Contents

What's New

Version 17 summary of new features:
- New ribbon menu buttons.
- Fix only in LC0.
- Fabric and cable mass added to databases.
- Fabric and cable mass can be added to all load cases as dead load.
- Help button context sensitive.
- Improved text spacing in plot tables.
- Help file includes more example model [tutorial videos](#)

Version 16 summary of new features:
- Auto correct mesh orientation of mirrored and rotated meshes.
- New "Poly Number" plot added to check model options.
- Plot headers added to individual plots.
- 3 additional text size options providing finer control for plot results.
- Table splits updated and arranged more intuitively and saved on exit.

Version 15.1 maintenance update:
- Pressure load up/down title block sometimes reported incorrectly ..... fixed
- Results filter not working for XY, XYZ, and PList reactions .... fixed
- Rhino 7 draws text too large depending on templates used .... fixed

Version 15 summary of new features:
- The solver has been upgraded to providing robust solutions without the need for the superseded dual-loop solver.
- New "Pinned Pole" has been introduced.
- New "Free pole or beam" has been introduced.
- Simulate beam poly special status has been removed.

Introduction

Performing FEA (Finite Element Analysis) 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.
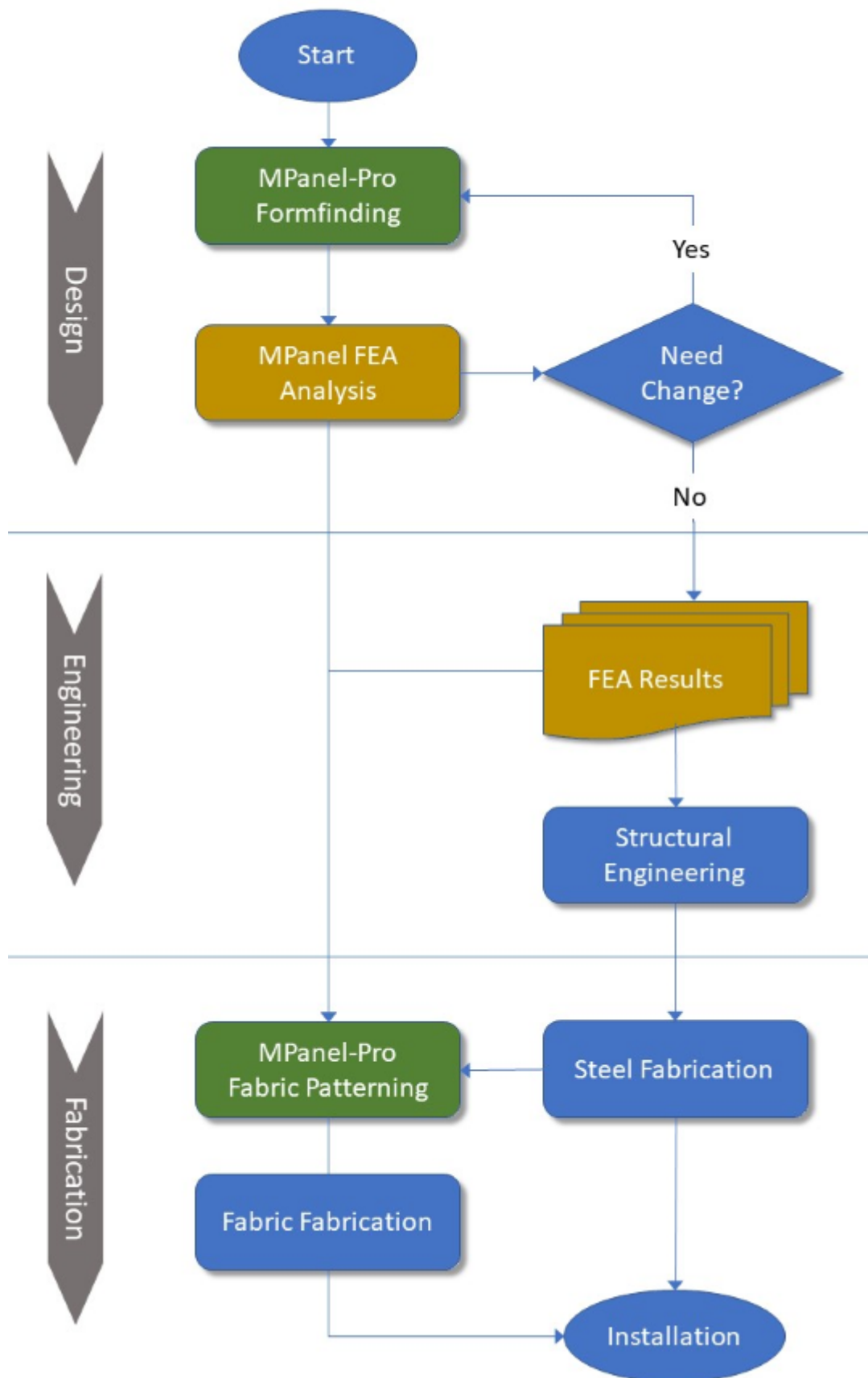
This program works around these problems by supplying a simple, step by step interface called MPanel FEA, which runs a powerful non-linear solver. The solver is robust and generates good solutions even when large parts of the membrane are slack. Program operation and concepts are kept simple and rapidly produce good results that are clear and understandable for a wide range of users.

The purpose of the program is to perform an analysis of the membrane by applying loads to determine:

- Stresses in the membrane (to determine suitable fabric to suit strength requirements)
- Cable tensions (to determine suitable cable size to suit strength requirements)
- Deflections (to ensure fabric doesn't strike any existing structures and to ensure against ponding)
- Reactions (to be passed to a structural engineer to design suitable steel members, fittings, fixtures and foundations)

Ideally, an initial FEA analysis will be performed during the concept design stage to determine these results and a more detailed analysis will be carried out at the same time as structural engineering design is undertaken.

PHASES OF A FABRIC STRUCTURE PROJECT

Start

Design

MPanel-Pro Formfinding

MPanel FEA Analysis → **Need Change?**

Yes

No

Engineering

FEA Results

Structural Engineering

Fabrication

MPanel-Pro Fabric Patterning ← **Steel Fabrication**

Fabric Fabrication

Installation

General operation of the program

In general, on each model, you use the program tabs in order from left to right of the primary menu shown below. This provides a natural work flow to an FEA problem, from defining the model, specifying the material properties, specifying the loads, finding a solution, and outputting the results. The tools and options menu's are contextual with different tools/options available depending on which primary menu tab is selected - non usable tools/options are greyed out.
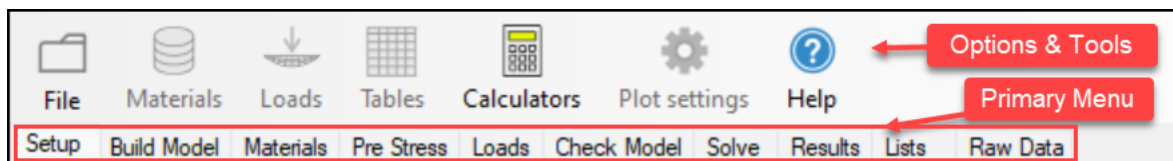
Models are created in an AutoCAD or Rhino drawing similar to a standard MPanel model. The model is read into FEA (refer Build Model) and all free mesh edges have cables added to them.

The model is written to a file called "myname.fem" and consists of triangular plane stress elements and links. The elements and links have properties set such as stiffness, pre-stress, etc.

In the same file the settings used to make the model, on the Build Model page, are stored.

Up to twelve load cases can be defined, with load case 0 being pre stress only. These are also stored in "myname.fem"

The file is sent to a geometrically non linear solver, that computes the displacements of the model nodes to meet the specified load case. The results are written to a file called "myname_lcx.out", where x is the load case number.

The results from the output files can then be written back to the CAD drawing as coloured models showing stresses, displacements, etc.

See examples:

The program can be re sized and minimised using the usual windows methods. If you have limited screen size available the button images can be hidden to save space.
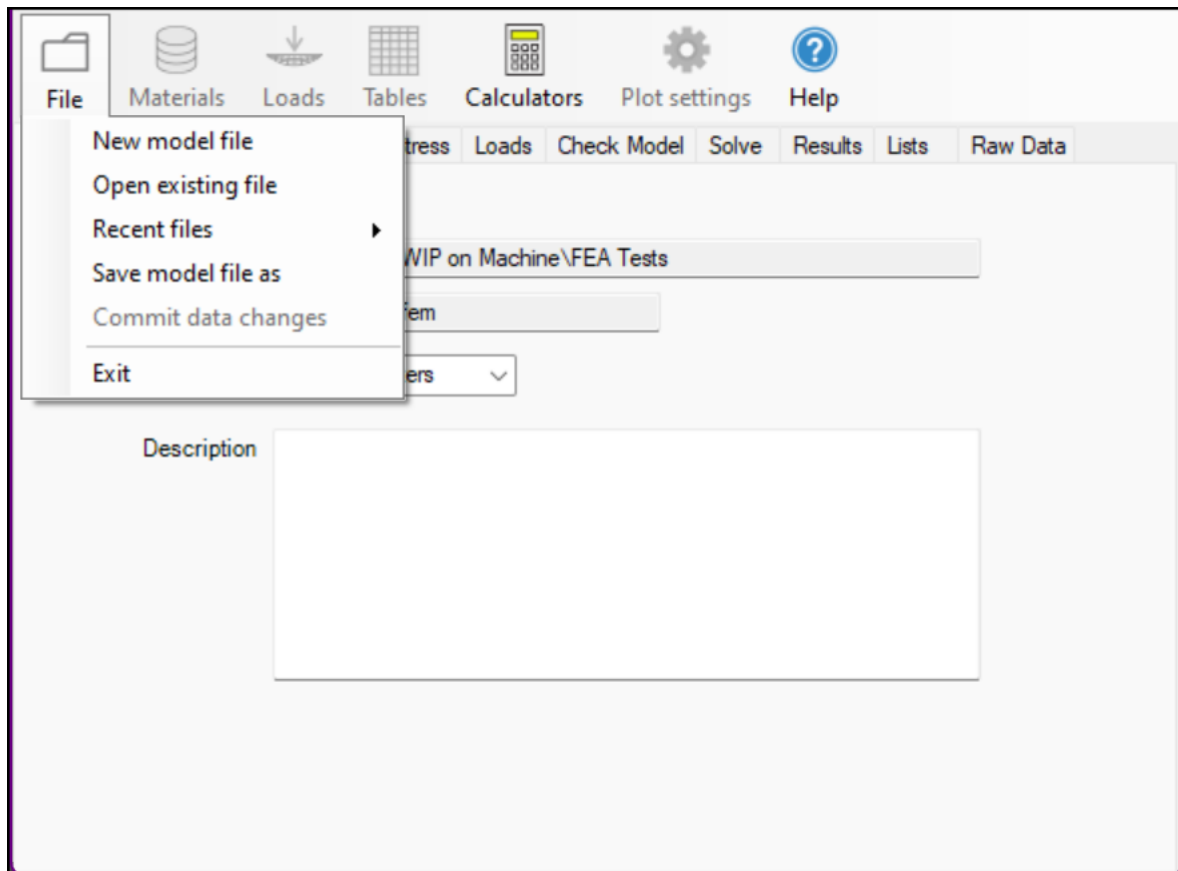
Help

Help contents

✓ Show button images

Hide button images

About

Model Name

The first step in a new project is 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 but can be useful when reviewing the FEA at a later date or when comparing different analysis.

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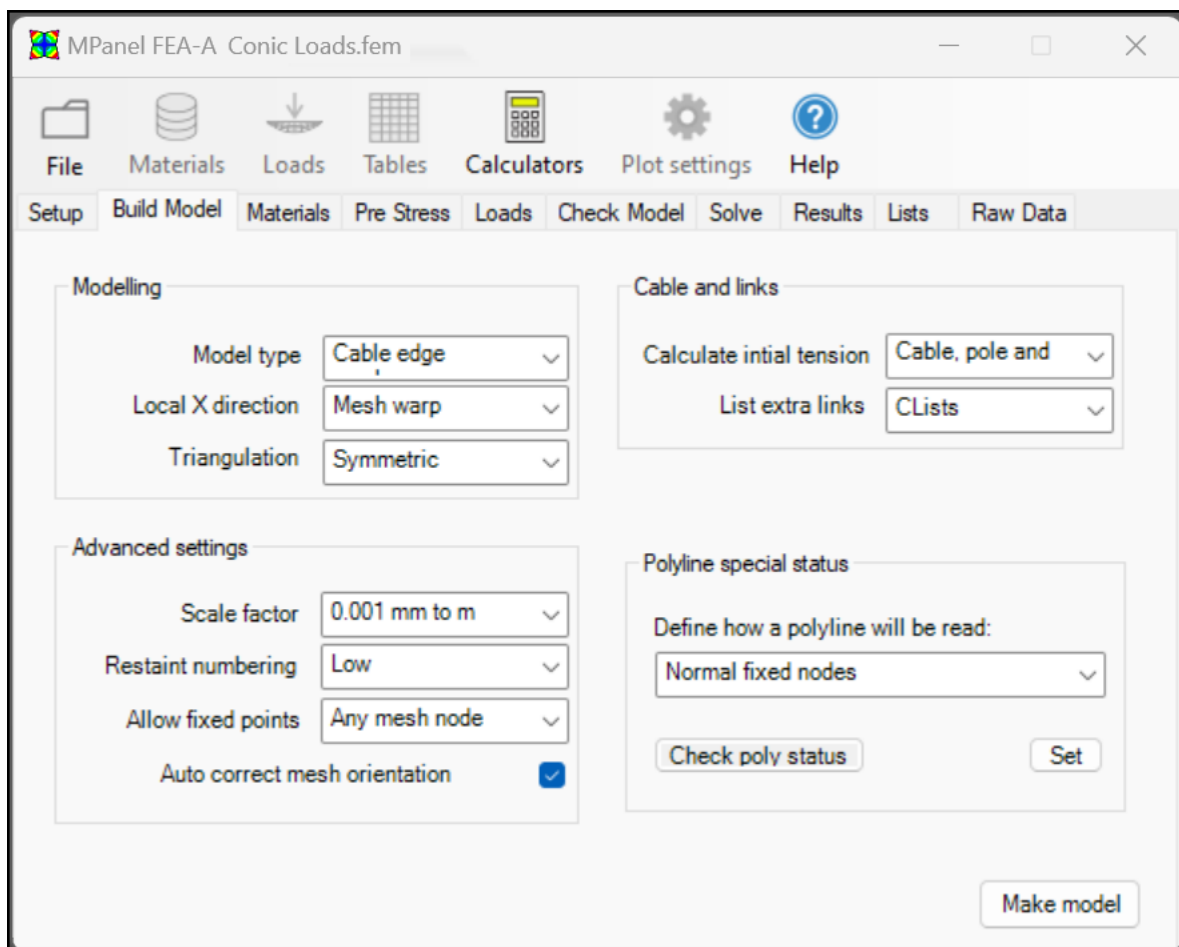
The "Make Model" button will read a selected drawing from an open CAD drawing consisting of:

Meshes   to define the fabric surface
3DPolys   to define the system fixed points on mesh edges or line ends and cantilever, free or pinned poles.
Lines   to define any guys including compression elements (poles) backstay cables and sail corner links (turnbuckles etc.)

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.

3DPoly special status settings should be applied as required prior to reading the model into FEA (see below).

Make Model and Optimization.

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

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

The FEA model is optimized 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 reasonably fast times, 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 optimizer will sort out the model into separate regions, and optimise each one separately.

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

Modeling 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 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 millimeters 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.

- Auto correct mesh orientation

CAD mirroring, copying and 3D rotating can result in the incorrect/inconsistent mesh orientations when they are read into FEA. Pressures are applied normal to mesh surfaces so it is important that surfaces are oriented consistently. This option if selected allows FEA to correct these inconsistencies.

Cables and links
- Calculate initial tension

Normally the FEA build will find any cables on mesh edges, and calculate an appropriate initial tension. Often the cables will terminate in a corner with a single guy, link or corner adjuster (e.g. turnbuckle or toggle), 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 (backstay cables or rods) 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
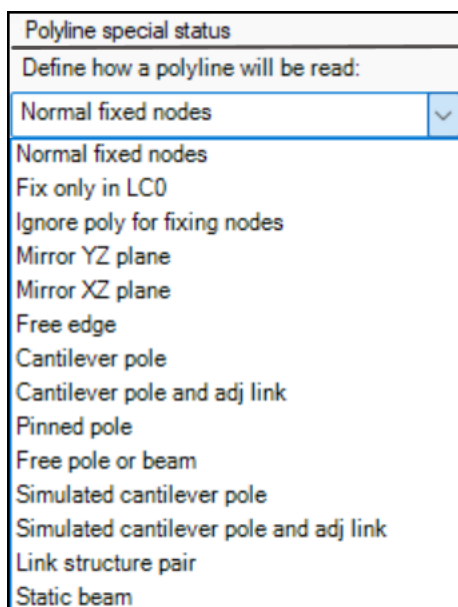
Normally Polylines are read as a series of restraints. Optionally polylines can be selected then "Set" to apply special status so FEA will interpret it as follows:

```
Fix only in LC0              Fixes the poly fixing nodes only in LC0,
                             but uses the poly colour to define Plists
                             useful for vault structures mid arches.


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.


Cantilever pole              Reads a poly with multiple vertices as a
cantilever pole/beam
                              fixed at the base, allowing multiple sail
connections
                              (at pole vertices)and allowing pole top
```

movement during

the application of loads (see video links
below)

Cantilever pole
cantilever pole/beam
with adjustable links
connections

Reads a poly with multiple vertices as a
fixed at the base, allowing multiple sail
(at pole vertices)using adjustable links to
top movement during pre stress and the

accommodate the pole

application of loads

Pinned pole
(set in concrete)

poly with more

Reads the poly as a beam fixed at the bottom
and allowed to bend the poly needs to be a 3D
than 2 nodes (suggest 7-11)

Free pole or beam
no end fixing

nodes (suggest 7)

Reads the poly as an beam allowed to bend with
the poly needs to be a 3D poly with more than 2

Simulate cantilever pole
simulates

at the top.

Reads the poly as a link with a top spring that
a pole set in concrete, with a point stiffness

(previously called Read as encastre pole)

Simulate cantilever pole
uses adjustable
with adjustable links
during pre stress

adjustable links)

An alternative cantilever pole simulation that
links to accommodate the pole top movement
(previously called Read as encastre pole with

Link structure pair
links,

Makes a pair of polys to be modeled as a set of
typically to allow a top ring to move

Static beam
resolves them into two

beam that is

Collects the forces along a polyline and
forces, one at each end this simulates a stiff
fixed at it's ends.

Video: **Fix only for LC0**
Video: **Analyze hypar with fixed connections**
Video: **Analyze hypar with corner links to fixed connections**
Video: **Analyze hypar with link poles and backstays**

Video: **Analyze hypar with corner links to link poles and backstays**
Video: **Analyze hypar with cantilever poles**
Video: **Analyze hypar with cantilever poles with adjustable corner links**
Video: **Analyze hypar with simulated cantilever poles**
Video: **Analyze hypar with simulated cantilever poles with adjustable corner links**
Video: **Analyzing multiple sails connected to a free pole and fixed connections**
Video: **Analyzing multiple sails connected to a free pole and cantilever poles**
Video: **Analyzing multiple sails connected to a pinned pole and cantilever poles**

- **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 defaults 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.
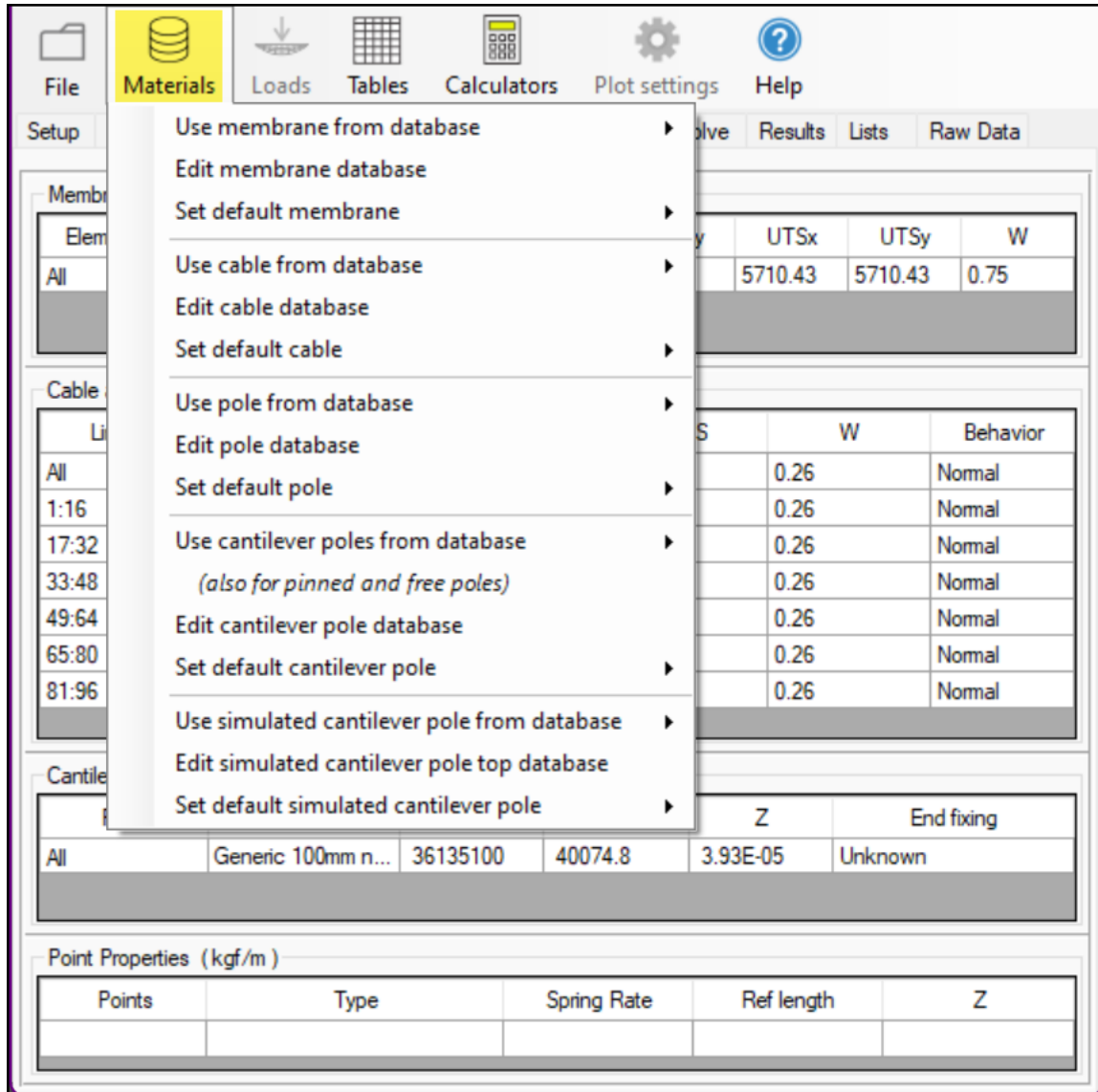
When in the Materials tab selecting the Materials menu tool provides access to the materials databases to use values from the database, edit the database and set the default values that will be used during Build Model.
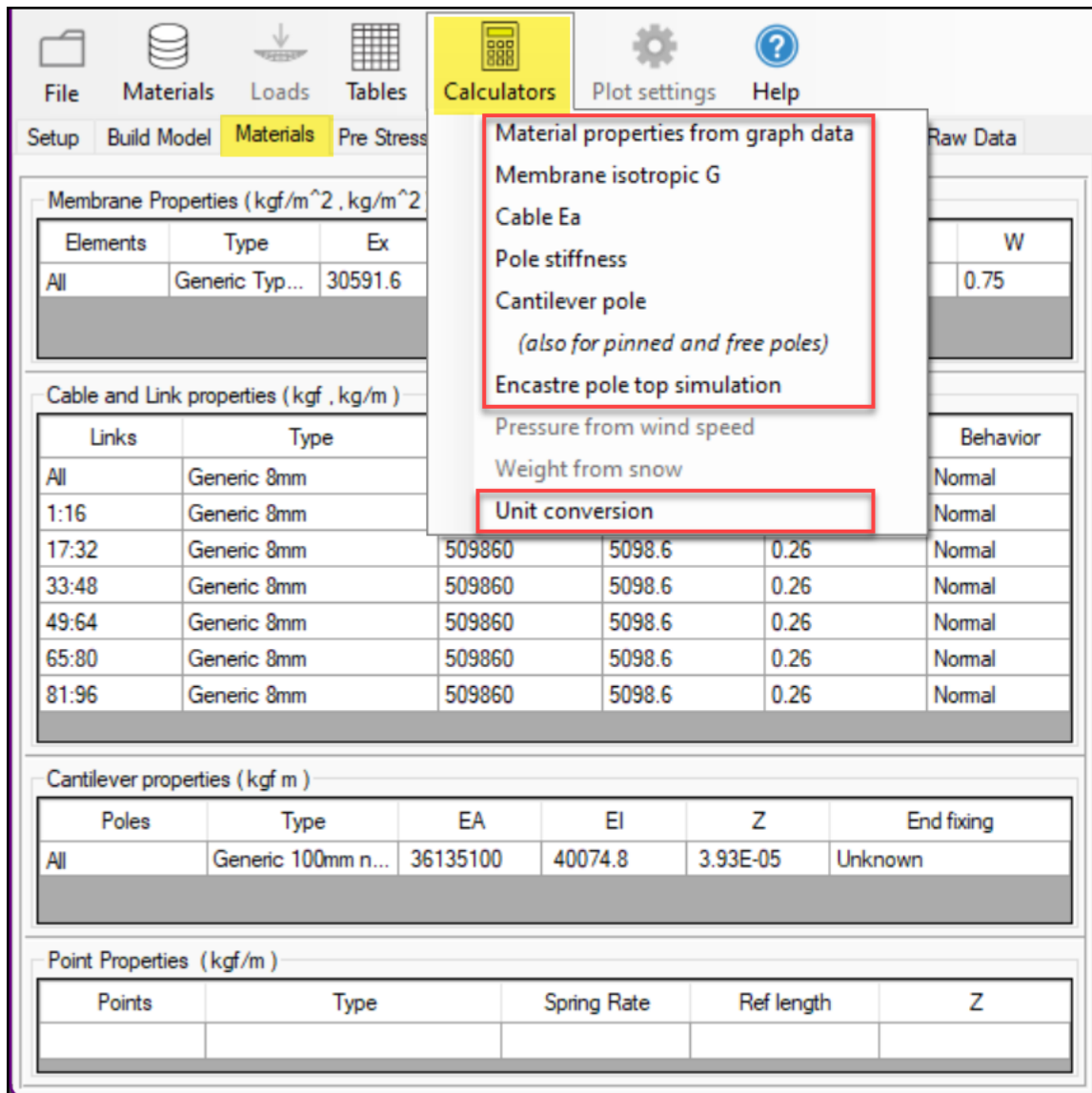
When in the Materials tab selecting the Calculators menu tool provides access to calculators to facilitate determining material properties.

- **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.

W

Is the weight of the fabric (either from the materials database or manually entered). These figures are used during the analysis if you select "include dead weight" when solving.

- **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.

W

Is the weight of the cables (either from the materials database or manually entered). These figures are used during the analysis if you select "include dead weight" when solving.

Behavior

The link can be behave as :
 Normal a link that accept tension only
 Compression a link that can accept tension and compression
 Adjustable a link that can vary it's length during load case zero, to simulate a turnbuckle

- **Cantilever Properties**

Applies to cantilever poles used in the model:

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.

Ea

Ea is the lengthwise area stiffness. A calculator is provided to give this figure from the poles diameter and wall thickness.

EI

Flexural rigidity is derived by multiplying the Material E x Area Moment (I). A calculator is provided to give this figure from the poles diameter and wall thickness.

Z

Section modulus of the pole. This is used to calculate the bending stress on a cantilever pole. If this value is zero then the reported bending stress will also be zero. A calculator is provided to give this figure from the poles diameter and wall thickness.

- **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.

Z
The section modulus of the beam. This is used to calculate the bending stress on an encastre pole. If this value is zero then the reported bending stress will also be zero.

Pre Stress

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 e.g. the Tensinet Design Guide suggests a value of 1.3% of fabric tensile value as a minimum.

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 (AutoCalc = 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
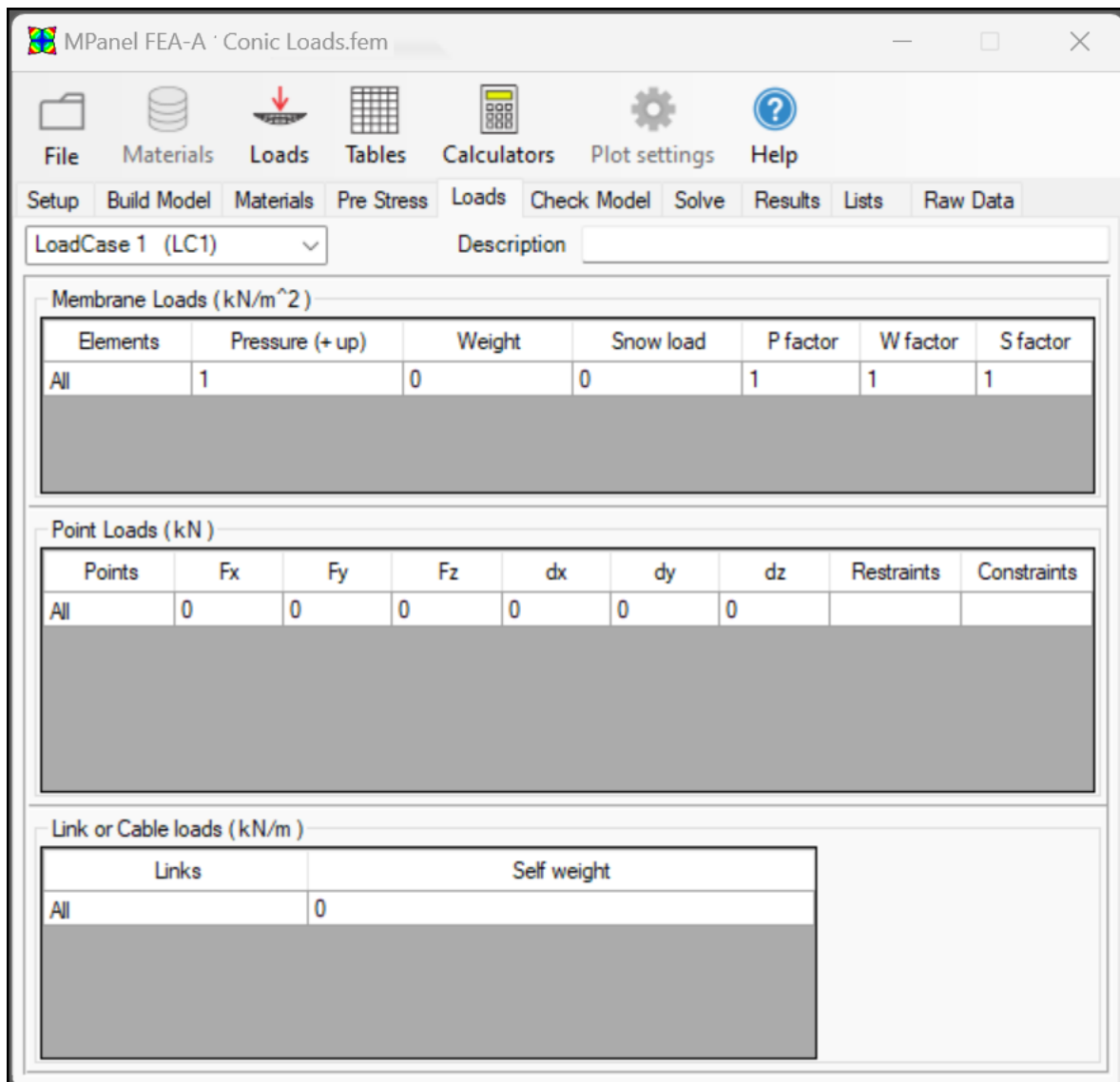
See also:
[Using MPanel FEA tables](#)
[Using MPanel FEA database libraries](#)
[Using Mpanel FEA calculators](#)

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 saved to and downloaded from the loads library (Load settings menu).

A description of the load case can be added.

There are three loading tables:

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

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
```
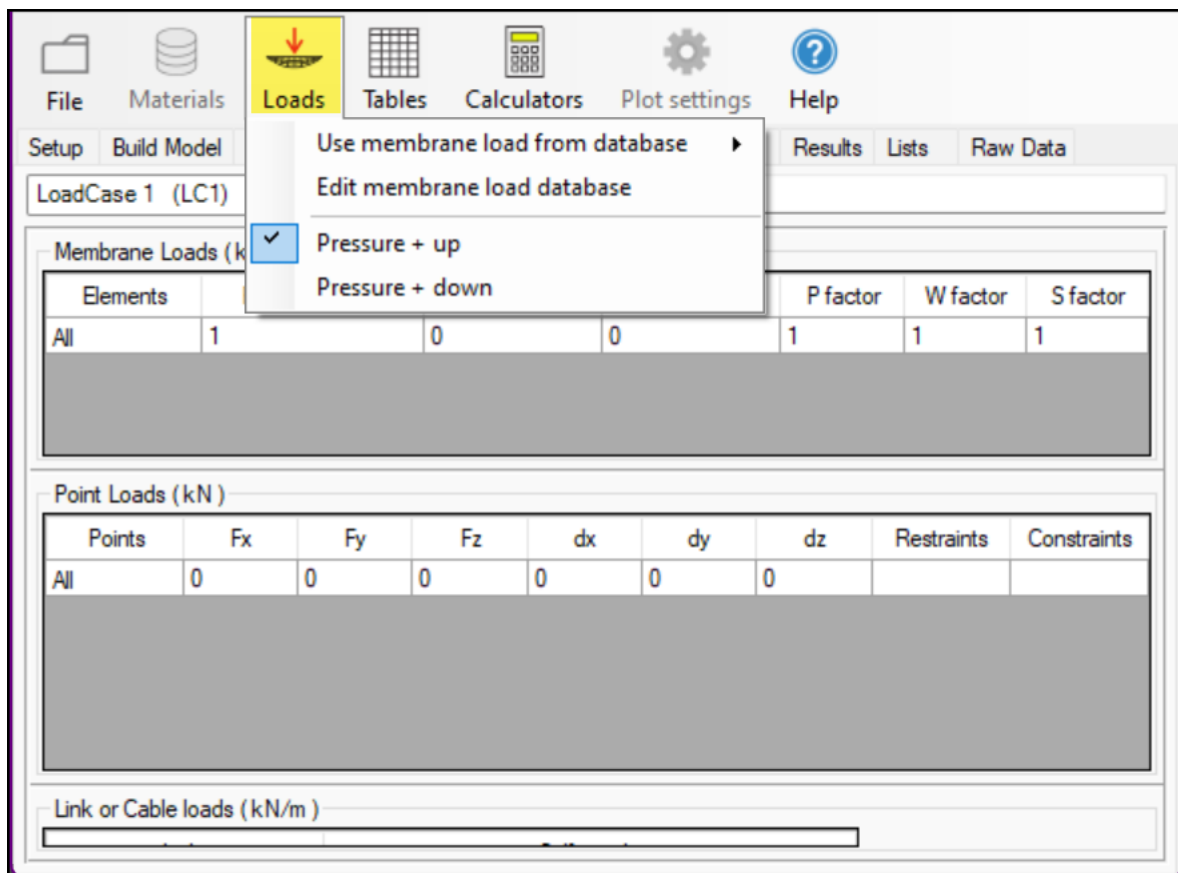

The Loads options key allow you to save and re-use load cases to a database as well as defining if pressure load values are applied up or down to suit the local building standards in different countries.
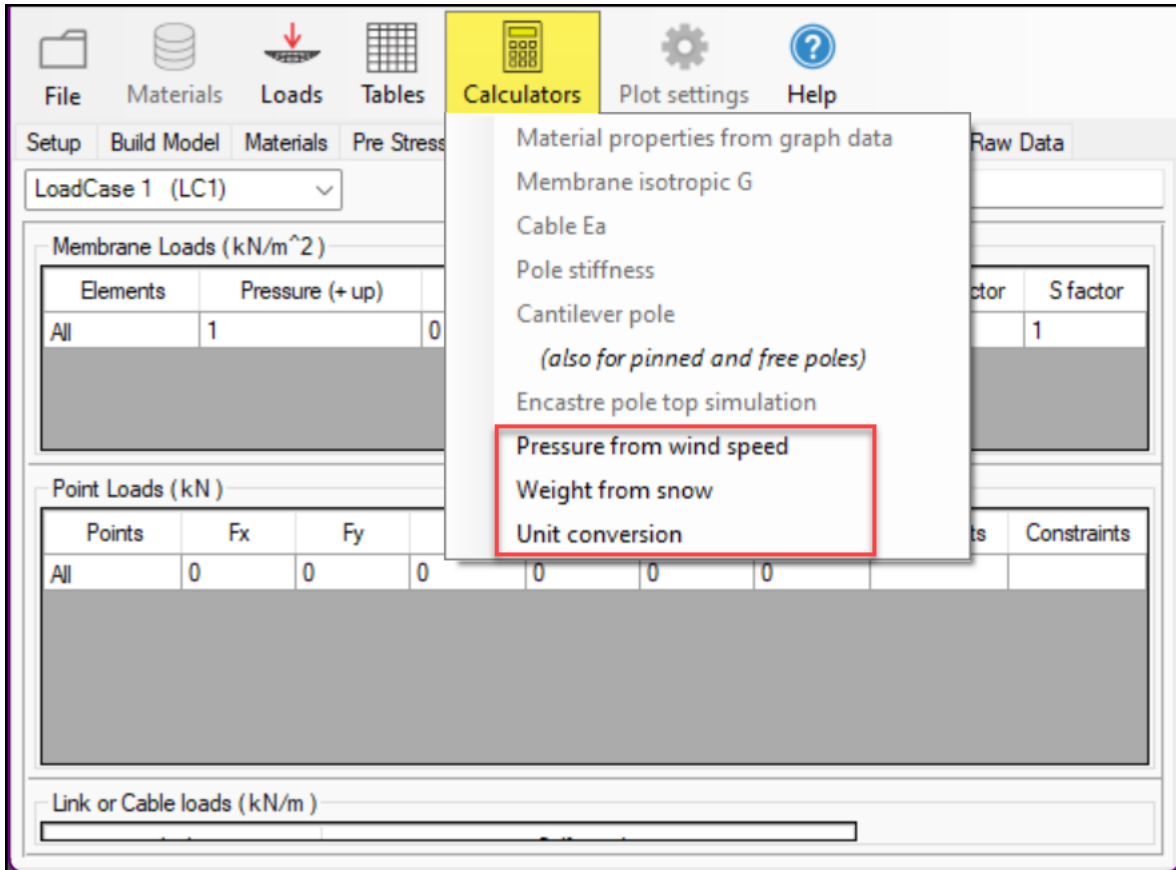
Selecting the Calculator tool when in the Loads Tab provides relevant options to assist with determining loads.

- **Membrane loads**

Pressure
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 can be specified with positive value giving an upward force on each mesh, or with positive values giving downward force on each mesh. To change this option use the menu item "Load settings" A calculator is provided to give the pressure from a known wind speed and coefficient of pressure.

Weight
A weight, representing a constant weight per unit 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. This provides a manual option to add weights including fabric self weight if the weights are not included in the materials tables. Normally add fabric self weight (values shown in the materials table) during the analysis by selecting "include dead weight" when solving.

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 snow depth and density.

Load Factors
These are used to multiply the Pressure, Weight and Snow loads by factors to vary loads in complicated loading cases and to conform with local building codes.

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 wind uplift of 2 applied to the whole model, and no weight or snow, would have a table entry of:

```
Elements    Pressure (+ up)     Weight    Snow load    P
factor  W factor   S factor
All          2                   0         0            1
1           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 structural 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.
Slip cable... allows a cable edge to slip though 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. This provides a manual option to add cable or other weights, if the cable weights are not included in the materials tables. Normally add cable self weight (values shown in the materials table) during the analysis by selecting "include dead weight" when solving.

Check Model
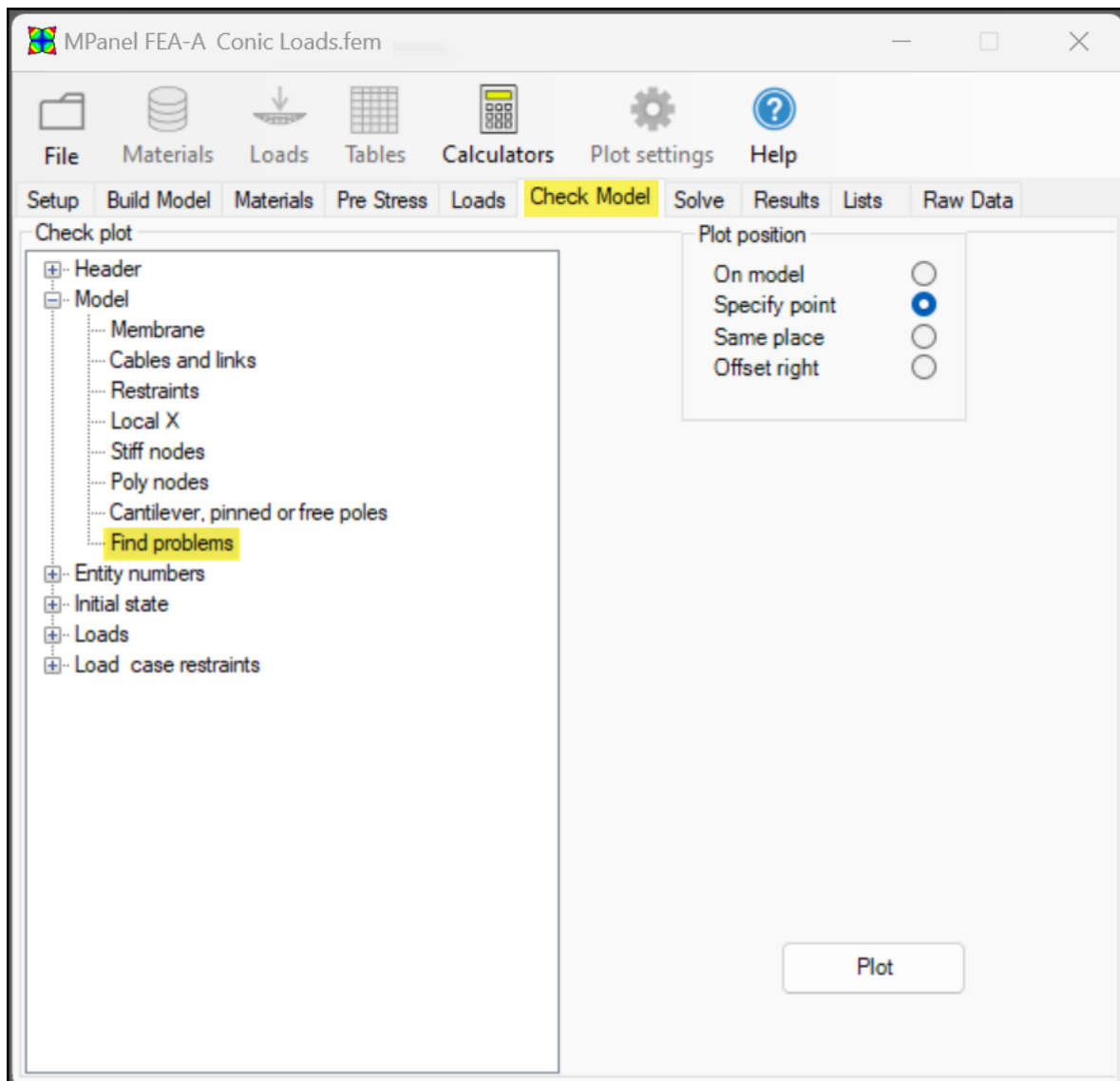
It is important to check the model before solving to ensure you are analyzing what you think you are.

The check model tab provides a variety of drawings/plots 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:

Header
- 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.

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)
- Local X   Shows the direction that is assigned as local X.
- Stiff nodes   Shows the points with stiffness added to them
- Polyline nodes   Shows the nodes for each polyline by the program
- Cantilever Poles Shows each polyline defined as cantilever poles by the program
- Find Problems   Shows any problems detected with this model (see notes below)

Entity Numbers

- Membrane elements Identifies the membrane element numbers.
- Cable Numbers   Identifies the link numbers, that make up the cables and guys.
- Restraint Numbers Identifies nodes that are restraints.
- Stiff node numbers Identifies nodes that are stiff nodes.
- Cantilever poles   Identifies numbers of poles defined as cantilever poles.
- Poly numbers   Identifies poly's in the model and their numbers.
- Poly node numbers Identifies poly node numbers.
- Node numbers   Identifies all node numbers.

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 and link tensions
- Initial Forces   Shows the initial forces

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

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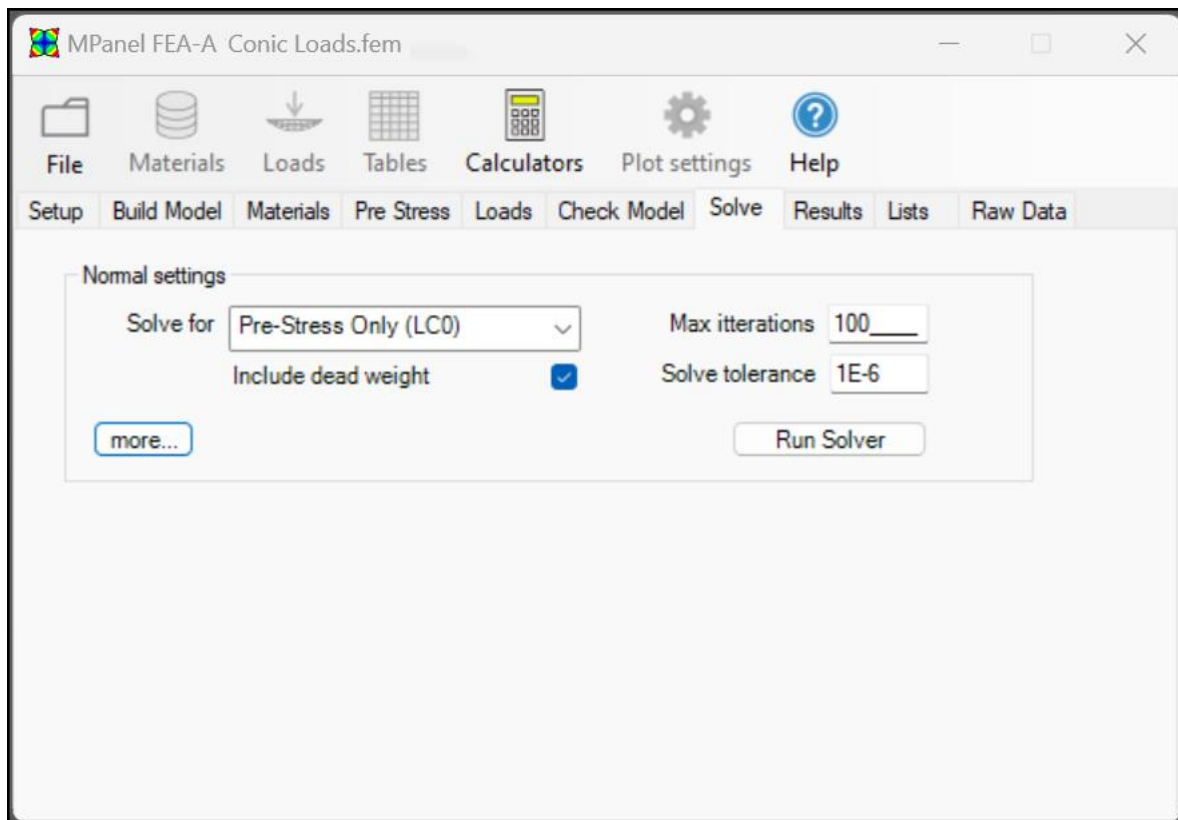
This starts the solver program which calculates the displaced node positions and element stress for the required loading. The progress of the solution is reported in a graphical window. The model geometry, and stresses are updated on each iteration, this allows for a geometrically non linear solution,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

Include dead weight will include the weight of the fabric and cables (values shown in the materials tables will be used).

If the model has reasonable material property values and low loads, the solution will converge quickly, in less than 10 iterations for example. If the model involves substantial shape change and deformation then the solution will converge slower, and may need 250 or more iterations to solve.

If the model includes Cantilever poles then the solver may also require additional iterations especially if adjustable links are selected.

The residual is graphed in numbers like 1E-3, which means 0.001 and this should be used for cantilever poles and more complex models. The default tolerance for a solution is 1E-6 which is OK for simple models.

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 and changes to the model. materials or pre-stress may be required.
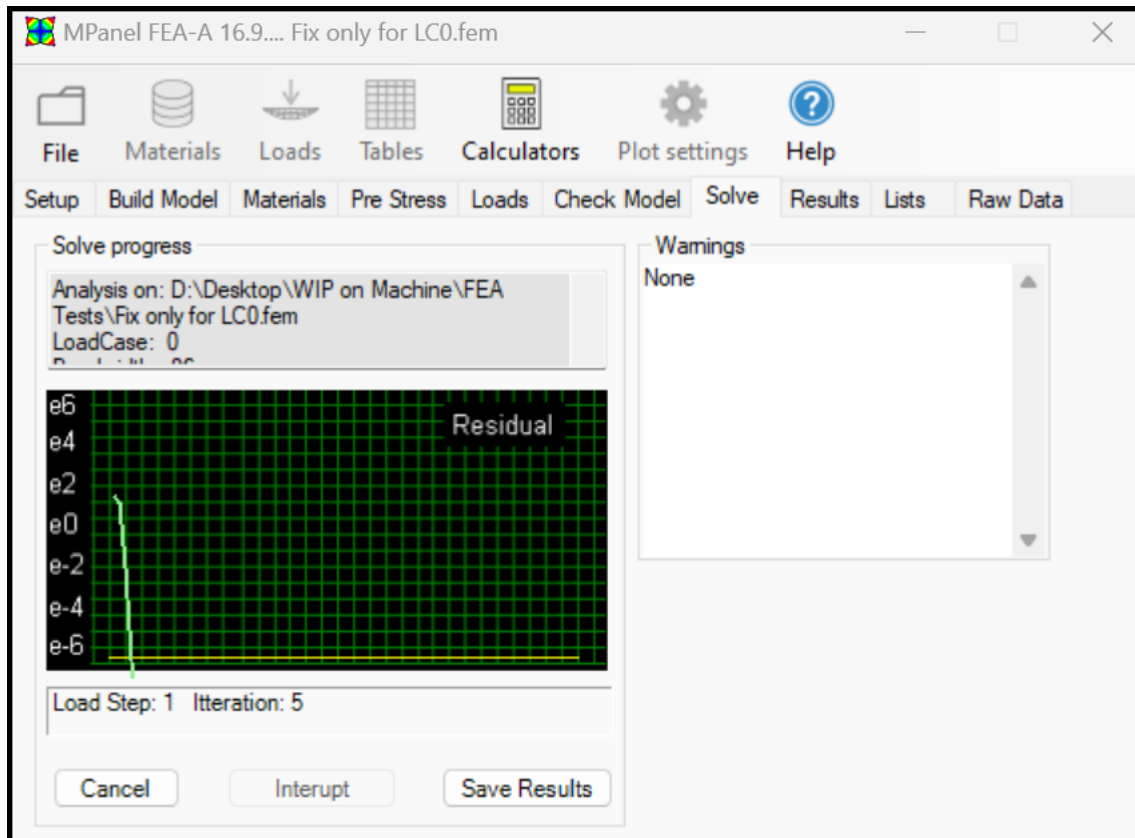
When the solver has finished, click on "Save Results" to write the solution to the output file. Or "Cancel" to discard the results.

In difficult to solve cases it may help to specify that the load is applied in a series of smaller steps by changing the load steps (available under more settings). For example, a load step setting of 2 will apply half the load to the model and solve then apply the balance of the load in step 2.
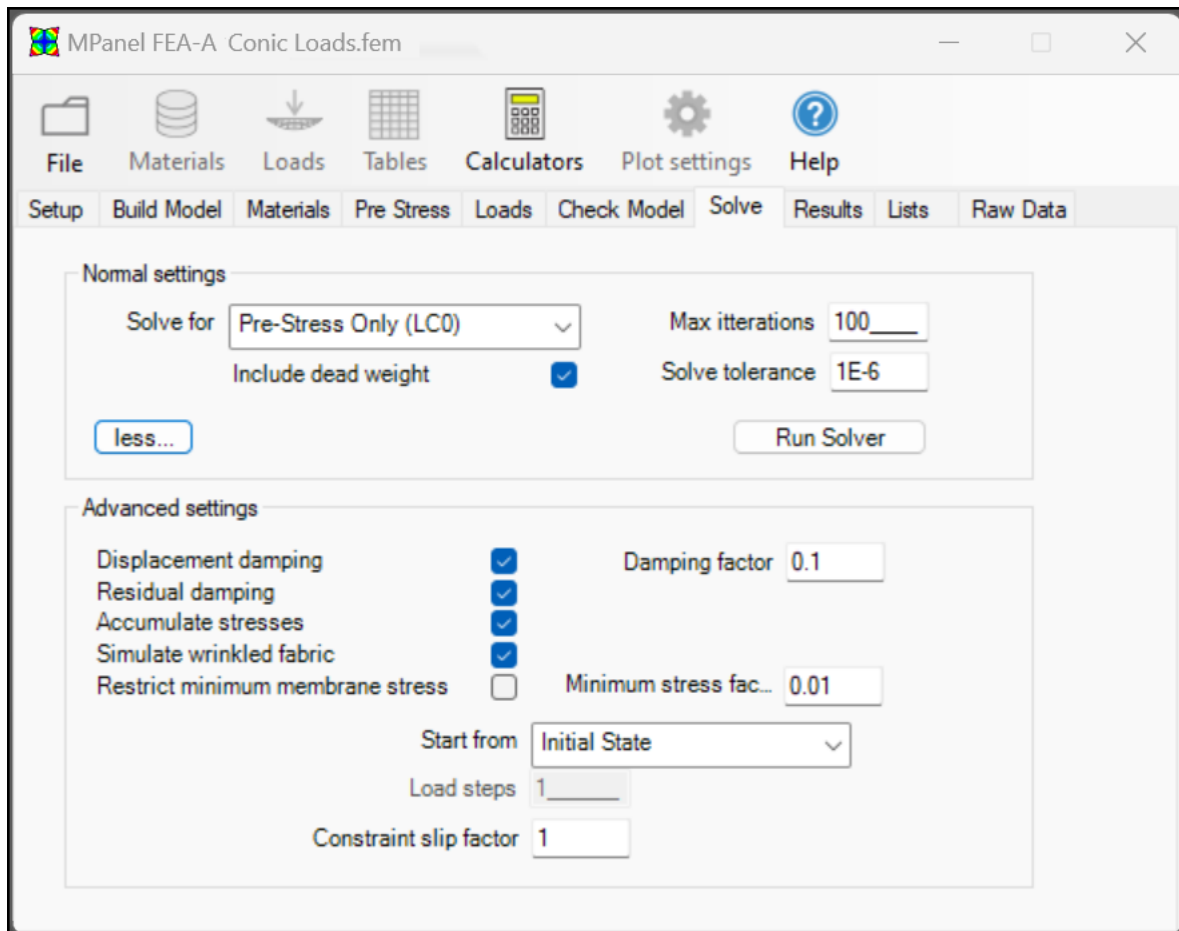
Damping.

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 "CC", for "cable compression"

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.

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.

Restrict minimum membrane stress. This is used with models where large parts of the membrane are slack, preventing a valid solution. By specifying a minimum membrane stress, as a percentage of the membrane prestress (0.01 = 1%), the solver can solve to produce results that will be a good approximation of the real world result.

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.
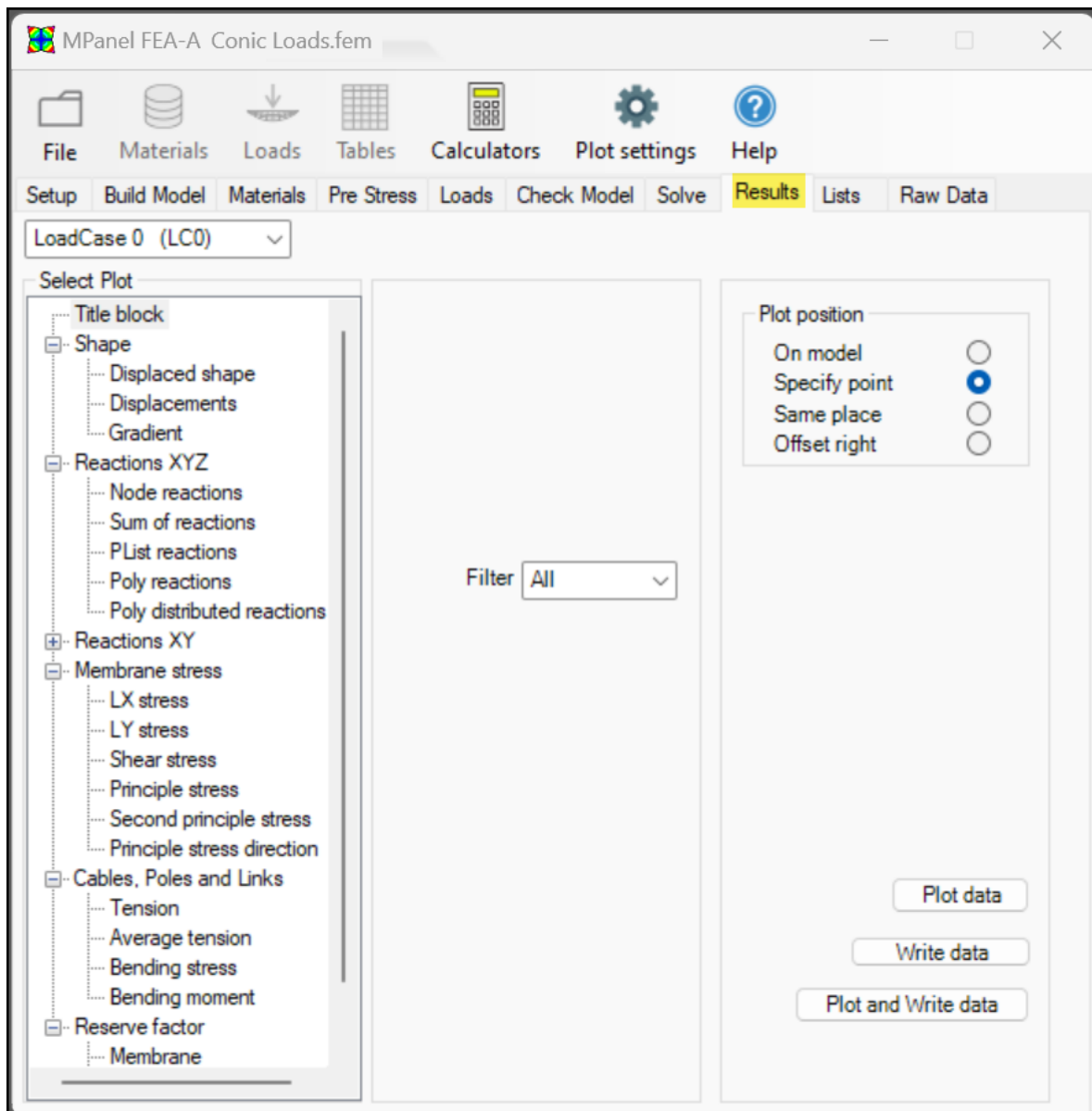
Draw Results

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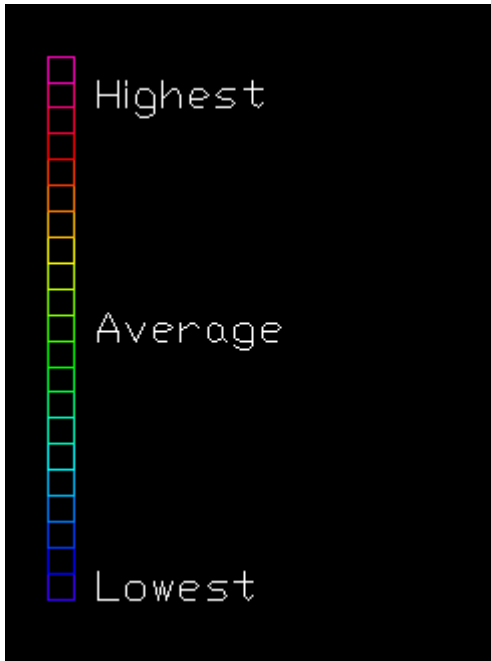
(With a special case of white indicating slack fabric in the principle stress plots.)

 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** XYZ   reactions at each of the fixed points.
- **Sum Reactions** XYZ   sum of the reactions for a set for the
complete model
- **PList sum reactions XYZ**   sum of reactions for each PList
- **Poly sum of reactions XYZ**   sum of reactions for each polyline in
the model
- **Poly distributed reactions XYZ**   the distributed reactions along the
polyline

- **Reactions** XY   reactions at each of the fixed points.
- **Sum Reactions** XY   sum of the reactions for a set for the
complete model
- **PList sum reactions XY**   sum of reactions for each PList
- **Poly sum of reactions XY**   sum of reactions for each polyline in
the model
- **Poly distributed reactions XY**   the distributed reactions along the
polyline

- **Membrane LX Stress**   stress in the local X direction
- **Membrane LY Stress**   stress in the local Y direction

- **Membrane Shear Stress**   shear stress between the local X and local Y 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 and link tension**   the tension in each link
- **Average cable tension**   average tension in a group of links that make up a cable

- **Bending stress**   maximum bending stress in an encastre beam
                                        (only valid if your encastre pole has a Section Modulus value)
- **Bending moment**   bending moment plot in a cantilever pole
- **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](#).

The text size can be set in the plot settings, to be a fixed value or as a percentage of the model average element size providing control over the plot results.

Plot text size and resolution

Text style	Percentage			Fixed size		
Small	100	% of default small size	☑	50	height (m)	☐
Medium	50	% of default medium size	☑	75	height (m)	☐
Large	100	% of default large size	☑	500	height (m)	☐
Scale	100	% of default scale size	☑	5	height (m)	☐
Title	50	% of default title size	☑	10	height (m)	☐
General	50	% of default general size	☑	2	height (m)	☐

Number of significant figures   5

Use scientific notation   ⦿

Use engineering notation   ○

OK   Cancel

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 visualizing the mesh movement. An original, non-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 represented by a node (e.g. 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 visualize 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
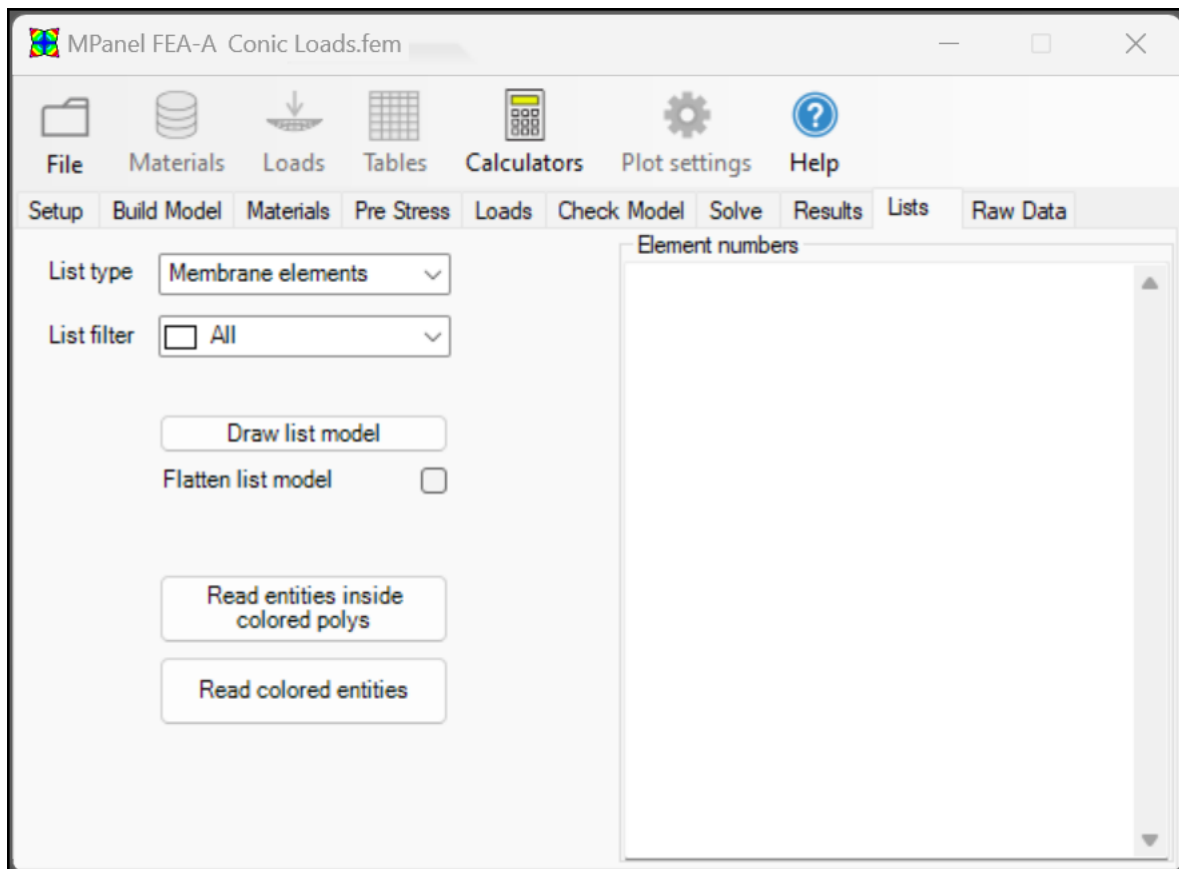
Lists allow a group of elements, links or points to be defined, and referred to as a single unit.

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.

Video: **Create PLists form model example**

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
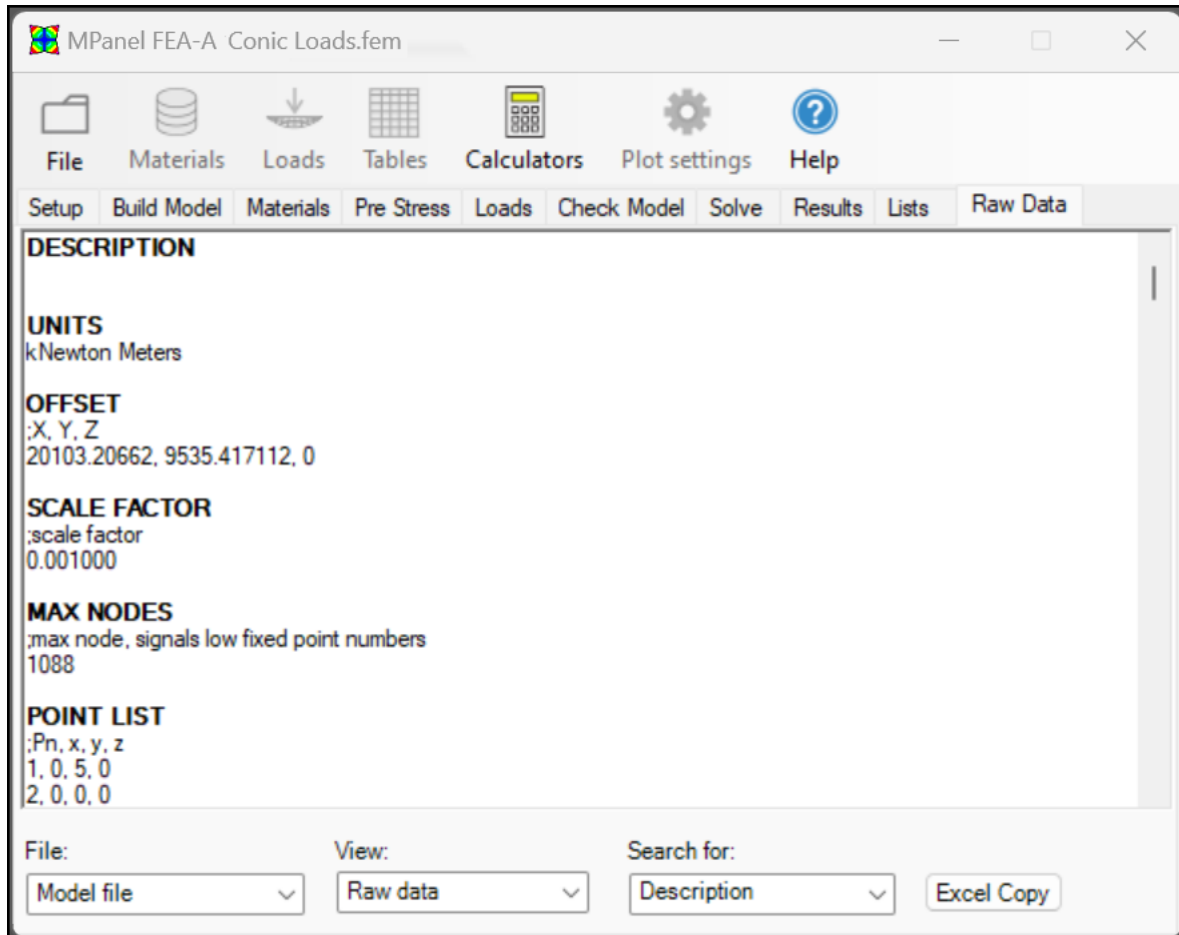
The model file - any solved output files can be viewed in a text window. They consist of the sections, which can be selected using the "Search for" box.

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 output file reactions can be 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

MAX NODES If present, specifies the maximum number of nodes, and that the nodes are arranged with the fixed points at the bottom of the list with low numbers. If not present then the fixed points are at the top of the list, with high numbers

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 pre-stress applied to the membrane, in LX and LY

PRESTRESS CABLE pre-stress applied to the cable

MEMBRANE PROPERTIES material properties for membrane

CABLE PROPERTIES stretch properties for the cable.

PRESTRESS CABLE ADJUSTMENT recalculates the cable prestress according to the catenary radius.

PRESTRESS STIFF ADJUSTMENT recalculates the stiff point back force according to the catenary radius and fabric prestress values.

LISTS records the elements in each list.

LOAD DESCRIPTION (X) records the description given for each load case.

POLYLINE NODES records the node numbers for each polyline

CANTILEVER POLES records the node numbers for each polyline which is defined as a cantilever pole.

CANTILEVER POLES PROPERTIES material properties for cantilever poles.

(where X is the load case number for load cases 2 through 12)

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

AVERAGE LINK TENSION the force in each group of links that make up a cable

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, for each polyline, and for any defined PLists.
SUM REACTION CENTER an point in space that the sum of reactions can be considered as acting through.
POLE VIRTUAL REACTIONS the XYZ force that the model exerts on the poles shown at the top of the pole.

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

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 Vaults

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

Video: **Building model to read into FEA**
Video: **Read in model and apply loads, solve and plot results**
Video: **Review results and copy to spreadsheet**

This video shows analysis of a more complex model using metric units:

Video: **FEA analysis of 3 bay barrel vault**

- Example Hypar Models

This shows a hypar with 4 catenary 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 modeling the hypar with the seams running in that direction with an anisotropic material.

Video: **FEA analysis of hypar sail**

The following videos show analysis of a several different hypar configurations - the related CAD files are linked below:

Rhino Drawing: **Floating Hypar**
AutoCAD Drawing: **Floating Hypar**
Rhino Drawing: **Hypar models**

AutoCAD Drawing: **Hypar models**

Video: **Analyze hypar with fixed connections**
Video: **Analyze hypar with corner links to fixed connections**
Video: **Analyze hypar with link poles and backstays**
Video: **Analyze hypar with corner links to link poles and backstays**
Video: **Analyze hypar with cantilever poles**
Video: **Analyze hypar with cantilever poles with adjustable corner links**
Video: **Analyze hypar with simulated cantilever poles**
Video: **Analyze hypar with simulated cantilever poles with adjustable corner links**
Video: **Analyzing multiple sails connected to a free pole and fixed connections**
Video: **Analyzing multiple sails connected to a free pole and cantilever poles**
Video: **Analyzing multiple sails connected to a pinned pole and cantilever poles**

- Example Cone tent models

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

Video: **Building model to read into FEA**
Video: **Read in model and apply loads, solve and plot results**
Video: **Analyse fabric lifting off umbrella frame**

This video shows analysis of a conic with pinned and cable restrained centre post. In this example we demonstrate using MLists to apply different pressures to winward and leeward sides of umbrella:

Video: **FEA analysis of conic with pinned centre post**

This video shows analysis of a dual conic with several different pressure bands:

Video: **FEA analysis of dual conic with complex pressure loads**

This video shows analysis of a conic to determine pole top reactions:

Video: **FEA analysis of conic pole top reactions**

This video shows analysis of a conic with a cable supported (flying) mast:

Video: **FEA analysis of conic with flying mast**

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.

In situations where the program is unable to resolve corner tensions correctly colouring the links and poles before making the model will allow simple allocation of materials and pre-stress values based on the Lists.

- **Simulated Cantilever Poles (previously Encastre poles)**

Encastre poles simulate cantilever poles that are 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 dependent 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 "Simulate Cantilever Poles using stiff nodes".
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 dependent 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.

- **Simulated Cantilever Poles adjustable links using stiff nodes (previously Encastre poles using adjustable links)**

The method described above for encastre pole is technically the correct method of simulation, as the FEA analysis starts in the "as installed" condition, with a back force on the pole tops. However this method does not show the pole top movement during installation.

To allow the pole top movement to be visible we can use a modeling entity called an adjustable link. This entity maintains constant tension during the solve for load case zero, and then behaves as a normal link for all other load cases.

So when this modeling method is used there is no back force applied to the pole tops, and the pole tops move in during the solve sufficiently to develop the required back force.

When solving for other load cases the existing solve for load case zero is automatically used as a starting point.

- **Link structure pairs**

A conic's top ring is usually modeled 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 beam**

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 modeling stability the links are pre tensioned, and small stabilizing 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.

- **Static beam**

A static beam is a polyline. Forces along the polyline are collected and presented at the 2 ends simulating a stiff beam fixed at both ends and is typically used when modeling marquees and similar pop-up and temporary tents.

- **Cantilever Pole and adjustable links**

Cantilever poles are fixed at the ground and free to move at the top and allow for multiple sail attachments. Cantilever poles are modeled using a 3D polyline with multiple vertices (e.g. 11). Multiple sails can be connected to the pole provided they are connected at one of these pole vertices.

A correctly balanced FEA model should have a back force specified at the pole sail attachments representing the force developed in the pole by bending during installation. However, use of this back force means that the bending of the pole during installation will not be shown. To allow the pole bending to be shown, instead of specifying a back force, we require an adjustable link that has constant tension during load case zero. This bends the pole during the solve for load case zero, and then behaves as a normal link for all other load cases. In practical terms the adjustable link is like a turnbuckle used to tighten the sail to its prestress condition.

So when this modeling method is used the pole tops move in during the solve sufficiently to develop the required back force.

When solving for other load cases load case zero is suggested as a starting point.

Video: **Analyze hypar with fixed connections**
Video: **Analyze hypar with corner links to fixed connections**
Video: **Analyze hypar with link poles and backstays**
Video: **Analyze hypar with corner links to link poles and backstays**

Video: **Analyze hypar with cantilever poles**
Video: **Analyze hypar with cantilever poles with adjustable corner links**
Video: **Analyze hypar with simulated cantilever poles**
Video: **Analyze hypar with simulated cantilever poles with adjustable corner links**
Video: **Analyzing multiple sails connected to a free pole and fixed connections**
Video: **Analyzing multiple sails connected to a free pole and cantilever poles**
Video: **Analyzing multiple sails connected to a pinned pole and cantilever poles**

Using Calculators

Several simple calculators are provided to help generate suitable values for material properties and loads. They can be used for direct entry of values into the material tables, or for calculating values for the databases. 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.

- **Cantilever Pole.**

This calculates the pole lengthwise stiffness (EA), flexural rigidity (EI) and section modulus (Z).

- **Simulated Cantilever Pole (previously 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.

The section modulus "Z" is also calculated, this is used to calculate the bending stress in an encastre pole. Note that encastre poles that were added to the database before version 13 will not have this value, and it should be added to the database if you want the pole bending stress to be calculated.

- **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.
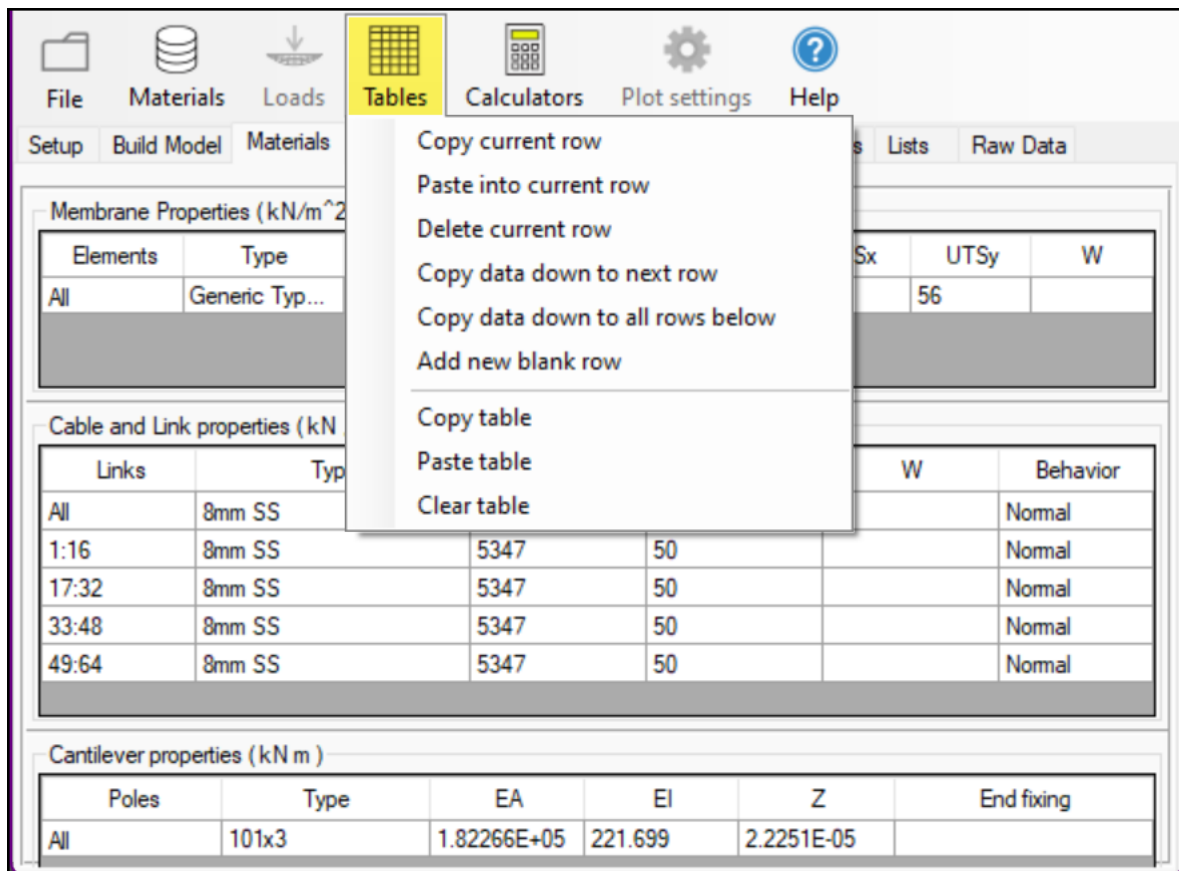
- **Unit conversion.**

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.

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 the databases

There are databases for:
 Membrane properties
 Cable properties
 Pole properties
 Cantilever Pole properties
 Simulated Cantilever Pole (previously called Encastre pole) properties
 Simulated Beam properties
 Loads.

Each database comprises:

 Standard entries
  these cannot be edited
  when used they have the word "Generic" shown with the name
  they may have a name in the format   xxx | yyy
   the xxx part is the description for imperial units, yyy is the description for metric units

 User entries
  these can be edited in the database editor
  and the values entered are automatically converted for use in other unit systems.

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 (click the + button), change (click the tick/check button), or delete (click the - button) materials. To assist with determining the material values calculators are available which can past the calculated value into the database record.

Note: when you add a database entry, the values are stored in a units neutral format, and then converted into the units that you have selected to use in the analysis. During the conversion they may be a small round off error in the values.

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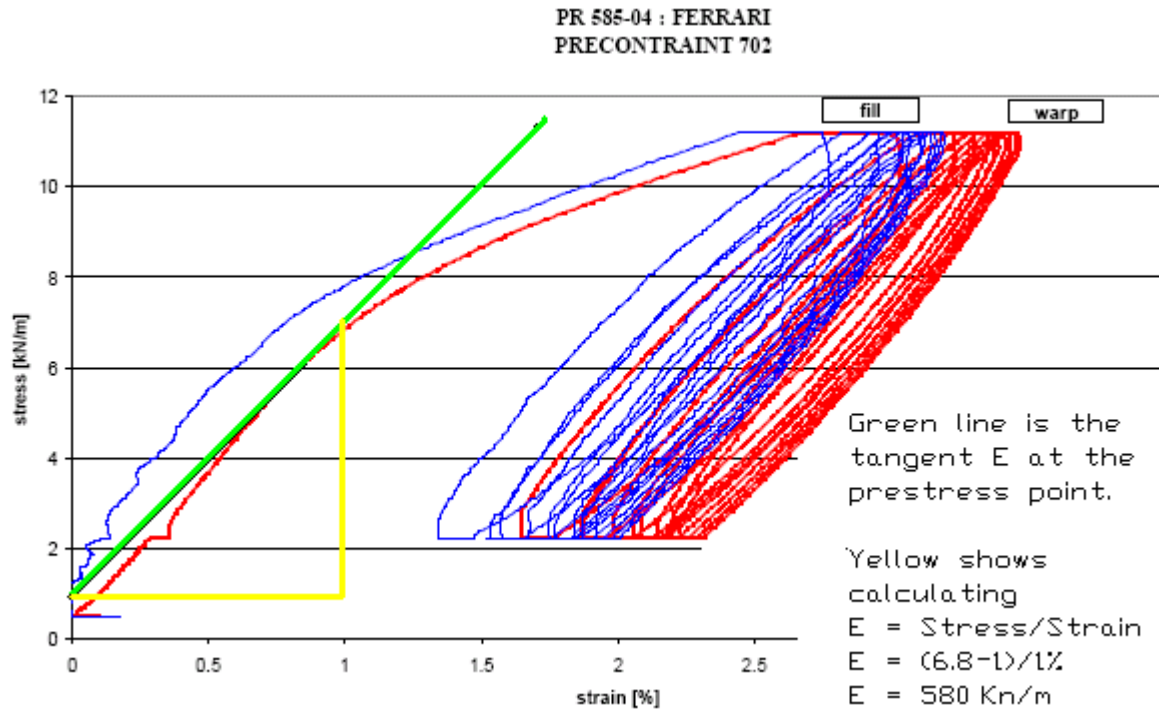
We can obtain the tangential E value from the graph in the manner shown, either using the "first loading" part of the curve, or the "after settling" part of the curve. You can see that the E value is similar for these two situations.

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 stretch graphs at various stress/strain ratios are available.

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.

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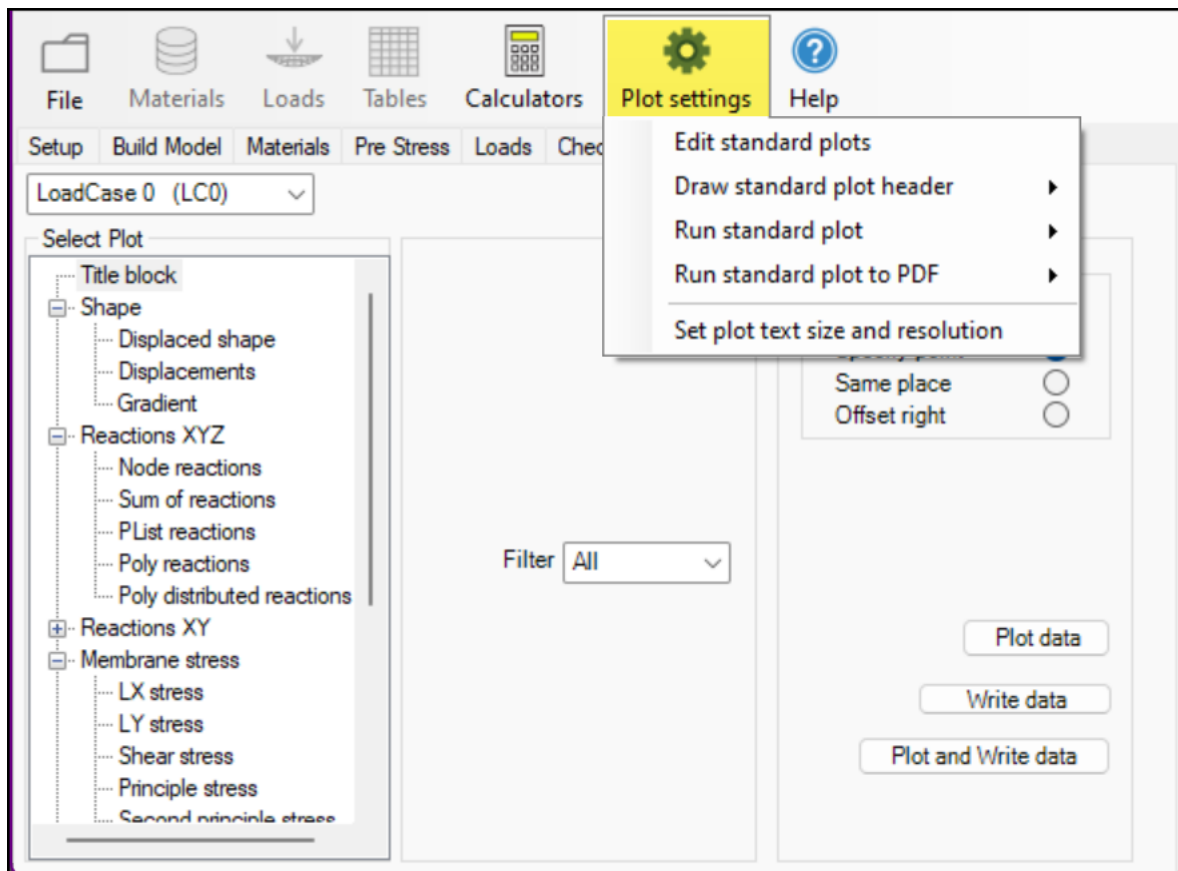
Standard reports let you set up multiple plots on a fixed horizontal spacing. This allows the generation of reports with the plots in a regular grid, with one row for each load case.

At least 2 standard reports should usually 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
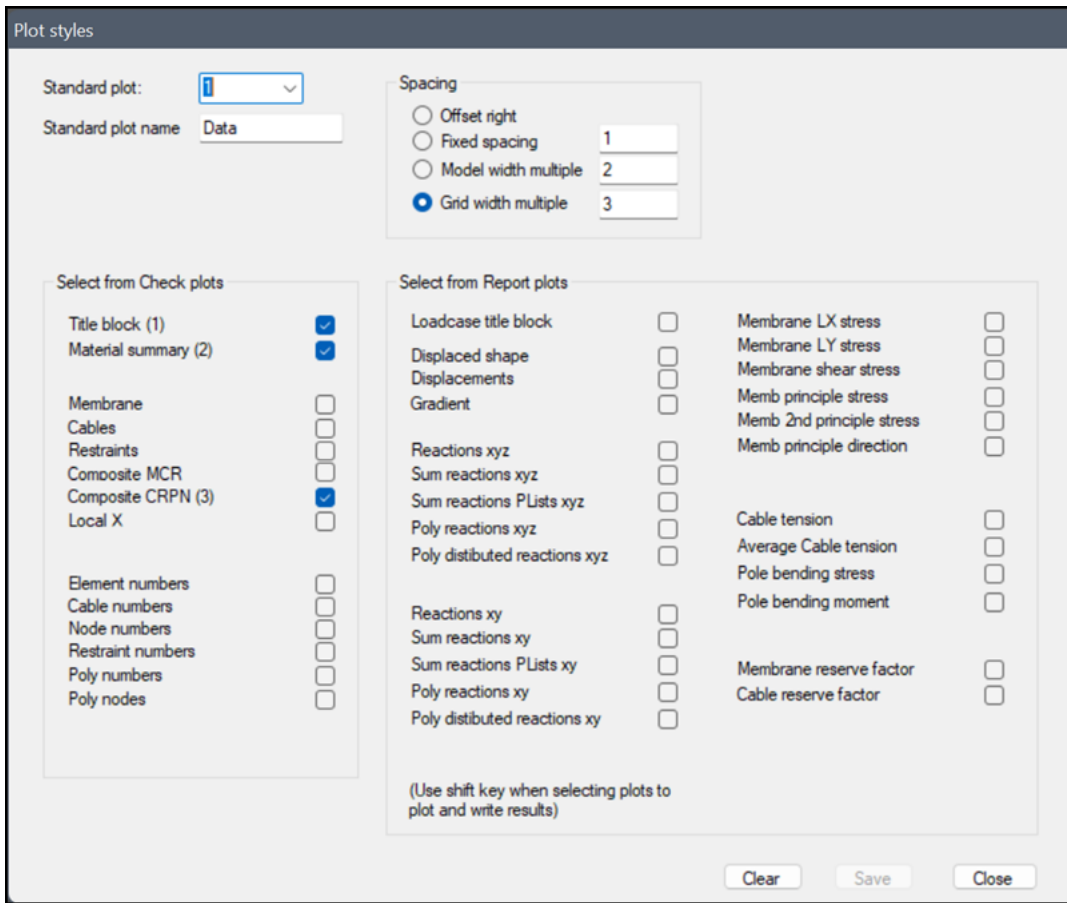
The plots can be edited by clicking on the menu item Edit Standard plots and filling in the form shown above.

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.

Grid reports

If the report type is specified as "grid", then the following additional features will help to make a structured grid style report:

 You can plot out a line of grid headers, that contain a descriptive title of each plot.
 When plotting the grid headers you will be prompted for the insertion point in the drawing

 When you plot the actual reports, they are drawn directly beneath the header, with a box grid drawn around each 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.

Standard reports can also be sent directly to a pdf file.

Video: **Grid reports**

License Agreement

MPanel Software Solutions LLC License Agreement

This is a legally binding software License agreement (the "License") between the following parties:

(1)   MPanel Software Solutions LLC of 3615 Queens Hwy., Carlsbad, NM 88220 U.S.A. ("MPanel", "we" or "us" or "Licensor"). We are a company registered in New Mexico with identification number 821926203; and
(2)   You (either as an individual or a single entity), being a user of the MPanel Software as defined below.

 PLEASE READ THIS LICENSE CAREFULLY.

IF YOU DO NOT ACCEPT THE TERMS OF THIS LICENSE, DO NOT INSTALL OR USE THE MPANEL SOFTWARE.

BY USING THE MPANEL SOFTWARE YOU AGREE TO BE BOUND BY THIS LICENSE.

MPanel Software means any software provided with this License, such as by way of download, on disk, flash drive or through any other media or format, including any software, code, interfaces or Content and whether owned by us or by a third party.

Content means any content, text, documents, fonts, graphics, images, video, footage, audio, media files, data, templates, clipart, photographs, names, images, trademarks, links, or similar.

Your Rights and Permitted Use

1.   You are licensed to use the MPanel Software as expressly set out in this License from your first use of the MPanel Software until termination of this License.

2.   No rights in the MPanel Software are sold to you; your right is solely to use the MPanel Software within the terms of this License. All rights not expressly granted to you are reserved to MPanel.

3.   The License granted to you is limited, non-transferable and non-exclusive. Except to the extent any rights cannot be excluded by law, you are permitted:

(i) if you are a private individual, to download, install, use and run for personal use, one (1) copy of the MPanel Software on a computer running Microsoft Windows ("Windows Computer") that you personally own or control at your location for each seat authorized by MPanel.

(ii) if you are a commercial enterprise, to download, install, use and run one (1) copy of the MPanel Software for use by a single individual on a Windows Computer that they use that you own or control at your location for each seat authorized by MPanel. A maximum of 10 seats can be authorized per license and separate licenses are required for each additional location you operate equipment owned by you.

(iii) if you are an educational institution, to download, install, use and run, one (1) copy of the MPanel Software for use by a single individual on a Windows Computer that they use that you or they own or control for each seat authorized by MPanel. Commercial use is not permitted.

4.   Except to the extent expressly allowed in this License:
a.   you are not permitted to run, or allow to be run, the MPanel Software over a network or any other means by which it can be used simultaneously on more than one computer; and
b.   you have no right to rent, lend, lease, sell, supply, transfer or distribute, transfer, redistribute or sub license the MPanel Software.
c.   the license is restricted to use for your own purpose and is not to be used for the sale of design services to other potential license holders.

5.   The MPanel Software is protected by United States and international copyright and other applicable laws. All title and intellectual property rights, including copyright, in the MPanel Software are owned by MPanel or its licensors.

6.   You must not use the MPanel Software in a way that may infringe the copyright of a third party, such as by copying, reproducing, adapting, publishing, issuing copies of, communicating, exhibiting, transmitting, or distributing copyright materials without permission of the copyright owner. It is your responsibility to ensure that your use of the MPanel Software does not infringe the rights of third parties; you should take any necessary legal advice in this respect.

7.   Where you use the MPanel Software to display or access Content, you acknowledge that the Content may be owned by third parties, may be protected by copyright and/or other intellectual property rights and may be subject to conditions of use. Other than as expressly set out in this License, you are not granted any rights to use such Content and no representation is made that any Content will continue to be made available to you. MPanel is not responsible for any third party Content and any access to it provided by MPanel is provided solely as a convenience and the provision of such access does not imply any endorsement by MPanel.

8.   The MPanel Software may be supplied in different versions or subject to further specific License terms. For example, the MPanel Software may be supplied:
a.   as a trial, locked, demonstration or other limited use version ("Trial Version"); or
b.   as a student, teacher, OEM/Bundle Purchase version or other version dependent on status or intended use ("Customer Specific Version"); or
c.   as part of a volume or maintenance License or as part of use of other services

provided by MPanel ("Volume License").

In the case of all Trial Versions, Customer Specific Versions, Volume Licenses or software supplied under other specific arrangements, then this License will still apply but shall be subject to any terms on which such MPanel Software is supplied including any terms set out on the website from which the MPanel Software is downloaded or on the packaging of the MPanel Software or otherwise as agreed with MPanel. In the case of Trial Software, this License will terminate after any period specified as part of the trial or demonstration. In the case of Customer Specific Software, this License will only take effect if and to the extent at the time of purchase you satisfy any stipulated status requirements; if you are in any doubt as to whether your status at the time of purchase entitles you to use a Customer Specific Version then please contact MPanel support for further information in advance of making a purchase.

9.   Except to the extent necessary for the normal and permitted use of the MPanel Software or to the extent that such right cannot be excluded by law or to the extent required to be permitted by licensing terms governing use of open source components that may be included with the MPanel Software:

a.   You have no right (and shall not permit any third party) to copy, adapt, amend, reverse engineer, decompile, disassemble, attempt to derive the source code of, create derivative works of, decrypt, modify, distribute or make error corrections to any aspect of the MPanel Software in whole or in part; and

b.   You have no right to a copy of any of the MPanel Software, or of any algorithms or code or other material relating to the MPanel Software.

10.   Where you are supplied with any updates, upgrades or new versions of the MPanel Software, the terms of this License shall apply to the use of such updates, upgrades or new versions unless new License terms accompany such updates, upgrades or new versions, in which case the new License terms will apply to all use of the updated, upgraded or new version of the MPanel Software. Nothing in this License shall require MPanel to supply you with updates, upgrades or new versions.

11.   You will comply with all laws and regulations when downloading, installing, and using the MPanel Software, including any laws and regulations specific to your location of download. You must not use the MPanel Software in any way to publish or transmit any illegal material or links to such material, nor to infringe the rights (including intellectual property rights) of any third party.

12.   MPanel (or its licensors) may suspend, remove, modify, or disable (or impose limits on) access to the currently available MPanel Software at any time without notice and without liability to you.

Consent to Use of Data

13.   During normal MPanel Software operation, standard computer information is passed to our server, including Hardware ID, Computer Name, and User Name. This information will be used by MPanel and its subsidiaries and affiliates solely for program authorization. This information will not be transferred to third parties. You agree that MPanel and its affiliates

may use any information you give to us as part of product support and other services provided to you, if any, related to the MPanel Software solely to improve products or to provide customised services or technologies to you and will not disclose this information in a form that personally identifies you.

Termination

14.	This License will terminate automatically on any material breach by you of its terms.

15.	On termination, you must cease all use of the MPanel Software and delete all copies in your possession.

16.	Any provision of this License which expressly or by implication is intended to come into or continue in force on or after termination of this License shall remain in full force and effect.

WARRANTIES AND LIMITS OF LIABILITY

17.	NOTHING IN THIS LICENSE SHALL OPERATE TO EXCLUDE OR LIMIT EITHER PARTY'S LIABILITY FOR:
a.	DEATH OR PERSONAL INJURY CAUSED BY ITS NEGLIGENCE; OR
b.	FRAUD; OR
c.	ANY OTHER LIABILITY WHICH CANNOT BE EXCLUDED OR LIMITED UNDER APPLICABLE LAW.
ALL TERMS OF THE LICENSE SHALL TAKE EFFECT SUBJECT TO THIS.

18.	LOCAL LAWS MAY APPLY TO PROTECT THE RIGHTS OF INDIVIDUALS WHO PURCHASE THE MPANEL SOFTWARE IN THEIR INDIVIDUAL CAPACITY OUTSIDE A COMMERCIAL ENTERPRISE, SUCH AS CONSUMERS. NOTHING IN THIS LICENSE SHALL OPERATE TO EXCLUDE OR REDUCE SUCH PROTECTION. IF YOU THINK THIS MAY APPLY TO YOU, PLEASE CONSULT A CONSUMER ADVICE ORGANISATION.

19.	THE MPANEL SOFTWARE ARE SUPPLIED TO YOU "AS IS" AND "AS AVAILABLE" WITH ANY AND ALL FAULTS AND DEFECTS AND WITHOUT ANY WARRANTIES BY MPANEL OR ITS LICENSORS OF ANY KIND.

20.	IT IS YOUR RESPONSIBILITY TO ENSURE THE MPANEL SOFTWARE PERFORM THE PURPOSE FOR WHICH THEY ARE REQUIRED AND WILL FUNCTION PROPERLY IN THE ENVIRONMENT IN WHICH THEY ARE TO BE USED. THEIR USE IS AT YOUR SOLE RISK AND YOU ARE RESPONSIBLE FOR ANY COST OF REPAIR, CORRECTIONS OR SERVICING. YOU ACKNOWLEDGE INSTALLATION OF THE MPANEL SOFTWARE MAY AFFECT THE USABILITY OF THIRD PARTY SOFTWARE OR THIRD PARTY SERVICES. . IT IS YOUR RESPONSIBILITY TO ensurE that any structure designed and built using the MPANEL Software is built to a satisfactory standard. In particular, the materials, corner connecting hardware, poles and ground footings should be sufficient to resist expected environmental loads such as wind and snow with an adequate safety margin. YOU ARE responsible for meeting any applicable building codes and regulations, obtaining any required building permits, and meeting any

applicable engineering specifications. Note that the MPANEL Software supplies no guidance or information on the matters mentioned in this paragraph at all.

21.   WITHOUT LIMITATION, PLEASE NOTE THAT NO REPRESENTATION OR WARRANTY IS GIVEN BY MPANEL OR ITS LICENSORS THAT:
a.   THE MPANEL SOFTWARE WILL MEET YOUR REQUIREMENTS; OR
b.   ANY INFORMATION OR DATA PROVIDED WITH THE MPANEL SOFTWARE IS ACCURATE, ERROR FREE OR COMPREHENSIVE; OR
c.   OPERATION OF THE MPANEL SOFTWARE WILL BE UNINTERRUPTED OR ERROR FREE; OR
d.   ANY CONTENT AVAILABLE FOR DOWNLOADING FROM US IS FREE FROM VIRUSES OR ANY OTHER CONTAMINATION OR DESTRUCTIVE FEATURES; OR
e.   YOU WILL ENJOY THE USE OF THE MPANEL SOFTWARE FREE FROM INTERFERENCE; OR
f.   THE MPANEL SOFTWARE WILL CONTINUE TO BE MADE AVAILABLE OR MAINTAINED OR SUPPORTED OR THAT ANY DEFECTS WILL BE CORRECTED; OR
g.   THE MPANEL SOFTWARE WILL BE COMPATIBLE WITH ANY THIRD PARTY SOFTWARE, THIRD PARTY SERVICES, THIRD PARTY EXTERNAL SERVICES OR MPANEL EXTERNAL SERVICES;
h.   THE MPANEL SOFTWARE OR MPANEL EXTERNAL SERVICES OR THIRD PARTY EXTERNAL SERVICES WILL BE AVAILABLE OR ACCESSIBLE IN ANY LANGUAGE OR FROM ANY LOCATION OR ARE APPROPRIATE FOR USE IN ANY LOCATION.

22.   TO THE EXTENT POSSIBLE UNDER THE APPLICABLE LAW, ALL CONDITIONS, WARRANTIES OR OTHER TERMS WHICH MIGHT HAVE EFFECT BETWEEN THE PARTIES OR BE IMPLIED OR INCORPORATED INTO THIS LICENSE OR ANY COLLATERAL CONTRACT, WHETHER BY STATUTE, COMMON LAW OR OTHERWISE, ARE HEREBY EXCLUDED, INCLUDING ANY IMPLIED CONDITIONS, WARRANTIES OR OTHER TERMS AS TO SATISFACTORY QUALITY, FITNESS FOR PURPOSE OR THE USE OF REASONABLE SKILL AND CARE.

23.   MPANEL AND ITS LICENSORS WILL NOT BE LIABLE FOR ANY LOSS OR DAMAGE CAUSED BY A DISTRIBUTED DENIAL-OF-SERVICE ATTACK, VIRUSES OR OTHER TECHNOLOGICALLY HARMFUL MATERIAL THAT MAY INFECT ANY SITE, COMPUTER EQUIPMENT, COMPUTER PROGRAMS, DATA OR OTHER PROPRIETARY MATERIAL DUE TO YOUR USE OF THE MPANEL SOFTWARE.

24.   MPANEL AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY LOSSES OR DAMAGES WHICH MAY BE SUFFERED BY YOU (OR ANY PERSON CLAIMING UNDER OR THROUGH YOU), WHETHER THE SAME ARE SUFFERED DIRECTLY OR INDIRECTLY OR ARE IMMEDIATE OR CONSEQUENTIAL, AND WHETHER THE SAME ARISE IN CONTRACT, TORT (INCLUDING NEGLIGENCE) OR OTHERWISE HOWSOEVER, WHICH FALL WITHIN ANY OF THE FOLLOWING CATEGORIES: I) SPECIAL DAMAGE EVEN THOUGH MPANEL WAS AWARE OF THE CIRCUMSTANCES IN WHICH SUCH SPECIAL DAMAGE COULD ARISE; II) LOSS OF PROFITS; III) LOSS OF ANTICIPATED SAVINGS; IV) LOSS OF BUSINESS OPPORTUNITY; V) LOSS OF GOODWILL; OR VI) LOSS OR CORRUPTION OF DATA.

25.   THE TOTAL LIABILITY OF MPANEL, WHETHER IN CONTRACT, TORT (INCLUDING NEGLIGENCE) OR OTHERWISE AND WHETHER IN CONNECTION WITH THIS LICENSE OR ANY

COLLATERAL CONTRACT, SHALL IN NO CIRCUMSTANCES EXCEED THE TOTAL SUMS PAYABLE BY YOU TO US FOR USE OF THE MPANEL SOFTWARE.

26.   YOU AGREE THAT, IN ENTERING INTO THIS LICENSE, EITHER YOU DID NOT RELY ON ANY REPRESENTATIONS (WHETHER WRITTEN OR ORAL) OF ANY KIND OR OF ANY PERSON OTHER THAN THOSE EXPRESSLY SET OUT IN THIS LICENSE OR (IF YOU DID RELY ON ANY REPRESENTATIONS, WHETHER WRITTEN OR ORAL, NOT EXPRESSLY SET OUT IN THIS LICENSE) THAT YOU SHALL HAVE NO REMEDY IN RESPECT OF SUCH REPRESENTATIONS AND (IN EITHER CASE) MPANEL SHALL HAVE NO LIABILITY OTHER THAN IN ACCORDANCE WITH THE EXPRESS TERMS OF THIS LICENSE.

Government End Users

27.   The MPanel Software and related documentation are "Commercial Items", as that term is defined at 48 C.F.R. §2.101, consisting of "Commercial Computer Software" and "Commercial Computer Software Documentation", as such terms are used in 48 C.F.R. §12.212 or 48 C.F.R. §227.7202, as applicable. Consistent with 48 C.F.R. §12.212 or 48 C.F.R. §227.7202-1 through 227.7202-4, as applicable, the Commercial Computer Software and Commercial Computer Software Documentation are being licensed to U.S. Government end users (a) only as Commercial Items and (b) with only those rights as are granted to all other end users pursuant to the terms and conditions herein. Unpublished-rights reserved under the copyright laws of the United States.

Miscellaneous

28.   This License and the License granted pursuant to this License are personal to you and except where expressly permitted you shall not assign the benefit of or any interest or obligation under this License.

29.   Apart from MPanel's licensors and authorised distributors, a person who is not a party to this License has no right under the Contracts (Rights of Third Parties) Act 1999 or otherwise to enforce any term of this License. The consent of any third party is not required for any variation (including any release or compromise of any liability under this License) or termination of this License.

30.   All disclaimers of warranties, limitations and exclusions of damages, rights to be indemnified by you and dispute resolution shall apply to MPanel's licensors, suppliers, vendors, authorised distributors and resellers, all of which shall be deemed express third party beneficiaries thereof.

31.   If any court or competent authority finds that any provision of this License (or part of any provision) is invalid, illegal or unenforceable, that provision or part-provision shall, to the extent required, be deemed to be deleted, and the validity and enforceability of the other provisions of this License shall not be affected.

32.   Except where expressly stated, no variation of this License shall be effective unless it is in writing and signed by the parties (or their authorised representatives).

33. No failure or delay by a party to exercise any right or remedy provided under this License or by law shall constitute a waiver of that or any other right or remedy, nor shall it preclude or restrict the further exercise of that or any other right or remedy.

34. This License (including any documents referred to in the License) contains the whole agreement between the parties relating to the subject matter hereof and supersedes all prior agreements, arrangements and understandings between the parties relating to that subject matter.

35. a) References to MPanel and you includes their permitted successors and assigns; b) references to statutory provisions include those statutory provisions as amended or re-enacted; c) references to one gender includes a reference to the other genders; d) references to "including" or "includes" shall be deemed to have the words "without limitation" inserted after them; e) words in the singular include the plural and those in the plural include the singular; and f) a person includes a natural person, corporate or unincorporated body (whether or not having separate legal personality) and that person's personal representatives, successors or permitted assigns.

36. This License and any connected dispute shall be governed by and construed in accordance with the laws and courts of New Mexico, United States. Where any translations of this License are provided, the content of the English version shall take priority in cases of conflict, to the extent not prohibited by any local laws.

37. You agree that You will not export or transmit the MPanel Software or any Applications, directly or indirectly, to any restricted countries or in any manner that would violate United States laws and regulations as shall from time to time govern the license and delivery of technology abroad by persons subject to the jurisdiction of the United States government, including the Export Administration Act of 1979, as amended, and any applicable laws or regulations issued thereafter.

Contact List and Technical Support

Technical support is available to customers who:

 Have a valid annual technical support contract
or purchased MPanel FEA within the previous 12 months
or are evaluating an MPanel FEA demo product.

In general, it is best to send the drawing and the fem file showing the question as an attachment to the email, as this will allow us to respond specifically to your question. We will usually mark up the drawing and send it back to you or we will send a short video which addresses the question.

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

[Video Tutorials](#)