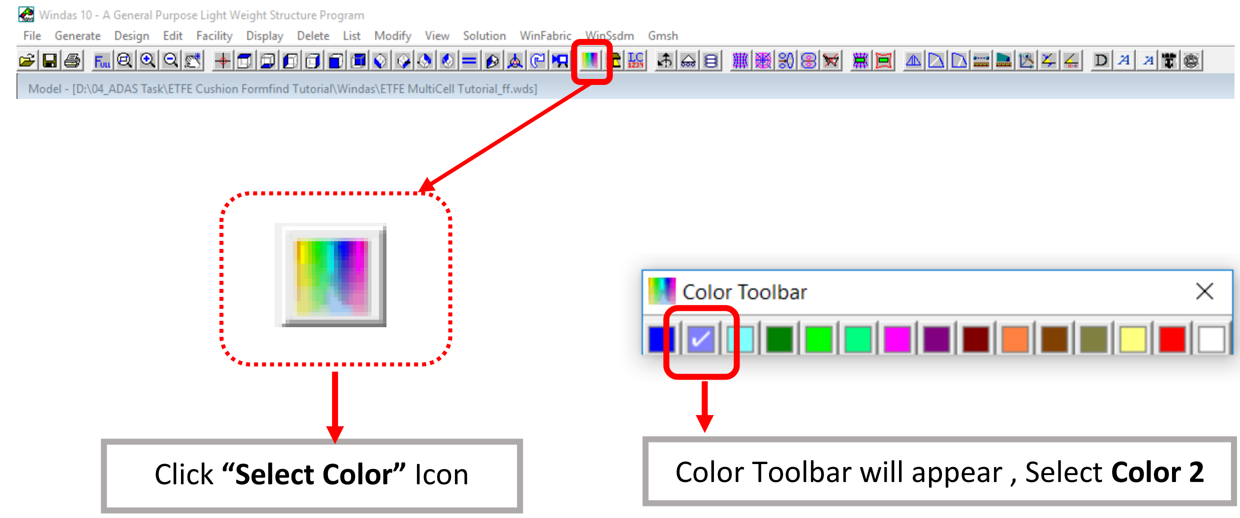
After the support applied, it is a good practice to check if there is any double nodes. Click **Facility | Check | Nodes**. The next step we will inflate the mesh by using this command **WinFabric | Inflated Mesh Volume Form Finding**. Then, Inflated Mesh Volume Form Finding Dialog Box will appear in your worksheet.



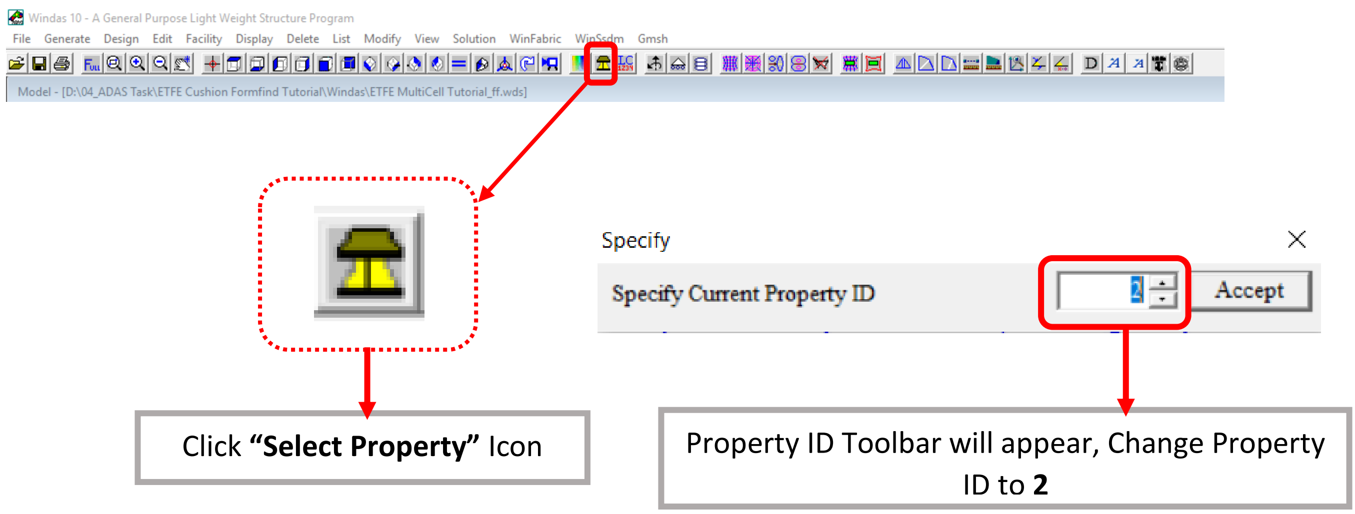
Then click **Accept**.



Now we have the Top Layer of the cushion. Every ETFE Cushion contains minimum of 2 layer (Top Layer and Bottom Layer). We have already generated the Top Layer, the next step is to generate the Bottom Layer. Before we generate the Bottom Layer, we need to change the **Color**.

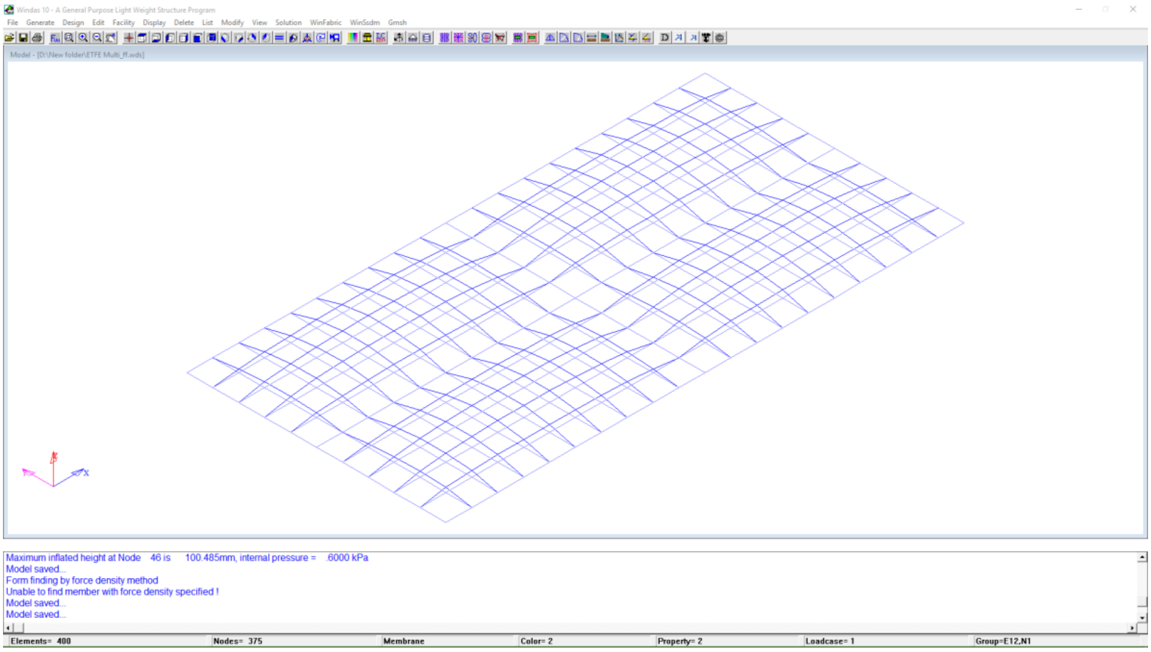


After changing the **Color**, we need to change the **Property ID**.

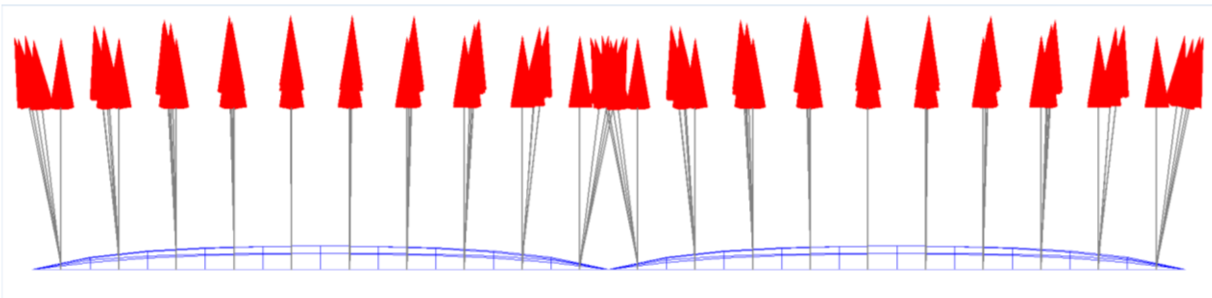


Now we are ready to generate the Bottom Layer using Mesh command.

Click **Generate | Mesh | Mesh Definition | Run**. The Bottom Layer will appear in different color (Color 2) as shown below;

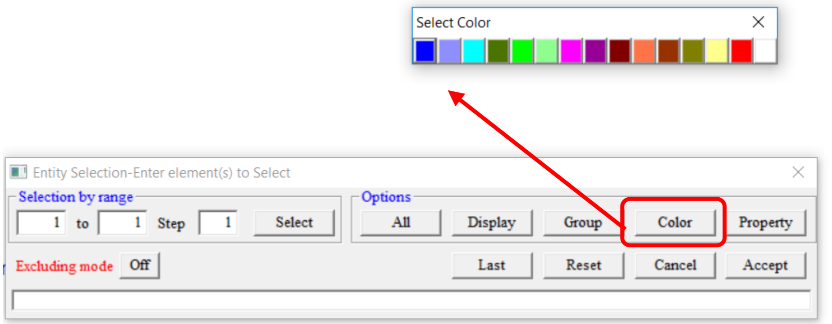


We need to check the Membrane Normal Direction using **Display | Surface Normals**.

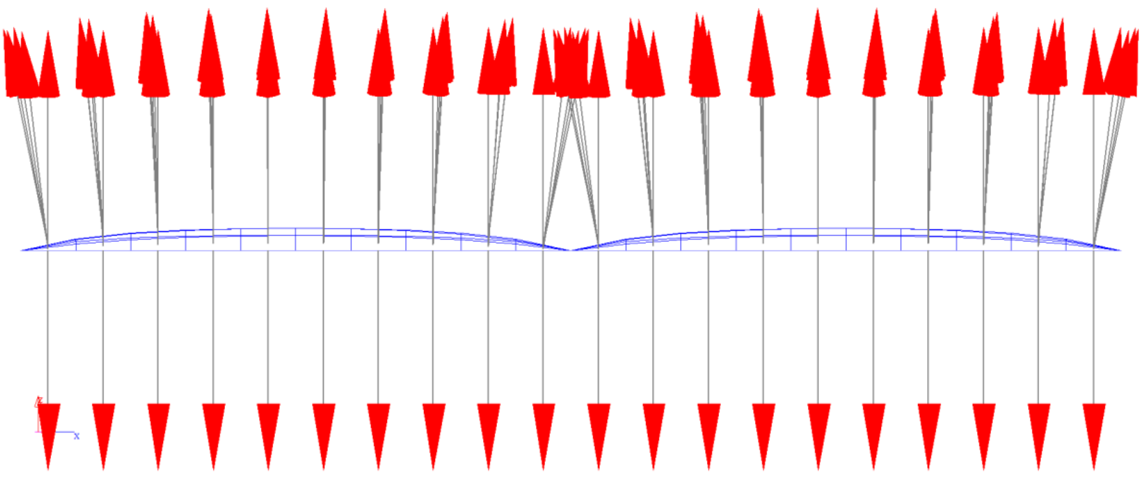


Both layers have the same Z+ direction.

For the Bottom Layer, the Z direction should be pointed downward, so we need to change the direction of Bottom Layer into Z- direction. Click **Edit | Surface Normals | -Z**



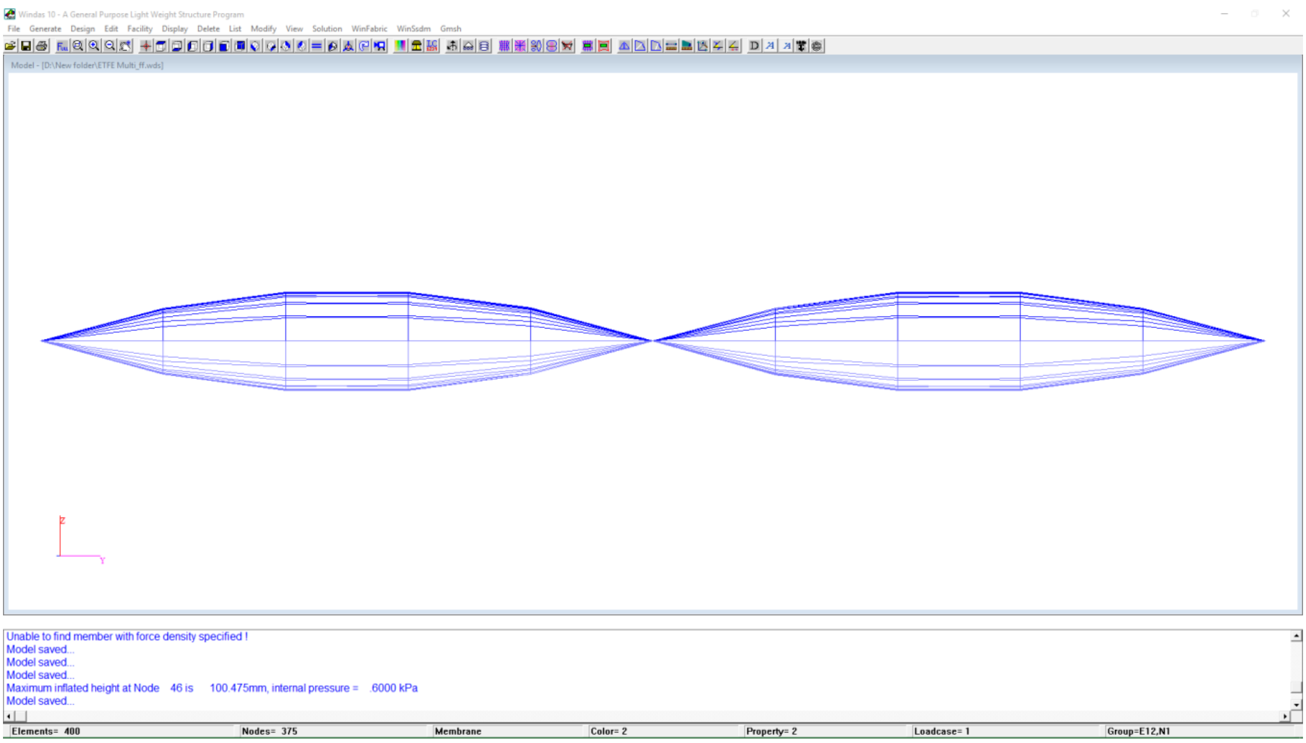
Click **Color** and select **Color 2** then **Accept**.



The Z direction of the Bottom Layer has been changed into Z-.

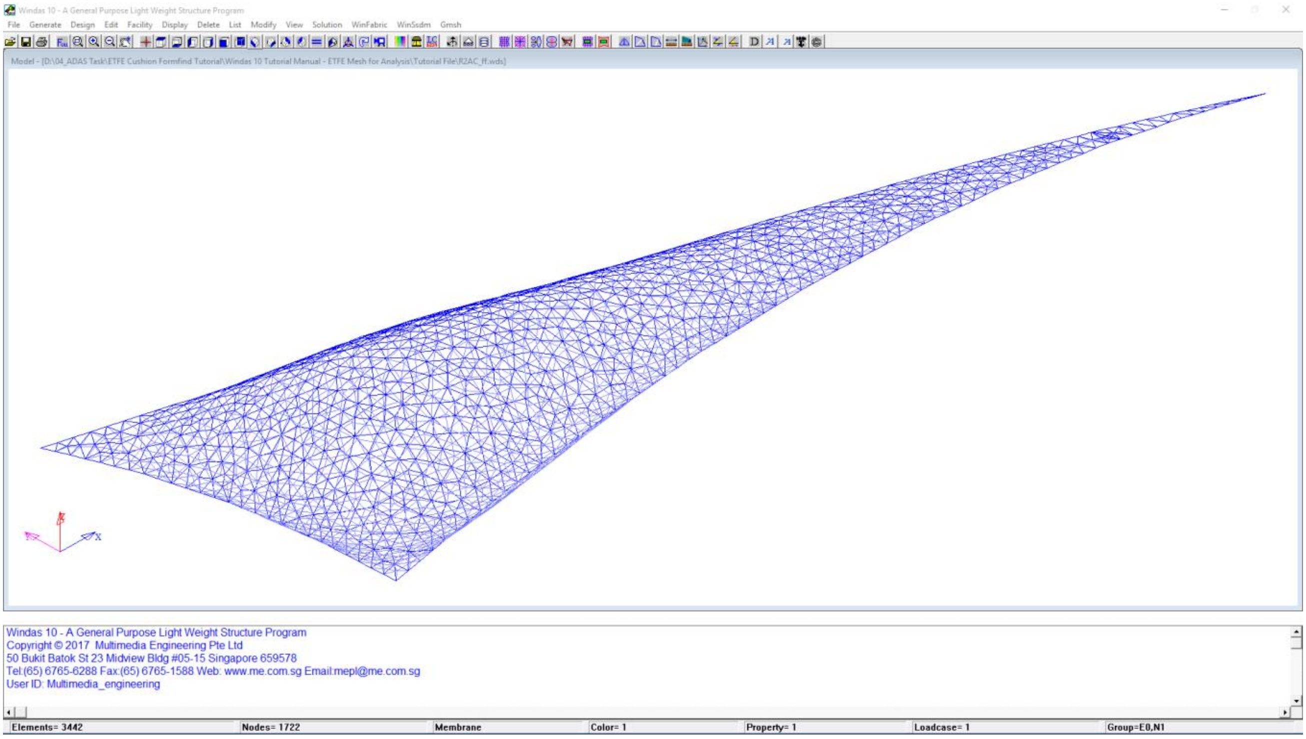
Now, we can reinflate the cushion, using **WinFabric | Inflated Mesh Volume Form Finding** , then click **Accept**.

We can see the cushion inflated as shown below;

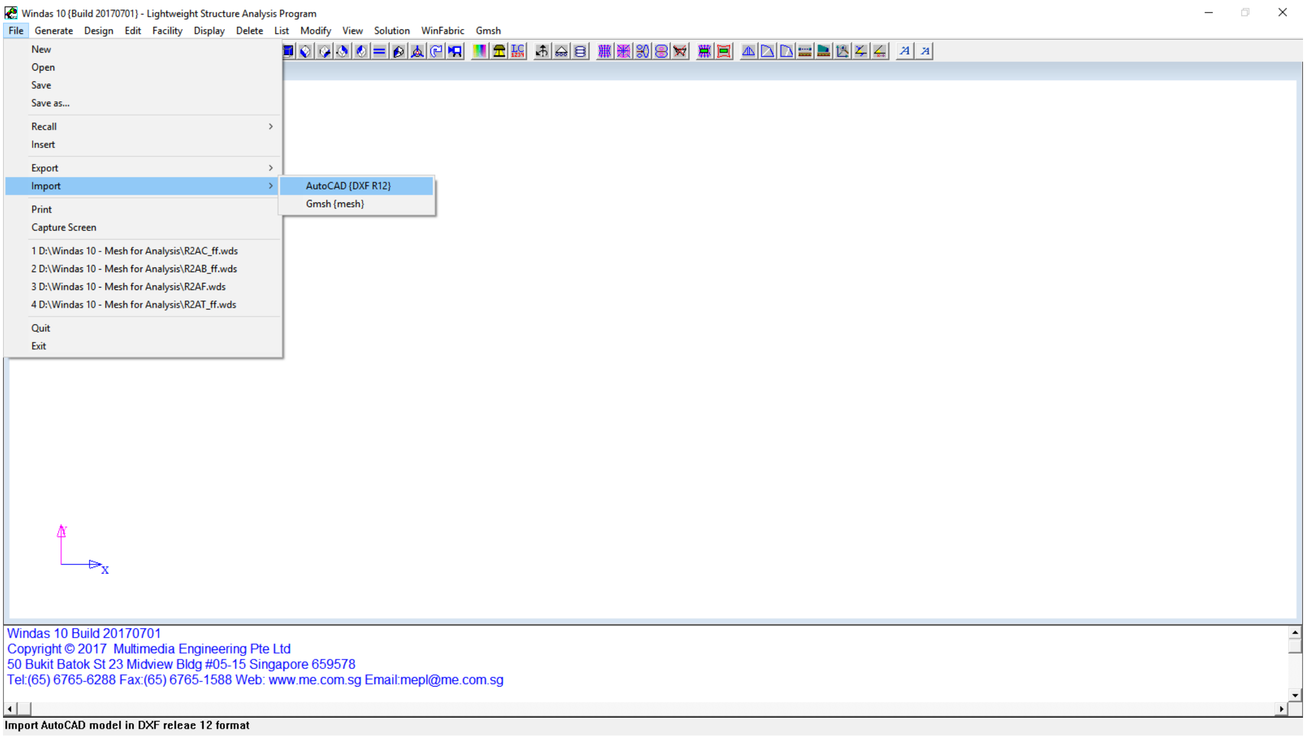


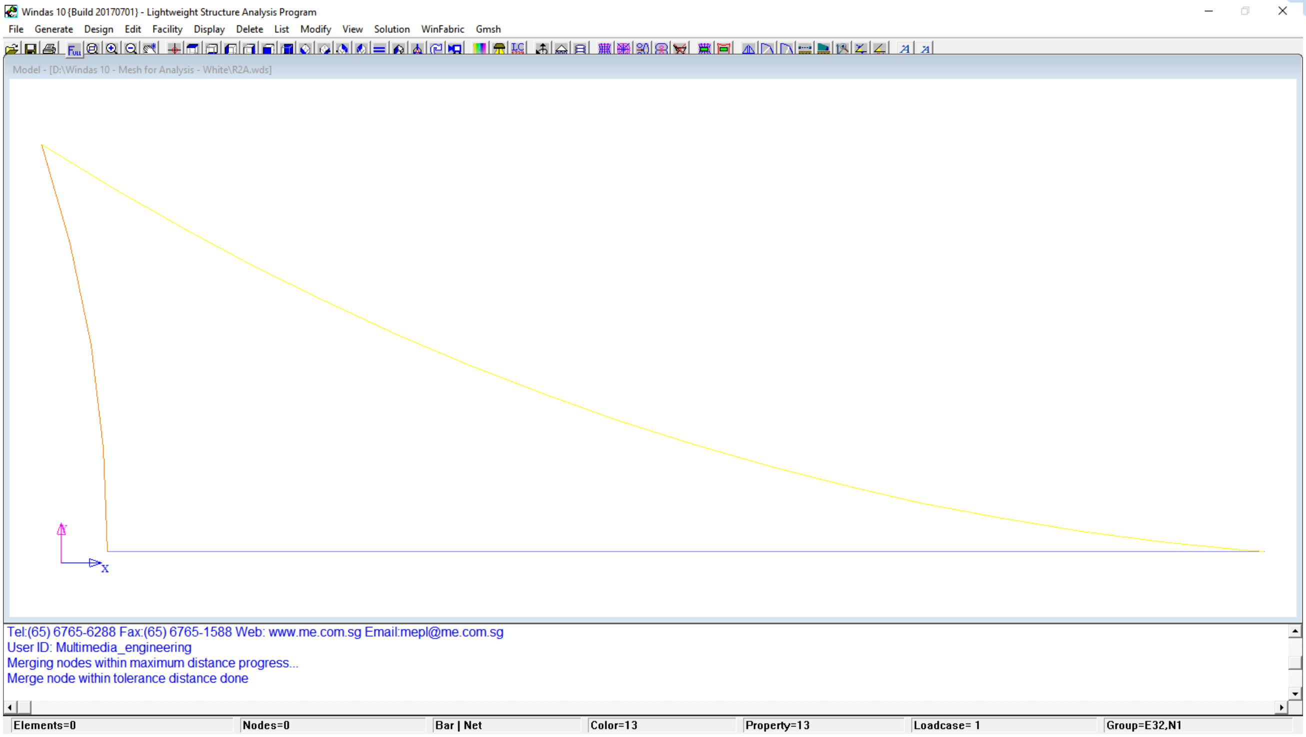
17.4. ETFE Mesh for Analysis

The example of ETFE Mesh for Analysis will be generated in Windas as shown below;

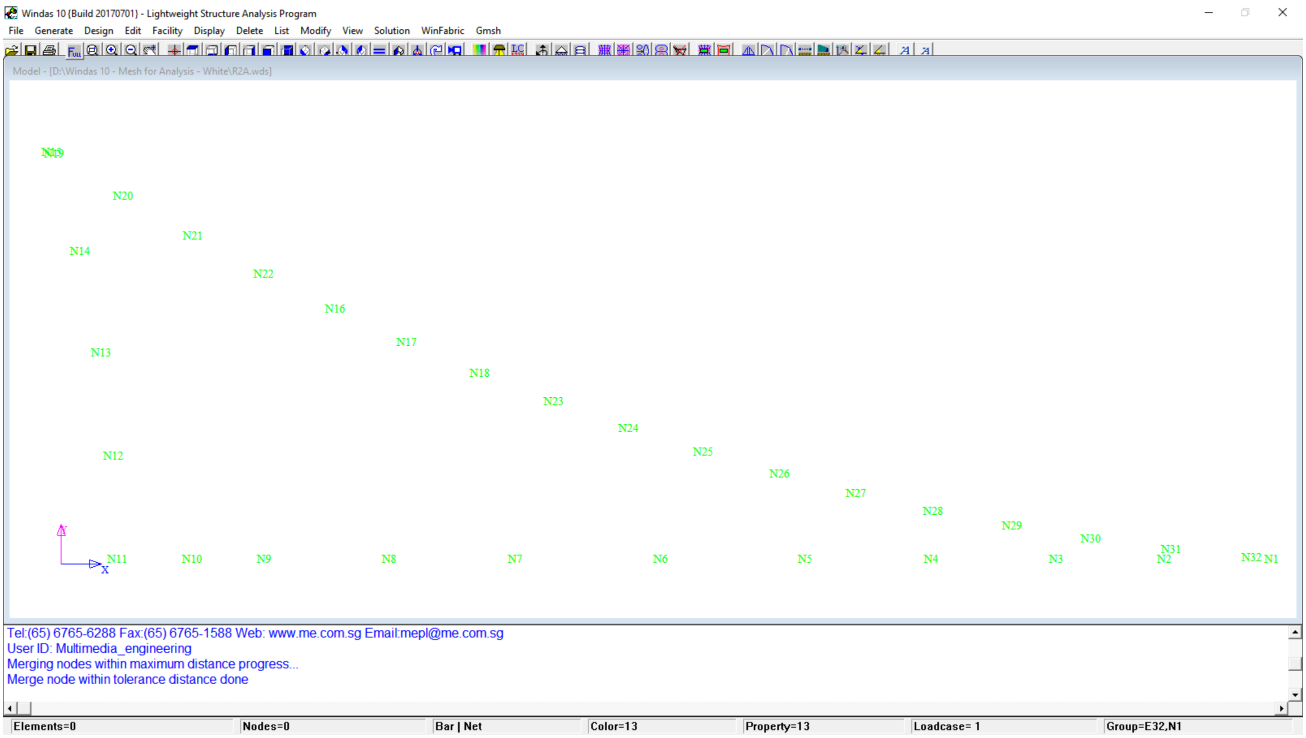


Click **File | Import | AutoCAD {DXF12}** to import the system line into Windas.

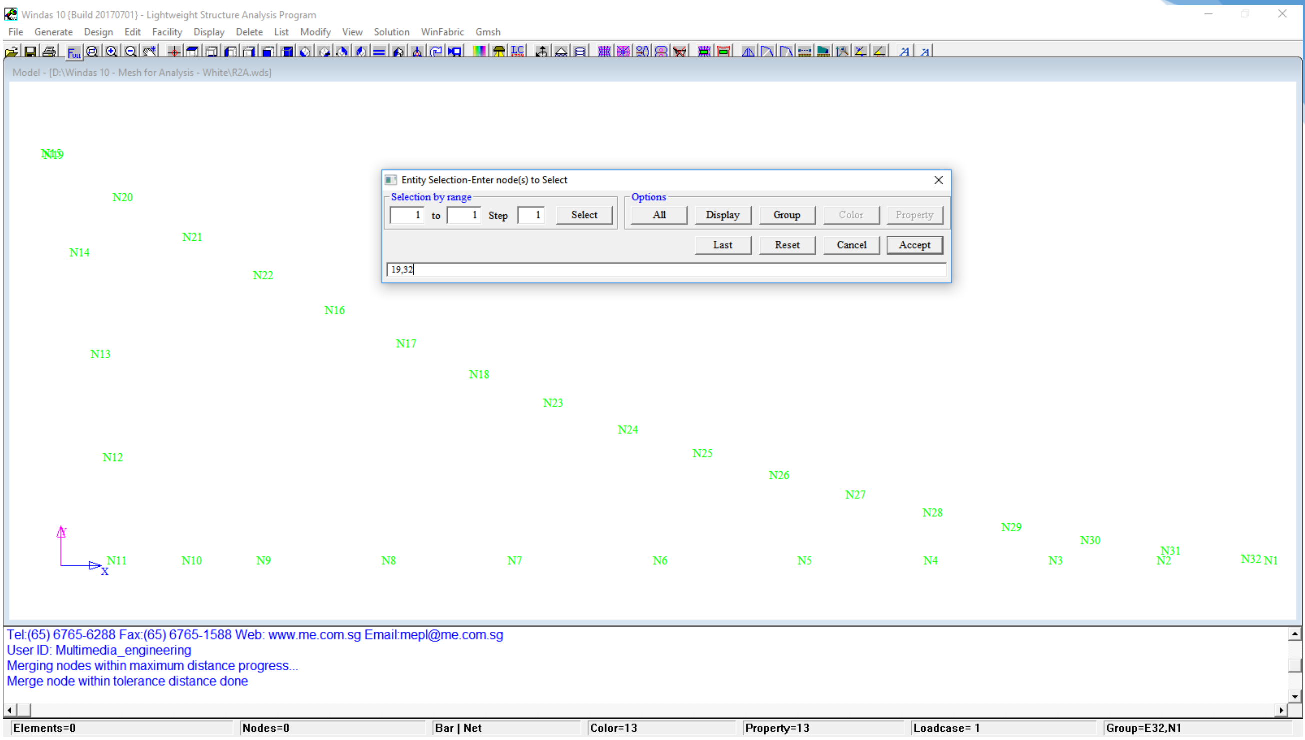




After the system line imported, click **Delete | Elements** select **All** then click **Accept** to delete the segmented lines since we only need the nodes.

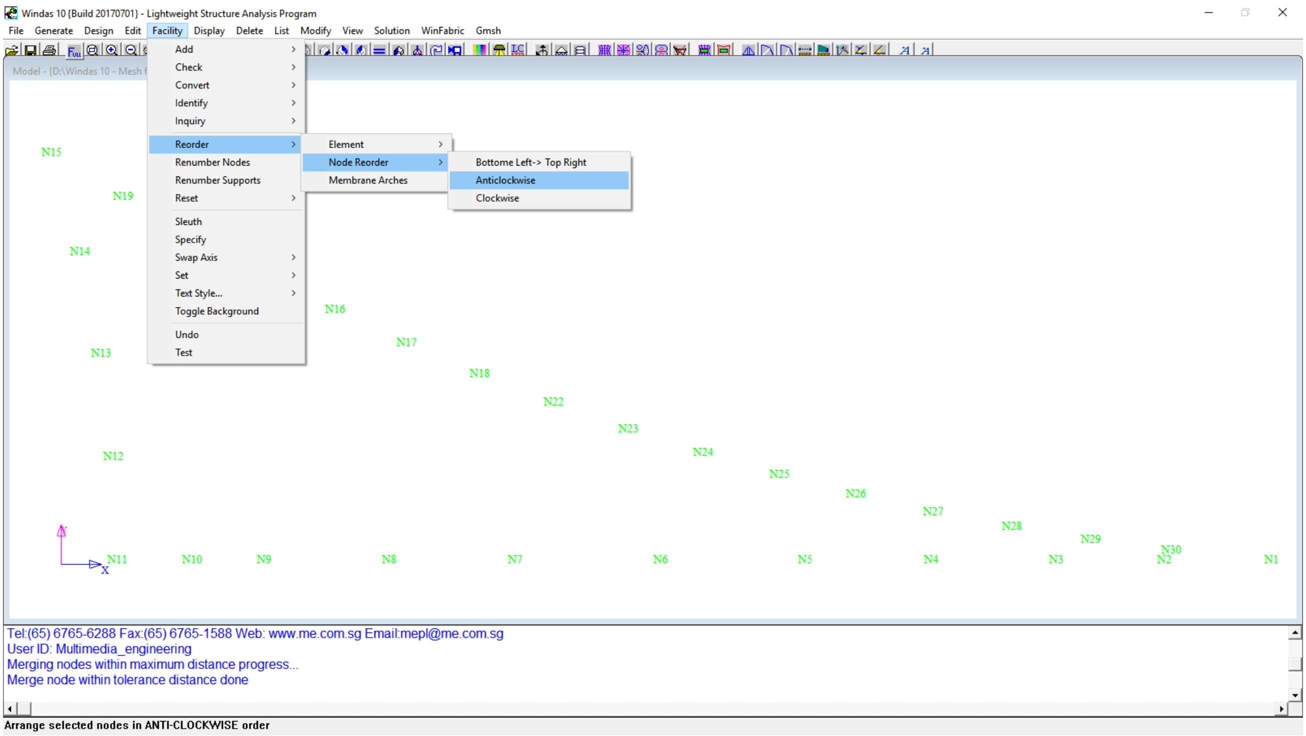


Then, please check if there are some nodes too close to each other. As for this case, node N19 & N32 are the unnecessary nodes. Click **Delete | Node** to delete the unnecessary nodes (nodes which are too close to each other).

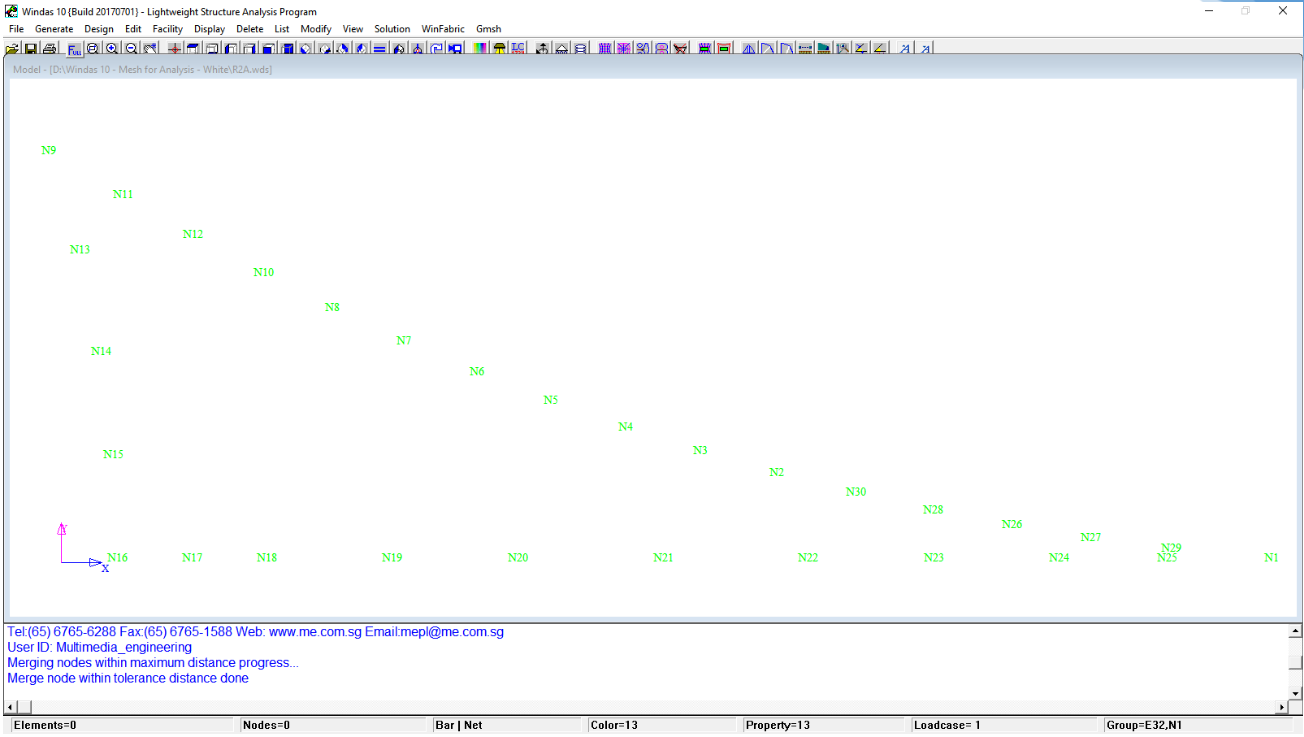


After we deleted the unnecessary nodes, we need to reorder the nodes in anti-clockwise.

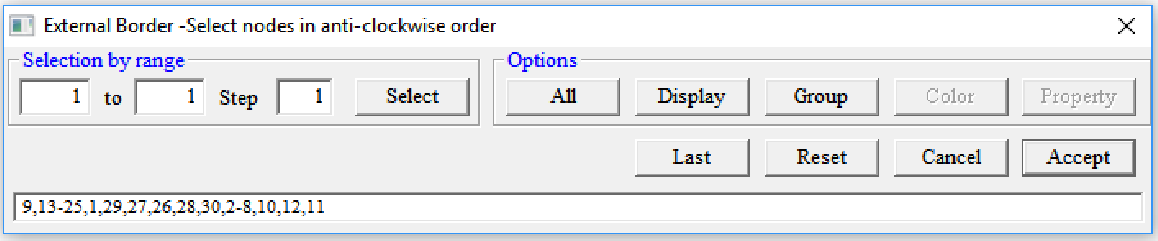
Click **Facility | Reorder | Node Reorder | Anti-clockwise** select **All** then click **Accept**.

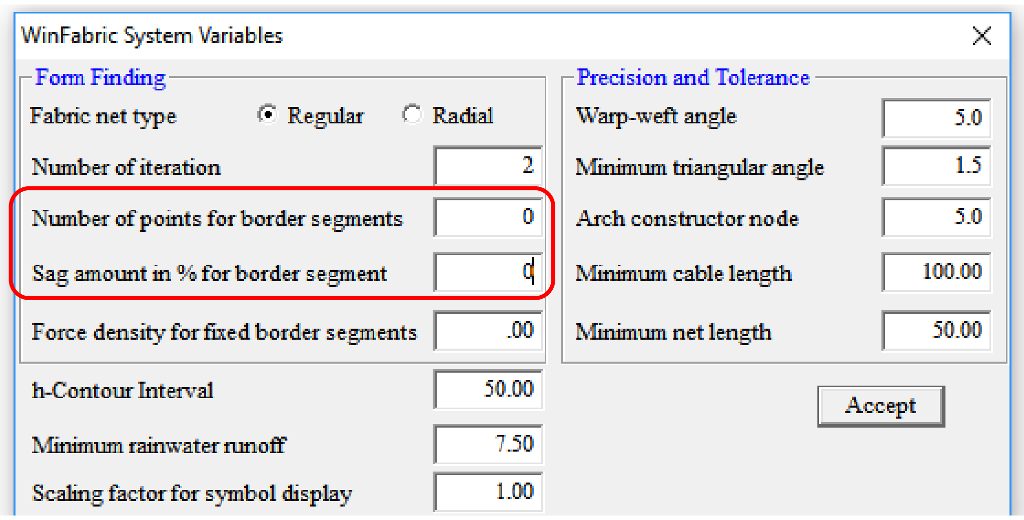


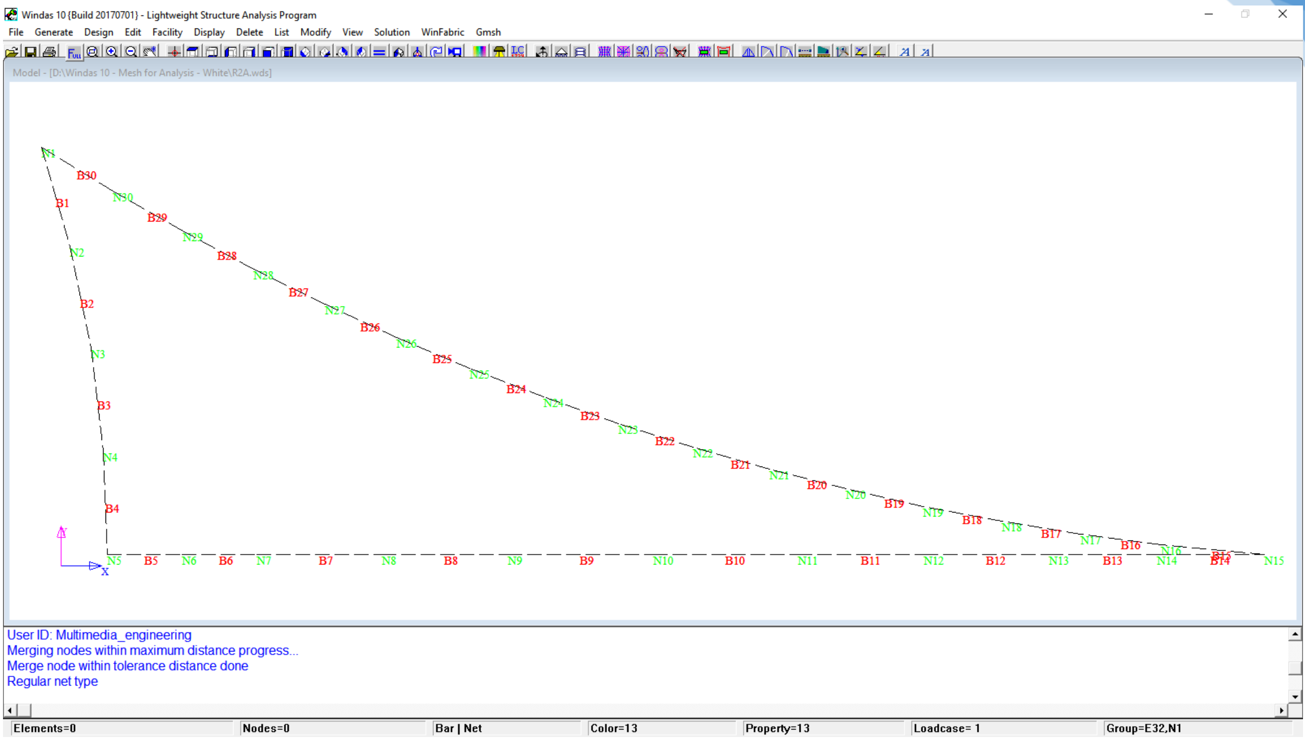
If the nodes are not in anti-clockwise order after we do the reorder several times, then we need to proceed manually by inputting nodes in anti-clockwise order for the next step.



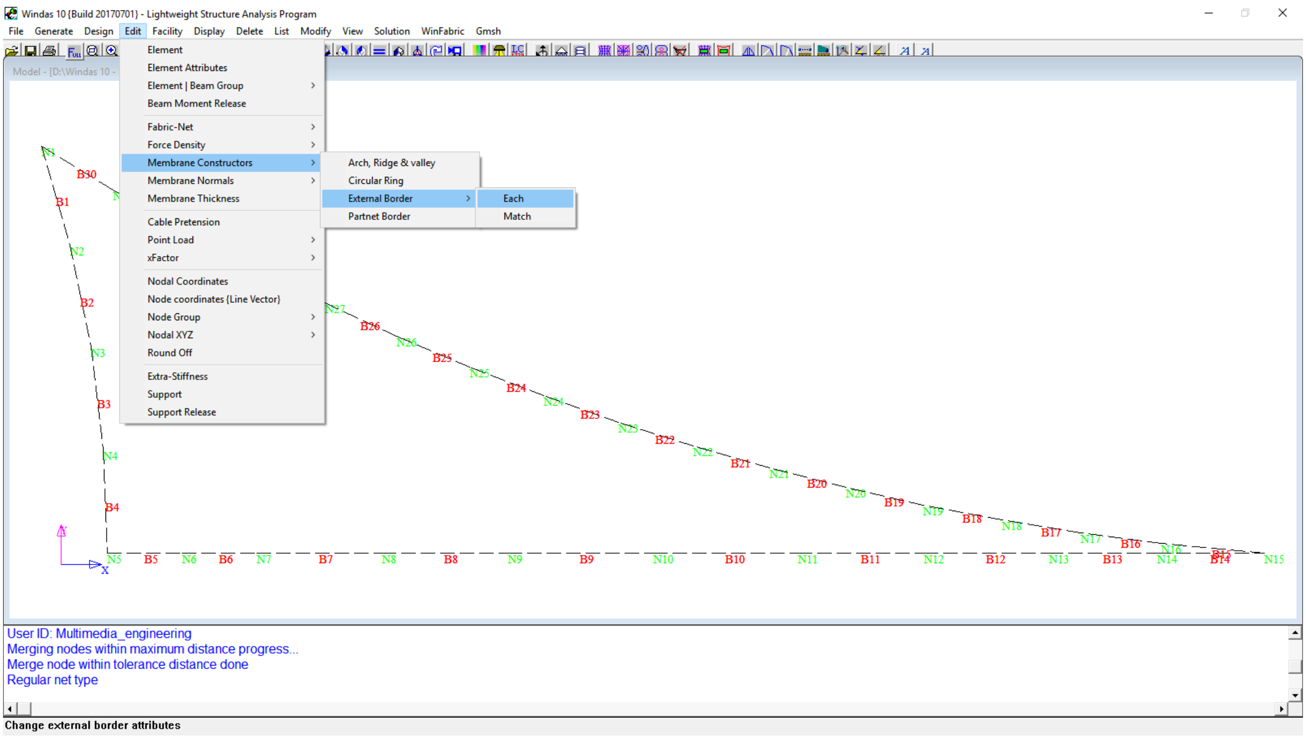
We need to generate the external border based on the nodes. Click **Generate | Membrane Constructors | External Border**, insert the nodes in anti-clockwise order, then click **Accept** and a dialog box will pop up. Change the **Number of points for border segments** into **0** and change the **Sag amount in % for border segments** into **0**. Click **Accept**.



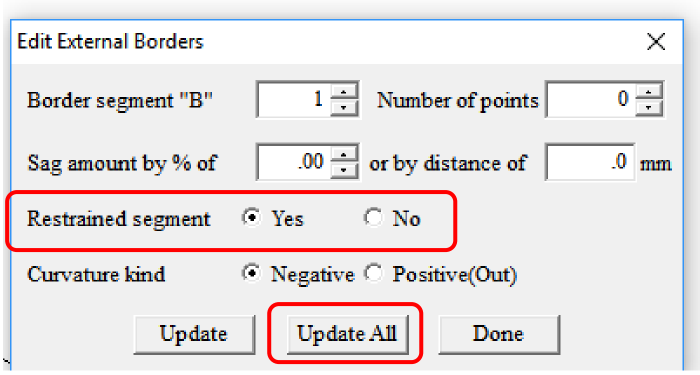


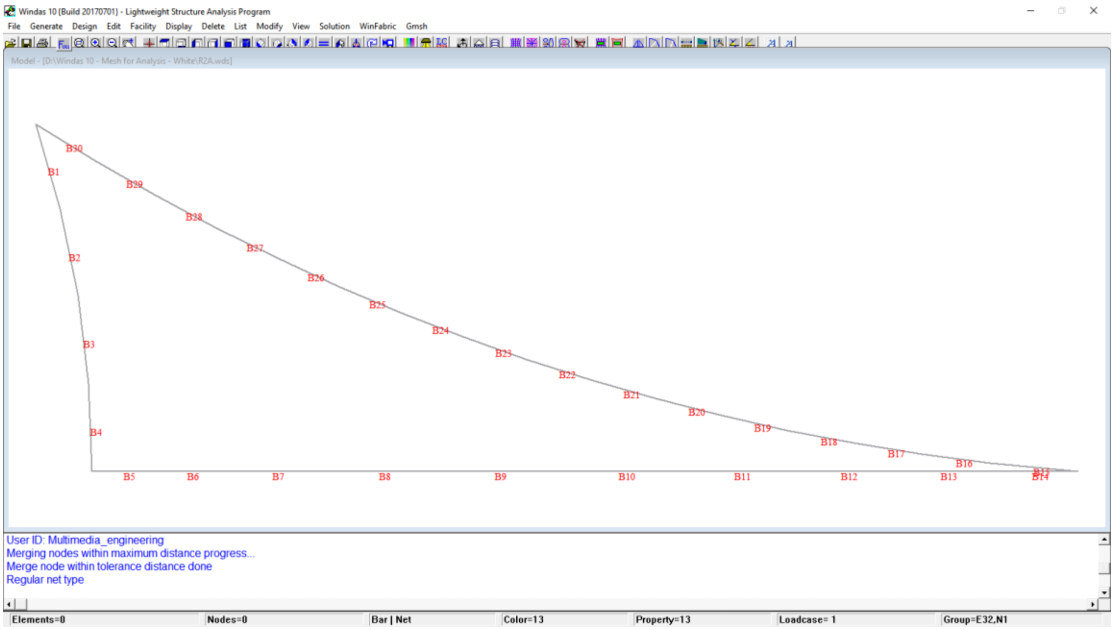


In order to have a fixed border following the cushion system line, click **Edit | Membrane Constructors | External Border | Each** set the **Restrained segment** into **Yes** and then click **Update All**.

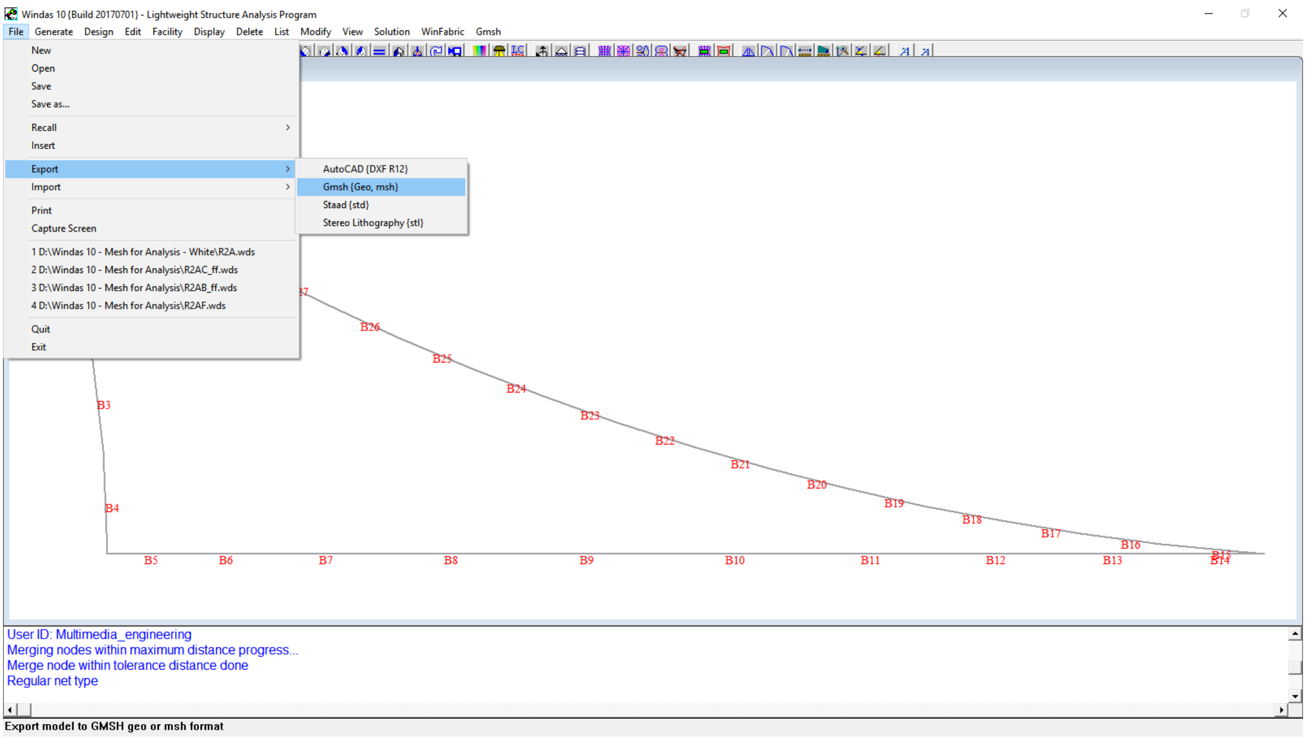


Then, we will get this model with fixed system line as shown below;

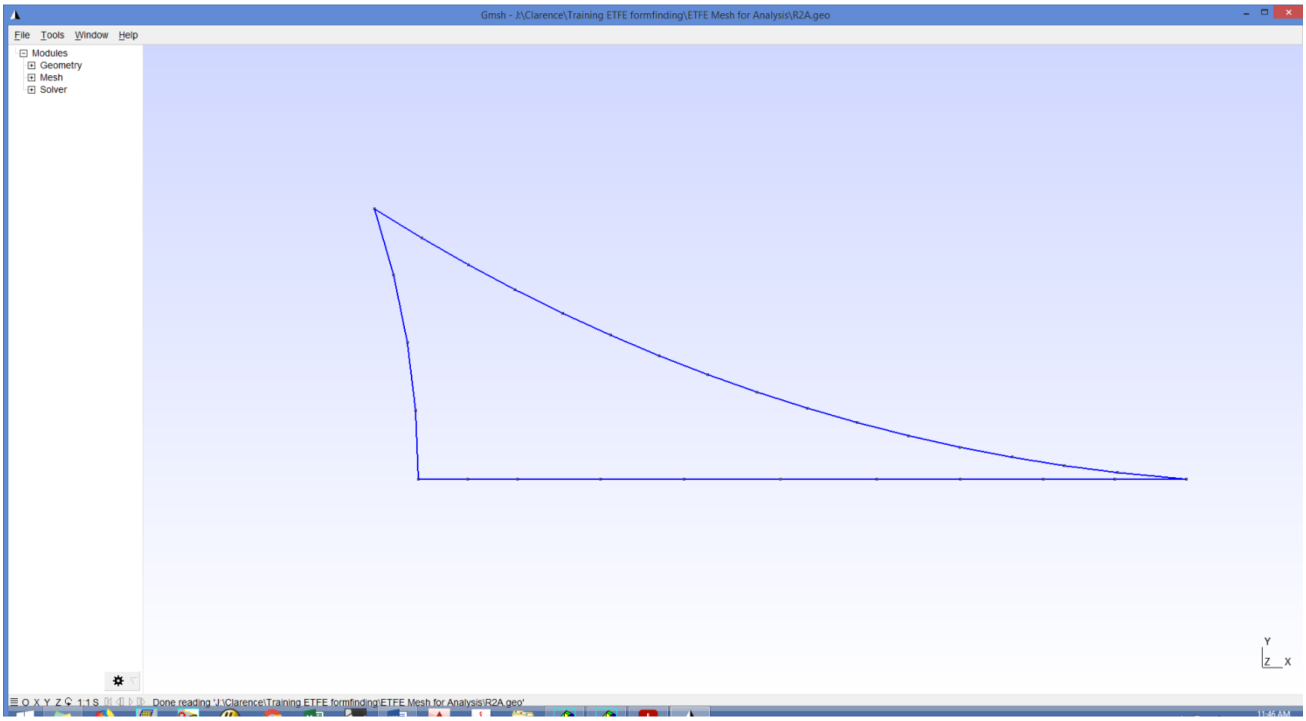




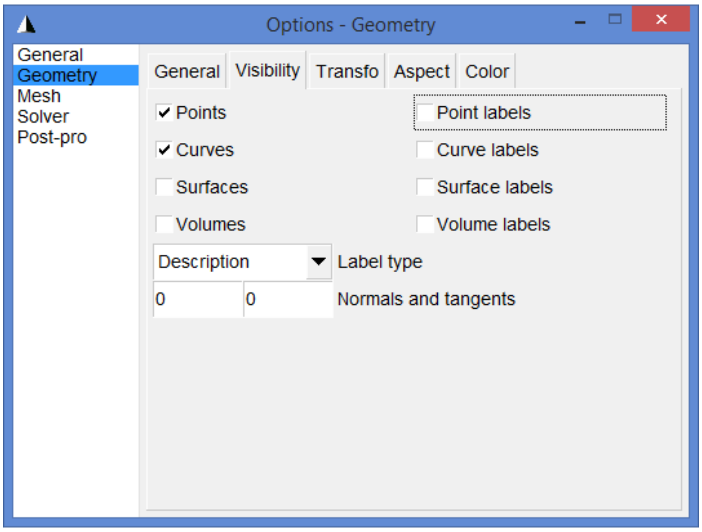
Now, we need to export this model into Gmsh **.geo** format. Click **File | Export | Gmsh**.

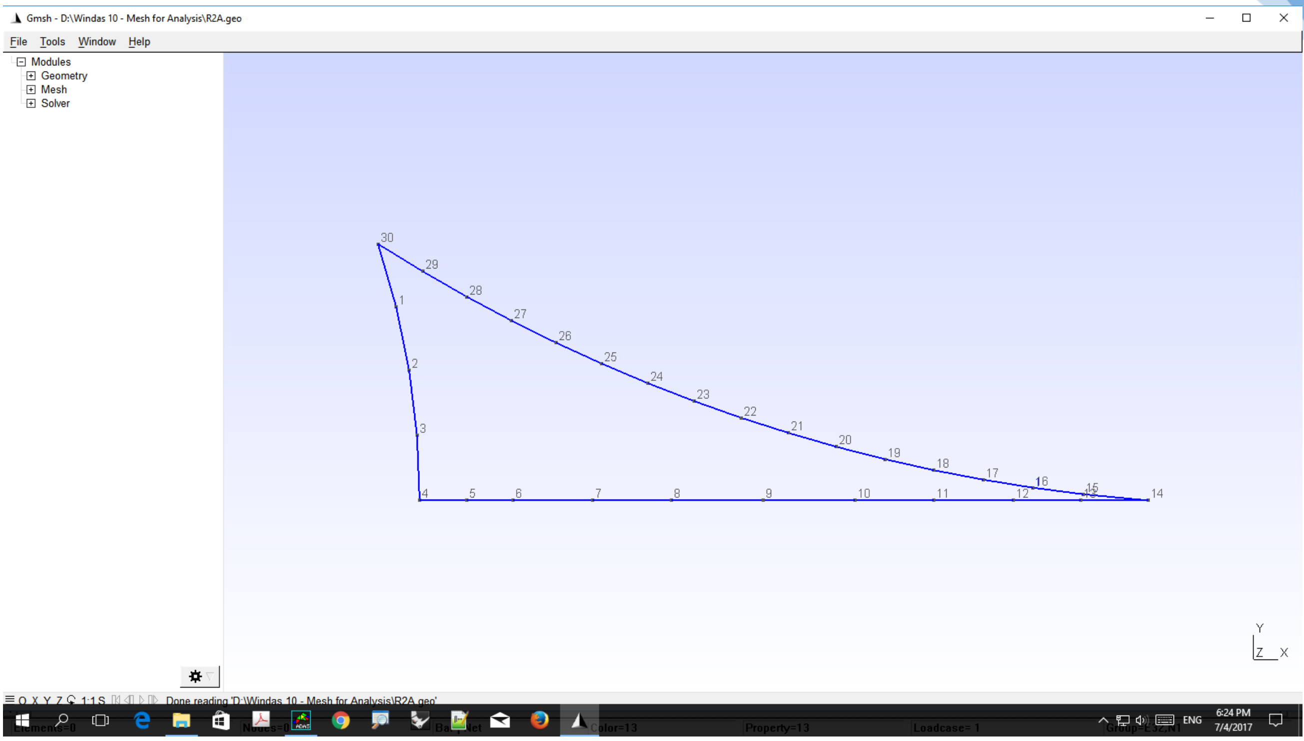


In the next step, we will move into Gmsh software interface. Go back to your working folder and find the exported **.geo** file. Click right on this file, choose open with, choose **gmsh.exe.** If you can not find this software in the default program, click **look for another app in this PC** | **(C:) | ADAS | Gmsh | gmsh.exe | open**. The program should be able to run and show the default interface.

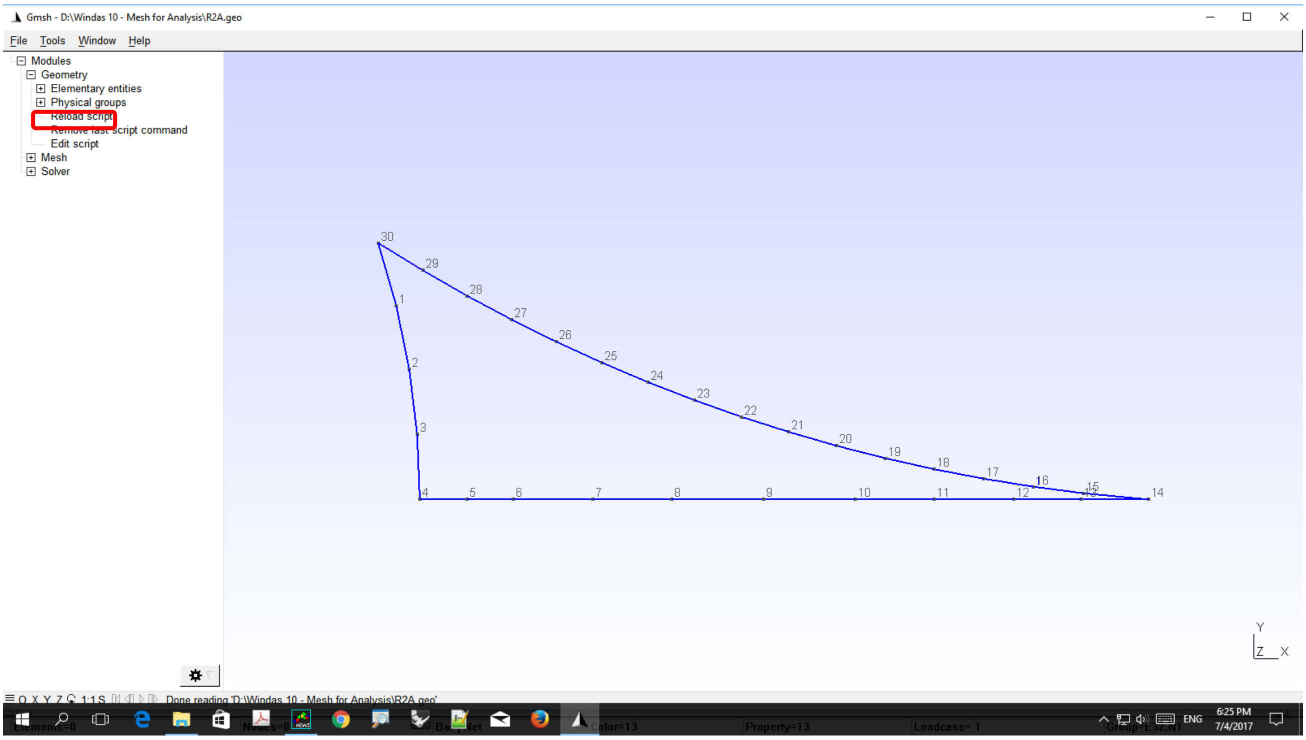


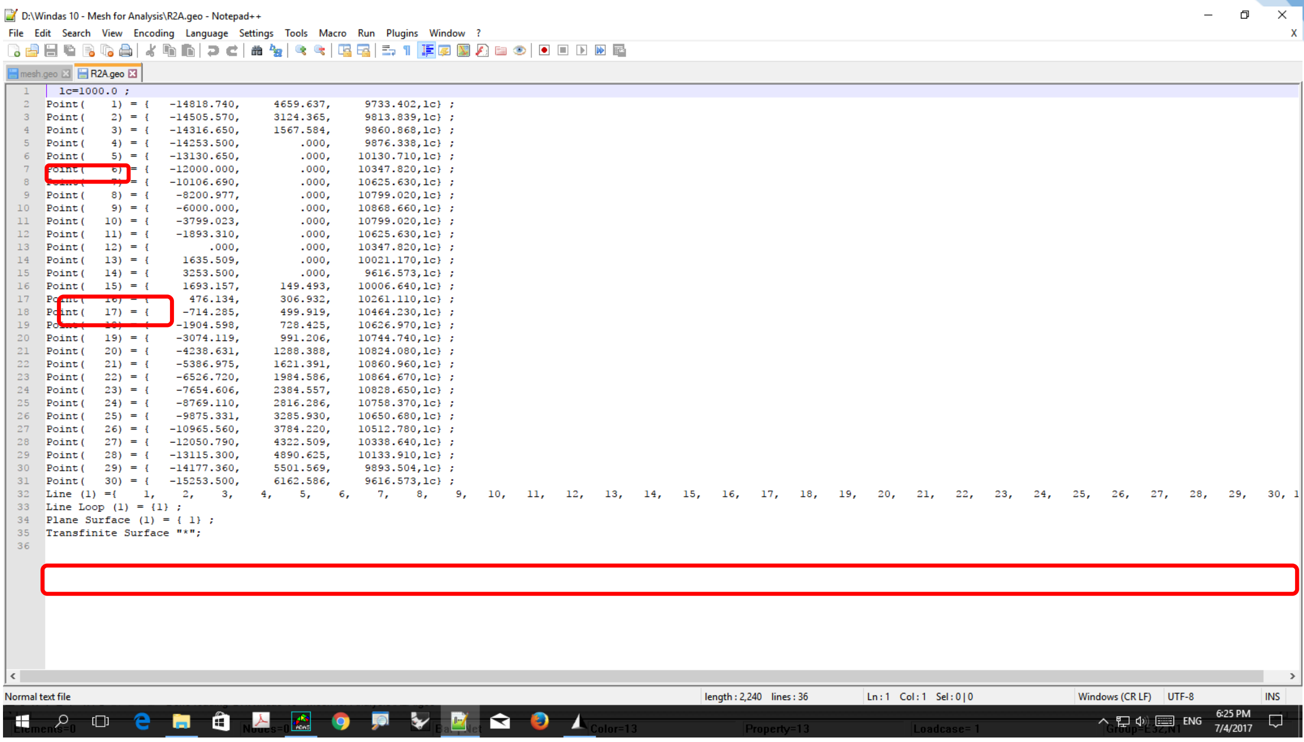
For convenience, we can choose to show the nodal point numbers by clicking **Tools | Options |** tick on **Point Labels.**





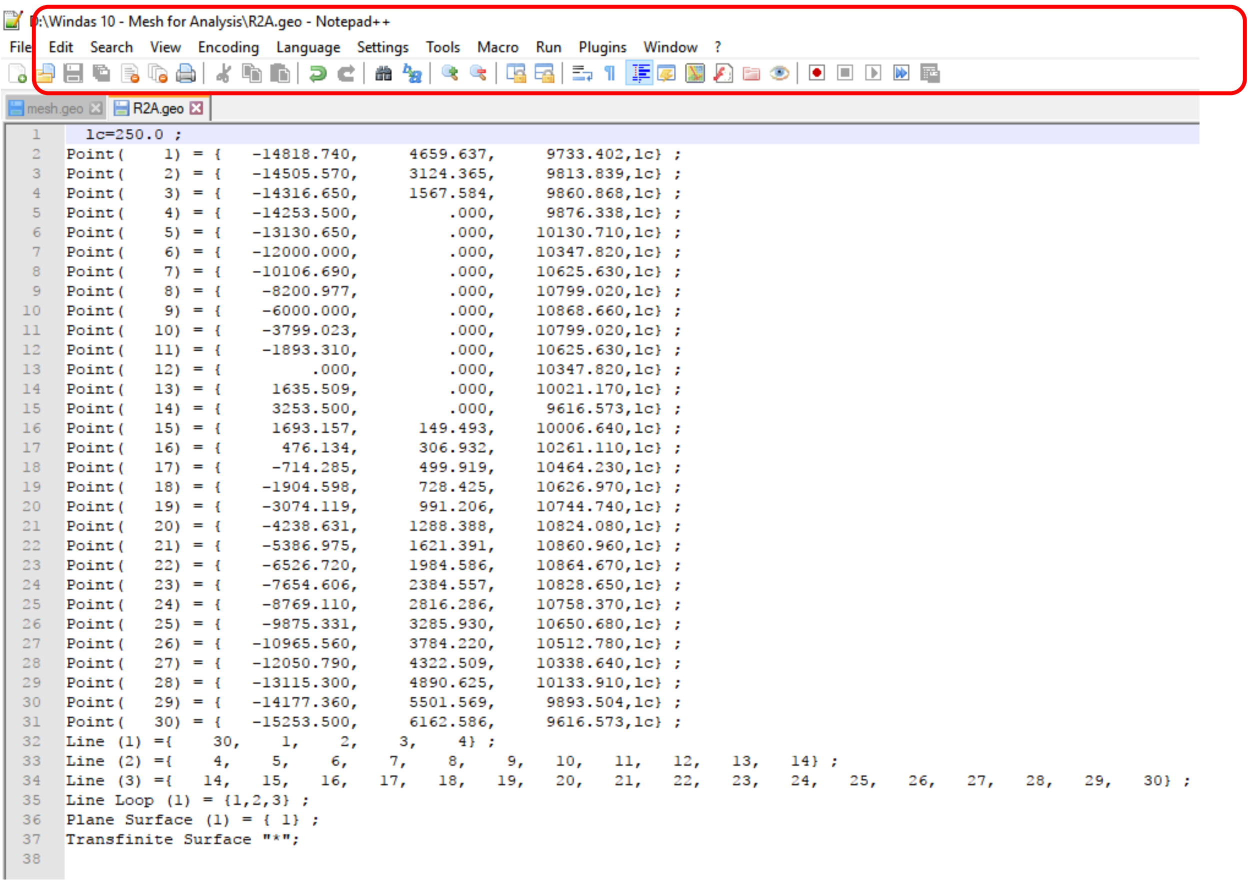
After this, click **Modules | Geometry | Edit Script** and edit the notepad list **Notepad++**.





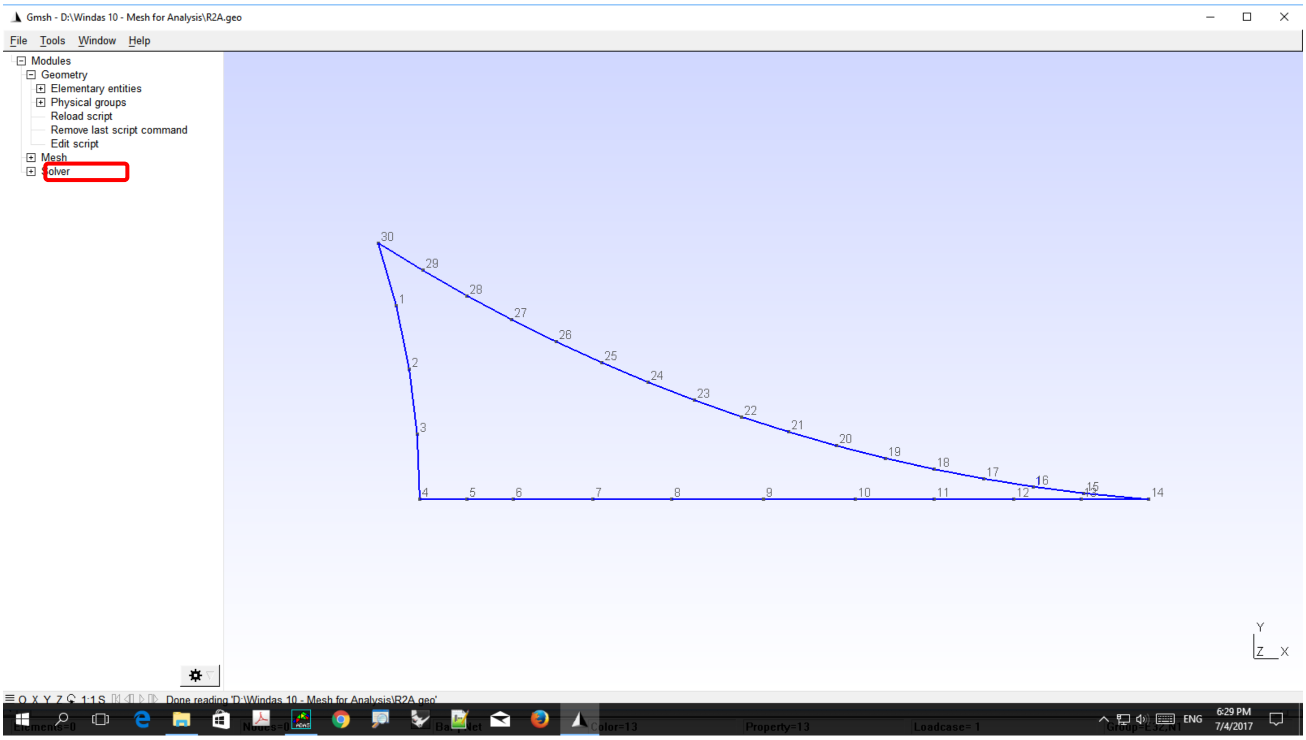
```
lc=1000.0 ;
Point(    1) = {   -14818.740,     4659.637,     9733.402,lc} ;
Point(    2) = {   -14505.570,     3124.365,     9813.839,lc} ;
Point(    3) = {   -14316.650,     1567.584,     9860.868,lc} ;
Point(    4) = {   -14253.500,         .000,     9876.338,lc} ;
Point(    5) = {   -13130.650,         .000,    10130.710,lc} ;
Point(    6) = {   -12000.000,         .000,    10347.820,lc} ;
Point(    7) = {   -10106.690,         .000,    10625.630,lc} ;
Point(    8) = {    -8200.977,         .000,    10799.020,lc} ;
Point(    9) = {    -6000.000,         .000,    10868.660,lc} ;
Point(   10) = {    -3799.023,         .000,    10799.020,lc} ;
Point(   11) = {    -1893.310,         .000,    10625.630,lc} ;
Point(   12) = {         .000,         .000,    10347.820,lc} ;
Point(   13) = {     1635.509,         .000,    10021.170,lc} ;
Point(   14) = {     3253.500,         .000,     9616.573,lc} ;
Point(   15) = {     1693.157,      149.493,    10006.640,lc} ;
Point(   16) = {      476.134,      306.932,    10261.110,lc} ;
Point(   17) = {     -714.285,      499.919,    10464.230,lc} ;
Point(   18) = {    -1904.598,      728.425,    10626.970,lc} ;
Point(   19) = {    -3074.119,      991.206,    10744.740,lc} ;
Point(   20) = {    -4238.631,     1288.388,    10824.080,lc} ;
Point(   21) = {    -5386.975,     1621.391,    10860.960,lc} ;
Point(   22) = {    -6526.720,     1984.586,    10864.670,lc} ;
Point(   23) = {    -7654.606,     2384.557,    10828.650,lc} ;
Point(   24) = {    -8769.110,     2816.286,    10758.370,lc} ;
Point(   25) = {    -9875.331,     3285.930,    10650.680,lc} ;
Point(   26) = {   -10965.560,     3784.220,    10512.780,lc} ;
Point(   27) = {   -12050.790,     4322.509,    10338.640,lc} ;
Point(   28) = {   -13115.300,     4890.625,    10133.910,lc} ;
Point(   29) = {   -14177.360,     5501.569,     9893.504,lc} ;
Point(   30) = {   -15253.500,     6162.586,     9616.573,lc} ;
Line (1) ={    1,    2,    3,    4,    5,    6,    7,    8,    9,   10,   11,   12,   13,   14,   15,   16,   17,   18,   19,   20,   21,   22,   23,   24,   25,   26,   27,   28,   29,   30, 1
Line Loop (1) = {1} ;
Plane Surface (1) = { 1} ;
Transfinite Surface "*";
```

In the **Notepad++**, set the **characteristic length (lc)** into **250**. Then break and create lines by editing the script, as for this case; **Line(1), Line(2), Line(3)**. Edit the loop according the amount of lines; **Line Loop {1,2,3}** . Proceed as shown below;

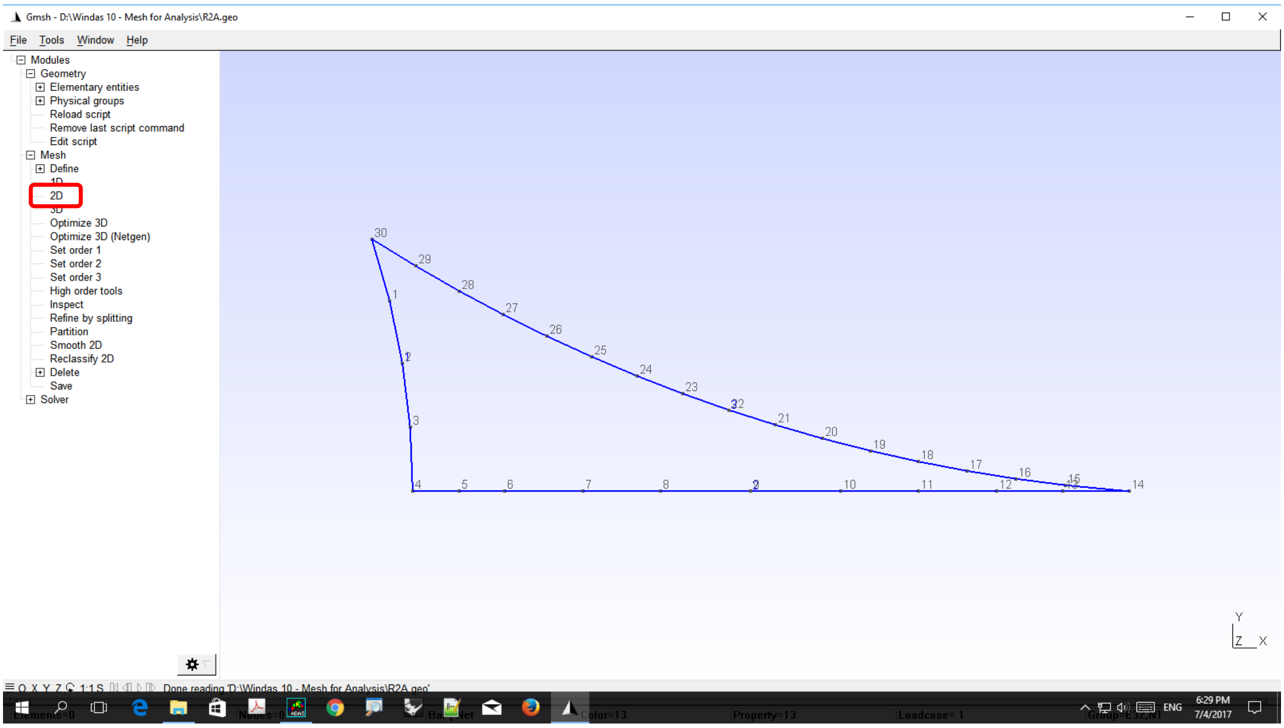


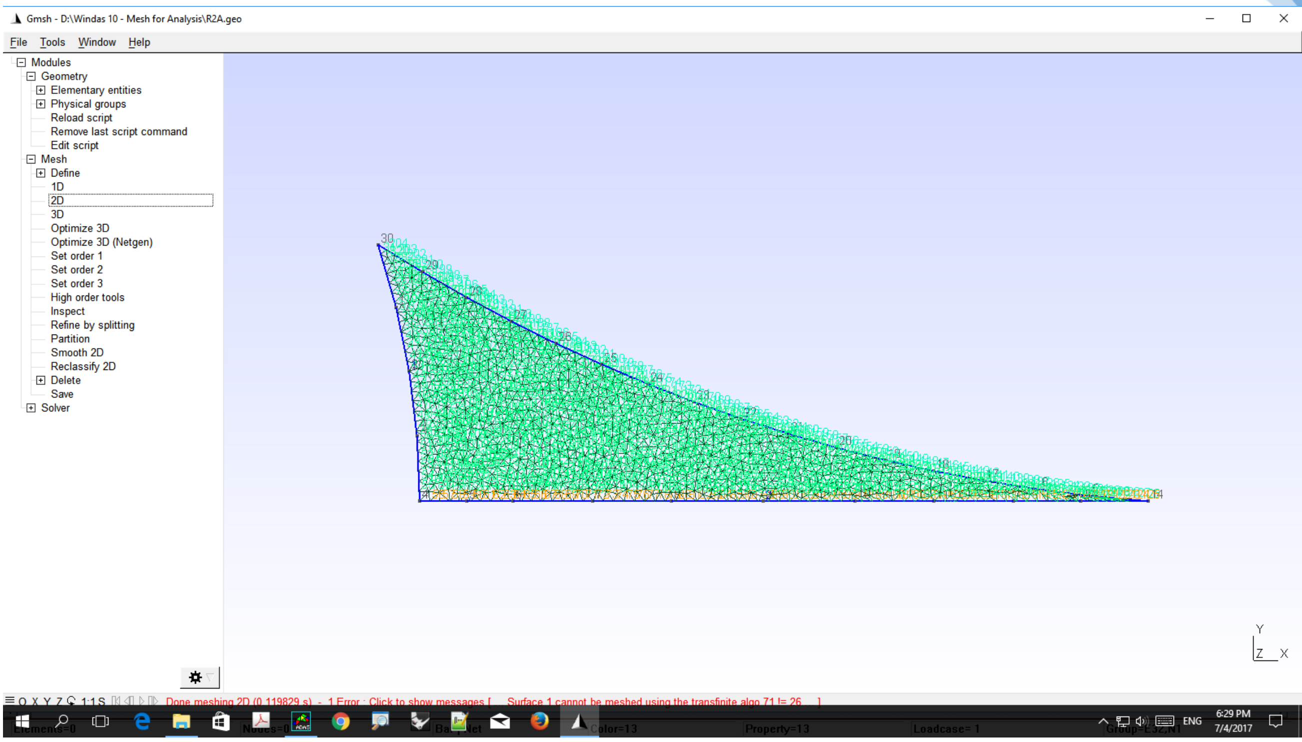
```
lc=250.0 ;
Point(    1) = {   -14818.740,     4659.637,     9733.402,lc} ;
Point(    2) = {   -14505.570,     3124.365,     9813.839,lc} ;
Point(    3) = {   -14316.650,     1567.584,     9860.868,lc} ;
Point(    4) = {   -14253.500,         .000,     9876.338,lc} ;
Point(    5) = {   -13130.650,         .000,    10130.710,lc} ;
Point(    6) = {   -12000.000,         .000,    10347.820,lc} ;
Point(    7) = {   -10106.690,         .000,    10625.630,lc} ;
Point(    8) = {    -8200.977,         .000,    10799.020,lc} ;
Point(    9) = {    -6000.000,         .000,    10868.660,lc} ;
Point(   10) = {    -3799.023,         .000,    10799.020,lc} ;
Point(   11) = {    -1893.310,         .000,    10625.630,lc} ;
Point(   12) = {         .000,         .000,    10347.820,lc} ;
Point(   13) = {     1635.509,         .000,    10021.170,lc} ;
Point(   14) = {     3253.500,         .000,     9616.573,lc} ;
Point(   15) = {     1693.157,      149.493,    10006.640,lc} ;
Point(   16) = {      476.134,      306.932,    10261.110,lc} ;
Point(   17) = {     -714.285,      499.919,    10464.230,lc} ;
Point(   18) = {    -1904.598,      728.425,    10626.970,lc} ;
Point(   19) = {    -3074.119,      991.206,    10744.740,lc} ;
Point(   20) = {    -4238.631,     1288.388,    10824.080,lc} ;
Point(   21) = {    -5386.975,     1621.391,    10860.960,lc} ;
Point(   22) = {    -6526.720,     1984.586,    10864.670,lc} ;
Point(   23) = {    -7654.606,     2384.557,    10828.650,lc} ;
Point(   24) = {    -8769.110,     2816.286,    10758.370,lc} ;
Point(   25) = {    -9875.331,     3285.930,    10650.680,lc} ;
Point(   26) = {   -10965.560,     3784.220,    10512.780,lc} ;
Point(   27) = {   -12050.790,     4322.509,    10338.640,lc} ;
Point(   28) = {   -13115.300,     4890.625,    10133.910,lc} ;
Point(   29) = {   -14177.360,     5501.569,     9893.504,lc} ;
Point(   30) = {   -15253.500,     6162.586,     9616.573,lc} ;
Line (1) ={    30,    1,    2,    3,    4} ;
Line (2) ={    4,    5,    6,    7,    8,    9,   10,   11,   12,   13,   14} ;
Line (3) ={   14,   15,   16,   17,   18,   19,   20,   21,   22,   23,   24,   25,   26,   27,   28,   29,   30} ;
Line Loop (1) = {1,2,3} ;
Plane Surface (1) = { 1} ;
Transfinite Surface "*";
```

File | Save the edited notepad and switch back to Gmsh interface. Then click on **Modules | Geometry | Reload Script** to update the script.

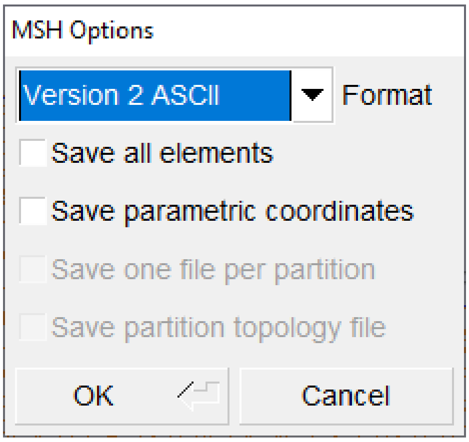


Click **Modules | Mesh | 2D** to generate 2D Mesh.



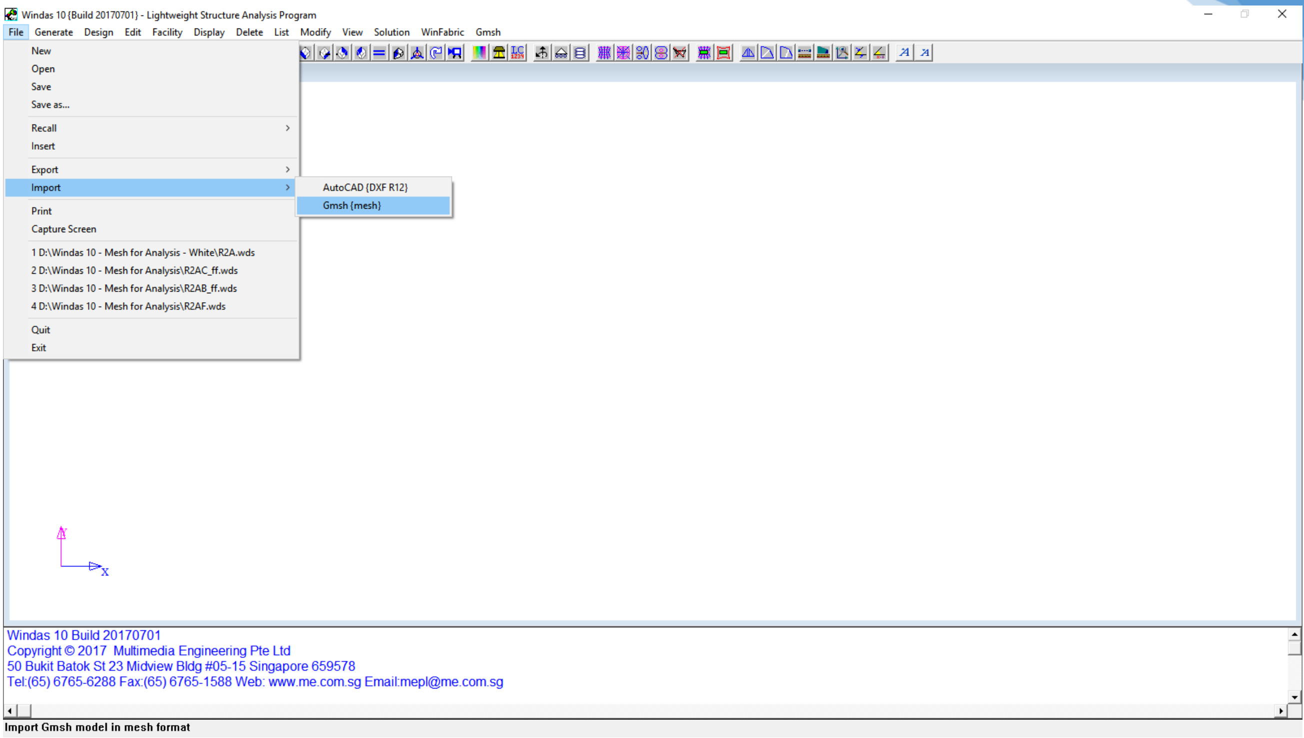


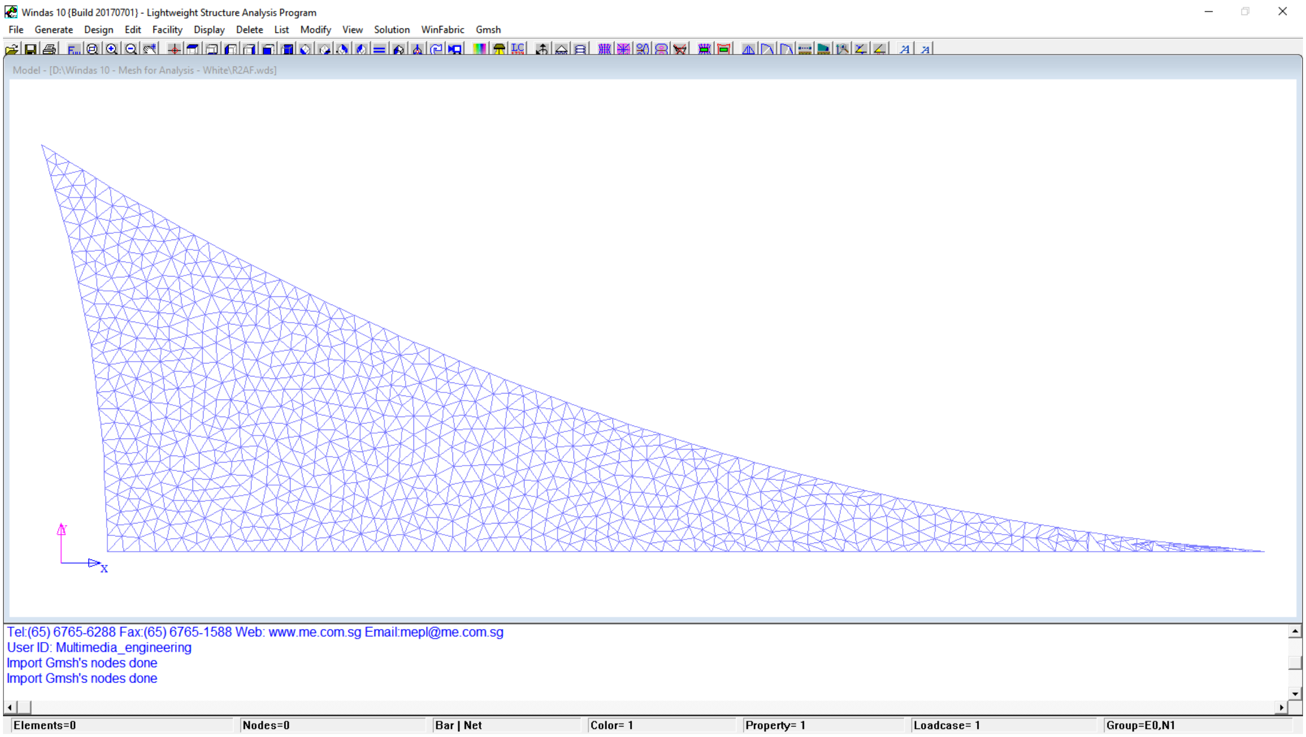
Click **File | Export |** choose .**msh** | choose **Version 2 ASCII** format.



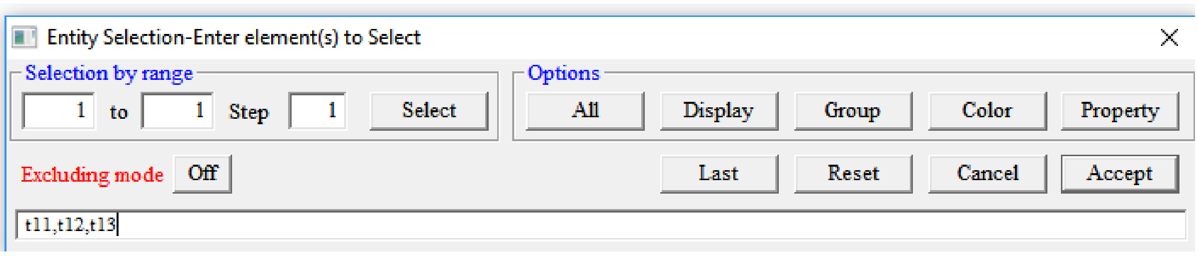
We will have a **.msh** file generated.

After we saved the mesh file in Gmsh, we need to switch back into Windas interface and create a new file by clicking **File | Import | Gmsh{msh}** | choose the **.msh** file. Save the **.wds** file as **filenameFlat.wds** to indicate it is the flat mesh which serves as the base mesh for inflation at later phase.

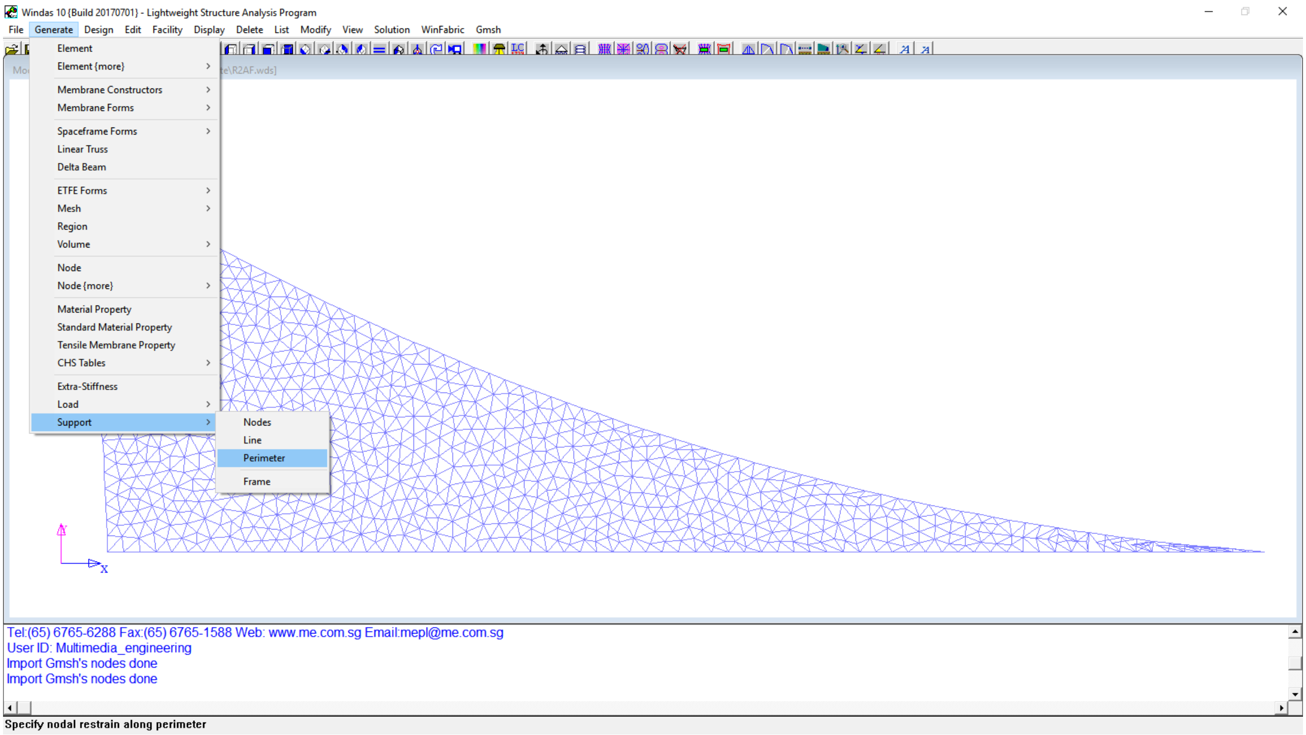


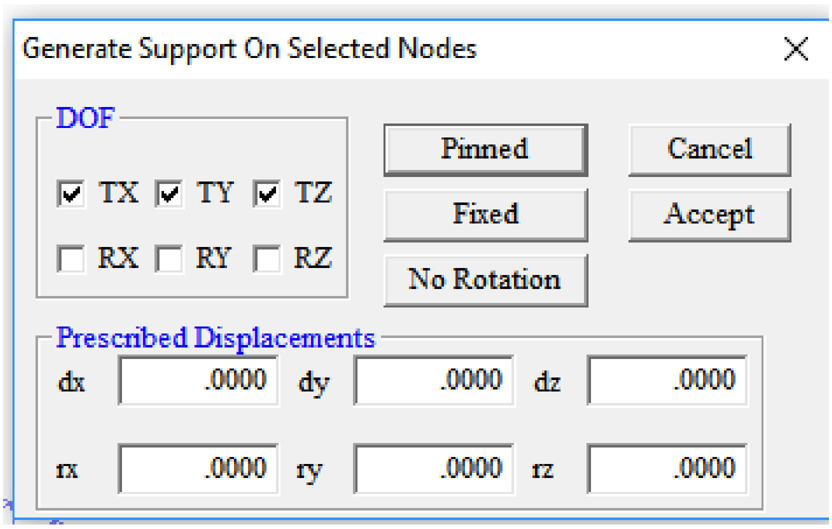


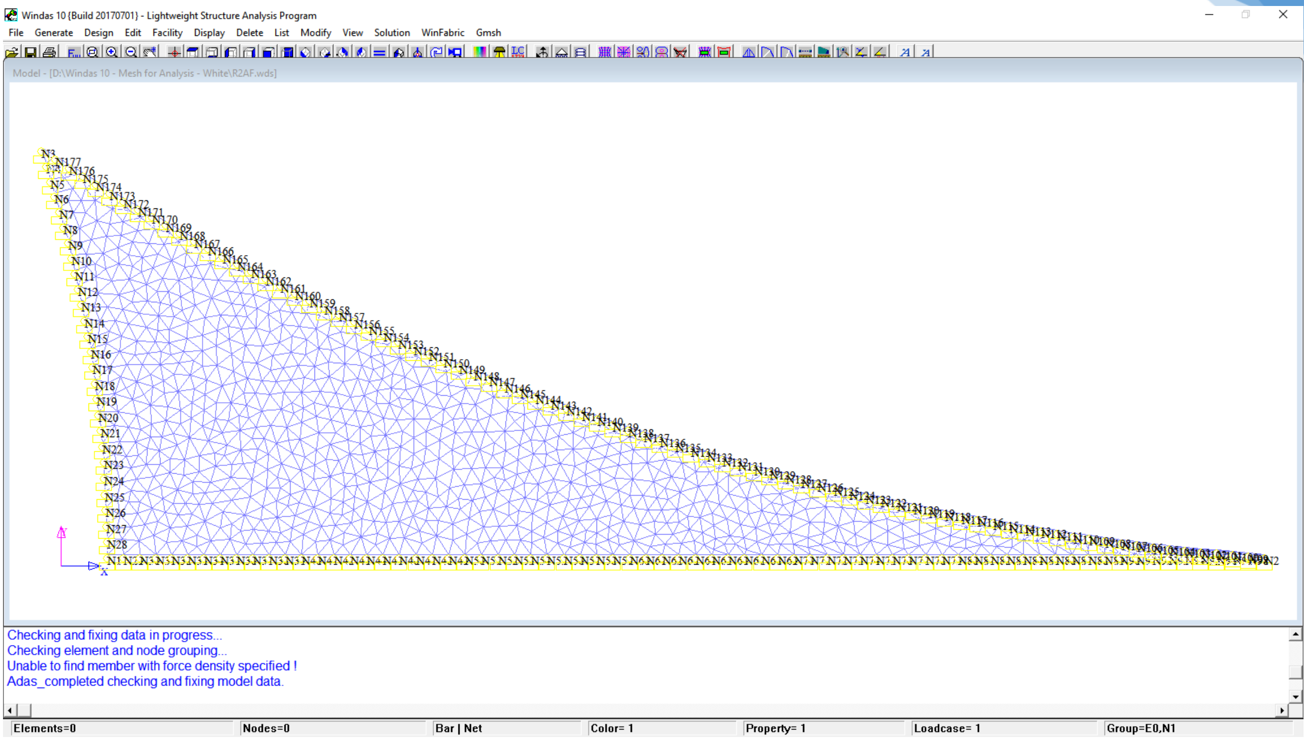
Then, click **Delete | Element** type **t11, t12, t13** and click **Accept**.



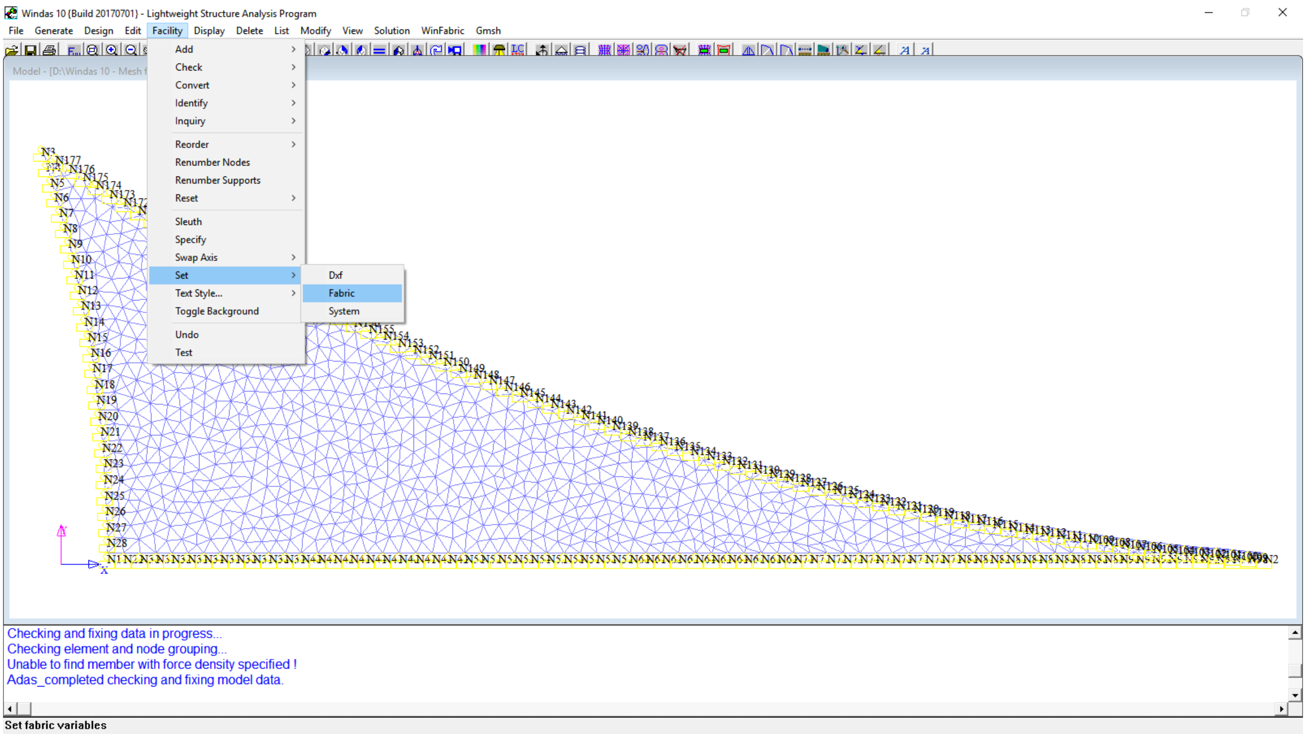
After this, click **Delete | Node | All** to refresh the nodes sequences since we deleted some elements in the previous step. In the next step, we need to have support along the system line since that will be the frame of the ETFE Cushion. Click **Generate | Support | Perimeter** select **Pin** then click **Accept**.

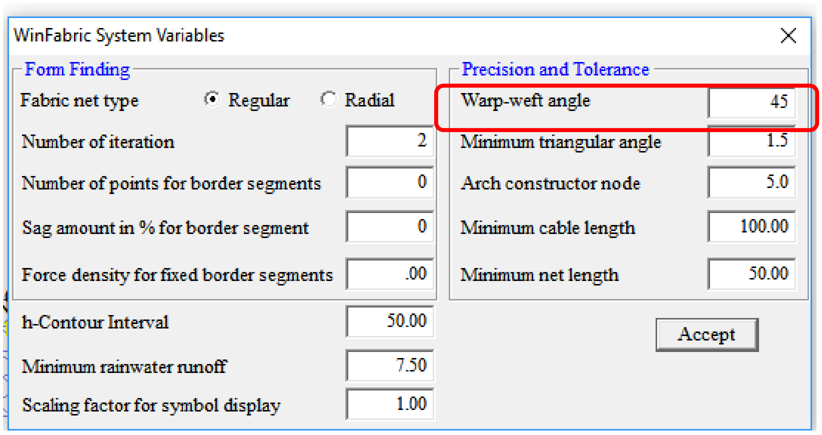




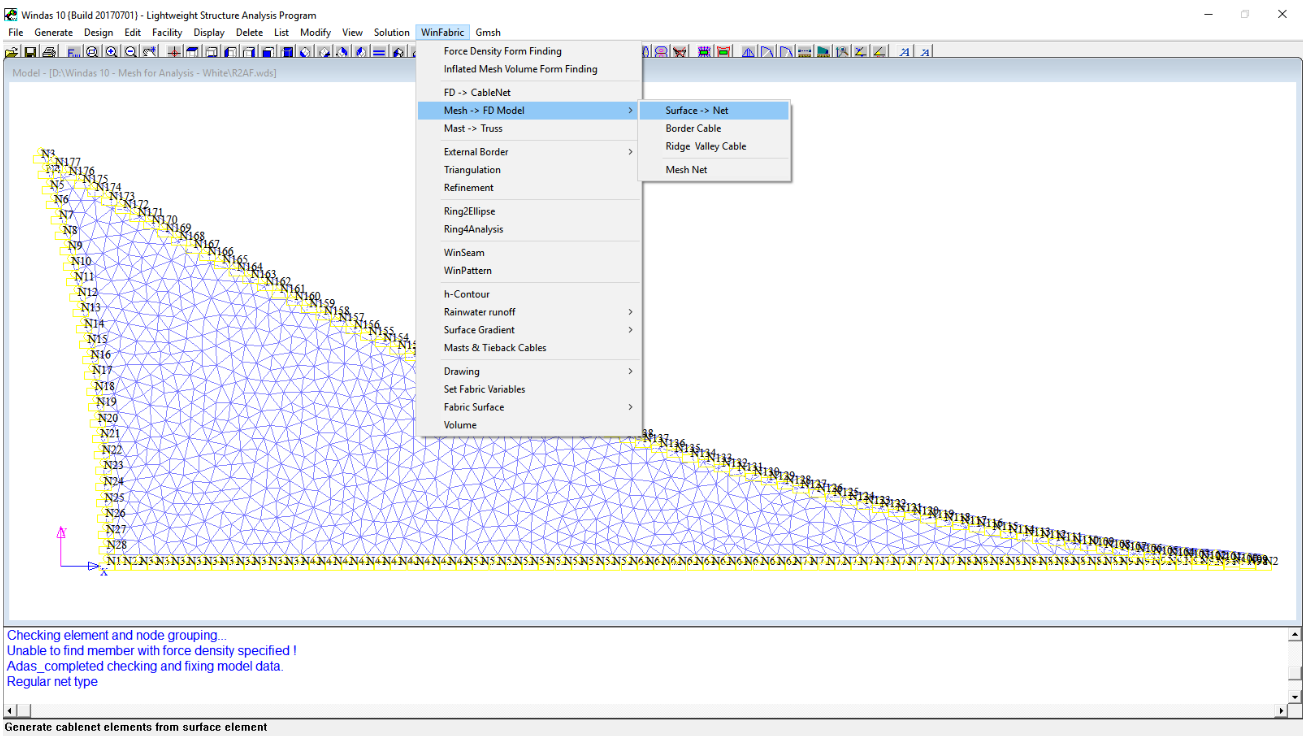


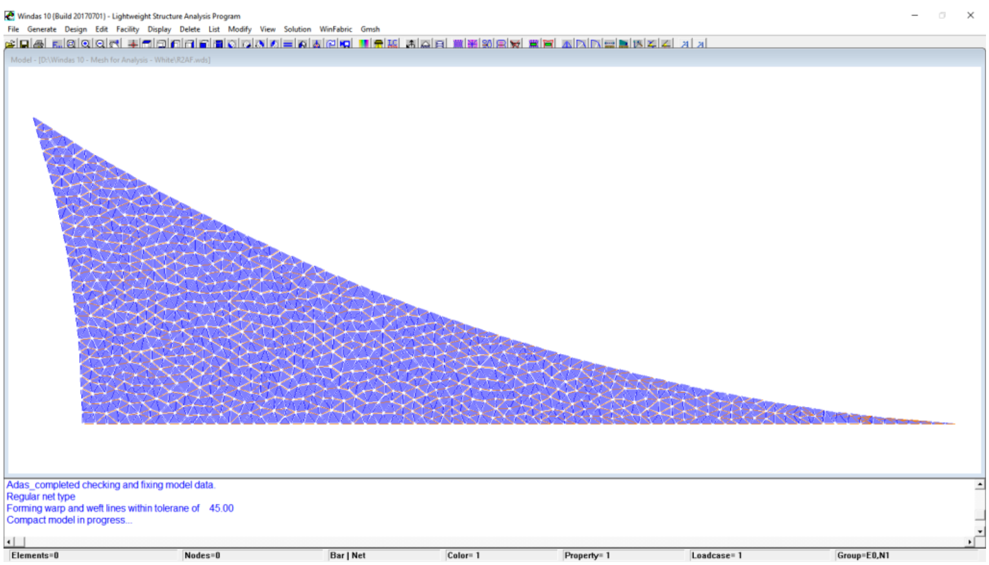
After the support are generated, click **Facility | Set | Fabric** set the **Warp-weft angle** to **45**.



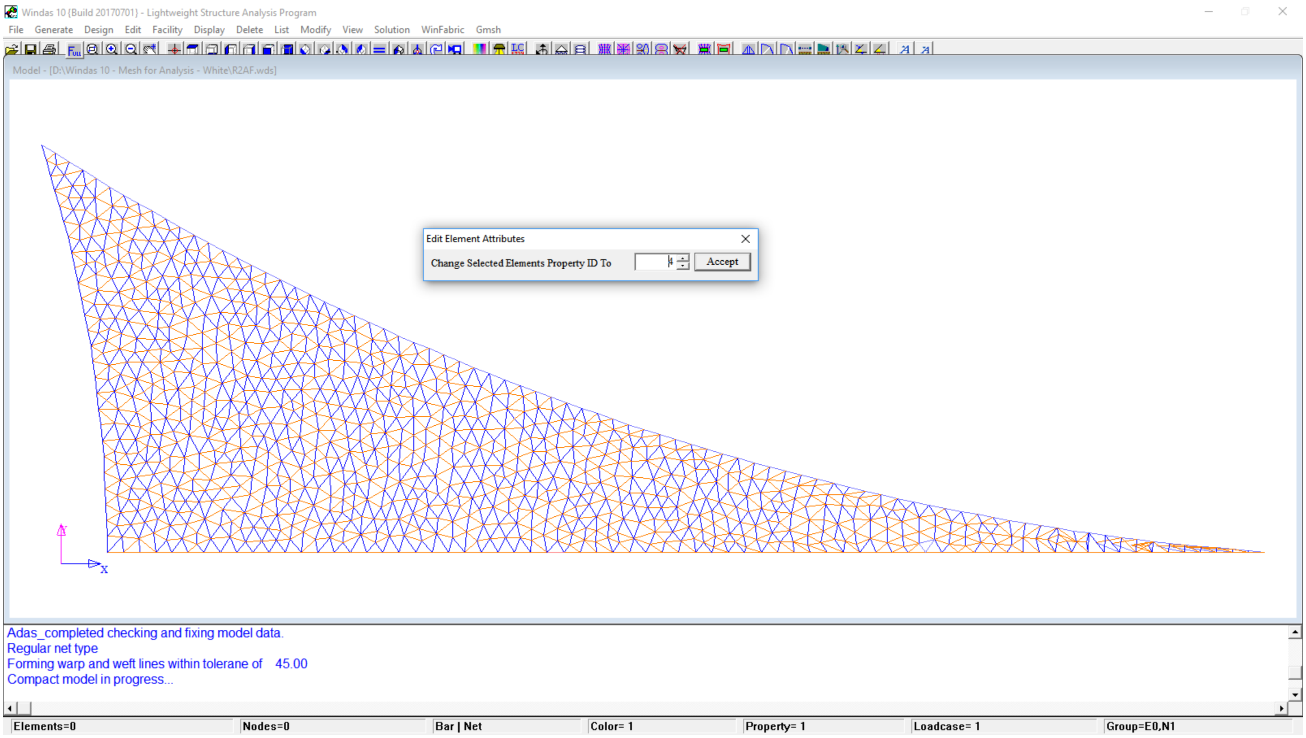


Then, click **WinFabric | Mesh > FD Model | Surface > Net**.

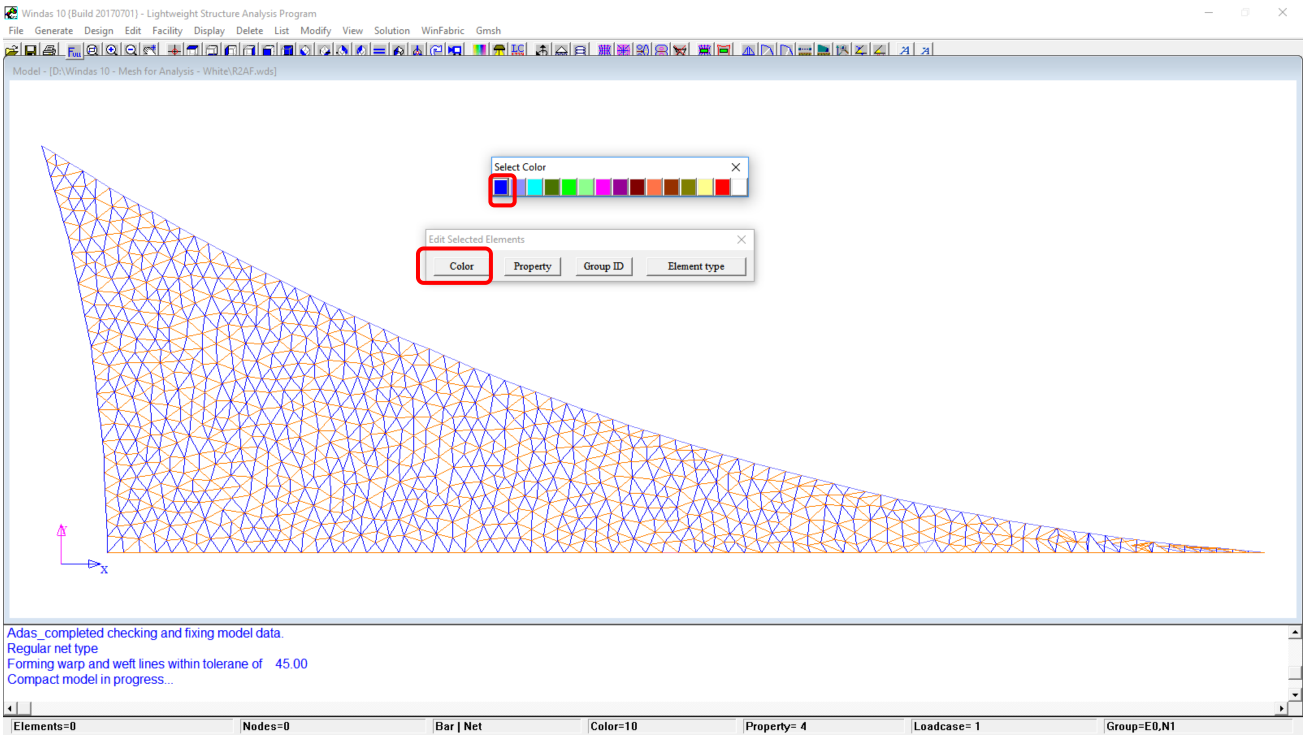


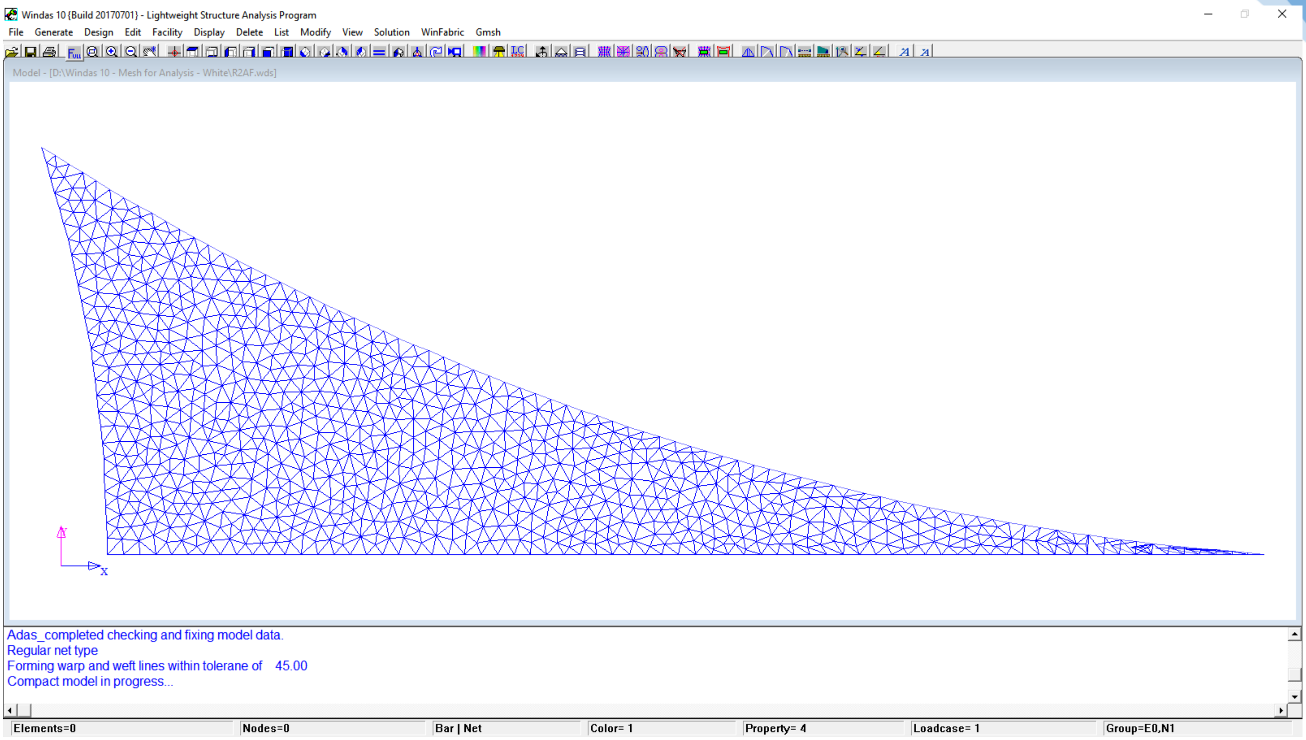


We need to have a flat mesh for inflation. Initially we will treat the mesh as tensile membrane in order to get the mesh shape generated. Click **Edit | Element Attributes** select **All** and change **Property** into **Property 4**.

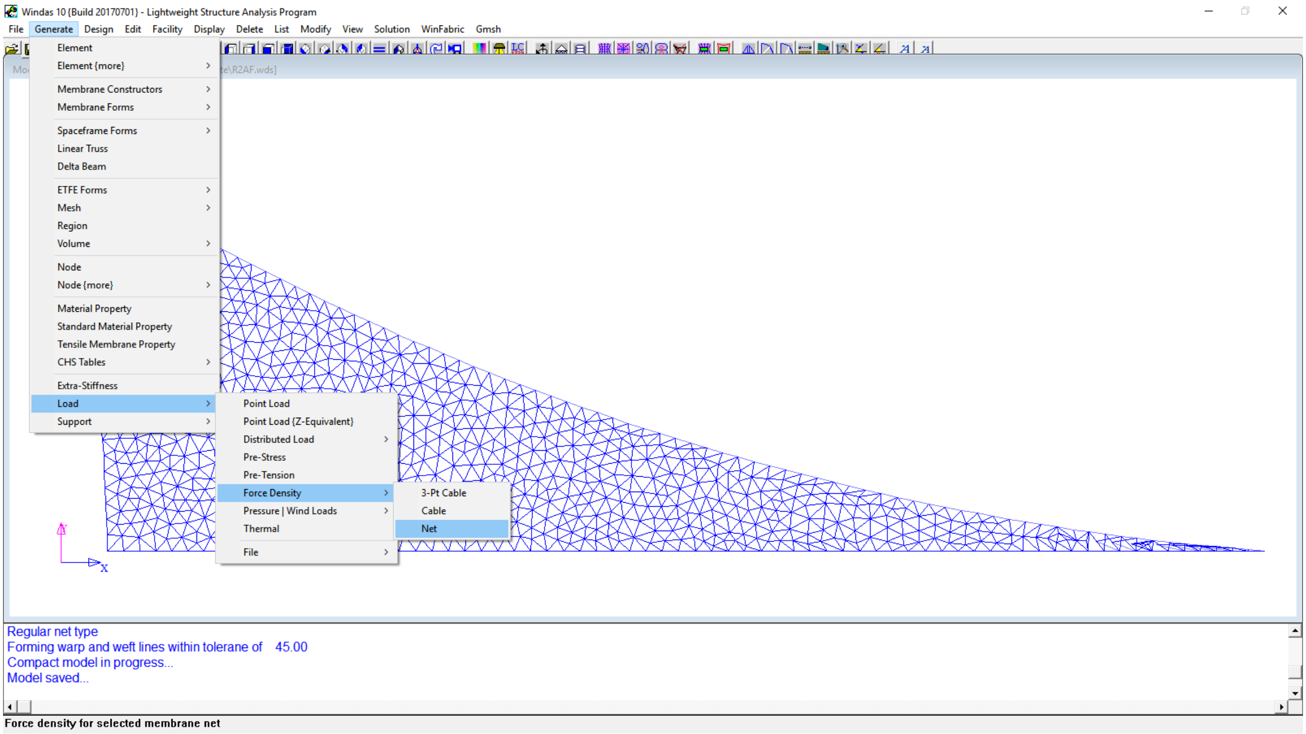


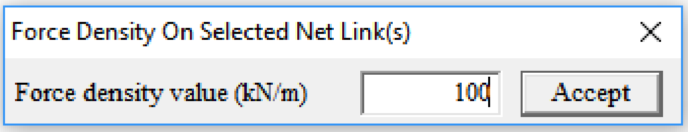
Then, click **Edit | Element Attributes** select **Color 10** and change **Color** into **Color 1**.



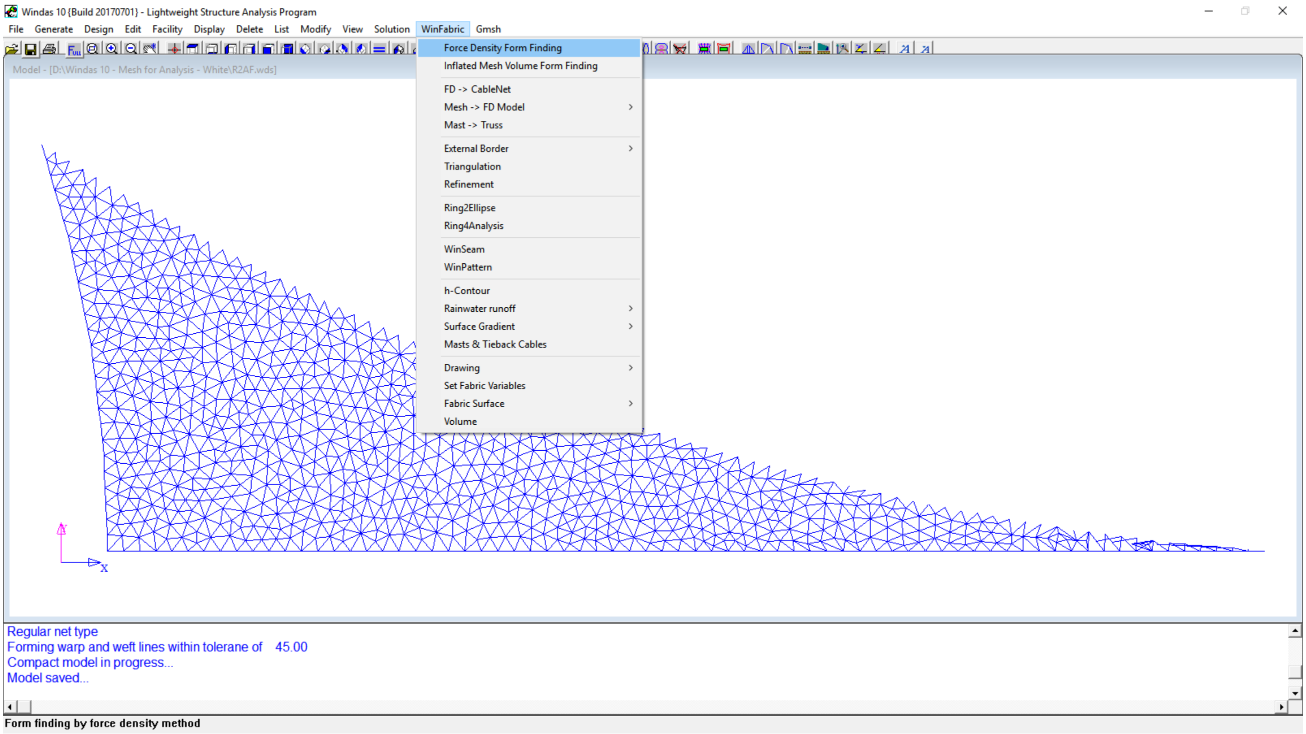


After we change the **Color** and **Property**, click **Generate | Load | Force Density | Net** select **Color 1** and set the pre-stress to **100 kN/m**.

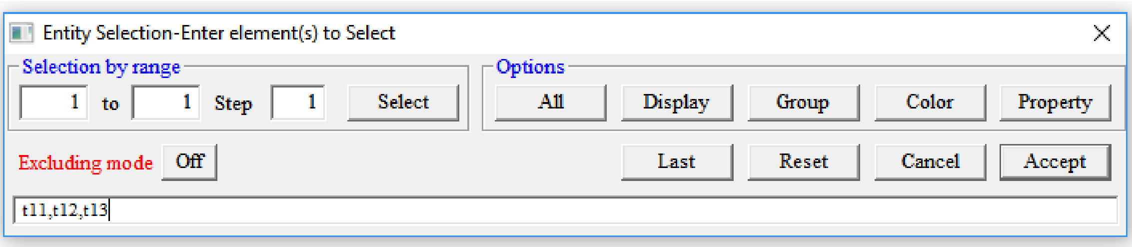




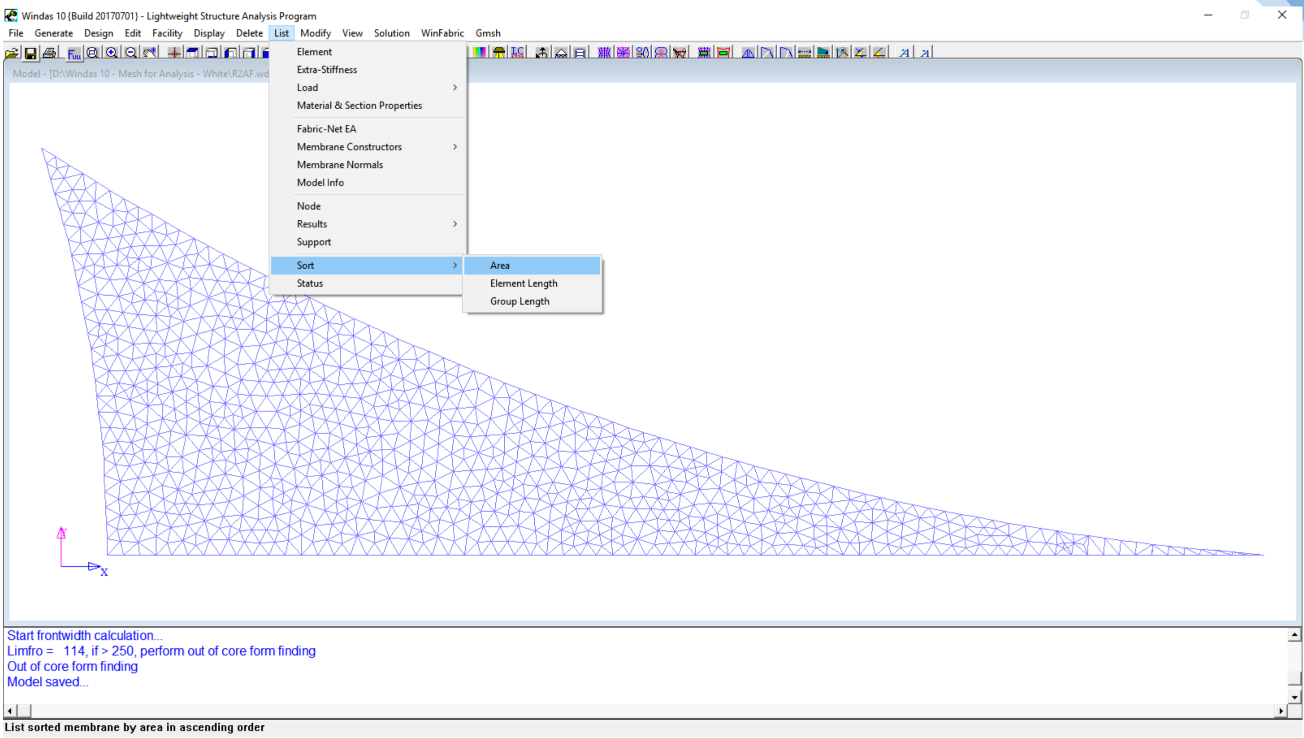
In the next step, click **WinFabric | Force Density Form Finding** then **Accept** with the default parameter.

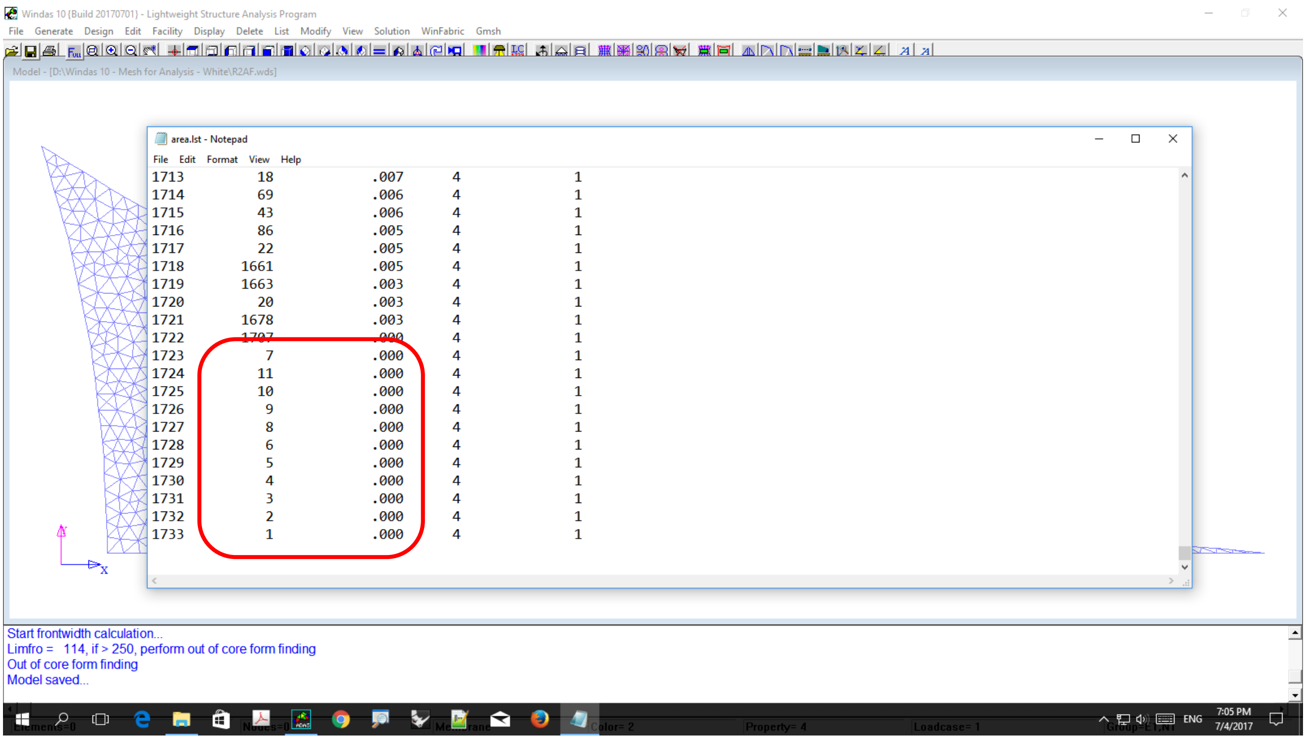


After the form finding, click **Delete | Element** type **t11, t12, t13** and **Accept** to delete the tensile membrane element properties since we intend to treat the mesh as ETFE cushion mesh.

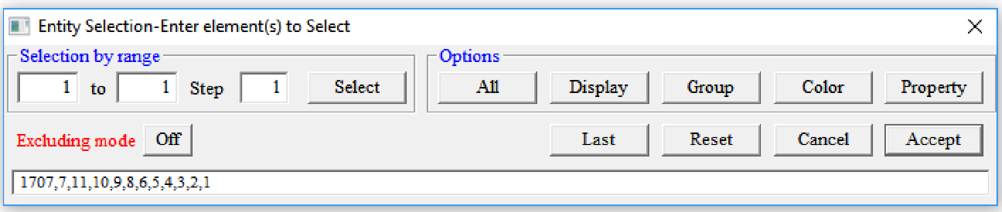


Now, click **Delete | Node** select **All** and then **Accept** to remove unnecessary nodes and refresh the nodes sequence. Click **List | Sort | Surface Area** to check for unusual area (with the value of 0.000), a notepad list will be generated. Please take note for elements ID number which have the area value of 0.000.



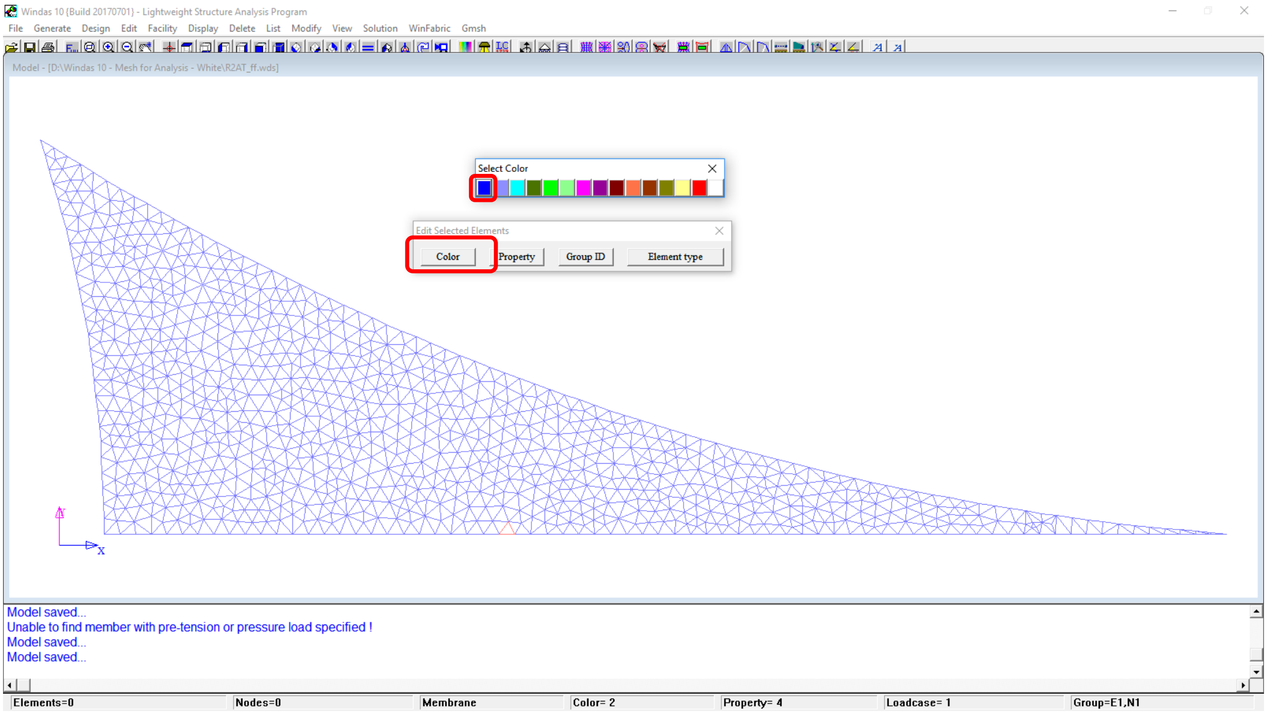


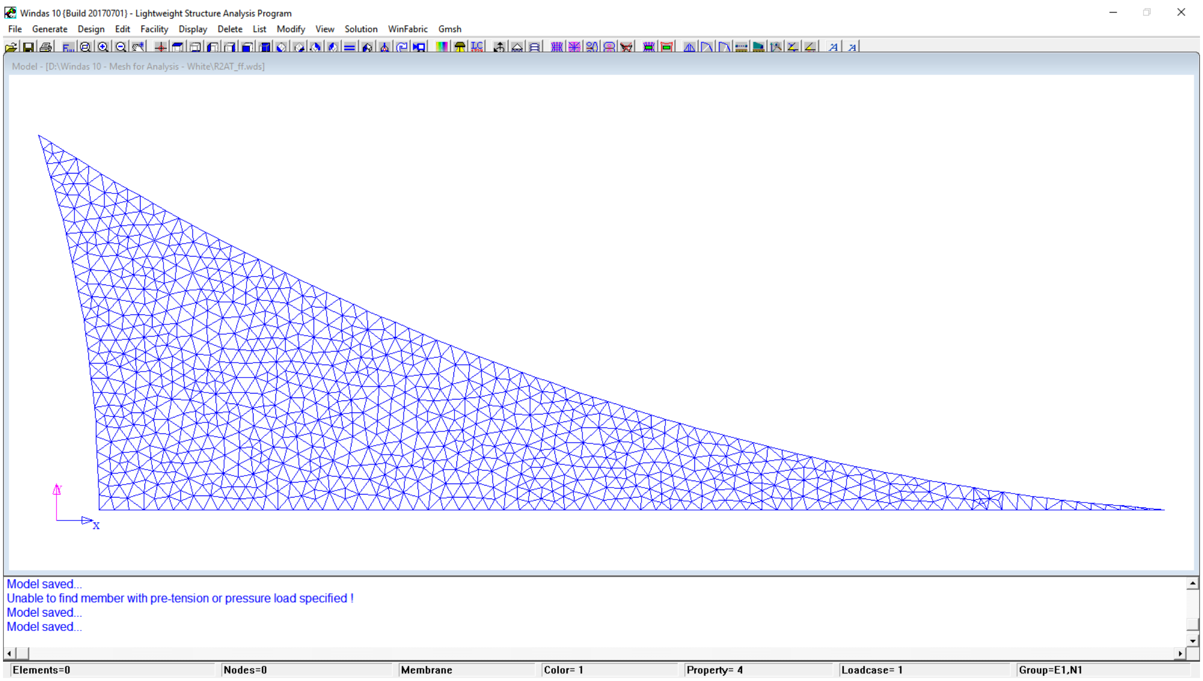
Click **Delete | Element** to delete the elements which have area with 0.000 value. As for this case it is node {N1707, N7, N11, N10, N9, N8, N6, N5, N4, N3, N2, N1}. Then click **Accept**.



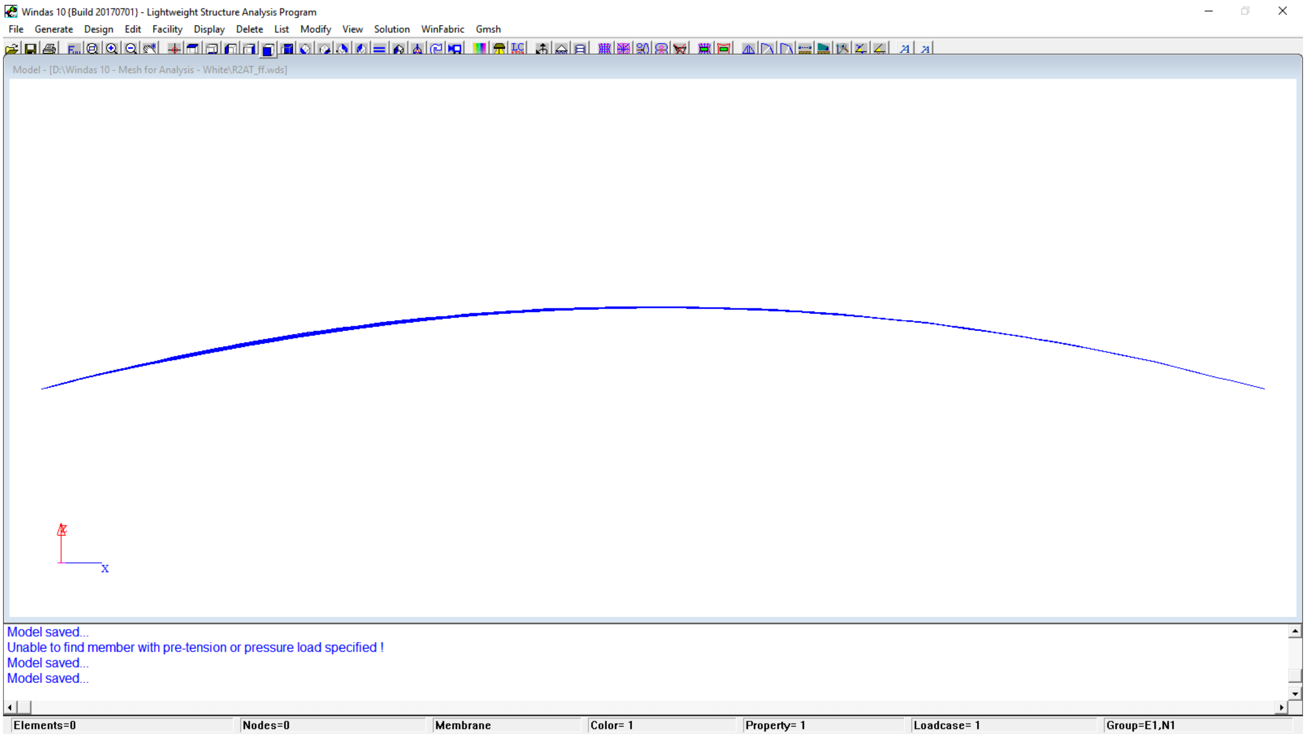
Then, click **Delete | Node** select **All** to refresh the nodes sequence and click **List | Sort | Area** for checking one more time to make sure there is not any area with 0.000 value. Save the file by clicking **File | Save** and then save it again as **filenameTop_ff.wds** by clicking **File | Save As** for next step where we will inflate the mesh into top layer part of cushion.

In order to do inflation for **Top** Layer, click **Edit | Element Attributes** select **All** and set **Color** into **Color 1** (Color 1 is for Top Layer).

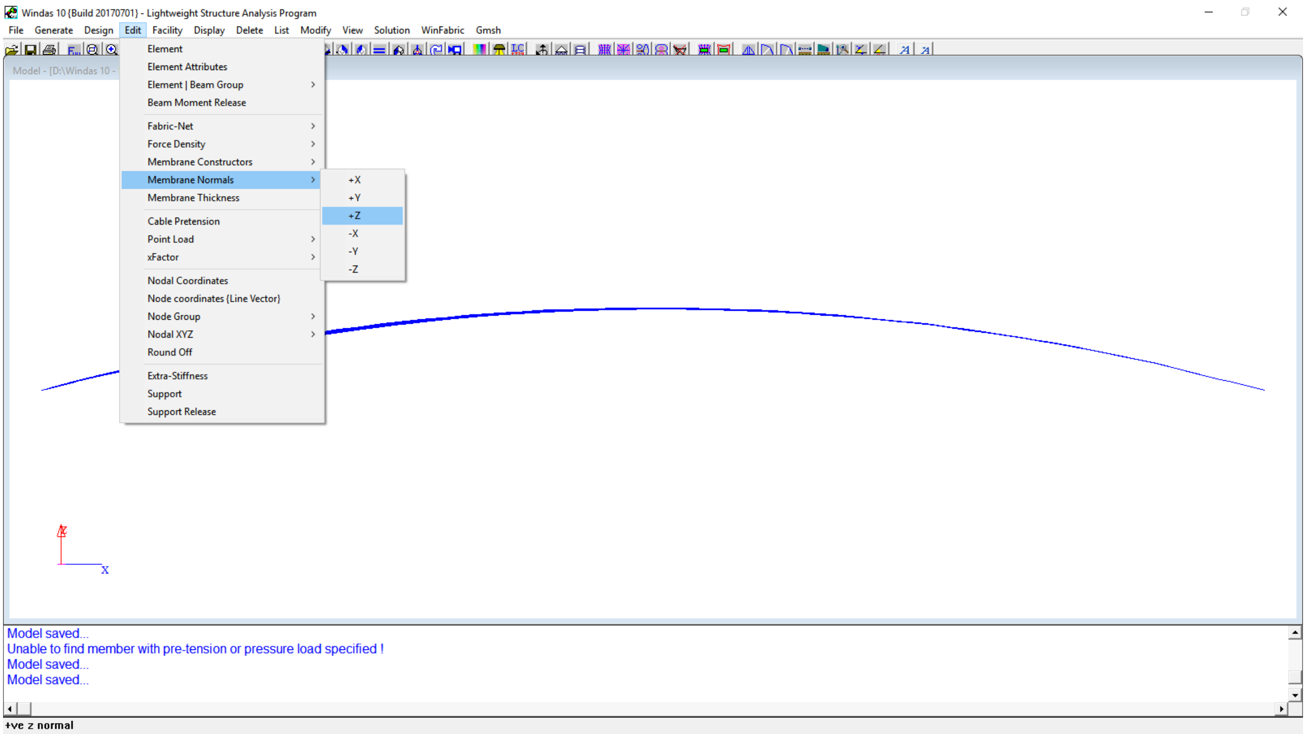


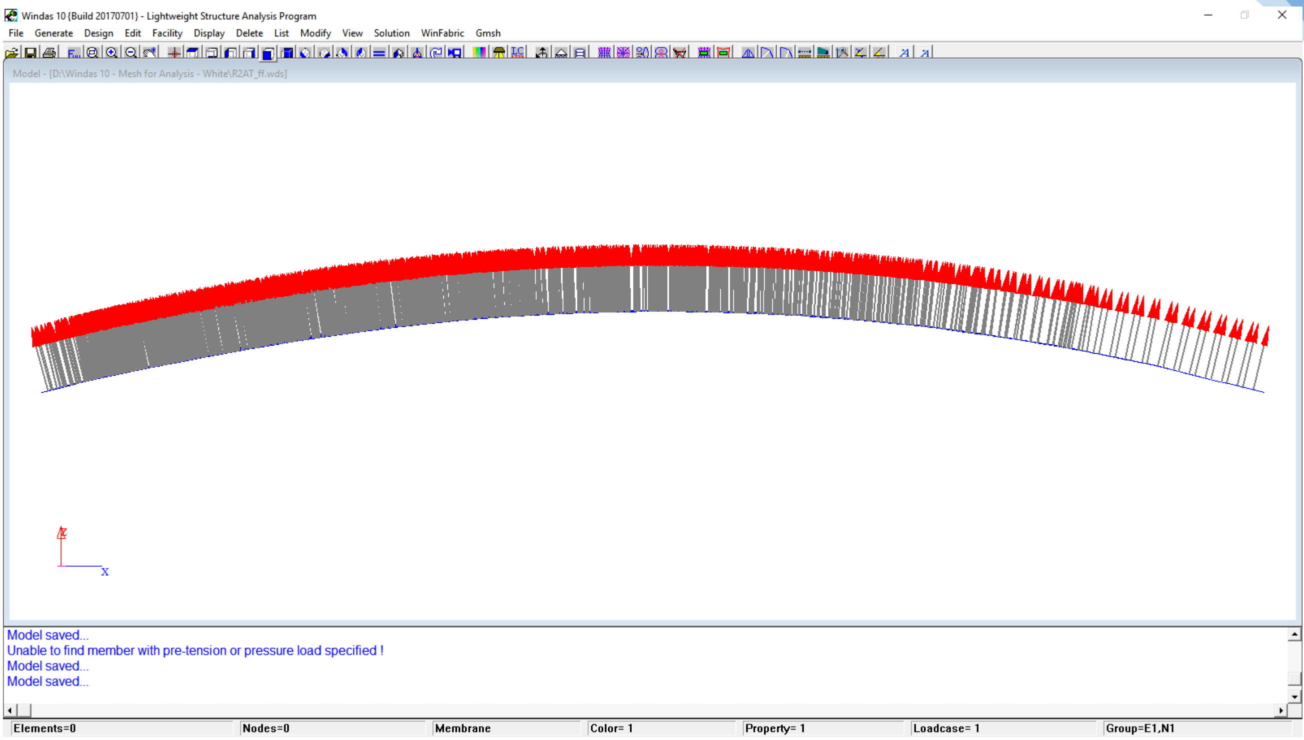


After this, click **Edit | Element Attributes** select **All** and set **Property** into **Property 1** (Property 1 is for Top Layer). Then, change the view into **Front View** for our convenience in checking the mesh.

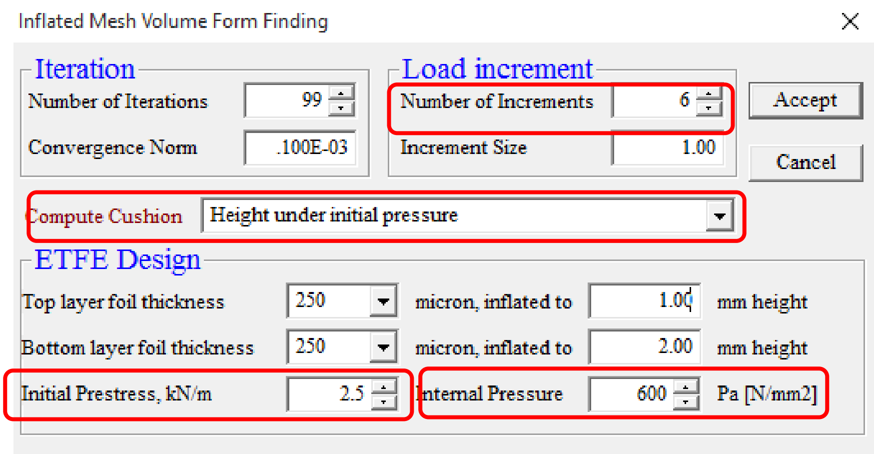


Change the membrane normals (for Top Layer it is **+Z**) by clicking **Edit | Surface Normals | +Z** and select **All** then click **Accept**.

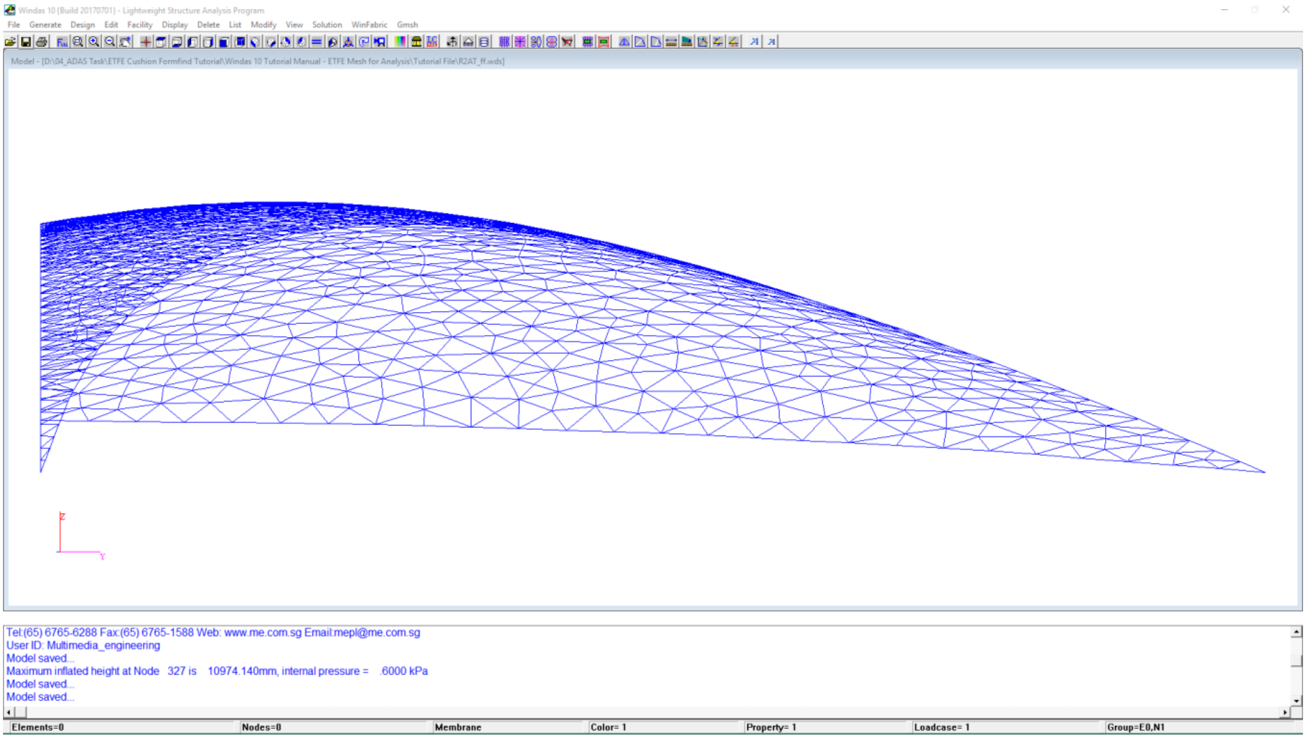




Click **WinFabric | Inflated Mesh Volume Form Finding** set the **Compute Cushion** into **Height under Initial Pressure**, set **Number of Increments** into **6**, set **Initial PreStress** to **2.50 kN/m** and set the **Internal Pressure** with **600 Pa** then click **Accept** (Please note that the unbalanced force should converge into 0 or maximum decimal place with power of 9, if the result is not converging please check the model and re-do again).



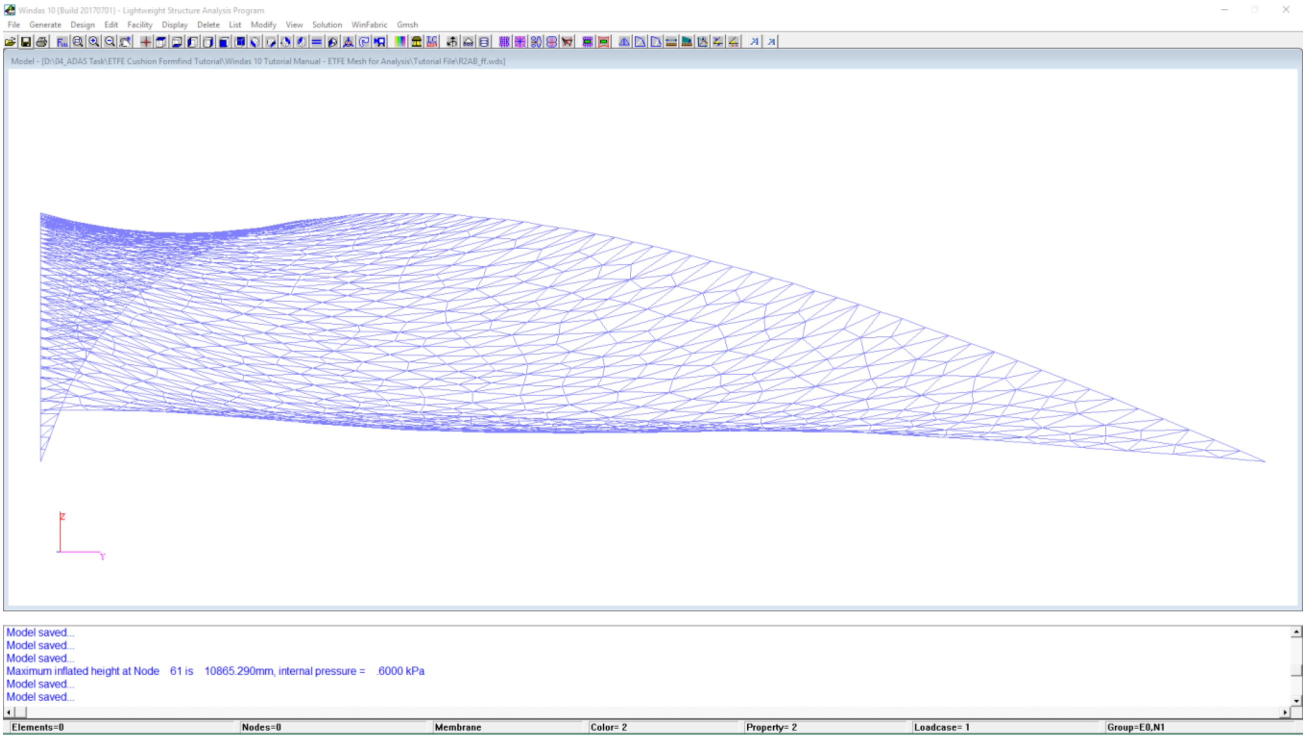
After this, the mesh will be inflated into the cushion shape as shown below. Click **File | Save** and we will proceed to the next phase where we will do inflation for Bottom Layer.



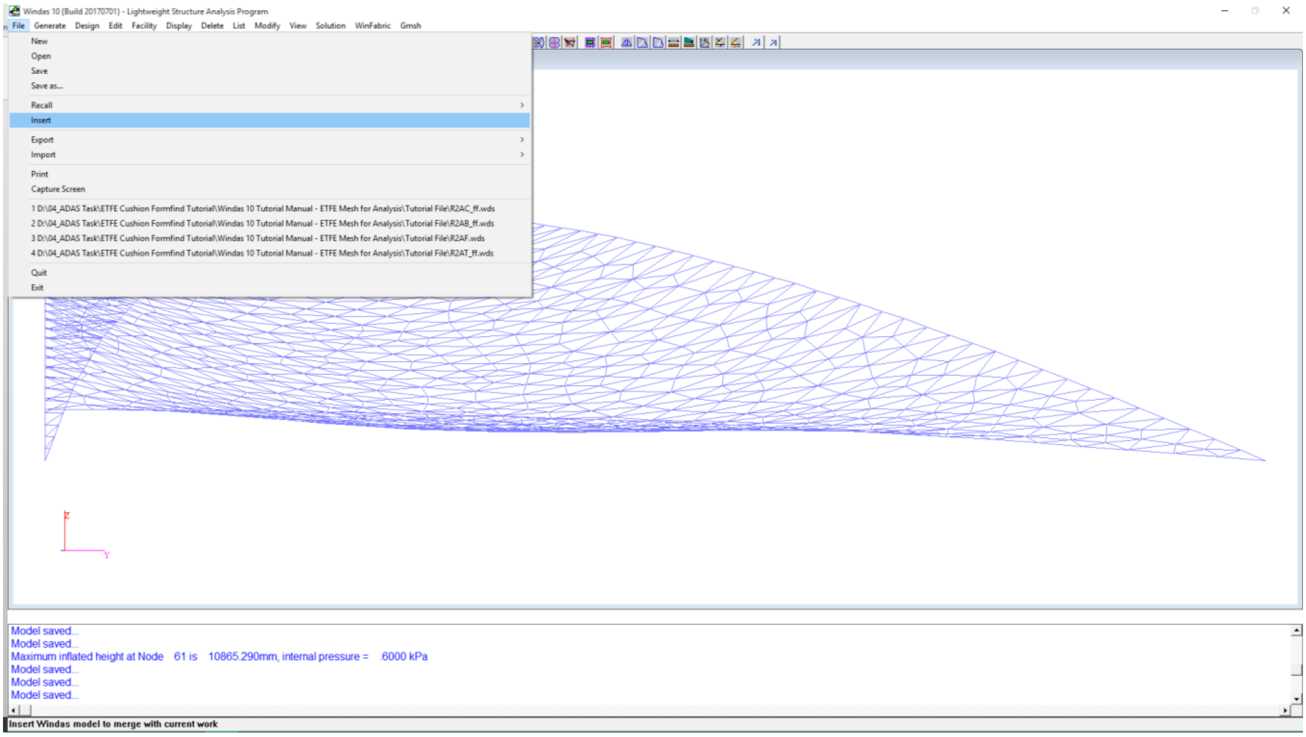
As for the inflation of **Bottom Layer**, click **File | Open** select the **filenameFlat.wds** (the flat mesh which serves as the base mesh for inflation). Then, click **File | Save As filenameBottom_ff.wds**. After this, please repeat the procedure of **Top Layer** inflation for **Bottom Layer** inflation. The only thing that is different for **Bottom Layer** is as stated below;

- **Edit | Element Attributes** select **All** and set **Color** into **Color 2** (Color 2 is for Bottom Layer)
- **Edit | Element Attributes** select **All** and set **Property** into **Property 2** (Property 2 is for Bottom Layer)
- Change the membrane normal (for Bottom Layer it is **–Z**) by clicking **Edit | Surface Normals | -Z** and select **All** then click **Accept**

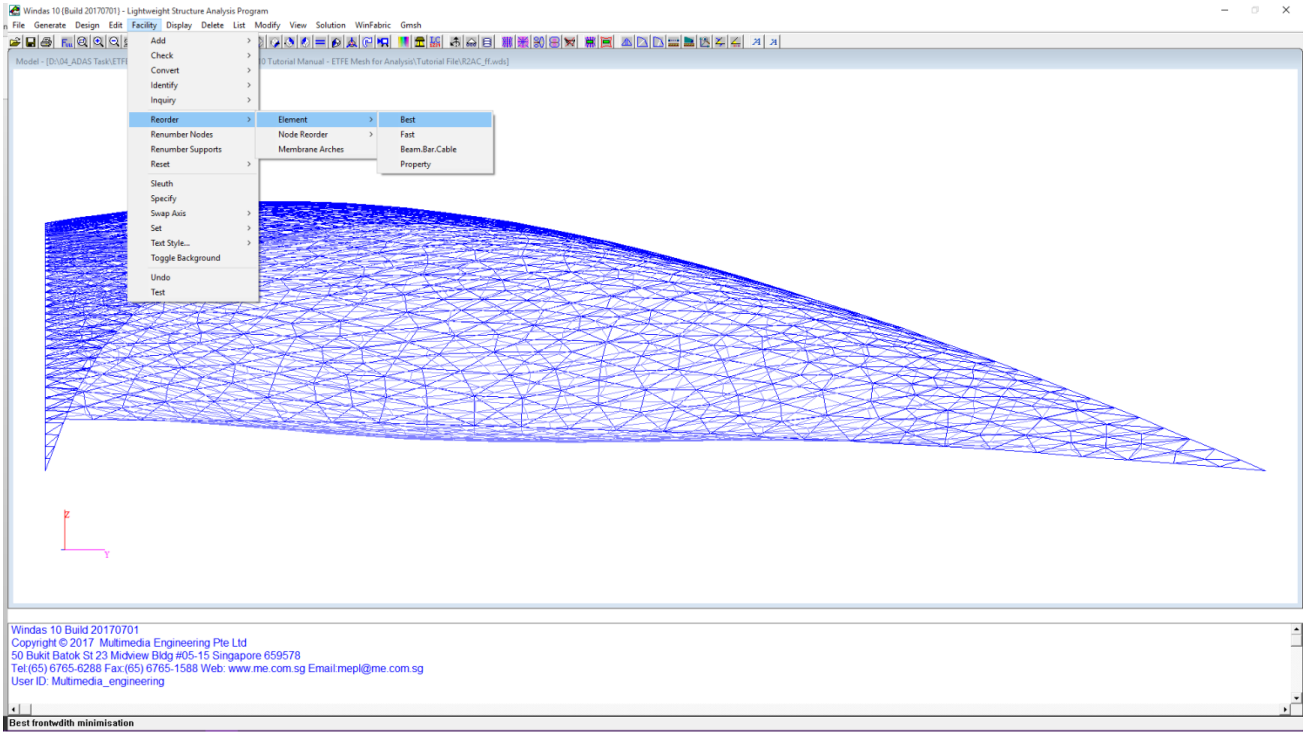
After this step, click **Winfabric | Inflated Mesh Volume Form Finding** and **Accept** with the same parameter as **Top Layer**. Then save the file by clicking **File | Save**. The inflated **Bottom Layer** mesh is as shown below;

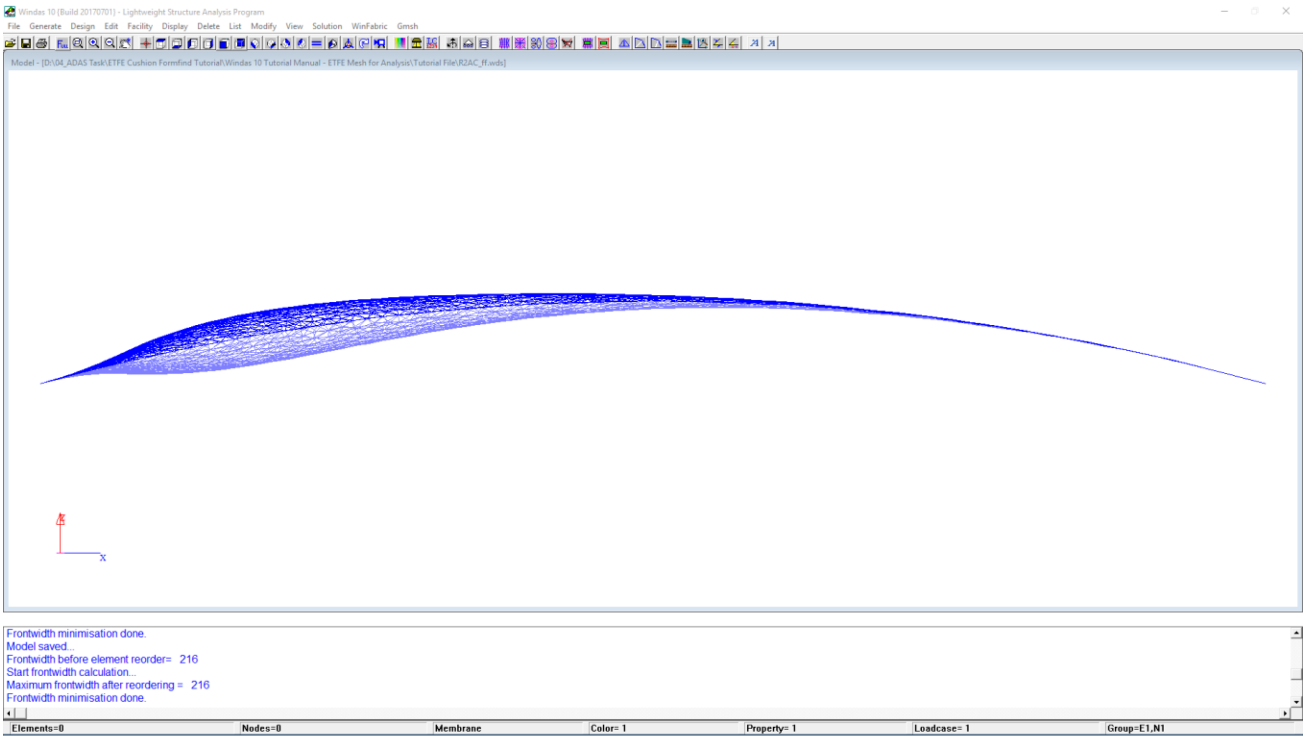


Now, we have both **Top Layer** and **Bottom Layer** mesh for ETFE cushion. We need to combine them into a single cushion for analysis purpose in the next phase. Click **File | Save As filenameCombination_ff.wds**. Then, click **File | Insert** to insert Top Layer model.



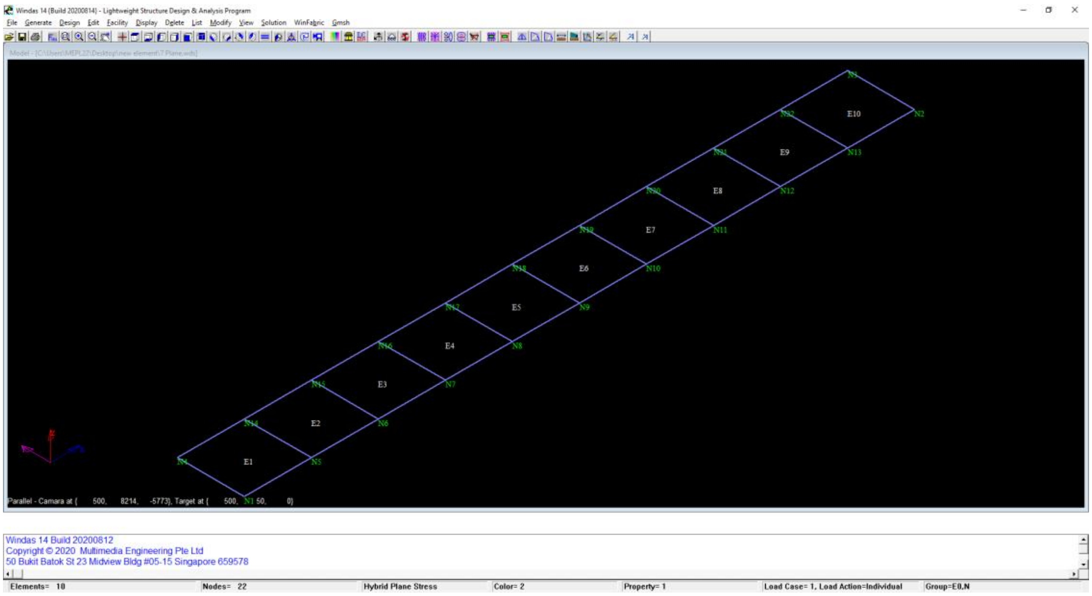
After this, click **Facility | Reorder | Element Reorder | Best** and proceed to click **WinFabric | Inflated Mesh Volume Form Finding** with the same parameter as before. Then, save the file by clicking **File | Save**.





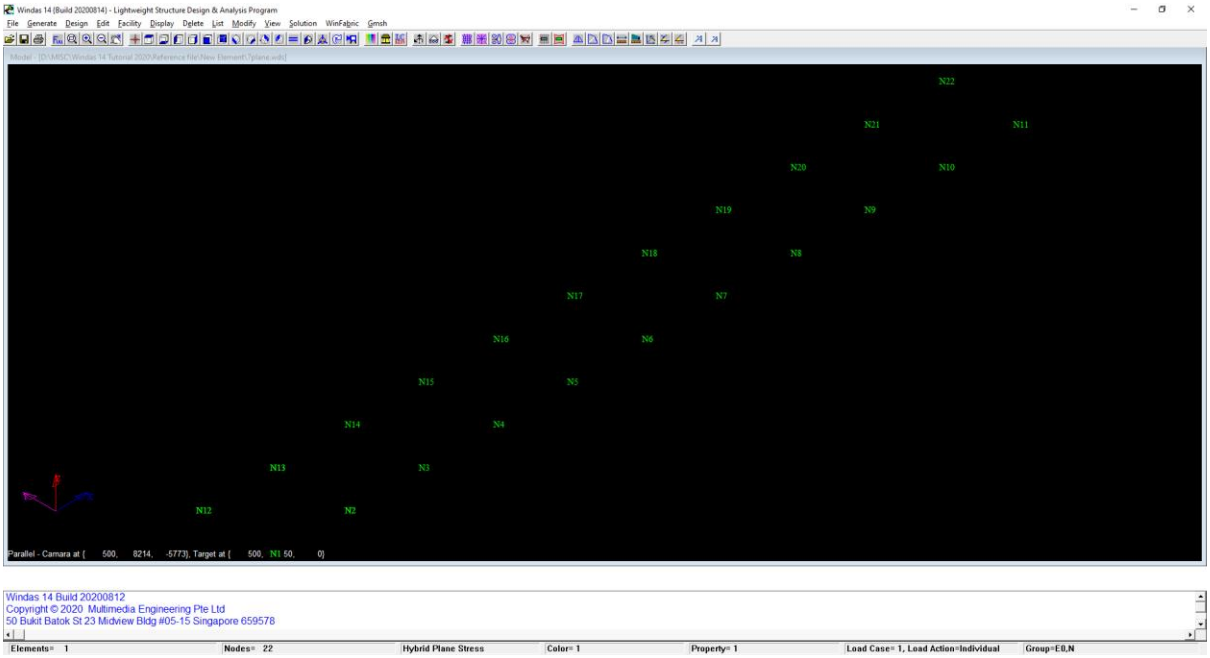
18. New Element

18.1. Hybrid Plane Stress

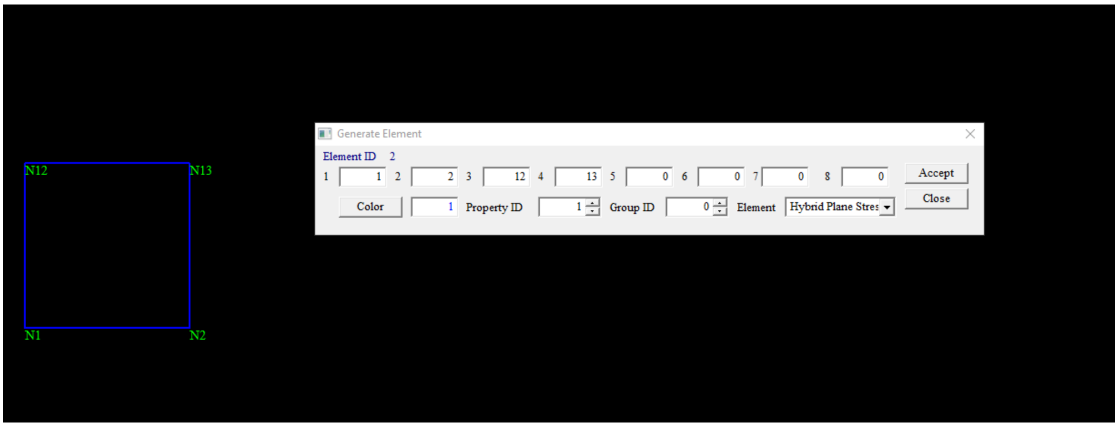


In order to construct this Hybrid Plane Stress, we must first construct the nodes accordingly.

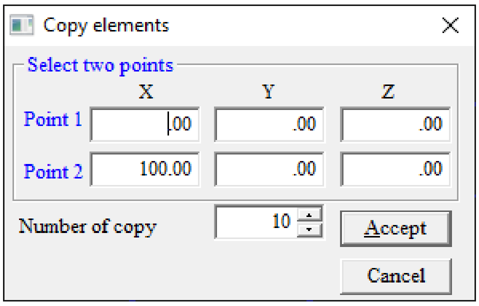
Node	x-coord	y-coord	z-coord
1	0	0	0
2	100	0	0
3	200	0	0
4	300	0	0
5	400	0	0
6	500	0	0
7	600	0	0
8	700	0	0
9	800	0	0
10	900	0	0
11	1000	0	0
12	0	100	0
13	100	100	0
14	200	100	0
15	300	100	0
16	400	100	0
17	500	100	0
18	600	100	0
19	700	100	0
20	800	100	0
21	900	100	0
22	1000	100	0



After constructing the desired nodes, we can now construct the elements. Click **Generate Element.** In the command box change the element type to **Hybrid Plane Stress**. Always remember to key in the corresponding nodes in anti-clockwise order.



Once the first Hybrid Plane Stress element is constructed, we can simply copy this element to the adjacent nodes. Click **Modify | Copy | Element** and when the command box appears, click the element. Choose the first point as Node 1 and the second point as Node 2. Copy 10 times.



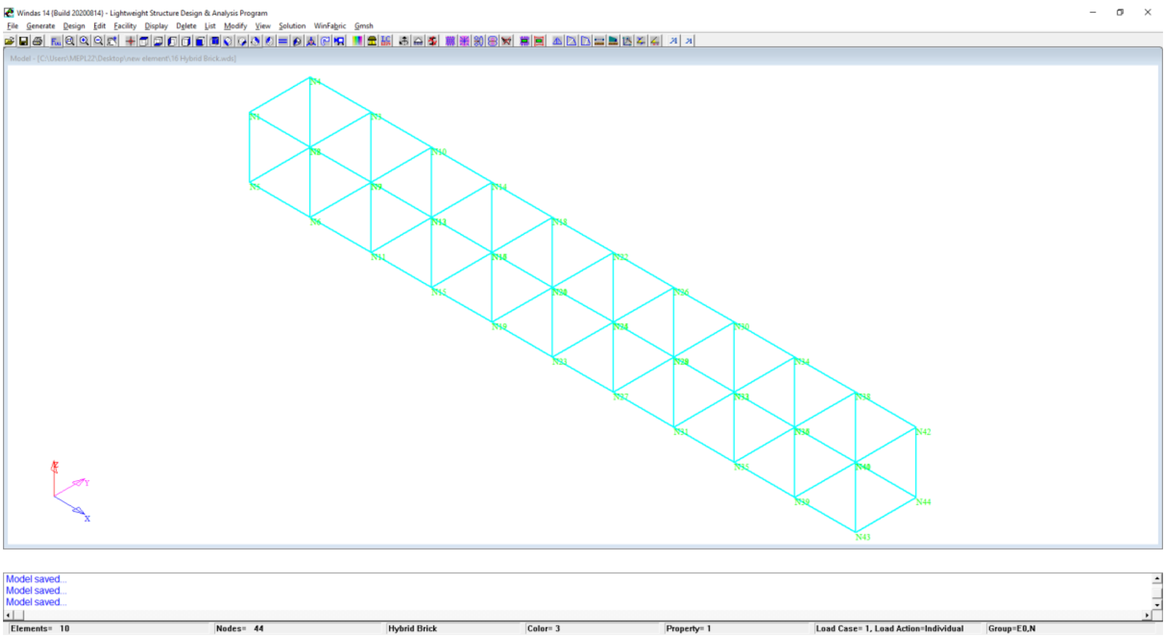
Click **List | Element** and check this reference below to see whether the **Hybrid Plane Stress** has been generated correctly.

Windas Text Editor - [D:\MISC\Windas 14 Tutorial 2020\Reference file\New Element\7plane_element.lst]

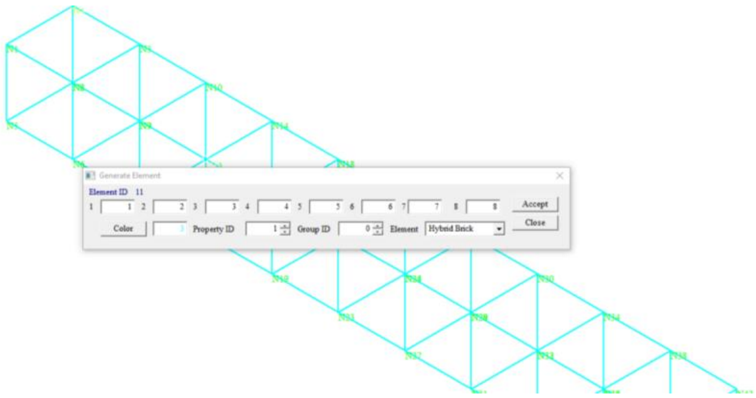
Line	Element Number	Element Type	Property Number	Group Number	Color ID	Area m2	Description	Nodes			
1	1	7	1	0	1	.010	Hybrid Plane Stress	1	2	13	12
2	2	7	1	0	1	.010	Hybrid Plane Stress	2	3	14	13
3	3	7	1	0	1	.010	Hybrid Plane Stress	3	4	15	14
4	4	7	1	0	1	.010	Hybrid Plane Stress	4	5	16	15
5	5	7	1	0	1	.010	Hybrid Plane Stress	5	6	17	16
6	6	7	1	0	1	.010	Hybrid Plane Stress	6	7	18	17
7	7	7	1	0	1	.010	Hybrid Plane Stress	7	8	19	18
8	8	7	1	0	1	.010	Hybrid Plane Stress	8	9	20	19
9	9	7	1	0	1	.010	Hybrid Plane Stress	9	10	21	20
10	10	7	1	0	1	.010	Hybrid Plane Stress	10	11	22	21
11	11	7	1	0	1	.010	Hybrid Plane Stress	11	23	24	22
				Total Area =		.110 m2					

Line:14 Col:66 NUM INS

18.2. Hybrid Brick

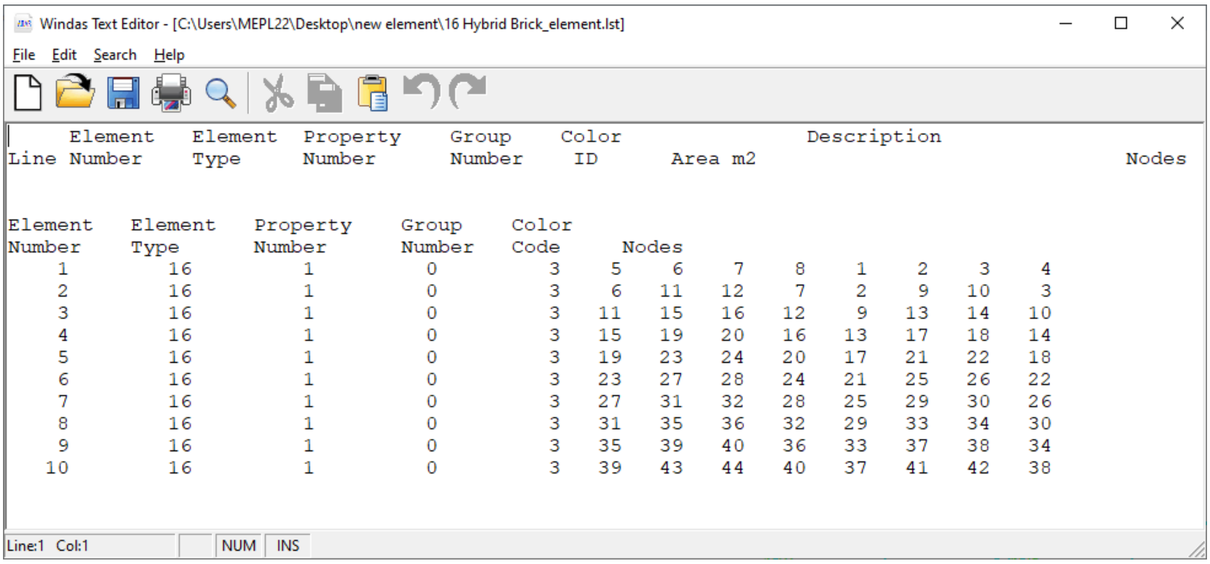


Similar to Hybrid Plane Stress, we need to construct the nodes and then construct the elements by keying in the nodes in anti-clockwise order.



Node	x-coord	y-coord	z-coord
1	0	0	0
2	0	0	1000
3	1000	0	1000
4	1000	1000	1000
5	0	1000	1000
6	1000	0	0
7	1000	1000	0
8	0	1000	0
9	2000	0	1000
10	2000	1000	1000
11	2000	0	0
12	2000	1000	0
13	3000	0	1000
14	3000	1000	1000
15	3000	0	0
16	3000	1000	0
17	4000	0	1000
18	4000	1000	1000
19	4000	0	0
20	4000	1000	0
21	5000	0	1000
22	5000	1000	1000
23	5000	0	0
24	5000	1000	0
25	6000	0	1000
26	6000	1000	1000
27	6000	0	0
28	6000	1000	0
29	7000	0	1000
30	7000	1000	1000
31	7000	0	0
32	7000	1000	0
33	8000	0	1000
34	8000	1000	1000
35	8000	0	0
36	8000	1000	0
37	9000	0	1000
38	9000	1000	1000
39	9000	0	0
40	9000	1000	0
41	10000	0	1000
42	10000	1000	1000
43	10000	0	0
44	10000	1000	0

Click **List | Element** and check this reference below to see whether the **Hybrid Brick** has been generated correctly.



Windas Text Editor - [C:\Users\MEPL22\Desktop\new element\16 Hybrid Brick_element.lst]

File Edit Search Help

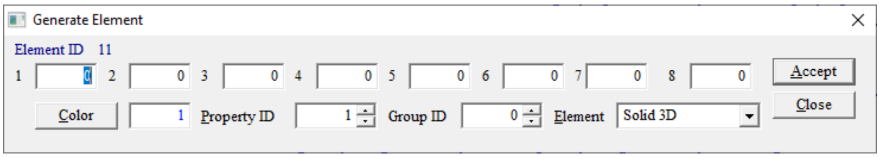
Line | Element Number | Element Type | Property Number | Group Number | Color ID | Area m2 | Description | Nodes

Element Number	Element Type	Property Number	Group Number	Color Code	Nodes							
1	16	1	0	3	5	6	7	8	1	2	3	4
2	16	1	0	3	6	11	12	7	2	9	10	3
3	16	1	0	3	11	15	16	12	9	13	14	10
4	16	1	0	3	15	19	20	16	13	17	18	14
5	16	1	0	3	19	23	24	20	17	21	22	18
6	16	1	0	3	23	27	28	24	21	25	26	22
7	16	1	0	3	27	31	32	28	25	29	30	26
8	16	1	0	3	31	35	36	32	29	33	34	30
9	16	1	0	3	35	39	40	36	33	37	38	34
10	16	1	0	3	39	43	44	40	37	41	42	38

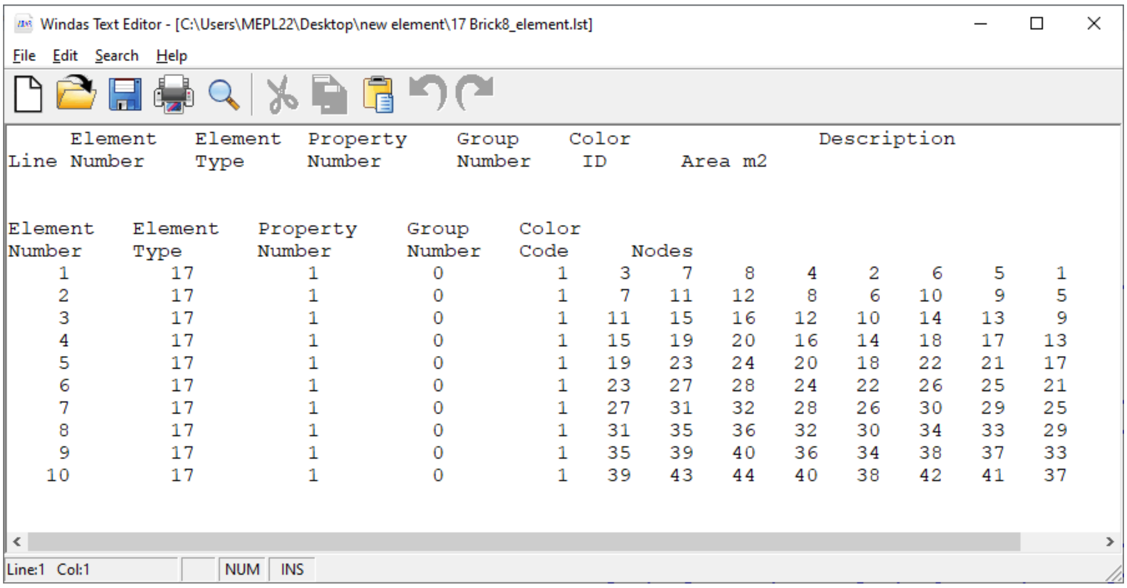
Line:1 Col:1 NUM INS

18.3. Brick 8 (Solid 3D)

Generate the same nodes and elements as shown for hybrid brick. However, change the element type to **Solid 3D**.



Another way to do create Brick 8 from Hybrid brick, we can simply click **Edit | Element Attributes** choose **All | Element Type | Solid 3D.** Click **List | Element** and check this reference below to see whether the **Brick 8 / Solid 3D** has been generated correctly.



Windas Text Editor - [C:\Users\MEPL22\Desktop\new element\17 Brick8_element.lst]

File Edit Search Help

Line | Element Number | Element Type | Property Number | Group Number | Color ID | Area m2 | Description

Element Number	Element Type	Property Number	Group Number	Color Code	Nodes							
1	17	1	0	1	3	7	8	4	2	6	5	1
2	17	1	0	1	7	11	12	8	6	10	9	5
3	17	1	0	1	11	15	16	12	10	14	13	9
4	17	1	0	1	15	19	20	16	14	18	17	13
5	17	1	0	1	19	23	24	20	18	22	21	17
6	17	1	0	1	23	27	28	24	22	26	25	21
7	17	1	0	1	27	31	32	28	26	30	29	25
8	17	1	0	1	31	35	36	32	30	34	33	29
9	17	1	0	1	35	39	40	36	34	38	37	33
10	17	1	0	1	39	43	44	40	38	42	41	37

Line:1 Col:1 NUM INS

19. Load Analysis of A Simple Hypar

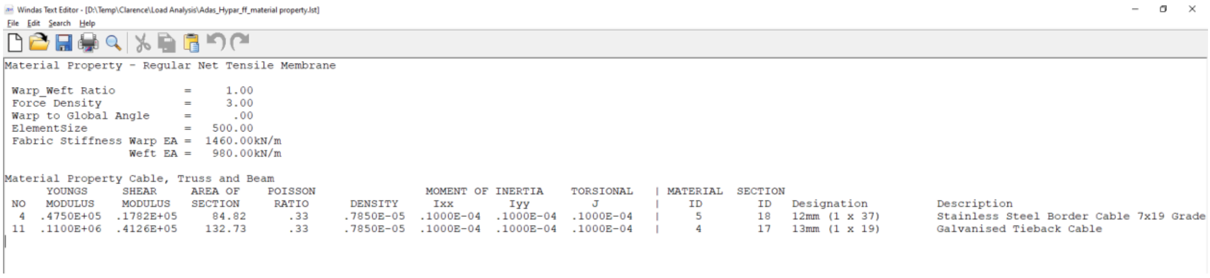
In this tutorial, you will learn how to perform load analysis of a simple hypar. We will use **a simple hypar** model that has been created in the previous tutorial.

19.1. Material Property

In structural analysis, the elastic modulus of material section properties needs to be specified in order to perform computer analysis. Similarly, we need to specify the fabric stiffness of the fabric in the warp and weft directions and also the cable properties so that the computer can run the analysis for a tensile membrane structure.

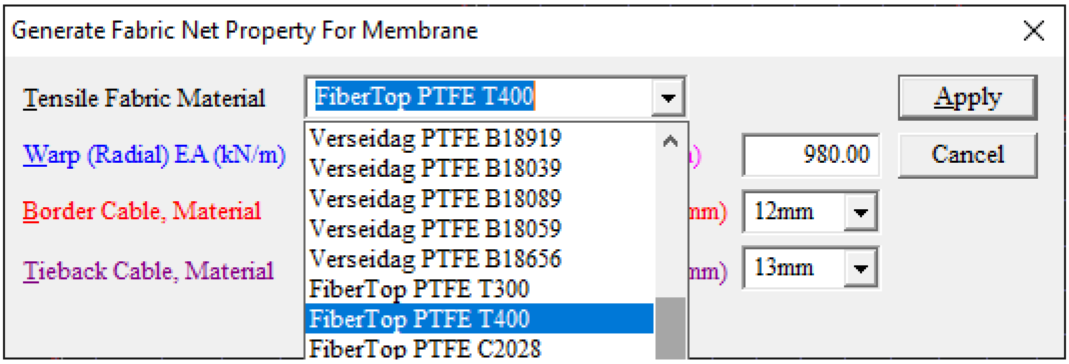
The fabric stiffness of a given fabric is obtained from bi-axial testing of the fabric material.

After performing form-finding, Windas automatically assigns material and section properties to the model. Use **List | Material & Section Properties** command to see the default values given. For the hypar constructed previously, the values given should be as shown below.



```
Material Property - Regular Net Tensile Membrane

 Warp_Weft Ratio          =     1.00
 Force Density            =     3.00
 Warp to Global Angle     =      .00
 ElementSize              =   500.00
 Fabric Stiffness Warp EA =  1460.00kN/m
                  Weft EA =   980.00kN/m

Material Property Cable, Truss and Beam
      YOUNGS     SHEAR     AREA OF    POISSON                 MOMENT OF INERTIA      TORSIONAL   | MATERIAL  SECTION
 NO   MODULUS    MODULUS   SECTION     RATIO      DENSITY      Ixx        Iyy           J        |    ID        ID   Designation          Description
  4  .4750E+05  .1782E+05    84.82       .33    .7850E-05  .1000E-04  .1000E-04  .1000E-04   |     5        18   12mm (1 x 37)        Stainless Steel Border Cable 7x19 Grade
 11  .1100E+06  .4126E+05   132.73       .33    .7850E-05  .1000E-04  .1000E-04  .1000E-04   |     4        17   13mm (1 x 19)        Galvanised Tieback Cable
```

The default fabric is **FiberTop PTFE T400 with 1 mm stainless steel border cable and 13mm galvanised tieback cable.** Use **Generate | Material & Property | Tensile Membrane property** to verify or change the default material and section properties.



Please note that the unit of fabric stiffness is kN/m width and is directional dependent.

It is advisable to perform biaxial test to determine the fabric stiffness along the warp and weft directions for each batch of material received.

For reference**, fabric property** of some of the most commonly used fabric materials for the construction of tensile membrane structures are given in table below.

					Fabric Weight	Fabric Stiffness, kN/m		Tensile Strength, kN/m	
Base Cloth	Coating	Brand	Model	Type	g/m2	Warp	Weft	Warp	Weft
PVC	PvdF	Duraskin	B4951	I	800	600	300	60	60
		Duraskin	B4617	II	900	1000	500	88	79
		Duraskin	B4915	III	1100	1500	900	115	102
		Duraskin	B4618	IV	1300	2000	1250	149	128
		Duraskin	B4092	V	1450	2200	1400	196	166
PVC	Pvdf	Ferrari	502	0	590	680	680	56	56
		Ferrari	702	I	750	680	680	56	56
		Ferrari	1002	II	1050	750	750	84	84
		Ferrari	1202	III	1050	1000	1000	112	112
		Ferrari	1302	IV	1350			160	140
Glass Fiber	Teflon	Verseidag	18039	I	800	1195	1097	70	70
		Verseidag	18089	III	1150	2200	1150	116	116
		Verseidag	18059	IV	1550			150	130
Glass Fiber	Teflon	FiberTop	T400	I	850	1460	980	124	104
		FiberTop	C2028	II	1182	1738	1028	182	147
		FiberTop	C1008	IV	1320	1513	1315	187	186
		FiberTop	C1028	V	1560	1504	1039	188	222
ePTFE	Teflon	Tenara	3T20	I	630	600	300	60	58
		Tenara	4T20	II	830	600	300	84	80

Also for reference, the recommended **pre stress level** on different fabric is given in the table below:

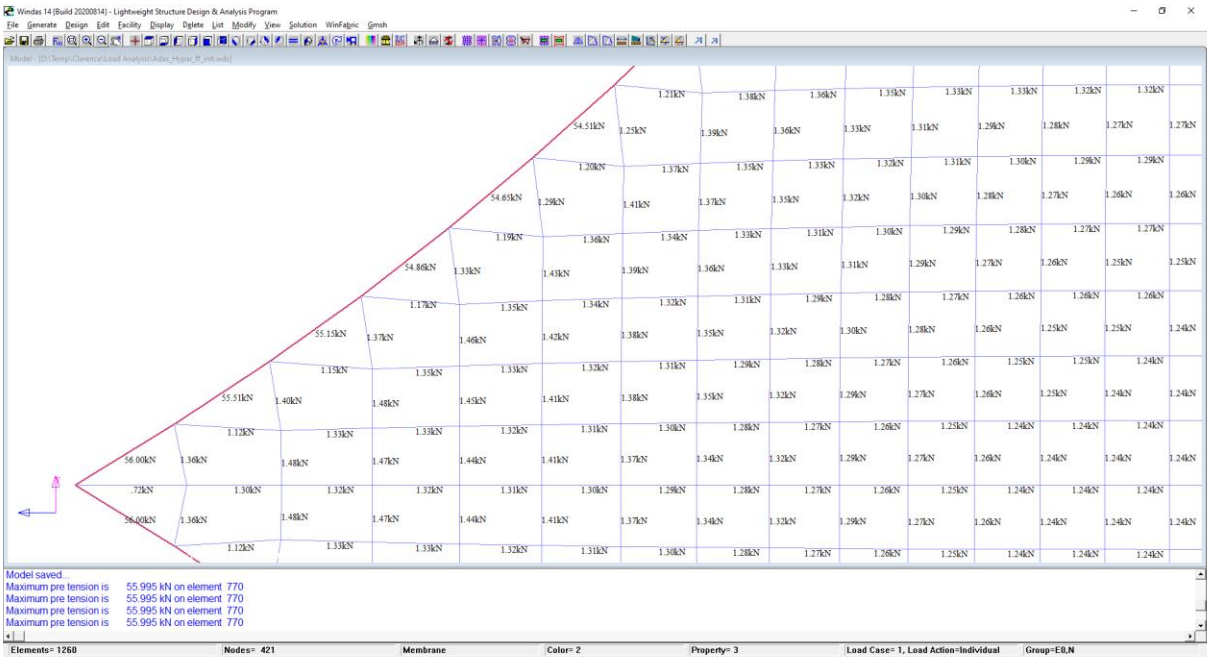
				Tensile Strength, kN/m		Recommended Prestress kN/m					
Material	Manufacture	Model	Type	Warp	Weft	Warp			Weft		
						Min	Max	*Use*	Min	Max	*Use*
PVC	Verseidag	B4951	I	60	60	0.5	4.0	*1.0*	0.5	4.0	*1.0*
		B4617	II	88	79	1.0	5.0	*1.0*	1.0	5.0	*1.0*
		B4915	III	115	102	1.5	7.0	*2.0*	1.5	6.0	*2.0*
		B4618	IV	149	128	2.0	8.0	*2.0*	2.0	8.0	*2.0*
		B4092	V	196	166	3.0	10.0	*3.0*	3.0	10.0	*3.0*
PVC	Ferrari	502	0	56	56	0.5	4.0	*1.0*	0.5	4.0	*1.0*
		702	I	60	56	0.5	4.0	*1.0*	0.5	4.0	*1.0*
		1002	II	84	84	1.0	5.0	1.0	1.0	5.0	2.0
		1202	III	112	112	1.5	7.0	*2.0*	1.5	7.0	*2.0*
		1302	IV	160	140	2.0	10.0	*3.0*	2.0	9.0	*3.0*
PTFE	Verseidag	18039	I	70	70	2.0	5.0	*2.0*	2.0	5.0	*2.0*
		18089	III	116	116	2.0	10.0	*3.0*	2.0	9.0	*3.0*
		18059	IV	150	130	2.0	9.0	*4.0*	2.0	8.0	*4.0*
ePTFE	Gore Tenera	3T20	I	60	58	1.5	4.0	*2.0*	1.5	4.0	*2.0*
		4T20	II	84	80	2.5	5.0	*3.0*	2.5	5.0	*3.0*

19.2. Materialization

The basic concept of force density approach to tensile membrane design is to reduce the membrane surface to a cablenet representation. The fabric modulus is approximated by an equivalent EA value. In short, *Materialization is a process of idealization of a membrane surface as a cablenet system with the pre-tension that is equivalent to the pre-stress of the membrane.*

Use **Winfabric | FD -> Cablenet** command to do this materialization. Notice that after materialization is done, he model name changed from **ADAS_hypar_ff.wds** to A**DAS_Hypar_ff_init.wds.** The addition of init indicates that the cablenet model is in a state of *self-equilibrium*

Click **Display | Load Values | Pre-tension** to show the pretension on the initial equilibrium cablenet model.



Pre-tension on the cablenet model is equivalent to the pre-stress on the force density model.

External loads applied to a tensile membrane structure are resisted by its pre-tension (which is the same as fabric pre-stress). So, higher pre-stress value is required to resist larger external loads. The optimum pre-stress of a given membrane structure is not known until you have performed the load analysis.

19.3. Loadings

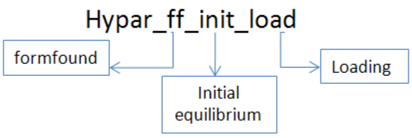
Membrane structure are normally subjected to the following loads :

- <u>Dead load</u> due to fabric, cables, fittings, and clamping plates is generally taken as 0.02kN/m^2
- <u>Rain load</u> due to rain water flow on the membrane surface is assumed to be 10mm thick or 0.10 kN/m^2 vertically downward
- <u>Wind load</u> - depends on the project location. In Singapore, the mean hourly wind speed is 22m/sec.

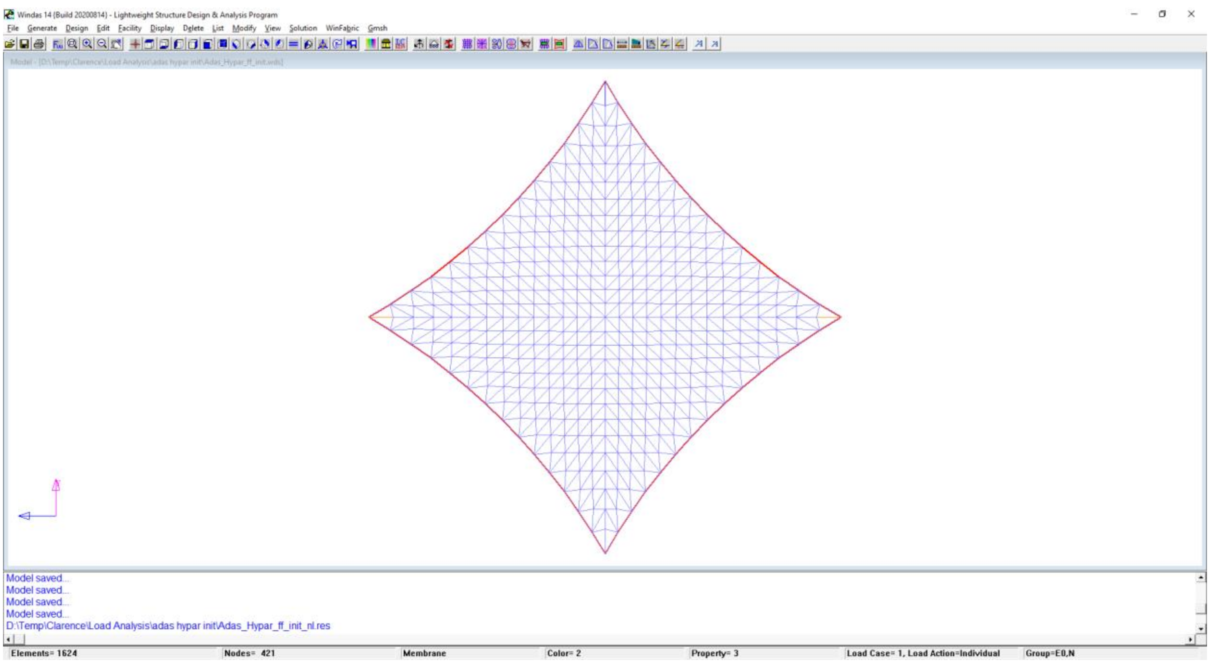
Wind loading is normally the most critical and least accurate out of the three loadings. For big and complex membrane structure, the wind pressure coefficient over the membrane surface is determined from wind tunnel test or a CFD Analysis which will be discussed furthermore.

The loadings for membrane analysis are to be input as combinations of individual loadings. Do not confuse this with the load combination as in the structural analysis.

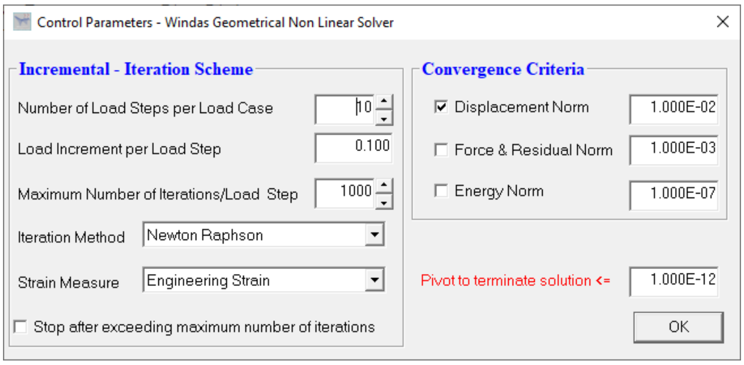
Save the model as **ADAS_Hypar_ff_init_load** before conducting the full structural analysis. The **ADAS_Hypar_ff** model and **ADAS_Hypar_ff_init** model might be needed later.



The membrane surface is represented by quadrilateral surface elements. To perform load analysis, these quadrilateral surface element need to convert into triangular elements. Click **Winfabric | Triangulation** to proceed with this procedure.



Perform load analysis with the **Solution | Nonlinear {Tensile Membrane, ETFE}** command. Accept default by clicking **OK**. Click **F1** right away when the analysis is done to view the analysis results. Alternatively, click **List | Results | All | Nonlinear {Tensile Membrane, ETFE}.**



Check your result to make sure that all the nodal displacements are zero and the reactions at the system points are as shown below.

Node	Code	Reaction X-Axis (kN)	Reaction Y-Axis (kN)	Reaction Z-Axis (kN)
1	111	0.00	-91.91	33.54
2	111	91.92	0.00	-33.54
3	111	-0.00	91.91	33.54
4	111	-91.92	0.00	-33.54
		=========	=========	=========
		0.00	0.00	0.00
		=========	=========	=========

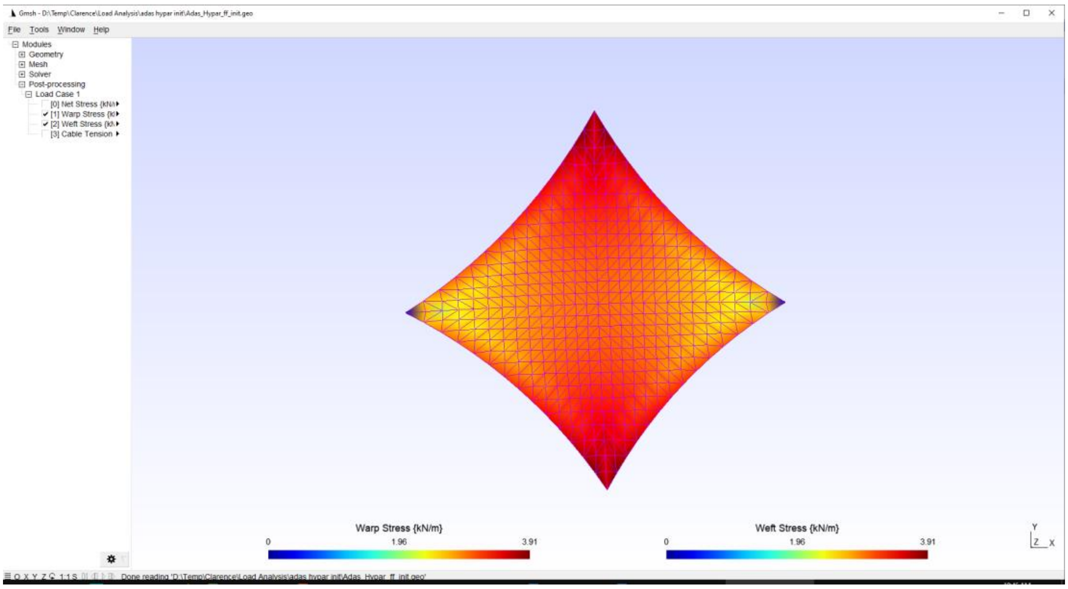
(Note : this is an important step to do as we need to make sure that the hypar is really in equilibrium state. Analysis will not be accurate otherwise.)

Another important checking is checking if we have all complete properties of a hypar. Click **List | Element** and check all the properties accordingly.

Below is shown the general reference of how Windas recognize the elements.

Representation	Color	Color ID	Property ID
Fabric Net (Warp/Radial)	Blue	1	1
Fabric Net (Weft/Ring)	Cyan	10	2
Border Cable	Red	13	3
Membrane Surface	Light Blue	2	

Click **Gmsh | Fabric Stress Plot** to check that the stress in the warp and weft varies from 2.18 kN/m to 3.91 kN/m.

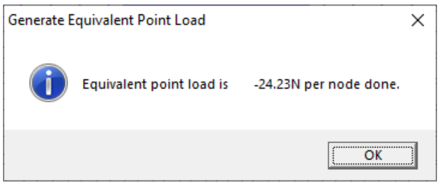


Recall that in Winfabric, the loadings are input as combination of individual loadings.

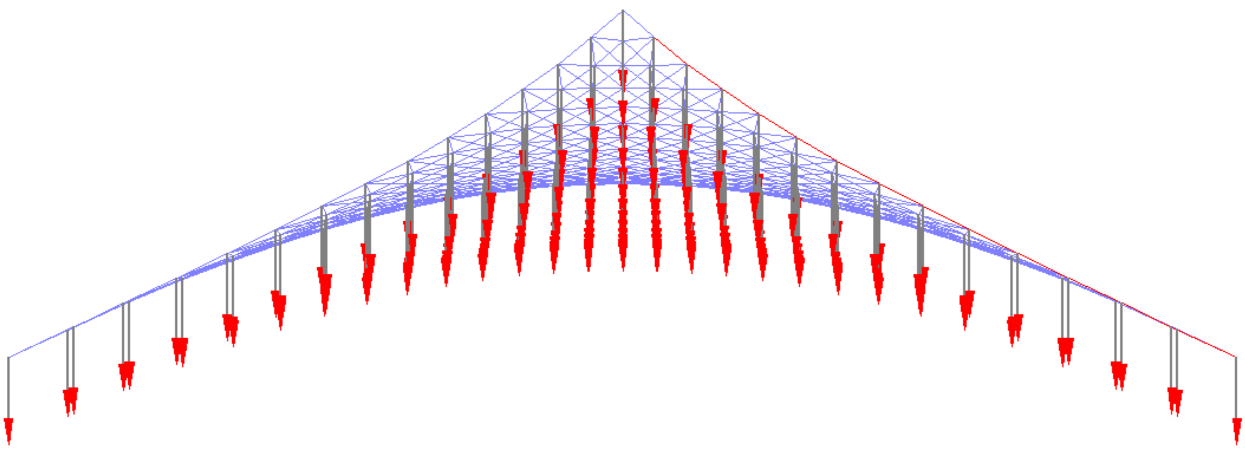
19.3.1. Load Combination 1 : Pre-stress + Dead Load + Rain Load

The design load for load combination 1 is taken as 0.12kN/m^2. Apply the load as equivalent point load using the **Generate | Load | Point Load (Z-equivalent)** command.

Windas calculates the equivalent point load automatically and apply to each nodal point.



Adjust the view and click **Display | Load | Point Load | Load case 1** to check whether the load has been applied in the correct direction.



Perform load analysis with **Solution | Nonlinear {Tensile Membrane, ETFE}.**

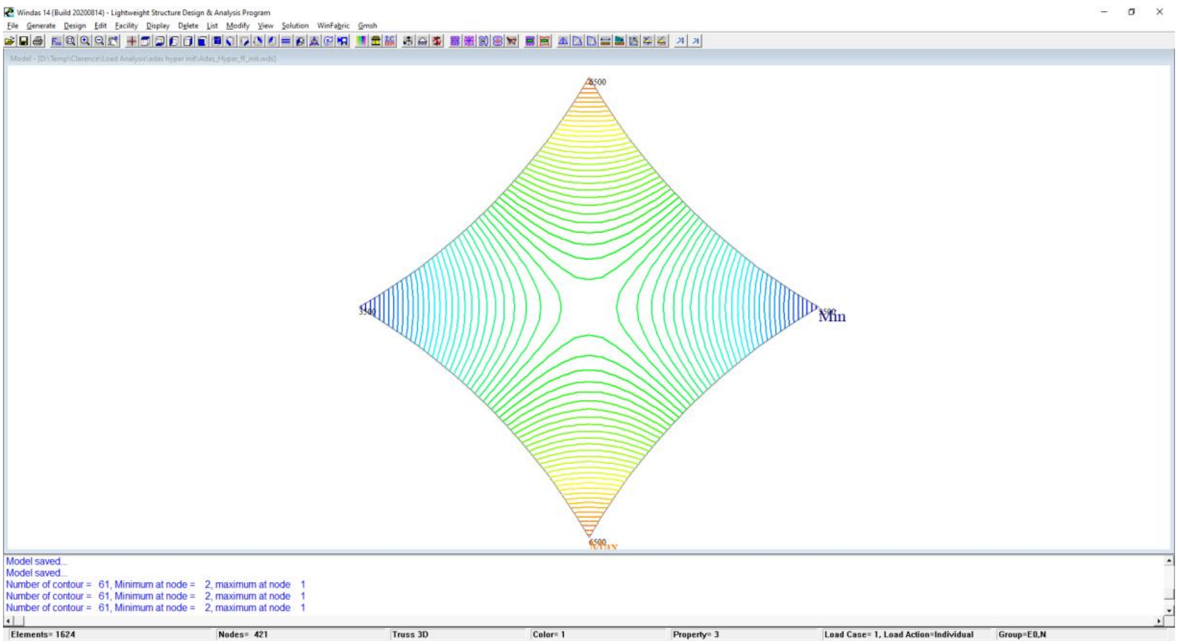
Click **List | Results | Movements | Maximum** command to check the maximum displacements and locations.

```
maximum movement listings
=========================
Loadcase   1
Maximum x-displacement of         3.964 at node   322
Maximum y-displacement of         2.788 at node    19
Maximum z-displacement of       -14.473 at node   318
```

Under dead load and live load, it is important to check that there is no slacking of the membrane.

If slacking occurs, you need to undo the form finding, increases the pre-stress level and perform form finding again. Repeat the load analysis again.

We can also check for the possibility of water ponding with the **WinFabric | h-contour** command.



A closed contour indicates a strong possibility of water ponding.

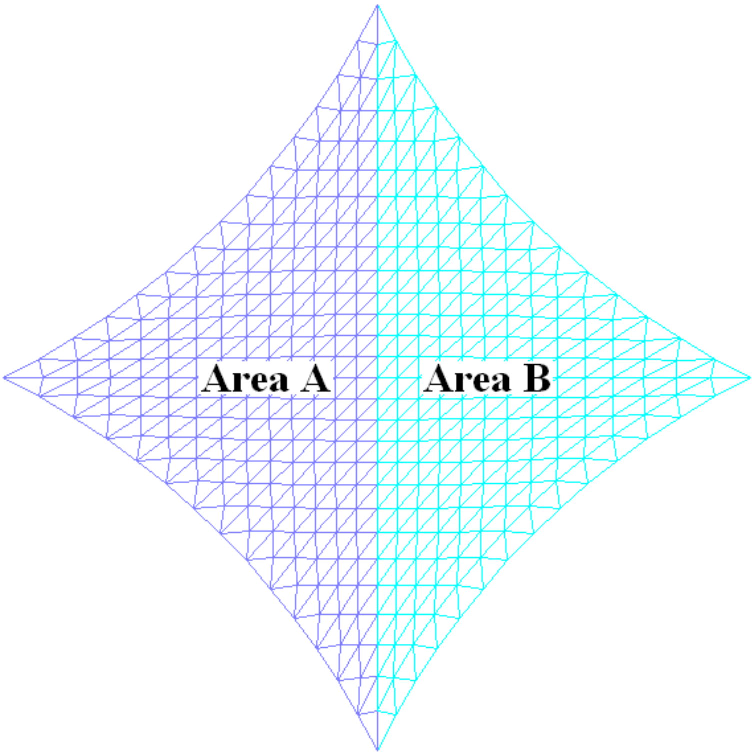
Please note that the loading for load case 1 is Pre-stress + Dead Load + Rain Load.

19.3.2. Load Case 2 : Pre-stress + Dead Load + Wind Load

Now we will focus more on the application of wind loading. The wind pressure acting on a membrane surface depends on its shape and the site wind speed. Wind pressure coefficient, which is a function of its shape, is normally determined by wind tunnel test. It can also be estimated from CFD Analysis. The wind pressure over the membrane surface is not a constant. For this reason, it is necessary to divide the membrane surface into sub-regions.

In this tutorial, we divide the membrane surface into two sub-regions for wind pressure and wind suction. Use the **WinFabric | Fabric Surface | Color | X-Range** command to change the color attributes of the membrane elements in Area B to light blue (Color ID 3).

Select Node ID 1 and 2 to define the node range.



The membrane surface is now divided into two sub-regions, area A and B respectively. This is a simplified representation of wind load sub-regions. Now we need to determine the wind pressure and wind suction acting on Region A and B. Both will be calculated according to BS 6399-2:1997 using simplified standard method.

<u>Dynamic Classification</u>

Building type factor K_b = 8

H = 6m

Dynamic augmentation factor, C_r = 0.12 < 0.25, therefore BS 6399-2 can be used.

Basic wind speed, V_b = 22 m/s

Wind factors :

Altitude factor, S_a = 1.00

Direction factor, S_d = 1.00

Seasonal factor, S_S = 1.00

Probability factor, S_p = 1.00

Site wind speed, $V_s = S_a\ S_d\ S_S\ S_p$

V_s = 22m/s

Closest distance from sea, = 5 km

Effective height, = 6 m

Terrain and Building Factor (Table 4), S_b = 1.528

Effective wind speed, $V_e = V_s\ S_b$ = 22 x 1.528 = 33.616 m/s

Dynamic pressure, $q_s = 0.613\ V_e^2 = 0.613 \times 33.616^2 / 1000 = 0.693$ kN/m^2

Net surface pressure for free-standing canopies, $p = q_s\ C_p\ C_a$

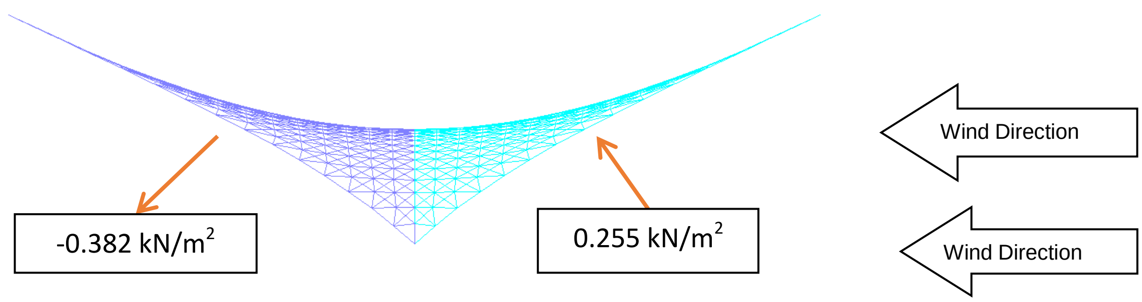
(Clause 2.1.3.3 of BS 6399-2: 1997)

C_p for pitch angle α = +10° is +0.4 and -0.6

Size effect factor (Clause 2.1.3.4) is about 0.92 for diagonal dimension of 14m.

Wind Load (Pressure) = 0.693 x 0.4 x0.92 = **0.255** kN/m^2

Wind Load (Suction) = 0.693 x 0.6 x0.92 = **-0.382** kN/m^2



Now, the case above is only for the wind load. Recall that this loadcase 2 also included dead load and wind load in the analysis.

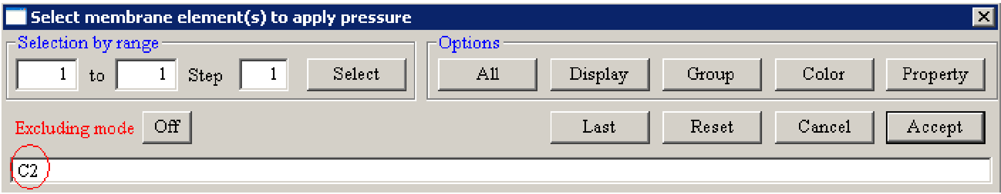
Hence, a uniformly distributed surface load of **0.384 kN/m²** is applied on Area A in the **negative z-direction** based on this calculation :

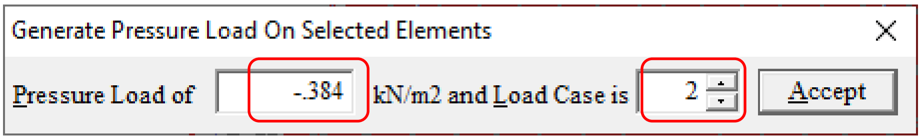
Wind suction + Dead Load = 0.382 + 0.02 = -0.384 kN/m²

Whereas, a uniformly distributed surface load of **0.253 kN/m²** is applied to Area B in the **positive z-direction** based on this calculation :

Wind pressure + Dead Load = 0.255 - 0.02 = 0.253 kN/m²

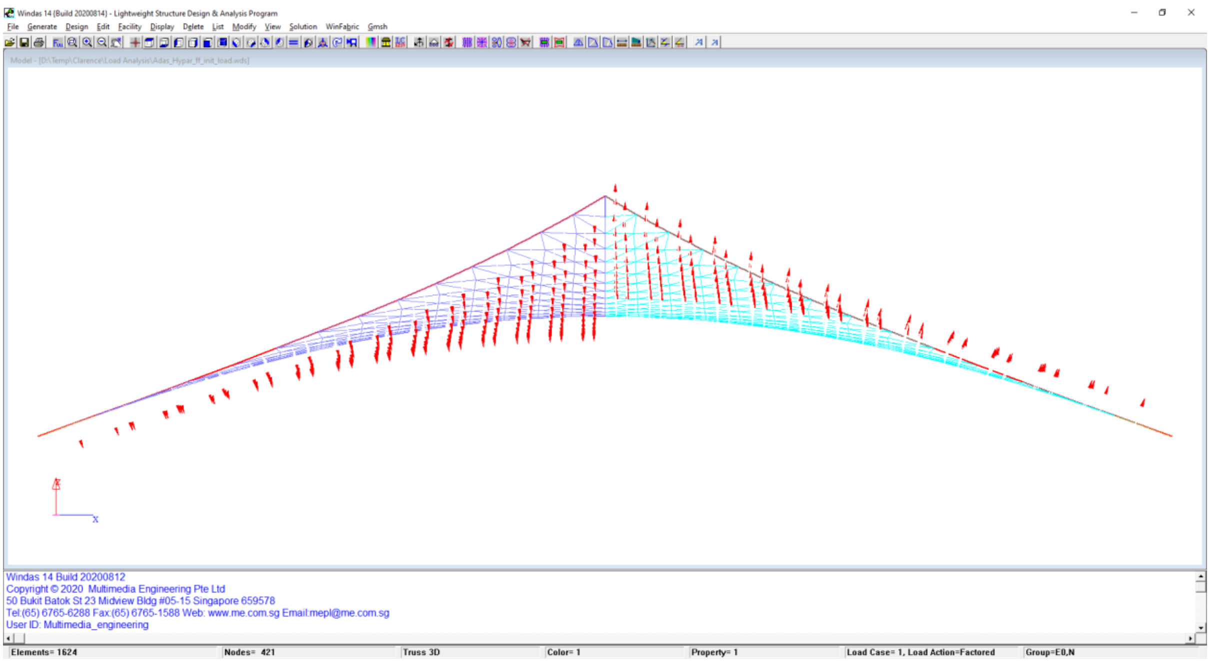
Use the **Generate | Load | Pressure / Wind Loads | Each** command to apply of **0.384 kN/m²** load to membrane elements with color ID 1. Repeat the same steps for color ID 2 with of **0.253 kN/m²** load.





Pressure loading is always perpendicular to the surface.

Click **Display | Load | Pressure Wind Load** to confirm the applied load direction for load combination 2.



19.3.3. Load Case 3: Pre-stress + Dead Load + Rain + Wind

You should be able to work out the applied load on Area A and B for this load case by now.

A uniformly distributed surface load of **-0.504kN/m²** is applied over the area A representing the dead load + rain + wind suction load.

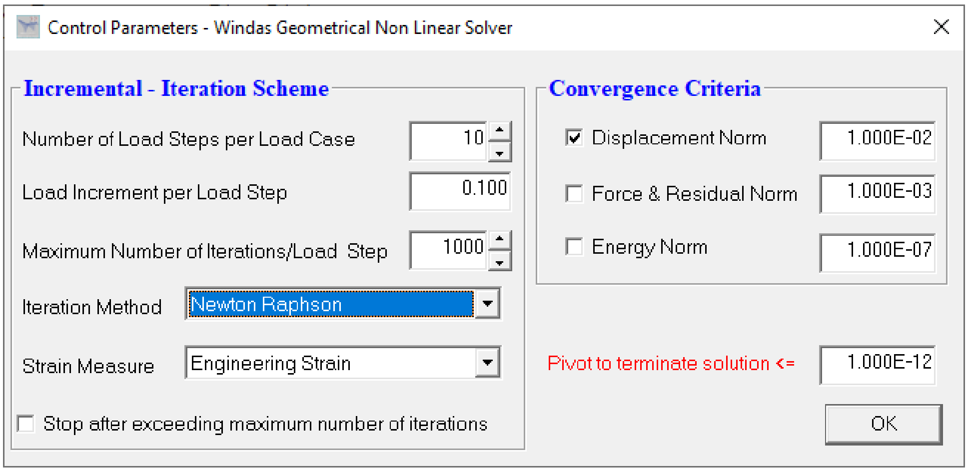
Applied Load = -0.382-0.02-0.10 = -0.504 kN/m²

A uniformly distributed surface load of **0.135kN/m²** is applied over area B representing the dead load + wind pressure.

Applied Load = 0.267-0.02-0.10 = 0.135 kN/m²

19.4. Load Analysis

Perform nonlinear membrane analysis of all three load cases with the **Analysis | Nonlinear {Tensile Membrane, ETFE}** command. Accept the default settings.



The solution type is always Newton – Raphson for tensile membrane load analysis.

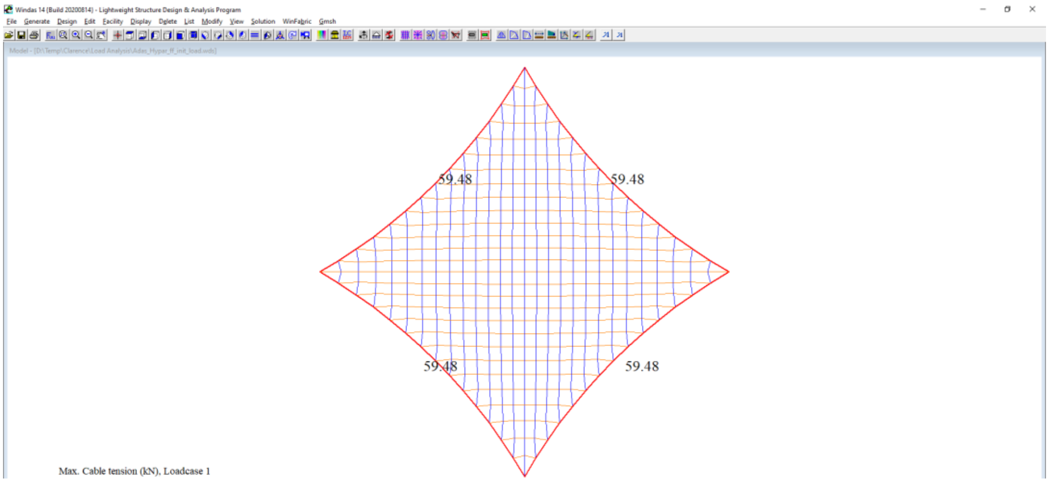
Click **List | Results | Movement | Maximum** to compare the results with this sample below.

```
Maximum Displacement List
=========================
Loadcase   1 Loadcase Name = <Load Name  1>
Maximum x-displacement of        -3.549 at node  104
Maximum y-displacement of        -2.146 at node  230
Maximum z-displacement of       -26.137 at node   82

Loadcase   2 Loadcase Name = <Load Name  2>
Maximum x-displacement of        71.270 at node  397
Maximum y-displacement of       -22.226 at node  190
Maximum z-displacement of      -239.994 at node  397

Loadcase   3 Loadcase Name = <Load Name  3>
Maximum x-displacement of        75.572 at node  397
Maximum y-displacement of       -23.405 at node  190
Maximum z-displacement of      -273.056 at node  397
```

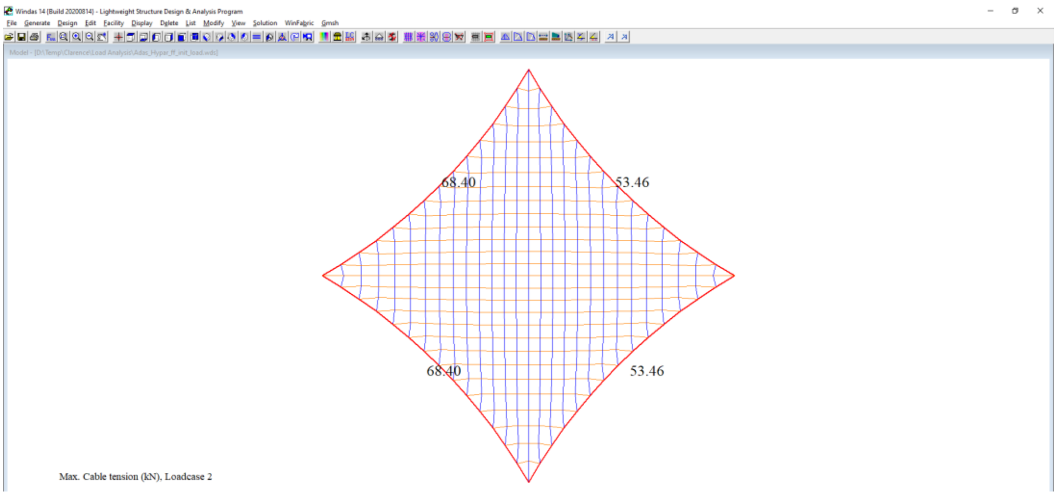
Maximum Factored Border Cable Tension can also be obtained.

Display the factored border cable tensions with the **View | Result | Unfactored | Cable tension (max)** command. Check for all 3 load combinations.

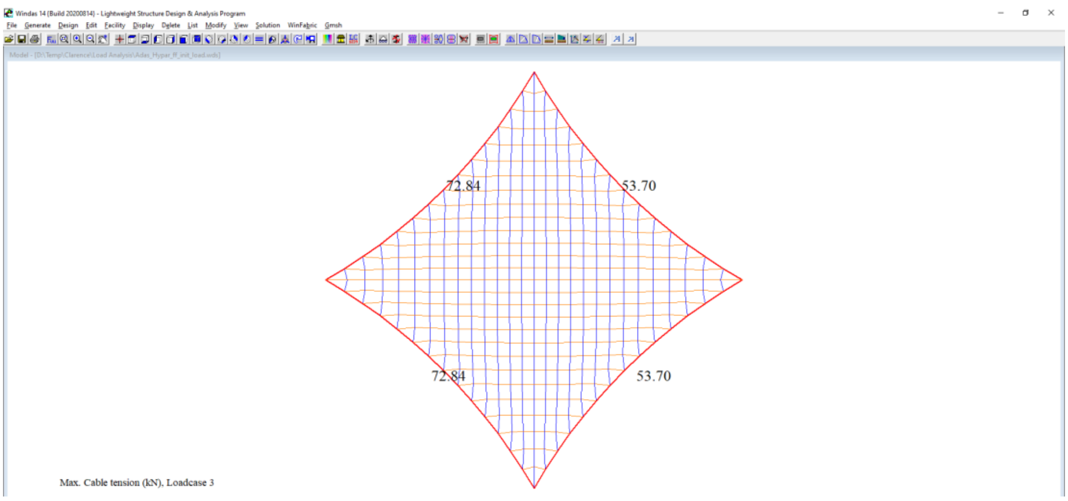
Load case 1 :



Load case 2 :



Load case 3 :



The support reactions can also be obtained.

Use the **List | Results | Reactions | All** command to list the reaction forces.

```
Windas Text Editor - [D:\Temp\Clarence\Load Analysis\Adas_Hypar_ff_init_load_reaction 1.lst]
File  Edit  Search  Help

Project : D:\Temp\Clarence\Load Analysis\Adas_Hypar_ff_init_load
Reaction forces listing
=======================
Load    Node      COORDINATES                                   Fixity      REACTION(kN)
Case    Number    X-COORD            Y-COORD        Z-COORD     Code             Rxx            Ryy            Rzz
  1       1            .000        -7071.068       6500.000      111             .000        -98.050         36.250
  2                                                                             6.350        -98.660         36.530
  3                                                                             7.010       -104.610         39.370
  1       2        7071.068             .000       3500.000      111           86.690           .000        -31.150
  2                                                                            87.030           .000        -34.810
  3                                                                            82.470           .000        -32.440
  1       3            .000         7071.068       6500.000      111             .000         98.050         36.250
  2                                                                             6.350         98.660         36.530
  3                                                                             7.010        104.610         39.370
  1       4       -7071.068             .000       3500.000      111          -86.690           .000        -31.150
  2                                                                          -103.340           .000        -32.840
  3                                                                          -100.100           .000        -31.040


 ===== Minimum and Maximum Reactions

       Minimum Rxx =      -103.340kN at Node      4 Maximum Rxx =       87.030kN at Node       2
       Minimum Ryy =      -104.610kN at Node      1 Maximum Ryy =      104.610kN at Node       3
       Minimum Rzz =       -34.810kN at Node      2 Maximum Rzz =       39.370kN at Node       1
```

19.5. Design Check and Dimensioning (as reference, not checked with Windas 14)

Design Information

Prior to design check and dimensioning, you need to have some idea about how the membrane is fabricated and assembled together.

The fabric Verseidag B18039B has a tensile strength of 70kN/m in the warp and the weft directions respectively.
Depend on the membrane type, the following are the high stress locations when dimensioning are

Location	Analysis Stress kN/m		Safety Factor	Design Stress (kN/m)	Direction	Comment
Max. stress at high point ring	-	x	1.5	-	Warp/Weft	
Max. stress at border cable (pocket)	-	x	1.5	-	Warp/Weft	A
Max. stress at clamping (System Point)	-	x	1.5	-	Warp/Weft	B
Max. stresses at seam	-	x	1.5	-	Weft	C
Max. stress at ridge/valley boundary	-	x	1.5	-		

required.

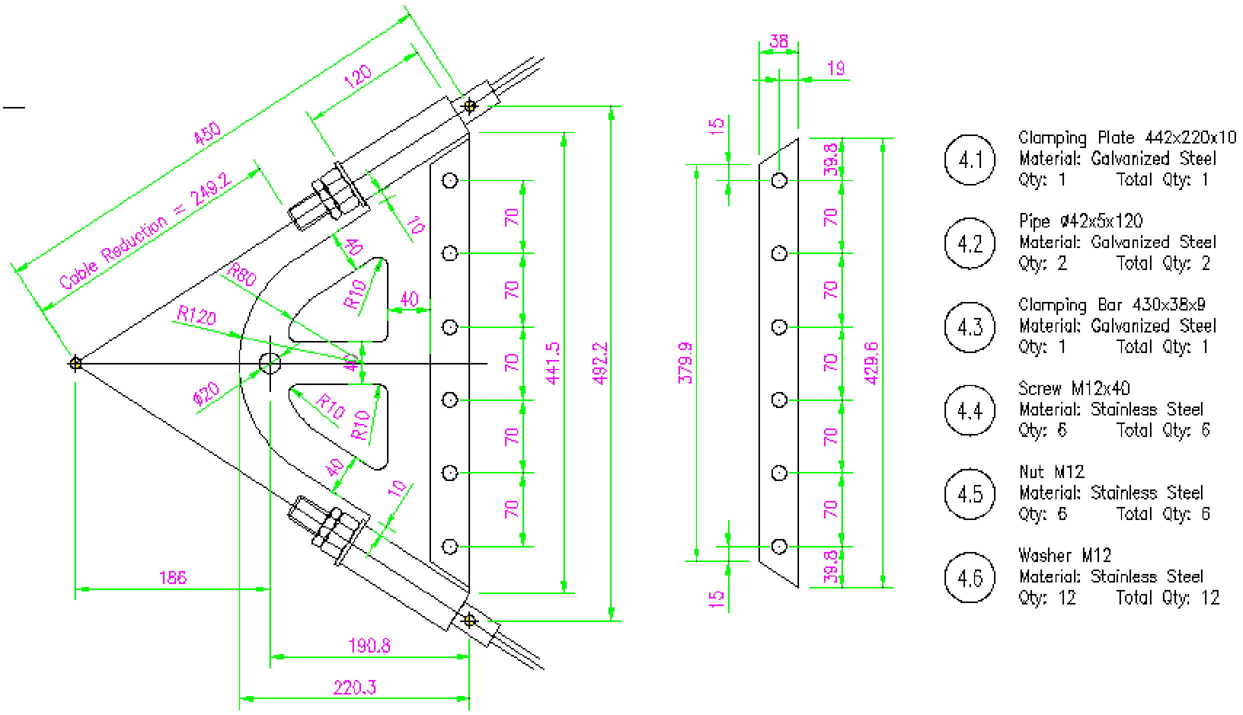
We normally start our design by determining the location of seam lines. Seam lines are line where the fabric panels are heated sealed or welded together. If the raw fabric comes in roll of 2.5m then the width of the fabric panel should be less than 2.5m. As welded seam lines are weaker then the parent material, it is important to check the strength of the seam and decide on the seam width.

High fabric stresses are normally found at the fabric corners and at the high point ring. It is necessary to reinforce these areas with double or more fabric layers. For this reason, we shall start with corner plate design first.

Corner Plate Design

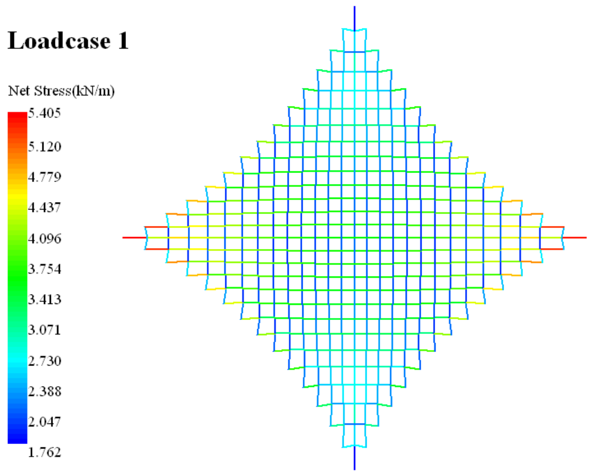
Corner plate is also known as clamping plate is use to connect the membrane to the steelwork. A typical corner plate detail is shown in the figure below. A corner plate comprises of a semi-circular profile plate with two tubes attached at both ends. The angular distance between the two tubes is determined from the membrane model.

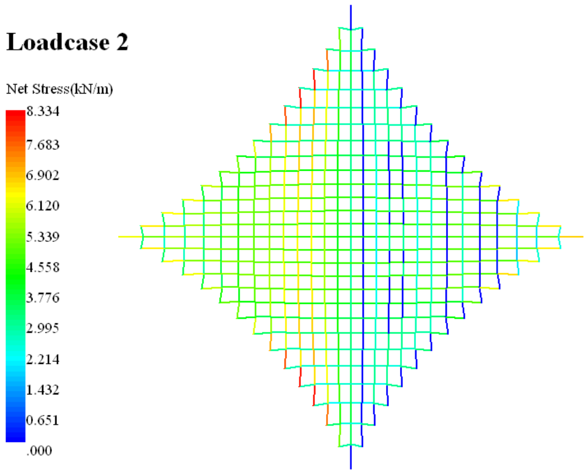
In this example, we assume the size of the corner plate at a distance 450mm from the system points.

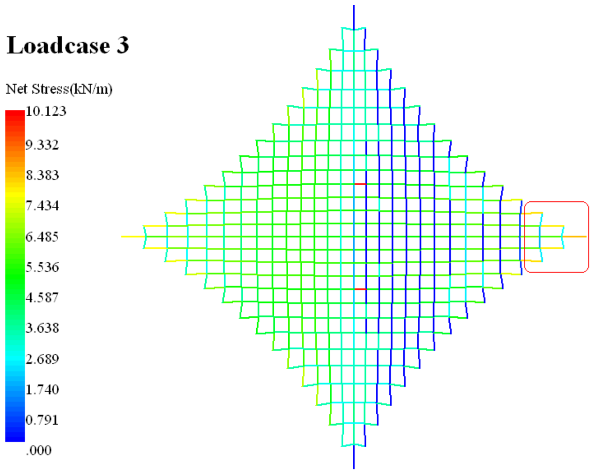


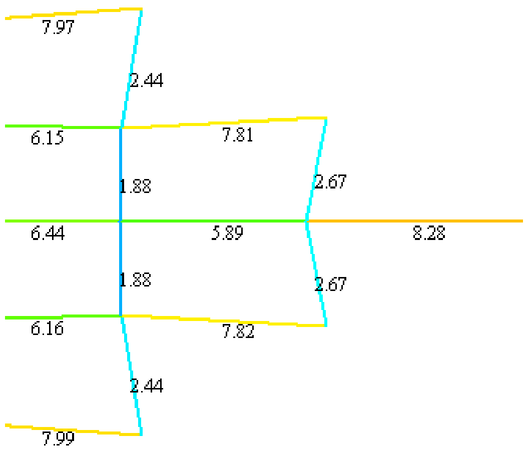
Fabric Stress At the corner

Use the **View | Result | Net Stress | All** command to study the fabric stress at the corners for each load case.









Maximum fabric stress at the corner is 8.3 kN/m. The design stress is 1.5 x 8.3 kN/m = 12.45 kN/m.

Corner Reinforcement

A corner reduction factor of 2.9 is used to take care of long term behavior, weather, high temperature, fabric's quality and membrane detail.

Recommended number of reinforcement = 2

Reinforced fabric strength = 70 x 2 = 140 kN/m

Limiting Corner Strength = 140/2.9 = 48 kN/m.

$$Corner\ strength\ \ check = \frac{Design\ Stress\ (12.45)}{Limiting\ \ Corner\ Strength} = 0.26 < 1.0\ O.K.$$

Border Cable Selection

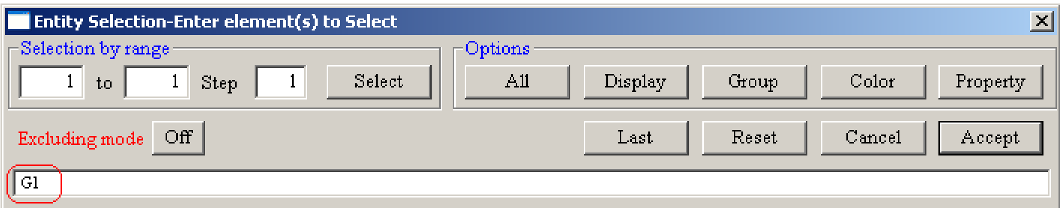
The easiest item to design for a membrane is the border cables. Select the **Design | Design Check | Border Cable** command to automatically determine the required border cables.

In this example, all four border cables should be of the same diameters due to wind loading. Select 14mm diameter 6 x 37 IWRC Galvanized Cable with a Min. breaking strength of 126 kN as the border cables.

Border Cable Length

The length of the cable for this membrane should be the same. The cable elements for the first border or segment have a group ID of 1. Use the **Display | Element | Normal** command to display border cable elements 1.

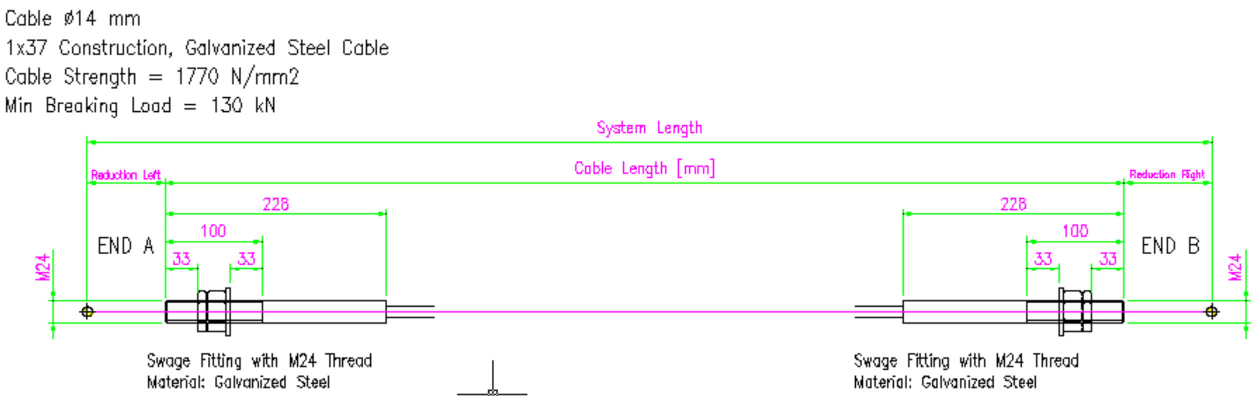
Use the **List | Element** command to find the cable length of segment 1. The cable length should be about 10.934m



Line	Element Number	Element Type	First Node	Second Node	Property Number	Group Number	Color Code	Member Length
1	2	13	18	1	3	1	14	835.15
2	8	13	32	18	3	1	14	808.12
3	21	13	45	32	3	1	14	787.63
4	42	13	57	45	3	1	14	772.03
5	71	13	68	57	3	1	14	760.75
6	108	13	78	68	3	1	14	753.43
7	153	13	87	78	3	1	14	749.84
8	206	13	95	87	3	1	14	749.84
9	267	13	102	95	3	1	14	753.43
10	336	13	108	102	3	1	14	760.75
11	413	13	113	108	3	1	14	772.03
12	498	13	117	113	3	1	14	787.63
13	591	13	120	117	3	1	14	808.12
14	692	13	2	120	3	1	14	835.15
							Sum =	10933.90

Border Cable With Terminal Ends

Each border cable is to be custom made to fit with the corner plate. A typical border cable comes with terminal at both ends. You are to get the terminal dimensions from the cable manufacturer and produce fabrication drawing as shown below:

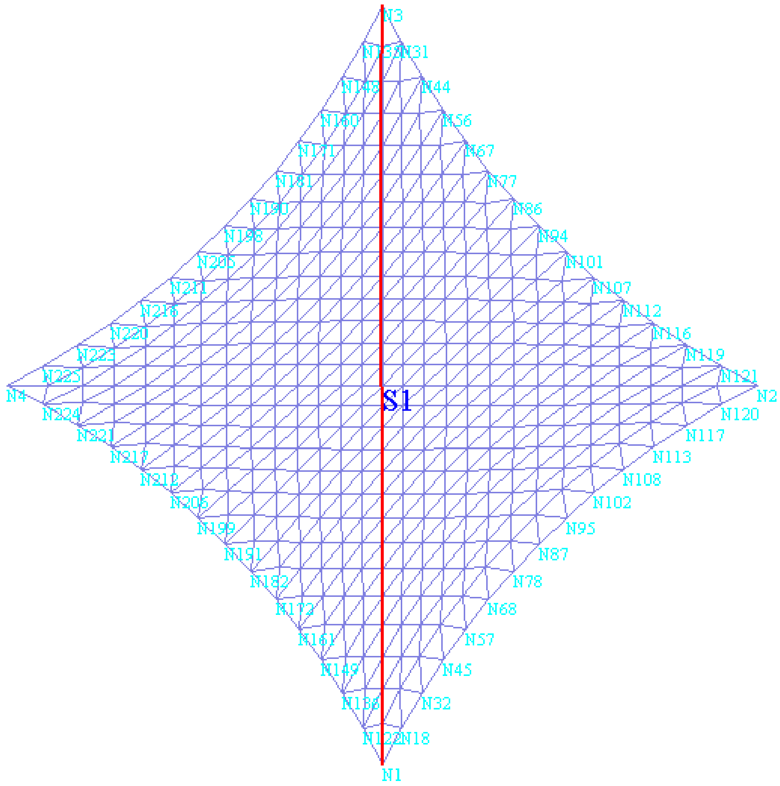


I.D.	QTY	CUTTING FORCE (kN)	CABLE LENGTH (mm)	REDUCTION LEFT (mm)	REDUCTION RIGHT (mm)	SYSTEM LENGTH (mm)	END A	END B
ECB1	1	10 kN	10484	249	249	10934	CPB1	CPB2
ECB2	1	10 kN	10484	249	249	10934	CPB2	CPB3
ECB3	1	10 kN	10484	249	249	10934	CPB3	CPB4
ECB4	1	10 kN	10484	249	249	10934	CPB4	CPB1

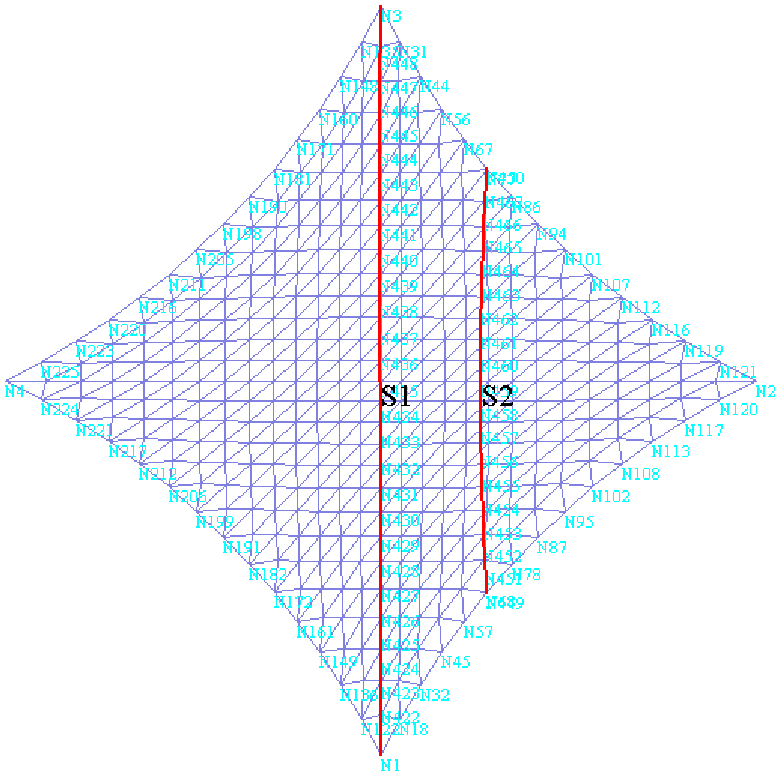
The system length is the length from one system point to another. The reduction length is the length adjustment for the corner plate. The cable length should be the length when the terminal is half open.

Seam line

Start seam line design with the **WinFabric | WinSeam** command. A new window appears. Select the **Generate | Seam Line | XyNet Geodesic | 2 Points** command to create a seam line passing through system point ① and ③.



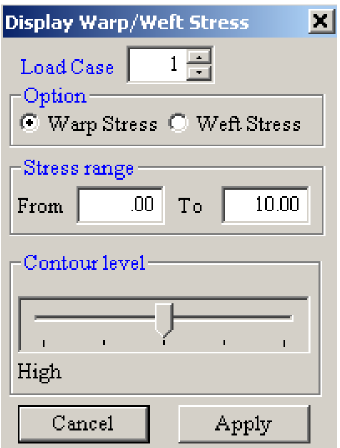
Select the **Generate | Seam Line | XyNet Geodesic | Offset** command to generate a seam line to the right of seam line S1.

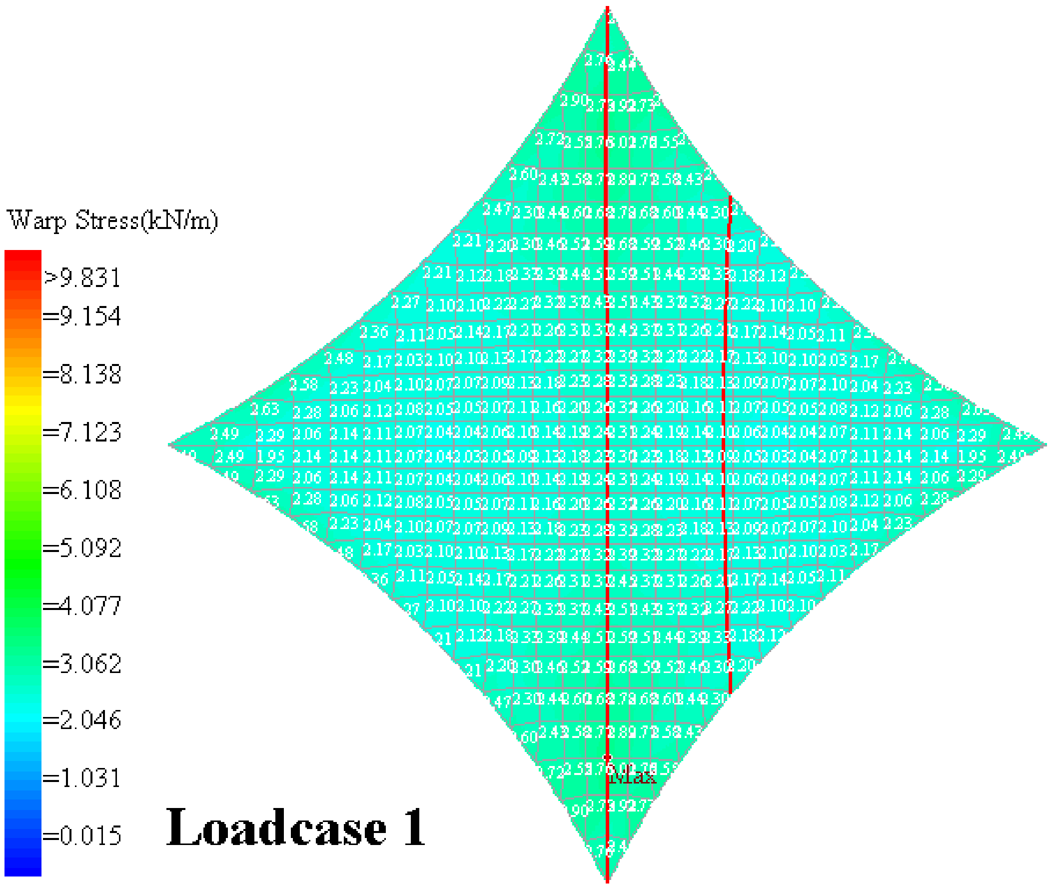


With two seam lines, it is possible to construct a fabric panel for patterning. We shall discuss about patterning in Chapter Twelve.

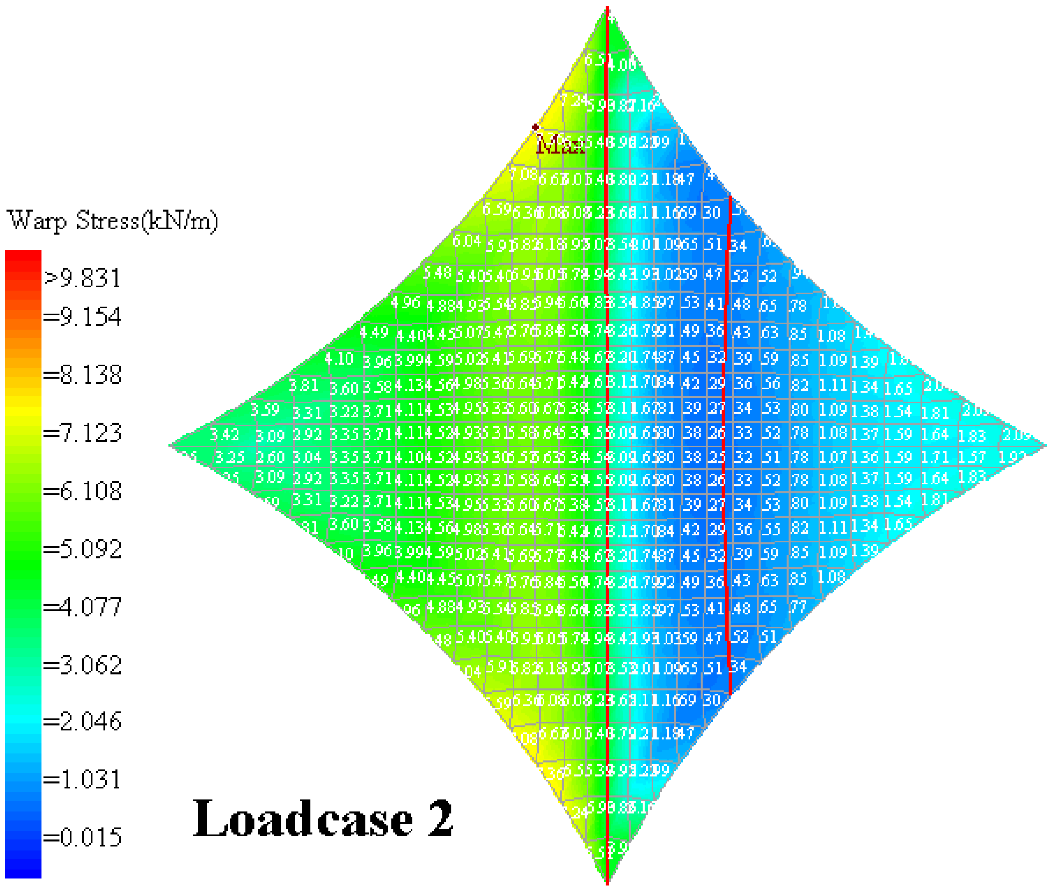
Seam Line Stress

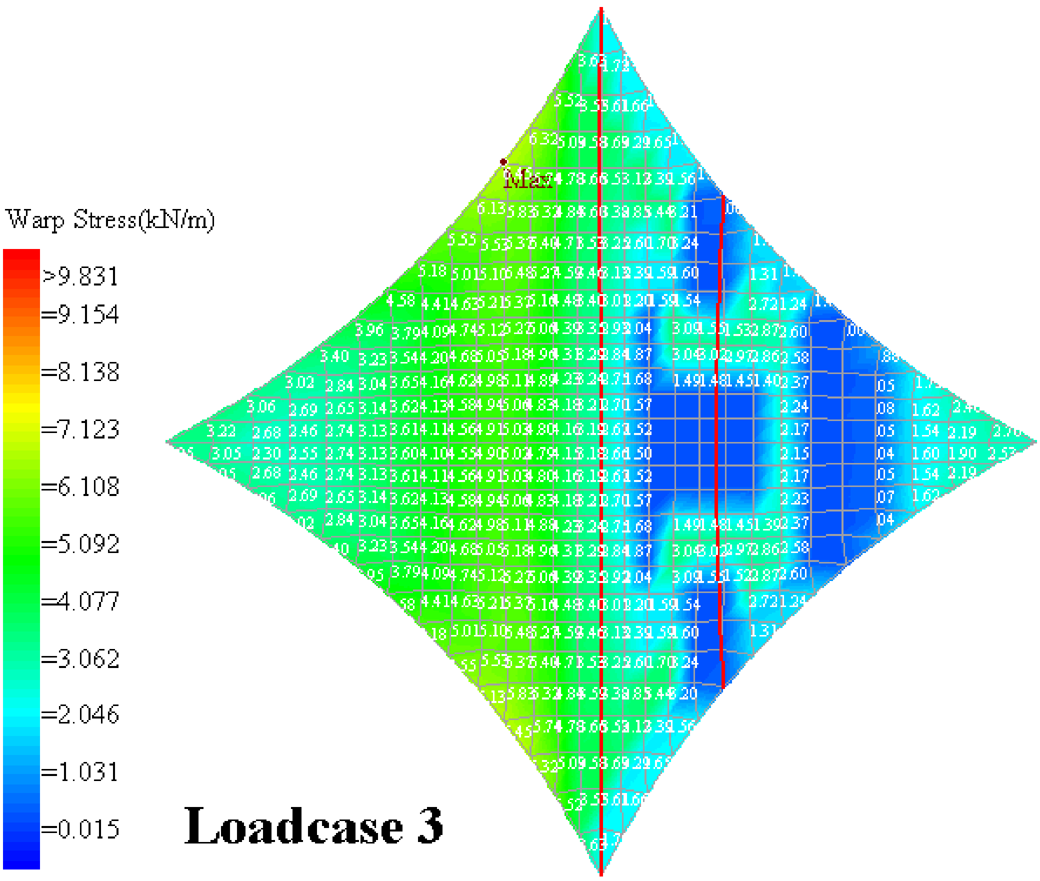
Use the **View | Fabric Stress** command to plot the fabric stress next to the seam lines.





Repeat for load case 2 and 3.


Warp Stress(kN/m)
>9.831
=9.154
=8.138
=7.123
=6.108
=5.092
=4.077
=3.062
=2.046
=1.031
=0.015
Max
Loadcase 2



Identify the greatest fabric stress along the seam line at 450mm away (corner plate length) from the system points. In this example, we shall take 7.0kN/m for seam design.

Seam Design.

The seam strength reduction factor of 3.9 is used to take care of long term behavior, weather, high temperature, fabric's quality and membrane detail.

Limiting Seam Strength = 70/3.9 = 17.9 kN/m.

$$Seam\ strength\ check = \frac{Design\ Seam\ Stress\ (7.0)}{Limiting\ \ Seam\ Strength} = 0.39 < 1.0 \quad O.K.$$