# Introduction

Performing FEA analysis on tensile structures is complicated by several factors. The results are influenced by the fabric properties, the fabric pre stress and the cable initial tension, and the seam orientation in the model. Under loading the model may deflect by large amounts, part of the membrane may go partially or completely slack, and this may result in unusual deflected shapes.

Often the fabric properties are only poorly known, and are subject to visco elasticity and other time and history related effects. The pre-stress levels are at best estimated, and the seam orientation may not be known at this stage in the design.

This program works around these problems by supplying a simple, step by step interface called MPanel FEA, which runs a powerful non linear solver. The solver is robust and generates good solutions even when large parts of the membrane are slack.   By keeping the operation and concepts simple, it is hoped to produce results that, if not completely accurate, are clear and understandable.

Model Name
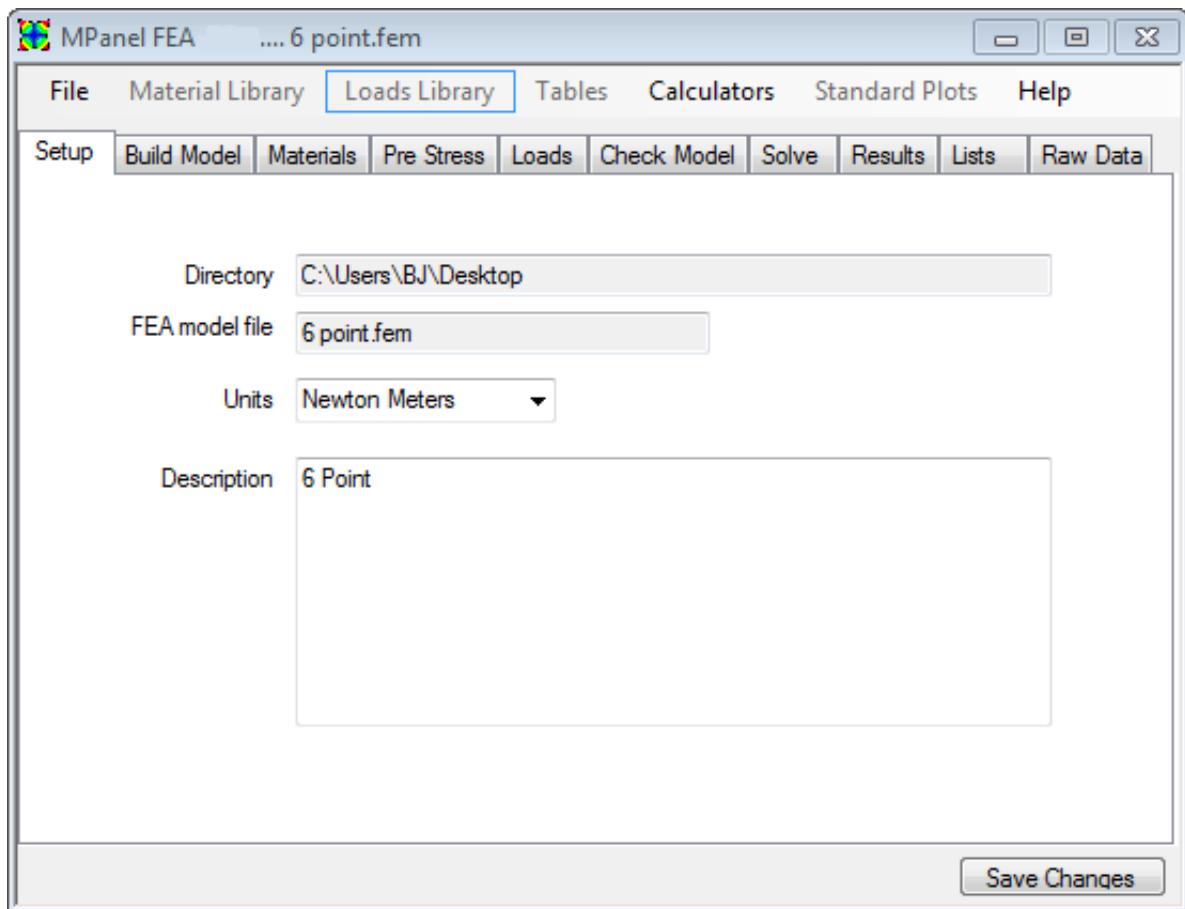
First, you have to specify a model name. Do this by using the **"File... New"** from the menu. Usually you will end up with several model files, all in the same directory.

To open an existing model you can either use the **"File... Open Existing"** or the **"File... Reopen Recently Used"**

You can specify the Units used in the analysis. This will provided units in the property names, which can help with understanding the required values. And it will put in default property values that are appropriate for the units chosen.

Note: The units must be the same as the drawing units. So if you specify "Kgf Meters" then the drawing must be in meters, not millimeters, centimeters, or inches.

Optionally you can enter a text description of the model. This is stored but not used by the program,.

Build Model
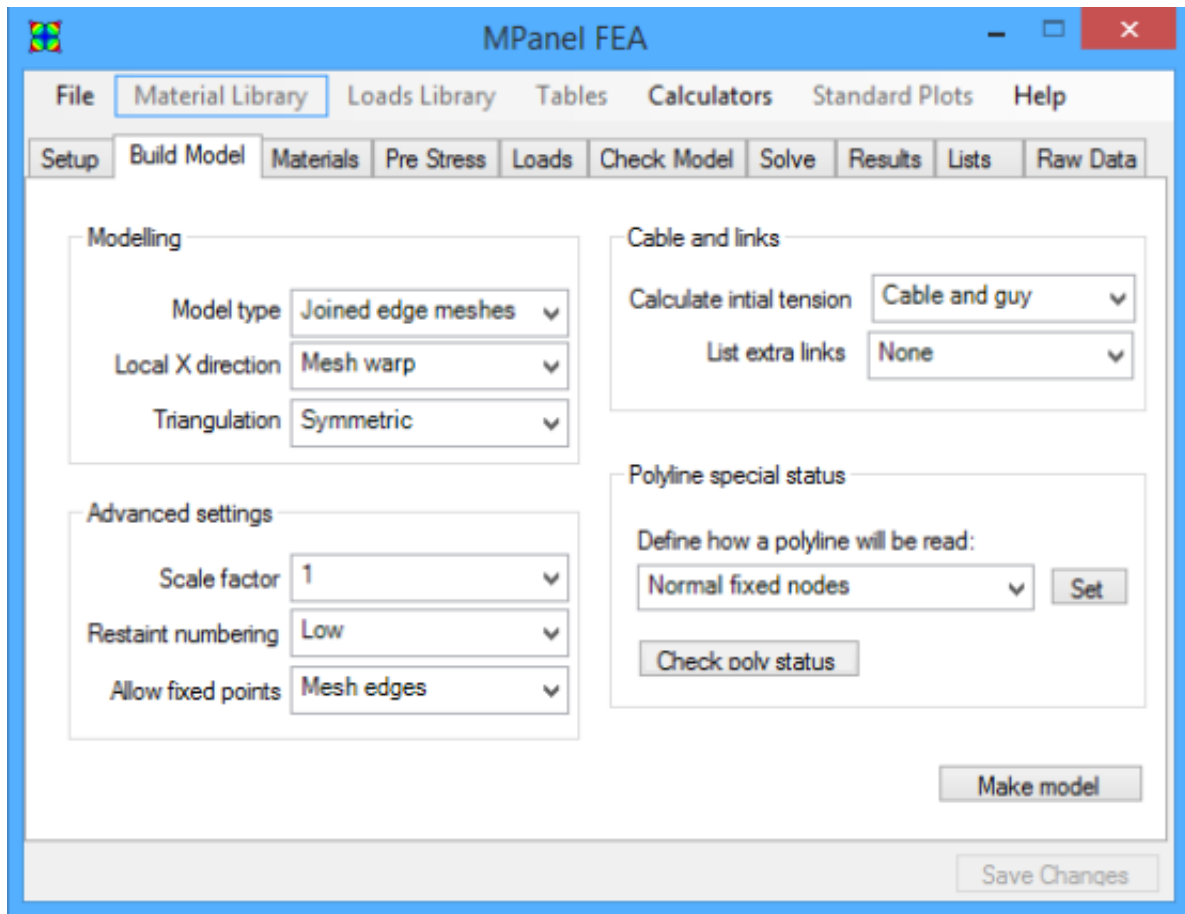
The "Make Model" button will read a drawing from CAD consisting of:

Meshes   to define the fabric surface
3DPolys   to define the system fixed points on mesh edges or line ends
Lines   to define any guys

A normal MPanel model usually meets these conditions.

In most cases you can just select the MPanel model and click the "Make Model" button to make the FEA model. Option settings adjust the FEA models to cope with more advanced requirements.

Make Model and Optimisation.

Select the whole model, and click on the "Make Model" button.

The FEA model will be built and default materials specified. If you are rebuilding an existing model there will be the option to keep the existing materials, loads and lists.

The FEA model is optimised for fast solving by numbering the nodes in a particular order, to give the minimum bandwidth. . A message box will report on the model build, including the bandwidth and number of regions. Bandwidths up to about 150 solve in reasonable fast time, and this is sufficient to cope with a double cone tent consisting of 8 13*13 meshes.

The model may be in several regions that are not connected to each other, or are connected only through fixed points, like a multi bay barrel vault. The optimiser will sort out the model into separate regions, and optimise each one separately..

If the option to retain any existing settings is rejected, the model file will be a clean build, using the default values to membrane and cables properties. All load cases will be deleted, and the Lists page cleared.

Modelling Options

- MPanel Model Type

Fixed edges: will add restraints to all edges.
Cable edges: will add a cable to all mesh edges, including internal mesh-mesh edges.
Joined edges: will add a cable to just the external mesh edges.

Usually this selection will be the same as the relaxation method used in MPanel.

- Local X direction

The FEA model Local X direction is set to be in line with the MPanel mesh warp, the weft, or diagonally.

This is meant to represent the seam direction, so that we can interpret X stress as parallel to the seams, and Y stress as across the seams. If at this stage the best seam orientation is not known then use "Align with warp"

The model Local Y direction is at right angles to the Local X direction, in the local mesh plane.

- Triangulation

Normally the FEA model generated will be mesh symmetrical if the meshes have odd Nsurf and Msurf numbers. Even numbers will work, but give some the model some extra difficulties in the corners.

The option to generate asymmetric meshes, which have all the elements laid in the same direction, allows comparison with other FEA products that use this method of meshing.

In models that have skinny corners, it is possible for some of the triangular elements to become very thin. This may prevent a model from solving, and in these cases it is possible to mesh to create the largest triangles.

Advanced Settings

- Scale factor

You may have a model drawn in millimetres that you want to FEA analyse in meters. You can specify a scale factor, from the drawing to the FEA, of 0.001   This will be used at Model Build, Check Model, and Draw Results so that the FEA analysis will work in the specified units, with a drawing in different units.

- Restraint numbering

Usually it is convenient to have the nodes numbered so that the restraints are low numbers, like 1, 2, 3. For compatibility with earlier FEA models you can also specify that restraints are high numbers.

- Allow restraints

The model building is faster if restraints are only allowed on mesh edges (or ends of lines). If you have restraints in the mesh interior then you can specify that these points should be checked also.

Cables and links

- Calculate initial tension

Normally the FEA build will find any cables om mesh edges, and calculate an

appropriate initial tension. Often the cables will terminate in a corner with a single guy, link or corner adjuster, and the initial tension for that can also be calculated.

In some models this calculation may not be required, so can be turned off.

In models with pole and guy tripods in the corners you can specify that the FEA should find these, and calculate initial tensions and compressions.

- List extra links

Normally the FEA will list any cables or significant links in the link table. Sometimes you want to list any links defined by CLists, or any links not otherwise listed. There are options for both of these.

- Polyline special status

Normally Polylines are read as a series of restraints. Optionally each polyline can be "tagged" to make the FEA interpret it as follows:

```
Ignore Poly              Specifies ignore the poly when fixing
nodes,
                         but uses the poly colour to define Plists.

Mirror YZ Plane          Mirror the model about the YZ plane, ie a
east/west mirroring.
                         (this adds a restraint in the X direction
only at these nodes)

Mirror XZ Plane          Mirror the model in the XZ plane,  ie
north/south mirroring.
                         (this adds a restraint in the Y direction
only at these nodes)

Free Edge                Suppresses the addition of a cable on the
this edge.


Mark pole tops           Makes lines that the polyline is snapped
to be modelled as
                         a pole set in concrete, with a point
stiffness at the top.
                         (more detailed description)

Mark link structure pair Makes a pair of polys to be modelled as a
set of links,
                         typically to allow a top ring to move
                         (more detailed description)

Simulate beam            Makes a set of links along the polyline
```

```
that are
                                                    triangulated to
act like a beam.
```

Lists

During the FEA model build colours can be used to define Lists.

MLists   (lists of Membrane elements)   are defined by coloured meshes
CLists   (lists of Cable or Links)   are defined by coloured lines
PLists   (Lists of Points or Nodes) are defined by coloured 3Dpolys

MLists can be used to specify different material properties to some meshes, or
specify a load on them.
CLists can be used to specify different material properties to some links, which could
be guys, poles, or cable sections.
PLists can be used to identify components such as cone tent top ring, to find the total
reaction on the component.

The colour definitions for list are described in the [Lists section of this help file.](#)
Note also that the colours can be overridden by using layer names. Again this is
described fully in the Lists section.
The lists generated during the FEA model build can be checked and edited on the
Lists tab later.
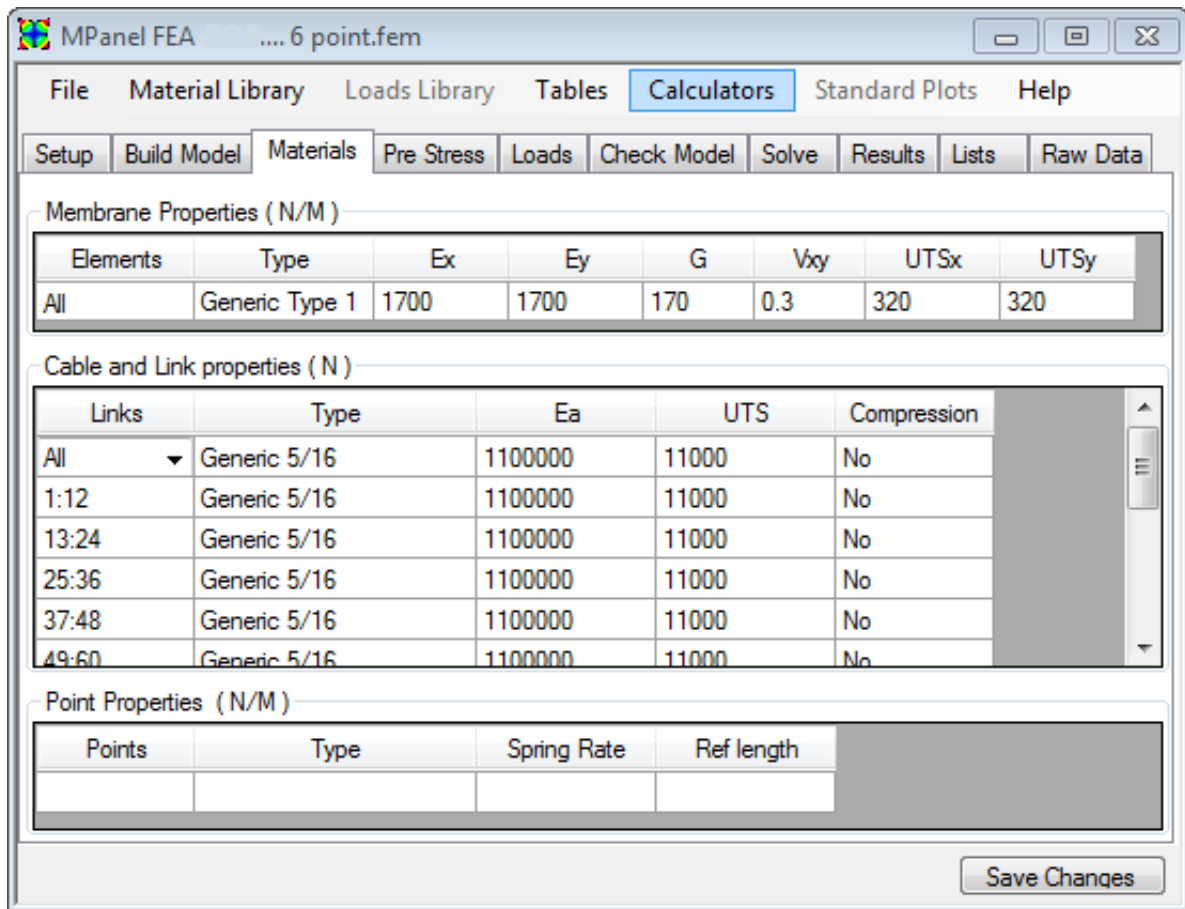
Material Properties

The default values used when a new model is built are initially a Type I pvc coated polyester membrane, and 8mm (5/16") diameter cables. (The default can be changed by the user)

For basic models, the same Membrane and Cable properties are used for the whole model.

The membrane or cable properties can be changed by hand, or by loading in a new set of values from the Materials library.

For more complicated models, different membrane or cable properties can be specified for different elements or cable links, on additional rows in the table. See "Additional Table Entries" at the end of this page.

After making any changes to the values in the form, you have to click on the "Save Changes" button to write them to the model file.

- **Membrane Properties**

Type
This is a short name to remind the user what material is being used. This name is stored, but not used by the program.

Ex   Ey
The Ex and Ey are the tangential values of the stiffness obtained from test data at the expected pre-stress. They may be previously known, or obtained from manufacturers data. If the manufacturer supplies the data in the form of a graph, the E values can be obtained using this method, or using the Material properties calculator. Note that the E is quoted as force per unit length, i.e. the fabric thickness is ignored. Usually Ex is equivalent to E warp on the fabric.

This amounts to the fabrics non linear stretch characteristics being approximated by a stretch that is linear around a chosen point, and that chosen point is the pre-stress.

G
The G value is the shear stiffness of the membrane. This figure is hard to obtain. If the material is isotropic, like a Mylar film, the G value can be calculated by the program. If the material was like a loosely tied net, then G would be zero. A value of E/10 has been used as the default, and represents a woven fabric with an infill coating.

In some situations it is useful to model with an isotropic material, even when the fabric is not isotropic. For instance, when trying to determine the principle stress direction, or trying to get some results for a model that is difficult to solve. A calculator is provided that will give the G value needed to make the material isotropic.

Vxy
Vxy is the Poisson ratio. In a film membrane, the values would be close to 0.3. In a fabric this can be understood as the crimp interchange. The value can be estimated from biaxial tests. Values between zero and 0.5 are suitable. Technically Vxy is the strain ratio of the contraction in X caused by a stretching strain in Y. (Vyx is calculated by the program).

UTSx   UTSy
If the ultimate tensile stress of the fabric is supplied, then a fabric reserve factor (safety factor) can be plotted from the results. These figures are not used during the

analysis, but are referenced in the Results plot of membrane reserve factor.

- **Cable Properties**

Type
This is a short name to remind the user what cable is being used. This name is stored, but not used by the program.

Ea
Ea is the cable effective stiffness. This is the cable E times it's area. Practically it is the tension that would double the cable's length, if it didn't break. A calculator is provided to give this figure from the cable's effective E and it's diameter.

UTS
If the ultimate tensile strength of the cable is supplied, then a cable reserve factor (safety factor) can be plotted from the results. These figures are not used during the analysis, but are referenced in the Results plot of cable reserve factor.

Compression
A tick box to specify if the link can accept compression. If the link is modelling a cable segment then it can't. If it's modelling a pole then tick this box.

- **Point properties**

(Note: point properties are not often used. Their main use is to model pole tops where the pole is set in concrete.They act as a zero length spring, trying to return the point to it's original position)

Type
This is a short name to remind the user what the point is being used for. This name is stored, but not used by the program.

Spring Rate

Spring rate is the force, returning the point toward it's original position per unit of displacement. A [calculator](#) is provided to give this figure from the pole dimensions.

Ref Length

The reference length is the length of the pole that the spring is being attached to. If this value is available, and the pole top is obtained from the library, this value will be used to calculate the spring rate.

Pre Stress
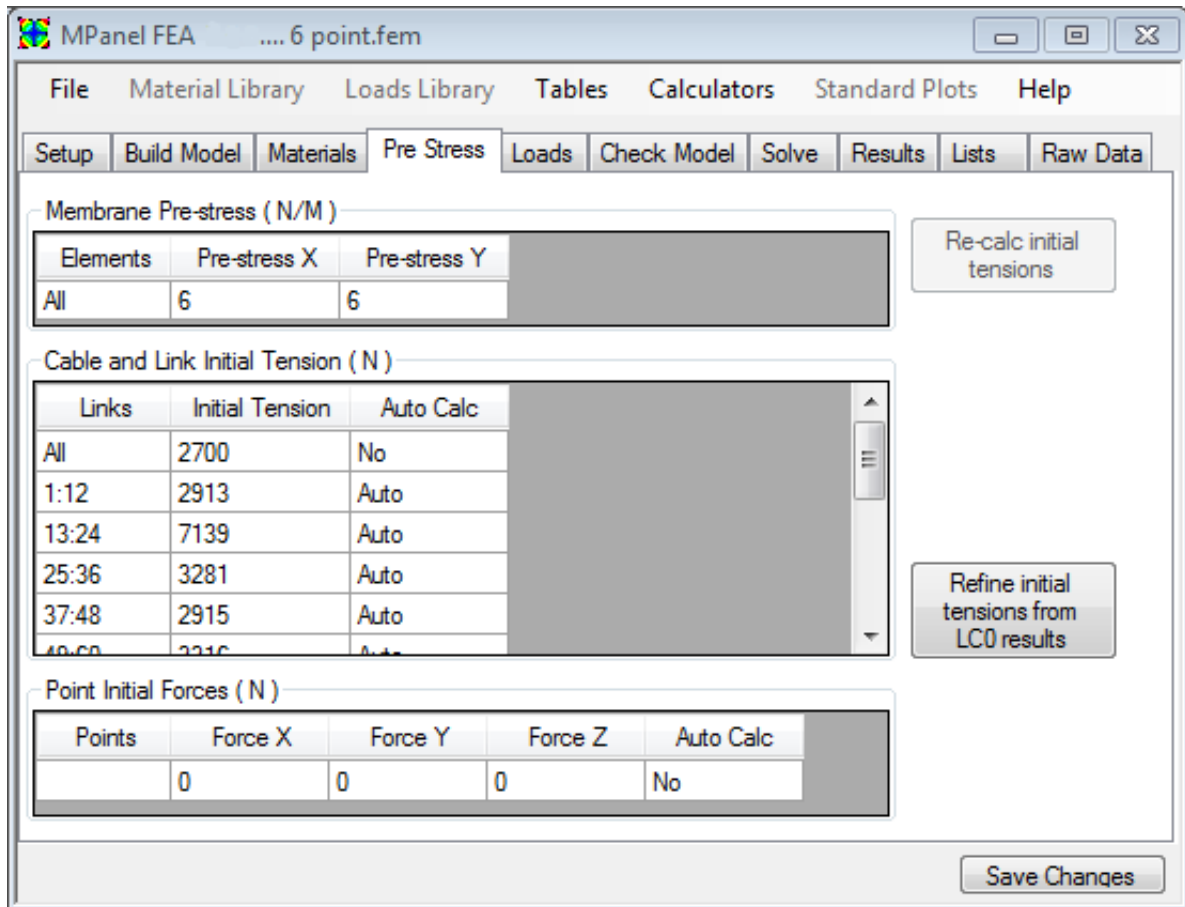
See also:
Using MPanel FEA tables
Using MPanel FEA database libraries
Using Mpanel FEA calculators

It is important to start an FEA analysis of a tensile structure in the "as built" condition, ie with the membrane stresses, cable and guy tensions, that exist in the structure after it has been installed. (If a reasonable initial stress and tension isn't used then the FEA solution may not be valid)
But this gives a problem, as these stresses and tensions are what we wanted the FEA to produce.

So we have a two step process:

a)   Using simple analytic methods, estimate the initial stresses and tension, and enter them in the Pre stress page.

b)   Solve for load case zero (pre stress only) to refine the actual stresses and tensions.

MPanel FEA will attempt to calculate the pre stress conditions based on simple analysis as follows:

Membrane Pre-stress X and Pre-stress Y

These are the values that determine the pre stress for the structure.
By default they are stress values that would be produced by a 1% stretch in the membrane in X and Y directions.
Often these values are higher than common use, and should be reduced to the design membrane pre stress.

Cable and Link initial tension

The cable initial tension for edge cables is automatically calculated by the program as:

Cable Tension = Edge Radius * Membrane Stress (from the first line in the Membrane properties table)

The cable tension is further calculated for links that are connected to edge cables, such as corner links.

The initial tension value is recalculated during "Save Changes" or when the "Re-calc initial tensions" button is clicked.

The automatically calculated cables have "Auto" specified in the table. This can be changed to "No" to override the automatic tension calculation, or to "Auto * X" to modify the calculated tension by the factor X.   This can be used in situations where you believe the initial tension to be different from that calculated by the program.

If the cable and link structure is too complicated for the program to solve analytically (AutoCal = No) then the user has to supply a reasonable initial tension value.

It is possible specify a reasonable initial tension value, then solve for load case 0, then use the solved tension to update the initial tensions. Specify AutoCalc = Refine for the individual cables, and then click the "Refine initial tension from LC0 results" button. Repeating this process several times will usually obtain initial tensions that result in no significant model movement when solving for load case 0.

Point Initial forces.

(Note: Point Initial forces are not often used. Their main use is to model pole tops where the pole is set in concrete.)

When a poles are set in concrete, and a membrane is tensioned between the pole tops, the "as built" initial condition is that the poles are bent inwards, with a resulting outward force on each pole top.

The initial pole top force is automatically calculated by the program if the pole top structure is reasonable simple. If it is not calculated (Auto = No) then the user has to supply a reasonable initial force value.

External Loads

Load Cases
The loads can be specified for up to twelve load cases. You might specify a couple of wind load cases and a couple of snow load cases. The unloaded case, with just pre stress, is automatically set as load case zero.

The element loadings are specified through a grid. On basic model the elements are set to "All" and just a single line of the grid is used to specify the loading. To specify loadings that vary across the model, you usually first set up some lists.

Load cases can be edited by hand, or downloaded from the loads library.

A description of the load case can be added.

There are three loading tables:

```
Membrane loads     specify membrane loads such as wind,
snow and self weight

Point loads        specify loads that refer to points,
such as point forces,
                   movements, restraints and constraints

Cable loads        specify cable loads such as self
weight
```

- **Membrane loads**

Pressure (wind uplift)
A pressure, representing the suction effect of the wind, can be applied. The value to use may be obtained from local building codes, or there is a calculator to give the pressure in terms of the wind speed and pressure coefficient. The pressure is applied so that a positive value gives an uplift on each mesh. A calculator is provided to give the pressure from a known wind speed and coefficient of pressure.

Self Weight
A weight, representing the fabric weight, or a constant weight per unit are like an ice film, can be applied. The weight is applied vertically down, in the -Z direction. The element load is calculated from the element area, regardless of the element orientation.

Snow load
This uses the local horizontal projected to calculate the element loading, which gives a better simulation of snow loads. A calculator is provided to give the load from a known wind speed and coefficient of pressure.

Load Factor
This is used to multiply the Pressure, Self Wheight and Snow loads by factor to allow for easy varying of the loads in complicated loading cases.

Example

In a simple loading situation, where the same pressure/weight is applied to the whole

model, the grid consists of one line, with the keyword "ALL" used in the elements column. So a loading with an uplift of 2 applied to the whole model, and no weight, would have a table entry of:

```
Elements     Pressure    Self Weight    Snow load    Load
Factor
All             2            0             0           1
```

To specify more complicated loading involving additional rows in the table, see the section Using Tables.

- **Point Loads**

These are only used when building advanced models, and can be completely ignored for basic model analysis.

Direct point loads are seldom applied in tensile structures, but are included for completeness.

Node movements can be applied to restraints to simulate the movement of the support structure. You could run the FEA, and obtain the reaction forces for a given load. These reaction forces could be used by a steelwork engineer to produce the steelwork displacements under load. these displacements could be entered in the table, and the FEA analysis re run, to refine the results..

Additional restraints can be useful in several situations. For example, if a multi bay barrel vault rests on steel arches, the suction load may be unsupported by the middle arches, whilst the snow load will be supported. So the additional restraints could be specified during just the snow load analysis.

Constraints are used in situations where the membrane is not firmly fixed to a restraint. Constraints can be set up to:

    Allow up movement... so a constraint stops movement downward
    Allow down movement... so a constraint stops movement upward
    Allowed specified movements, positive for upward movement and negative for downward movement.
    Sideways only.... so the node is only allowed to move in the local membrane plane
    Slip... allows the membrane to slip over a restraint by a limited amount.

The definition of upward and downward for each node is normal to the original

surface at that point. When a constraint has been invoked, nodal movement along the local normal is restrained and a nodal reaction will be generated, but movement in other planes is permitted. This simulates fabric slipping on the support surface.

- **Cable loads**

Self Weight
A weight, representing the cable or link weight, can be specified as the weight per unit length   The weight is applied vertically down, in the -Z direction.

Check Model
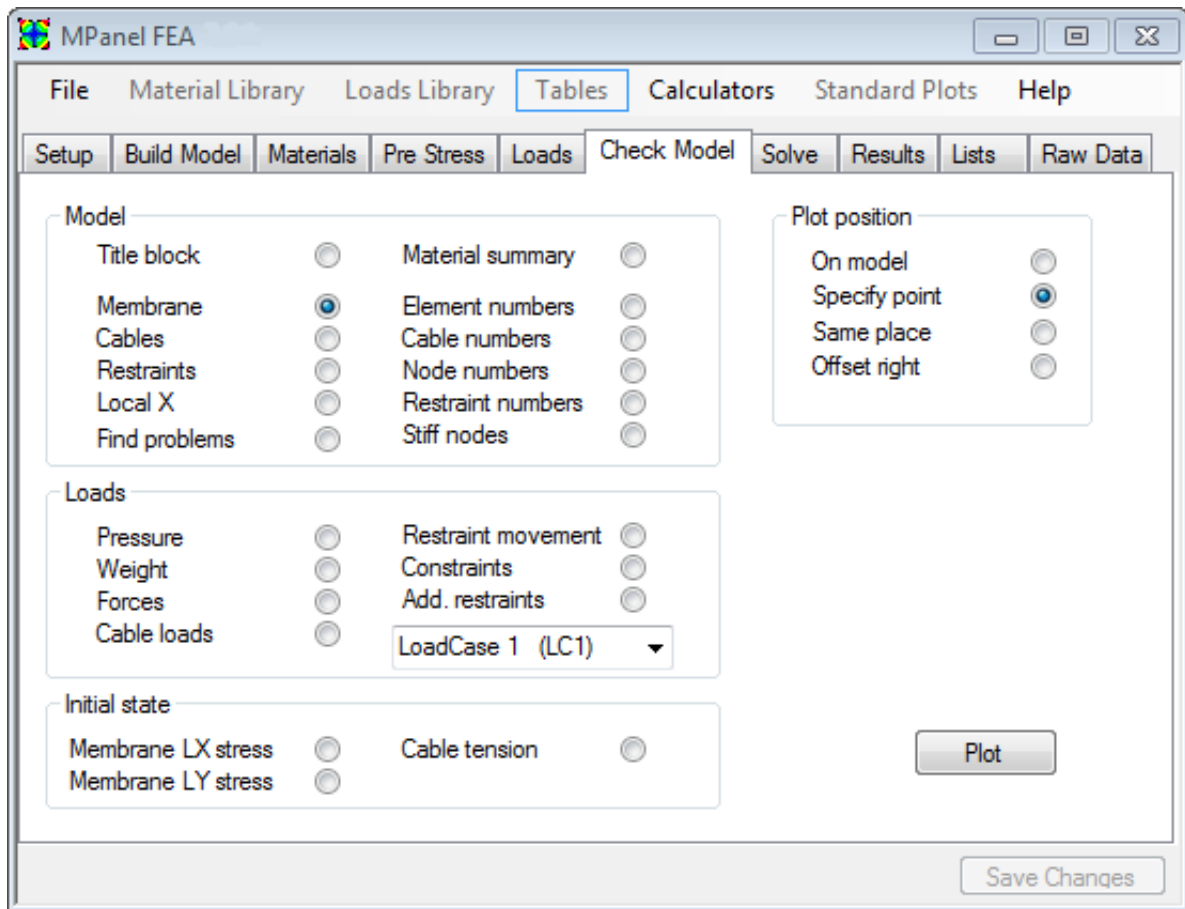
The model should be checked before solving, to ensure that you are analysing what you think you are.

The check model will create a variety of drawings to help check the model.

The drawings can be drawn:

- Exactly on top of the original model
- At a user specified point on the drawing
- At the same place as the previous drawing
- Offset right from the previous drawing.

The available drawings are:

- Title block   Model name, file name, and units used in a

large plot.

- Material summary   Details the material properties and pre stress used in the model.

- Membrane model   Shows the elements in the model.

- Cables   Shows the link in the model, either used as cables or guys. (See note below)

- Restraints   Shows the system fixed points as coloured circles. (See note below)

- Node numbers   Identifies the node numbers. Used to identify specific displacements in the output file

- Restraint Numbers Identifies just the nodes that are restraints

- Element numbers   Identifies the element numbers. Used to identify specific stresses in the output file.

- Cable Numbers   Identifies the link numbers, that make up the cables and guys.

- Local X   Shows the direction that is assigned as local X.

- Stiff nodes   Shows the points with stiffness added to them

And for each load case you can check:

- Pressures   Shows any pressure loads assigned in the model.

- Weight   Shows any loads in the direction of Gravity, such as self weight or snow.

- Forces   Shows any point or node forces in the model.

- Add. Restraints   Any additional restraints specified on the Loads pages

- Restraint Movement Any restraint movement specified on the Loads page

- Constraints   Any constraints specified on the Loads page

- Cable Loads   Shows any cable loads assigned in the model

- Find Problems   Shows any problems detected with this model

The Initial stress state can also be plotted for:

- Membrane LX Stress   Shows the membrane LX stress

- Membrane LY Stress   Shows the membrane LX stress

- Cable Tension   Shows the initial cable tension

Notes on Cables
The line showing a cable section or link is coloured as follows:

```
Cyan        Cables with the initial tension auto calculated
Magenta     Cables or links that the program could not calculate the
initial tension
```


Notes on Restraints
The circle showing a restraint is coloured as follows:

```
Red         Normal restraint, fixed in X, Y, and Z
Green       Partial Restraint, fixed in X
Blue        Partial Restraint, fixed in Y
Cyan        Partial Restraint, fixed in Z
Gray        Any other partial restraint
```


Notes on Find problems
Find problems draws out a model with at least the restraints and cables, drawn in black or white. Any problems are then coloured in the following manner:

```
Red circle      Duplicate node, can be caused by models with
duplicate meshes.
                (fix:  remove duplicate meshes from original
model)
```

Cyan circle   Nodes that are close together, but not coincident.
                       (fix: correct small errors in original model
                        so poly nodes and mesh nodes exactly match)

Magenta circle   Solitary nodes that not connected to the rest of
the model
                       (fix: possible model errors, or extra points
on a 3dpoly that aren't needed)

Cyan line   Duplicate line, will show half of expected link
tension
                       (fix: remove duplicate lines from original

model)

Red line   Zero length line, will prevent a solution.
                       (fix: remove zero length lines from original

model)

Cyan mesh element   Small element in the mesh, may cause slow
convergence
                       (fix: make a more even mesh in the original

model.

   may have to accept this with this triangular

sails)

Red mesh element   Collapsed element in the mesh, probably will not
solve.
                       (fix: rebuild with the meshing option
"Largest triangles" selected)

Solve
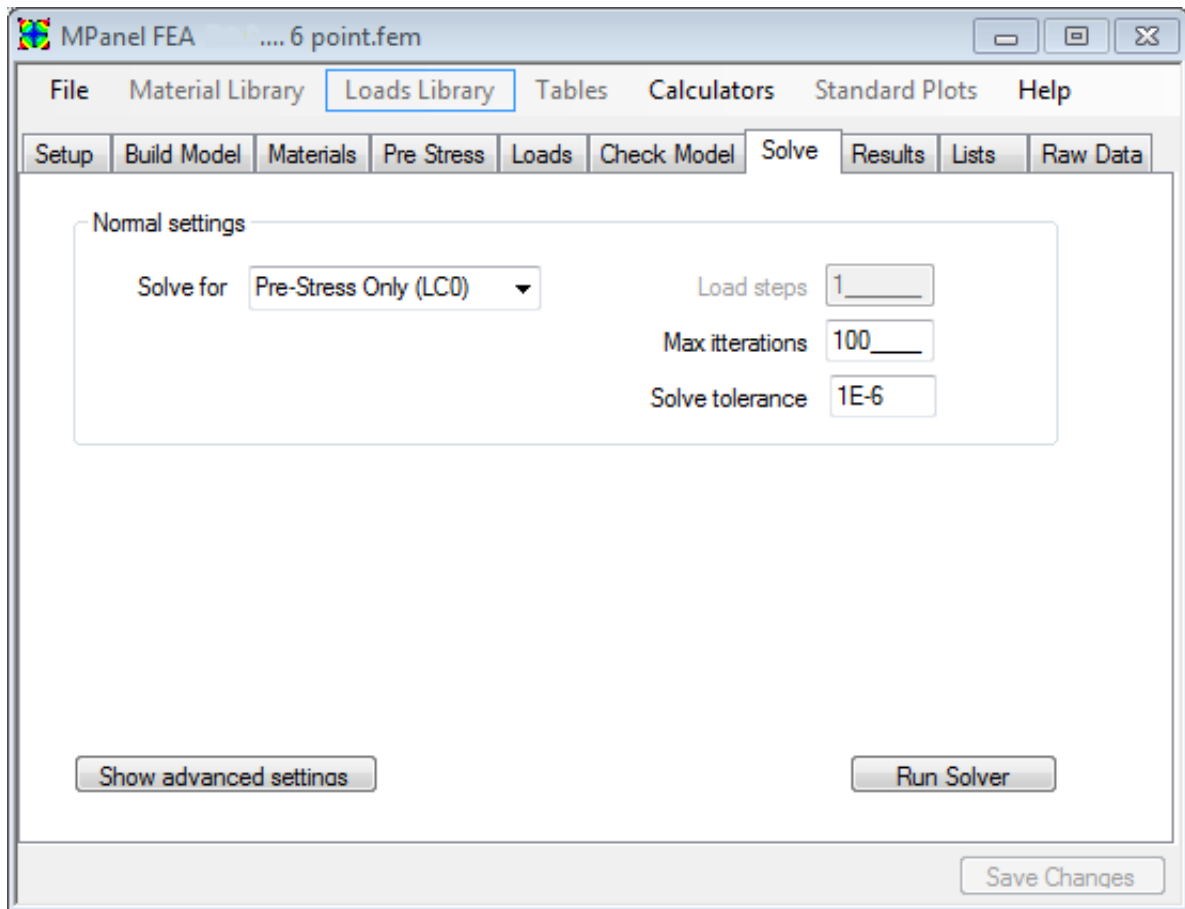
This starts the solver program , which calculates the displaced node positions and element stress for the required loading. The progress of the solution is reported in a separate window. The model geometry, and stresses are updated on each iteration, this allows for a geometrically non linear solution, with large displacements being allowable.

Choose which load case you want to solve for. Load Case 0 always exists, and is the pre stress only situation. The other load cases can be defined on the Loads tab.
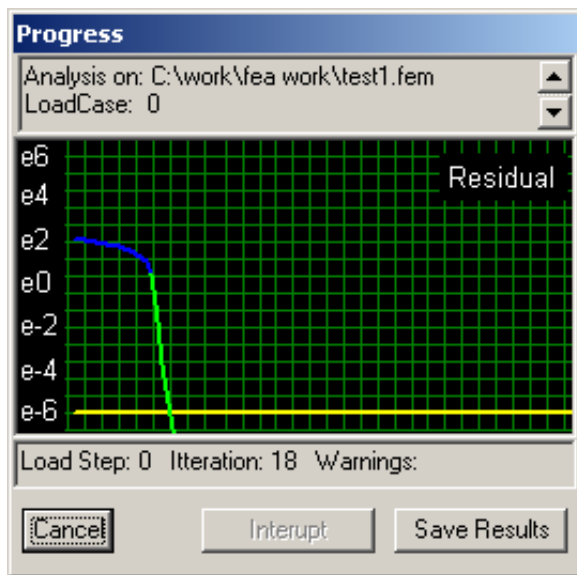
If the model has reasonable material property values and low loads, the solution will converge quickly, in less than 10 iterations, as shown above. If the model involves substantial shape change and deformation then the solution will converge slower, and may need 250 or more iterations to solve.

The residual is graphed in numbers like 1E-3, which means 0.001 The default tolerance for a solution is 1E-6

Typically residuals below 1E-2 will give numerical answers that are as accurate as 1E-6. Allowing the solution to continue to 1E-6 is just a confirmation that the model is converged and solved correctly. On hard to solve models, that go into a low level oscillation, a solution better than 1E-2 should be OK.

If the model has property values or loads that do not lead to a solution then the residuals may increase, or hang at some value. The results obtained in this case are not valid.

When the solver has finished, click on "Save Results" to write the solution to the output file. Or "Cancel" to discard the results.

In difficult to solve cases it may help to specify that the load is applied in a series of smaller steps.

Damping.

Two methods of damping are used to prevent models from going unstable during the solve.

Note: When damping is used, the convergence becomes much slower, so be sure to check there are enough iterations to get the residual low enough for a valid solution.

- If an element shows a large displacement in it's position, compared to the element size, all of the displacements are reduced (damped), and the graph line is coloured Blue.

- If an iteration results in an increase in the residual, then all the displacement are reduced (damped), and the graph line is coloured Cyan.

Warnings

- If an element has zero or negative stress in its lowest stress direction, it will be modified to be able to carry stress at right angle to that direction. This allows the analysis to continue with slack fabric in one direction, with the loads being carried by hoop tension in the other axis. This reported in the solve window with the warning "Z". (The wrinkled fabric can be seen in a plot of the Second Principle Stress, where it will be coloured White).

- If an element has zero or negative stress in all directions, the element will be modified so that it cannot carry stress. This reported in the solve window with the warning "ZZ". (The slack fabric can be seen in a plot of the Principle Stress, where it will be coloured White).

- If a node is surrounded by elements that have zero or negative stress in all directions, it will be skipped in the calculation, and the warning "S" is added to the residual report. If this occurs in the final step, be aware that the position of, and stresses around, a node that was skipped will not be correct.

- If a link or cable segment tries to go into compression, and it has not been defined as being able to carry compression, then the link will be set to zero stress and zero stiffness, This is reported in the solve window with the warning "NB", for "negative bar"

Normal Load Cases

There are three cases that you normally solve for:

Static Case
This has the membrane and cable pre-stress only. The solution will represent the redistribution of stresses that occurs on erection.This is normally called Load Case

0   It may be surprising that there is any change in the stresses at all, but they are there because:

In a normal MPanel model, the membrane stress increases as the mesh lines get closer together… but we may still use a simple compensation that applies the same initial pre-stress to the whole structure.

The cable pre-stress may be different from the pre-stress assumed in the MPanel model

The 3D membrane shape may not be exactly correct, as MPanel uses several approximations in its shape finding algorithm.

(Although the fabric weight should be included in the static case, it has such a small effect that it can usual be ignored without changing the stress by a noticeable amount. If you want to consider it, then you can add it in as an external load. For Ferrari 702 is about 0.75 Kg / M^2)

Uplift Case
This has an even pressure applied to the structure to produce an uplift. This is used to represent the suction effect of the wind.

Down force Case
This has a vertical load applied evenly over the structure. This is used to represent a snow load, a heavy rain, or self weight.

Advanced Settings

You can specify a load case to use as the starting point for solving.
So if we had solved Load case 3 for a pressure of 50, and needed a solution for load case 4 with a pressure of 75, you could save some computation time by using load case 3 as the starting point. This is not usually needed.

You can also specify the following advanced settings:

Use Displacement damping. Used to limit the displacement of the model on each iteration to a fraction of the smallest element length in the model. The fraction is set in the "Damping Factor" box.

Use Residual damping. Used to limit the displacement of the model on each iteration if the displaced shape increases the residual.

Use Accumulate Stresses. This allows the stresses to be accumulated into each element on each iteration, and the new shape of the element used going into the next iteration. This has the benefit of keeping the element strain on each iteration small,

but it can lead to instability in some situations.

Use Simulate wrinkled fabric. This allows the program to temporarily alter the membrane properties in elements with zero stress. This will usually speed up the solve when there is zero stress, but it can lead to instability in some situations.

Constraint slip factor. (Default = 1), this sets how much of the in plane movement is applied to the model each iteration. The in plane movement simulates fabric slip on a constraint. Setting a value lower than one may help in getting a solution in a difficult case. Setting a value of zero means that no slip is allowed after the constraint has been activated.

Changing settings during the solve

During the solve you can interrupt the process from the solve window. This allows you to modify some of the solver parameters and then continue. The parameters that you can change are:

- Enable / Disable the displacement damping
- Enable / Disable the residual damping
- Change the Maximum number of iterations
- Change the Solve tolerance.
- Save the current solution, even though it has not met the solve tolerance
- When a number of load steps are specified, skip to the next load step.
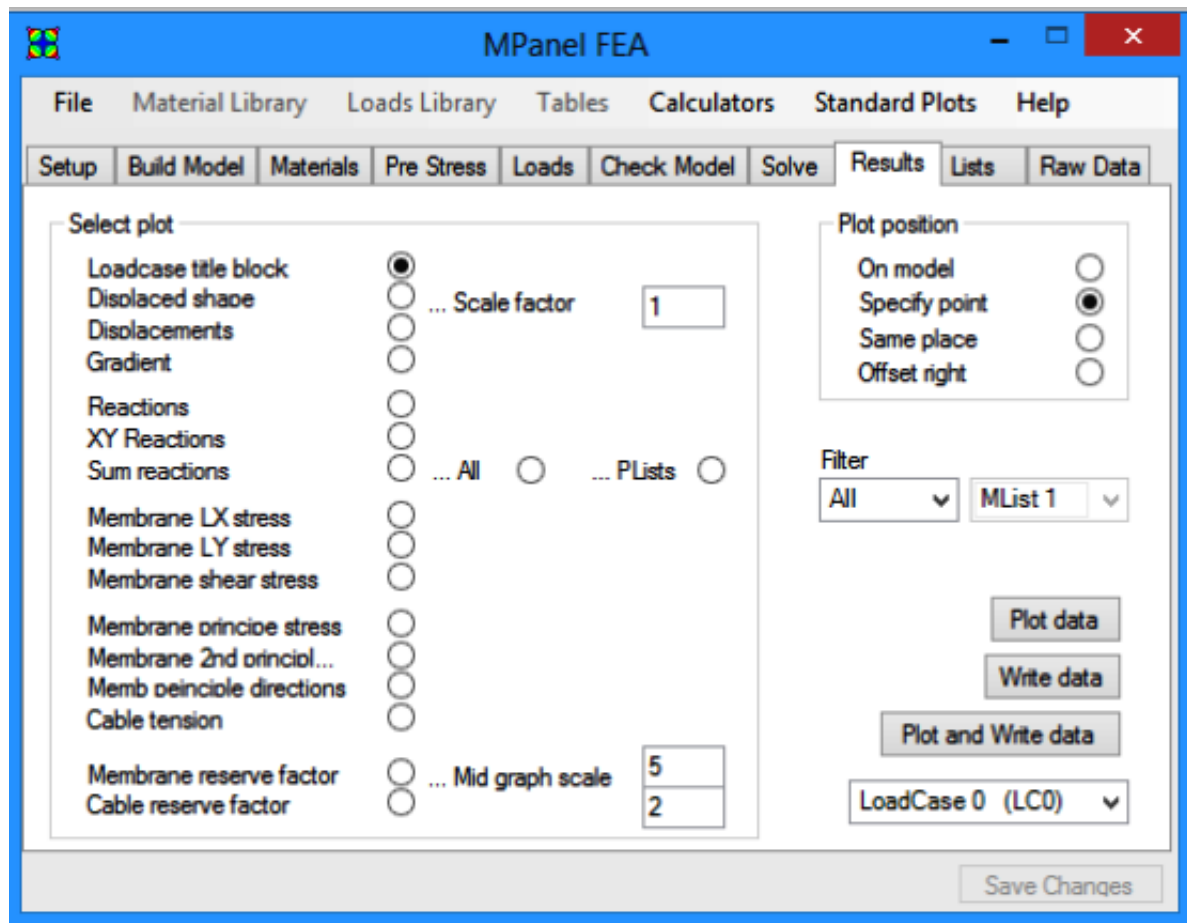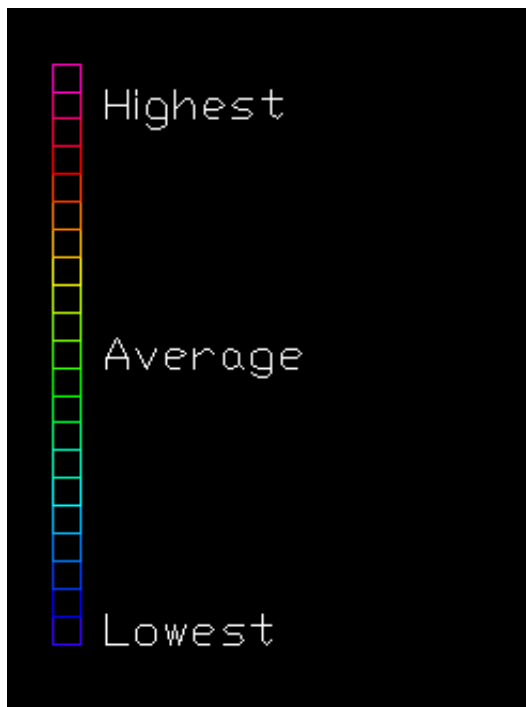
Draw Results

Draw results will create a variety of drawings and text printouts to show the results of the analyses.

It can either draw a scaled graph, or write in the actual values for the selected data, or both.

The coloured plot is scaled and colour coded like this:

(With a special case of white indicating slack fabric in the principle stress plots.)

It can create the following output drawings:

- **Load case title block**   a text plot with the load case
  number and details of the applied loads

- **Displaced Shape**   solved position of the mesh
  and links (notes below)

- **Displacements**   plus or minus displacement
  form the original shape

- **Gradient**   local gradient direction and
  value (notes below)

- **Reactions**   reactions at each of the fixed
  points.

- **XY Reactions**   X and Y reactions at each of
  the fixed points (notes below)

- **Sum Reactions**   sum of the reactions for a set
  of points or the complete model

- **Membrane LX Stress**   stress in the localX

direction

- **Membrane LY Stress**   stress in the localY
  direction

- **Membrane Shear Stress**   shear stress between the
  localX and locally directions

- **Membrane Principle Stress**   highest local stress,
  irrespective of direction. (notes below)

- **Membrane Second Principle Stress** lowest local stress,
  irrespective of direction. (notes below)

- **Membrane Principle Direction**   indicates the principle
  stress direction. (notes below)

- **Cable Tension**   tension in the cables or
  guys.

- **Membrane Reserve Factor**   Plots the reserve factor
  (safety factor) for the membrane (notes below)

- **Cable Reserve Factor**   Plots the reserve factor
  (safety factor) for the cable (notes below)

General Notes on plotting results

Results can be filtered so just part of the model results are shown. This can help with seeing more detail on the coloured plots in part of the model.

Standard reports can be defined that contain a series of plots. These are useful when creating a report to document the load response. More information on [standard reports is here](#).

Further Notes on specific plots.

- Displaced Shape
The displacement from the original values can be scaled. A scale of 1 gives "true" results, a larger scale factor may help in visualising the mesh movement. An original, un-displaced copy of the model is also drawn out for reference, coloured Green'

- Gradient

Draws a arrow like symbol showing the local direction water would flow on the displaced surface, colour coded for the gradient. A group of arrows all pointing in wards indicates a ponding area.

- XY Reactions

This is the overturning reaction for a fixed point representing the pole top, or the overturning reaction at the top of a retaining wall.

- Sum Reactions

This will sum the reactions for either the whole model, or for sets of points defined by PLists. The reaction direction is shown by a line starting at a point that roughly represents the point of action of the force. The point of action is prone to round off errors when the total reaction is small, so should be treated accordingly.

- Membrane Principle Stress

Useful when the main stress varies in direction from the warp or weft direction, to get a good picture of the maximum fabric stress. Is shown as white if the fabric is totally slack in all directions in that elements

- Membrane Second Principle Stress

This can be used to show elements in the mesh that are slack, ie zero stress in this direction. Typically this will cause wrinkles in the fabric parallel to the principle stress direction,

- Membrane Principle Direction

This can be used to interpret the Principle Stress, used to visualise the stress flow, and can be used to determine the best direction to run the seams.

- Membrane Reserve Factor

This plots out the membrane reserve factor (safety factor) that is calculated from the membrane $UTSx / Sx$ or $UTSy / Sy$ whichever is lower. In use you set the Mid Graph Scale to the reserve factor that you want to achieve... for example 5. Then in the plotted results, membrane elements that are OK (ie at a reserve factor greater than 5) will be plotted out Green through to Blue. Membrane elements that fail to meet the required reserve factor (ie less than 5) will be plotted out toward the Red and Magenta colours.

- Cable Reserve Factor

This plots out the cable reserve factor (safety factor) that is calculated from the cable UTS / cable Tension.. In use you set the Mid Graph Scale to the reserve factor that you want to achieve... for example 2. Then in the plotted results, cable links that are OK (ie at a reserve factor greater than 2) will be plotted out Green through to Blue. Cable links that fail to meet the required reserve factor (ie less than 2) will be plotted

out toward the Red and Magenta colours

Lists
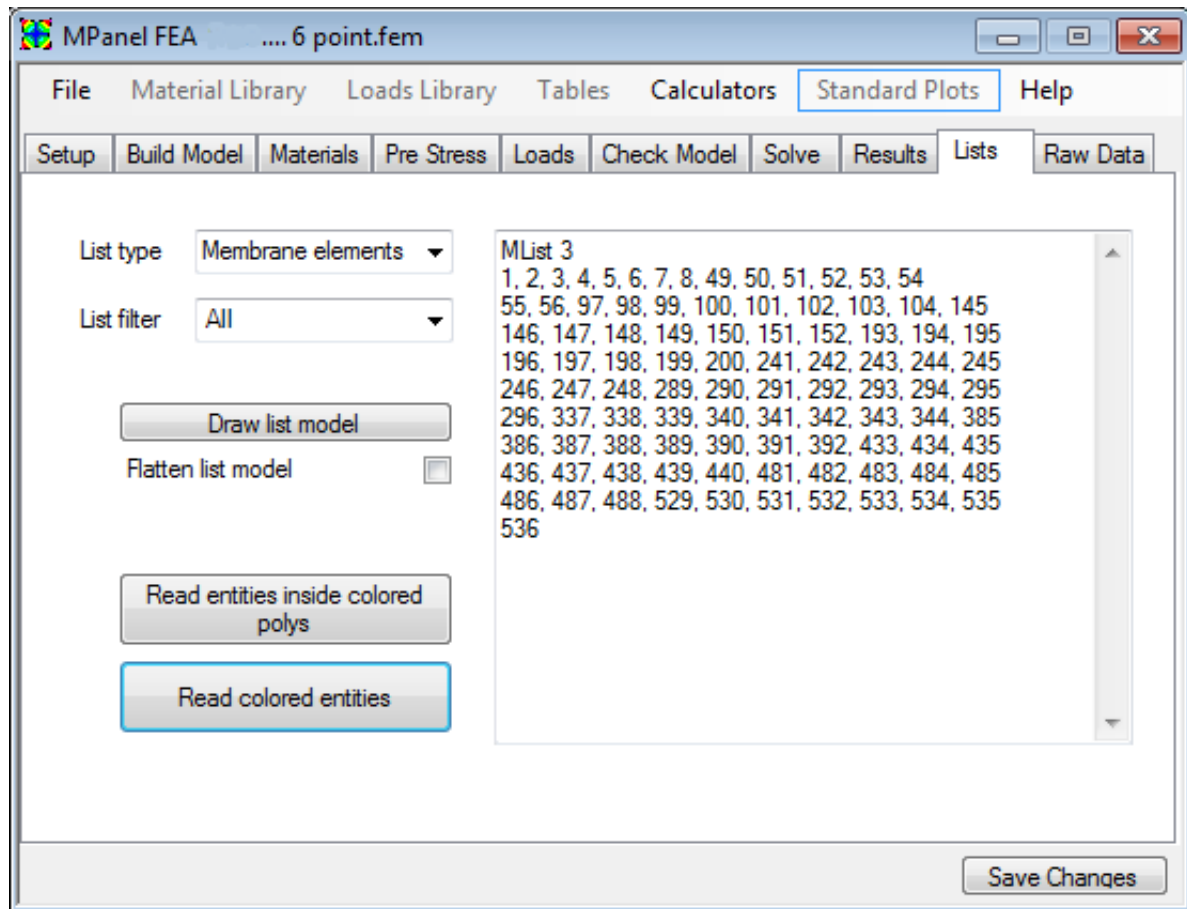
Lists allow a group of elements, links or points to be defined, and referred to as a single unit.

Thy can be used to

defining loading regions for pressure or weight loading.

define parts of the membrane built with a different material or

prestress

define cables with a different stiffness or prestress

define links used as struts

define points to accept point loading or accept additional restraints.

define points to sum the reactions for.

The lists can be created in this tab, or they can be created during the model build by using coloured entities.

There are two methods to create lists on this tab

Using coloured entities

Start by drawing out the element, link or point model, using the "Draw List Model" button. This will draw out a model, with numbers, in an un-grouped drawing. By having an un-grouped drawing, it is easier to select just part of the model. If the model contains parts which are substantially vertical, it can help to "flatten" the List image.

Then colour parts of the model, according to the which list you want them to be in. So List 1 would be coloured Red, List 2 would be Yellow, etc. (note that the colour must directly defined, ie not ByLayer or ByBlock). A description of the list colours is at the end of this help topic.

(Alternatively instead of colouring parts of the model, layer names can be used instead. So membrane elements on Layer "MList 3" will be assigned to MList 3 irrespective of their colour)

Then select the coloured drawing, and use the "Read coloured entities" button to create the comma separated list of numbers.

To check the list, you can use the "Draw List Model" button again. And then perhaps change some of the list regions, and read them in again.

Once you have the lists correct, save them to file using the "Save Changes" button.

Note: usually you can work on all the lists at the same time. If some lists overlap, you will have to work on them separately. You do this by selecting the list you want to work on in the List Options.

Using coloured polylines

Start by drawing out the element, link or point model, using the "Draw List Model" button. If the model contains parts which are substantially vertical, it can help to "flatten" the List image.

Draw a coloured polyline (2d or 3d) around the area that you want to colour, when viewed from above.

(Alternatively instead of colouring the polylines, layer names can be used instead. So polyline on Layer "MList 3" will assign all elements inside to MList 3 irrespective of

the polyline colour)

Then select the list model, and the coloured polyline, and click on the "Read entities inside coloured polyline" button.

If the polyline was coloured Red, all the entities inside the polyline would be written to List 1.

Multiple lists can be defined at one reading by using several coloured polys, which will often look like contours around a hill.

But note the following:
The Lists are assigned in the numerical order, List 1, then List 2, then List 3 etc
So the polys should be coloured in the same order, with the Red contour at the "top of the hill", the Yellow next , etc

An advantage of using coloured polys to define lists is that if the model has to be rebuilt, it is possible that the new model will not use the same entity numbering. By having an overlay of coloured polys the lists can be easily rebuilt.

- List colour definitions

The list number (MList 1, CList 2, etc) is defined by the colour, and 24 list numbers are supported. The colour definitions are:

List number	Colour	AutoCAD ACI	Rhino RGB		
1	Red	1	255	0	0
2	Yellow	2	255	255	0
3	Green	3	0	255	0
4	Cyan	4	0	255	255
5	Blue	5	0	0	255
6	Magenta	6	255	0	255
7	Lt Red	11	255	127	127
8	Lt Yellow	51	255	255	127
9	Lt Green	91	127	255	127
10	Lt Cyan	131	127	255	255
11	Lt Blue	171	127	127	255
12	Lt Magenta	211	255	127	255
13	Dk Red	12	204	0	0
14	Dk Yellow	52	204	204	0
15	Dk Green	92	0	204	0
16	Dk Cyan	132	0	204	204

17	Dk Blue	172	0	0	204
18	Dk Magenta	212	204	0	204
19	Vlt Red	13	204	102	102
20	Vlt Yellow	53	204	204	102
21	Vlt Green	93	102	204	102
22	Vlt Cyan	133	102	204	204
23	Vlt Blue	173	102	102	204
24	Vlt Magenta	213	204	102	204

Note that the colour must directly defined, ie not ByLayer or ByBlock. Any colours which are not exactly the ACI number shown above will be ignored.

Raw Data
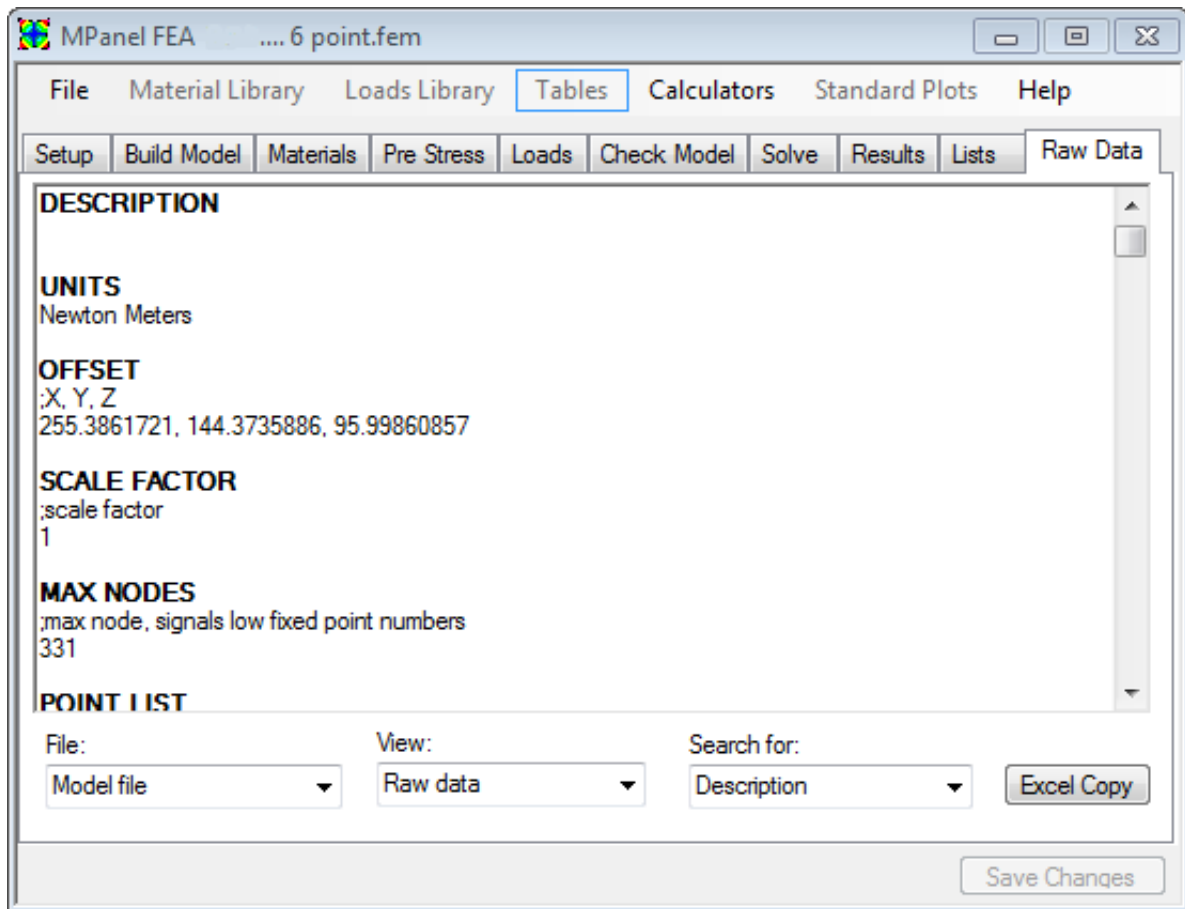
The model file any any solved output filescan be vied in a text window. They consist of the sections, which can be selected using the "Search for" box.

An "Export for Excel" button formats the data in an Excel friendly style, and copies the selected it to the windows clipboard. This can by pasted into excel, and will have the correct columns, etc. This has use for performing further calculations on the output data, allowing further post processing.

Instead of viewing all the raw data, you can specify the view as "Restraint List" or "Reaction List". This gives just the information needed to export the model nodes and forces on those nodes to another program, such as a steelwork analysis program. In addition the otput file reactions can ve vied organised into PList groups.

Model File

DESCRIPTION   text description of the models
UNITS   units used in model

OFFSET   xyz location of the first point in the model
SCALE any scale factor used in reading the drawing
POINT LIST xyz location of each point referenced to the first point
REGION LIST   the points making up each triangle, and RH or LH triangle
LINK LIST   the points making up each link
RESTRAINTS LIST   fixed points, with the freedom described in X,Y,Z

POINT LOADS (X)   forces applied to individual points. Only used for program testing
ELEMENT LOADS (X) forces applied to triangular regions.
PRESTRESS MEMBRANE (X) pre-stress applied to the membrane, in LX and LY
PRESTRESS CABLE (X) pre-stress applied to the cable
MEMBRANE PROPERTIES (X) material properties for membrane
CABLE PROPERTIES (X) stretch properties for the cable.

PRESTRESS CABLE ADJUSTMENT recalculates the cable prestress according to the catenary radius.
LISTS records the elements in each list.

(where X is the load case number for load cases 2 through 6)

Editing data by hand.

These entries can be altered by hand to define more advanced models. Edit them in the style of the existing settings. There must be at least one blank line between sections. To preserve any hand changes during the normal "Save Changes" button operation, preface them with the "@" symbol.

If the same data applies to many elements or links, it can be entered like   1:20, mydata   (Where mydata is to be set on elements or links 1 through 20.)

The data is read sequentially, so entries of   "ALL, mydata"   followed by   " 5, mydata2" will set mydata on all items except item 5, which will have mydata2 set.

Output File

DISPLACEMENTS   the xyz displacement of each node from it's original position
STRESSES the stresses in each element in the local XY coordinates
LINK TENSION the force in each link, or cable element
REACTIONS   the XYZ force that the model exerts on each fixed point.
SUM REACTIONS   the vector sum of the reactions, summed for the whole model, and any defined PLists.

SUM REACTION CENTER an point in space that the sum of reactions can be considered as acting through.
LAST STORED STRESSES used during solves starting from an existing load case
LAST STORED ELEMENT FACTORS used during solves starting from an existing load case

Advanced model building notes.

- **Resolve corner tensions**

Normally the program will resolve the corner forces and apply them to a single corner link. If the option to resolve corner tensions is selected then the program will try to resolve the corner forces and apply them to a set of links attached to the corner.

If the geometry at the corner is suitable then 3 links can be determined, and any links with a negative initial tension will be assigned as poles. If the geometry is locally flat then 2 links can be determined. The calculation for multiple links depends on the membrane having the same membrane stresses Sx and Sy, if this isn't the case then the calculations will only be a first approximation.

The links assigned as poles will use the default pole properties from the pole database.
The links assigned as guys will use the default cable properties from the cable database.

This algorithm may give an incorrect interpretation of a set of links in some models, so you can turn the option off. In particular poorly angled guys may be assigned a negative initial tension, which would lead to them being written into the links table incorrectly as poles.

- **Encastre poles**

Encastre poles are simulated with a pole that is pin joined at each end, and an additional node stiffness at the pole top. The node stiffness is specified as a spring rate, and is calculated to give the same force / deflection characteristics as an encastre pole in bending.

The value of the spring rate for simple tubular poles can be obtained from the Encastre pole calculator. It is very dependant on the pole length.

To correctly start a FEA analysis from the "as built" configuration, we also need to specify the force generated at the pole tops by tightening the cables during installation. This is specified in the Point Initial Forces table. If this isn't specified then the analysis for each load case will include an unintended slackening as the pole tops move inward to absorb the corner forces. The force vector at each pole top can be obtained by performing an initial fea analysis with the pole tops fixed, and entering those pole top reactions into the Point Initial Forces table.

The program will do all of this automatically if the pole tops are identified with a 3dpoly, that has special status of "Mark Encastre Pole tops".
The pole links and pole top spring rate will use the default pole properties from the encastre pole database.
The initial forces will be calculated analytically and entered in the Point Initial Forces table.

Since the spring rate is very dependant on the pole length, automatic assignment of properties by the program uses the "Spring rate for unit length" value (if it is available) and then calculates the actual spring rate from the pole length. When this is done the pole length is noted in the Point Initial Forces table as the Reference length.

- **Link structure pairs**

A conic's top ring is usually modelled by fixed points, and this is appropriate when the top ring is fixed in position by stiff members, such as steel framework or external guys. When the top ring is attached to a pole that is pin joined to ground then loading may cause the top ring to move sideways. This can be simulated by building a "inverted tepee" arrangement of links that are only fixed at the ground. These links structures can be built by hand., or automatically by the program.

To make the links automatically, use an Mpanel model with the top ring defined by a single 3dpoly. Draw in another 3dpoly to represent the pole, with just 2 nodes (3 for Rhino users). Select the pole poly and the top ring poly and mark them with special status as a Link Structure Pair". You will also need to provide a 3dpoly to fix the bottom of the pole.

During the FEA model build the 3dpolys for the pole and top ring will be replaced with an "inverted tepee" of links. The link stiffness will be the default cable stiffness, with Allow compression = Yes. (We do not use the default pole stiffness, as the pole is represented by numerous links acting together)

There can be any number of link structure pairs, the program sorts out which pole goes with which top ring by proximity.

- **Simulate beams**

A beam can be simulated by a truss made from pin jointed links. The truss triangulation are based on the polyline nodes, and the simulation is most accurate with evenly spaced nodes. The beam is created with pin jointed ends, to simulate an

encastre beam the end two nodes should be fixed.

The stiffness of the links can be set to achieve a defined EI value for the beam. The link stiffness is automatically read during the FEA model build, but can also be calculated with the EI calculator supplied. The truss deflection under load will be in reasonable agreement with a real beam's deflection, but should be checked if the deflection is critical.

To ensure modelling stability the links are pre tensioned, and small stabilising forces added to the outer nodes to balance the forces.

The beams links are automatically assigned to CList 24, which helps with filtering the simulation out during report drawing.

Using Tables
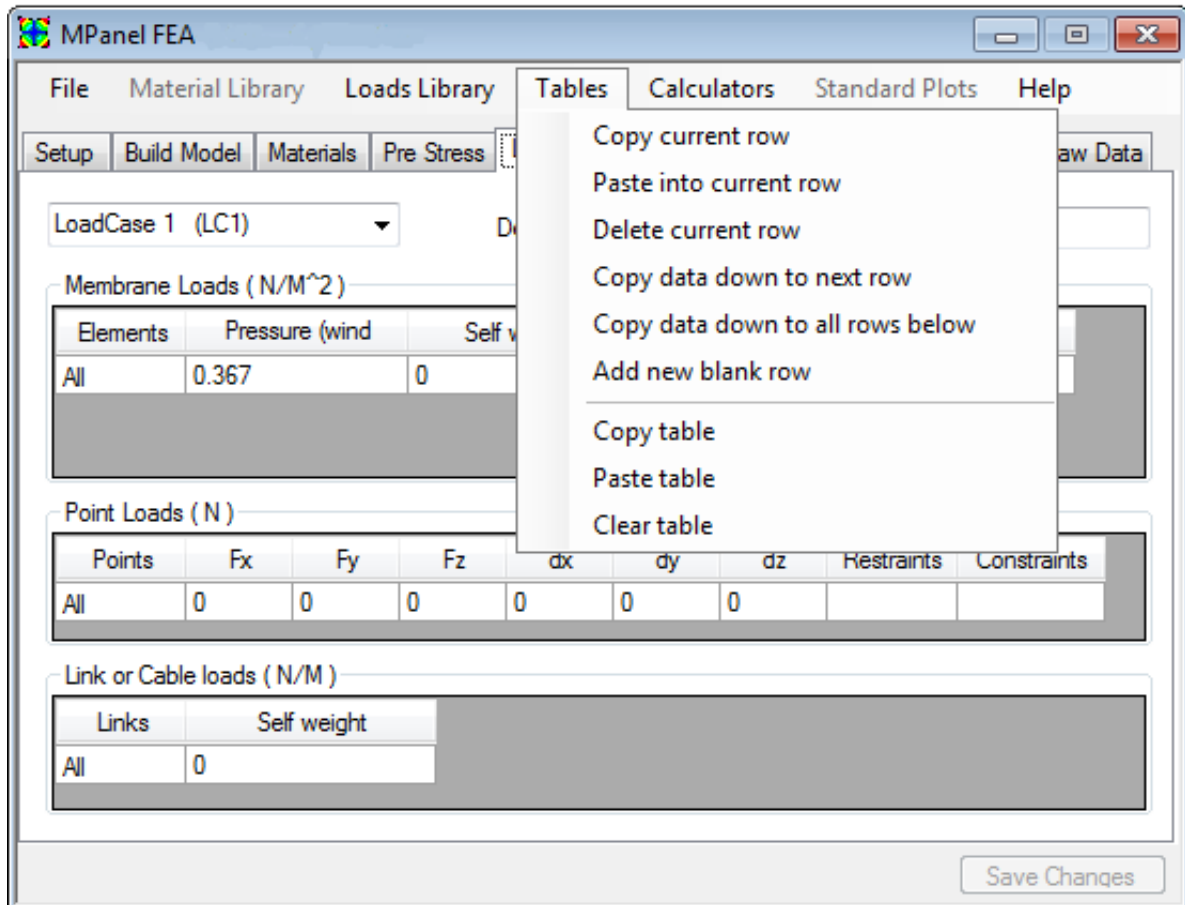
The tables can be sized vertically by hovering the mouse over the gap between the tables, and selecting the sizing bar.

The table columns can be sized horizontally by hovering the mouse on the column header, and selecting the sizing arrow.

The left column defines which elements, links or points the row refers to. Example entries are:

```
All              (all the elements)
3:7              (elements 3 to 7)
MList 3          (all the elements in MList 3)
```

The tables are read from the top downward, so a table entry for the membrane loads of:

Elements	Pressure	Self Weight	Snow load	Load Factor
All	120	0	0	1

```
3:7             60              0               0               1
```

puts an pressure of 120 on all elements, except for elements 3 through to 7, which have a pressure of 60

The table editing menu can be selected from the menu bar, or opened by right clicking on a table.

The tables editing functions are:

```
Copy current row                        copies the row
values in an Excel compatible format
Paste into current row                  pastes in the row
values
Delete current row                      deletes the current
row
Copy data down to next row              copies the row
values down, except for the Element numbers
Copy data down to next row              copies the row
values down, except for the Element numbers
Copy data down to all rows below    copies the row
values to all rows below
Add new blank row                       adds a new blank row
at the end of the table
-----------------
Copy table                              copies the table
values in an Excel compatible format
Paste table                             pastes in the table
values
Clear table
```

Using Database libraries
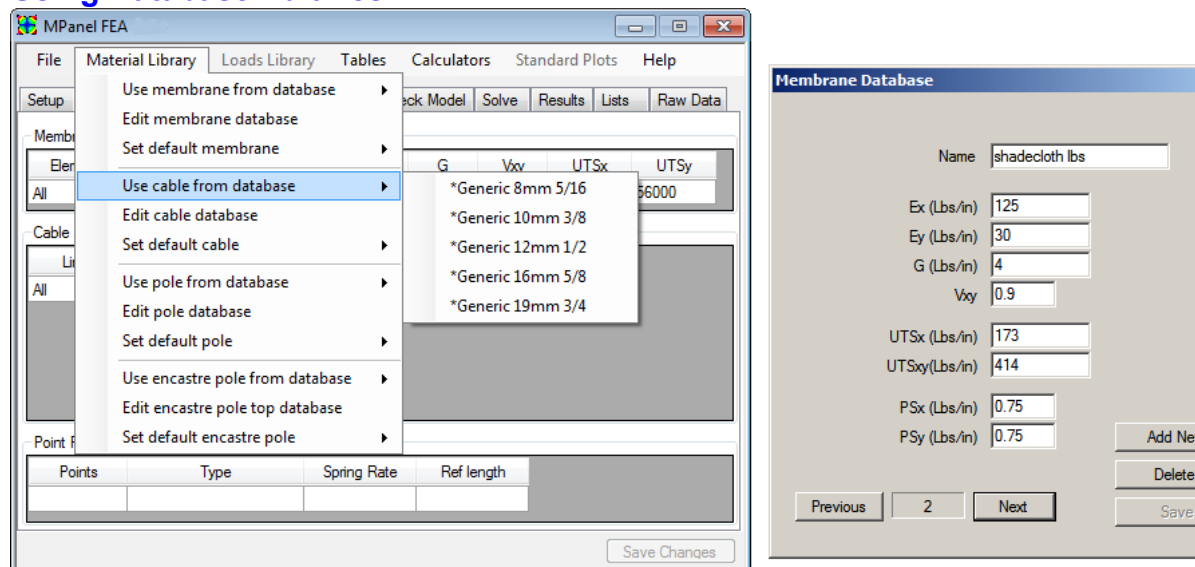
There are libraries for:
> Membrane properties
> Cable properties
> Pole properties
> Encastre pole simulation properties
> Loads.

Each library consists of :
> Standard entries, starting with *Generic
> User entries from the database.

To use a material, put the cursor in the table row where you want the data, and then click on the menu item like "Use membrane from database" to select the required material.

To set a default material, to be used on the next FEA model build, click on the menu item like "Set default membrane" to select the required material.

To edit the database entries, click on the menu item like "Edit membrane database" to open the editing form.
Then you can add, change, or delete materials.

Note: you can only add properties in the units you are working in. . If you intend to work in two unit systems you should make two database entries, with distinctive names. If you try to use a set of properties in the wrong unit system the program will warn you.

Using Calculators

Several simple calculators are provided to help generate suitable values for material properties and loads. These are described below.:

- **Membrane Properties from graph data**

Given a stress/strain graph for a material, this will calculate the Ex and Ey from the graph slopes.

If only a single graph at 1:1 stress loading is available you will have to estimate of Vxy, often a value of 0.3 is suitable.
The value of G cannot be determined for a simple stretch graph , but is estimated as 10% of the average E value.

If stress/strain graphs are available at different stress ratios, such as 1:1 and 2:1 , this data can be used to refine the estimate for Vxy. If the material behaves like a linear orthotropic material a Vxy value will be calculated with a low value for the graph slope miss match percentage, perhaps around 10%.   Much higher miss matches indicate that the material does not behave like a linear orthotropic material (or that the graph data is wrong) and the best we can do is to use an estimated Vxy.

If stress/strain graphs are available at several stress ratios, such as 1:1 2:1 and 1:2 , then several values for Vxy can be obtained, and the average used for final material calculations.

- **Membrane Isotropic G**

This calculates the G value needed to make the membrane behave like an isotropic material such as a thin film. It reads the Ex. Ey and Vxy values from the selected line in the membrane table, and the calculated result can be automatically pasted into the selected line.

True isotropic materials have the same Ex and Ey, so the calculation with differing Ex and Ey gives a "quasi" isotropic material.

- **Cable Ea**

The cable and links table uses the Ea value for link stiffness. This value would result

directly from a stretch test of cable or webbing. If the manufacturer supplied the effective E value for the cable instead, this calculator will give the Ea from the effective E and the cable diameter . The calculated result can be automatically pasted into the selected line in the cable and links table.

- **Pole stiffness.**

This calculates the lengthwise stiffness (Ea) of simple tubular poles. The calculated result can be automatically pasted into the selected line in the cable and links table.

- **Encastre pole simulation.**

This calculates the spring rate needed to simulate the bending of a pole set in concrete, and also the pole lengthwise stiffness. The calculated results can be automatically pasted into the Point stiffness or Cable links table.

If the spring rate is going to be added to the pole top materials library, it is better to store the "SR for unit length". This lets the program adjust the spring rate for the pole length in the table. This value is obtained by entering a pole length of 1.

- **Beam simulation.**

This calculates the link stiffness needed to simulate a beam with a specified EI as a truss. The calculated results can be automatically pasted into the Cable links table.

The truss deflection under load will be in reasonable agreement with a real beam's deflection, but should be checked if the deflection is critical.

- **Pressure from wind speed.**

This calculates the pressure caused by a wind speed at a defined coefficient of pressure (Cp). Cp values can be obtained from wind tunnel tests, past experience, or from building codes. The calculated result can be automatically pasted into the selected line in the element loads table. Really these values should be used for estimation purposes, and the actual pressure required by your building code should be used instead.

- **Snow load**

This calculates the pressure caused by a depth of snow at a defined snow density. The calculated result can be automatically pasted into the selected line in the element loads table. Really these values should be used for estimation purposes, and the actual snow loading required by your building code should be used instead.

- **Unit conversion.**

This converts values commonly used in tensile structure engineering from one units system to another. The results can by coped to the clipboard, and then pasted where needed.

Standard Reports
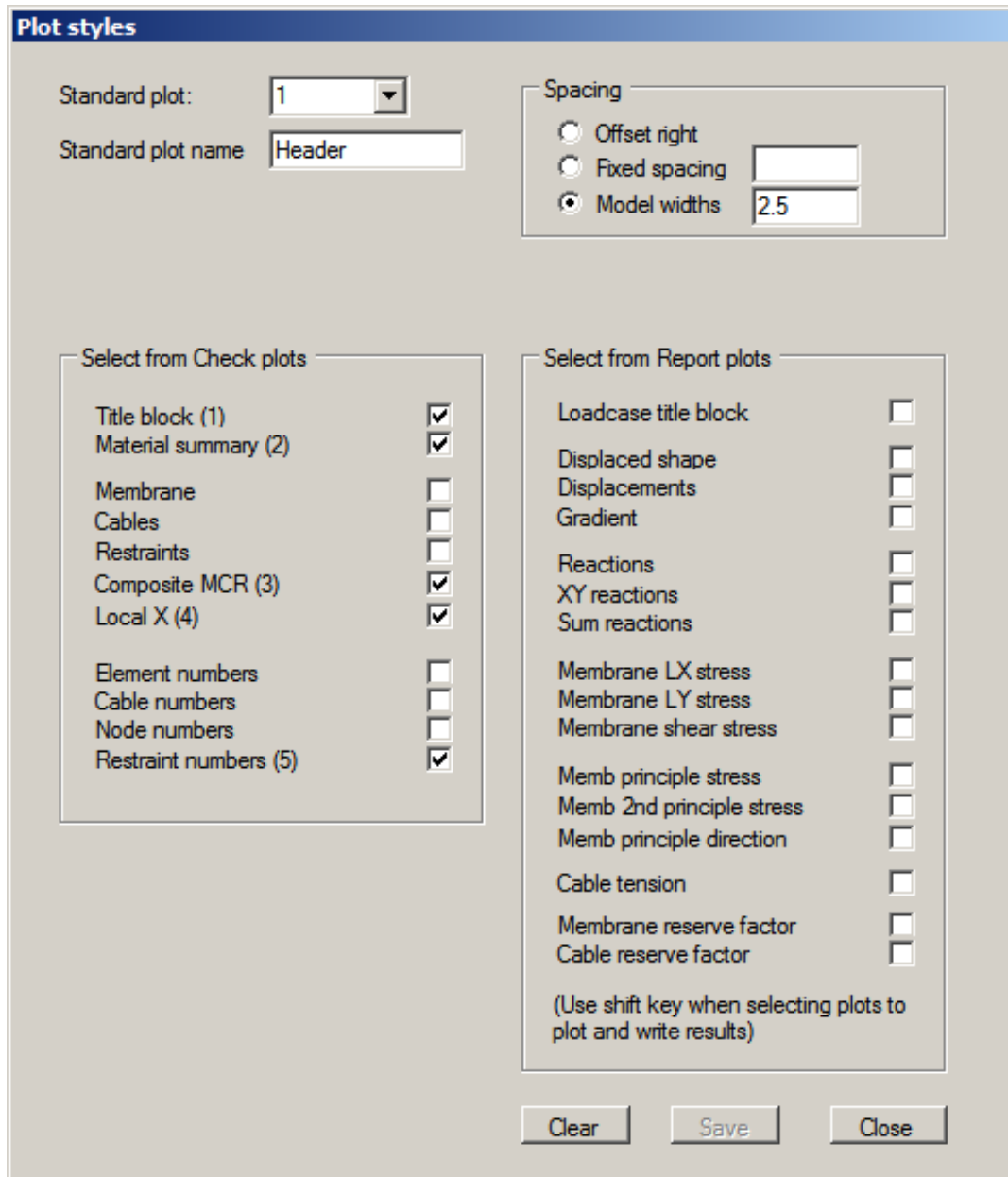
Standard reports let you set up multiple plots on a fixed horizontal spacing. This allows the generation of reports with the plots in a regular grid, with one row for each load case.

At least 2 standard reports should be defined:

Header
                contains the information needed at the top of a report. Typically

includes:

Title block
Material Summary
Composite MCR (membrane, cables and restraints)

Body

contains the information needed on. Typically includes:
Load case title block
Displaced shape
Membrane LX stress
Membrane LY stress
Reactions.

A report is then generated by clicking on the menu item Run Standard plots and selecting:

Header
Body with results set to load case 0
Body with results set to load case 1
etc

The plots can be edited by clicking on the menu item Edit Standard plots and filling in the form shown above.
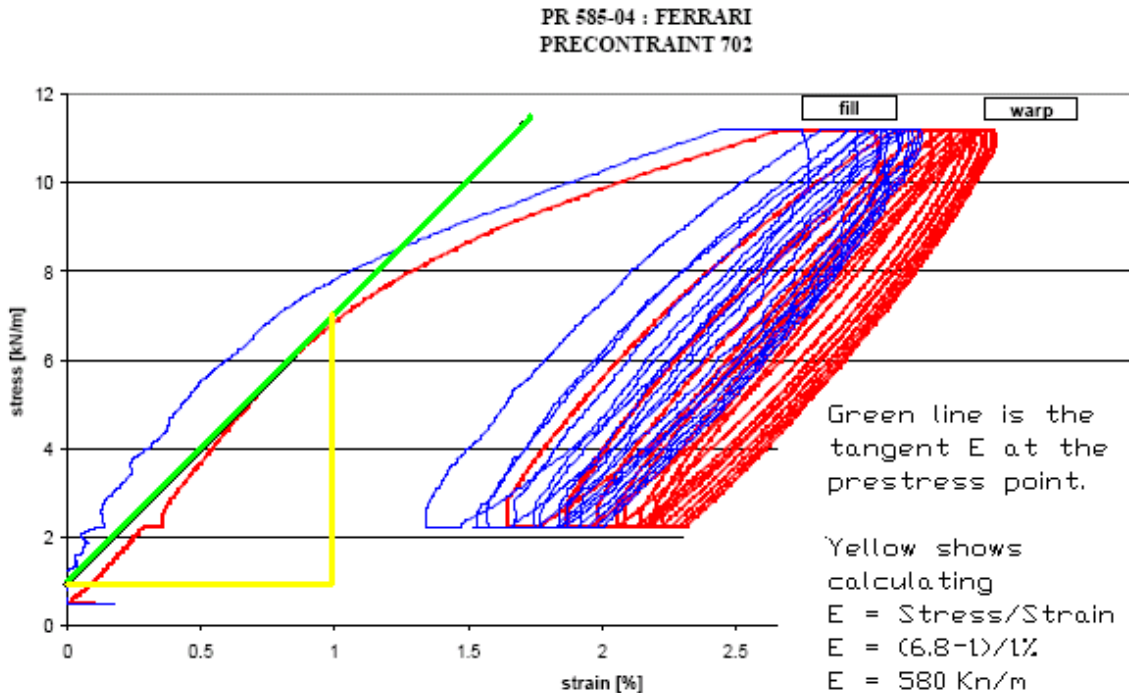
The spacing between the plots can be set a fixed distance, a multiple of the model width, or each plot offset right.

When using Report plots you can plot and write the results by holding down the shift key when selecting the plot.

Standard reports can be run using shortcut keys like Ctrl+1 (holding the "Ctrl" key down and pressing the "1" key) directly from the reports page.

Determining E values

In some situations we have to determine the E values from graphical data. Often the graph will be a biaxial stress/strain graph at 1:1 weft/warp tension ratio, like the graph shown here:

PR 585-04 : FERRARI
PRECONTRAINT 702

Green line is the tangent E at the prestress point.

Yellow shows calculating
E = Stress/Strain
E = (6.8-1)/1%
E = 580 Kn/m

We can obtain the tangential E value from the graph in the manner shown, either using the "first loading" part of the curve, or the "after settling" part of the curve. You can see that the E value is similar for these two situations.

This gives us an Ewarp value in this case of 580 kN/M

However this figure is a mix of two effects. Since the fabric is loaded bi-axially, at a 1:1 weft/warp tension ratio, the stretch in the warp direction for a given stress is reduced by the Poisson effect from the stretch in the weft direction.

To separate these two effects we would need graphs from a different kind of test, such as two graphs, loaded bi axially at different weft/warp tension ratios. From this we could derive the stiffness(E) values and the poisons ratios.

But often we will have some idea of the Poisson ratio, from experience with similar fabrics, or from simple tests. In a fabric it is essentially the crimp interchange, and may have a value close to 0.3

Once we know the Poisson ratio(v), we can reduce the measured E values from the 1:1 biaxial test to allow for this effect. The reduction for the general case is somewhat complicated, but for the situation where Ewarp = Efill then we have:

E real = E graph * (1 - v^2) / (1 + v)

With a Poisson ratio v = 0.3 this amounts to a factor of 0.7 so our Egraph = 580 becomes Ereal = 406

This would be the value for Ewarp or Ex and in this case Efill or Ey is similar in value.

There is a calculator to help with the more general case when Ex does not equal Ey, and also when

biaxial strech graphs at various stress/strain ratios are availiable.

There is no sense in being too accurate about these figures, as the next batch of fabric, at a different temperature, may show slightly different results. But it does help the FEA solution to reduce the stiffness, whilst keeping the prestress value the same.

Examples

Some example drawings are supplied with this help file, that are annotated and show the steps taken to perform an FEA analysis.
Also included are the model and output files for these examples.

- Example Barrel Vault

This shows a barrel vault, with fixed edges.
Built in Imperial units (inches, lbs)
Analysed for wind loading

- Example

This shows a hypar with 4 catenary's edges.
Built in metric units(meters, kN)
Analysed for wind

Looks at the situation where there is uncertainty as to the best direction to run the seams, determining this from the principle stress direction, and then modelling the hypar with the seams running in that direction with an anisotropic material.

- Example Cone tent

This shows a cone tent with cable edges.
Built in metric engineering units(meters, kgf)
Analysed for wind and snow.

Contact List and Technical Support

Support questions should be sent to:
support@mpanel.com

Alternatively you can contact members of the support team directly.

Andrew Askwith, Meliar Design, UK
andrew@meliar.com

Tim Akes, CAD Effects LLC,   USA
takes@cadeffects.com

Alan Stewart, Structureflex, Australia
alan@structureflex.com.au