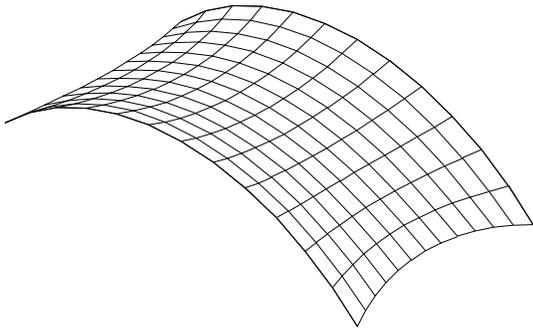


Applying FEA to a tensile structure

The starting point for any FEA analysis is a good understanding of what results we expect. This can be from prior experience with similar structures, or by an estimate of the stresses. Then we can use the FEA to get more exact figures for the deflections and stresses, whilst using our broad picture of what the results should be as a reality check on the FEA results.

So first we will describe how a tensile structure reacts to loads, and then run through an estimation of the expected loads, and then look at an example of using FEA on a simple structure.

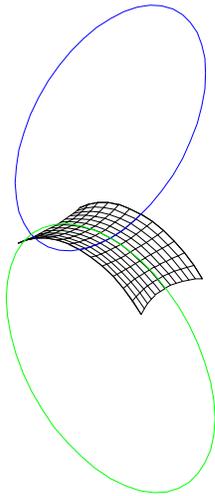


We will use a simple 3m by 6m barrel vault roof, between two arch supports, with cables on the bottom ends, as our model for this discussion.

The AutoCAD drawing this note was based on is also attached, called "Applying FEA"

/cont

How we expect a tensile structure to react to loads



A tensile structure's shape is maintained by the balance of two hoop stresses along the principle directions of curvature, shown in Blue and Green in the drawing.

These hoop stresses are the result of pre stressing the structure, using compensation, and of the structures shape, which determines the two hoop radii.

As loads are applied by wind suction, snow load, etc, they cause one of the hoop stresses to increase, and the other to decrease. (This change in stress will stretch or shrink the material, giving a different structural shape, but that's a further complication).

At some load value the structure will loose all hoop tension in one direction, and will then behave locally as an inflated structure. This can be the point where large deflections start.

So the pre-stress value is the main value that determines how the structure reacts to loads.

A high pre stress gives a stiff structure that will resist applied loads with low defections, but will need a strong material and supporting structure.

A low pre stress gives a structure that will deflect more with loads, but can be built with a weaker material and supporting structure.

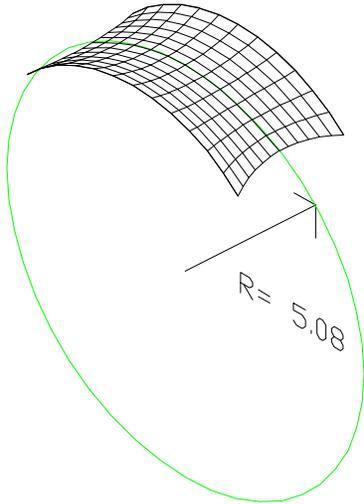
As a tensile structure, an external load will increase the fabric stress in one direction, and reduce it in the other.

Above a critical load, part or all of the structure will loose pre stress, and become an inflated structure, at least locally.

Estimating Structural Stresses

An estimate of the pre-stress needed to cope with an applied pressure, with the membrane just about to go slack in one direction, can be obtained from the cylinder stress equation:

$$\text{Skin Stress} = \text{Hoop Radius} * \text{Pressure.}$$



So looking at the barrel vault shown,

applying a suction of 0.25 kN/M^2
(equivalent to 45mph wind at $CP=1$)

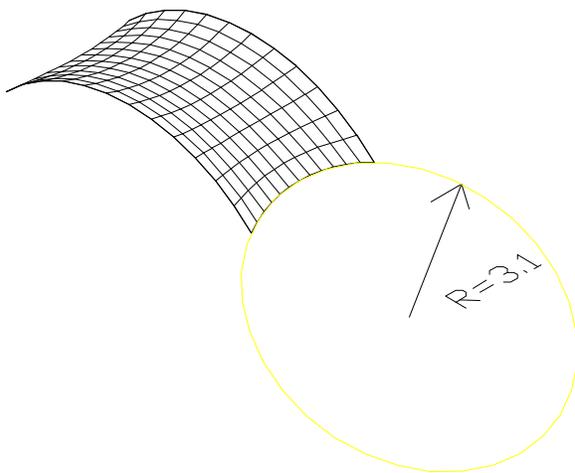
with a hoop radius of 5 M in the hoop that will be
withstanding suction,

giving a required pre-stress of approximately 1.2 kN/M .

Above this loading we would expect parts of the structure to become inflated, with larger deflections.

We can estimate the cable tension from the equation:

$$\text{Cable Tension} = \text{Skin Stress} * \text{Catenary Radius}$$



For the pre stress case we have a skin
stress of 1.2 Kn/M ,

Catenary radius of 3.1 M

giving a cable tension of 3.7 Kn .

The main structural reactions will be at the corners from the cable, plus a distributed load on the supporting frame from the membrane stress. These can easily be calculated.

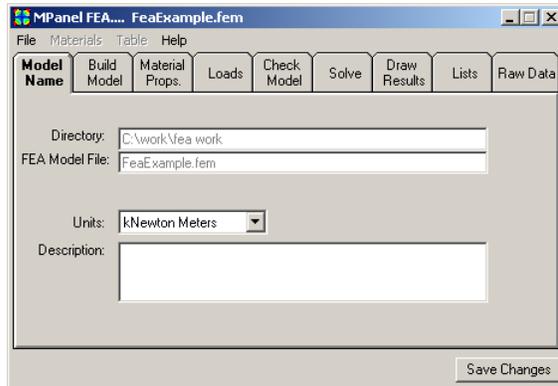
Note: the mesh hoop radiuses, and the catenary radius, can be obtained from the drawing as shown, or MPanel will report them by using the Info button on the mesh and on a 3Dpoly representing the edge cable.

/cont

Running an FEA Analysis.

The FEA Interface program is organised as a series of Tabs. To run an analysis you go through the tabs in order

Model Name

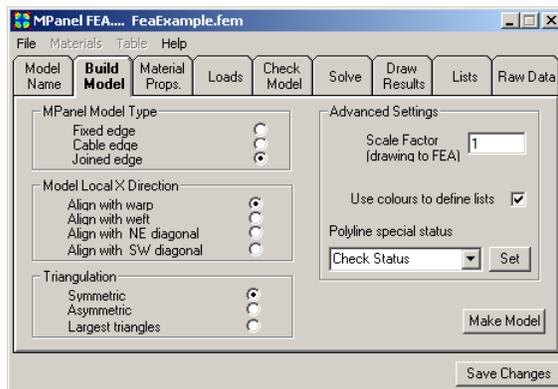


First we open the FEA Interface, and choose a file name. This will set the filename for the model file and the output file.

Also we choose a units system. This will start us off with some sensible default values. Here we choose to work in kNewtons and Meters.

It is important to note that the AutoCAD model must be drawn in the units that have been specified in the FEA. This model is a 3M by 6M model. If it was drawn in millimetres, as a 3000mm by 6000mm model, then the FEA would read it as a 3000M by 6000M model. You may have to scale your AutoCAD model to suit.

Build Model

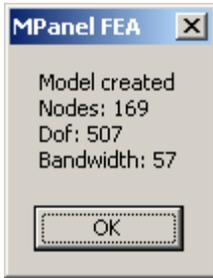


The model definition is the same as an MPANEL model, and on model with some cable edges all fixed points must be specified with a 3D poly.

This is not an problem as we have the 3Dpolys defining the framework already in the drawing.

The FEA model will output the stress results in terms of a localX and localY direction. Usually we want the localX to align with the model warp, so we select this option.

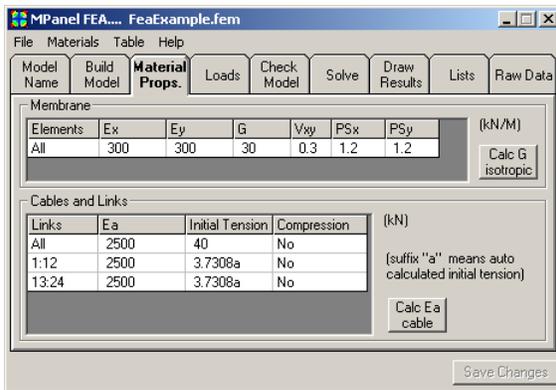
Then we select the model in AutoCAD, and make the model.



This window reports on the model build. The Bandwidth is an indication of how big the model looks to the FEA. Above a bandwidth of 150 the FEA solving becomes quite slow. But in this case it is just fine.

At this point the model file .fem is written to the disk.

Material Properties



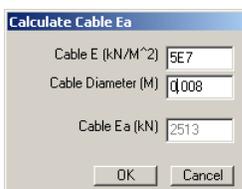
We start out with the default values that will be roughly correct for the unit system shown.

And then we alter the figures to suit our model case more exactly.

Ex and Ey are the stiffness values for the material, once it is pre stressed to the expected working stress. This value can be obtained from the fabric manufacturer, or from fabric tests, or from experience with similar materials.

Vxy is the Poisson ratio, which in a fabric is related to the crimp interchange. Values between 0 and 0.5 are suitable.

G is the shear stiffness value. On a loose weave fabric this may be close to zero, on a film it would be close to half the E value. This figure could be obtained from the manufacturer, or measured, but is often estimated as 10% of the E value.



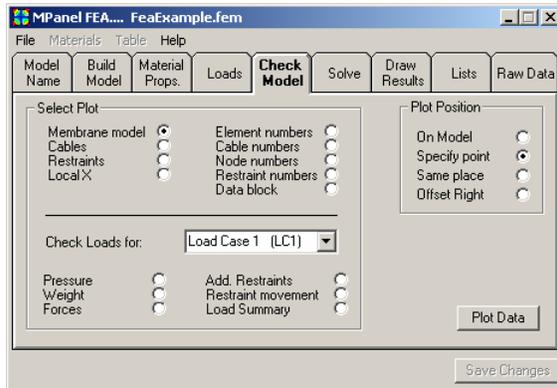
Ea is the cable effective E value. This is calculated from the cable area and the cable E, or it may be known from previous work. A calculator for Ea is provided, and is shown here with an 8mm cable.

We change the Membrane pre stress values to the 1.2 that we choose earlier.

We could also set the cable initial tension value to 3.7, the value that we estimated earlier. But for simple edge cables, the program will calculate this for us automatically, as shown by the "a" after the initial tension value.

And then we click on "Save Changes" to write the new values to the model file.

Check the Model

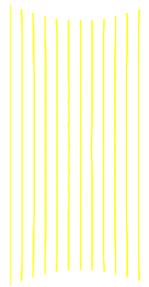
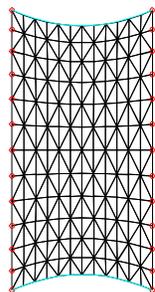


For now we will skip the loads tab, and check the model we have so far.

We will now draw out the Membrane Model.

We can choose where to draw the model... for this case choose "Specify Point".

Click on 'Plot Data' and click in the AutoCAD drawing to define that point.

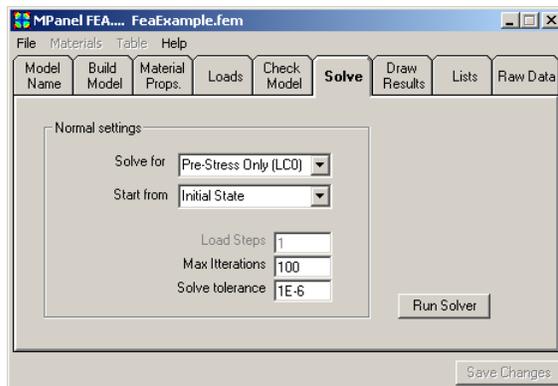


```
FeaExample.fem
Ex,EyGN/M= 400, 400
Vxy,Vyx = 0.3, 0.3
G0N/0= 40
Fxx, FyyGN/M= 1.2, 1.2
CreGN= 2513.256
```

We then draw in the cables and restraints, using 'Same Place' as plot position, and the localX direction, using 'Offset Right' as the plot position, to make sure that we have the model that we were expecting.

Also we draw the Data Block that summarises the model name, properties and loads.

Solve for Pre Stress

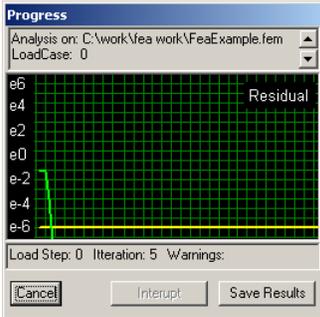


First we solve for load case 0, just the pre stress

This step is needed because the pre stress values entered for the membrane and cable are "first try" values.

In a simple structure like this they will be pretty close to the final values, but in more complicated structures the model will have to move to achieve a pre stress balance.

So we make sure we are solving for load case 0. Then we run the solver.



This is the solver progress window.

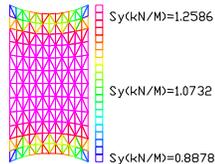
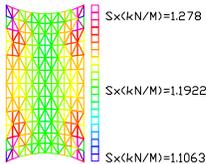
The green line is the graph of the residual.

The residual is a measure of how far the model is out of force balance. Typically when this figure is less than $1E-3$ the model is pretty well force balanced, and the results are valid.

We have specified $1E-6$ to make sure of the result.

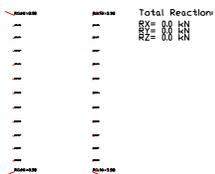
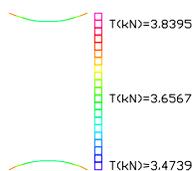
So now you click on Save Results, and an output file is written with the stress and displacement results.

Draw Results



So now we plot out the results from the output file, to check the pre stress load case.

We will look at S_x , S_y , the cable tension, and the reactions.



The model has adjusted the stresses a little to get a pre stress force balance.

S_x values are all close to the initial pre stress of 1.2

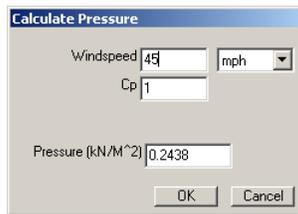
S_y varies more, with the lowest figures near the cable corners, a difficult area to model.

The Cable tension agrees well with the estimate of 3.7

The reactions all look correct, and in particular the total reaction is zero. The total reaction should equal the total external load, so its value gives a good check that the solution was valid.

Loads

Now we will look at some load cases. First we look at the suction load caused by a 45mph wind.

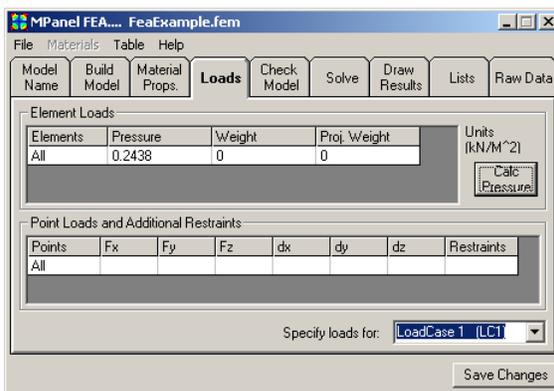


The 'Calculate Pressure' dialog box shows the following settings: Windspeed is 45 mph, Cp is 1, and the resulting Pressure is 0.2438 kN/M^2. There are OK and Cancel buttons at the bottom.

The pressure calculator works out the pressure that 45 mph will apply, in the correct units, for a given coefficient of pressure. In this case it is 0.25 kn/m^2

The coefficient of pressure to use may be specified in local building codes, or may be known from wind tunnel tests.

A value of 1 is probably conservative.



The 'MPanel FEA... FeaExample.fem' dialog box shows the 'Loads' tab selected. The 'Element Loads' table is populated with the following data:

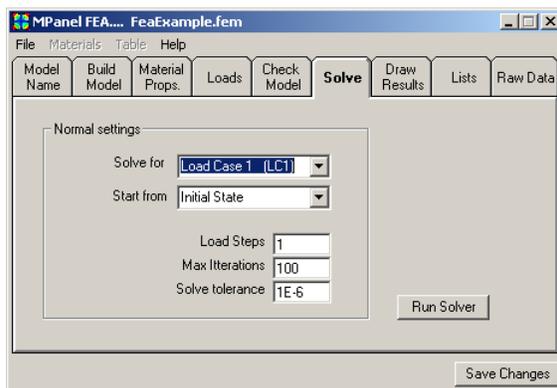
Elements	Pressure	Weight	Proj. Weight	Units (kN/M^2)
All	0.2438	0	0	

Below the table, there are sections for 'Point Loads and Additional Restraints' and a 'Specify loads for:' dropdown menu set to 'LoadCase1 (LC1)'. A 'Save Changes' button is at the bottom right.

The value is entered in the pressure column, and applied to All elements.

(You can also apply to just part of the structure, but that's a more advanced analysis).

This is defining Load Case 1.



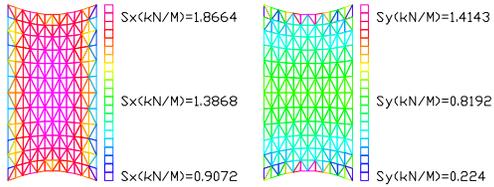
The 'MPanel FEA... FeaExample.fem' dialog box shows the 'Solve' tab selected. The 'Normal settings' section includes the following options:

- Solve for: Load Case 1 (LC1)
- Start from: Initial State
- Load Steps: 1
- Max Iterations: 100
- Solve tolerance: 1E-6

A 'Run Solver' button is located at the bottom right, and a 'Save Changes' button is at the very bottom.

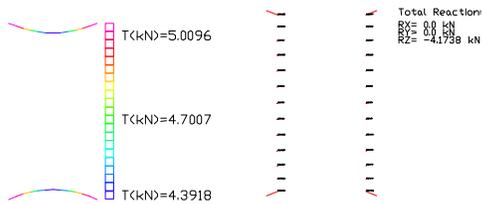
Then we solve in the same manner as before, except that now we solve for load case 1.

Now we can look at the results for this load case.



The Sx has increased, and the Sy decreased, as expected.

The membrane is still in tension, but the Sy is dropping down close to zero.

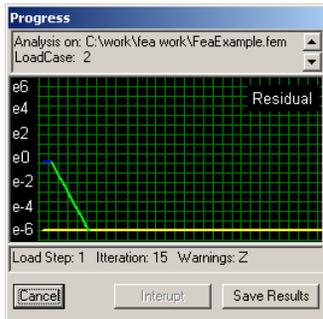


The cable tensions have increased, and the reactions now show a resultant force in Z, which will equal the projected area times the applied pressure.

There is zero X and Y reactions as the model is symmetrical.

Load Case 2

Now let's change the pressure load to 0.5 kn/m^2 , equivalent to 65 mph wind suction, save the changes, and run the solver as before. We called this load case 2.



Now the solution has some warnings.

The Blue graph line indicates large displacements, with some damping being applied.

The Z warning means zero stress in some elements.

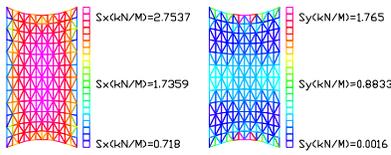
In these situations the model takes more iterations to solve.

Usually allowing enough iterations will get a solution.

In some cases we need to split the load into several smaller load steps to get a solution.

And in some combinations of loads and properties, the model will not solve... the residual will not come down to our desired figure of $1\text{E-}6$.

But in this case we have a valid solution as the residual has come down to $1\text{E-}6$



Now the S_y is close to zero on much of the structure.

The elements that have gone slack are best shown up by plotting the Second Principle stress.

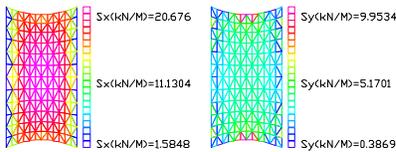
This is the lowest stress in the element, irrespective of direction, and the slack fabric is marked in white/black. This will be wrinkled fabric.



Load Case 3

Lets now go to an extreme case, and look at loading the structure to 5 kn/m^2 . This is equivalent to a wind of 200 mph.

There is no real requirement to solve at this high a pressure, but it is instructive to see what happens to the model.

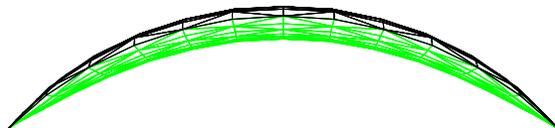


Now the S_x and the S_y are both increasing, although some areas of the model still have slack fabric.

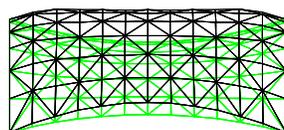
To see what is happening we plot out the displaced shape twice, with a scale factor of 1 in white/black, and with a scale factor of zero in green.



This gives us a comparison between the displaced shape and the original shape.



Looking closer at the displaced shape, in side and front views, we can see that under this extreme loading the membrane has completely inverted, and is now operating as an inflated structure.



Under more normal loading, just parts of the structure may invert, but this illustrates an extreme case.