Windas 14 Tutorial Manual 2020

August 2020

Table of Contents

| Table | £ / | | |
|----------------|-----------------|---|----------|
| lable | 9 OT (| CONTENTS | F |
| 1. III | LTOQUC | | |
| 1.1. | For | | |
| 1.2. | Me | mbrane Forms | 5 |
| 2. AS | Simple | XY Net / Regular Net | 6 |
| 2.1. | Sys | tem Points | 6 |
| 2.2. | Ext | ernal Membrane Border | 7 |
| 2.3. | Per | form Form Finding | 8 |
| 2.4. | Mc | del Representation and Editable Features | 9 |
| 2.4 | 4.1. | Undo Form Finding | |
| 2.4 | 4.2. | Changing Mesh Origin and Check Point Stop | |
| 2.4 | 4.3. | Changing Fabric Mesh Width | |
| 2.4 | 4.4. | External Border Curvature | |
| 2.4 | 4.5. | Straight External Border | |
| 2.4 | 4.6. | Form Finding Checkpoint | |
| 2.4 | 4.7. | Positive Border Curve | 15 |
| 2.4 | 4.8. | Warp Angle to Global | 15 |
| 3. A S | Simple | Radial Membrane Net | |
| 3.1. | Sys | tem Point | |
| 3.2. | Ext | ernal Membrane Border | |
| 3.3. | Hig | h Point Ring | |
| 3.4. | For | m Finding | |
| 3.5. | Rad | lial Net Division Method | |
| 3.5 | 5.1. | Border Node Method | |
| 3.5 | 5.2. | Meridian Angle Method | 20 |
| 3.6. | Rad | lial Hoop Pre-Stress Ratio | 21 |
| 3.7. | Tilt | ed Mast | |
| 4. Ra | adiatin | g Net | |
| 4.1. | Svs | tem Points | |
| 4.2 | Ext | ernal Membrane Border | 22 |
| <u></u> ۲ ک | For | m Finding | |
| т.э. Д Л | Mo | dified Form | 23 24 |
| ч.ч. 5 D~ | | ault Forms | 24 عد |
| ј. Dd | ۷ اے ایا د+م | ndard Parrel Vault Form | 25 |
| J.I. | Jld C+- | ndard Parrol Vault with Eros Porder | 25 |
| 5.2. | Sta | | |
| 5.3. | IVIC | unied Barrei vault with Free Border | |

| 6. | Нура | ar Forms | 34 |
|-----|-------------|--|-----|
| 6. | 1. | A simple hypar | 34 |
| 6. | 2. | Four Hypar Sail | 35 |
| 7. | Radia | al Cones | 37 |
| 7. | 1. | Circular Base | 37 |
| 7. | 2. | Polygonal Base | 39 |
| 7. | 3. | Elliptical Tent | 42 |
| 8. | Umb | rella | 44 |
| 9. | Surfa | асе | 46 |
| 9. | 1. | Spline Surface | 46 |
| 9. | 2. | Revolved Surface | 47 |
| 9. | 3. | Sine Curve | 49 |
| | 9.4. | Sine And Cosine Curves | 49 |
| 10. | XY | ′ Mesh | 50 |
| 10 |).1. | System Point | 50 |
| 10 |).2. | External Membrane Border | 50 |
| 10 |).3. | Form Finding | 51 |
| 11. | XY | ' Net with Circular Cut-out | 53 |
| 11 | 1. | A Triple Cone | 53 |
| | 11.1 | 1. High Point Rings | 54 |
| | 11.1 | 2. Form Finding | 55 |
| 12. | Be | nded Membrane Form Finding | 57 |
| 13. | Ar | ch Ridge & Valley | 61 |
| 13 | 8.1. | External Arch | 61 |
| 13 | 8.2. | Internal Arch | 63 |
| 13 | .3. | Ridge and Valley Cable | 65 |
| 14. | Re | estrained Border in Radial Net | 68 |
| 15. | Re | estrained Border in Regular Net | 70 |
| 16. | Rc | osetta Cutout | 75 |
| 17. | М | ulti-cell Cushions Form Finding | 80 |
| 17 | '.1. | Standard Regular Forms / Random Shape Form | 80 |
| 17 | .2. | ETFE Single Cushion | 86 |
| 17 | .3. | ETFE MultiCell Cushions | 95 |
| 17 | . 4. | ETFE Mesh for Analysis1 | .03 |
| 18. | Ne | ew Element1 | .28 |
| 18 | 8.1. | Hybrid Plane Stress1 | .28 |

| 18.2. | Hybrid Brick | |
|---------|--|-----|
| 18.3. | Brick 8 (Solid 3D) | 132 |
| 19. Loa | d Analysis of A Simple Hypar | 133 |
| 19.1. | Material Property | 133 |
| 19.2. | Materialization | 136 |
| 19.3. | Loadings | 137 |
| 19.3.1 | . Load Combination 1 : Pre-stress + Dead Load + Rain Load | 139 |
| 19.3.2 | . Load Case 2 : Pre-stress + Dead Load + Wind Load | 141 |
| 19.3.3 | . Load Case 3: Pre-stress + Dead Load + Rain + Wind | 144 |
| 19.4. | Load Analysis | 144 |
| 19.5. | Design Check and Dimensioning (as reference, not checked with Windas 14) | 146 |
| Desigr | Information | 146 |
| Corne | r Plate Design | 147 |
| Borde | r Cable Selection | 149 |
| Seam | line | 150 |



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:



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.

| Generate Node | | | | | × |
|--|----------------------|-------------|-------|-------|-----|
| Coordinate System • Cartesian • Spherical | Input Unit Om Omm | Origin x | .00 Y | .00 Z | .00 |
| Enter Coordinates 0,0,0 | | | | | |

| Node ID | Х | 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.

| WinFabric System Variables | | | | | |
|--|-----------------------------|--|--|--|--|
| Eorm Finding | Precision and Tolerance | | | | |
| Fabric <u>n</u> et type • Regular • Radial | Warp-weft angle 5.0 | | | | |
| Number of iteration 2 | Minimum triangular angle | | | | |
| Number of points for border segments 8 | Arch constructor node 5.0 | | | | |
| Sag amount in % for border segment | Minimum cable length 100.00 | | | | |
| Force density for fixed border segments .00 | Minimum net length 50.00 | | | | |
| h-Contour Interval 50.00 | Accept | | | | |
| Minimum rainwater runoff 7.50 | | | | | |
| Scaling factor for symbol display 1.00 | | | | | |

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.



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:

| Generate Regular Net | X |
|---|-------------------------------|
| Warp-Weft stress ratio 1.0 🔹 | Prestress (warp), kN/m 1.00 👻 |
| Fabric mesh width, mm | Warp angle to global |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | n point O Yes O No Accept |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -1000.00 Y - | 1000.00 Stop at check point 0 |

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

| Generate Regular Net | × |
|---|-------------------------------|
| Warp-Weft stress ratio 1.0 💌 | Prestress (warp), kN/m 1.00 💌 |
| Fabric mesh width, mm 1000.00 🔹 | Warp angle to global |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | apoint O Yes O No |
| Automatic form finding 📀 | Yes C No Cancel |
| Mesh origin at X -1250 Y | -1250 Stop at check point 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)

| Generate Regular Net | × |
|---|-------------------------------|
| Warp-Weft stress ratio 1.0 💼 | Prestress (warp), kN/m 1.00 📩 |
| Fabric mesh width, mm | Warp angle to global |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | m point C Yes • No Accept |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -750.00 Y | -750.00 Stop at check point 0 |
| | |

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.

| Xynet Fo | Xynet Form Finding - Net Surface 🛛 🔀 | | | | | |
|----------|--------------------------------------|-------------------------|-----------------------|--|--|--|
| ? | Some nets have no associated me | embrane surface, Contin | ue with form finding? | | | |
| | Yes | No | | | | |

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.

| Edit external borders | | × |
|-----------------------|----------------------------|----------|
| Border segment "B" | 3 📩 Number of points | 8 |
| Sag amount by % of | 7.5 or by distance of | 750.0 mm |
| Restrained segment | C Yes 💿 No | |
| Curvature kind | • Negative C Positive(Out) | |
| Update | Update All Done | |

Perform form finding with the same parameters below,

| Generate Regular Net | x |
|---|-------------------------------|
| Warp-Weft stress ratio 1.0 👘 | Prestress (warp), kN/m 1.00 🔹 |
| Fabric mesh width, mm 1000 🔹 | Warp angle to global |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest syster | n point C Yes • No |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -1000.00 Y - | 1000.00 Stop at check point 0 |

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)

| Edit external borders | | × |
|-----------------------|----------------------------|----------|
| Border segment "B" | 1 📑 Number of points | 8 |
| Sag amount by % of | or by distance of | 750.0 mm |
| Restrained segment | • Yes C No | |
| Curvature kind | • Negative C Positive(Out) | |
| Update | Update All Done | |

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.

| Generate Regular Net | X |
|---|-------------------------------|
| Warp-Weft stress ratio 1.0 👘 | Prestress (warp), kN/m 1.00 🔹 |
| Fabric mesh width, mm | Warp angle to global0 |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | n point OYes ONo |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -1000.00 Y | 1000.00 Stop at check point 0 |

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.

| Xy Net Form Finding Parameters | | | | |
|--|--|--|--|--|
| Warp-Weft stress ratio 1.0 + Prestress (warp), kN/m 1.00 + | | | | |
| Eabric mesh width, mm 250 · Warp angle to global .00 · | | | | |
| Minimum internal net point from border 25.00 | | | | |
| Align membrane center to nearest system point C Yes • No | | | | |
| Automatic form finding | | | | |
| Mesh origin at X -250.00 Y -250.00 Stop at check point : | | | | |

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.

| Edit external borders | X | | | | |
|---|-------|--|--|--|--|
| Border segment "B" | 8 | | | | |
| Sag amount by % of 10.0 - or by distance of | .0 mm | | | | |
| Restrained segment 🕑 Yes 🗢 No | | | | | |
| Curvature kind C Negative Positive(Out) | | | | | |
| Update Update All Done | | | | | |

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 | Х | 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.

| WinFabric System Variables | | | × |
|--|---|---------------------------|--------|
| Form Finding | 1 | Precision and Tolerance — | |
| Fabric net type 🔹 C Regular 💽 Radial | | Warp-weft angle | 5.0 |
| Number of iteration 3 | | Minimum triangular angle | 1.5 |
| Number of points for border segments 8 | | Arch constructor node | 5.0 |
| Sag amount in % for border segment 10.0 | | Minimum cable length | 100.00 |
| Force density for fixed border segments 1.00 | | Minimum net length | 50.00 |
| h-Contour Interval 50.00 | | · | Accept |

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.

| Radial Net Form Finding Parameters | | | | |
|---|--|--|--|--|
| Parameters Element size in radial direction 1000.00 ÷ | | | | |
| Meridian angle | | | | |
| Radial/Hoop pre-stress ratio | | | | |
| Pre-stress in radial direction, kN/m 1.0 | | | | |
| Start angle | | | | |
| Division by 🖲 Border node 🛛 Meridian angle | | | | |
| Constant radial force density? C Yes 📀 No | | | | |
| Automatic form finding? | | | | |
| Cancel Apply Radial is the warp and hoop is the weft direction | | | | |

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.

| Edit External Borders × | | | | | |
|--|--|--|--|--|--|
| Border segment "B" 1 : Number of points 1 | | | | | |
| Sag amount by % of 10.00 r or by distance of 1000.0 mm | | | | | |
| Restrained segment C Yes C No | | | | | |
| Curvature kind | | | | | |
| Update Update All Done | | | | | |



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.

| Radial Net Form Finding Parameters |
|---|
| Parameters Element size in radial direction |
| Meridian angle |
| <u>R</u> adial/Hoop pre-stress ratio 2.0 <u>↓</u> |
| Pre-stress in radial direction, kN/m 1.0 |
| Start angle |
| Division by C Border node 📀 Meridian angle |
| Constant radial force density? C Yes . • No |
| Automatic form finding? • Yes • No |
| <u>Cancel</u> <u>Apply</u> |
| Radial is the warp and hoop is the weft direction |



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.



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.

| Edit Fabric Top Ring | | × |
|---------------------------|-----------|--------|
| Top Ring ID 1 💌 Radius | 1000.00 | Apply |
| Top Ring Center .00 Y .00 | Z 3000.00 | Cancel |
| Base of Mast 1000.00 Y | Z .00 | |

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.

| Generate Node | | | | | × |
|--|------------|-------------|------|-------|-----|
| Coordinate System • Cartesian • Spherical | Input Unit | Origin X | 00 Y | .00 Z | .00 |
| Enter Coordinates 0,0,0 | | | | | |

| Node ID | Х | 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**.

| WinFabric System Variables | | | | × |
|---|--------|------------------|-------------|----------|
| Form Finding | | Precision and To | lerance ——— | |
| Fabric net type 💿 Regular 🔿 | Radial | Warp-weft angle | | 5.0 |
| Number of iteration | 3 | Minimum triangu | lar angle | 1.5 |
| Number of points for border segments | 8 | Arch constructor | node | 5.0 |
| Sag amount in % for border segment | 10.0 | Minimum cable le | ngth [| 100.00 |
| Force density for fixed border segments | 1.00 | Minimum net len; | gth [| 50.00 |
| h-Contour Interval | 50.00 | | Acc | ept |
| | | | | <u> </u> |

4.3. Form Finding

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

| Radial Net Form Finding Parameters 🛛 🛛 🗙 | | | |
|--|-------------------------|--|--|
| -Parameters | | | |
| Element size in radial dire | tion 1000.00 - | | |
| Meridian angle | 10.00 | | |
| Radial/Hoop pre-stress ra | tio 2.0 🔹 | | |
| Pre-stress in radial direction | on, kN/m 1.0 | | |
| Start angle | .00 💌 | | |
| Division by 💿 Border nod | le 🔿 Meridian angle | | |
| Constant radial force densi | ty? 🔿 Yes 💿 No | | |
| Automatic form finding? | • Yes • No | | |
| Cancel | Apply | | |
| Radial is the warp and hoor | o is the weft direction | | |

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 | Х | 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.

| Generate Standard Membrane Barrel Vault Form | | | |
|--|----------------------------|-----------------|--|
| Overall dimensionsChord10000.000Height3000.000 | Оrigin X .000 Y .000 | Apply Cancel | |
| Span 50000.000 | Z .000 | | |
| Number of internal arches | 3 | | |

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

| WinFabric System Variables | | | |
|--|----------------------------------|--|--|
| Eorm Finding | Precision and Tolerance | | |
| Fabric <u>n</u> et type • Regular C Radial | Warp-weft angle 5.0 | | |
| Number of iteration 2 | Minimum triangular angle 1.5 | | |
| Number of points for border segments 8 | Arch constructor node 5.0 | | |
| Sag amount in % for border segment 7.5 | Minimum cable length 100.00 | | |
| Force density for fixed border segments 1.00 | Minimum <u>n</u> et length 50.00 | | |
| h-Contour Interval 50.00 | Accept | | |
| Minimum rainwater runoff 7.50 | | | |
| Scaling factor for symbol display 1.00 | | | |

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

| Xy Net Form Finding Parar | neters | | × |
|----------------------------|--------------|------------------------|----------|
| Warp-Weft stress ratio | 2.0 🗧 | Prestress (warp), kN/m | 1.00 |
| Fabric mesh width, mm | 1000.00 📫 | Warp angle to global | .00 |
| Minimum internal net point | t from borde | r 25.00 | |
| Align membrane center to : | nearest syst | empoint C Yes • No | Accept |
| Automatic form finding | (| •Yes O No | Cancel |
| Mesh origin at X | .00 Y | .00 Stop at check po | oint 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.

| Edit external borders | × |
|-----------------------|-----------------------------|
| Border segment "B" | 1 Number of points 8 |
| Sag amount by % of | 10.0 r by distance of .0 mm |
| Restrained segment | C Yes 💽 No |
| Curvature kind | 💿 Negative 🔿 Positive(Out) |
| Update | Update All Done |

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.



31

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

| Node ID | Х | 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 :

| Edit Fabric Arches | | Edit Fabric Arches | × |
|-------------------------|-------------------------------|---------------------------------|---------------|
| Arch ID | | Arch ID 1 💌 | |
| <u>B</u> egin 8183.918 | 1441.360 .000 Cancel | Begin 3889.018 3066.110 | .000 Cancel |
| Mid 9572.809 | 4026.270 900.000 Apply | Mid 5974.729 5777.689 | 200.000 Apply |
| <u>E</u> nd 10961.690 | 6611.189 1340.000 | End 8060.438 8489.270 | -600.000 |
| Edit Fabric Arches | | Edit Fabric Arches | × |
| Arch ID | | Arch ID | |
| <u>B</u> egin 11794.720 | .000 Cancel | <u>B</u> egin 15119.330 107.680 | .000 Cancel |
| <u>M</u> id 12697.350 | 2989.561 100.000 Apply | Mid 15647.360 2374.661 | 800.000 Apply |
| <u>E</u> nd 13599.980 | 5418.279 -600.000 | End 16175.390 4641.641 | 1190.000 |
| | Edit Fabric Arches | × | |
| | Arch ID 5 | | |
| | Begin 19140.880 .00 | 0 .000 Cancel | |
| | <u>M</u> id 19293.060 2071.40 | 1 100.000 <u>Apply</u> | |
| | <u>E</u> nd 19445.240 4142.80 | 1 -600.000 | |

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.

| Generate Hypar Membrane Form 🛛 🗙 | | | |
|----------------------------------|--------------|-------|--------|
| ⊢Hypar | Geometry | | |
| Width | 10000.0 | Depth | 3000.0 |
| Sag am | ount by % of | | 7.5 - |
| | Cancel | Apr | bly |

Click **Accept** and a hyper 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.

| Xy Net Form Finding Parameters | × |
|--|-------------------------------|
| Warp-Weft stress ratio | Prestress (warp), kN/m 3.00 + |
| Fabric mesh width, mm | Warp angle to global .00 + |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system point C Yes • No | |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -7500.00 Y -7500.00 Stop at check point 0 - | |

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.

| Generate Conical Tent | | × |
|---------------------------|--|---------------|
| Tent Geometry | Tent Parameters | Origin |
| Outer radius 15000.0 | Element size in 1000.0 radial direction | X .000 Apply |
| Inner radius 2500.0 | Meridian angle 7.50 | Y .000 Cancel |
| High Point 6000.0 Ring | Radial/Hoop 3.00 Prestress Ratio | Z .000 |
| | Prestress, kN/m 1.00 | |

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.



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.

| (| Generate Radial Net 🛛 🗶 |
|---|---|
| | Parameters |
| | Element size in radial direction |
| | Meridian angle |
| | Radial/Hoop pre-stress ratio |
| | Pre-stress in radial direction, kN/m 1.0 |
| | Start angle |
| | Division by 🔿 Border node 💽 Meridian angle |
| | Constant radial force density? 💿 Yes 🗢 No |
| | Automatic form finding? 💽 Yes 🔿 No |
| | Cancel Apply |
| | Radial is the warp and hoop is the weft direction |



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:

| Generate tent with elliptical ba | se | × |
|----------------------------------|---------------------------------------|--------|
| Ellipse dimensions | High point ring & mast | |
| Major, a 6950.000 | Ring center X 4200.00 Y .00 Z 4700.00 | Apply |
| Minor, b 4100.000 | Mast base X 2750.00 Y .00 Z -3700.00 | Cancel |
| Number of division 36 | Top Ring Radius 500.00 | |

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.

| Radial Net Form Finding Parameters 🛛 🗙 🗙 | Ì |
|---|---|
| Parameters | |
| Element size in radial direction 1000.00 | |
| Meridian angle | |
| Radial/Hoop pre-stress ratio | |
| Pre-stress in radial direction, kN/m 1.0 | |
| Start angle | |
| Division by 💽 Border node) 🔿 Meridian angle | |
| Constant radial force density? 🔿 Yes 🛛 💿 No | |
| Automatic form finding? 💿 Yes 🔿 No | |
| Cancel Apply | |
| Radial is the warp and hoop is the weft direction | |



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.

| Generate Umb | rella (isoparan | netric) | × |
|------------------------|------------------|--------------------|----------------|
| Tent Geometr | у | Membrane Border P | arameters |
| Breadth | 6000.0 | Sag amount by % of | 10.0 - |
| Width | 6000.0 | Number of points | 9 |
| Height | 2000.0 | Cancel | Apply |
| Ring Radius | 500.0 | | |
| | | | |
| Generate Umbr | ella | | × |
| Parameters - | | | |
| <u>E</u> lement size { | Radial Directio | n} 500 ÷ | <u>A</u> pply |
| <u>R</u> adial/Hoop I | PreStress ratio | 2.0 - | <u>C</u> ancel |
| PreStress in ra | idial direction, | kN/m 1.0 | |
| <u>M</u> eridian angl | e | 10.00 + | |
| Fixed all edges | ;? | ○ <u>Y</u> es | |





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.

| | Generate node along a sine wave | × |
|----------------------------|---|---|
| | Length for one cycle 10000.00 Peak value | 1000.00 |
| | Number of points representing a complete cycle | 24 💌 |
| | Number of cycles | OK |
| | | Cancel |
| N3 N4 N5 N6 N3 N2 N1 | 6 N7 N8 N9 N10 N11 N12 N13 N14 N15 | N25 N24 N16 N17 N18 N19 N20 N21 N20 N21 |

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.

| Node Selection - Ruled Membrane Surface | | | | × |
|---|----------|-----------|--------|----------|
| Selection by range 1 to 1 Step 1 Select | All Disp | lay Group | Color | Property |
| | La | st Reset | Cancel | Accept |
| 1-25 | | | | |



| Directional vector for ruled surface | | | | | | |
|--------------------------------------|----------------|----------|---|----|--------|--|
| -Select two | points —— X | Y | Z | | Cancel | |
| Point 1 | .00 | .00 | |)0 | Accept | |
| Point 2 | .00 | 10000.00 | | 0 | | |

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.

| | Generate node along a sine wave | |
|----------------|---|---------|
| | Length for one cycle 10000.00 Peak value 1000.00 | |
| | Number of points representing a complete cycle 24 💌 | |
| | Number of cycles | |
| | Cancel | |
| N5 N6 N7 N3 | N8 N9 N10 N11 | |
| N2 | N12 | |
| N1 | N13 | N25 |
| | N14 | N24 |
| | N15 | N23 |
| | N17 N18 N17 | N20 N21 |
| | 1416 NIS | / 1920 |

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.

| 🔲 Node Selection - Revolved Membrane Surfa 🗙 | | | | | |
|--|----------|---------|---------|--|--|
| Rotational Axis | | | | | |
| | X | Y | Z | | |
| Point 1 | .00 | .00 | .00 | | |
| Point 2 | .00 | .00 | 1000.00 | | |
| Rotations | al angle | 15.00 🔹 | Accept | | |
| Number o | freplica | 24 🕂 | Cancel | | |





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.

| Generate membrane sine surface 🛛 🗙 | | | | |
|------------------------------------|------------------|---------|--|--|
| Length for one cycle 1000 | 0.00 Peak value | 1000.00 | | |
| Number of points representing | 24 🔹 | | | |
| Number of cycles | 3 . [| OK | | |
| Surface depth along y-axis | 10000.00 | Cancel | | |



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

| 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 |

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

10.2. External Membrane Border

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

| External Border -Select nodes in anti-clockwise order | × |
|---|---------------------------------|
| Selection by range 1 to 1 Select All | Display Group Color Property |
| | Last Reset Cancel Accept |
| 1-8 | |
| | |
| WinFabric System Variables | × |
| Form Finding | |
| Fabric net type 💿 Regular 🔿 Radial | Warp-weft angle 5.0 |
| Number of iteration | 2 Minimum triangular angle 1.5 |
| Number of points for border segments | 8 Arch constructor node 5.0 |
| Sag amount in % for border segment 10 | 0.0 Minimum cable length 100.00 |
| Force density for fixed border segments | 00 Minimum net length 50.00 |
| h-Contour Interval 50. | 00 Accept |
| | |

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



53

Copyright © 2020 Multimedia Engineering Pte Ltd

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

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**.

| WinFabric System Variables | × |
|---|------------------------------|
| - Eom Finding | Precision and Tolerance |
| Fabric net type 📀 Regular 🔿 Radial | Warp-weft angle 5.0 |
| Number of iteration 2 | Minimum triangular angle 1.5 |
| Number of points for border segments 8 | Arch constructor node 5.0 |
| Sag amount in % for border segment 7.5 | Minimum cable length 100.00 |
| Force density for fixed border segments .00 | Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |
| Minimum rainwater runoff 7.50 | |
| Scaling factor for symbol display 1.00 | |

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,



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.

| Xy Net Form Finding Parameters | X |
|---|---|
| Warp-Weft stress ratio 3.0 | Prestress (warp), kN/m 1.00 + |
| Fabric mesh width, mm | Warp angle to global .00 - |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | m point ○ <u>Y</u> es ⊙ <u>N</u> o <u>A</u> ccept |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at \underline{X} -1200.00 \underline{Y} | -8400.00 Stop at check point 0 |

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.

| Formfinding of Xynet with circular cutout | × |
|---|------------------------------|
| Specify factor for radial effect on Ring ID 1 | 4.00 <u>→</u> <u>A</u> ccept |

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

| Ring ID 1 - Formfinding of Xynet with | × | |
|---------------------------------------|-----|--------|
| Number of weft lines for Ring ID 1 | 8 - | Accept |

A triple cone membrane will then be formed.





σ×

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.

| WinFabric System Variables | × |
|---|------------------------------|
| r <u>F</u> orm Finding | Precision and Tolerance |
| Fabric <u>n</u> et type 📀 Regular C Radial | Warp-weft angle 5.0 |
| Number of iteration 2 | Minimum triangular angle 1.5 |
| Number of points for border segments 8 | Arch constructor node 5.0 |
| Sag amount in % for border segment 7.5 | Minimum cable length 100.00 |
| Force density for fixed border segments .00 | Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |
| Minimum rainwater runoff 7.50 | |
| Scaling factor for symbol display 1.00 | |



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.

| Xy Net Form Finding Parameters | × | | | |
|--|-------------------------------|--|--|--|
| Warp-Weft stress ratio 1.0 📩 | Prestress (warp), kN/m | | | |
| Fabric mesh width, mm 500.00 | Warp angle to global .00 + | | | |
| Minimum internal net point from border | 25.00 | | | |
| Align membrane center to nearest system point C Yes • No | | | | |
| Automatic form finding | Yes C No Cancel | | | |
| Mesh origin at X -500.00 Y | -500.00 Stop at check point 0 | | | |

| Windas 14 (Build 20200814) - Lightweight Stri File Generate Design Frit Facility Disp | ucture Design & Analysis Program Jav Delete List Modify View Solution WinFahri | r Grede | | | | | - ø × |
|--|---|----------|----------|-------------|--------------------------------------|------------|----------|
| | | | | | | | |
| Model - ID\\Temp\Clarence\3,1,13,a,Flatten For | rm Finding ff.wds] | | | | | | |
| | | | | | | | |
| | | | | B5 | | | |
| | | | | | | | |
| Front width is 66, if > 250, perform | out of core form finding | | | | | | <u>^</u> |
| Out of core form finding | | | | | | | |
| Model saved | | | | | | | |
| Model saved | | | | | | | |
| Elements= 2254 | Nodes= 783 | Membrane | Color= 2 | Pronetty= 3 | Load Case= 1. Load Action=Individual | Group=E0 N | |

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 | Х | 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.

| WinFabric System Variables | | | × |
|---|--------|--------------------------------|--------|
| _ <u>F</u> orm Finding | | <u>Precision and Tolerance</u> | |
| Fabric <u>n</u> et type • Regular O | Radial | Warp-weft angle | 5.0 |
| Number of iteration | 2 | Minimum triangular angle | 1.5 |
| Number of points for border segments | 8 | Arch constructor node | 5.0 |
| Sag amount in % for border segment | 7.5 | Minimum cable length | 100.00 |
| Force density for fixed border segments | .00 | Minimum net length | 50.00 |
| h- <u>C</u> ontour Interval | 50.00 | | Accept |
| Minimum rainwater runoff | 7.50 | | |
| Scaling factor for symbol display | 1.00 | | |

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.

| Generate regular membrane arch | | | | | x |
|--------------------------------|----------|------|------|--------|---|
| Begin | .00 | .00 | .00 | OK | |
| Mid | 5000 | 1500 | 3000 | Cancel | |
| End | 10000.00 | .00 | .00 | | |

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.

| Edit - Fabric Arch, Ridge and Valley Cable | | | | |
|--|----------|--------------|---------------|----------------|
| Arch ID | 1 📩 Type | 🔿 External 🤅 | • Internal) O | Ridge & Valley |
| Start Point | .00 | .00 | .00 | Apply |
| Mid Point | 5000.00 | 5000.00 | 3000.00 | Cancel |
| End Point | 10000.00 | 10000.00 | .00 | |

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.

| Generate Regular Net | × |
|---|---------------------------------|
| Warp-Weft stress ratio 1.0 👘 | Prestress (warp), kN/m 1.00 🔺 |
| Fabric mesh width, mm 500.00 | Warp angle to global0 💌 |
| Minimum internal net point from border | 5.00 |
| Align membrane center to nearest system | n point OYes • No |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at X -500.00 Y | -500.00 Stop at check point 0 🐳 |

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 | Х | 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.

| Xy Net Form Finding Parameters | × |
|--|-------------------------------|
| Warp-Weft stress ratio 1.0 + | Prestress (warp), kN/m 1.00 + |
| Fabric mesh width, mm | Warp angle to global00 |
| Minimum internal net point from border | 25.00 |
| Align membrane center to nearest system | n point C Yes C No |
| Automatic form finding | Yes C No Cancel |
| Mesh origin at \underline{X} -12000.00 \underline{Y} - | 1000.00 Stop at check point 0 |

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.

| Generate ridge and valley cable | | | | | × |
|---------------------------------|-----|----------|---------|--------|---|
| Begin | .00 | .00 | 7500.00 | OK | |
| Mid | .00 | 7500 | 6000 | Cancel | |
| End | .00 | 15000.00 | 7500.00 | | |



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.

| Tent - Polygon | × |
|--------------------|----------------------------|
| Geometry | Membrane Border Parameters |
| Base Radius 2500.0 | Sag amount by % of 10.0 📑 |
| Height 1500.0 | Number of points 9 |
| Ring Radius 250.0 | Cancel Apply |
| Polygon Hexagon 💌 | |

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

| R | adial Net Form Finding Parameters |
|---|--|
| F | Parameters |
| : | Element size in radial direction |
| : | Meridian angle |
| | Radial/Hoop pre-stress ratio |
| | Pre-stress in radial direction, kN/m 1.0 |
| | Start angle 00 ÷ |
| D | ivision by 🖲 Border node 🔿 Meridian angle |
| c | onstant radial force density? C Yes 📀 No |
| А | utomatic form finding? 💿 Yes 🔿 No |
| | Cancel Apply |
| R | adial is the warp and hoop is the weft direction |
| | |

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.



| 💳 Genera | Generate regular membrane arch | | | |
|--|--------------------------------|---------|-----|--------|
| Select tw | o points — X | Y | Z | Accept |
| Point 1 | 2500.00 | .00 | .00 | Cancel |
| Point 2 | 1250.00 | 2165.06 | .00 | |
| Arch height at mid point from ground 1000.00 | | | | |

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

| Edit - Fabric | Arch, Ridge ar | nd Valley Cable | 2 | x |
|---------------|----------------|-----------------|------------|----------------|
| Arch ID | 1 📩 Туре | • External (| Internal C | Ridge & Valley |
| Start Point | 2500.00 | .00 | .00 | Apply |
| Mid Point (| 2000.00 | 1500.00 | 1000.00 | Cancel |
| End Point | 1250.00 | 2165.06 | .00 | |

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.

| Edit external borders | X |
|---|----------|
| Border segment "B" 1 Number of points | 9 |
| Sag amount by % of 10.0 👘 or by distance of | 250.0 mm |
| Restrained segment 💽 Yes 🔿 No | |
| Curvature kind C Negative Positive(Out) | |
| Update Update All Done | |

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



Copyright © 2020 Multimedia Engineering Pte Ltd

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 | Х | 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.

| WinFabric System Variables | × |
|---|------------------------------|
| Form Finding | Precision and Tolerance |
| Fabric net type 💿 Regular 🔿 Radial | Warp-weft angle 5.0 |
| Number of iteration 2 | Minimum triangular angle 1.5 |
| Number of points for border segments 8 | Arch constructor node 5.0 |
| Sag amount in % for border segment 10.0 | Minimum cable length 100.00 |
| Force density for fixed border segments .00 | Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |



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.

| Generate regular membrane arch | | | | |
|---|--|-------------------|----------|------------------|
| -Select tw | o points —— | | | |
| | x | Y | Ζ | Accept |
| Point 1 | .00 | 2000.00 | .00 | Cancel |
| Point 2 | 2500.00 | 2000.00 | .00 | 2 |
| Arch height at mid point from ground 750.00 | | | | |
| | | | | |
| 🧮 Genera | te regular m | embrane arch | | X |
| Genera Select tw | te regular m 10 points — | embrane arch | | × |
| Genera Select tw | te regular m ro points | embrane arch Y | Z | Accept |
| Genera Select tw Point 1 | te regular m ro points X .00 | embrane arch Y | Z .00 | Accept Cancel |
| Genera Select tw Point 1 Point 2 | te regular m ro points X .00 2500.00 | Y 00 00 | Z .00 | Accept Cancel |

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

| Edit - Fabric | Arch, Ridge an | d Valley Cable | | × |
|---------------|----------------|----------------|------------|----------------|
| ArchID | 2 🕂 Type | • External | Internal C | Ridge & Valley |
| Start Point | .00 | .00 | .00 | Apply |
| Mid Point | 1250.00 | -500.00 | 500.00 | Cancel |
| End Point | 2500.00 | .00 | .00 | |

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.

| Edit external borders | | × | |
|---|-------------------------|----------|--|
| Border segment "B" | 4 - Number of points | 8 | |
| Sag amount by % of | 1.0 📑 or by distance of | 250.0 mm | |
| Restrained segment | C Yes 💿 No | | |
| Curvature kind 💿 Negative 😳 Positive(Out) | | | |
| Update | Update A11 Done | | |

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

| Xy Net Form Finding Parameters | × |
|--|--------|
| Warp-Weft stress ratio 1.0 📩 Prestress (warp), kN/m | 1.00 🔺 |
| Fabric mesh width, mm | • 00. |
| Minimum internal net point from border 25.00 | |
| Align membrane center to nearest system point O Yes 💿 No | .ccept |
| Automatic form finding • Yes • No | ancel |
| Mesh origin at X 2750.00 Y 2750.00 Stop at check point | |



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).

| Edit Force Density Of Selected Elements | × |
|---|---|
| Selection by range | tions All Display Group Color Property |
| Excluding mode Off | Last Reset Cancel Accept |
| C1 | |
| Force Density On Selected | l Net Link(s) |
| Force density value (kN/m) |) 2.00 Accept |

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.

| Edit Nodal Coordinates | | | | | | | | |
|------------------------|------------|---------|--------|--|--|--|--|--|
| Node ID 4 Nod | le Group 🔽 | | Apply | | | | | |
| Х 2500.000 Ү | 4000.000 z | 250.000 | Cancel | | | | | |

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

| $\left[\right]$ | | N38 | N | 37 | N36 | N35 | N34 | N33 | N32 | N31 | N30 |
|------------------|-------------|-----|---------|------------------|-------------------|---------|---------|------------|--------|----------|-----|
| | | | 📑 Entit | y Selection-Ente | r node(s) to Sele | ect | | | | × | |
| \downarrow | | | Selecti | to 1 S | Step 1 | Select | All Dis | play Group | Color | Property | / |
| | | | | \ | | | L | nst Reset | Cancel | Accept | |
|) | | | 130-38 |) | | | | | | | |
| | $\langle $ | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | _ | | | | | | | |
| | | | | | Select two r | node ID | | × | | | |
| | | | | 82 | elect node ID | 1 5 | and 2 4 | Apply | | | |

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

| Entity Selection-Enter node(s) to Select | | × |
|--|---------------------|----------|
| Selection by range Options | | |
| 1 to 1 Step 1 Select All | Display Group Color | Property |
| | Last Reset Cancel | Accept |
| | | |
| 30-38 | | |



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.

| 💳 Entity Selection-Enter n | ode(s) to Select | | | | | × |
|----------------------------|--------------------|-------------|---------|-------|--------|----------|
| Selection by range | | Options ——— | | | | |
| 1 to 1 Step | p 1 Select | A11 | Display | Group | Color | Property |
| | | | Last | Reset | Cancel | Accept |
| 30-38 | | | | | | |
| | | | | | | |
| | | | | | | |
| | Set Selected Nodes | To Same Y | | × | | |
| | Enter new Y value | 40 | 100 A | .pply | | |

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.

| Node ID | Х | 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 | Node (more) | Cartesian** command to create this system points.

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.

| WinFabric System Variables | | | × |
|---|--------|--------------------------|--------|
| Form Finding | | Precision and Tolerance | |
| Fabric net type 🛛 C Regular 💽 I | Radial | Warp-weft angle | 5.0 |
| Number of iteration | 2 | Minimum triangular angle | 1.5 |
| Number of points for border segments | 8 | Arch constructor node | 5.0 |
| Sag amount in % for border segment | 10.0 | Minimum cable length | 100.00 |
| Force density for fixed border segments | .00 | Minimum net length | 50.00 |
| h-Contour Interval | 50.00 | Aco | cept |



Save the model as Rosetta.

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

| Parameters | |
|---|-------------------|
| Element size in radial direction | |
| Meridian angle | |
| Radial/Hoop pre-stress ratio | |
| Pre-stress in radial direction, kN/m 1.0 | |
| Start angle | |
| Division by 🖲 Border node 🛛 C Meridian angle | |
| Constant radial force density 💽 Yes 🔘 No | |
| Automatic form finding? •• Yes •• No | |
| Cancel Apply | |
| Radial is the warp and hoop is the weft direction | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | $\langle \rangle$ |
| | > |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |

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.



77

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.



| Windas Eight (Build 08.2016) Eile Generate Design Draw Erlit Faci | A ility Dicelary Delete List Mor | A Spaceframe & Tensile Membrane Design System (By View Solution WinFabric WinSolm WinCl | Copyright © 2015 Multimedia Engineeri | ng Pte Ltd | | | - 0 × |
|--|-------------------------------------|--|---------------------------------------|-------------|-------------|--------------|----------|
| | | | | | | | |
| Model - [E:\01 FOR TUTORIAL\test\test1.wds | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| NB | N9 | N10 | N11 | | N12 | N13 | NI |
| | | | | | | | |
| | | | | | | | |
| | | N14 | | | N3 | | |
| | N7 | | | | | N2 | |
| | | | | | | | |
| | | N6 | | | N4 | | |
| | | | | | | | |
| | | | | | | | |
| | | | 203 | | | | |
| | | | | | | | |
| 4 | | | | | | | |
| → × | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| Model saved | ice done | | | | | | - |
| Merge node within tolerance distant | ice done | | | | | | _ |
| Model saved | | | | | | | - |
| Elements= 0 | Nodes= 14 | Bar Net | Color= 1 | Property= 1 | Loadcase= 1 | Group=E16.N1 | <u> </u> |

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.

| E Wind | as Eight (Build 08.2016) merate Design Draw Edit Facili | A Space tv Display Delete List Modify V | frame & Tensile Membrane Design System iew Solution WinFabric WinSodm WinClad | Copyright © 2015 Multimedia Engineering Pte Ltd | | | - a × (|
|--|---|--|--|---|-----|-----|----------|
| i Ma | Element Element (more) | | - 6 4 6 4 1 2 16 4 4 4 9 | | 6 | | |
| | Membrane Constructors AMembrane Forms | > > | | | | | |
| | Spaceframe Forms 2 Linear Truss Delta Beam | | | | | | |
| | ETFE Forms | | | | | | |
| | Mesh 3 | 3-Node Triangle | | | | | |
| | Region Volume | 4-Node Quad 6-Node ISO-P Triangle | | | | | |
| | Node Node (more) | 8-Node ISO-P Quad 9-Node Bubble Function | N10 | NII | N12 | N13 | N1 |
| | Material Property Standard Material Property Tensile Membrane Property | Mesh Definition 3 | > Open Run | | | | |
| | Extra-Stiffness Load 3 Support 3 | 7 | N14 | | NJ | N2 | |
| | | | N6 | | Ni | | |
| | | | | Nő | | | |
| | 4 x | | | | | | |
| User IE Mergin Merge Model Mergin Merge | Multimedia_engineering ig nodes within maximum dist node within tolerance distanc saved ig nodes within maximum dist node within tolerance distance | ance progress te done ance progress te done | | | | | - |
| < Open m | esh definition file | | | | | | <u> </u> |

| M Mes | h Definition Tabl | e | | | | | | | | | | | × |
|-------|-------------------|----|----|----|----|----|----|----|----|----|----|---|------|
| | Region | N1 | N2 | N3 | N4 | N5 | N6 | N7 | N8 | NX | NY | ^ | |
| 1 | 1 | 8 | 7 | 6 | 14 | 10 | 9 | 0 | 0 | 10 | 10 | 1 | |
| 2 | 2 | 6 | 5 | 4 | 3 | 12 | 11 | 10 | 14 | 20 | 10 | 1 | Sort |
| 3 | 3 | 4 | 2 | 1 | 13 | 12 | 3 | 0 | 0 | 10 | 10 | 1 | 0.11 |
| 4 | | | | | | | | | | | | | Quit |
| 5 | | | | | | | | | | | | | Evit |
| 6 | | | | | | | | | | | | | EXIL |
| 7 | | | | | | | | | | | | | |
| 8 | | | | | | | | | | | | | |
| 9 | | | | | | | | | | | | | |
| 10 | | | | | | | | | | | | | |
| 11 | | | | | | | | | | | | | |
| 12 | | | | | | | | | | | | | |
| 13 | | | | | | | | | | | | | |
| 14 | | | | | | | | | | | | | |
| 15 | | | | | | | | | | | | | |
| 16 | | | | | | | | | | | | | |
| 17 | | | | | | | | | | | | | |
| 18 | | | | | | | | | | | | | |
| 19 | | | | | | | | | | | | | |
| 20 | | | | | | | | | | | | | |
| 21 | | | | | | | | | | | | - | |
| 22 | | | | | | | | | | | | - | |
| 23 | | | | | | | | | | | | - | |
| 24 | | | | | | | | | | | | - | |
| 25 | | | | 1 | | | | | | | | V | |

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.

| Generate Fabric Net Property For Membrane X | | | | | | | |
|---|---|------------|---------------------------|--------|--|--|--|
| Tensile Fabric Material | Tensile Fabric Material FiberTop PTFE T400 | | | | | | |
| Warp (Radial) EA (kN/m) | 1460.00 Weft (Ring) EA (kN/m) 980.00 Cancel | | | | | | |
| Border Cable, Material | Stainless | • D | iameter (mm) 12mm 🗨 | | | | |
| Tieback Cable, Material | Galvanised | – D | iameter (mm) 13mm 🗨 | | | | |
| | | | | | | | |
| | | | | 1 | | | |
| WinFabric System Variables | | | | × | | | |
| Form Finding | | | Precision and Tolerance - | | | | |
| Fabric net type 📀 Re | gular 🛛 🔿 Ra | adial | Warp-weft angle | 5.0 | | | |
| Number of iteration | Γ | 2 | Minimum triangular angle | 1.5 | | | |
| Number of points for border | segments | 8 | Arch constructor node | 5.0 | | | |
| Sag amount in % for border s | segment | 7.5 | Minimum cable length | 100.00 | | | |
| Force density for fixed borde | r segments | .00 | Minimum net length | 50.00 | | | |
| h-Contour Interval | | 50.00 | [| Accept | | | |
| Minimum rainwater runoff | Γ | 7.50 | L | | | | |
| Scaling factor for symbol dis | play | 1.00 | | | | | |

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;

| WinFabric System Variables | × |
|---|------------------------------|
| -Form Finding | Precision and Tolerance |
| Fabric net type 📀 Regular 🔿 Radial | Warp-weft angle 45.0 |
| Number of iteration 2 | Minimum triangular angle 1.5 |
| Number of points for border segments 8 | Arch constructor node 5.0 |
| Sag amount in % for border segment 7.5 | Minimum cable length 100.00 |
| Force density for fixed border segments .00 | Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |
| Minimum rainwater runoff 7.50 | |
| Scaling factor for symbol display 1.00 | |

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 perin 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.

| Xy Net Form Finding Parar | neters | | × |
|---------------------------|-----------------|----------------------|-------------|
| Warp-Weft stress ratio | 1.0 🕂 | Prestress (warp), kN | /m 1.00 • |
| Fabric mesh width, mm | <u>68 .84</u> ÷ | Warp ingle to globa | 1 .00 - |
| Minimum internal net poir | t from border | 25.00 | |
| Align membrane center to | nearest syste | mpoint CYes 🖲 No | Accept |
| Automatic form finding | (| Yes C No | Cancel |
| Mesh origin at X | .00 Y | .00 Stop at chec | k point 0 • |

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;

Bottom Layer

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 \rightarrow High Point \rightarrow 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 .



87

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.**

| Windas Eight (Build 08.2016) File Generate Design Draw Edit Facility | A Display Delete List Modify | Spaceframe & Tensile Membrane Design View Solution WinFabric WinSsdm | System WinClad | Copyright © 2015 Multimedia | Engineering Pte Ltd | - | o x |
|---|---|---|-------------------|-----------------------------|---------------------|-------------|----------|
| FIQQQE + J | | = 🖉 🛦 🖻 🖳 📕 🏛 👪 🎄 | | | 44 D A A 80 | | |
| Model - [D:\ETC\Tutorial\WINDAS\cushion\2tes | t.wds] | | | | | | |
| | | | | | | | |
| | | | | | | | |
| N7 | | | | | | | |
| (250) | | | | | | | |
| | | | N6 | | | | |
| | | | | | | NS | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| 2.0 | | | | | | | |
| | | | | | | N4 | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| 4 | | | | | | | |
| - Dy | NI | | | N2 | | | N3 |
| | | | | | | | |
| | | | | | | | |
| 50 Bukit Batok St 23 Midview Bldg #0 Tel:(65) 6765-6288 Fax:(65) 6765-15 | 15-15 Singapore 659578 88 Web: www.me.com.sg.F | mail mepl@me.com.sq | | | | | _ |
| User ID: Multimedia_engineering | | | | | | | |
| Merging nodes within maximum distar Merge node within tolerance distance | done | | | | | | - |
| | | | | | | | <u> </u> |
| Elements= 0 | Nodes= 8 | Bar Net | Color= 1 | Property= 1 | Loadcase= 1 | Group=E8,N1 | |

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.

| 🛃 Windas File Gen | Eight (Build 08.2016) erate Design Draw Edit Facili | A Spa ty Display Delete List Modify Vi | eframe & Tensile Membrane Design System Copyrigi Solution WinFabric WinSsdm WinClad | ht © 2015 Multimedia Engineering Pte Ltd | - 0 | × |
|----------------------|--|---|---|--|-----|----------|
| 26 | Element | | 0 a C 9 11 1 1 1 4 4 4 4 4 4 1 1 1 1 1 1 1 1 | | | |
| Mo | Element (more) | wds] | | | | |
| | Membrane Constructors Membrane Forms | | | | | |
| | Spaceframe Forms Linear Truss | • | | | | |
| | Delta Beam | | | | | |
| | ETFE Forms | | N6 | | | |
| | Mesh | 3-Node Triangle | | | | |
| | Region | 4-Node Quad | | | N5 | |
| | Volume | 6-Node ISO-P Triangle | | | | |
| | Node | 8-Node ISO-P Quad | | | | |
| | Node {more} | 9-Node Bubble Function | | | | |
| | Material Property | Mesh Definition > | Open | | | |
| | Standard Material Property | | Kun | | | |
| | Tensile Membrane Property | | | | | |
| | Extra-Stiffness | | | | | |
| | Load | | | | N4 | |
| | Support | | | | | |
| | | | | | | |
| | <u>a</u> | N1 | | 272 | | N2 |
| | └──⊳ _X | 281 | | 172 | | |
| | | | | | | |
| | | | | | | |
| Copyrig | tight A Spacename & ren ht © 2016 Multimedia Engi | sile wembrane system | | | | |
| 50 Buki | t Batok St 23 Midview Bldg | #05-15 Singapore 659578 | | | | _ |
| Tel:(65) | 6765-6288 Fax:(65) 6765-1 | 588 Web: www.me.com.sg Em | l:mepl@me.com.sg | | | |
| User ID | Multimedia_engineering | | | | | • |
| | ch definition file | | | | | <u> </u> |

Then a new window will appear as shown below.

| M Mes | h Definition Table | | | | | | | | | | | | × |
|-------|--------------------|----|----|----|----|----|----|----|----|----|----|---|------|
| | Region | N1 | N2 | N3 | N4 | N5 | N6 | N7 | N8 | NX | NY | ^ | |
| 1 | | | | | | | | | | | | | |
| 2 | | | | | | | | | | | | | Sort |
| 3 | | | | | | | | | | | | | Quit |
| 4 | | | | | | | | | | | | | Quit |
| 5 | | | | | | | | | | | | | Evit |
| 6 | | | | | | | | | | | | | LXII |
| 7 | | | | | | | | | | | | | |
| 8 | | | | | | | | | | | | | |
| 9 | | | | | | | | | | | | | |
| 10 | | | | | | | | | | | | | |
| 11 | | | | | | | | | | | | | |
| 12 | | | | | | | | | | | | | |
| 13 | | | | | | | | | | | | | |
| 14 | | | | | | | | | | | | | |
| 15 | | | | | | | | | | | | | |
| 16 | | | | | | | | | | | | | |
| 17 | | | | | | | | | | | | | |
| 18 | | | | | | | | | | | | | |
| 19 | | | | | | | | | | | | | |
| 20 | | | | | | | | | | | | | |
| 21 | | | | | | | | | | | | | |
| 22 | | | | | | | | | | | | | |
| 23 | | | | | | | | | | | | | |
| 24 | | | | | | | | | | | | | |
| 25 | | | | | | | | | | | | ~ | |

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.

| M Mest | Definition Table | | | | | | | | | | | | × |
|--------|------------------|----|----|----|----|----|----|----|----|----|----|---|------|
| | Region | N1 | N2 | N3 | N4 | N5 | N6 | N7 | N8 | NX | NY | ^ | |
| 1 | þ | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 10 | 6 | | |
| 2 | | | | | | | | | | | | | Sort |
| 3 | | | | | | | | | | | | | 0.11 |
| 4 | | | | | | | | | | | | | QUIT |
| 5 | | | | | | | | | | | | | Evit |
| 6 | | | | | | | | | | | | | EXIL |
| 7 | | | | | | | | | | | | | |
| 8 | | | | | | | | | | | | | |
| 9 | | | | | | | | | | | | | |
| 10 | | | | | | | | | | | | 1 | |

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





 Moder satved...
 R1
 X9
 N10
 N11
 N12
 A2
 N13
 N14
 N15
 N16
 N3

 moder satved...
 Checking and fixing data in progress...
 Checking and fixing data in progress...
 Checking and fixing model data
 Image: Checking and fixing model data
 Image: Checking and fixing model data

 Image: Name
 Color= 1
 Property= 1
 [Loadcase= 1
 [Group=E8,N]

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



92

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**

| | Select Color | | | × |
|---|----------------|-------|-------|----------|
| Entity Selection-Enter element(s) to Select | | | | × |
| Selection by range | s 1 Display | Group | Color | Property |

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

| Inflated Mesh Volume Form Finding | × |
|--|--|
| Iteration Number of Iterations 99 ÷ Convergence Norm | Load increment Number of Increments Increment Size 1.00 Cancel |
| Compute Cushion Final Form to rech g | viven height and pressure |
| Top layer foil thickness 250 • | micron, inflated to 350.00 mm height |
| Bottom layer foil thickness 250 | micron, inflated to 350.00 mm height |
| Initial Prestress, kN/m 2.00 | Internal Pressure 300 Pa [N/mm2] |

| Flemente= 120 | Nodes= 122 | Membrane | Color= 2 | Property= 2 | Loadcase= 1 | Group=E0.N1 | • |
|--|----------------------------------|---|---|--------------------------|--------------------------|-------------|---|
| Maximum inflated height at Node Model saved | 43 is 624.055mm, intern | al pressure = .3000 kPa | | | | | |
| Maximum inflated height at Node Model saved Model saved | 43 is 785.197mm, intern | al pressure = .3000 kPa | | | | | - |
| x | | | | | | | |
| ¢r | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| | | | | | | | |
| Model - [D:\ETC\Tutorial\WINDAS\cushio | n\test_ff.wds] | | | | | | |
| <u> = </u> | | | 700 10 10 10 10 10 10 10 10 10 10 10 10 1 | | | | |
| Windas Eight (Build 08.2016) File Generate Design Draw Edit Fa | acility Display Delete List Modi | A Spaceframe & Tensile Membrane De fy View Solution WinFabric WinS | sign System isdm WinClad | Copyright © 2015 Multime | edia Engineering Pte Ltd | - 0 | × |
| - | | | | | | | |

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.

| tiement | | | |
|--|---|----------|-----------|
| Element (more) | d Tutorial\Windas\ETFE Tutorial.wds] | | |
| Membrane Constructors | > | | |
| Membrane Forms | > | N/ | 89 |
| Spaceframe Forms | > | | |
| Linear Truss | | | |
| Delta Beam | | | |
| ETFE Forms | > | | |
| Region | 3 -Node Inangre | | |
| Volume | > 6-Node ISO-P Trianole | | |
| | 8-Node ISO-P Quad | | |
| Node Node (more) | 9-Node Bubble Function | | |
| Material Descents | Mesh Definition > Open | | |
| Standard Material Property | Run | | |
| Tensile Membrane Property | | | |
| Extra-Stiffness | | | |
| Load | > | | |
| Support | | | |
| | · · | 81 | Né |
| å ⊳x | , , | NI | 26 |
| A Total State Stat | 11 Engineering Pie Ltd Engineering Pie Ltd Engineering Pie Ltd Engineering Pie Ltd Engineering Pie Ltd Engineering Pie Ltd Engineering Pie Ltd | NI NI | X6 |
| A Satok SI 23 Michael Satok SI 23 Michael Multimedia_engineer Multimedia_engineer | T Figureeing Pie Ltd Biog into: 15 Singapore 059578 Biog into: www.mc.com.g Email.mept@me.com.sg ing ing ing distance progress. | 83 | 26 |

| M Mes | h Definition Tabl | e | | | | | | | | | | | × |
|-------|-------------------|----|----|----|----|----|----|----|----|----|----|----------|------|
| | Region | N1 | N2 | N3 | N4 | N5 | N6 | N7 | N8 | NX | NY | ^ | |
| 1 | 1 | 1 | 2 | 3 | 4 | 0 | 0 | 0 | 0 | 10 | 5 | Ť. | |
| 2 | 2 | 2 | 5 | 6 | 3 | 0 | 0 | 0 | 0 | 10 | 5 | 1 | Sort |
| 3 | 3 | 4 | 3 | 7 | 8 | 0 | 0 | 0 | 0 | 10 | 5 | 1 | |
| 4 | 4 | 3 | 6 | 9 | 7 | 0 | 0 | 0 | 0 | 10 | 5 |] . | Quit |
| 5 | | | | | | | | | | | • | | Evit |
| 6 | | | | | | | | | | | | | EXIL |
| 7 | | | | | | | | | | | | | |
| 8 | | | | | | | | | | | | | |
| 9 | | | | | | | | | | | | | |
| 10 | | | | | | | | | | | | 1 | |
| 11 | | | | | | | | | | | | 4 | |
| 12 | | | | | | | | | | | | 4 | |
| 13 | | | | | | | | | | | | | |
| 14 | | | | | | | | | | | | | |
| 15 | | | | | | | | | | | | | |
| 16 | | | | | | | | | | | | 4 | |
| 17 | | | | | | | | | | | | 4 | |
| 18 | | | | | | | | | | | | 4 | |
| 19 | | | | | | | | | | | | - | |
| 20 | | | | | | | | | | | | - | |
| 21 | | | | | | | | | | | | - | |
| 22 | | | | | | | | | | | | - | |
| 23 | | | | | | | | | | | | 1 | |
| 24 | | | | | | | | | | | | 1 | |
| 20 | | | | | | | | | | | | _ | |

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**

| | Sel | ect Color | | | × |
|---|---------|-----------------------|-------|--------|----------|
| | | $\mathbf{\mathbf{N}}$ | | | |
| Entity Selection-Enter element(s) to Select | Ortions | | | | × |
| Selection by range 1 to 1 Step 1 Select | All | Display | Group | Color | Property |
| Excluding mode Off | | Last | Reset | Cancel | Accept |
| | | | | | |

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.





104

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).

| as 10 {Build 20170701} - Light | tweight Structure Analysis Program | | | | | | | - 0 |
|--|------------------------------------|-----------------------------------|----------|---------------------|-----------|-----|--------------|-------|
| nerate Design Edit Faci S = 0 0 0 0 | | | | | | | | |
| [D:\Windas 10 - Mesh for An | nalysis - White\R2A.wds] | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| | | | | | | | | |
| (1 3) | | | | | | | | |
| | | Entity Selection-Enter node(s) to | Select | | × | | | |
| N20 | | - Selection by range | Options | | 1 | | | |
| | 2721 | 1 to 1 Step 1 | All | Display Group Color | Property | | | |
| N14 | N21 | | | Last Reset Cance | Accept | | | |
| | N22 | 19,32 | | | | | | |
| | | | | | | | | |
| | N | 16 | | | | | | |
| | | N17 | | | | | | |
| N13 | | | | | | | | |
| | | N18 | | | | | | |
| | | | N23 | | | | | |
| | | | N24 | | | | | |
| N12 | | | N | 25 | | | | |
| | | | | N26 | | | | |
| | | | | N2 | 7 | | | |
| | | | | | N28 | N29 | | |
| 4 | | | | | | | N30 | |
| ▶ N11 | N10 N9 | N8 N7 | N6 | N5 | N4 | N | 13 N2 | N32 N |
| | | | | | | | | |
| | | | | | | | | |
| 6765-6288 Fax:(65) | 6765-1588 Web: www.me.co | m.sg Email:mepl@me.com.sg | | | | | | |
| Multimedia_enginee | ering | | | | | | | |
| node within tolerance | distance done | | | | | | | |
| | | | | | | | | |
| | Nedeo=0 | Rev Net | Color=13 | Property=13 | Loadcace= | 1 | Croup=E22 N1 | |

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**.

| 📧 External Border -Select nodes in anti-clockwise order 🛛 🗙 | | | | | |
|---|---------|---------|-------|--------|----------|
| Selection by range | Options | | | | |
| 1 to 1 Step 1 Select | A11 | Display | Group | Color | Property |
| | | Last | Reset | Cancel | Accept |
| 9,13-25,1,29,27,26,28,30,2-8,10,12,11 | | | | | |

| WinFabric System Variables | × |
|--|--------------------------------|
| Form Finding | Precision and Tolerance |
| Fabric net type 💿 Regular 🔿 Radial | Warp-weft angle 5.0 |
| Number of iteration | 2 Minimum triangular angle 1.5 |
| Number of points for border segments | Arch constructor node 5.0 |
| Sag amount in % for border segment | Minimum cable length 100.00 |
| Force density for fixed border segments .0 | 0 Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |
| Minimum rainwater runoff 7.50 |) |
| Scaling factor for symbol display 1.0 | |



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





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++.



| <pre>Part = Part = Part</pre> | | 🐚 🚔 강 배h 🗈 구 C | 📾 🏣 🔍 🔍 🕎 | 3 🗟 🛼 11 📰 | 🗾 🚺 📓 | • 💌 | | 1 | | | | | | | | | | | | | | | | |
|---|---------|---|---------------|---------------|--------|-----|-----|-----|-------|-------|-------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|---|
| <pre>late100.0; / point(1) = (-1402.740, 465.637, 973.402,1c); Point(2) = (-1402.740, 465.637, 973.402,1c); Point(3) = (-1402.640, 1267.847, 980.368,1c); Point(5) = (-1402.640, 1267.847, 980.368,1c); Point(5) = (-1200.000, .000, 1037.20,1c); Point(5) = (-1200.000, .000, 1037.20,1c); Point(5) = (-200.77, .000, 1065.680,1c); Point(5) = (-200.77, .000, 1065.680,1c); Point(1) = (-1093.30, .000, 1001.70,1c); Point(1) = (-1094.38, .000, 1001.70,1c); Point(1) = (-1094.38, .000, 1001.170,1c); Point(1) = (-1094.38, .000, 1001.170,1c); Point(1) = (-1094.38, .000, 1004.170,1c); Point(1) = (-1094.38, .000, 1006.1c); Point(1) = (-1094.38, .000, 1066.60,1c); Point(2) = (-6874.70, 1086.660,1c); Point(2) = (-6874.70, 1086.560,1c); Point(2) = (-6874.70, 1086.560,1c); Point(2) = (-6874.70, 1086.560,1c); Point(2) = (-1095.560, 2384.577, 1085.660,1c); Point(2) = (-1095.560, 2384.577, 1085.660,1c); Point(2) = (-1107.360, 500.1660, 680.1c); Point(2) = (-1205.100, 2384.270, 1085.170,1c); Point(2) = (-1205.100, 2384.270, 1085.170,1c); Point(2) = (-1217.360, 500.1660, 680.1c); Point(2) = (-1211.360, 500, 640.1675, 1033.360,1</pre> | 🖂 🖂 R2 | A.geo 🗵 | | | | | | | | | | | | | | | | | | | | | | |
| <pre>bint () = (-1480.760, 468.637, 973.402.10); bint () = (-1480.760, 1364.368, 981.839.10); bint () = (-1435.350, 0.000, 1374.360,10); bint () = (-1435.350, 0.000, 1097.380,10); bint () = (-1305.660, .000, 1098.660,10); bint () = (-1006.660, .000, 1098.660,10); bint () = (-1006.660, .000, 1098.660,10); bint () = (-1098.050, .000, 1098.660,10); bint () = (-1335.350, .000, 1098.660,10); bint () = (-1335.550, .000, 1098.660,10); bint () = (-336.611, 1286.380, 1086.660,10); bint () = (-338.611, 1286.380, 1086.660,10); bint (2) = (-338.611, 1286.380, 1086.660,10); bint (2) = (-358.675, 1621.391, 1086.560,10); bint (2) = (-758.66, 75, 1621.391, 1086.560,10); bint (2) = (-758.381, 238.538, 1086.660,10); bint (2) = (-358.675, 1621.391, 1086.560,10); bint (2) = (-758.381, 238.538, 1086.660,10); bint (2) = (-758.381, 238.538, 1088.660,10); bint (2) = (-1318.530, 0.974.1032, 1080,108); bint (2) = (-1318.530, 0.984.663, 0.10); bint (2) = (-1318.530, 0.688.664, 0.1038.664, 0.10); bint (2) = (-1318.530, 0.688.664, 0.10); bint (2)</pre> | 1c=1000 | 0.0 ; | | | | | | | | | | | | | | | | | | _ | | | _ | _ |
| Peart () = (-1431.650.570, 3124.365, 99.3.835.10; point () = (-1431.650.570, 3124.365, 99.335.10; point () = (-1431.650, 00, 1006, 1907.335.10; point () = (-1200.000, 1006, 1007.000, 1007.00; point () = (-1200.000, 1006, 1007.000, 1007.00; point () = (-1000.000, 1006, 1007.00; 1007.00; point () = (-1000.000, 1007.00; 1007.00; 1007.00; point () = (-1000.000, 1007.00; 1 | oint(| 1) = { -14818.740, | 4659.637, | 9733.402,1c) | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>bdnt () = (-1433.6.65, 167.584, 980.88,1c); bdnt () = (-1453.60,006, 1013.710.1c); bdnt () = (-1453.60,006, 1013.710.1c); bdnt () = (-1331.30.60,006, 1025.630.1c); bdnt () = (-2000.877,000, 1025.630.1c); bdnt () = (-2000.877, 1025.630.1c); bdnt (20) = (-2000.870, 4022.505, 1033.640.1c); bdnt (20) = (-2000.877, 1025.630, 1051.700.1c); bdnt (20) = (-1000.530.730, 1051.700.1c); bdnt (20) = (-1000.530.730, 1051.700.1c); bdnt (20) = (-1000.530.730, 1051.700.1c); bdnt (20) = (-1200.730, 4250.560, 1033.640.1c); bdnt (20) = (-1200.73</pre> | oint(| 2) = { -14505.570, | 3124.365, | 9813.839,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>bint { y = { -14253.500,000, 9976.333.1c); bint { y = { -1313.650,000, 10137.1c0,1c); bint { y = { -1313.650,000, 10037.1c0,1c); bint { y = { -1300.100,000, 10078.1c0,1c); bint { y = { -2000.100,000, 10078.1c0,1c); bint { y = { -2000.100,000, 10078.1c0,1c); bint { 1 y = { -1393.310,000, 10078.1c0,1c); bint { 1 y = { -1393.311, 128.188, 10048.1c0,1c); bint { 2 y = { -3074.311, 128.188, 10048.1c0,1c); bint { 2 y = { -3075.331, 328.530, 10650.460,1c); bint { 2 y = { -3075.331, 328.530, 1050.160,1c); bint { 2 y = { -13050.760, 4532.1c0, 1051.760,1c); bint { 2 y = { -13050.760, 4532.1c0, 1051.770,1c); bint { 2 y = { -13050.760, 4532.1c0, 1051.770,1c); b</pre> | oint(| 3) = { -14316.650, | 1567.584, | 9860.868,1c) | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>chart 0; = { -1330.650;00; 1030.710,1c); </pre> | oint(| $4) = \{ -14253.500, $ | .000, | 9876.338,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>Suft = 0</pre> | oint(| 5) = (-13130.650, | .000, | 10130.710,1c) | 7 | | | | | | | | | | | | | | | | | | | |
| <pre>datt 0) = { -10166.680,000, 10625.680,10 ; outt 0) = { -200.077,000, 10789.060,10 ; outt 0) = { -200.000,000, 10880.660,10 ; outt 0) = { -10333,500,000, 10625.680,10 ; outt 0) = { -10333,500,000, 10625.680,10 ; outt 0) = { -10533,500,000, 10625.680,10 ; outt 0) = { -10533,500,000, 10621.680,10 ; outt 0) = { -10536,975, 100,483, 10066.660,10 ; outt 0) = { -10374,115, 199.1.206, 10744.780,10 ; outt 0) = { -10336,975, 1601,335, 10660.680,10 ; outt 0) = { -10536,975, 1601,335, 10660.680,10 ; outt 0) = { -10533,500, 4625, 560, 1053, 500,10 ; outt 0) = { -10533,500, 4855, 500, 4037, 10 ; outt 0) = { -11315,300, 4890,655, 10133,500,10 ; outt 0) = { -11375,300, 5801,580, 9893,500,10 ; outt 0) = { -11375,300, 5801,580, 9893,500,10 ; outt 0) = { -11375,300, 5801,580, 9893,500,10 ; outt 0) = { -11575,300, 5801,580, 9893,500,10 ; outt 0) = { -11575,300, 5801,580, 9893,500,10 ; outt 0) = { -1253,500, 580, 586, 586, 5801,500 ; ranglinute Burface ***; Hen Surface (1) = { (1) ; ranglinute Burface ***; </pre> | oint (| 6) = (-12000.000, | .000, | 10347.820,1c) | 7 | | | | | | | | | | | | | | | | | | | |
| <pre>chart (b) = { -28200.577, . 000, 1098.600,10 ; shart (1) = { -2830.507, . 000, 1098.600,10 ; shart (1) = { -283.500, . 000, 10021.170,10 ; shart (1) = { -283.500, . 000, 10021.170,10 ; shart (1) = { -283.500, . 000, 9616.573,10 ; shart (1) = { 1283.500, . 000, 9616.573,10 ; shart (1) = { 1283.500, . 000, 9616.573,10 ; shart (1) = { 1283.500, . 000, 9616.573,10 ; shart (1) = { 1283.500, . 000, 9616.573,10 ; shart (1) = { 1283.57, 1125, 1026.670,10 ; shart (1) = { -386.575, 1121.381, 1088.460,10 ; shart (2) = { -386.575, 1121.381, 1088.460,10 ; shart (2) = { -4286.675, 1121.381, 1088.460,10 ; shart (2) = { -4286.675, 1121.381, 1088.460,10 ; shart (2) = { -766.100, 2316.266, 10785.70,10 ; shart (2) = { -7676.100, 2316.266, 10785.70,10 ; shart (2) = { -7676.100, 2316.266, 10785.70,10 ; shart (2) = { -10055.660, 3344.520, 1033.100,10 ; shart (2) = { -10055.600, 3744.200, 10312.760,10 ; shart (2) = { -1315.300, 4890.625, 1033.800,10 ; shart (2) = { -1315.300, 4890.655, 9818.573,10 ; shart (2) = { -14077.300, 580.1666, 9818.673,10 ; shart (2) = { -14077.300, 580.1666, 9818.6173,10 ; shart (2) = { -14077.300, 580.1665, 9818.673,10 ; shart (</pre> | | <pre># = { -10106.690,</pre> | .000, | 10625.630,1c) | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>chart { 9 = { -6000.000, . 000, 1098.660,10 ; chart { 10 = { -3795.023, 0.00, 1098.660,10 ; chart { 11 = { -1893.310, 0.00, 1097.203,10 ; chart { 13 = { 1853.505, 0.00, 1097.203,10 ; chart { 13 = { 1853.505, 0.00, 1097.203,10 ; chart { 13 = { 1853.505, 0.00, 1097.203,10 ; chart { 15 = { 1964.505, 725.425, 10626.970,10 ; chart { 20 = { -3078.51, 1621.303,10 ; chart { 20 = { -3078.51, 1621.303,10 ; chart { 21 = { -5556.720, 1984.505, 10624.670,10 ; chart { 23 = { -7656.100, 2984.505, 10634.670,10 ; chart { 23 = { -7656.100, 2984.505, 1083.705,10 ; chart { 23 = { -7656.100, 2984.505, 10933.705,10 ; chart { 23 = { -9675.31, 3285.503, 10631.780,10 ; chart { 23 = { -9675.31, 3285.503, 10631.780,10 ; chart { 25 = { -11075.300, 5501.565, 993.504,10 ; chart { 25 = { -11075.300, 5501.565, 993.504,501 ; chart { 25 = { -11075.300, 5501.565, 995.504.571,201 ; chart { 25 = { -1105.500, 660.505.906.571,201 ; chart { 25 = { -1105.500, 660.505.906.571,201 ; chart { 25</pre> | oint(| 8) = { -8200.977, | .000, | 10799.020,1c) | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>clant (10 = { -3759.03, . 0.00, 10799.020,10; ; clant (11 = { -1893.03, . 0.00, 10247.820,10; ; clant (12 = { . 0.00, . 0.00, 10247.820,10; ; clant (13 = { 1635.050, . 0.00, 10247.820,10; ; clant (13 = { 1635.050, . 0.00, 10247.820,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (13 = { 1635.050, . 0.00, 10247.80,10; ; clant (23 = { 1635.050, . 0.00, 480,0.60,10; ; clant (23 = { 1635.050, . 0.024.800,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 1064.600,10; ; clant (23 = { 1635.050, . 0.286, 50, 1065.600,10; ; clant (23 = { 1635.050, . 0.286, 50, 1065.600,10; ; clant (23 = { 1635.050, . 0.286, 50, 1065.25, 10133.500,10; ; clant (23 = { 1635.050, . 0.486.650, 1569, . 9893.504,10; ; clant (23 = { 1635.050, . 0.486.650, 1569, . 9893.504,10; ; clant (20 = { 1, 235.500, . 0.486.650, 1569, . 9893.504,10; ; clant (10 = { 1, 1; ; ransfinite Surface ***; </pre> | oint(| 9) = { -6000.000, | .000, | 10868.660,1c) | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>cant (1) = { -1893.310,</pre> | oint(| 10) = (-3799.023, | .000, | 10799.020,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>chart (12) = { .000, .000, 10347.850,10; chart (12) = { .000, .000, 10347.850,10; chart (14) = { .358.600, .000, 9616.573,10; chart (14) = { .358.600, .000, 9616.573,10; chart (15) = { .0074.138, 366.640,10; chart (15) = { .0074.138, 951.265, 10566.460,10; chart (15) = { .0074.138, 951.265, 10566.470,10; chart (12) = { .0074.34, 12838, 10620.460,10; chart (12) = { .0074.534, 12838, 10620.460,10; chart (12) = { .0074.534, 12838, 10620.460,10; chart (23) = { .0074.534, 12838, 10620.460,10; chart (23) = { .0075.560, 3764.470,10; chart (25) = { .0075.560, 3764.120, 1052.470,10; chart (25) = { .0075.560, 3764.120, 1033.640,10; chart (25) = { .0075.560, 3764.120, 1033.640,10; chart (25) = { .0075.500, 652, 1033.640,10; chart (27) = { .12050.790, 4322.509, 1033.640,10; chart (28) = { .0175.500, 652, 1033.640,10; chart (29) = { .01275.500, 652, 1033.640,10; chart (29) = { .01275.500, 652, 1058, 9983.594,10; chart 2007 () = { .012; lane Surface () = { .013; lane Surface () = { .013; lane Surface () = { .013; lane Surface () = { (1); lane Surface () = { (1); lane Surface () = { (1); lane Surface () = { .013; lane Surface () = { .013; lane</pre> | oint(| 11) = (-1893.310, | .000, | 10625.630,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>chart (13) = { 1383.609,000, 10021.179.10; ; chart (13) = { 1383.609,000, 9616.573.10; ; chart (15) = { 1383.609,000, 9616.573.10; ; chart (17) = { 147.138, 306.927, 1024.439, 1016.649, 10; ; chart (17) = { 1-101.859, 723.435, 1026.4970, 10; ; chart (19) = { 1-3074.119, 991.206, 10744.740, 10; ; chart (21) = { -5386.975, 1621.381, 10280.580, 10; ; chart (21) = { -5386.975, 1621.381, 10280.580, 10; ; chart (23) = { -7553.609, 302.570, 1021.660, 10; ; chart (23) = { -7553.609, 302.5590, 10651.790, 10; ; chart (23) = { -7553.609, 302.570, 1021.660, 10; ; chart (23) = { -7553.609, 302.570, 1021.660, 10; ; chart (23) = { -7553.609, 302.509, 10651.790, 10; ; chart (23) = { -7553.609, 302.509, 10651.790, 10; ; chart (23) = { -1573.33, 3255.580, 10651.790, 10; ; chart (23) = { -1573.33, 3255.580, 10651.790, 10; ; chart (23) = { -1573.33, 3255.580, 10651.790, 10; ; chart (23) = { -1553.530, 666.573, 102; ; chart (23) = { -1553.530, 666.558, 9635.504, 10; ; chart (23) = { -1553.530, 666.558, 9635.504, 10; ; chart (30) = { -15533.500, 6165.589, 983.504, 10; ; chart (30) = { -15533.500, 6165.589, 983.504, 10; ; chart (30) = { -15533.500, 6165.589, 983.504, 10; ; chart (30) = { -15533.500, 6165.589, 983.504, 10; ; chart (30) = { -1553.3500, 6165.589, 983.504, 10; ; chart (30) = { -1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (30) = { (-1553.3500, 6165.589, 983.504, 10; ; chart (-1</pre> | oint (| $12) = \{ .000, .0$ | .000, | 10347.820,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>chat [14] = { 3.55:300,000, 942.57,102; chat [15] = { 165:157, 164.33, 1026.640,L0; chat [17] = { 175:157, 164.33, 1026.640,L0; chat [17] = { 175:157, 164.33, 1026.640,L0; chat [17] = { 175:157, 164.33, 1026.657,102; chat [17] = { 175:157, 164.740,10; chat [17] = { 175:157, 164.740,10; chat [17] = { 175:158, 1064.670,10; chat [17] = { 175:158, 1064.650, 106.660,10; chat [17] = { 175:158, 1064.650, 106.660,10; chat [17] = { 175:158, 1064.650, 107:10; chat [17] = { 175:158, 106, 105:168, 9983.594,10; chat [17] = { 175:158, 106, 105:158, 107:100; chat [17] = { 175:158, 106, 105:158, 9983.594,10; chat [17] = { 175:158, 106, 105:158, 106, 105:158, 105, 105; chat [17] = { 175:158, 106, 105:158, 106, 105:158, 106, 105:158, 106, 105; chat [17] = { 175:158, 106, 105:158, 106, 105:158, 106, 107; 106, 107; 106, 107; 106, 107; 106, 107; 107; 107; 107; 107; 107; 107; 107;</pre> | oint(| $13) = \{ 1635.509, $ | .000, | 10021.170,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>chat 13 = { 19 = 1 19 = 1 19 = 1 19 = 1 19 = 1 19 = 1 19 = 1 19 = 1 109 = 19 100 = 001, 10 10</pre> | oint(| $14) = \{ 3253.500, $ | .000, | 9616.573,1c) | ; | | | | | | | | | | | | | | | | | | | |
| <pre>htt 10 = {</pre> | oint (| 15) = { 1693.157, | 149.493, | 10006.640,103 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>htt (1) = (</pre> | LIIC (| 10) - (4/6.134, | 306.932, | 10261.110,103 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>Sect 100 1 = 00 1</pre> | int (| 17) - (-/14.285, | 499.919, | 10464.230,10) | | | | | | | | | | | | | | | | | | | | |
| <pre>bint (2) = (33(4,215), 391.206, 10/41.40,10); bint (2) = (423.63,1, 1268,306, 1004,400,10); bint (2) = (3586,975, 1621.381, 10060.406,10;); bint (2) = (3656,606, 2384.557, 10020.665,10;); bint (2) = (9754.606, 2384.557, 10020.665, 10;); bint (2) = (10765,960, 3794.230, 10510.760,10;); bint (2) = (10765,960, 3794.230, 10510.760,10;); bint (2) = (113115.300, 4690.655, 10133.500,10;); bint (2) = (113115.300, 4690.655, 10133.500,10;); bint (3) = (11553.500, 6162.556, 9616.573,10;); bint (3) = (11, 7, 10, 19, 10, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 2 bint (3) = (-11, 7, 10, 10, 10, 10, 10, 10, 10, 10, 10, 10</pre> | | -1904.598, | /20.425, | 10626.970,103 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>sint: 11 = { - 5386.075, 1621.391, 10860.960.10;; sint: 23 = { -5386.075, 1621.391, 10860.960.10;; sint: 23 = { -7654.606, 2384.567, 10826.650.10;; sint: 25 = { -9675.331, 3285.980, 10650.680.10;; sint: 25 = { -9675.331, 3285.980, 10650.680.10;; sint: 27 = { -12065.650, 3784.200, 10338.640.10;; sint: 28 = { -1315.300, 4580.655, 10338.640.10;; sint: 29 = { -14177.360, 5501.565, 9593.504.10;; sint: 10 = { 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 15, 20, 21, 22, 23, 24, 25, 26, 27, 28, 2; sint: Loop(1) = { 1, ; sansfinite Surface (1) = { 1 ; sansfinite Surface ***; </pre> | int (| 20) = (-4228 621 | 1200 200 | 10924 090 101 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>Lam: (1) = { -0535.70, 1084.65, 1084.670,10;; Lam: (2) = { -0535.70, 1084.65, 1084.670,10;; Lam: (2) = { -0576.110, 2315.266, 10756.370,10;; Lam: (2) = { -0776.110, 2315.266, 10756.370,10;; Lam: (2) = { -010765.560, 3784.220, 10512.760,10;; Lam: (2) = { -010765,560, 3784.220, 10512.760,10;; Lam: (2) = { -01077.360, 5501.565, 9893.640,10;; Lam: (1) = { -01553.500, 6162.566, 9616.573,1c;; Ine (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, 2 Ine Loop (1) = { 1, 1; r Lam: Surface "**;</pre> | int (| 21) = (-5296 075 | 1621 201 | 10860 860 101 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>cont(23) = { -7654.606, 2394.557, 10835.650.1c); cont(23) = { -7654.606, 2394.557, 10835.650.7c); cont(25) = { -875.331, 3285.930, 10650.660.1c); cont(27) = { -12050.790, 4322.509, 10338.640.1c); cont(27) = { -1215.00, 4590.625, 10338.640.1c); cont(28) = { -1315.300, 4590.625, 10338.10,1c); cont(28) = { -14177.360, 5501.569, 983.504.1c); cont(28) = { -14215.300, 6162.586, 983.504.1c); cont(30) = { -1525.300, 6162.586, 986.573.1c); ine (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, 2 ine for [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, 2 ine for [1] = { 1; ; lane Surface (1) = { 1; ; ransfinite Surface ***;</pre> | int (| 22) = 1 -526.720 | 1984 586 | 10864 670 1c1 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>cint(29 = (9769.110, 2916.206, 10750.370.1c); cint(25 = (9765.331, 3235.580, 10650.600.1c); cint(27 = (10565.560, 3764.220, 105312.760.1c); cint(27 = (123115.300, 4890.625, 10133.400.1c); cint(29 = (14371.530, 501.565, 9933.564.1c); cint(29 = (14371.530, 501.565, 9933.564.1c); cint(30 = (15253.500, 6162.586, 9616.573.1c); cint(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, 2 ine Loop(1) = (1); lane Surface "*";</pre> | int (| 23) = / -7654 606 | 2384 557 | 10828 650 1c1 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>oint(25) = {9875.331, 3285.930, 10650.660.1c1; oint(25) = {10965.560, 3781.220, 10512.780.1c1; oint(27) = {12050.790, 4322.509, 10338.640.1c1; oint(29) = {1315.00, 4590.665, 1033.940.1c1; oint(29) = {1525.350, 6162.5369, 9683.594.1c1; oint(30) = {1525.350, 6162.5369, 9616.573.1c1; int [10] = { 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, 2 int coording = { 1, 2; int coording = { 1, 2; int coording = { 1; int coordi</pre> | int (| (24) = (-8769, 110) | 2816 286 | 10758 370 1c1 | 1 | | | | | | | | | | | | | | | | | | | |
| <pre>cont(25) = (-10965.560, 3704.220, 10512.760.1c); cont(27) = (-12050.760, 4322.509, 1033.640.1c); cont(23) = (-13115.300, 4890.625, 10133.940.1c); cont(23) = (-1417.360, 5501.569, 9933.504.1c); cont(30) = (-1.5233.500, 6162.586, 9616.573.1c); ine(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, 2 ine(1) = (1); ransfinite Surface "*";</pre> | int (| $(25) = \{ -9875, 331 \}$ | 3285,930. | 10650.680.1c3 | 4 | | | | | | | | | | | | | | | | | | | |
| <pre>cint = 27 = { -12056.790, 4322.509, 10338.640.10; : oint = 27 = { -1215.0, 790, 4322.509, 10338.640.10; : oint = 29 = { -1315.300, 4690.625, 10133.940.10; : oint = 29 = { -14177.360, 5501.569, 9933.594.10; : oint = 0 = { -15253.500, 6162.5369, 9616.573.10; : ine (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, 2 ine Surface (1) = { 1, ; ransfinite Surface ***; </pre> | int (| $26) = \{ -10965, 560, -10065, 560, -10000, -10000, -10000, -10000, -100000, -10000, -10000, -100000, -100000, -10000, -1000000, -10000000000$ | 3784.220. | 10512,780,103 | 4 | | | | | | | | | | | | | | | | | | | |
| <pre>cont(29) = { -13115.300, 4890.625, 10133.910.1c; cont(29) = { -1417.300, 501.565, 9935.504.1c; cont(30) = { -15253.500, 6162.586, 9616.573.1c; cont(30) = { -1.5253.500, 6162.586, 9616.573.1c; ine(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, 2 ine(1) = { 1, 2; ine Loop(1) = { 1}; ransfinite Surface "*";</pre> | int (| $(27) = \{ -12050, 790 \}$ | 4322.509. | 10338.640.1c3 | 4 | | | | | | | | | | | | | | | | | | | |
| int(29) = { -14177,360, 5501.569, 9933.504,10; int(30) = { -15253.50, 6162.569, 9616.573,10; ine (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, 2 ine Surface (1) = { 1 ; iane Surface (1) = { 1 ; ianefinite Surface ***; | int (| (28) = (-13115, 300) | 4890.625. | 10133.910.1c3 | | | | | | | | | | | | | | | | | | | | |
| oint(30) = { -1.5253.500, 6162.586, 9616.573,10}; ine(1) = { -1.5253.500, 6162.586, 9616.573,10}; ine(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, 2 ine boop (1) = { 1, 1 ; ransfinite Surface "**; | int(| $29) = \{ -14177, 360, $ | 5501.569. | 9893.504.1c1 | | | | | | | | | | | | | | | | | | | | |
| ine (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, 2 ine Loop (1) = (1); lane Surface (1) = (1); ransfinite Surface "**; | int (| 30) = { -15253,500, | 6162.586. | 9616.573.1c1 | | | | | | | | | | | | | | | | | | | | |
| <pre>ine loop (1) = (1) ; hans Surface "**";</pre> | ne (1) | ={ 1, 2, 3, | 4. 5. 6 | . 7. 8. | 9. 10. | 11. | 12. | 13. | 14. 1 | 5. 16 | . 17. | 18. | 19. | 20. | 21. | 22. | 23. | 24. | 25. | 26. | 27. | 28. | 29. | |
| lane Surface (1) = { 1) ; ransfinite Surface "**; | ne Loop | $(1) = \{1\};$ | | | | | | | | | | | | | | | | | | | | | | |
| ransfinite Surface ***; | ane Sur | <pre>rface (1) = { 1} ;</pre> | | | | | | | | | | | | | | | | | | | | | | |
| | ansfini | ite Surface "*"; | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | 1 |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | Î |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | |

111

In the Notepad++, set the characteristic length (Ic) 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;

| | e:\W | /indas 10 - | Mesh fo | r Ana | lysis\R2 | A.geo - | Notepa | d++ | | | | | | | | | | | | | | |
|------|-------|-------------|---------|-------|----------|---------|--------|----------|---------|-------|---------|------------------|-------|-------|-----|-----|-----|-----|-----|-----|------|---|
| File | Ec | dit Search | h View | En | coding | Langu | iage S | Settings | Tools I | Macro | Run Plu | ugins Wi | ndow | ? | | | | | | | | |
| | -0 | | a na 4 | | k Bi | In s |) C | - m h | | | | 1 12 7 | z 🔊 🕽 | A = 4 | D I | | | | | | | |
| | C | | | - | -0 0 | | _ | | | | | | | | | | | | | | | _ |
| | mesh. | .geo 🗵 🔚 | R2A.geo | | | | | | | | | | | | | | | | | | | |
| | 1 | lc=25 | 50.0 ; | | | | | | | | | | | | | | | | | | | |
| | 2 | Point (| 1) | = | { - | 14818. | .740, | 4 | 659.63 | 7, | 9733. | 402,1c} | ; | | | | | | | | | |
| | 3 | Point (| 2) | = | { -: | 14505 | .570, | 33 | 124.36 | 5, | 9813. | 839,1c} | ; | | | | | | | | | |
| | 4 | Point (| 3) | = | { - | 14316. | .650, | 1 | 567.584 | £, | 9860. | 868,1c} | ; | | | | | | | | | |
| | 5 | Point(| 4) | = | { -: | 14253 | .500, | | .000 | ο, | 9876. | 338,1c} | ; | | | | | | | | | |
| | 6 | Point(| 5) | = | { - | 13130. | .650, | | .000 | ο, | 10130. | 710 ,1 c} | ; | | | | | | | | | |
| | 7 | Point(| 6) | = | { - | 12000. | .000, | | .000 | ο, | 10347. | 820,1c} | ; | | | | | | | | | |
| | 8 | Point(| 7) | = | { -: | 10106. | .690, | | .000 | ο, | 10625. | 630,1c} | ; | | | | | | | | | |
| | 9 | Point(| 8) | = | { · | -8200 | .977, | | .000 | ο, | 10799. | 020,1c} | ; | | | | | | | | | |
| 1 | .0 | Point(| 9) | = | { - | -6000 | .000, | | .000 | ο, | 10868. | 660,1c} | ; | | | | | | | | | |
| 1 | .1 | Point(| 10) | = | { - | -3799. | .023, | | .000 | ο, | 10799. | 020,1c} | ÷ | | | | | | | | | |
| 1 | .2 | Point(| 11) | = | { · | -1893 | .310, | | .000 | ο, | 10625. | 630,1c} | ; | | | | | | | | | |
|]] | .3 | Point(| 12) | = | { | | .000, | | .000 | ο, | 10347. | 820,1c} | ÷ | | | | | | | | | |
| 1 | .4 | Point(| 13) | = | { | 1635. | .509, | | .000 | ο, | 10021. | 170,1c} | ; | | | | | | | | | |
| 1 | .5 | Point(| 14) | = | { | 3253 | .500, | | .000 | ο, | 9616. | 573,1c} | ; | | | | | | | | | |
| 1 | .6 | Point(| 15) | = | { | 1693. | .157, | | 149.493 | з, | 10006. | 640,1c} | ; | | | | | | | | | |
| 1 | .7 | Point (| 16) | = | { | 476. | .134, | | 306.932 | 2, | 10261. | 110,1c} | ; | | | | | | | | | |
| 1 | .8 | Point(| 17) | = | { | -714 | .285, | | 499.919 | э, | 10464. | 230,1c} | ; | | | | | | | | | |
| 1 | .9 | Point(| 18) | = | { - | -1904 | .598, | | 728.42 | 5, | 10626. | 970,1c} | ; | | | | | | | | | |
| 2 | 20 | Point (| 19) | = | { - | -3074 | .119, | 9 | 991.200 | 6, | 10744. | 740,1c} | ÷ | | | | | | | | | |
| 2 | 1 | Point(| 20) | = | { · | -4238 | .631, | 12 | 288.380 | з, | 10824. | 080,1c} | ÷ | | | | | | | | | |
| 2 | 2 | Point (| 21) | = | { - | -5386 | .975, | 10 | 621.39 | L, | 10860. | 960,1c} | ; | | | | | | | | | |
| 2 | 3 | Point(| 22) | = | { · | -6526 | .720, | 19 | 984.580 | 6, | 10864. | 670,1c} | ; | | | | | | | | | |
| 2 | 24 | Point(| 23) | = | { · | -7654. | .606, | 2: | 384.55 | 7, | 10828. | 650,1c} | ÷ | | | | | | | | | |
| 2 | 5 | Point (| 24) | = | { · | -8769. | .110, | 28 | 816.280 | 6, | 10758. | 370,1c} | ; | | | | | | | | | |
| 2 | 6 | Point(| 25) | = | { · | -9875 | .331, | 33 | 285.930 | ο, | 10650. | 680,1c} | ; | | | | | | | | | |
| 2 | 27 | Point(| 26) | = | { - | 10965. | .560, | 3' | 784.220 | ο, | 10512. | 780,1c} | ; | | | | | | | | | |
| 2 | 8 | Point (| 27) | = | { -: | 12050. | .790, | 4: | 322.509 | э, | 10338. | 640,1c} | ; | | | | | | | | | |
| 2 | 9 | Point(| 28) | = | { - | 13115. | .300, | 48 | 890.62 | 5, | 10133. | 910,1c} | ÷ | | | | | | | | | |
| 3 | 0 | Point (| 29) | = | { - | 14177. | .360, | 5 | 501.569 | э, | 9893. | 504,1c} | ; | | | | | | | | | |
| 3 | 31 | Point(| 30) | = | { -: | 15253. | .500, | 6. | 162.580 | 6, | 9616. | 573,1c} | ; | | | | | | | | | |
| 3 | 32 | Line (1 | 1) ={ | | 30, | 1, | 2, | з, | 4} | ; | | | | | | | | | | | | |
| 3 | 33 | Line (2 | 2) ={ | | 4, | 5, | 6, | 7, | 8, | 9, | 10, | 11, | 12, | 13, | 14} | ; | | | | | | |
| 3 | 34 | Line (3 | 3) ={ | 1 | 4, : | 15, | 16, | 17, | 18, | 19, | 20, | 21, | 22, | 23, | 24, | 25, | 26, | 27, | 28, | 29, | 30}; | |
| 3 | 35 | Line Lo | oop (1 |) = | {1,2 | ,3}; | | | | | | | | | | | | | | | | |
| 3 | 86 | Plane S | Surfac | e (| 1) = | { 1} ; | ; | | | | | | | | | | | | | | | |
| 3 | 37 | Transfi | inite | Sur | face | *"; | | | | | | | | | | | | | | | | |
| 3 | 88 | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |

File | Save the edited notepad and switch back to Gmsh interface. Then click on Modules |



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





113

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

| MSH Options | | | | | | | | |
|------------------------------|--------------|--|--|--|--|--|--|--|
| Version 2 ASCII Format | | | | | | | | |
| Save all elements | | | | | | | | |
| Save parametric coordinates | | | | | | | | |
| Save one file pe | er partition | | | | | | | |
| Save partition topology file | | | | | | | | |
| ок <= | Cancel | | | | | | | |

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.

| | | | | 114 |
|---|--|---|--------------|----------|
| Windas 10 (Build 20170701) - Lightweight Structure Anal File Generate Design Edit Facility Display Delete New | lysis Program e List Modify View Solution WinFabr | ic Gmsh 1] <u>태국</u> 등 · · · · · · · · · · · · · · · · · · | | - • × |
| Open Save | | | | |
| Save as | | | | |
| Recall | > | | | |
| Export | | | | |
| Import | AutoCAD {DXF R12} | | | |
| Print | Gmsh {mesh} | | | |
| Capture Screen 1 D:\Windas 10 - Mesh for Analysis - White\R2A.wds 2 D:\Windas 10 - Mesh for Analysis\R2AC_ff.wds 3 D:\Windas 10 - Mesh for Analysis\R2AB_ff.wds | | | | |
| 4 D:\Windas 10 - Mesh for Analysis\R2AF.wds | | | | |
| Quit Exit | | | | |
| å ⊾⊳ _x | | | | |
| Cell (65) 6765-6288 Fax: (65) 6765-1588 Web Import Gmsh model in mesh format @ Windas 10 (Build 20170701) - Lightweight Structure Anal File Generate Design Eait Facility Display Delet Import Gmsh method Color Import Delet | lysis Program List Modify View Solution WinFabr | ic Gmsh | elect of all | - 0 X |
| 에 바람이 있는 것이 없는 것이 없 있 | nin (a a a a a a a a a | | | |
| | | | | |
| Tet:(65) 6765-6288 Fax:(65) 6765-1588 Wet: Jser ID: Multimedia_engineering moot Gmsh's nodes done moot Gmsh's nodes done | o: www.me.com.sg Email.mepl@me | .com.sg | | <u>م</u> |
| Internet and the second s | | | | • • |

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

| Entity Selection-Enter element(s) to Select | × |
|---|----------------------------------|
| Selection by range | ┌ Options |
| 1 to 1 Step 1 Select | All Display Group Color Property |
| Excluding mode Off | Last Reset Cancel Accept |
| t11,t12,t13 | |

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**.



| Generate Support On Selecter | d Nodes | × |
|------------------------------|-------------|--------|
| DOF | Pinned | Cancel |
| V TX V TY V TZ | Fixed | Accept |
| | No Rotation | |
| Prescribed Displacements | | |
| dx .0000 dy | .0000 dz | .0000 |
| nx .0000 ry | .0000 rz | .0000 |





| WinFabric System Variables | × |
|---|------------------------------|
| Form Finding | Precision and Tolerance |
| Fabric net type 📀 Regular 🔿 Radial | Warp-weft angle 45 |
| Number of iteration 2 | Minimum triangular angle 1.5 |
| Number of points for border segments 0 | Arch constructor node 5.0 |
| Sag amount in % for border segment 0 | Minimum cable length 100.00 |
| Force density for fixed border segments .00 | Minimum net length 50.00 |
| h-Contour Interval 50.00 | Accept |
| Minimum rainwater runoff 7.50 | |
| Scaling factor for symbol display 1.00 | |

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





Copyright © 2020 Multimedia Engineering Pte Ltd

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.





119

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**.

| Element | DIERCONSERACE EX AGE ##988 # # ADE # ADE # ADE A |
|----------------------------|---|
| Element {more} | re\RZAF.wds] |
| Membrane Constructors | |
| Membrane Forms | |
| Spaceframe Forms | > |
| Linear Truss | |
| Delta Beam | |
| ETFE Forms | > |
| Mesh | |
| Region | |
| /olume | |
| Node | |
| Node {more} | · AAAA |
| Material Property | KAT RIX ADD |
| Standard Material Property | |
| Tensile Membrane Property | KAXXXXXXXXXXXXXXX |
| CHS Tables | · KATATATATATA |
| Extra-Stiffness | KADAA KAAAA KAAAA KAAAAAAAAAAAAAAAAAAAA |
| Load | Point Load |
| Support | Point Load [Z-Equivalent] |
| KARK (| |
| | Pre-Stress KAN XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX |
| KARA | |
| KXXXX | Presure I Wind Loads Cable Cable |
| x XXXXX | Themail Net Net A A A A A A A A A A A A A A A A A A A |
| | |
| ∟ ⊳ _x | |
| | |
| | |
| iet tune | |
| warp and weft lines within | n tolerane of 45.00 |
| model in progress | |
| ved | |
| | |
| | |

| Force Density On Selected Net Link(s) | | | | | |
|---------------------------------------|-----|--------|--|--|--|
| Force density value (kN/m) | 100 | Accept | | | |

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

parameter.

| Windas 10 (Build 20170701) - Lightweight Structure Analysis Program File Generate Design Edit Facility Display Delete List Modify View Solution V | VinFabric Gmsh | - 0 X |
|--|---|----------|
| | Force Density Form Finding | |
| Model - [D:\Windas 10 - Mesh for Analysis - White\R2AF.wds] | Inflated Mesh Volume Form Finding | |
| | FD -> CableNet Mesh -> FD Model >> Mast -> Truss | |
| KAL I | External Border > Triangulation Refinement | |
| | Ring2Ellipse Ring4Analysis | |
| | WinSeam WinPattern | |
| | h-Contour Rainwater runoff >> Surface Gradient >> Masts & Tieback Cables Drawina >> | |
| | Ser Fabric Variables Fabric Surface > Volume | |
| | <u></u> | |
| Regular net type Forming warp and weft lines within tolerane of 45.00 Compact model in progress Model saved | | <u>م</u> |
| Form finding by force density method | | • • |

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.

| Entity Selection-Enter element(s) to Select | × |
|---|------------------------------|
| Selection by range Options | |
| 1 to 1 Step 1 Select All | Display Group Color Property |
| Excluding mode Off | Last Reset Cancel Accept |
| t11,t12,t13 | |

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.



1 🚛 🗛 🚥 🤤 💼 🏛 🛄 🧔 🌍 🗩 😵 📓 🕿 🥹 🖉

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**.

へ 空 (1) ENG 7:05 PM 7/4/2017

| Entity Selection-Enter element(s) to Select | × |
|---|------------------------------|
| Selection by range Options | |
| 1 to 1 Step 1 Select All | Display Group Color Property |
| Excluding mode Off | Last Reset Cancel Accept |
| 1707,7,11,10,9,8,6,5,4,3,2,1 | |

Copyright © 2020 Multimedia Engineering Pte Ltd

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.

| Windas 10 {Build 20170701} - Lightw File Generate Design Edit Facilit | eight Structure Analysis Program ty Display Delete List Modify View | Solution WinFabric Gmsh | | | | - | o x |
|--|--|-------------------------|-----------|-------------|-------------|-------------|----------|
| | Iniainini piniaiaiai | a 💷 🔊 🛦 (2) 💶 🔳 🖶 🔛 💡 | 146 #¥998 | | | | |
| Model - [D:\Windas 10 - Mesh for Ana | lysis - White\R2AT_ff.wds] | | | | | | |
| Č, x | | | | | | | _ |
| Model saved | | | | | | | |
| Unable to find member with pre Model saved | e-tension or pressure load specifi | ed ! | | | | | |
| Model saved | | | | | | | |
| | | | | | | | |
| Elements=0 | Nodes=0 | Membrane | Color= 1 | Property= 1 | Loadcase= 1 | Groun=F1.N1 | <u> </u> |

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).

| Inflated Mesh Volume Form Finding X | | | | | | | | |
|--|--|--|--|--|--|--|--|--|
| Iteration Load increment Number of Iterations 99 • Convergence Norm .100E-03 | | | | | | | | |
| Compute Cushion Height under initial pressure | | | | | | | | |
| Top layer foil thickness 250 micron, inflated to 1.00 mm height | | | | | | | | |
| Bottom layer foil thickness 250 💌 micron, inflated to 2.00 mm height | | | | | | | | |
| Initial Prestress, kN/m 2.5 + Internal Pressure 600 + Pa [N/mm2] | | | | | | | | |

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.

| Windas 10 (Build 20170701) - Lightweight Structure Analysis Program File Generate Design Edit Excility Display Detete List Modify View Solution WinFabric Gmsh | - 0 X |
|--|---------------------|
| New | NRS EE ANNELIKAZ XX |
| Open | |
| Save | |
| Save as | |
| Recall | |
| Insert | |
| Export | |
| Import | |
| Print | |
| Capture Screen | |
| 1 D:\04_ADAS Task\ETFE Cushion Formfind Tutorial\Windas 10 Tutorial Manual - ETFE Mesh for Analysis\Tutorial File\R2AC_ff.wds | |
| 2 D:\04_ADAS Task\EFFE Cushion Formfind Tutorial\Windas 10 Tutorial Manual - EFFE Mesh for Analysis\Tutorial File\R2AB_ff.wds | |
| 4 D/04 ADAS Task/ETFE Cushion Formfind Tutorial/Windas 10 Tutorial Manual - ETFE Mesh for Analysis/Tutorial File/R2AT ff.wds | |
| 0.3 | |
| bit | |
| | |
| Model saved | |
| Model saved | |
| Maximum initiated height at Node 61 is 10865.290mm, internal pressure = .6000 kPa Model saved | |
| Model saved | |
| Model saved | - |
| | |

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 | |

| Elle Generate Design Edit Ev | ntweight structure Design & Analysis Program cility <u>D</u> isplay D <u>e</u> lete <u>List</u> <u>M</u> odify <u>V</u> iew <u>S</u> olution | WinFabric Gms | h | | | | | | - 0 ^ |
|----------------------------------|---|---------------|-----|------|-----------|----------|------|------|-------|
| | | a 🖻 🖪 🚺 🖠 | - M | 8881 | 📰 🖿 🖪 🎸 🚄 | <u> </u> | | | |
| Model - [D:\MISC\Windas 14 Tutor | ial 2020;Reference file;New Element;?plane.wds] | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| | | | | | | | | | |
| Parallel - Camara at { 500, | 8214, -5773), Target at [500, NI 50, | 0} | | | | | | | |
| | | | | | | | | | |
| Windas 14 Build 20200812 | En Contraction Dia List | | | | | | | | • |
| 50 Bukit Batok St 23 Midvie | w Bldg #05-15 Singapore 659578 | | | | | | | | - |
| 10 | | | | | | | | | • |

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.

| 🔳 Сору | elements | | × |
|------------|-------------|------|--------|
| _ Select t | wo points — | | |
| | х | Y | Z |
| Point 1 | ļ00 | .00 | .00 |
| Point 2 | 100.00 | .00 | .00 |
| Number | of copy | 10 - | Accept |
| | | | Cancel |

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

| 🔤 Wi | ndas Text Editor - | D:\MISC\Windas | 14 Tutorial 2020\Refe | erence file\New Ele | ement\7plane_el | ement.lst] | | | | - | | × |
|-----------------------|--------------------------------|----------------|-----------------------|---------------------|-----------------|------------|---------------------|-------|----|----|----|----|
| <u>F</u> ile <u>E</u> | dit <u>S</u> earch <u>H</u> el | p | | | | | | | | | | |
| | 🔁 🔒 🖣 | þ 🔍 🗡 | , i) 🖥 🖌 |)(| | | | | | | | |
| | Element | Element | Property | Group | Color | | Description | | | | | |
| Line | Number | Туре | Number | Number | ID | Area m2 | | 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 r | m2 | | | | | |
| | | | | | | | | | | | | |
| | | | | | | | | | | | | |
| Line:14 | Col:66 | NUM IN | 45 | | | | | | | | | |
| | | 1 1 2 2 2 1 2 | | | | | | | | | - | // |

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.

| as Wi | indas Text | Editor - [C:\User | rs\MEPL22\Desktop\nev | v element\16 Hyb | rid Brick_elem | ent.lst] | | | | | | | - | | × |
|-----------------------|---------------------------|-------------------|-----------------------|------------------|----------------|----------|------|-------|----|-------|-------|----|----|----|-----|
| <u>F</u> ile <u>E</u> | <u>E</u> dit <u>S</u> ear | ch <u>H</u> elp | | | | | | | | | | | | | |
| \Box | 2 |] 븕 🤇 | ة 🖬 🎖 🕻 | | | | | | | | | | | | |
| | Elem | ent Ele | ement Proper | ty Gro | up Co | olor | | | D | escri | otion | | | | |
| Line | Numb | er Typ | Number | Num | ber 1 | ED | Are | əa m2 | | | | | | No | des |
| Flow | ont | Flomont | Broporty | Crown | Color | | | | | | | | | | |
| Numb | ent | Tupe | Number | Number | Code | N | odee | | | | | | | | |
| N'unite | 1 | 16 | 1 | 0 | 3 | 5 | 6 | 7 | 8 | 1 | 2 | з | 4 | | |
| 1 | 2 | 16 | 1 | ő | 3 | 6 | 11 | 12 | 7 | 2 | 9 | 10 | 3 | | |
| 1 | 3 | 16 | 1 | ő | 3 | 11 | 15 | 16 | 12 | 9 | 13 | 14 | 10 | | |
| 1 | 4 | 16 | 1 | ő | 3 | 15 | 19 | 20 | 16 | 13 | 17 | 18 | 14 | | |
| 1 | 5 | 16 | 1 | õ | 3 | 19 | 23 | 24 | 20 | 17 | 21 | 22 | 18 | | |
| 1 | 6 | 16 | 1 | 0 | 3 | 23 | 27 | 28 | 24 | 21 | 25 | 26 | 22 | | |
| 1 | 7 | 16 | 1 | 0 | 3 | 27 | 31 | 32 | 28 | 25 | 29 | 30 | 26 | | |
| 1 | 8 | 16 | 1 | 0 | 3 | 31 | 35 | 36 | 32 | 29 | 33 | 34 | 30 | | |
| 1 | 9 | 16 | 1 | 0 | 3 | 35 | 39 | 40 | 36 | 33 | 37 | 38 | 34 | | |
| 1 | .0 | 16 | 1 | 0 | 3 | 39 | 43 | 44 | 40 | 37 | 41 | 42 | 38 | | |
| | | | | | | | | | | | | | | | |
| Line:1 | Col:1 | 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**.

| 💽 Generate Element | | | | × |
|------------------------|-------------|----------------|--------------------|-----------------|
| Element ID 11 1 2 0 | 3 0 4 | 0 5 0 6 | 0 7 0 8 | 0 Accept |
| <u>C</u> olor 1 | Property ID | 1 · Group ID 0 | • Element Solid 3D | ▼ <u>C</u> lose |

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 | t Editor - [C:\Users | \MEPL22\Desktop\new | v element\17 Brick | k8_element.lst | :] | | | | | | - | | × |
|---------------------------------------|----------------------|---------------------|--------------------|----------------|------|------|-------|----|--------|-------|----|----|----|
| <u>F</u> ile <u>E</u> dit <u>S</u> ea | arch <u>H</u> elp | | | | | | | | | | | | |
| 🗋 🔁 🛛 | _ 🖶 🔍 | . 🔏 🖬 🖥 | 50 | | | | | | | | | | |
| Elem | ment Eler | ment Proper | ty Gro | up Co | olor | | | De | escrip | otion | | | |
| Line Numb | ber Typ | e Number | Num | ber : | ID | Are | əa m2 | | - | | | | |
| | | | | | | | | | | | | | |
| Flement | Floment | Property | Group | Color | | | | | | | | | |
| Number | Type | Number | Number | Code | N | odes | | | | | | | |
| 1 | 17 | 1 | 0 | 1 | 3 | 7 | 8 | 4 | 2 | 6 | 5 | 1 | |
| 2 | 17 | 1 | õ | 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 | N | | | | | | | | | | | | 11 |

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.



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.

| Generate Fabric Net Proper | ty For Membrane | × |
|--|--|------------------|
| <u>T</u> ensile Fabric Material | FiberTop PTFE T400 | • <u>Apply</u> |
| \underline{W} arp (Radial) EA (kN/m) | Verseidag PTFE B18919 Verseidag PTFE B18039 | ^) 980.00 Cancel |
| <u>B</u> order Cable, Material | Verseidag PTFE B18089 Verseidag PTFE B18059 | nm) 12mm 💌 |
| <u>T</u> ieback Cable, Material | Verseidag PTFE B18656 FiberTop PTFE T300 | nm) 13mm 💌 |
| | FiberTop PTFE T400 | - |

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.

| | | | | | Fabria | Fat | oric | Ten | sile |
|-------------|---------|-----------|-------|------|--------|-------------|-------------|-------------|-------------|
| | | | | | Weight | Suiir kN | iess, /m | Strer kN | igtn, /m |
| Base Cloth | Coating | Brand | Model | Type | g/m2 | Warn | Weft | Warn | Weft |
| Dase Olotin | Obating | Diand | | турс | 9/112 | waip | vvon | waip | VVCIL |
| PVC | PvdF | Duraskin | B4951 | I | 800 | 600 | 300 | 60 | 60 |
| | | Duraskin | B4617 | II | 900 | 1000 | 500 | 88 | 79 |
| | | Duraskin | B4915 | | 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 | | 1050 | 1000 | 1000 | 112 | 112 |
| | | Ferrari | 1302 | IV | 1350 | | | 160 | 140 |
| | | | | | | | | | |
| Glass Fiber | Teflon | Verseidag | 18039 | I | 800 | 1195 | 1097 | 70 | 70 |
| | | Verseidag | 18089 | | 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 | | 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 | | Prestress kN/m | | n | | | |
|----------|-------------|-------|------|---------------------------|------|----------------|------|---------|------|------|---------|
| Material | Manufacture | Model | Туре | Warp | Weft | | Warp | | Weft | | |
| | | | | | | Min | Max | Us e | Min | Max | Us e |
| 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 | | 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 | | 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 | | 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 :

- Dead load due to fabric, cables, fittings, and clamping plates is generally taken as 0.02kN/m²
- <u>Rain load</u> due to rain water flow on the membrane surface is assumed to be 10mm thick or 0.10 kN/m² 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}.**

| Control Parameters - Windas Geometrical Non Linear Solver | | | | |
|---|--------------------------------|-----------|--|--|
| Incremental - Iteration Scheme | Convergence Criteria | | | |
| Number of Load Steps per Load Case | Displacement Norm | 1.000E-02 | | |
| Load Increment per Load Step 0.100 | □ Force & Residual Norm | 1.000E-03 | | |
| Maximum Number of Iterations/Load Step 1000 - | Energy Norm | 1.000E-07 | | |
| Iteration Method Newton Raphson | | | | |
| Strain Measure Engineering Strain | Pivot to terminate solution <= | 1.000E-12 | | |
| ☐ Stop after exceeding maximum number of iterations | [| ОК | | |

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 | Reaction Y-Axis | Reaction Z-Axis |
|------|------|-----------------|-----------------|-----------------|
| | | (kN) | (kN) | (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/m². Apply the load as equivalent point load using the **Generate | Load | Point Load (Z-equivalent)** command.

| Generate Equivalen | t Point Load | l i i i i i i i i i i i i i i i i i i i | | × |
|--------------------|--------------|---|-----|--------|
| Pressure Load of | -0.12 | kN/m2 and Load Case is | 1 - | Accept |

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 Maximum y-displacement of | 3.964 2.788 | at node at node | 322 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.

Dynamic Classification

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, | $S_{\rm S} = 1.00$ | | | | |
| 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) | | | | | |

 $C_{\rm p}$ 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) = $0.693 \times 0.4 \times 0.92 = 0.255 \text{ kN/m}^2$ Wind Load (Suction) = $0.693 \times 0.6 \times 0.92 = -0.382 \text{ kN/m}^2$



142

Copyright © 2020 Multimedia Engineering Pte Ltd

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/m^2 is applied on Area A in the **negative zdirection** based on this calculation :

Wind suction + Dead Load = $0.382 + 0.02 = -0.384 \text{ kN/m}^2$

Whereas, a uniformly distributed surface load of **0.253** kN/m^2 is applied to Area B in the **positive z-direction** based on this calculation :

Wind pressure + Dead Load = $0.255 - 0.02 = 0.253 \text{ kN/m}^2$

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.

| Select membrane element(s) to apply pressure | × |
|--|----------|
| Celection by range Options | |
| 1 to 1 Step 1 Select All Display Group Color | Property |
| Excluding mode Off Last Reset Cancel | Accept |
| | |
| | |
| Generate Pressure Load On Selected Elements | \times |
| Pressure Load of384 kN/m2 and Load Case is 2 : Accept | t |

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 = $-0.382-0.02-0.10 = -0.504 \text{ kN/m}^2$

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.

| Control Parameters - Windas Geometrical Non Linear Solver | × | |
|---|--------------------------------|-----------|
| Incremental - Iteration Scheme | Convergence Criteria | |
| Number of Load Steps per Load Case | Displacement Norm | 1.000E-02 |
| Load Increment per Load Step 0.100 | 🗖 Force & Residual Norm | 1.000E-03 |
| Maximum Number of Iterations/Load Step 1000 | Energy Norm | 1.000E-07 |
| Iteration Method Newton Raphson | | |
| Strain Measure Engineering Strain | Pivot to terminate solution <= | 1.000E-12 |
| ☐ Stop after exceeding maximum number of iterations | | ОК |

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 -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 -23.405 at node 190

Maximum z-displacement of -23.405 at node 397

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.

| inda 🔤 🔤 | Windas Text Editor - [D:\Temp\Clarence\Load Analysis\Adas_Hypar_ff_init_load_reaction 1.lst] | | | | | | | | | | |
|---------------------------|--|-------------------|-----------------------|-----------------|---------------|---------|------------|----------|---|---------|--|
| <u>F</u> ile <u>E</u> dit | <u>S</u> earch <u>H</u> elp | | | | | | | | | | |
| |) 🔒 🚔 | | 6 🖬 🕻 | 5 0 | | | | | | | |
| Projec Reacti | t : D:\Te on forces | mp\Clan listin | rence\Loa 1g == | d Analysis\Adas | _Hypar_ff_in: | it_load | | | | | |
| Load | Node | COOF | RDINATES | | | Fixity | REACTION (| kN) | | | |
| Case | Number | X-CC | DORD | 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 | 701 | 71.068 | .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 | -701 | 71.068 | .000 | 3500.000 | 111 | -86.690 | .000 | | -31.150 | |
| 2 | | | | | | | -103.340 | .000 | | -32.840 | |
| 3 | | | | | | | -100.100 | .000 | | -31.040 | |
| | Minimum | and Mar | kimum Rea | ctions | | | | | | | |
| | Minimum | Rxx = | -103. | 340kN at Node | 4 Maximur | n Rxx = | 87.030kN | at Node | 2 | | |
| | Minimum | Ryy = | -104. | 610kN at Node | 1 Maximur | n Ryy = | 104.610kN | at Node | 3 | | |
| | Minimum | Rzz = | -34. | 810kN at Node | 2 Maximur | n 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 | - | х | 1.5 | - | Warp/Weft | |
| Max. stress at border cable (pocket) | - | х | 1.5 | - | Warp/Weft | А |
| Max. stress at clamping | | | | | Warp/Weft | В |
| (System Point) | - | х | 1.5 | - | | |
| Max. stresses at seam | - | х | 1.5 | - | Weft | С |
| 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

| Entity Selection-Enter element(s) to Select | | × |
|---|------------------------------|------|
| Selection by range | Options | |
| 1 to 1 Step 1 Select | All Display Group Color Prop | erty |
| Excluding mode Off | Last Reset Cancel Acc | ept |
| GI | | |

| | Element | Element | First | secona | Property | Group | Color | Member |
|------|---------|---------|-------|--------|----------|--------|--------|-----------|
| Line | Number | туре | Node | Node | Number - | Number | Code | 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:



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.

| 🔲 Generate Seamline By Offset Method | | × |
|--------------------------------------|--------------------------------|----|
| Seamline ID 1 - Offset distance | 2000 📩 Number of offsets 🛛 1 📩 | OK |



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.

| Display Warp/Weft Stress 🛛 🗙 |
|------------------------------|
| Load Case 1 |
| • Warp Stress © Weft Stress |
| Stress range |
| From .00 To 10.00 |
| Contour level |
| |
| |
| High |
| Cancel Apply |



Repeat for load case 2 and 3.





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.

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