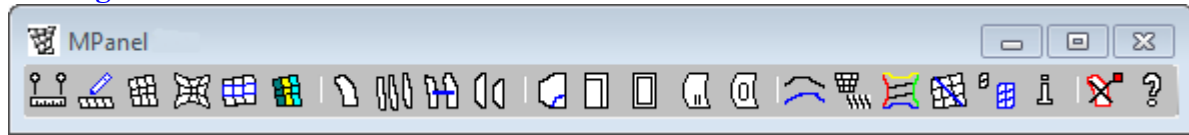


Getting Started



MPanel is a very flexible tool, used in different ways on different jobs. On your first use of MPanel you will be asked to set the units system that you use in your drawings. After that a typical job could go through the following steps:

- Start your Cad program and MPanel.
- Draw the model in your Cad program using specified entities.
- Relax the model.
- Produce visualizations for client approval.
- Estimate a materials bill.
- Do a stress estimation.
- Create the panels.
- Compensate the panels.
- Apply a seam allowance.
- Productionise the panels.
- Output the panels.

Start your Cad program and MPanel

To use MPanel you need to first start your Cad program and then MPanel. MPanel will appear as a floating toolbar, with buttons for each tool.

Basic MPanel use.

Left click on a toolbar button will run that tool with the current options.

Right click on a toolbar button will open the Options screen for that tool.

Holding the mouse over a button will show a drop down menu of the main tool options.

Left click on a drop down menu option will open the Options screen at the place for that menu option.

Help is available for each tool on the drop down menu as well.

Draw the model in your Cad program using specified entities

An outline drawing is made using 3D polylines as fixed system points, lines as guy lines, and meshes as surfaces to relax. The [Model Builder](#) tool can help with this. Often the modelling process starts with drawing a 3dpolyline around the fixed points, and building the model up from that. Also models can be built directly from the model library in the Model Builder.

Relax the model

Either [Relax Fixed Edge Mesh](#), [Relax Cable Edged Mesh](#) or [Relax Joined Edge Mesh](#) is used to relax the structure towards convergence. A few iterations are tried first to ensure that the model is correct, and then repeated relaxations until the model converges.

Produce visualisations for client approval

The relaxed model can be shaded or rendered in your Cad program. The drawing can be sent to other packages for walk-through's, and simulations, either in the native drawing format or as a DXF, drawing exchange format.

(If you are using AutoCAD, the drawing can be Published or Published to Web)

Estimate a materials bill.

The [Info](#) tool will provide the total mesh area and cable lengths by using the "Whole Model Report" option.

Do a stress estimation

MPanel is not a stressing package, but it does supply enough information for stress estimation to be made, as described in the [Stress Estimation](#) section. The "Whole Model Report" can also give useful indications of the possible stresses and cable tensions.

Create the panels

The panels can be created directly from the mesh, using [Mesh-Panel](#), or they can be defined by [cross sections or geodesic](#). These can be panelled directly using [Flat Panel](#), or they can be converted to another mesh with [mesh-lines](#), and then panelled.

Arrange the panels.

MPanel draws the panels directly under the mesh or cross sections they came from, to ensure that the panel location in the build is correctly known. Often it is useful to arrange the panels in an ordered set, horizontally or vertically, with the [Arrange Panels](#) tool.

Compensate the panels

The panels are shrunk by a defined amount to introduce pre-stress into the structure. The amount of [compensation](#) depends upon the material properties of your fabric.

Apply a seam allowance

A [seam allowance](#) is added to the panel, and optionally, sewing balance marks, guidelines, and slits.

Productionise the panels

Before the panels are sent to production they will need annotation, warp orientation lines and other information adding. This is done by the [Panel Production](#) tool.

Output the panels

If the panels are going straight to a cutter then the panel may need modifying so that the cut information is distinct from the annotations, etc. The [Panel Output](#) tool takes care of these modifications.

Glossary

Fabric Terms

Shear strain

The amount that a fabric is distorted such that a square in the fabric becomes a parallelogram or rhomboid

Warp

The direction along the length of a fabric

Weft

The direction across the width of a fabric

Tensile Structure Terms

Anticlastic

A 3D shape similar to a saddle, with positive and negative principle curvatures

Cross section

The curve produced by cutting a 3D surface with a plane, often a vertical plane

Geodesic

The shortest path between two points on a curved 3D surface

MPanel Terms

Balance marks

Marks placed on the panel edges to help with lining up during manufacture

Coincident

The placing of one node directly on top of another node

Compensate

The deliberate shrinking of panels, to introduce pre stress when they are stretched to fit

Converge

Move towards a final stable shape

Interpolated polyline

A polyline approximation to a spline

Iteration

One run through the relaxation program.

Typically 100 iterations are done in one go.

Links

A structural member that has the properties of length and tension only.

Merge

The process of joining together two adjacent panels, that typically will not fit together exactly

Edge Mesh

A mesh drawn between 4 edges. The edges can be any non closed Cad entity

Corner Mesh

A mesh drawn between 4 corners

Fixed Edge Mesh

A mesh with all edges assumed to be fixed

Cable Edge Mesh

A mesh with all edges assumed to be cables

Joined Edge Mesh

A mesh with all edges assumed to be cables, unless they join another mesh at the corners.

Productionise

The preparation of a panel for production

Relax

The process of allowing the mesh and links to move in 3D space until a force balance is found at each node

Tick marks

Marks placed on the panel edges to help with lining up during manufacture

Slits

Cuts made in a seam to allow the folded seam to follow a curve

AutoCad specific terms.**Blocks**

A collection of entities that are to be regarded as a single object

Entities

A visible geometry object in AutoCAD

Nodes

The defining points in an entity, ie for a line the start and end points

Polylines

An entity composed of one or more connected line segments treated as a single object

3D polyline

A polyline with nodes in 3 dimensions of x, y and z

LW polyline

A polyline with nodes in 2 dimensions x and y, drawn in a 3 dimensional plane

2D polyline

A polyline with nodes in two dimensions, x and y, with a fixed z elevation

Spline

A smooth curve passing through a set of nodes

Visualization

The process of rendering, shading and viewing from different viewpoints.

Rhino specific terms.**Blocks**

A collection of entities that are to be regarded as a single object

Entities

A visible geometry object in Rhino

Point

The defining points for an entity, used to adjust the entity or to OSnap another entity to.

Or a point entity drawn in 3D space

Polylines

A curve with more than two points, with degree =1.

Spline

A smooth curve passing through a set of points

Visualization

The process of rendering, shading and viewing from different viewpoints.

General Settings

On the general settings tab we can define how MPanel draws entities in the your Cad drawing. You can make settings for:

- New Object Layer
- Old Object Colour
- New Object Colour
- Detail Colour
- Default Units
- Selection Filter

New Object Layer

Defines the layer that MPanel will draw new entities onto. If the layer doesn't exist, it will be created. Choice of:

Named Layer uses the layer name specified. Often set to "Development" or "MPanel"
Predefined Layers uses different predefined layers for Building ,Relaxing, Panelling, etc.
Preserve Object layers keeps the same object layer for meshes, lines, or derived objects.
Current Layer uses the current CAD drawing layer

Old Object

Defines the colour to identify the old object. Often set to "Red"
Can also be set to delete the old object.

New Object Colour

Defines the colour to draw the new object. Often set to "Current"
Can also be set to "By Layer" , so the display colour will be defined by the New Object Layer.
Can also be set to "Preserve colour" so the new object colour is the same as the object it was derived from.

Detail Colour

Defines the grey to draw panel detail, and extra cross sections in. Often set to a mid .
Additionally detail used to define marks during seaming can be coloured Orange.

Default Units

Allows you to set the default values, and prompts, to the unit system that you use. Select the units and click on "Set". If you have previously saved a set

Selection Filter

Allows you to filter the selection set by layer name. This can be useful when working in a dense drawing with a lot of extra details (columns, nuts, bolts etc) that are not required by MPanel.

Notes for advanced users:

If you normally use layer colours, MPanel will maintain them if you set:

New Object Layer = Preserve Layers
New Object Colour = ByLayer

For the fastest drawing of complicated models, use:

New Object Layer = Current
New Object Layer = Current
Colour Old Object = OFF

The predefined layers can be tailored by hand editing the Mpanel User Setting.ini file, in the users documents directory. By default they are:

Model Builder = "MP_Model"
Relaxed Models = "MP_Relax"
Basic Panels = "MP_Panel"
Compensated Panels = "MP_Comp"
Seamed Panels = "MP_Seam"
Stamped Panels = "MP_Stamp"
Utility Tools = "MP_Utility"

Contact List and Technical Support

Technical support is available to customers who:

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

In general, it is best to send the drawing 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

Support questions should be sent to:

support@mpanel.com

If you have a support contract quote the number on the subject line as this marks the email for higher priority.

Alternatively you can contact members of the support team directly.

Andrew Askwith, Meliar Design, UK

andrew@meliar.com

Tim Akes, CAD Effects LLC, USA

takes@cadeffects.com

Alan Stewart, Structureflex, Australia

alan@structureflex.com.au

Quick Reference (entities)

MPanel places special meaning on some drawing entities to define the model for relaxation. These entities are:

Mesh

A mesh is treated as a surface to relax. Meshes are usually created using the MPanel Model builder

3D Polyline

A 3D polyline is used to define fixed points on a relaxation. Mesh points or link ends that are coincident with the 3D polyline vertices are fixed during the relaxation. Fixed points are typically anchor points, tops of masts, or fixed edges on the relaxations.

Line

A line is used to define a link, which could represent a guy, mast or a section of a cable. Several links can be joined together to define a structure.

AutoCAD specific notes

Polylines, LW Polyline, 2D Polylines are flat polylines that are only used for panels or panel details. They are not recognised as entities in the 3d model.

A mesh can also be drawn directly in AutoCAD using commands such as EdgeSurf or 3D Mesh.

There is also limited read support for polyface and sub-dividable meshes, as long as they are regular.

(Sub-dividable mesh's where introduced in AutoCAD 2010, AutoCAD refers to them just as "Meshes")

Rhino specific notes

A mesh can also be drawn directly in Rhino using commands such as MeshPlane. Meshes have to be regular meshes, ie an ordered set of rectangular mesh elements, like a plane mesh.

Polylines are a curve with a degree of 1, and more than two points

Lines are a curve with a degree of 1 and exactly two points.

Quick Reference (tools)

[Site Layout](#)

Use this to help convert site measurements into a 3d draing in your CAD system

[Model Builder](#)

Set model options and click on "Model Builder" to create the prototype drawing. Move nodes around to meet your boundaries; you are then ready to relax the model. Or use the Model Library to draw a complete model to your specifications.

[Relax Fixed Edge Meshes](#)

Select some meshes. Run the tool to relax the mesh towards a stress balanced surface, with fixed edge boundaries.

[Relax Cabled Edge Meshes](#)

Select some meshes and some 3D polylines that define the structure fixed points. Run the tool to relax the meshes including the cable edges.

[Relax Joined Edge Meshes](#)

Select some meshes and some 3D polylines that define the structure fixed points. Run the tool to relax the meshes including the cable edges.

[Model Visualization](#)

Shows graphical model information such as gradient, curvature, sun shadow, etc.

[Panel between Polylines](#)

Select two 3D polylines, and run the tool to produce the flat panel development. Optionally select a set of 3D poly's and run the tool to produce a single merged panel.

[Panel Mesh](#)

Select a mesh. Click on "Panel Mesh" to produce a series of flat panel developments. Optionally the number of panels can be specified.

[Merge Panels](#)

Select panels "A" and "B" that are alongside each other. Clicking on "Merge Panels" will produce a merged panel created by joining "A" and "B" together.

[Arrange panels](#)

Arranges panels into an ordered set, optionally between lines representing the roll width.

[Panel Detail](#)

Allows panel detail, such as corner plates or graphic markers, to be added or edited in the panel. Draw the detail as polylines, then select the detail and the panel and Click on "Panel Detail"

[Compensate Panel](#)

To shrink the panel slightly to introduce pre stress. Select the panel and click on "Compensate Panel".

[Seam Allowance Panel](#)

To add material for a seam, and optionally add guidelines and guide marks to the panel. Select the panel and click on "Seam allowance".

[Panel Production](#)

To orient the panel, stamp it with a panel name and warp line, and create a node list. Select the panel and click on "Stamp Panel".

[Panel Output](#)

To make a panel suitable for direct cutting on a plotter/cutter. Select the panel and click on "Panel Output".

[Convert Polys](#)

Convert polylines, arcs, etc to 3D polys and change 3Dpoly properties. Select the entities and click on "Convert Polys"

[Convert Mesh/Poly](#)

Convert between a mesh and a set of polylines, and vica versa. Select the entities and click on "Convert Mesh/Poly"

[Cross Section](#)

To make cross section on meshes, often to panel between. Draw cutting lines above the meshes. Select the cutting lines, and the meshes, and click on "Cross Section".

[Change Mesh Density](#)

To change the msurf and nsurf of a mesh to reduce the time it takes to create a high definition model.

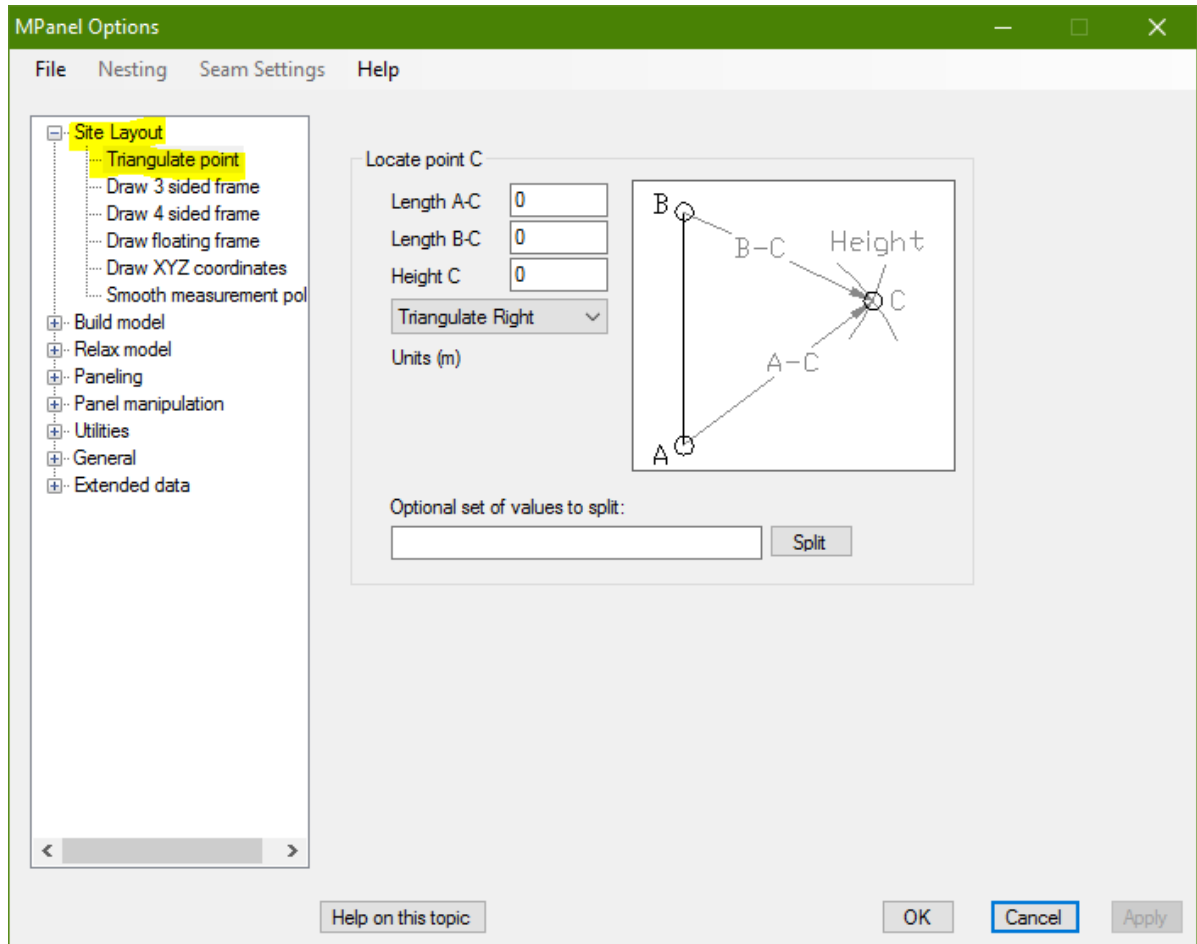
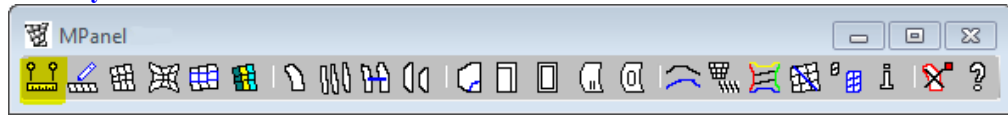
[Information Button](#)

Select a Mesh, 3D polyline, pair of touching 3D polylines, or a Panel and click on "Information". Or select the complete 3D model, and specify a "Whole Model Report".

[Delete Old Objects](#)

To delete the last old objects, ie the entities that MPanel read in the drawing before running a tool.

Site Layout



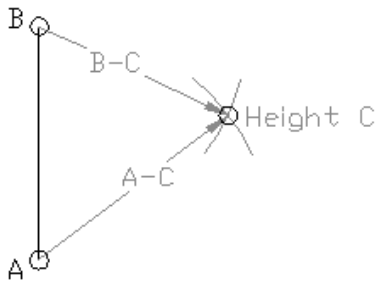
Site layout is a set of tools that assist with getting the site measurements converted into an outline CAD drawing. It can perform the following functions:

- Triangulate a point
- Draw 3 sided frame
- Draw 4 sided frame
- Draw 4 sided floating frame
- Draw XYZ coordinates
- Smooth machine measured data

Triangulate a point

A common measuring method is to define a point by measuring straight line distances from two known points, and measuring the height of the unknown point. At it's simplest this can be done with a tape measure and level, or electronic distance measuring equipment can be used.

We define points A and B as known, and supply lengths A-C B-C and height C. Running the tool and selecting points A and B in the drawing will create a an outline 3d poly to point C. You can select if this point is to the right or left of line A-B

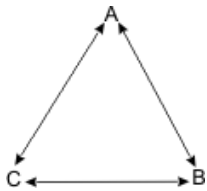


This process can be repeated many times to build up a 3d frame image from a set of site measurements. A set of site measurements would often be recorded in a spreadsheet, a facility to split a line containing the 3 values is supplied.

When the process is repeated many times, it is good practise to make the final measurements to a pair of points that have a known separation. Measuring this separation in the drawing will give an indication of the accumulated measurement errors.

Draw 3 sided frame

We define a triangle with points A, B, C and enter values for lengths A-B, B-C, C-A and heights of A, B, C. The program will then draw the triangle as a 3d poly frame. This can be used as the fixed points for a mesh model

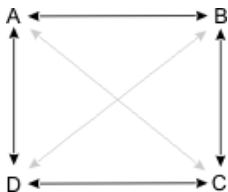


If A, B, C are poles, the most accurate measurements are obtained at the pole tops. This allows for any pole lean or manufacturing errors. Alternatively you can make the measurements at the pole bottoms.

The "Check Measurements" button will confirm that the measurements are consistent, ie that they can produce a valid triangle.

Draw 4 sided frame

We define a quadrilateral with points A, B, C, D and enter values for edge lengths A-B, B-C, C-D, D-A and diagonals A-C, B-D and heights of A, B, C, D. The program will then draw the quadrilateral as a 3d poly frame.



If A, B, C, D are poles, the most accurate measurements are obtained at the pole tops. This allows for any pole lean or manufacturing errors. Alternatively you can make the measurements at the pole bottoms.

The "Check Measurements" button will confirm that the measurements are consistent, ie that they can produce a valid quadrilateral. Because we have one more measurement than is strictly necessary to define the quadrilateral, we can also report the measurement accuracy.

Draw 4 sided floating frame

In some situations it is not possible to measure the heights of the 4 points A, B, C, D. It is still possible to define a frame

that will fit between the points, although the frame will not necessarily be oriented to match the points. So a membrane designed to fit the frame will also fit the A, B, C, D site points. It is also necessary to define which is the higher of the two diagonals.

The "Check Measurements" button will confirm that the measurements are consistent, ie that they can produce a valid quadrilateral.

Draw XYZ coordinates

Electronic measuring equipment, such as "Total Stations" or GPS survey equipment, can produce a list of X, Y, Z coordinates for the system points. The program can take in a list of these points and will draw a 3d poly frame joining the points.

The coordinates will usually be pasted from the windows clipboard. They can be copied from a spreadsheet, a text document, an email, etc. The program will use any non numeric character as a separator, so tabs, spaces, etc can be used. Optionally the 3d poly can be closed.

Any number of 3d poly frames can be drawn in one go, by separating the point data sets by at least 1 blank line.

Here is an example of a set of coordinates, representing the bottom frame and centre pole of a conic, that can be cut and pasted into the tool options:

```
Base poly
1      519.2511      -24.7030      2.0687
2      512.2049      -20.8973      2.700
3      514.6306      -15.5924      2.500
4      522.7164      -10.4028      2.1742
5      535.3071      -15.5924      0.0000
6      531.2642      -22.1659      1.5940
7      519.2511      -24.7030      2.0687
```

```
Centre pole
1      522.4318      -17.7856      5.3685
2      522.4318      -17.7856      0.0000
```

Smooth machine measured data

Electronic measuring equipment using mechanical, acoustic or laser measuring can produce a polyline trace of existing framework that the fabric needs to fit.

However there is often a "jitter" in the trace polyline caused by operator error / measurement inaccuracies.

This tool takes a slightly jittery polyline and creates a smooth curve through it. The curve can be:

- Nurbs curve (spline)
- Nurbs curve that lies flat in a plane
- Arc

For nurbs curves the number of smoothing points can be set.. a lower number gives a smoother curve. Often the ends of the curves must be preserved exactly to meet up with other curves, there is an option for this.

With high frames it is common to trace the underside of the frame, and then need to offset the curve to obtain the fabric touch line. An offset can be specified if the curve lies in a plane or is an arc.

- Create Mesh from Surface

Draw 3D Polyline

Model Builder will draw a 3D Polyline based on user input points.
(In AutoCAD this is an encapsulation of the 3D poly command)

Draw Edge Mesh

Model Builder will draw a mesh based on four boundary edges, which can be any curve like entities that join together at their ends. The density of the mesh is set in the options under Nsurf and Msurf. The edge first chosen defines the weft direction of the mesh, so the warp direction runs away from that edge. If panels were produced directly from the mesh then Msurf is one more than the number of panels in the mesh, and Nsurf is the number of points on the edge of each panel.

There is an option to draw a 3d poly along each mesh edge to fix the edges during relaxation. Any 3dpolys not needed (cable edges) can then be deleted.

Normally the edges defining a mesh should exactly at the corners. If there is a small gap the program will offer to "ignore" the gap and continue building the mesh. The user must then inspect the mesh to see if the corner error is acceptable.

To draw a mesh based on three boundary edges, skip the top edge when selecting the edges. You will be prompted to use the Enter key to skip the top edge and make a triangular mesh.

To draw a mesh based on two boundary edges, skip the left and right edges when selecting the edges, by using the Enter key when prompted for those edges

Normally a smooth surface mesh, based on a coons patch is used. There is an option to use a simpler mesh shape, called an interpolated mesh.

There is an option to fit a surface exactly to 4 edges, and smoothly though another poly called the surface poly (or scribble poly) with defines some internal points'

This can be used when the four edges of a surface have been traced with a mechanical digitiser, and then the surface shape has been mechanically traced as well.

The fitted surface is a B spline surface defined by a specified number of control points. The smaller the number of control points the smoother the surface will be, with more control points the surface will follow the internal points more closely. Often just 2 control points in the N and M directions are sufficient.

Draw Corner Mesh

Model Builder will draw a mesh based on four corners specified as user input points in the drawing. The Cad text window will prompt for the corners in order. The mesh is drawn such that the first two points define the weft direction and the warp direction runs away from those points. The density of the mesh is set in the options under Nsurf and Msurf. If panels were produced directly from the mesh then Msurf is one more than the number of panels in the mesh, and Nsurf is the number of points on the edge of each panel.

A square mesh of specified base width can be drawn by using the "Enter" key when prompted for the second user input point.

A triangular mesh can be drawn by using the "Enter" key when prompted for the forth user input point.

Draw Hypar (Sail Tent)

Model Builder will draw a flat sail as a mesh with system points on a 3D polyline between four corners specified as user input points in the drawing. The Cad text window will prompt for the corners in order. In addition a link representing a corner guy can be included at each corner. The corner link can simulate a hardware fitting like a D ring or EZ glide, or a corner plate. [See here](#) for a description of links that simulate the corner hardware.

A square mesh of specified base width can be drawn by using the "Enter" key when prompted for the second user input point.

A triangular mesh can be drawn by using the "Enter" key when prompted for the forth user input point.

Draw Conic (Cone Tent)

Model Builder will draw a conic with several meshes, a top ring and system points. The model is defined by option parameters including Base width, Ring diameter, and Model height. The model is drawn in at a user defined insertion point, with even dimensions all around. The system points and mesh corners can then be dragged or stretched to suit the model requirements that you have. The option Swap Warp/Weft will draw the mesh with the warp and weft directions swapped. The option "With fixed Edges" will draw in a series of 3DPolys for the fixed edges, and snap the mesh edges to them.

Sometimes you will need to create several prototype models to add together in a modular fashion to make your model. To adjust the prototype system points to suit your model, select the meshes and the system point line and drag the corner nodes to their new location.

(In AutoCAD, you can use the Stretch command to move the top ring and mesh attachments all at once, or you can select the nodes you want to move)

(In Rhino, you can turn the control points on and then select all the points you want to move)

Draw Pole Tent

Model Builder will draw a pole tent with several meshes and system points. The model is defined by option parameters including Base width, Number of meshes, and Top elevation. The model is drawn in at a user defined insertion point, with even dimensions all around. The system points and mesh corners can then be dragged or stretched to suit the model requirements that you have. The option Swap Warp/Weft will draw the mesh with the warp and weft directions swapped. The option "With fixed Edges" will draw in a series of 3dPolys for the fixed edges, and snap the mesh edges to them.

Fix Mesh Edges

Model Builder takes some meshes and 3Dpolys, and "fixes" the edges of the meshes to the 3Dpolys. There are 3 sub programs, which use the following rules:

Fix mesh edges in straight line between poly nodes. (used to fix mesh edges in straight lines)

If some mesh nodes are snapped to adjacent poly nodes (often the mesh nodes are the corners)

a new poly is drawn straight between the poly nodes with the same number of points

as the mesh edge gap

and the mesh edge nodes are snapped to the poly nodes.

(note that in Rhino a poly must have at least 3 nodes)

Snap mesh edges nodes to existing poly (used to fix mesh edges to a curved poly)

If some mesh nodes are snapped to some poly nodes

and the number of nodes "in the gap" equals the number of poly nodes "in the gap"

the mesh nodes "in the gap" are snapped to the poly nodes "in the gap"

(the mesh nodes do not have to be corner nodes)

Snap mesh edge nodes from existing poly to new poly (used to change mesh edges, like moving a conic top ring)

If mesh edges are fixed to an existing polyline

and a new polyline is also selected that is not connected to the meshes, and has the same number of

nodes

the mesh edges will be moved to the new polyline

Make Poly Float

Model builder can add a tag to 3D polylines to specify special handling during relaxation. The special handling allows 3D polys, to move in some restricted manner during relaxation, to achieve a force balance. The possible settings are:

Check	Reports on any existing tags
None	Sets the tags to no special handling
XY	Allows the 3D poly to move in the XY plane
Z	Allows the 3D poly to move in the Z axis
Rot X	Allows the 3D poly to rotate around the X axis
Rot Y	Allows the 3D poly to rotate around the Y axis
Rot Z	Allows the 3D poly to rotate around the Z axis

Note that a floating poly will not work correctly with connected to stiff links (guys).

Subtract Meshes

Model builder will take two meshes, and try to interpret one mesh as a hole in the other. If this is possible then a series of meshes will be drawn around the hole, that can be used as joined edge meshes during relaxation. Only the corners of the smaller mesh are checked, and they should be snapped to the vertices's of the larger mesh.

This function can be used to develop hypar sails with holes in them, or doorways in wall structures.

Draw Advanced Conic

Model Builder will take a circle representing the top ring, and a 3D polyline representing the base, and will draw in meshes between the two, with the mesh Msurf being varied on each mesh to achieve an equal mesh thread spacing on the 3d Poly. This can make a better model when the base width varies a lot. The average msurf is set in the options. The circle can be angled to any plane, and will be overdrawn with a 3dpolys. The option "With fixed Edges" will draw in a series of 3dPolys for the fixed edges, and snap the mesh edges to them.

Add 3D Detail

The main use for this tool is to assist in designing corner plates to terminate fabric corners and cables.

This tool will read a flat dxf detail drawing from a library, and insert it between two reference lines in the 3d drawing. The drawing reference lines are coloured green and blue, and the lines must touch at one end, so they form a V. The V can be in any xyz plane.

Alternatively the dxf detail can be inserted between mesh edges, where the mesh corner is identified by a short Cyan line, or by nodes on a 3d polyline.

The dxf drawing consists of 2d polylines that will be transferred to the 3D drawing, a dxf reference V in green and blue, and optionally some points. Entities in the dxf drawing can have special meaning depending on their colour or layer. This is described in the [detail dxf specification](#).

Some simple detail drawings are included during installation. You can use these as a starting point to draw your own dxf detail drawings.

Angular scaling can be:

- to fit between the drawing V lines.
- to fit in a mesh corner
- to fit in a mesh corner tangentially, ie allowing for the curvature of the mesh edges

Linear scaling can be:

- to fit the average length of the drawing V lines or cyan line
- to scale to a specified length

The scaled and oriented detail is drawn out as 3Dpolys, and optionally some points.

The 3Dpolys are the main entities in the inserted drawing, and are used to define the basic reinforcement plate shape. The optional points can be used to carry position information into the panelling process.

Add Corner Plate

Adds corner detail, including a clamp plate with spaced bolt holes, cable tubes, fabric reinforcement patches and seam cutouts for the cable termination. These are dimensioned from the front edge of the clamp pate, so varying the plate size varies all the dimensions parametrically.

The corner plate is usually inserted at a mesh corner, marked with a cyan line, or with the nodes of a 3d polyline. The insertion can be angularly scaled to fit the mesh corner. The effective "plate arc centre" can be at the mesh corner, or at a position defined by a tangents the mesh edge at the insertion point.

The corner plate is drawn as 3dpolys, which have tags in the extended data to define them as corner detail during the panelling process.

Add Corner Disk

Adds corner detail, including a disk with spaced bolt holes, fabric reinforcement patches and seam cutouts for the cable termination. These are dimensioned from the front edge of the clamp pate, so varying the plate size varies all the dimensions parametrically.

The distance of the disk from the mesh corner is adjusted to make the disk tangential to the mesh edges.

The corner disk is usually inserted at a mesh corner, marked with a cyan line, or with the nodes of a 3d polyline.

The corner disk is drawn as 3dpolys, which have tags in the extended data to define them as corner detail during the panelling process.

Orient to XY

This tool will take some 3dpolys, meshes, lines and points that are in an arbitrary xyz plane, and orient them so that they lie flat in the XY plane.

The main use of this tool is to take corner plate detail, that is drawn to fit the 3D model, and lay it flat for a metal work drawing.

The if the original objects lie in a plane, then the orient to XY will produce a perfectly flat 2D drawing.

If the original objects do not lie in a plane, then a "best fit" plane will be used, getting the oriented objects as flat as possible. This can be used for re orienting a mesh model that is largely vertical for cross sectioning.

Draw Library Model

[Described here](#)

Draw Excel Model

[Described here](#)

Read MPSD file

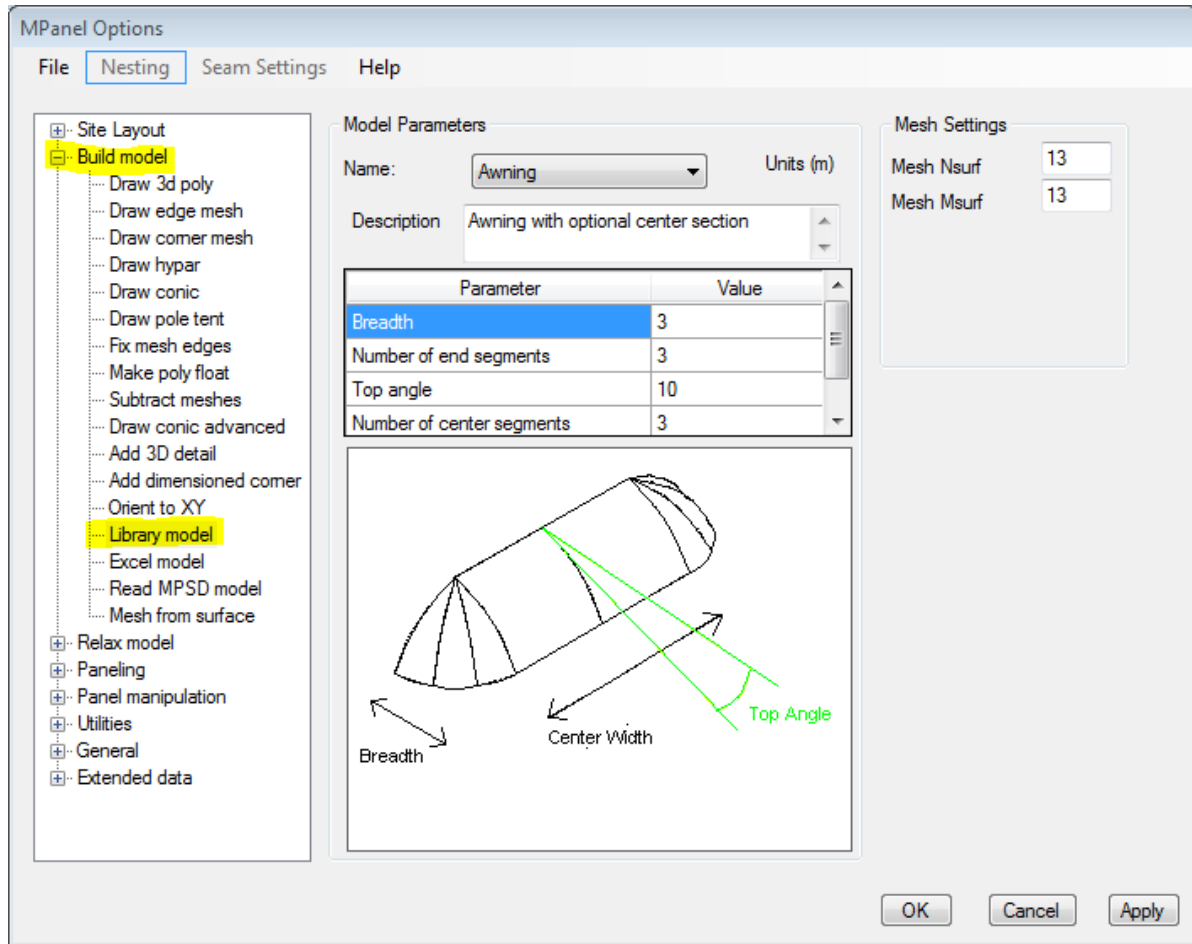
This tool will read an MPanel Shade Designer file (*.mpm) and build the prototype model described in the file. This allows users who obtain their site measurements with MPSD to build the model with MPanel, which gives more design and panelling options. The meshes will have extended data set for weft/warp stress ratio's and cable tension, so if you want to alter the model with the Mpanel options you will have to set the extended data back to the default setting, as [described here](#)

If the mpm file was a partial file (produced by the MPSD site checker web site) then the prototype model will be drawn with no extended data and no corner links.

Create Mesh from Surface

This tool will take a simple surface, and try to convert to a regular polygon mesh that can be used by MPanel. If the surface has 3 or 4 edges then the surface edges can be used directly. For more complicated surfaces the 3 or 4 edges will have to be drawn on the surface before using this tool.

Library Model Builder



Model builder will draw a model with meshes and 3dpolys defined in a library of standard designs. The required model is selected from a drop down list, and will be drawn out when the Model Builder button on the toolbar is clicked. Library models include:

- Awnings
- Barrel vaults
- Pole tents with valances
- Cable supported tents
- Modular cone tents
- Hypar sails

To build a Library model:

- In the MPanel Options specify the Model Builder to use Library models
- Choose the model you want from the drop down list.
- Change any parameters you need to suit your dimensions.
- Close the options and use the Model Builder button to create the model.

You can make your own copy of a library model template, by using the "Library Model.... Save as" tool on the File menu. This lets you build up a library of your own models, perhaps for company standard designs.

A description of the basic model templates is available here: [Library](#)
Other models may be added later.

Excel Models

Excel spreadsheets are supplied for all the library models, and can be used instead of the model library to define models.

The spreadsheets will be in your Documents/MPanel/Excel folder.

To use a spreadsheet for a design:

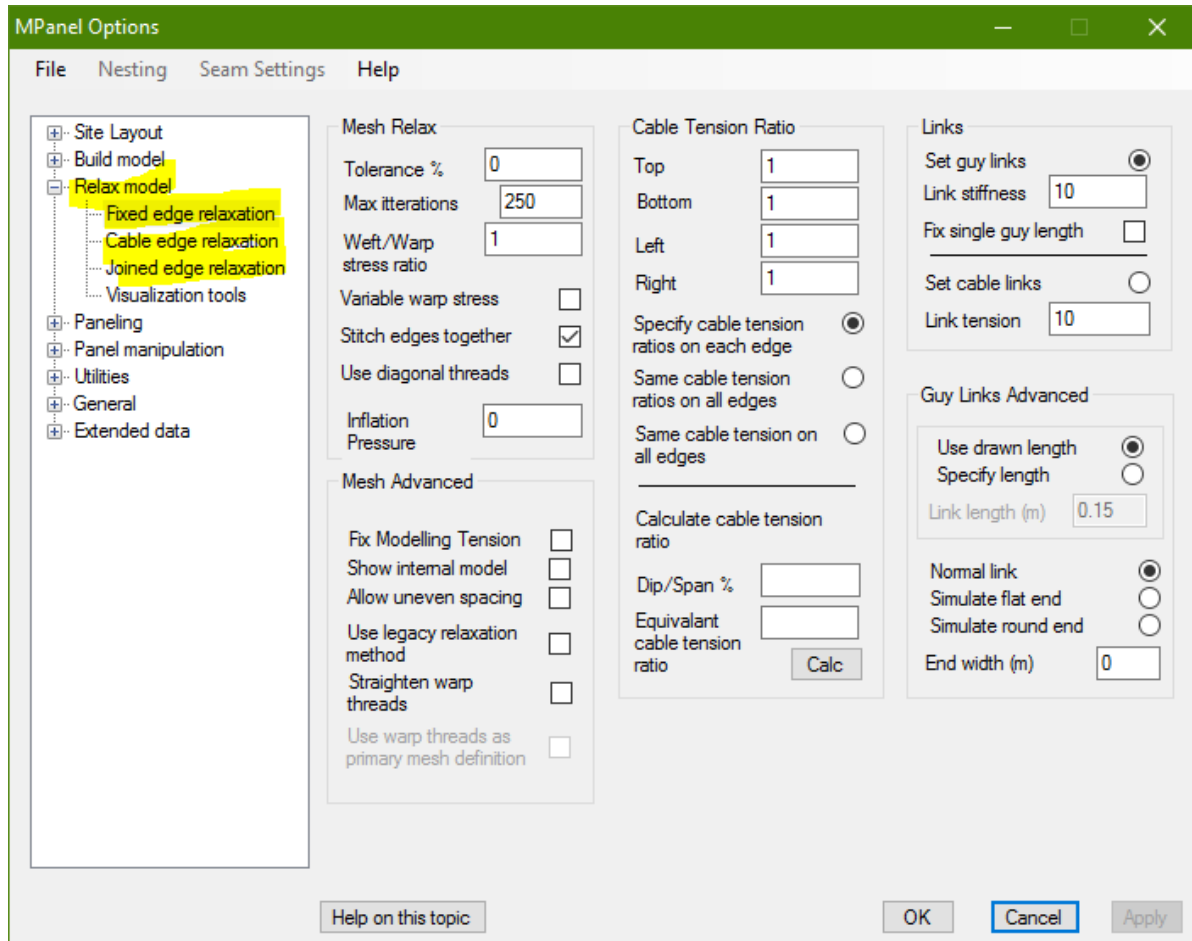
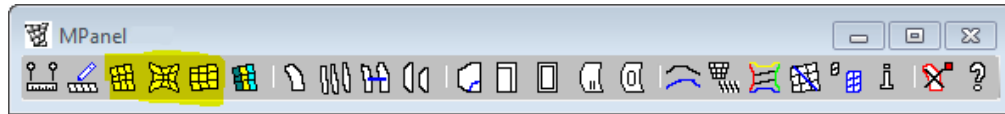
- Open the spreadsheet in Excel.
- Make any changes needed to the parameters.
- In the MPanel Options specify the Model Builder to use Excel models.
- Run the model builder to create the model.

An advantage of using spreadsheets as the design input is that the parameters can be calculated from a formula. So you might be making a range of tents, with the pole height determined by a formula involving width, breadth, or other factors.

To be read correctly, the spreadsheet has to follow the following rules:

- The spreadsheet must be open in the first running copy of Excel.
- Cell A1 must contain the text: **MPanel Design Sheet**
- The parameter names must be in column A
- The parameter values must be in column B

Relax Mesh



The three mesh relaxation tools can be used to relax one or several meshes, with the following boundary conditions:

- Fixed edge meshes: all mesh edges are fixed. Additional fixed points can be specified using 3D poly nodes.
- Cable edge meshes: all mesh edges contain cables. On edges with corners that join onto another mesh edge, with the same number of nodes on each, the edge nodes are usually stitched together with the option "Stitch edges together". Fixed points must be specified using 3D poly nodes.
- Joined edge meshes: if a mesh's corners are joined to another meshes edge, with the same number of nodes on each, then those edges are treated as a smooth joined edge. The nodes between the corners will be stitched together automatically. All other mesh edges contain cables. Fixed points must be specified using 3D poly nodes.

Additionally links can be used to restrain the movement of points in the mesh relative to other points. See [Using Links](#) for more information on this.

To relax the model, select all the meshes, 3Dpolys and links in the model, and click on the appropriate relaxation button.

Relaxation method

The mesh is relaxed towards a shape that has an even stress across the mesh surface, using a "Constant stress algorithm" (CSA). On a simple shape this will be an approximation of a soap bubble surface.

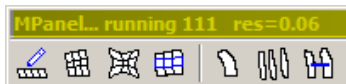
The shape of the surface is controlled by the weft/ warp stress ratio. This ratio can be set to achieve a variety of different anticlastic shapes. If ratios substantially different from 1 are used, caution is required to ensure that your fabric can physically take the stress ratio selected. Typically conics will require a weft/warp stress ratio between 0.33 and 0.66

In some cases, notably conics with small top rings, the constant stress surface is not visually appealing. In these situations you can increase the stress in the parts of the mesh where the mesh warp threads are closest, ie around the top ring, by selecting "Variable warp stress"

During relaxation a Cancel button is shown in the Mpanel text window. The relaxation can be interrupted with the cancel button (or by the ESC key when Mpanel has the focus) an option to draw out the partially relaxed mesh will be offered. This may be acceptable if the reported residual was low enough.

Technical descriptions of the relaxation methods used are available: [CSA method](#) and [Comparison of CSA and legacy methods](#)

To check that the mesh is fully relaxed



The number of iterations needed to fully relax a model depends on the geometry, the mesh density and the number of meshes.

To check if the mesh is fully relaxed you can check the residual mesh movement, reported on the MPanel toolbar during the relaxation, and in the MPanel text window afterwards. Or look at the old and new meshes after a fixed number of iterations and see if there is any shape change.

The residual represents the average percentage movement of the mesh during the last few iterations, less than 0.0001 usually indicates a converged mesh.

Mesh direction

The direction from the bottom of the mesh to the top is called the mesh warp direction, on the assumption that panels will be built in this direction. The direction across the mesh is called the mesh weft direction. To find the warp direction on a mesh that is already drawn use the [Information](#) button.

Optionally the internal model can have additional diagonal threads across the mesh cells. The tension in these threads is controlled by the weft/warp stress ratio, so this allows us to adjust the droop of hypar sails.

Cable names and shaping

The cable tension for the four mesh edges can be individually set in the options screen. "Cable tension bottom" refers to the cable tension on the bottom of the meshes, which may not be at the bottom of the screen. A cable tension of 1 means a tension in the cable equal to the total tension in the fabric on that edge, and will roughly make a cable arc with a radius the same as the mesh edge width. This method means that longer spans tend to get assigned more internal modelling tension, which is usually what is wanted to keep the cable edges a reasonable shape.

Optionally "Same cable tension ratios on all sides" can be specified. This will give approximately the same dip/span ratio on all the cables.

Optionally "Same cable tension on all sides" can be selected. This calculates the internal modelling tension based on the average edge length, so gives the same cable tension on each side. This should always be used when the cable edges pass through the corner, ie when you need even cable tensions on all edges.

Often cable edges are specified by the "Dip/Span" ratio. A calculator is provided to convert dip/span ratios into the equivalent cable tension. This calculation will only apply to cables that span one entire mesh edge. The calculator is reasonably accurate for commonly used dip/span ratios (5% to 20%) becoming less accurate at larger dip span ratios.

Guys and Cables

Lines in the drawing will be read as either links simulating guys or links simulating cable segments, depending on the

setting in the options. They can be used for corner guys, simulate corner plates or hardware, and the create cables in the mesh. A fuller description of using links is [given here](#).

Extended data

When the model is relaxed, mesh relaxation parameters are saved as extended data. This information can be inspected in the Extended data tab, and can be altered for individual meshes.

On the next relaxation any extended data will be read in, and this allows you to alter the parameters of individual meshes or just a section of the model.

Poly Float constraints

Normally a 3D poly defines a series of fixed points. But a polyline can also have a tag placed on it to make it behave differently during the relaxation. The tags are placed on using the Model Builder "Float Polys" tool. The tags can allow the polyline, as a whole, to:

Move in XY position for vertical poles)	(finds the conic top ring force balance
Move in Z	
Rotate around the X, Y or Z axis (finds the conic top ring balance angle for angled poles)	
Constrain points to be in straight line (to make a tent lace lines, or straighten a valley seam)	
Check constraints	(reports on the current poly tags)

Using these options it is possible to find the force balance position for vertical pole supported tents, the pole angle for asymmetric tents, of the position of a free floating stiff member, such as a centre ring, or constrain part of the mesh to lie in a straight line, such as for a tent lace line. The tag can be written to the poly with the "Set float" button in the options, or by closing the options and running the model builder Float poly tool.

Other relaxation options

Optionally the tension in each thread can be adjusted toward a constant fabric stress over the mesh. This can be of use in evening out stress variations, but may produce less pleasing shapes. When using constant stress, it is better to apply several small relaxations, to allow the tension matrix to be updated.

Optionally you can fix the guy length. If there are links in the model used as guys, they will stretch some during the relaxation process. Best practise is to let them stretch and use the finished link length as your guy specification, but you can force a fixed length onto single guys that are anchored at one end.

Optionally, a pressure can be applied across the mesh. This will result in a inflated shape. Depending on the geometry pressure settings from 0 to 1 will give reasonable inflated shapes. When using pressure, it is better to apply several small relaxations, to allow the tension matrix to be updated.

Optionally the legacy relaxation method can be used to replicate a model made before version 17.

Advanced relaxation options

Normally the mesh overall shaping is best defined by specifying that the warp threads define the model. This allows the weft threads to be at large angles to the warp threads without affecting the model shape. Optionally this preference can be turned off, and then both the warp and weft threads are used to determine the overall model shaping. This is mainly used to duplicate models made with earlier version of MPanel.

In some models it is useful to straighten the warp threads. This applies particularly to fixed edge models where there is a sudden change in mesh warp thread spacing, causing the warp threads to bow. The option "Straighten warp threads" can be used in these situations, but on some model shapes may have unexpected consequences.

Optionally the internal modelling tensions can be fixed at 1. This is only used in exceptional circumstances, such as transferring an MPanel mesh model to a links only model. See [Internal model](#) for a description of the modelling tension.

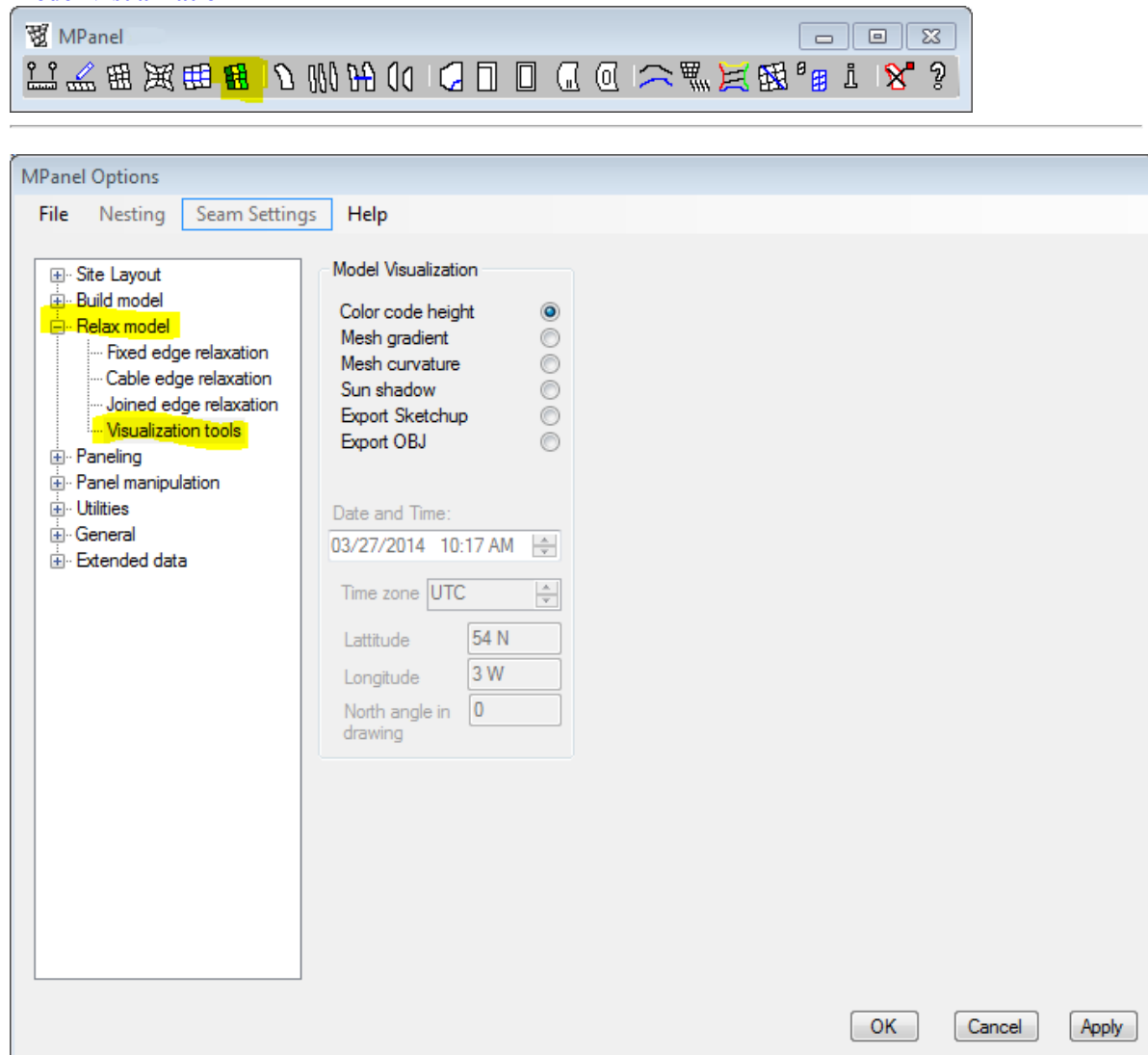
Optionally the mesh "even spacing" can be turned off. Normally MPanel uses an even spacing algorithm to make the

mesh more evenly spaced across the model. This can lead to problems in awkward shaped models, and can be turned off then.

Optionally the internal link model can be displayed. MPanel translates the drawing model into an internal model built solely from links with different assigned properties. This model can be drawn out for inspection, but has no further purpose. See [Internal model](#) for more information.

Optionally the model can be mirrored about a mesh edge. This option is set in the [extended data](#), and allows large symmetrical models to be created and relaxed more easily. The mirroring is done with a vertical mirror, ie as though you were looking down on the model. Usually the mirrored edge is fixed at it's corners. Theoretically a model could be mirrored on several edges at one time, but care is required in the resulting interpretation.

Model Visualization



Model Visualization is a tool that shows information about the model that can be useful for design and . The images produced by this tool are not usable as input for other MPanel tools, they are just for interpretation.

- Colour Height
- Mesh Gradient
- Mesh Curvature
- Sun Shadow
- Export Sketchup
- Export OBJ

Color Height

Draws a composite mesh that is colour coded for the local mesh elevation, from Blue at the bottom of the model to Magenta at the top. This can help in interpreting the shape of curved surfaces.

Mesh Gradient

Draws a set of lines that indicate the direction that water will travel on the mesh surface. Each line starts at a mesh node, and points in the 3d direction of the local steepest descent. The lines are colour coded to indicate the gradient, from flat (Blue) to gradients of 1:1 and above (Magenta).

Areas of possible ponding would be indicated by blue lines pointing in towards each other. but note that the gradient is shown for the mesh shape as designed, with no external loading. Low gradient areas can be expected to deflect downward under water / snow loading, so ponding may occur in the deflected shape even when it is not indicated in this gradient plot.

Mesh Curvature

Will take a set of meshes and draw crosses representing the local mesh curvature at each node.

The crosses each consist of a Green and Blue or Red lines,

The Green line shows the direction of the mesh principle curvature. It's length is proportional to the principle curvature.

The Blue line shows the direction of the mesh second principle curvature. It's length is proportional to the second principle curvature.

If the surface is locally inflated (ie the mesh curvatures have the same sign) then the second principle curvature is drawn in Red

This has use in showing flat spots in a model, in understanding the mesh shaping, and also in understanding the best direction to panel in.

Also any red lines would indicate a mesh shape that could not be supported by tensile forces alone.

The best direction to panel in is in line with the second principle curvature, ie along the Blue lines. This allows the seams to take care of as much of the principle curvature as possible in the panel edge curvature.

So looking at a mesh curvature plot:

- When the curvature is significant (large crosses)

- When the principle curvature is bigger than the second principle (Green line longer than Blue line)

 - Then you should panel along the Blue lines.

Sun Shadow

Draws out a flattened copy of the meshes on the Z=0 plane, in the position that the shadow would be. The shadow position is determined by the site latitude and longitude, the date and time of day, and the time zone that the site is in. A time and date stamp is added to the shadow drawing.

The latitude and longitude can be entered in degrees, minutes and seconds, or in decimal degrees (but is converted to and stored in decimal degrees). The suffix S, N, E, W can be used after the entry, so that a typical latitude and longitude would be 54.7 N 20.1 W

The time zone is the number of hours ahead or behind from UTC (Coordinated universal time, more commonly known as GMT for Greenwich mean time). So using this setting you can also allow for local summer time settings.

Usually a CAD drawing will have North straight up along the Y axis when in Top View. If this is not the case then the angle between North and the Y axis can be entered, in degrees clockwise, like a compass.

Export Sketchup

The model can be exported in mpx format, suitable for reading with Sketchup. The image in Sketchup can be rendered, shaded, fogged, etc.

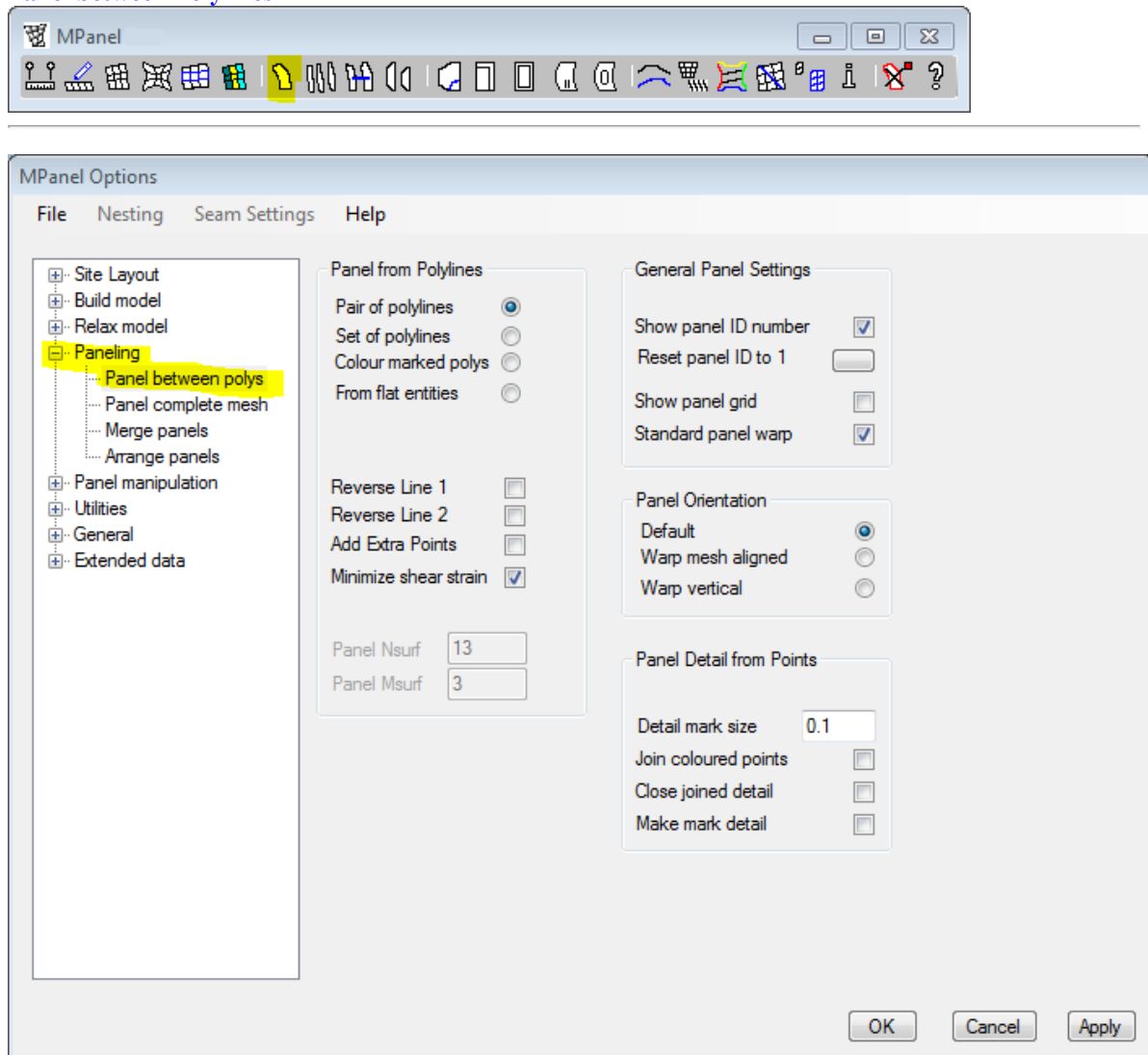
To read the exported file in Sketchup you need the Mpanel Import plug in. You can install this from the Mpanel Options menu under File... Import Sketchup plug in. Once this is installed the model can be shown in Sketchup using the menu item Plug ins... Import Mpanel model.

Make sure that the MPanel selected units are the same as the Sketchup units, or the model may be scaled.

Export OBJ

The model can be exported in obj format, suitable for reading with Awning Composer, and other software that read OBJ files. Once the model is read into Awning Composer you can render, add material textures, apply sun shadows, etc.

Panel between Polylines



Panel between Polylines will create a flat panel from 3D polylines. It can perform the following functions, as set in the options:

- Panel between a pair of polylines
- Panel a set of polylines
- Panel a colour marked set of polylines
- Panel from flat entities.

All panels can have their internal Panel ID number shown in large numbers on the panel for identification, This is automatically removed during panel output.

Panel between a pair of polylines

This tool takes two 3D polylines and computes the "tightest possible" surface between them. Locally this is a flat panel, but the whole panel will generally be curved.

The tool can accept 3D polylines with different numbers of nodes, and extra nodes will be inserted to make a panel with the same number of nodes on each side. This feature is selected in the options. The Polylines must be simple polylines, i.e. no arcs or smoothing. It can cope with 3D polylines that are drawn in different directions, i.e. one drawn "up" and one drawn "down", by reversing one or the other.

In general there is always an error when creating a 2D surface to fit between 3D boundaries. To minimise the errors select the option to "Minimise the shear strain". This both halves the error and avoids the error from compounding. The panelling errors are reported as the equivalent shear strain. See the [Modelling and Panelling Errors](#) topic. The shear strain is recorded in the panel as extended data.

The panel is drawn out in XY and should be viewed in TOP (XY) view.

Panel a set of polylines

This tool takes a set of 3D polylines, internally creates the panels between each pair, and merges the panels together to produce one panel that will typically have some end curvature. The shear strain reported for this operation is the sum of the panelling strains and the merging strains.

The lowest shear strains are obtained by using panel sets that merge together in pairs, ie 2, 4, 8 or 16 panels. These need 3,5,9 or 17 polylines in the set.

To obtain the lowest shear strain, the best merging point for each of the internal merges is computed separately. On some extreme merges this may cause unexpected panel shapes. If the "Minimum Strain Merge" is not checked then the same averagely good merging point will be used for all internal merges. The shear strain is recorded in the panel as extended data.

Detail points, in 3D, can be included with the polylines for panelling. They should be drawn as ordinary points, placed close to the local surface of the polys. During panelling an annotation mark will be added the correct 2D place. This feature can be used to position artwork, define cutouts, etc. Optionally, if the points are coloured, the annotation marks can be joined together to form a detail poly. Several different joined up sets can be defined by using different point colours. Points of the same colour are only joined if they are separated by half the panel length, so detail on two panel corners can be regarded as separate, even if they have the same colour.

Optionally, the points can be defined as mark detail, which is used to define the position of a mark line in the seam program.

Panel a colour marked set of polylines

This tool takes a set of 3D polylines and uses the "**Panel a set of polylines**" routine to make several panels in one operation. The panels edges are marked in the set of polys by colouring them a distinct colour.

For example, a set of black polylines could be panelled into 4 panels by colouring 5 of the polylines blue, to mark the panel edges. The colours used should be simple colours, like the colours in the CAD colour pick box.

A set of polys produced by the Cross Section tool, with the "extra polys" drawn in grey, will work with this tool to make a several panels in one operation.

Mark points are supported in the same way as in the "Panel a set of polylines" tool.

Panel from flat entities

There is a need to create panels directly from flat entities. These could be fill-in panels, non stressed panels, etc. With this tool any 3 or 4 simple entities, including lines, arcs, splines, 2dpolys and 3dpolys, can be used to define a panel. The entities should join at the corners, and should be oriented on the screen so that the top, bottom, left and right are distinct. If it is a triangular panel we assume that there is a bottom, left and right.

The panel will be generated with the number of points on the left and right sides defined by the Nsurf setting, and the number of points on the top and bottom defined by the Msurf setting. Usually the points will be evenly spaced along the panel.

If the original entities are 2dpolys or 3dpolys, and if they have the same number of points as specified in the nsurf and msurf, then the original entities nodes are copied to the new panel, without evenly spacing them. This allows a panel produced by a third party to be imported into MPanel exactly. This panel could then be compensated, seamed, and

output by MPanel.

General Notes

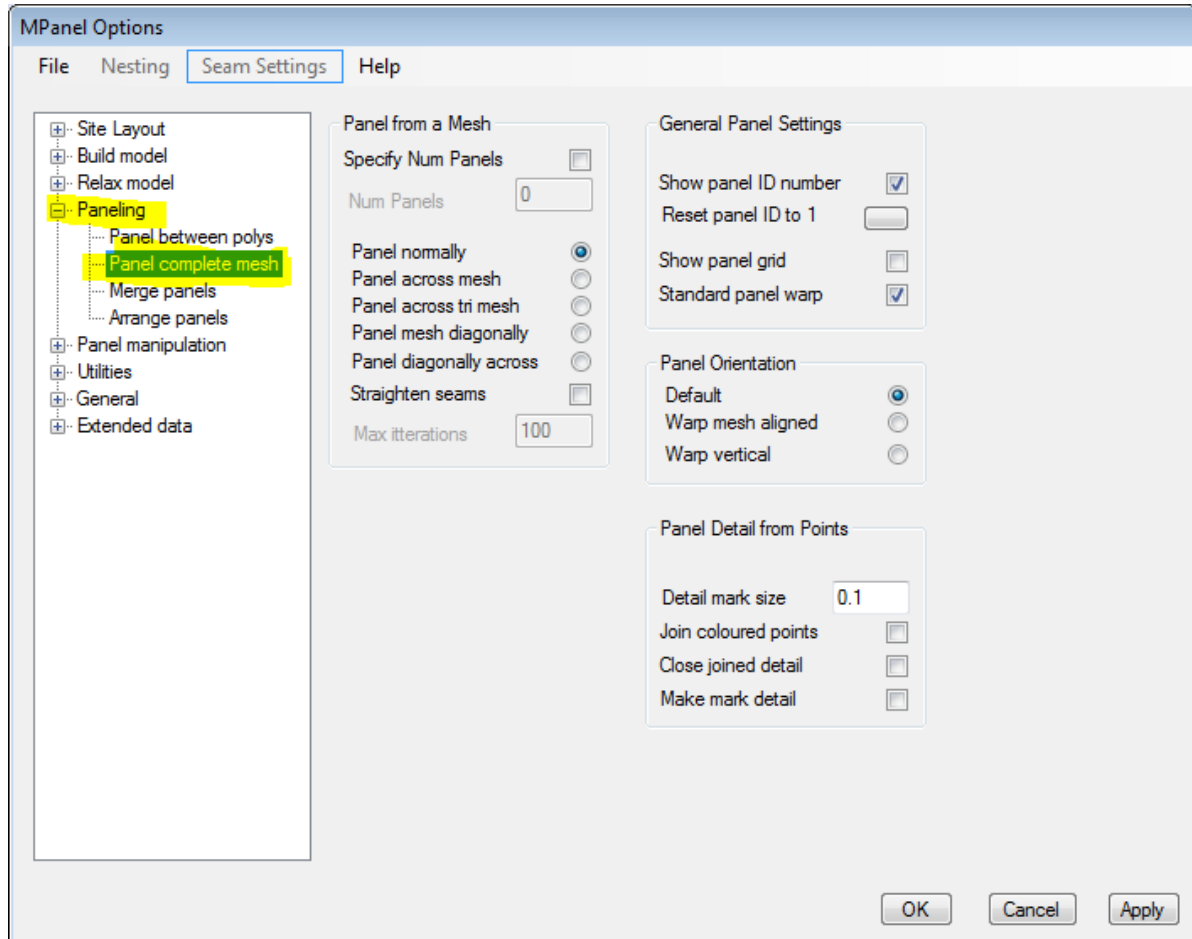
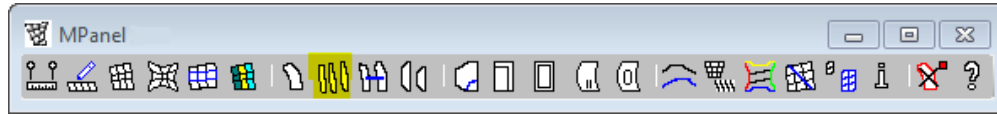
By default, the panels lie on top of the polylines that created them. If you choose 'warp vertical', then the panels are aligned with the warp vertical, so the left side of the panel is on the left side of the screen.

Panels can have their internal generation grid drawn out with them, by choosing 'Show Panel Grid'. These grids are derived directly from the 3D mesh, so they act as a placement grid for graphics, cutouts, etc.

Panels can be triangular, that is they can be missing a top or a bottom edge. These panels will still seam, compensate, stamp and output correctly. However the panels must have left and right edges to be processed correctly. If a set of polylines are panelled where the top ends all join together, the panel produced will be triangular.

Panels are issued an internal ID number, which is preserved through all panelling processes, and can be used to trace panels through the drawing. The Panel ID can be inspected with the MPanel Info tool. The panel ID can also be shown on the panel, in large print, by selecting the option "Show panel ID number". You can also manually reset the next panel ID number, but be aware that this might result in panels with duplicate internal panel numbers.

Panel Mesh



Panel Mesh takes a 3D Mesh, usually after relaxation, and produces a series of flat panels from it.

Optionally the number of panels to be produced from the mesh can be specified. If this specification is left blank then as many panels as there are in the mesh will be produced. The number of panels specified has to be a number that will fit evenly into the number of panels in the mesh. So if you have a mesh with 12 panels (ie $M_{surf} = 13$) you can produce 12, 6, 4, 3, 2 or 1 panels. If the model has been built with the wrong M_{surf} Mesh to create a set of panels directly, a message will tell what M_{surf} is needed and offer other panelling suggestions.

Panel Mesh reports the equivalent shear strain for each panel in the text window. See the [Modelling and Panelling Errors](#) topic. The shear strain is recorded in the panel as extended data.

Panels can be made as:

- **Panel normally**

Normal panels following the mesh N_{surf} threads. This usually gives curved seams, which by following the mesh threads will have an even spaced appearance.

Optionally you can specify "Straighten seams", which will try to make the seams straighter by using geodesics or cross sections. This will fail if the straighter seam lies outside one of the mesh edges and can result in panels with unacceptably narrow centre's, even when the straightening doesn't fail. Making geodesic seams is an iterative process, the default number of iterations is 100 which should be adequate for 13 by 13 meshes. Higher density meshes, or meshes with a lot of curvature, will need more iterations.

- **Panel across mesh**

The panels can be created across the mesh, so the panel seams follow the Msurf threads. In some cases this will give a more acceptable set of panels. Otherwise the same as Panel normally.

- **Panel across tri mesh**

With triangular meshes creating the panels across the mesh gives a top panel that is poorly defined, as it doesn't have a left side (In MPanel panels have to have a left and right side, but the top and bottom is optional.) And option is to panel across the mesh, whilst keeping the panel left and right edges aligned with the mesh left and right edges. This will produce an acceptable top panel on a triangular mesh. This option is called "Panelling across a tri mesh". Otherwise the same as Panel across mesh.

- **Panel diagonally**

Curved cross sections are drawn across the mesh diagonally, in a curve that will bisect the corner angles, to produce diagonal seams. The edge and internal seams will include each edge node, so the cable shape is accurately defined. There are no restrictions on the number of panels, as long as it's an even number, but specifying too many panels will result on poor panel end definition. The use of "Straighten seams can result in panels with unacceptably narrow corners, even when the straightening doesn't fail.

The panel warp will also be diagonal, so the panel edges that correspond to the mesh edges will be the left and right edges.

- **Panel diagonally across**

Curved cross sections are drawn across the meshes other" diagonal. Otherwise the same as Panel diagonally.

By default, the panels lie on top of the parts of the mesh that created them. If you select 'warp aligned to mesh' then the panels are aligned with the mean mesh direction. If you select 'warp vertical', then the panels are aligned with the warp vertical, so the left side of the panel is on the left side of the screen.

An assumption is made that the outside of the mesh is face up. Care is needed if part of the mesh is outside face down, as those panels will be drawn in mirror image. In this situation the mesh should be rotated in xyz to obtain panels that are the correct way up, as using mirrored panels may cause problems with other MPanel tools.

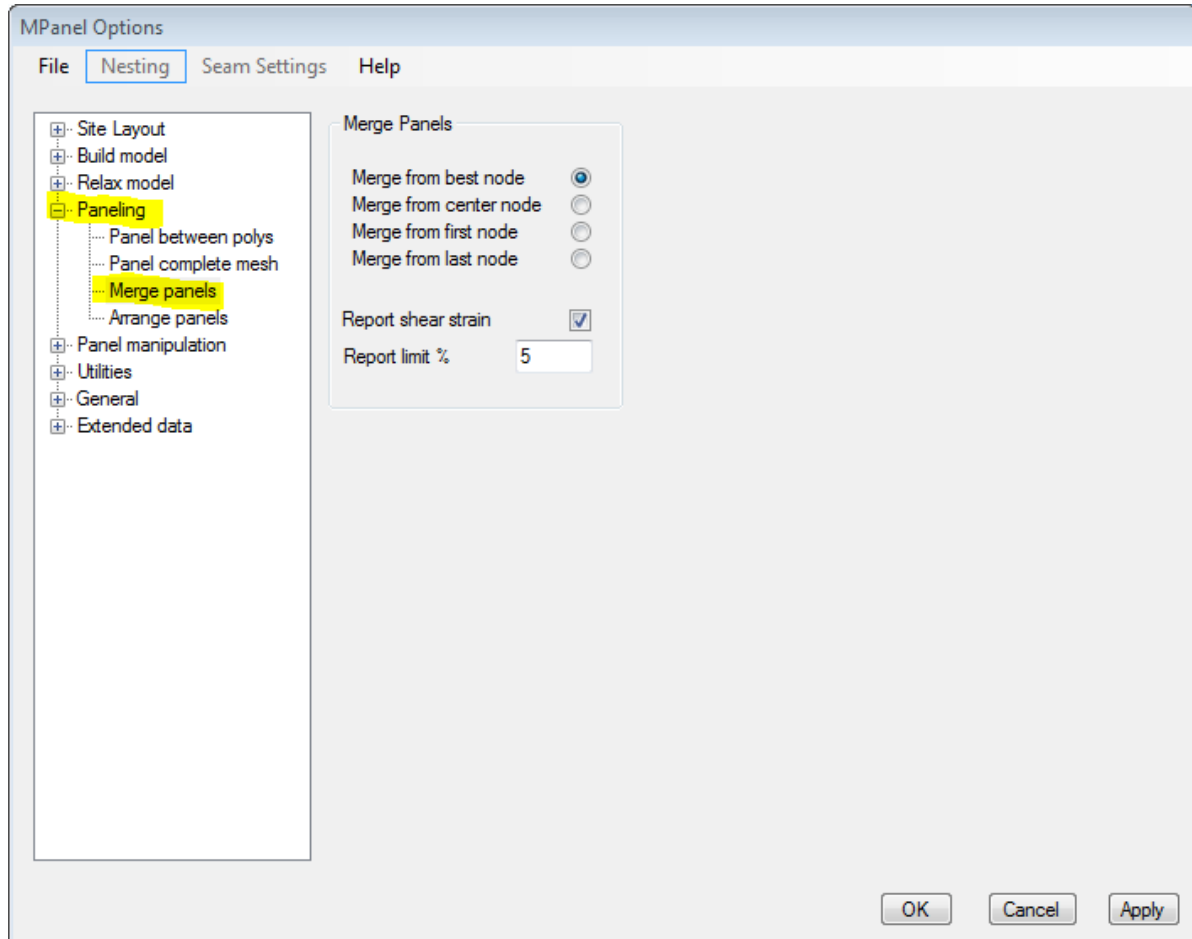
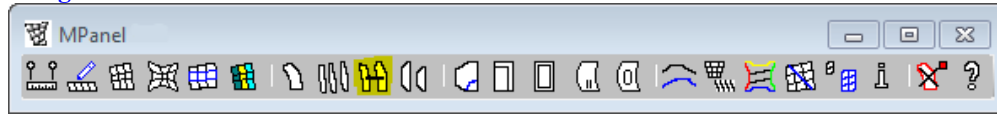
Panels can have their internal generation grid drawn out with them, by choosing 'Show panel grid'. These grids are derived directly from the 3D mesh, so they act as a placement grid for graphics, cutouts, etc.

Mark points, in 3D, can be included with the mesh for panelling. They should be drawn as ordinary points, placed close to the local surface of the mesh. During panelling they will be assigned to the correct panel, and an annotation mark added the correct 2D place. This feature can be used to position artwork, define cutouts, etc. Optionally, if the points are coloured, the annotation marks can be joined together to form a detail poly. Several different joined up sets can be defined by using different point colours. Points of the same colour are only joined if they are separated by half the panel length, so detail on two panel corners can be regarded as separate, even if they have the same colour. Optionally, the points can be defined as mark detail, which is used to define the position of a mark line in the seam program.

The panels are drawn out in XY and should be viewed in Top (XY) view.

Panels are issued an internal ID number, which is preserved through all panelling processes, and can be used to trace panels through the drawing. The Panel ID can be inspected with the MPanel Info tool.

Merge Panels



Merge Panels joins together two panels along their inner edges. The inner edges are identified to the program by making them sensibly close together, without the panels overlapping. Both panels are then selected and the program is run.

The distortion created by merging two different shaped panels together are all moved into the diagonals, causing shear strain. If your fabric can accept shear strain, (i.e. if it is like a net rather than like a film) then merging enables a flat panel to assume a 3D shape. The amount of shear strain you can accept depends on your fabric; on some fabrics it will not be possible to use any significant shear at all, but on an uncoated- coated nylon (F111) 20% shear has been used successfully. See the [Modelling and Panelling Errors](#) topic.

You can merge panels that have been created by merging panels, i.e. with panels A, B, C, D you merge A and B to make AB and merge C and D to make CD. You can then merge AB and CD to make ABCD. The shear strains for each of the panels to be merged are read in from extended data, and added to the shear strain from the merge, to produce a total shear strain for the panel. This is reported in the MPanel text window, and recorded in the panel as extended data.

The edge nodes that the panel merging starts from affects the merged panel shape, and the shear strain. Usually you want to let the program choose the node that gives the lowest shear strain. In some situations you want to specify that the merging starts at the top, the bottom, or the middle of the panel.

In some situations you want to force the merging to start at the edge first node, last node, or in the centre, there are options for these choices.. (The only common use for this is to minimise the distortion on part of the panel, for instance when a reinforcement patch is being made over two panels)

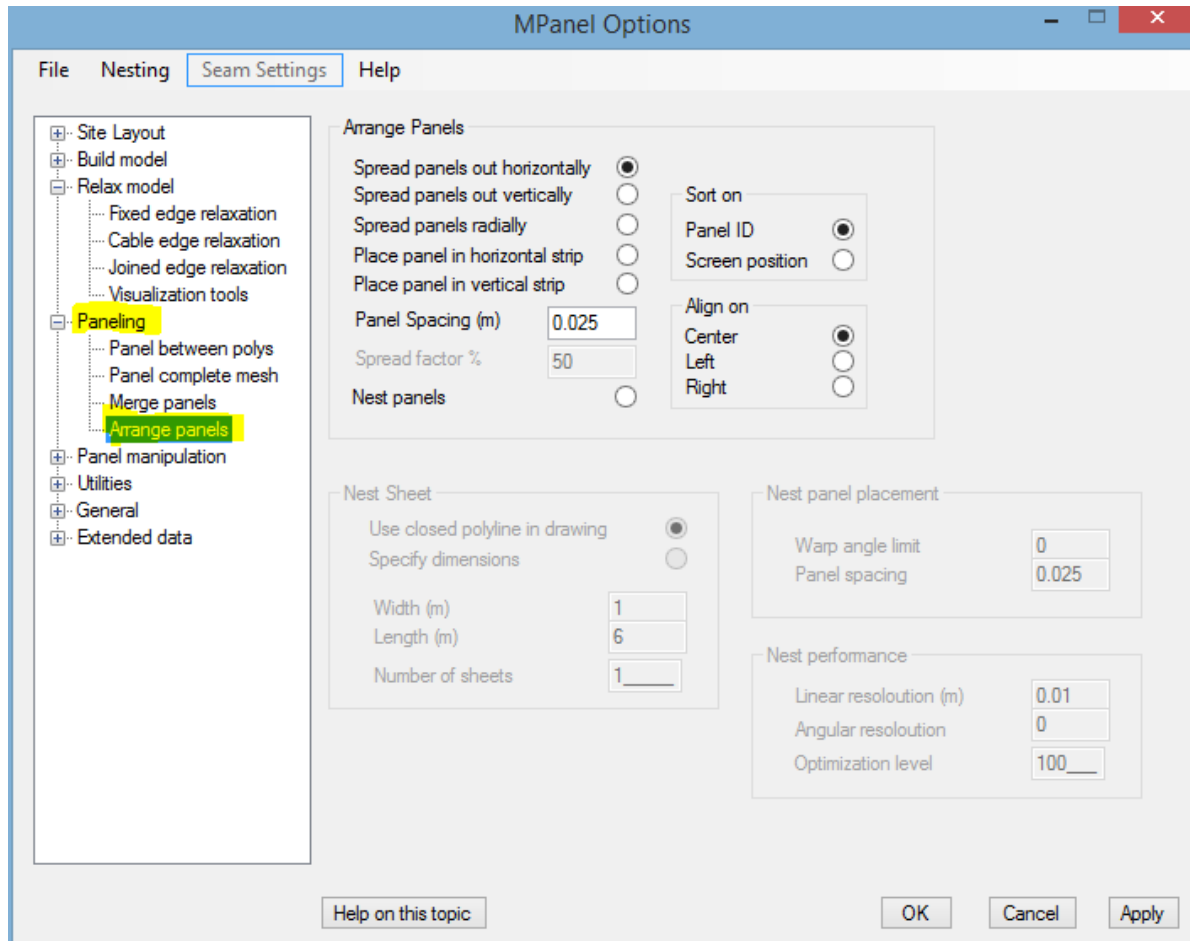
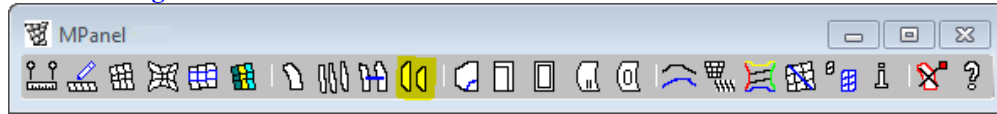
With a low shear stiffness material (i.e. like a net) this technique lets you create very sculpted 3D shapes from a single

flat panel.

Any detail polylines in the panel, such as internal grids or added detail, will be moved and distorted with the panels.

Merged panels are new panels that are issued an internal ID number, which is preserved through all panelling processes, and can be used to trace panels through the drawing. The Panel ID can be inspected with the MPanel Info tool.

Panel Arrange



Panel Arrange will arrange a set of panels using the following methods.

- Spread Panels out Horizontally
- Spread Panels apart Radially
- Place Panel in Horizontal strip
- Place Panel in Vertical strip
- Nest Panels

Spread Panels out Horizontally

This will take a set of panels and draw out a copy of the panel in a row, each panel aligned vertically, with a specified spacing between each panel. The order they are drawn is determined by the panel internal ID number, or the original panels bottom left X coordinate

The vertical alignment can be based on the panel centre line, the panel left edge, of the panel right edge.

This has use when a set of panels, which may be partially overlapping, are produced from a mesh, and need to be spread out to allow further MPanel operations to be carried out on them.

Spread Panels out Vertically

This will take a set of panels and draw out a copy of the panel in a column, each panel aligned vertically, with a specified spacing between each panel. The order they are drawn is determined by the panel internal ID number, or the original panels bottom left Y coordinate

The horizontal alignment can be based on the panel centre line, the panel left edge, of the panel right edge.

This tool is used to pre aligning panels for nesting when the panels have no warp line.

Spread Panels apart Radially

This will take a set of panels and spread them apart radially, from the average center of all the panels. The amount of spreading can be specified as a percentage. This has use in spreading panels apart whilst maintaining their original positions relative to each other, like in a set of panels for a conic.

Using this tool is can be a good start when creating alignment marks during stamping.

Place Panel in Horizontal strip

This will take a single panel, and make a copy of it inside a pair of horizontal guide lines, that represent the fabric roll or the cutting table, in a position that avoids any panels that are presently inside the guide lines, and tries to minimise wasted material.

In use you select the guide lines, the panels already inside the guide lines, and the new panel you want to place. Then run the tool, and a copy of the new panel will be drawn inside the guidelines. You can also specify a minimum spacing between panels.

The algorithm used to find the best position for the new panel is not very strong, and you will sometimes be able to see a better position for the panel by eye. In that case move the panel using normal CAD methods, and then continue placing panels in the strip using this tool.

The panels are not rotated by small angles during this operation. This ensures that the intended warp direction in the panel is aligned with the warp direction of the roll. So you may need to rotate your set of panels to align with the roll before using this tool. This tool will rotate the panel by 180 degrees if that helps to get a better position. If you want to rotate the panels by small angles to obtain a better position, you can do this using normal CAD methods, and then continue placing panels in the strip using this tool.

Place Panel in Vertical strip

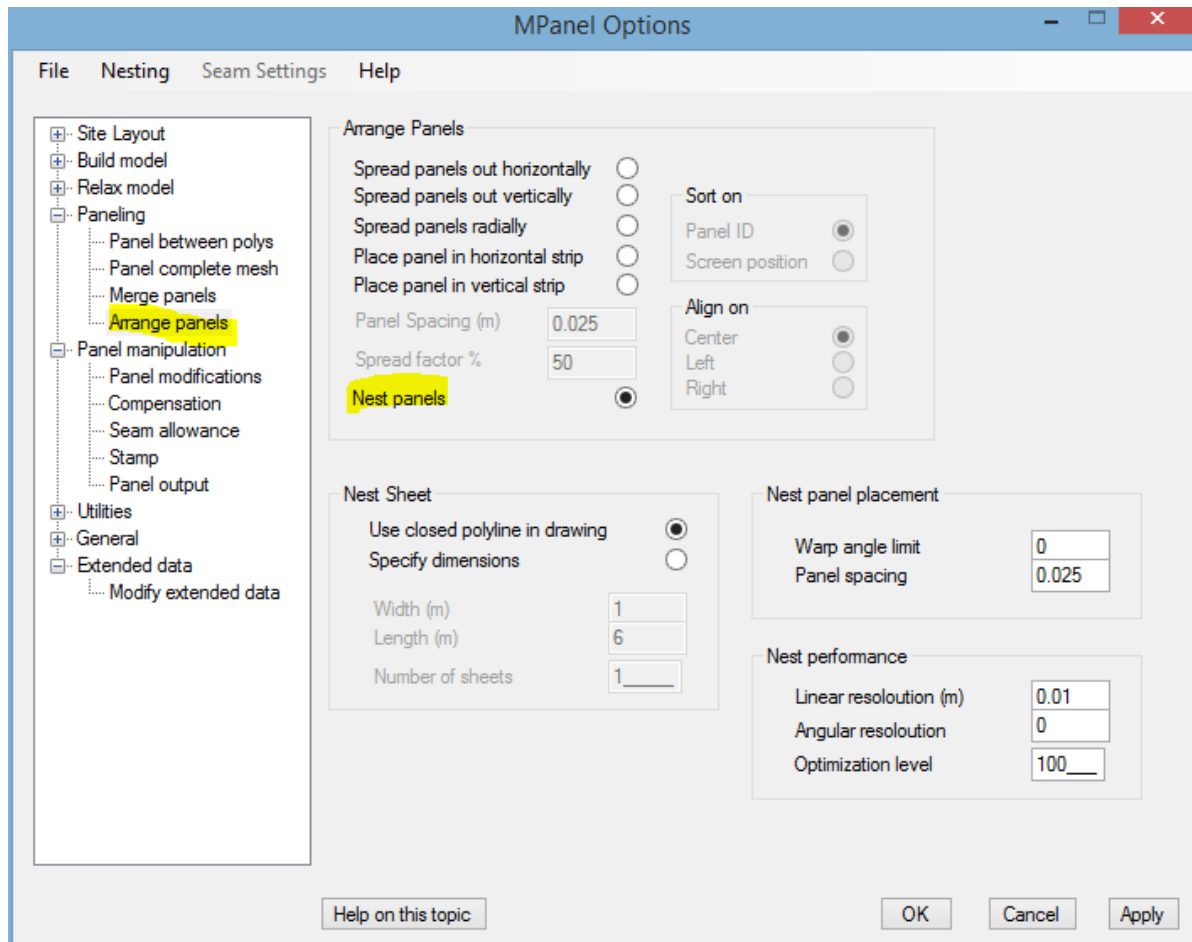
This tool works the same as above, except that the guide lines should be drawn vertically

Nest Panels

This tool performs nesting on a set of panels to achieve fabric usage savings. Nesting is an optional module, a licence for it's use is needed. The nesting module is described on [this page](#).

Note 1: The Output tool strips all extended data from the panels, and modifies the panels ready for automatic cutting and plotting. For this reason panels that have been through the Output tool cannot be arranged. So the process is to arrange the panels on the roll, or nest them, and then run the Output tool on the panels.

Nesting



MPanel offers a simple interface to a powerful nesting library, supplied by Optimal Programs (www.optimalprograms.com)

The nesting library is an optional add in to MPanel, and a separate licence for it's use must be obtained from Meliar Design.

Nesting is performed on the 'Arrange Panels" button. The settings for the nest are specified in the "Nest" menu item in the Options toolbar, shown here.

Specified Sheet Nesting.

In simple nesting, the cutter table size is specified in the Nesting options. The panels to be nested are selected, and the program run. The panels will be nested into sheet sizes, and drawn out below the panel set.

The number of sheets can be specified, and as many panels as can be nested onto those sheets will be used. Or you can specify a large number of sheets (100) and then all the panels will be nested.

The spacing between the panels can specified.

The warp angle limit is the number of degrees that the panel is allowed to rotate from the basic panel angle during nesting. Large values, like 20 degrees, will give a tighter nest, but will allow the finished panel warp to be different from the design panel warp, which may cause difficulties with seam puckering, uneven stretching, etc.

The basic angle of the panel, ie the result if you specify a warp angle limit of 0 degrees, is either:

- As the panel is drawn on the screen, for panel without a warp line (ie, the panels should be horizontal before nesting)

- With the warp line horizontal, for panels that have a warp line added during the stamping operation.

Nesting into a closed polyline

In this case you draw the sheet boundary in CAD as a closed polylines. The sheet drawing and the panels are then selected, and the nest will arrange the selected panels inside the sheet boundary.

This can be used to:

- Nesting onto a partially used sheet or offcuts.
- Nesting around fabric faults.
- Nesting non critical small parts with a high warp angle limit around critical parts with a low warp angle limit.

Nesting Report.

After each nest, a report is sent to the MPanel text window in the following format:

```
Nested 7 of 12 panels
Used 2 of 10 sheets
Sheet 1 Panels: 4 Used Length: 95.5%
Sheet 2 Panels: 3 Used Length: 65.1%
Utilisation = 47.8%
```

Utilization is the used area percentage within the used length on each sheet.

Performance

Linear resolution affects how tightly the panels are nested. Low resolution values gives tighter nests, but will take longer to nest. A reasonable value to use is 1% of the sheet width.

Angular resolution is the number of degrees that the panel is tried at between the warp angle limits. So with a warp angle of 20 degrees and an angular resolution of 10 degrees, the panel will be tried at +/- 0, 10 and, 20. A smaller angular resolution will take longer to nest,

Optimisation level controls the numbers of attempts the nesting program uses to achieve the best nest. Increasing this number will increase the quality of the nest, but will take longer to nest. A value of 100 produces reasonable nests. Larger numbers will make better nests, but will take longer.

Progress Reporting

The nesting progress is shown by an increasing number in the Mpanel toolbar. The exact number that is needed to complete the nest depends on several factors, but is similar to the specified optimisation level.

A cancel button is shown in the MPanel text window to interrupt long running nests. Also the ESC key can be used when Mpanel has the focus.

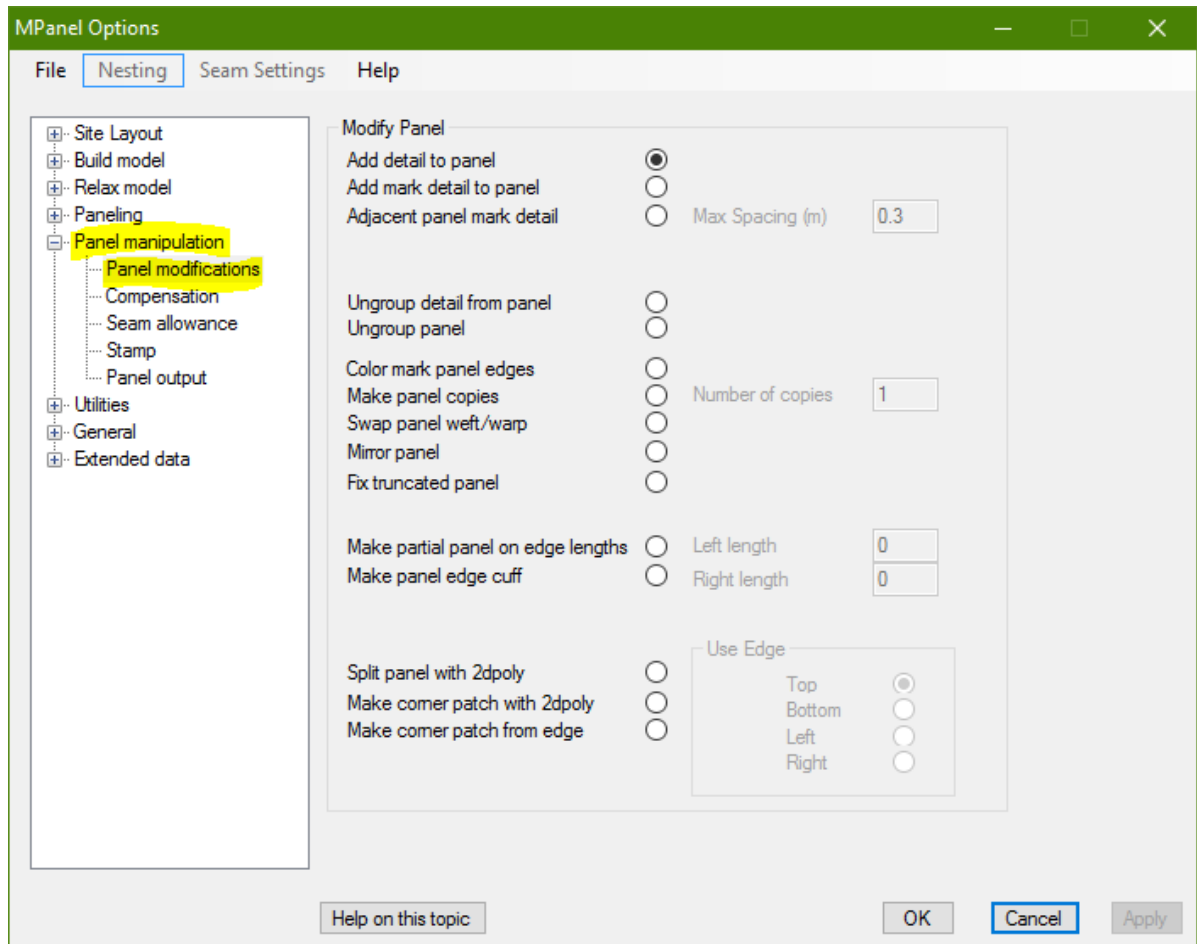
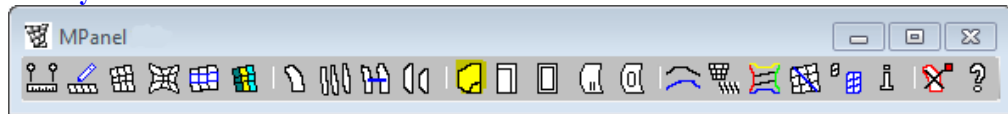
Nesting License.

The nesting library is supplied with a time limited license for evaluation, which will show a nag screen before each nest.

To licence the program, open the Nest Licence window and obtain the Computer ID. Send this to Meliar Design, and we will supply the Nest User and Nest Serial number.

Entering these in the Nest Licence window will licence the program permanently on this computer.

Modify Panel



The panel detail tools work on a single panel at a time, and perform of the following functions:

- Add Detail to Panel
- Add Mark Detail to Panel
- Mark Detail from Adjacent Panels
- Ungroup Detail From Panel
- Ungroup whole panel

- Colour Mark Panel Edges
- Make Panel Copy
- Swap Panel warp/weft
- Mirror Panel
- Fix truncated panel

- Split Panel with polyline
- Make partial from edge lengths
- Make panel edge cuff
- Make corner patch from polyline
- Make corner patch from edge

Add Detail to Panel

Add Detail to a Panel allows detail drawings made from polylines to be grouped with the panel. Select the panel and the required detail, and run the tool. The colour of the detail is set in the General Options. The details are correctly preserved through merging, compensation, seaming, stamping and output.

Note that the detail must be in the form of polylines (not 3Dpolys). The '[Poly conversion tool](#)' can be used to convert any other entities into polylines.

This can be used for example to add corner plate detail to panels by converting drawings of circles, arcs, etc to 2D polys with the poly conversion tool.

Or it can be used to draw in graphics or graphic markers, based on the panel grid.

Add Mark Detail to Panel

Add Mark Detail to a Panel works in a similar manner to the above, except that the seam allowance program will convert the detail into a mark line in the seam.

This allows information about graphic positions, structural reference points, etc to be carried through the panelling process to result in an edge mark in the final panel output.

The mark line will be generated by taking the middle node of the polyline and finding the closest point on the panel edge. So with a 3 point polyline node 2 is used, on a 2 point polyline node 1 (the start) is used. Usually the detail is close to an edge, to ensure an accurate mark.

Mark Detail from Adjacent Panels

Panel mark detail can be created from adjacent panels. This is useful if a main panel has several secondary panels that fit along one edge. Arrange the panels roughly as they would fit together, and colour the main panel Green. Select all the panels and run the tool. Detail marks will be added to the main panel to show where the adjacent panels will fit on the edge.

If there is a non zero setting for "max spacing", then additional detail marks will be added along the edges of all the panels, so that the detail marks are less than the max spacing apart.

Ungroup Detail From Panel

Detail can also be removed from a panel group. Apart from editing, the main use of this is to delete the internal panel grid after graphics or graphic markers have been added.

Ungroup whole panel

Also, the whole panel can be ungrouped. This occasionally has use when hand modifying panels. The entities can be regrouped using the Add Detail option.

Colour Mark Panel Edges

This draws a set of coloured polylines inside the panel, to identify which panel edges are top, bottom, left, right, according to the colour code:

Red	Left edge
Green	Right edge
Yellow	Top edge
Blue	Bottom edge

The coloured polylines cannot be used as a panel, they are just to indicate which edge is which on an existing panel.

Make Panel Copy

This makes a copy of the panel, with a new internal panel number. This is useful if you need several identical panels for the drawing, or for nesting. The number of copies can be specified. The new panels are drawn offset to the right of the original panel.

Swap Panel warp/weft

This makes a copy of the panel, with a new internal panel number, and with top_bottom swapped with left-right. This is useful if the original panel was created the wrong way around, or to help with merging panels "Top to bottom". The number of nodes on the panel edges are adjusted to meet panel rules. The new panel is drawn offset to the right of the original panel.

Mirror Panel

This makes a mirror copy of the panel, with a new internal panel number. This is useful when building a symmetrical model when only one half of the structure has been meshed. The new panel is drawn offset to the right of the original panel.

Fix truncated panel

When panelling with cross sections on the diagonal the corner panels are truncated due to the last cross section. This tool "fixes" the panel to include the missing part by extending the panel edges into a triangle. Note that this will change the orientation of the panel as the triangular panel must have left and right sides, so the inside edge of the panel becomes either a top or bottom edge.

Split Panel with polyline

The panel can be split into two, along a polyline that crosses the left and right (or top and bottom) panel edges. The new panels maintain the orientation of the original panels. The number of nodes on the each panel edge are adjusted to meet panel rules. This tool can be used for splitting panels into coloured segments, for artwork purposes, for making reinforcement patches, or bottom edge cable cuffs.

Make partial from edge lengths

A split panel is created, that is a specified length along the panel edges, from the top or bottom end of the panel. The new panels maintain the orientation of the original panel. This can be used for making top ring reinforcement patches, including start shaped reinforcements.

Make panel edge cuff

A small panel is created that is a specified trim length from an edge. This forms the basis of an edge cuff, which can be used to provide a cable tunnel on the mesh edge.

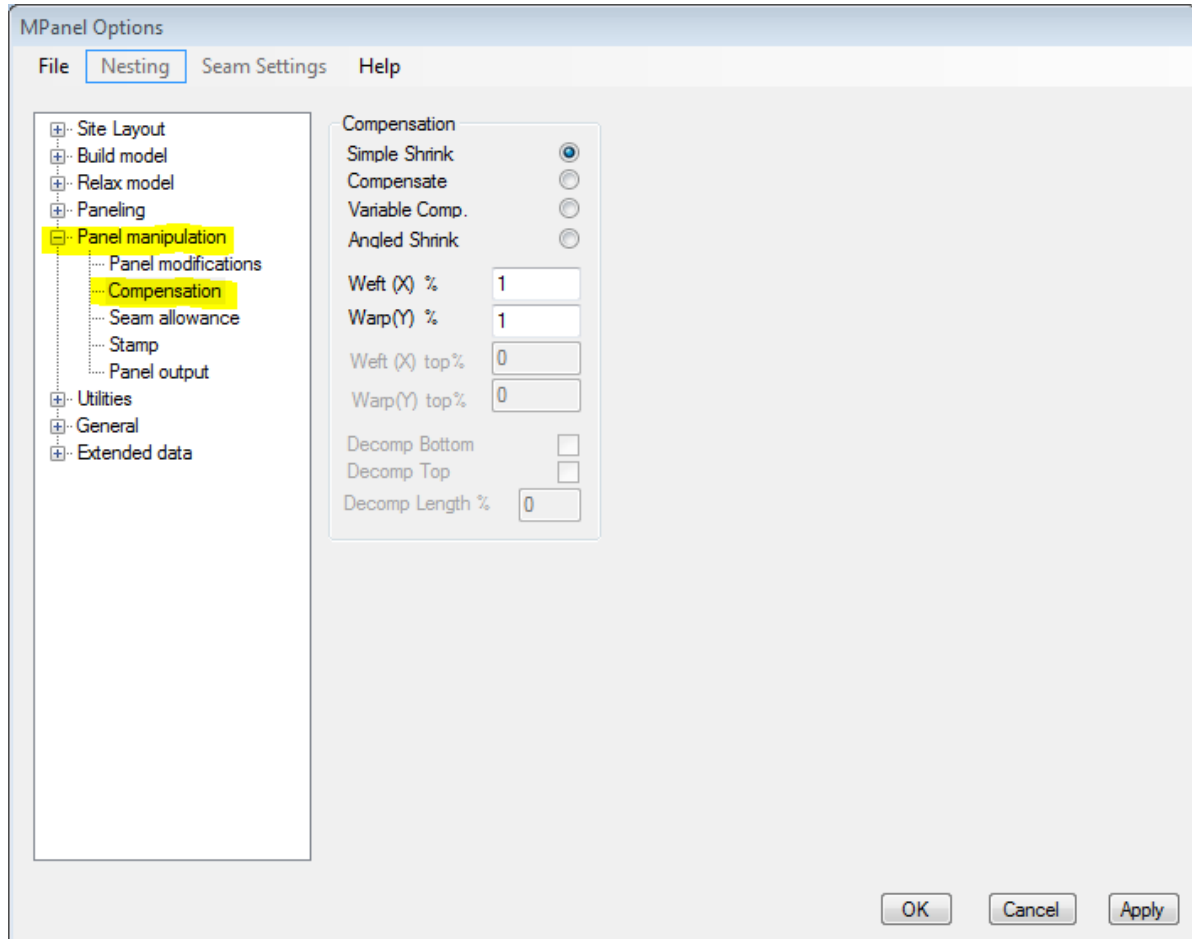
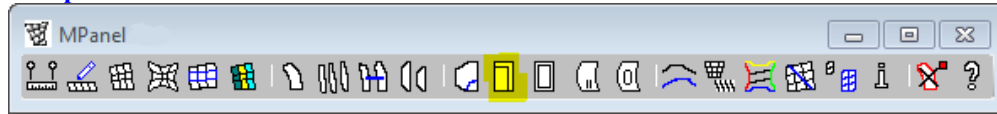
Make corner patch from polyline

A new panel can be defined by a polyline that crosses a corner of the original panel. The new panel always has a left, right, and top edge. This tool can be used to make reinforcement patches in panel corners. The number of nodes on the each panel edge are adjusted to meet panel rules

Make corner patch from edge

A new panel can be made from a panel edge, where the edge contains a significant angle change. This can be used to create a prototype reinforcement patch, that will be refined with other tools, for panels with an edge that spans a corner. The edge that the panel is derived from is specified in the options.

Compensate Panels



Compensate panels will shrink a panel or a set of panles by a controlled amount, to introduce pre-stress in the finished model. It can perform the following functions, as set in the options:

- Simple Shrink
- Compensate
- Variable Compensation
- Angled Shrink

Simple Shrink

This tool takes a panel and applies differential shrink to allow for pre-stress.

The panel Warp(Y) axis is the center line of the panel, running between the warp sides, and the panel Weft(X) axis is at right angles to this line. A 2% weft compensation will shrink the panel to 98% across the panel.

Compensate

This tool applies the Warp and Weft compensation locally, ie the Weft is locally across the panel and the Warp is locally along the panel. Optionally the weft compensation can be feathered out at the ends, over a percentage of the panel length.

The panel Warp(Y) is the curved line running halfway between the warp sides, and the panel Weft(X) axis is at local right angles to this line.

Variable Compensation

This tool applies the Warp and Weft compensation locally, ie the Weft is locally across the panel and the Warp is locally along the panel. The compensation value can vary linearly from the bottom of the panel to the top. Otherwise the operation is the same as Compensate.

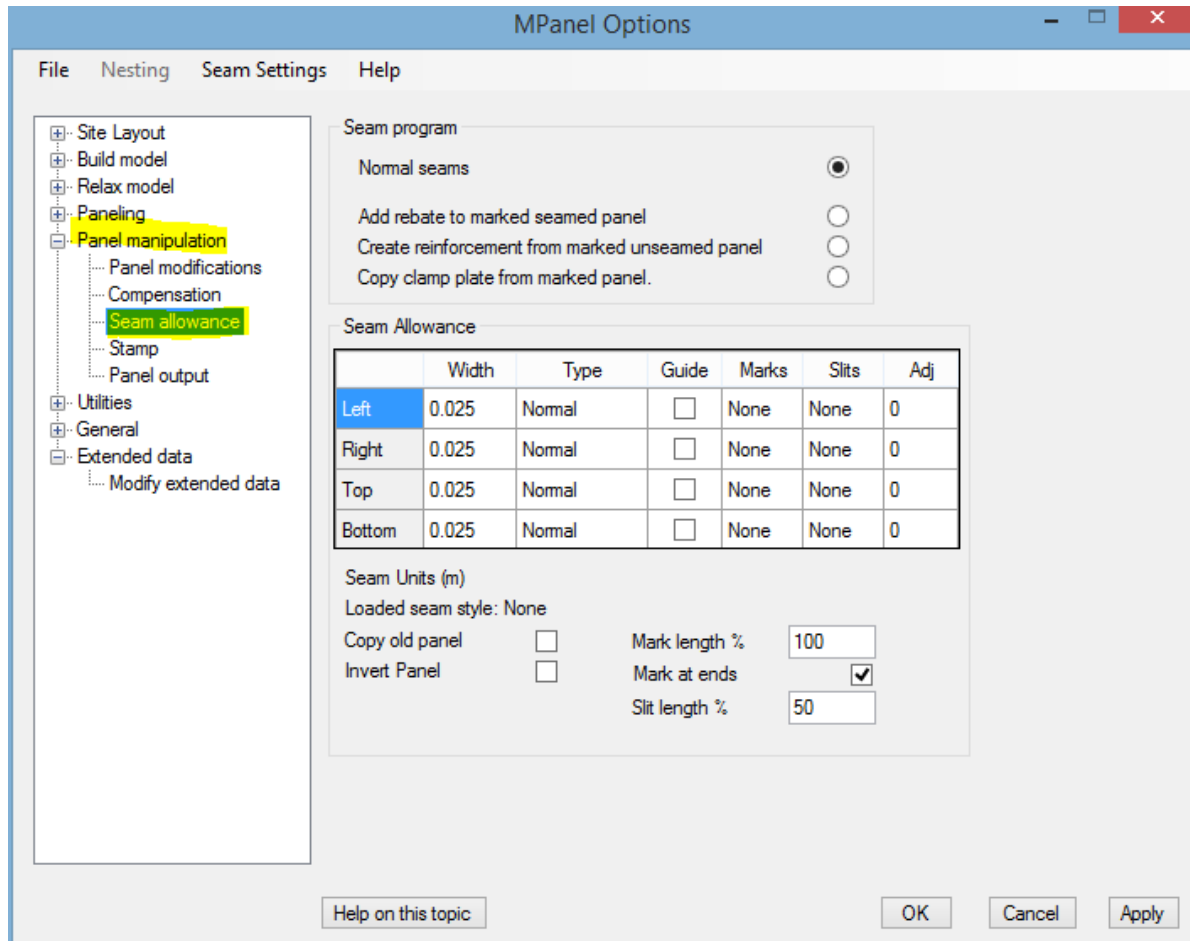
Angled Shrink

This tool applies the simple shrink method to the panel, but with the panel warp(Y) and the panel weft(X) axis being the Y and X axis on the screen. This allows the panel to be shrunk at an angle from the panel warp direction.

Standard Panel Warp Direction

Usually the "Standard Panel Warp Direction" checkbox should be ticked, as this forces the program to use a panel warp that lies between the two warp edges, which are marked on the panel in the extended data. In some situations you may want to override this, and use a panel orientation where the panel warp is midway between the most vertical panel edges as viewed on the screen in top view. You do this by setting 'Standard Panel Warp Direction = Off' under the 'Panel Tools' tab on the Options Screen.

Seam Allowance



This tool applies the seam allowance values entered in the options to the panel or to a set of panels. The seam allowance is entered in drawing units, ie if you are working in inches, the seam allowance is in inches, if you are working in millimetres the seam allowance is in millimetres.

In addition this tool can also create clamp plates, corner reinforcements, and rebates for cable cut outs from panels that have specially marked detail polys in them. These detail polys are generated by the Model builder tool "Add dimensioned corner"

Normal seams

The seam type can be specified for each edge, and optionally can generate a guideline to align the panel to during assembly. Different types of seams can be specified:

Normal	guideline is the original panel, usable for sewn edges
Lap Over	suitable for the edge on top in a lap seam
Lap Under	suitable for the edge underneath in a lap seam
Fold To	guideline to fold back to for a cable pocket
Butt	sets the mark length suitable for a butt seam
Offset	seam for very curved edges where parallel seams don't work
Offset Fold	cable pocket for very curved edges
Triangular	used to continue the fabric over a top ring

In addition the original panel can be added to the new panel as internal detail.

Tick Marks

Optionally, seam allowance can add tick marks, or balance marks to the new drawing. These can be used as a line up aid during production. Usually these are added along the warp, or long edge of the panel, but they can also be added on the top and bottom edges. The tick marks can be specified as:

A Number	like "7", which will generate 7 equally spaced tick marks on the side.
Std	rules for generating standard tick marks will be followed.
Nodes	tick mark is generated at each node along the edge
Detail	tick mark is generated from the centre node of any mark detail in the panel
None	no tick marks

Standard rules can be set under the "Seam Settings" menu item.
The position of the marks will be adjusted according to the type of guideline specified.

When the tick marks are specified by a number you can choose to have them starting and finishing at the panel ends, or starting part way into the panel. Often using marks partway into the panel causes less confusion, as the marks at the ends can be lost with some seam allowances.

Slits

Optionally Slits can be added to any edge. These are usually used on fold to edges to allow the edge to take a curved shape. The slit length is set as a percentage of the seam width. The slits are specified in the same manner as the tick marks.

Each seam edge also has an adjustment column, that will adjust the seam width in or out by a small amount. This has use when making a rope edge or catenary edge, when a fabric allowance is needed to go around the rope, without altering the real seam width.

The whole panel can be inverted, for marking on the underside.

Seam Styles

You can save any number of named seam styles for future use, by using the "Seam Styles" on the menu bar. So if you had a panel seam arrangement defined for a small cone tent, you could save it as "Small Cone". When you load in a seam style, the style name is displayed alongside the seam table.

If you saved seam styles on versions of MPanel before v13, then the named seam styles will still be available. If you did not name them, they will be named "UnNamed Style 1", etc.

Standard Alignment marks and slits.

Standard alignment marks and slits are defined by the following rule settings:

- Distance from start of edge to first mark
- Distance from end of edge to last mark
- Spacing between the marks
- Minimum number of marks.

Standard Panel Warp Direction

Usually the "Standard Panel Warp Direction" checkbox should be ticked, as this forces the program to use a panel warp that lies between the two warp edges, which are marked on the panel in the extended data. This then defines the top, bottom, left, right edges independently of the panel orientation on the screen. In some situations you may want to override this, and use a panel orientation where the top, bottom, left, right edges are as viewed on the screen in top view. You do this by setting Standard Panel Warp Direction = Off, accessed under the 'Panel Tools' tab.

Corner tools

Add rebate to seamed panel

This takes panel with specially marked detail polys, cuts a rebate from the seam to allow space for the cable connection, and truncates the corner. This should be applied to a panel that has a seam allowance already added, and the panel should have 1 or 2 reinforcement marks in place.

Create reinforcement panel

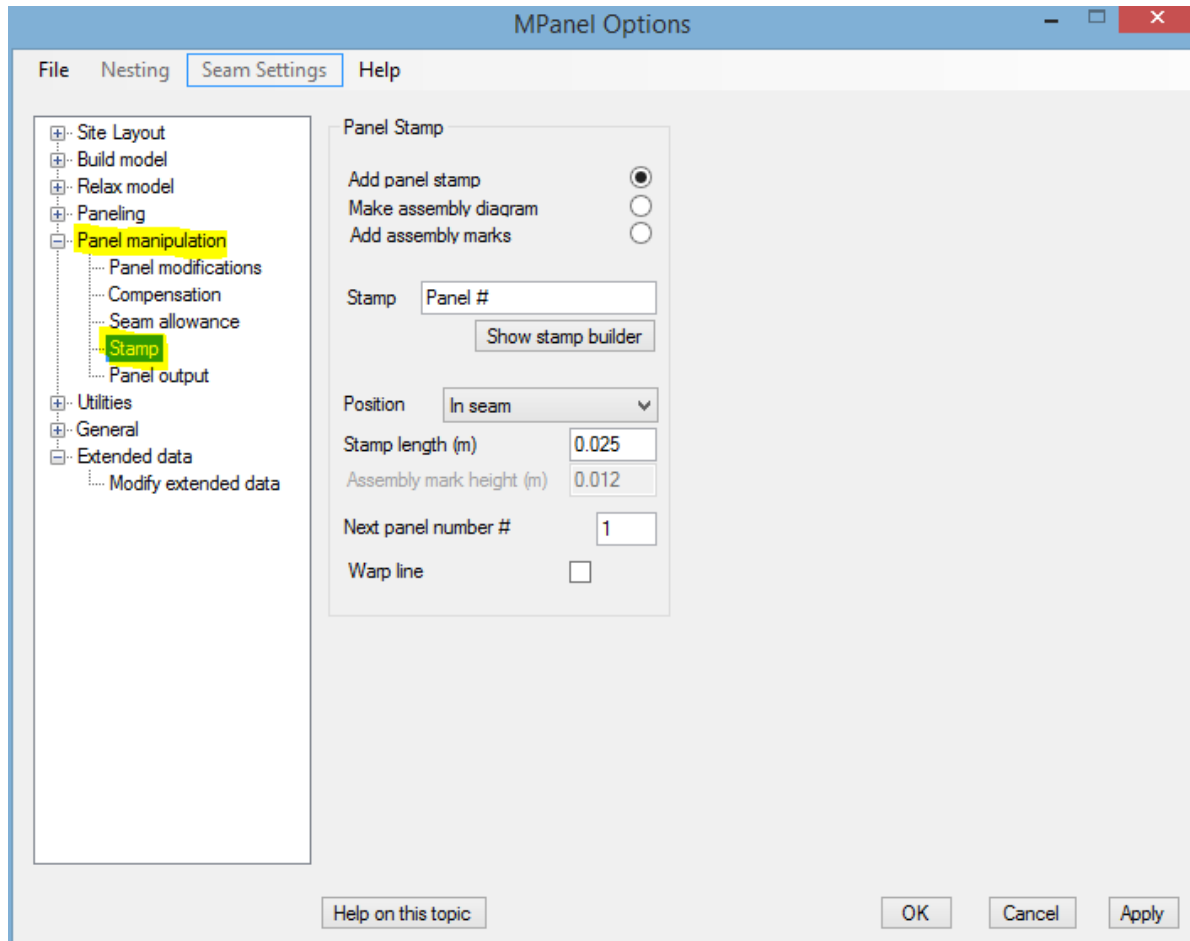
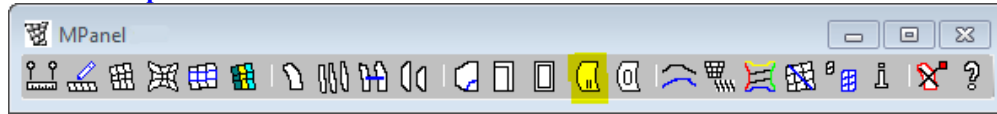
This creates a small reinforcement panel that will fit under the clamp plate, around the rebate marks, and trim to the panel finished edge. To ensure that the reinforcement fits to the panel finished edge this tool should be used on the panel before seams have been added.

Create clamp plate

This creates a prototype outline for the design of a metal clamp plate. It can be derived from the seamed or un-seamed panels. The outer edges of the clamp plate will be joined together in a continuous polyline. The bolt hole marks will be replaced with higher definition polylines representing a circle, with a centre mark.

The clamp will need further design work, including shaping the rear of the clamp plate to accept the connection to the pole. It may be useful to un-group the component parts for this, or to turn group selection on or off in the usual manner. (In AutoCAD with Ctrl Shift A)

Panel Stamp



Panel Stamp adds text and lines to the panel to assist with manufacture and assembly. There are 3 functions:

- Add panel stamp
- Make assembly diagram
- Add assembly marks

Add panel stamp

Plots the panel name on the panel in a choice of positions:

- Inside the seam, parallel to the bottom panel edge.
- Along to the panel centre line
- In the middle of the panel
- In a cut off tag outside the seam

Optionally a warp line indicating the warp direction of the panel can also be drawn along the panel centre line, near the bottom panel edge. The warp line is used to indicate the preferred panel orientation on the roll, and is used by nesting programs and cutter layout software to control the warp direction.

The length of the warp line is specified as the stamp length, and can be used as a final check that the panel was plotted / cut with no scaling.

The text is scaled to produce a stamp about the same length, even when no warp line is plotted. To force a smaller text size include some blank spaces at the end of the panel name.

The panel name can be based on:

- the unique panel ID number that MPanel assigns to each panel
- a specified number that is incremented with each stamped panel.

The full panel name can be built using the panel name builder, and can be simple or complicated, like:

P1
Panel 1
Job 15-83a Celtic Brown Panel 1

The panel name builder will remember that last used partial names, and optionally can also contain a user supplied list of names from the Mpanel User Setting.ini file, in the users documents directory, in the style of:

[STAMP NAMES PART 1]
Stamp 1 = My special panel name
Stamp 2 = Another special name

Make assembly diagram

An assembly diagram is a set of lines showing which panels edges join together. This tool will take a set of panels, arranged approximately as they would be assembled, and draws in a possible set of assembly lines. If the assembly lines are incorrect or incomplete, they can be editing in CAD by drawing simple lines.

The assembly line drawing can be used to indicate panel assembly for production, but it's main use is to define the position of panel assembly marks to be plotted at the next stage.

To get the best possible set of assembly lines arrange your panels so that:

- Panels do not overlap
- Joined edges are close together
(close means less than their edge length)
- Centre points of joined edges are approximately square on to each other
(square on means that the line will make an angle of more than 45 degrees to the

panel edges)

Make assembly mark

An assembly mark consists of text added in the panel seam where an assembly line touches the panel.

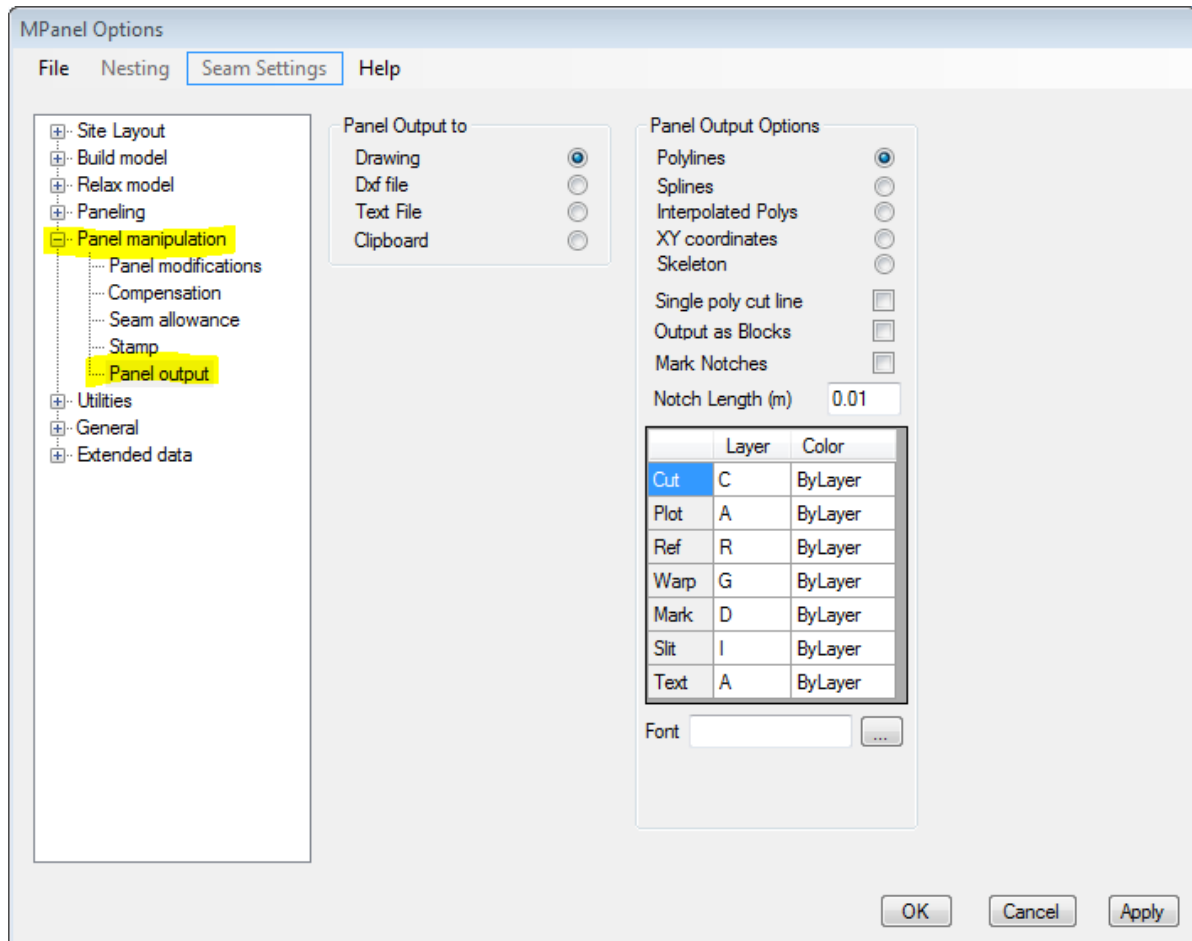
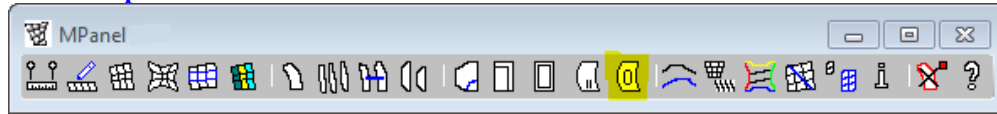
The assembly mark is a unique text of the form:

m xxx-yyy where xxx and yyy are the respective panel ID numbers.

The text height can be directly specified. Usually the panels will be assembled with the two "m" marks lined up.

Assembly marks can be used instead of, or in addition to normal panel stamping.

Panel Output



Panel Output takes a drawing of one or several panels, typically consisting of inner and outer polylines, tick marks and identifying text, and makes a copy of the panels in a form that is suitable for direct output to a cutter, plotter, or drawing grid.

The panel output can be to:

- **Drawing** the present Cad drawing
- **DXF file** a specified file in dxf release 14 format
- **Text File** a specified file in text format
- **Clipboard** the windows clipboard in a format that will paste into spreadsheets correctly.

The output type can be:

Polylines

A copy of the original polylines. Optionally they can be joined together into a single polyline per panel. Optionally tick marks can be converted into notches, cut into the edge, of a specified length. Optionally the panel can be drawn as a block. The drawing entities will be separated onto layers as defined by the layer table.

Splines

A spline is drawn through the nodes of the original polylines. Optionally the panel can be drawn as a block. The drawing entities will be separated onto layers as defined by the layer table.

Interpolated polylines

An new polyline is drawn for each edge, based on the spline that would pass through the nodes of the original polylines. Optionally they can be joined together into a single polyline per panel. Optionally tick marks can be converted into notches, cut into the edge, of a specified length. Optionally the panel can be drawn as a block. The drawing entities will be separated onto layers as defined by the layer table.

XY coordinates

The XY coordinates of the edge nodes are generated in the specified format. Fractional inch formats are supported, as well as decimal inch and settings suitable for metric measurements. A minimum X value can be specified to give a offset from the roll edge for the panel. If the output is to the drawing, a copy of the panel will also be drawn, with the node numbers marked.

Skeleton Panel

A line outline of the panel is drawn, with each edge having its length, and the dip of the edge curve from the line mid point specified. This can be used for panel checking, or can be used to generate by hand the edge curves for simple panels.

Colours and Layers

Normally all output is drawn with the colour set to 'By Layer'. This lets you assign different colours to the output layers, to see that the panel is correctly split into the different layers. If you need to suppress a layer, just give it a blank layer name.

The panel entities will be separated onto different layers. the default layer names are:

Cut curve	Layer C
Annotation	Layer A
Lineup marks	Layer D (drill)
Warp Line	Layer G (grain)
Reference	Layer R
Slits	Layer I
Text	Layer A

Use of the above features creates panels that comply with the AAMA specification for DXF output.

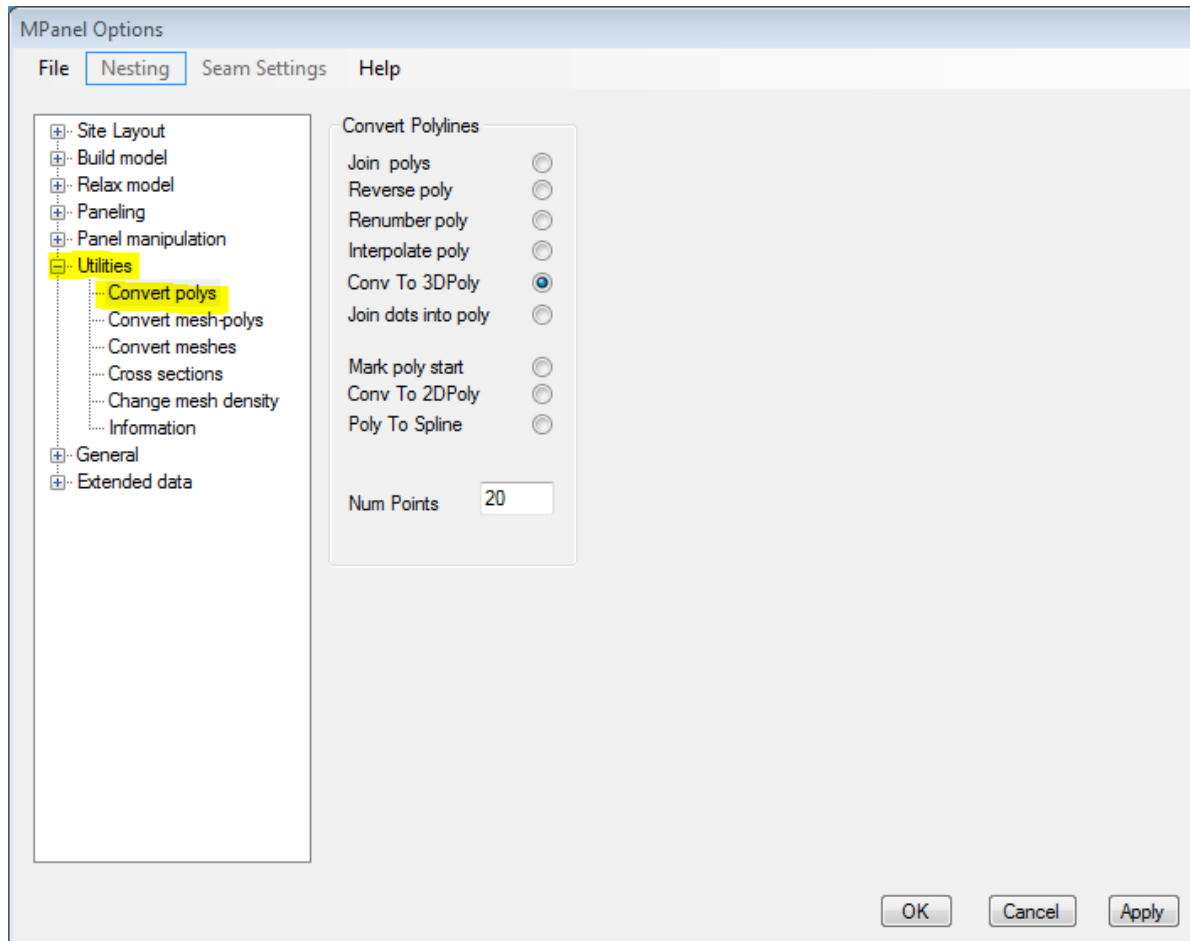
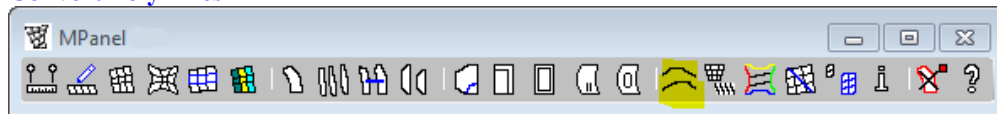
Additionally the colour can be set individually for each layer. This has use if the cutter / plotter you are uses colours rather than layers to define the actions of each layer.

Fonts

Normally the stamp text font is not specified. This allows the cutter / plotter to use it's preferred font, and should result in high speed plotting of the text. With some cutter / plotters it is better to specify the font, which can be done in the options. The font specified should be one that is available to the plotter / cutter. Usually the font can be selected from the windows font table. Non window fonts (such as the AutoCAD shx fonts) can be specified by pre-pending an asterisk. So *txt will specify the AutoCAD font txt.shx, if that is available on the cutter plotter.

Some cutters are slow when drawing text. Specifying "MP vector font" substitutes polylines for the text outlines, and can offer a useful speed increase.

Convert PolyLines



Convert polylines can perform different operations on 3Dpolys.

Join Polys

Will join any number of 3D polylines together. It will cope with small gaps, by moving a polyline end to cover the gap. The direction of the new polyline is the same as the direction of the first 3Dpoly selected for joining. If the joined 3Dpoly makes a closed figure, then the start and end of the joined 3Dpoly will be the start of the first 3Dpoly selected for joining.

Convert to 3dPoly:

A polyline, polyline with embedded arcs, arc, circle, ellipse, elliptical arc, line or spline will be converted to a 3Dpolyline. The number of points in the 3Dpoly is set in "Num Points". This tool has particular use as many MPanel tools require 3D polylines for input. Supports multiple conversions at the same time.

Reverse Poly

Will reverse the direction of a 3D polyline. Supports multiple 3Dpolys at the same time. Will also work with lines.

Renumber poly

Will redraw the 3D poly with the specified number of points. If there is a sharp corner in the 3D polyline then a node will be placed on that corner. More than 1 sharp corner will generate a warning. Supports multiple 3Dpolys at the same time.

Interpolate poly

Will redraw the 3D poly with the specified number of points. The points will be spaced along a spline that runs through the original points. In general this will give a better shaped 3Dpoly than "Renumber 3Dpoly", as long as the original

3Dpoly was smooth. Supports multiple 3Dpolys at the same time.

Convert to 3Dpoly

A polyline, polyline with embedded arcs, arc, circle, ellipse, elliptical arc, line or spline will be converted to a 3Dpolyline. The number of points in the 3Dpoly is set in "Num Points". This tool has particular use as many MPanel tools require 3D polylines for input. Supports multiple conversions at the same time

Join dots into Poly

Takes a set of points (dots) and joins them together into a polyline, on the basis that the closest points are joined together. Generally this method will give an acceptable poly, but it is possible for loops or kinks to form when the dot spacing varies.

Convert to 2Dpoly:

A 3Dpoly, polyline with embedded arcs, arc, circle, ellipse, elliptical arc, line or spline will be converted to a polyline. The number of points in the poly is set in "Num Points". The Z elevation is set to zero. This tool has particular use when adding panel detail, as MPanel require polylines panels. Supports multiple conversions at the same time.

(In AutoCAD, the 2Dpoly is drawn as a LWpolyline)

(In Rhino, the 2Dpoly is drawn as a polyline, with the Z value of each point set to zero)

Poly to Spline:

A 3Dpoly will be converted to a spline. It has use when outputting lines with a small number of data points, but requiring a smooth curve.

Mark poly start:

A marker triangle will be drawn at the start of each poly, to show it's direction. This is useful when making cross sections into panels or meshes.

Will also work with lines.

AutoCAD Note:

Although the tools are designed to work primarily with 3dpolys, the following tools will also work with 2dpolys (called polylines or LWpolylines in AutoCAD)

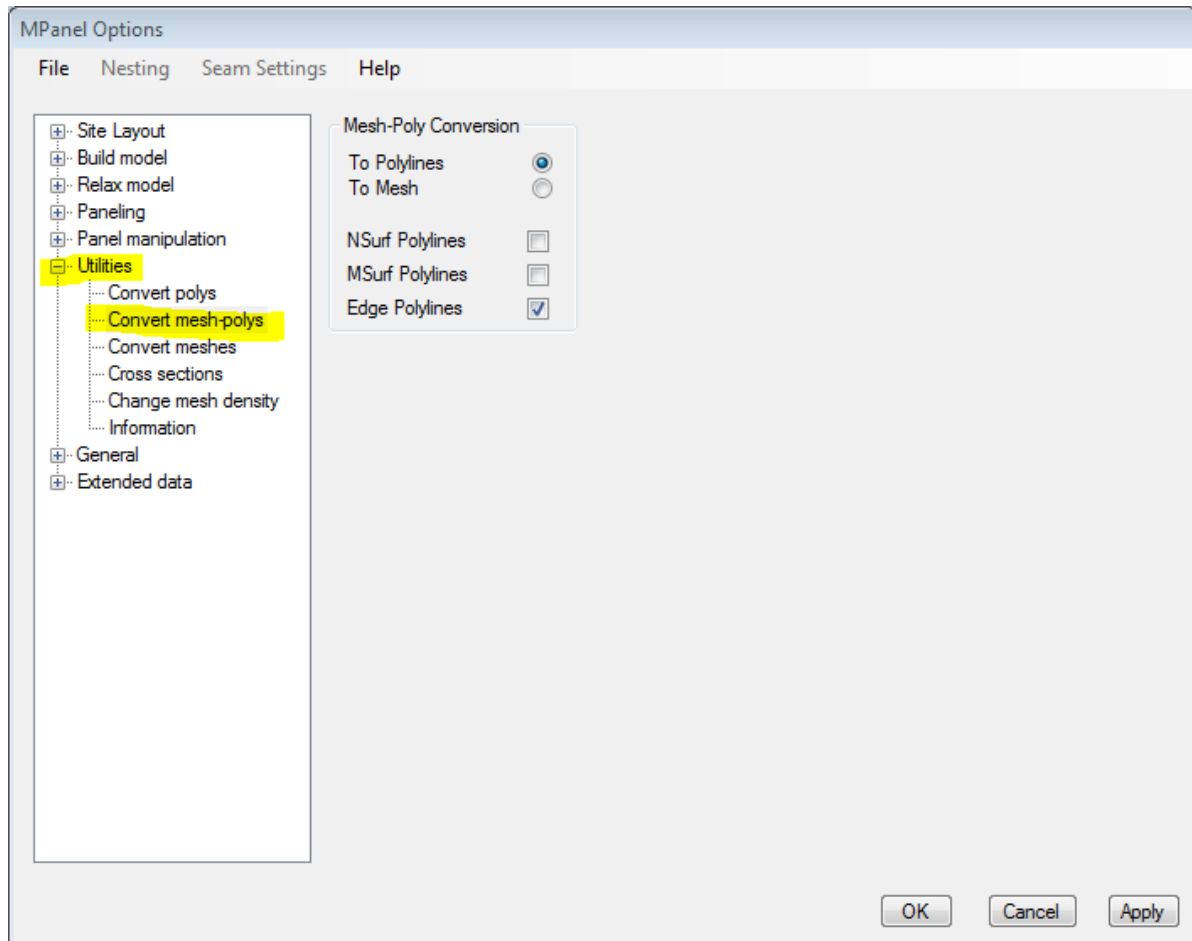
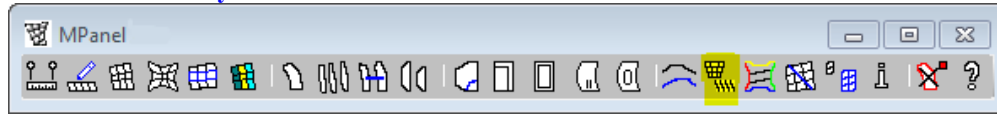
Join polylines
Reverse polylines
Renumber polylines
Interpolate polylines
Poly to spline
Mark poly start.

However in reading the 2dpolys the following restrictions apply:

Elevation is ignored
Polyline OCS is ignored
Embedded arcs are ignored

In Rhino there is no difference between a 3dpoly and a 2dpoly, so this section does not apply to Rhino users.

Convert Mesh/Polylines



Convert Mesh/Polylines can perform these functions:

- Convert Meshes into Polylines
- Convert Polylines into meshes

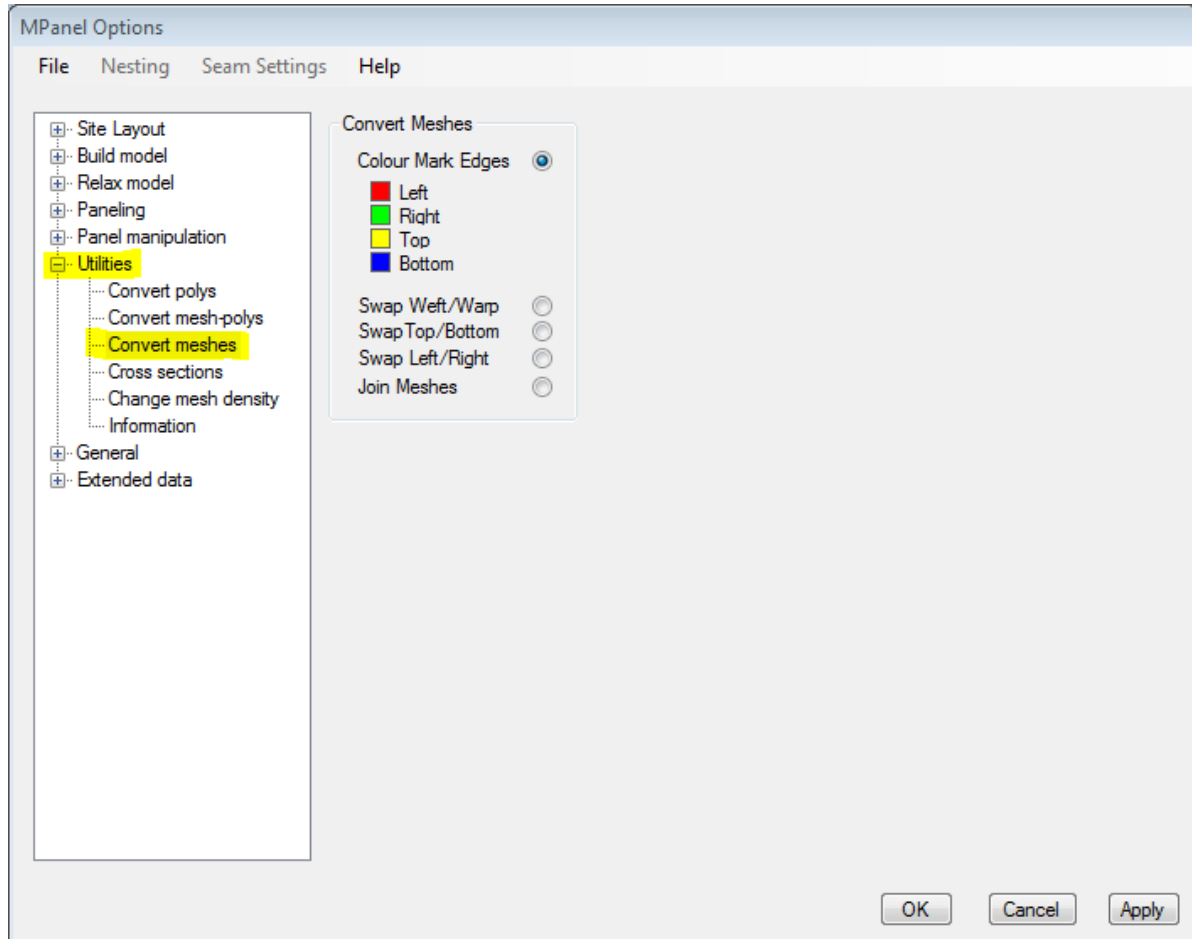
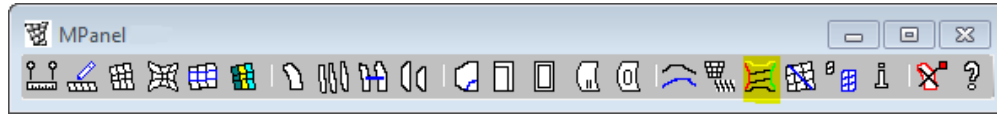
Convert Meshes into Polylines

Will convert a mesh into a series of 3D polylines. Either the NSurf, Msurf, or the Edge polylines can be drawn. This tool will work on multiple meshes, and any duplicate polylines are suppressed. Optionally the polylines drawn can be grouped together.

Convert Polylines into meshes

Will take a group of 3D polylines and convert them into a Mesh. If the mesh is too 3D curved, the polylines may be meshed in an incorrect order. The solution to this is to create 2 or more meshes, each of which will sort out its polylines correctly, and then use the Join Meshes command.

Convert Meshes



Convert Meshes is a modelling tool that can report and alter the orientation of meshes, helping to get consistent model orientation, and offering a different direction to panel in. The tool can perform the following functions:

- Colour Mark Edges
- Swap Weft and Warp
- Swap Left and Right
- Swap Top and Bottom
- Join Meshes

Colour Mark Edges

This will take any number of meshes, and draw a coloured outline just inside the mesh. This allows inspection of a model to see if the meshes are oriented correctly, and it can identify edges to determine which edge a cable is on. The colour coding is:

Red	Left edge
Green	Right edge
Yellow	Top edge
Blue	Bottom edge

Swap Weft and Warp

Will redraw the mesh with the weft and warp directions swapped. This has particular use when you want to panel a

complete mesh along the "Mesh weft" direction.

Swap Left and Right

Will redraw the mesh with the left and right edges swapped

Swap Top and Bottom

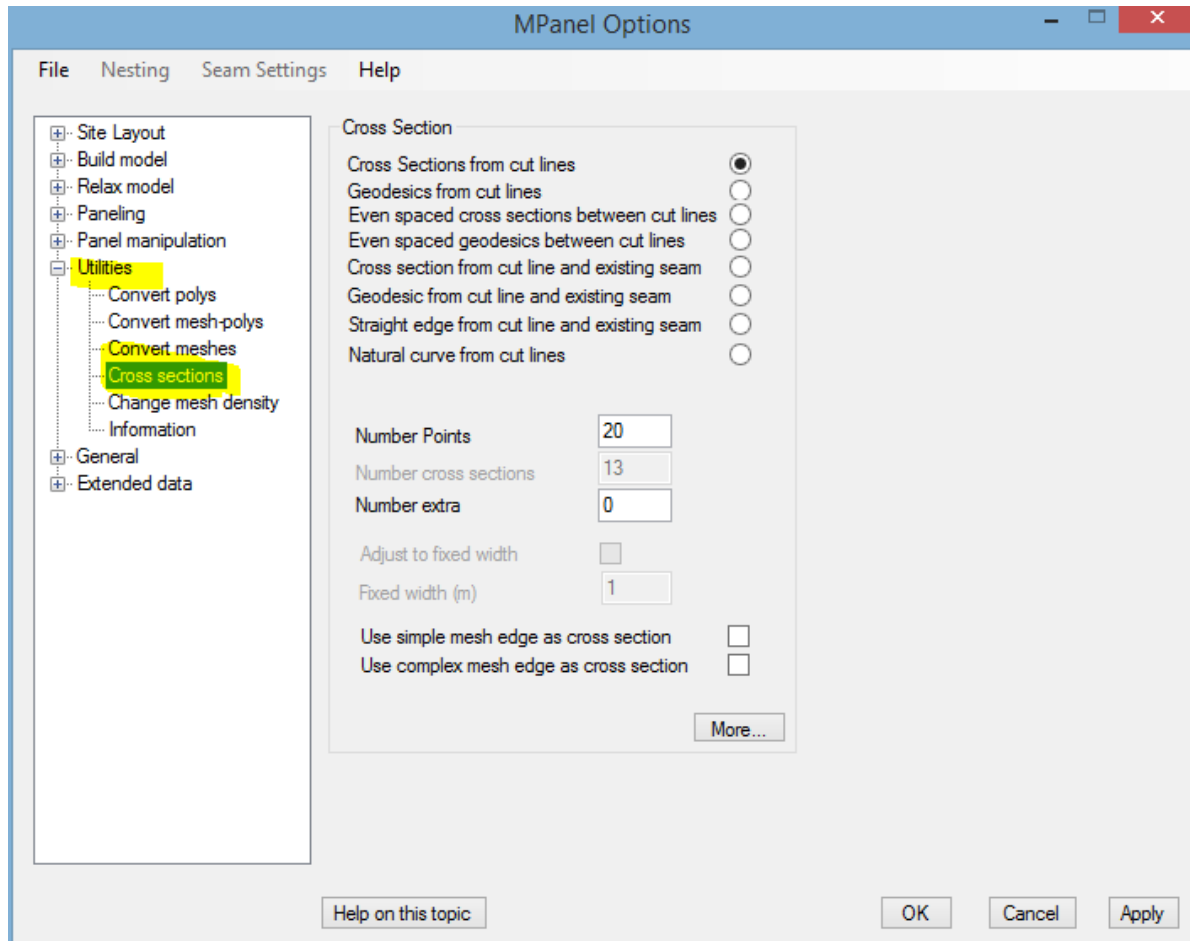
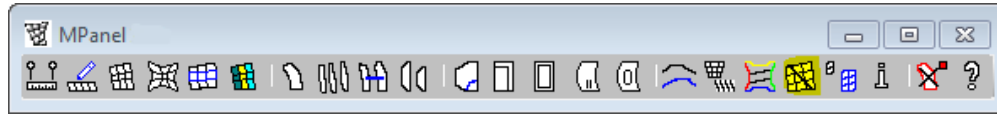
Will redraw the mesh with the top and bottom edges swapped

Note: Extended data that existed on the meshes is preserved by all the "swap" options of this tool, so special settings for weft/warp stress ratio, or cable tension ratios, will be preserved.

Join Meshes

Will take a set of meshes with common edges, and if possible redraw them as a single mesh. This has use in recreating meshes, and in making whole model meshes that look smoother in renderings and visualisations. If possible the original mesh warp direction is preserved.

Cross Section



Cross Section can be used to make cross sections across several meshes, that can be used for panel seams, building new meshes, or drawing lofting curves. It can perform the following functions, as set in the options:

- Cross Sections from cut lines
- Geodesics from cut lines
- Even spaced Cross sections from cut lines
- Even spaced Geodesics from cut lines
- Cross section between cut line and existing seam
- Geodesic between cut line and existing seam
- Straight edge from cut line and existing seam
- Natural curve from cut lines

with further modifying options of:

- Use mesh edge as cross section.
- Add extra cross sections
- Fixed spacing

and more advanced settings are available with the More... button

Cross Section from cut lines

Cross Section takes any number of lines as cutting lines, and drops an imaginary vertical plane down from the lines through any number of meshes. This is usually achieved by viewing the meshes and polylines from above in XY view, and drawing in the cutting lines. The cutting lines and the mesh(s) are selected, and the cross section tool run.

If the number of cross section points is not specified, the tool will put in as many as possible. If the number is specified, they will be evenly spaced between the cross section points. The cross sections will be pointing in the same direction as the cut line.

Cross Section will create the cross sections as 3D polylines. These cross sections can be used as panel seams, lofting lines, or polylines to build new meshes from.

If the cross sections are close to a mesh edge, you should visually check the result to ensure that the cross section is inside the mesh. The program does not check for this.

The start of the cross section is the point in the mesh closest to the cut line start, similarly with the end. In heavily curved cross sections you may have to adjust the z value of the cut line ends to obtain the required cross section.

If a cut line lies directly over a mesh edge, optionally the mesh edge can be used instead of the cross section.

Extra cross sections can be specified that will be spaced between the supplied cut lines. These extra cross sections will normally be curved to fit between the main cross sections.

Geodesic

Geodesics are curves whose end points are defined by the cutting line and the meshes, but whose inner points follow a geodesic line across the meshes. This is achieved by allowing the line to relax in the meshes. This can produce lines that are useful for panelling difficult shapes with a minimum of material wastage.

There can be any number of lines as cutting lines, across any number of meshes

The number of points in the geodesic line is set in the options. You should visually check the result to ensure that the geodesics near the mesh edges have stayed inside the mesh, the program does not check this.

The starting curve for a geodesic is the cross section, so your cut lines should be able to define a good cross section to obtain a valid geodesic.

If a cut line lies directly over a mesh edge, optionally the mesh edge can be used instead of the cross section.

Making geodesics is an iterative procedure, the number of iterations is set in the Advanced options.

Extra cross sections can be specified that will be spaced between the supplied cut lines. These extra cross sections will normally be curved to fit between the main cross sections.

Even Spaced Cross sections or Geodesics.

This creates a set of cross sections that are spaced between two cut lines. The number of cross sections produced is specified in the options. The spacing is adjusted so that the panels produced from the cross sections would be the approximately the same widths, ie the cross sections are evenly spaced in the 3d model. The width achieved is reported in the text window

The start and finish points of the cut lines for the intermediary cross sections lie on direct lines between the start and finish points of the two supplied cut lines. In some cases this means that the supplied cut lines need to be extended, to ensure that the full mesh is caught by the intermediary cut lines.

If one of the two supplied cut lines lie directly over a mesh edge, optionally the mesh edge can be used instead of the cross section.

Extra cross sections can be specified that will be spaced between the evenly spaced cross sections. These extra cross sections will normally be curved to fit between the main cross sections.

Cross sections or Geodesics between a cut line and an existing seam.

This creates a set of cross sections that lay between an existing cross section and the cut lines. This can be used to make the "last" set of cross sections to a mesh edge when making cross section sets one at a time.

you can specify fixed spacing, which will create fixed width panels. The method is to select the meshes, an existing 3Dpoly that is a seam, and a cut line that is in approximately the correct position for the next seam. Specify the required panel width as the spacing. When the tool is run the cut line will be adjusted to produce a new seam poly that will have the correct spacing; so when you panel between the old and new seams the panel width will be close to the spacing figure.

This procedure is then repeated to produce the next seam. The panel width produced will be more accurate if the cut line is in approximately the correct position.

Extra cross sections can be specified that will be spaced between the supplied cut lines. These extra cross sections will normally be curved to fit between the main cross sections.

Straight Edge

Straight Edge is used to assist in creating panels on which one seam will be a straight edge. The method is to select the meshes, an existing 3Dpoly that is a seam, and a cut line that defines the intended seam position. When the tool is run the new cross section will be adjusted to achieve a seam that will give a straight edge on the cut line side of the panel., the other edge will be curved.

This has use with some coated fabrics, where the coating is left of a strip on the roll edge to allow good material welding.

This procedure is then repeated to produce the next seam, using the new seam as the existing 3Dpoly for the next seam.

Straight edge can be used with Fixed Spacing and with Extra Cross Sections.

Natural cross sections

Natural cross sections make curved cross sections, defined by a cut line and a mesh, that mimic the path of the threads in the mesh. If the cut lines coincide with thread ends then these make cross sections with the same curved path as the "Panel a complete mesh" tool. This results in aesthetically pleasing seams, with each seam echoing the mesh shape.

Cut line direction warning

If adjacent cut lines in a set are pointing in different directions then the cross sections will also be pointing in different directions. This will give difficulty in panelling, as the panelling program will generate a "twisted" panel. A warning message is given in this situation.

Options

Use mesh edge as cross section

A simple mesh edge can be used as a replacement to a cross section. Draw a cut line that exactly passes through the mesh corners, and the mesh edge will be used in place of the cross section. The mesh edge poly will have it's number of nodes adjusted to the cross section number of points, and the direction changed to agree with the cut line direction.

A complicated mesh edge can be used as a replacement to a cross section. Draw a cut line that exactly passes through the some mesh edge nodes, possibly on different meshes. The shortest path between the mesh edges nodes over all the mesh external edges, will be used in place of the cross section. The mesh edge poly will have it's number of nodes adjusted to the cross section number of points, if there is a sharp bend in the mesh edge poly a node will be placed on the bend, and the mesh edge poly direction is forced to agree with the cut line direction.

Extra cross sections

Extra cross sections are used to improve the panel shaping by forcing the panel shape to pass through internal cross sections as well as the edge cross sections. They are drawn out in grey, and are curved to fit between the edge cross sections.

A set of cross sections, including extra cross sections, can be panelled in one operation use the "Panel a coloured set of polylines" tool

Advanced Options

Use 3dpoly as cut line

To create more complicated cross sections, it is possible to use a 3Dpolyline as the cutting line. This allows the creation of pseudo curved cross sections, when viewed from above. This method can also help to make cross sections to fill the gap between a curved mesh edge, and the first straight cross section.

Allow 3Dface as mesh.

It is also possible to use a set of 3Dfaces as the mesh that the cross section is based on. These are usually produced when a model is imported into the Cad program. This can allow panelling or further modelling on non MPanel geometry. (In AutoCAD a polyface mesh can then be exploded into a set of 3dfaces)
(In Rhino any irregular mesh can be used as a set of 3Dfaces)

Use curved extra cross sections

Extra cross sections will normally be curved to fit between the main cross sections. This is an option to use straight extra cross sections instead.

Use mesh N lines

Use mesh M lines

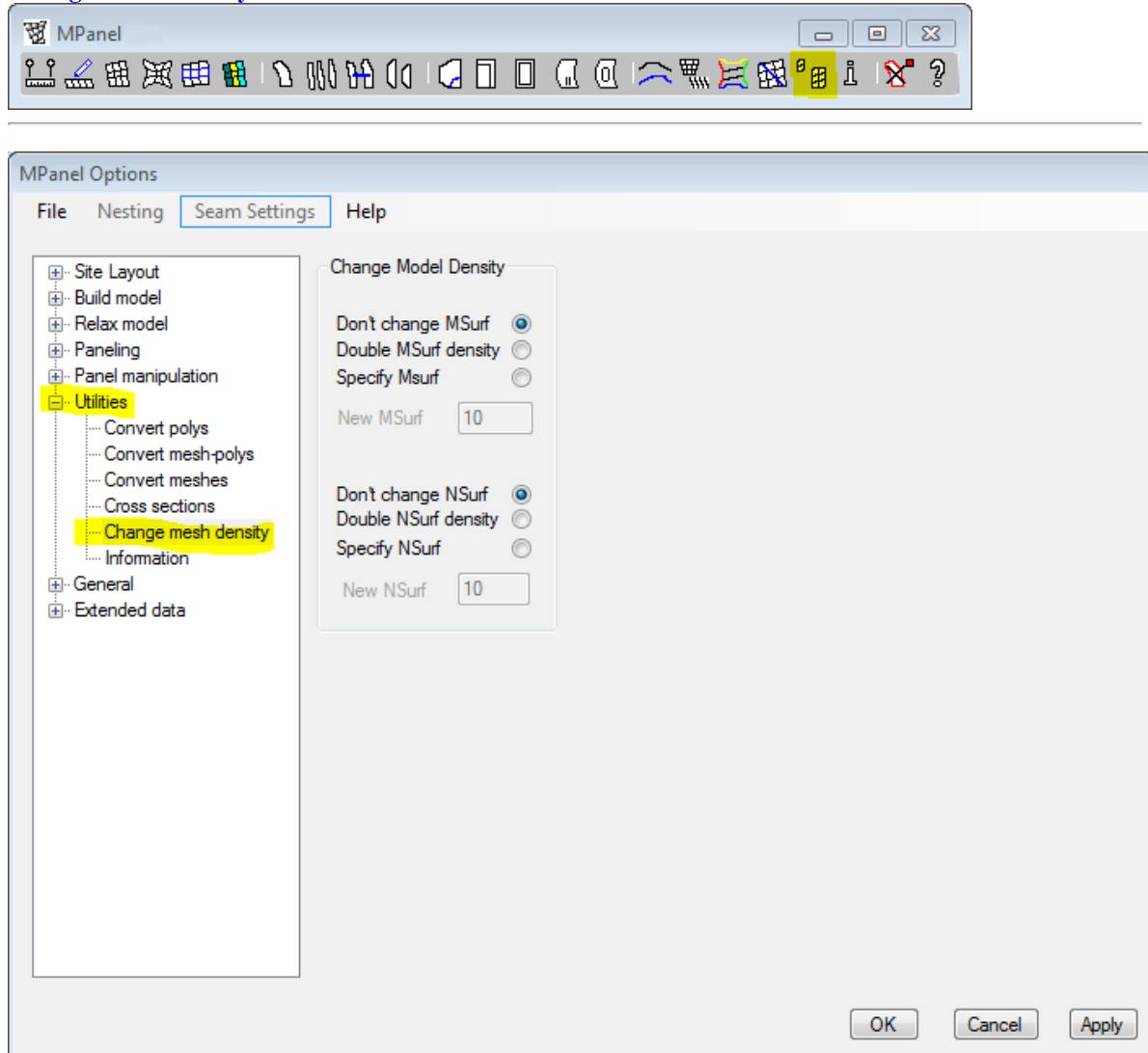
Normally all the mesh threads (N lines and M lines) are used to find intersections with the cut lines, to create the cross section. In some situations it is useful to restrict the intersection to only NLine or Mlines.

For example, when radially cross sectioning a conic, using only the MLine intersections will create a cross section that more closely follows the mesh surface. By also specifying the number of points as zero (which turns off the interpolation routine) this can give cross sections that will make a large panel with a rather lower shear strain than the usual method.

Geo iterations.

Making geodesics is an iterative procedure, the number of iterations is set here, Usually 100 is sufficient, more may be needed if the geodesic is very curved, or has a lot of vertices.

Change Mesh Density



Change Mesh Density is a model building tool that can greatly reduce the time it takes to create a high definition model, and can also help to get the right density meshes for direct panelling.

Change Mesh Density can take any number of meshes and double their mesh msurf and nsurf density. This can be applied repeatedly to achieve high density meshes. Note that the high density meshes are created by mesh interpolation, so they will need to be relaxed again to smooth them out. (Technically by doubling the msurf density, the msurf is changed to $2 * \text{msurf} - 1$, and the same for nsurf.)

Optionally, you can change the mesh nsurf and msurf to any specified values. This allows more control over mesh density changes. In particular it allows models to be rebuilt to suit a certain number of panels.

If there is a 3Dpoly on the mesh edge, the number of nodes on the 3Dpoly will be adjusted to suit the new mesh by interpolation.

If the 3D polys that are attached to the mesh edges can be recognised as circles or arcs, a new 3Dpoly is drawn with the correct number of points based on the arc. This maintains the models boundary shape when the boundaries are part of a circular arc. The arc can be in any orientation. This will help when altering the mesh density on cone tent models with a split top ring.

A mesh that is attached to a polyline will remain attached to all the polyline nodes during this interpolation.

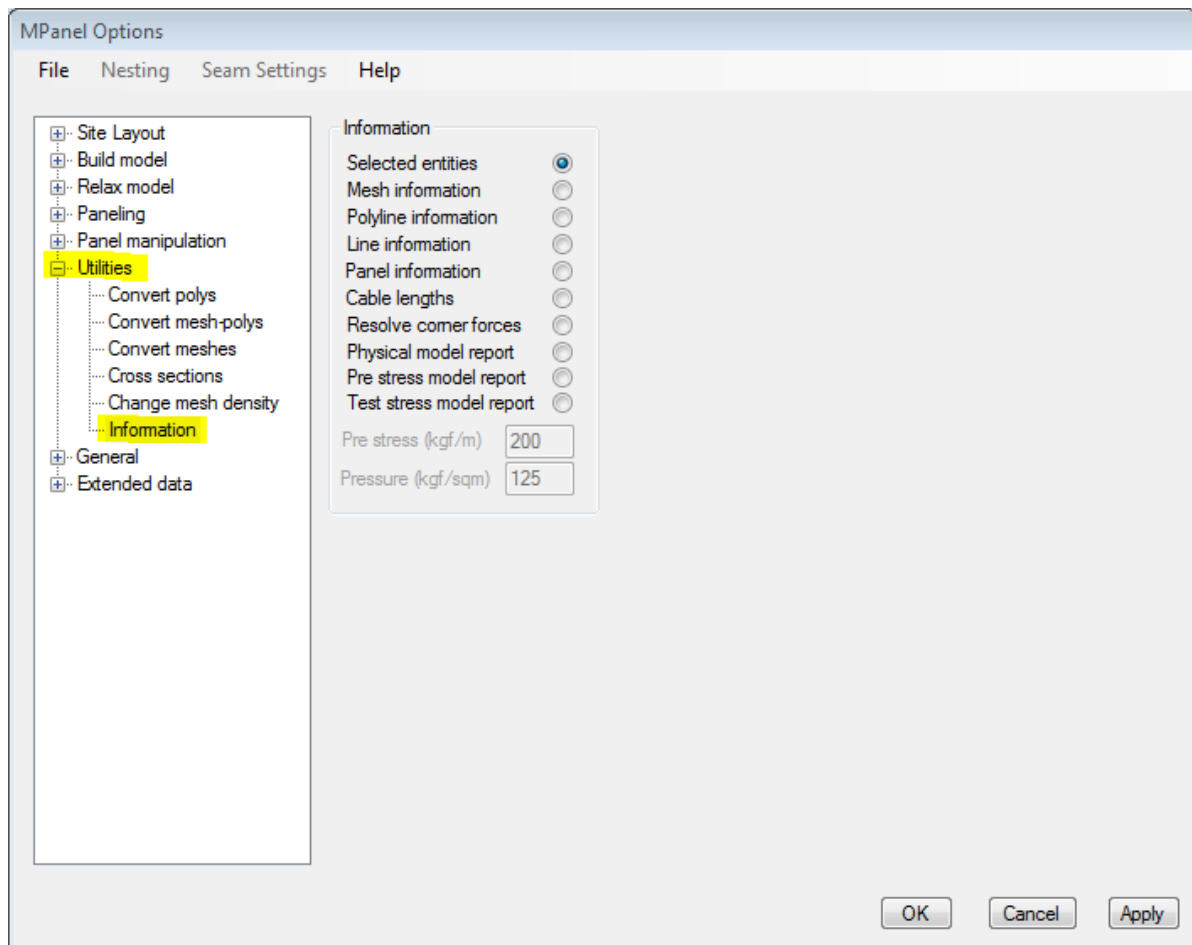
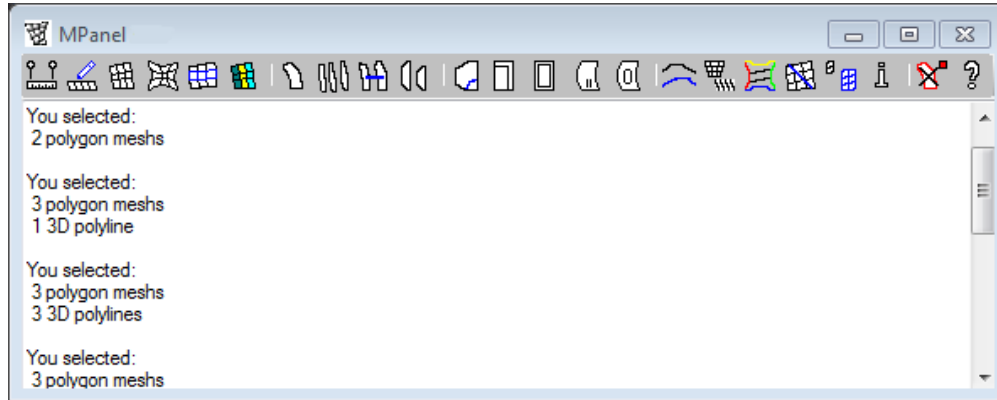
The result of this is that you can draw your models very crudely, using say $2*3$ meshes, with a minimum amount of snapping needed to build the model. You can relax the model, check it, and then apply the Change Mesh Density tool to

the whole model to increase the definition. The new model can now be relaxed again.

Extended data that existed on the meshes is preserved by this tool, so special settings for weft/warp stress ratio, or cable tension ratios, will be preserved.

Note: The method used for generating the new mesh is a linear interpolation of the old mesh, and makes no attempt to smoothly follow a curved surface. For this reason the model must always be relaxed again, to get a good surface to panel from.

Information Button



The Information button supplies information about entities that is not easily obtained from the CAD program. The information available is:

Drawing Entities

A list of the number of each kind of entity found is shown. This is useful if you are checking a model: you may know that your model has 4 meshes and 2 3D polys, but if the info tool also reports some lines, points, text, etc. you could be accidentally selecting some other entities as well.

Mesh

Area, calculated for the 3D shape of the mesh. This has use in estimating membrane material usage. The area is reported in the default units specified in MPanel, No checks are made on the actual drawing units.

Warp Direction, reported as the degrees clockwise from the Y-axis (like a compass). For example, a mesh direction of 90 degrees means that the top of the mesh is on the right of the screen when viewed in XY. This affects the warp/weft stress ratio, the panel seam direction, and the compensation direction.

The mesh Nsurf and Msurf are reported.

The two minimum radius of curvature is reported, along with the direction of the smallest. This is useful when making stress estimations.

3D Poly

Length. This has use in estimating cable lengths.

Mean Radius, calculated by fitting an arc over the end points and the mid point, is of use for estimating cable tension.

Dip/Span ratio is reported. This ratio is often quoted in building codes and is therefore useful information to have when trying to meet those codes.

Vertex count. This has use when checking if cross sections can be panelled or meshed

Direction, reported as the degrees clockwise from the Y-axis (like a compass). For example, a polyline direction of 90 degrees means that the end of the polyline is on the right of the screen when viewed in XY. This has use when checking cross sections and edge polyline directions.

If the 3DPoly is closed then only the length, mean radius, and vertex count are reported.

Line

Line type. Whether it is assigned as a Guy Link or a Cable Link

Line length. If it is a Guy Link, it's original length is reported.

Direction, reported as the degrees clockwise from the Y-axis (like a compass)

Panel

Area calculated on the 2D flat panel as a true area, ie no allowance for waste.

Width gives the width of the panel, based on the panel being placed on the roll with the panel centre line along the warp.

Length gives the length of the panel, based on the panel being placed on the roll with the panel centre line along the warp.

Panel Warp Direction, reported as the degrees clockwise from the Y-axis (like a compass). For example, a warp direction of 90 degrees means that the top of the panel is on the right of the screen when viewed in XY. This has use when checking the orientation of panels for seaming, etc.

Panel Shear Strain reports the amount of shear strain distortion that will be applied to the panel when it shapes to the original 3D shape.

Panel internal ID number.

Cable Lengths

This will take a set of 3dpolys, representing cable edges, and add together their lengths, to give the total cable length. If advanced links have been used on the corners to simulate hardware widths, then these will have a correction for the cable lengths in their extended data, and the corrections are included in the calculation if the links are selected as well.

Resolve corner forces

Calculates the resultant force from two 3dpolys, representing cables, that join at a point. When used you will be prompted to supply the cable tensions, with the left cable first (left defined when looking towards the mesh centre from the corner). This has use in estimating anchor forces.

Physical Model Report

Select the meshes and 3dpolys that define the model, and this tool will draw an outline of the model meshes with the mesh area, projected area, and cable lengths and cable dip/spans noted. A summary report for the whole model is printed in the Mpanel text window. The results are scaled into meters or feet according to the default units specified in MPanel.

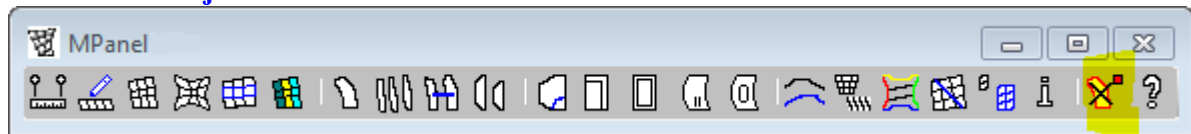
Pre stress Model Report

This takes a specified membrane pre stress and calculates cable tensions and corner forces based on simple static analysis. The results are scaled into meters or feet according to the default units specified in MPanel. [Background information and suggested values here.](#)

Test pressure Model Report

This takes a specified test pressure and estimates the fabric stresses it produces, and then the cable tensions and corner forces based on simple static analysis. The analysis assumes that there will be no shape change during the pressure loading, so the results can only be used as an approximate guide to the estimated tensions and forces. The results are scaled into meters or feet according to the default units specified in MPanel. [Background information and suggested values here.](#)

Delete Old Objects



Color old objects after drawing



Leave old objects after drawing



Delete old objects after drawing

This button deletes the last "old objects" that MPanel read before relaxing, panelling, etc.

(By default, when an MPanel tool runs, the old objects are left in the drawing and colored. This allows the user to check that the new objects have a correct relationship to the old objects)

The button image indicates how the program is set to handle future "old objects". They can be set to be:

- Left in the drawing and coloured
- Left in the drawing
- Deleted

In the first two cases, the old objects are added to an "old objects stack", and they can be deleted with this button on a "last used, first deleted" basis.

Using Links

Links can be of two types, the type is set the first time they are relaxed by an option setting under the 'Mesh Tools' tab.

- Guy Links: maintain an approximately constant length, irrespective of their tension
- Cable links: maintain a constant tension, irrespective of their length.

Guy Links

Guy links normally use the length they are drawn as, before relaxation, as their original length.

Optionally you can specify the link length in the options. This length applies to all links with "specified length", so provides an easy way to change the link length of all the links in a model.

To allow guy links to move, we have to let their length alter slightly, i.e. set a stiffness for them. This means that after relaxation, they will not be exactly the same length as before.

The link stiffness is set under options, and a value of 100 means that the link connected to a mesh is usually within 1% of its intended length after relaxation. Higher values of stiffness give greater control over the link length, but take more iterations to reach their relaxed position.

The default link stiffness value of 10 gets the links into a relaxed position after a few hundred relaxations. If greater control over the link length is required, the stiffness can then be increased, and the model relaxed again.

The link original length is stored in the extended data under the key name "Length", and can be edited.

Although links can be in compression (masts) or in tension (guys), compression links have poor modelling stability and often result in the whole model collapsing to the ground.

For an isolated link where one end of the link is a fixed point (i.e. a corner guy), you can optionally force the link length to stay exactly the same length. This causes a small modelling error, usually insignificant, but is useful when a number of identical links are required. Do this by ticking the 'Fix Guy Length' box under the 'Mesh Tools' tab.

Advanced Guy Links

Guy links can also simulate an end width, to allow for the truncation effect of fitting standard hardware such as D rings in the membrane corners. This is fully described [here](#)

Cable Links

A cable link has a tension assigned in the options. A cable link with a tension of 1 is the same as a warp mesh thread on a normal relaxation. So a series of cable links can be used to simulate a cable in an arbitrary position in a mesh.

The link tension is controlled in the options, and can be changed between relaxations. Or the tension can be adjusted on individual links in the extended data, allowing different parts of the model to have different cable tensions.

Note that the cable link tension specification is different from the mesh edge cable tension. The mesh edge cable tension definition is derived so that a cable tension ratio of 1 gives a catenary arc with a radius roughly equal to the catenary span, which typically means that an mesh edge cable with a tension ratio of 1 has much more tension than a cable link with a tension set to 1.

Link Models.

It is possible to use cable links and guy links to model free form cable nets, without any meshes at all, that can be relaxed and a force balance position found. Some care is needed in setting tension values realistically to obtain the desired result, without the model collapsing. Fixed points are specified with a 3Dpoly as usual.

As a simple example, a 2 link model with a guy link of stiffness 100, connected to a cable link of tension 7, will pull both into a straight line with the guy link stretched by 7%.

The link type can be changed in the extended data. Care should be taken to provide a reasonable setting for the link original length, stored as "value", for guy links. If set to default then the links present length will be used.

Link models can also incorporate meshes, leading to modelling of meshes supported by cable nets.

Links that simulate end width.

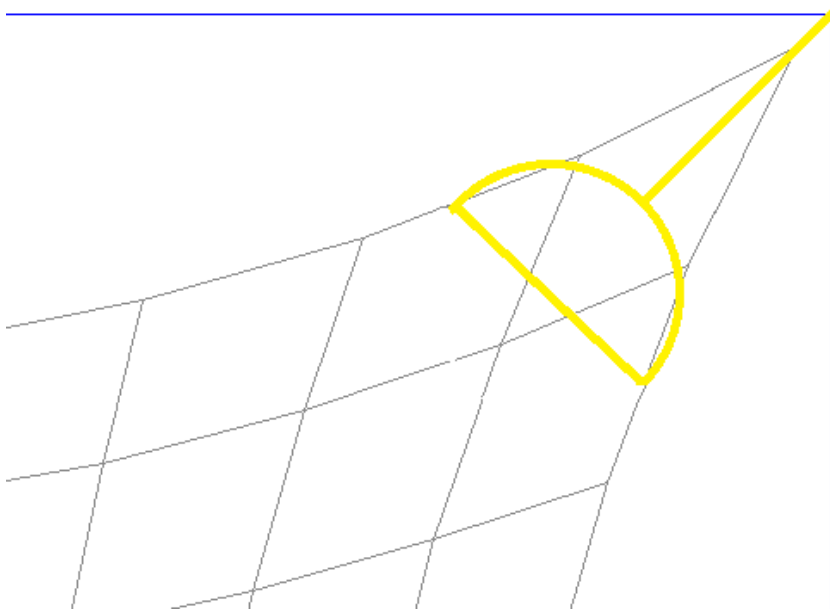
A specific problem arises when terminating membrane sail corners with standard hardware, such as D rings. The membrane corner has to be truncated to fit the hardware, and this shortens the corners by an amount depending on the corner angle. To allow for this an arbitrary length guy line is needed to make up the corner length.

This lead to the concept of an link that measures it's length between the fixed point and a point in the sail where the corner has a specified width. This allows the membrane to be designed, panelled, and then have the corners truncated to fit the hardware width, which will leave a membrane that will have the specified distance from the fixed points.

This is achieved by specifying the link as "Simulate flat end".

Simulate flat end

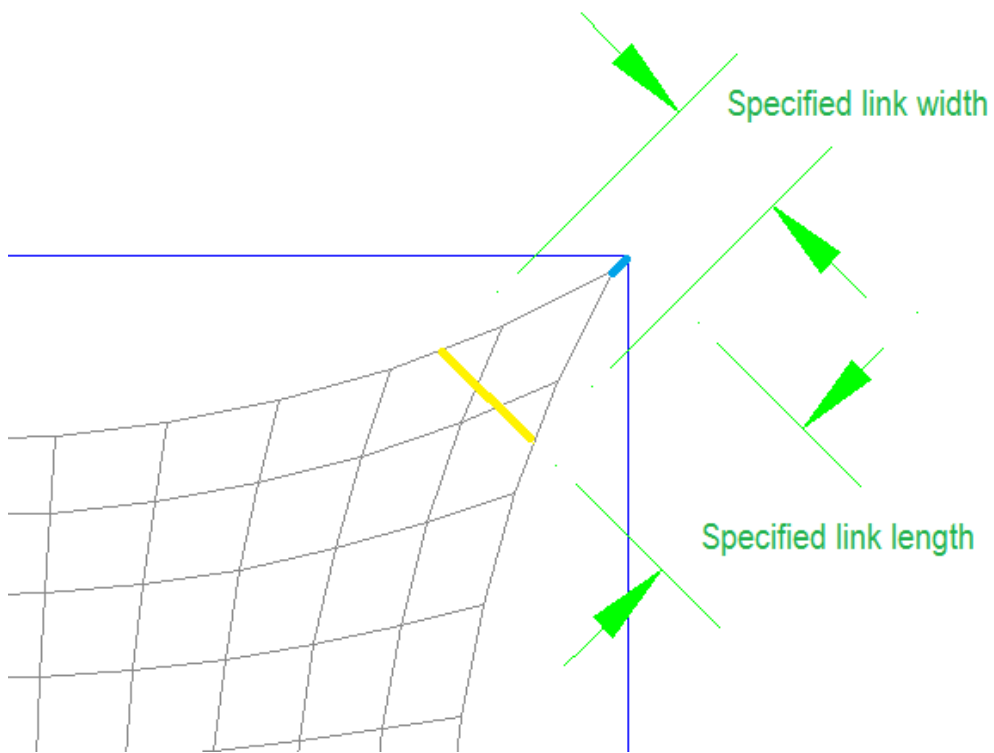
As an example, we have a hypar sail which will be terminated with a D ring and a snap link, shown in Yellow.



To model this we specify a flat end link in the corner that defines a link width and link length. The link length would be the internal length of the D ring and snap link coupled together. Then we relax the model to obtain the picture shown below. The relaxed link is shown in Cyan, and the points that the length and width are measured to are also shown.

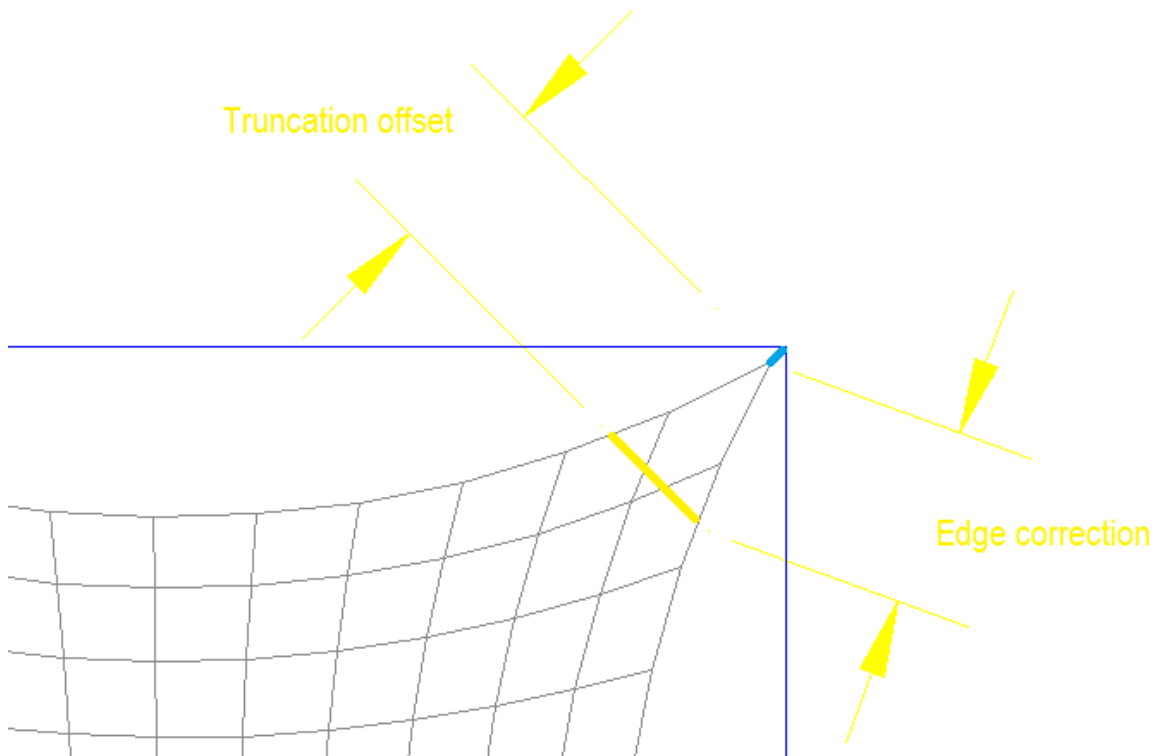
So the actual link length is reduced to allow for the sail truncation needed to accomadet the D ring width. In some cases the actual link length will become zero or negative, the program copes with this OK.

The Yellow line, representing the D ring width, is shown just for illustration, it does not appear in the CAD drawing.



So with the above drawing, if the sail was built and hemmed, and then truncated to fit the D ring, the snap link would be the right length to fit to the corner.

Additional information about the corner is stored on the links extended data, illustrated here.

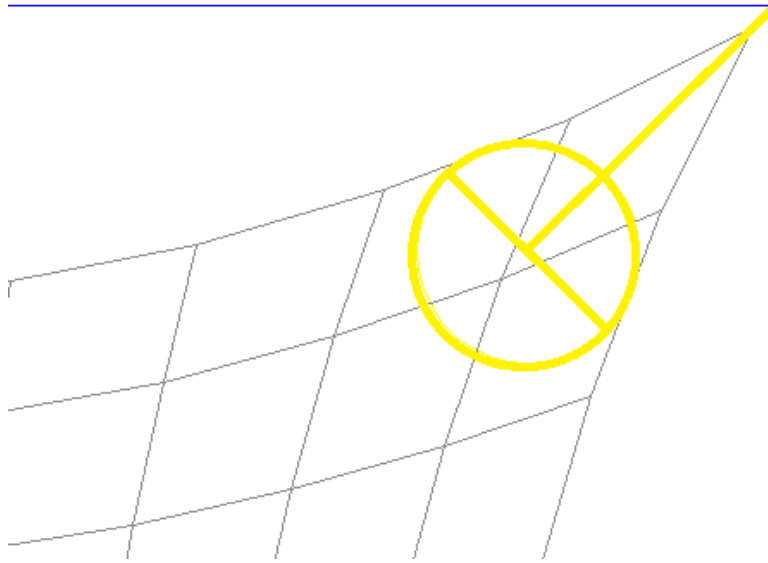


Edge Correction Amount the edge is shortened on each side of the corner.
This is used in correcting the cable length.

Link Offset Amount the membrane is truncated in the corner for the hardware to fit.
fit.
correct (this is measured in the 3d model, and will only be approximately correct on the panel due to compensation and shear distortion)

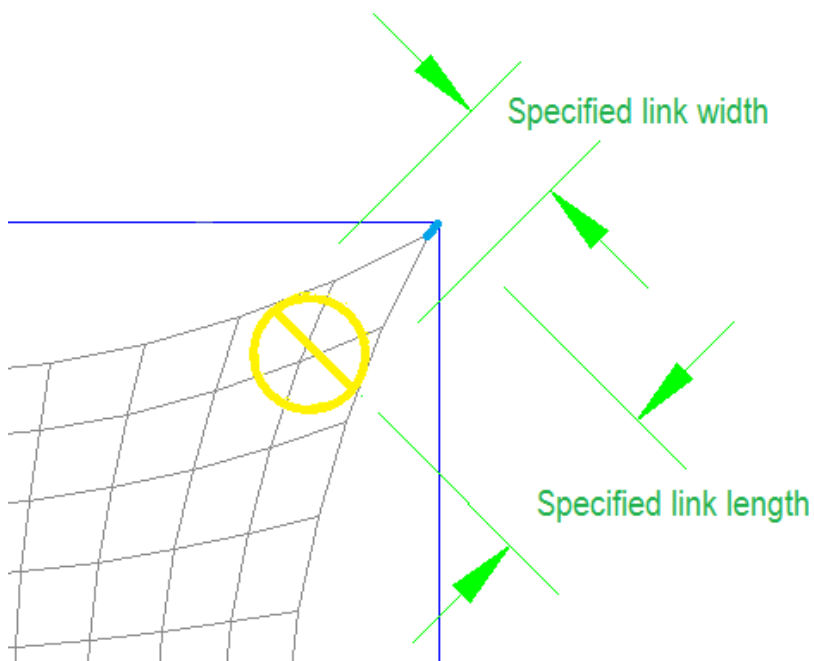
Simulate round end

A similar situation is where the corner hardware is built so that the cable passes around a circular groove in the hardware, like an Ez glide. In this case the circle has to fit in the membrane corner tangentially, as shown here.



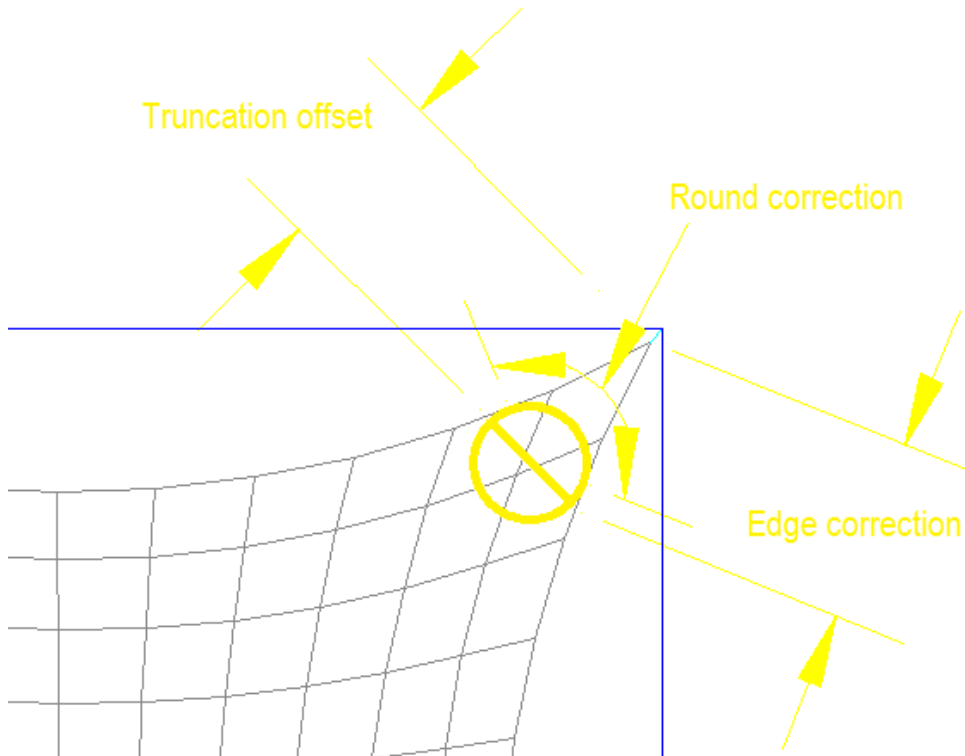
To model this we specify a round end link in the corner that defines a link width and link length. The link length would be the internal length of the Ez glide and snap link coupled together. The link width would be the diameter that we need to fit tangentially into the corner. Then we relax the model to obtain the picture shown below. The relaxed link is shown in Cyan, and the points that the length and width are measured to are also shown.

The Yellow circle and line, representing the Ez glide fitting, is shown just for illustration, it does not appear in the CAD drawing.



So with the above drawing, if the sail was built and hemmed, and then truncated to fit the EZ glide into the corner tangentially, the snap link would be the right length to fit to the corner.

Additional information about the corner is stored on the links extended data, illustrated here.



Edge Correction Amount the edge is shortened on each side of the corner. This is used in correcting the cable length.

Link Offset Amount the membrane is truncated in the corner for the hardware to fit. (this is measured in the 3d model, and will only be approximately

correct

on the panel due to compensation and shear distortion)

Round Correction Amount of cable used to go around the corner hardware.
This is used in correcting the cable length.
(only applies to round end links)

Modeling and Paneling Errors

There are several sources of error that are inherent in the modeling and panel flattening process. During the design the errors must be kept within reasonable bounds, which might reasonably be less than the intended fabric pre stress.

Modeling Error

The principal modeling error is the representation of a curved surface by a mesh consisting of flat faces joined at angles. This can lead to edge alignment problems. So if the edge is a semicircle and the mesh density say $12 * 12$, the semicircle would be approximated by a 12 segment polyline. This will give a 0.5% edge length discrepancy compared to the true arc.

The solution is to raise the model density. This is easy with the Change Mesh Density tool, but expect to need more iterations to achieve a fully relaxed model. Choose the lowest model density that will keep the errors at a reasonable level, and then the model should still relax fairly quickly.

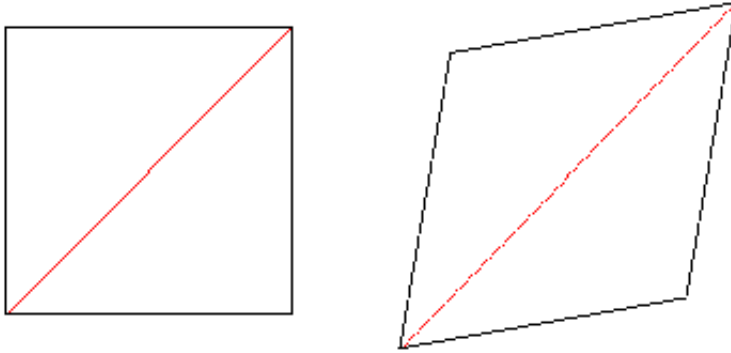
Paneling error

In general there is inevitably an error when flattening a 3D shaped fabric to a 2D panel. There are some exceptions, such as conic surfaces and surfaces of revolution that produce zero error, but for most anti clastic surfaces as used in tensile structures there will be flattening errors. MPanel uses a method to reduce the effects of these errors by sharing the errors equally in the bias directions of the panel. This error is then reported as the equivalent shear strain, see definition below. If the errors need reducing further, build with more seams.

Merging error.

When two panels are merged, if the joined seams are different curves, something has to give to let them join. The give is on the bias, and it is reported as the equivalent shear strain, see definition below.

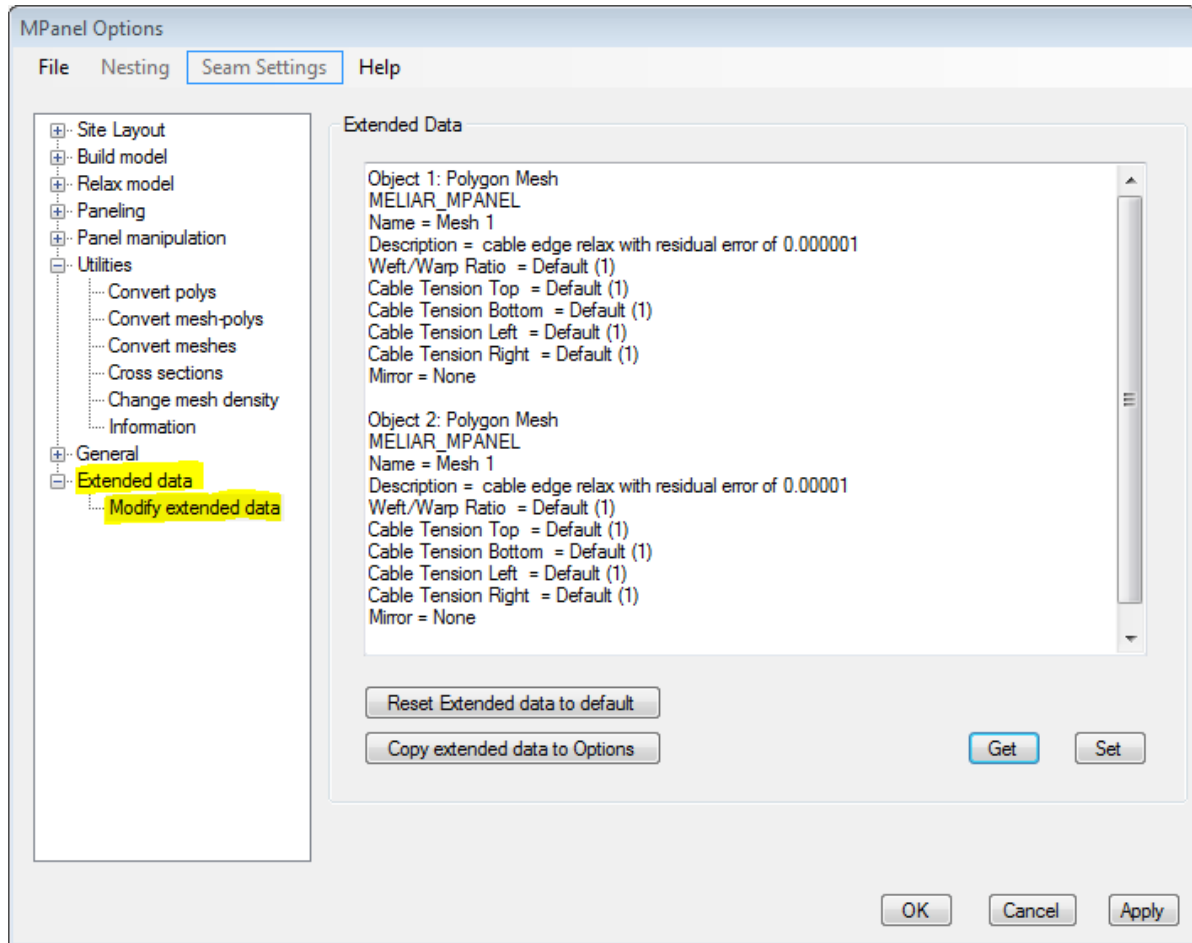
Shear Strain



Consider a small square of fabric in the middle of a panel, illustrated on the left. If it is distorted into the rhomboid on the right the amount of stretch or shrink in the diagonals, shown in red, gives a measure of the shear strain. This is the figure that MPanel reports as a percentage of the original diagonal length. So the shear strain shows how much your fabric will have to move on the bias to make the panels fit. Typically the shear strain is cumulative, so if you panel and then merge the shear strain is the sum of the strain at each stage.

If you prefer to see shear strain as an angular measurement, note that 1% diagonal change is roughly 1 degree of angular strain.

Extended Data



Extended data is used to record information about the mesh, link or panel drawn, and optionally to change that information to affect further relaxations.

The extended data editing is carried out on the "extended" tab in the options. Select the meshes, lines or panels and click the "Get" button. The extended data for all the selected objects will show in the text window.

To change any editable data, change the text and then click the "Set" button.

To set all extended data back to default setting, a "Set Default" button is supplied.

To avoid confusion when editing several objects at the same time a preferred method is to give significant names to each of the objects. For example select just the North mesh in a set of meshes, get its' extended data, change the object name from the default name to "North Mesh", and set the extended data. By giving each object a significant name the editing of a set of objects becomes clearer.

If several objects are selected with the same extended data (except their names) then the data will be presented as one item called "Multiple Objects", and the data can be edited and set on all the objects from this single entry.

Mesh

Editable fields and example values

Name = North mesh

Weft/Warp Ratio = Default (0.5)

Cable Tension Top = Default (1.3)

Cable Tension Bottom = Default (1.3)

Cable Tension Left = Default (2)

Cable Tension Right = Default (2)

Special = None

If the value contains "Default" , eg Weft/Warp ratio = Default, then the current option setting will be used on the next relaxation.

If the value contains a value in brackets, eg Weft/Warp ratio = Default (2), then that was the option setting value used on the last relaxation.

By altering these figures MPanel can use different settings for individual meshes.

Special values can set the model to mirror the model. Or they can set the mesh basic warp tension, which is normally=1.

Acceptable values are:

Special = MT or MB or ML or MR or W0.3

In addition if a special setting was used during relaxation then an "information" value is added to the special values. This is for information purposes only, and is not used as a setting. Information settings start with the letter I, and can be read as:

IV	constant stress used on a legacy relaxation
IS	soft spacing turned off
IC	even tension set on the cables
ID	diagonal warp used
IF	fix modelling tension used
IL	warp straightening used
IPx	a pressure of value x used
IFx	concentrate warp threads stress variation used, when selected x is presently always = 1

Links

Editable fields and example values:

Name = NE Guy

Maintain = Tension / Length

Value = 4 ... the length for a guy or the tension for a cable link

Panel

There are no editable fields for a panel.

The panel edges are marked as warp edges or weft edges

A description of the tool that created the panel, and the option settings used, and the shear strain, are recorded on the warp edges.

The panel internal ID number is recorded as Panel ID = X Further panel operations such as compensation, seaming, etc will give panel ID numbers like X.Y

Reset Extended data to Default

This will set the extended data for all the selected objects to Default. So the value used on the next relaxation will be the value in the Option. A particular use of this is to enable link lengths to be changed on the screen... normally the old link length would be recorded in the extended data, and used on the next relaxation.

Copy Extended data to Options

This will read the extended data window, and try to extract unambiguous values for the last used mesh weft/warp stress ratios, and the for cable tension ratios. If these are found then the option vales will be changed to match. This has use when working on an old model, to set the options to the same values as used previously.

Stress Estimation

Although MPanel is not a tensile structure stressing package, it can be used to roughly estimate fabric stress, edge loads, cable loads and anchor loads. This process is simplified by using the Model reports in the Info tool. For more comprehensive and accurate results, use the MPanel FEA add on module.

Here we describe:

- Estimating cable tension's and corner forces due to membrane pre stress.
- Estimating stresses under load using the whole model report.
- Estimating the static stresses using hand calculations.

Estimating tensions and forces due to pre stress.

This uses a given membrane pre stress, which is assumed to be the same all over the membrane. The cable tensions are then calculated from the mean arc radius of the catenary edges, and the cable tensions are vector summed to calculate the cable forces.

Suitable pre stress values:

(actual pre stress depends on the material, the manufacturers recommendations, the structural application, and the membrane size)

Shade cloth (Knitted HDPE)	=	10 kgf/m	=	6.5 lbf/ft
Type 1 PVC coated polyester	=	200 kgf/m	=	135 lbf/ft
Type 2 PVC coated polyester	=	250 kgf/m	=	168 lbf/ft

Estimating stresses under load using the model reports.

This uses a test pressure applied to a very simplified model, based on this premise that the fabric stress under a pressure load is similar to the hoop stress in an inflated cylinder, where the cylinder radius is equal to the mesh minimum radius. From the fabric stress the cable tension can also be estimated.

A description of the reasoning behind this method is appended below. Note that this method will tend to over estimate stresses (since we ignore membrane deflections, second principle carrying stress, cable orientation, etc) but will produce membrane stress and cable tension figures that are indicative of real values.

Suitable pressure loads:

Suction uplift from 50 mph (80 km/hr) wind 6 lbf/sqft	=	30 Kgf/sqm	=
Suction uplift from 100 mph (160 km/hr) wind 24 lbf/sqft	=	120 Kgf/sqm	=
Down pressure from 3 feet (1 meter) of snow 20 lbf/sqft	=	100 Kgf/sqm	=

Description of method:

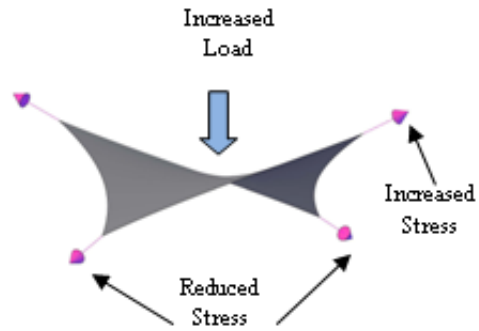
In a true anti-clastic tensile structure with pre-stress, there are two principle directions where the curvature is at a minimum.

Under external loading, the stress in one principle direction will increase, and the other will decrease. As the external loading increases, a point will be reached where the stress becomes zero in one principle direction.

(this assumes no deflection of the membrane)

At this point, the increased stress in the other principle direction is solely supporting the external load.

So this stress is similar to the hoop stress in an inflated cylinder, with the same radius.



Estimating the static stresses using hand calculations.

Fabric stress

At the geometric centre of the mesh, determine a reasonable stress for the fabric in warp. This could be 50 KG/M for example.

The weft stress at this point will be warp stress multiplied by the weft/warp stress ratio, as set in the Options. With a weft/warp ratio of .33 this could be 16 KG/M in our example.

If a normal relaxation has been performed, the stress at other parts of the mesh is related to the centre stress by the mesh density in warp and weft. So if a part of the mesh has its warp lines twice as close together as at the centre, then the warp stress there is twice as high, ie 100 KG/M in our example.

If the constant stress relaxation was chosen in Options, the stress ought to stay constant over the mesh irrespective of mesh density.

Edge Loads

The edge load, for example in supporting steelworks, is the reaction to the edge mesh stress. If the local mesh lines are substantially at right angle's to the edge, then the edge uniform density loading is numerically the same as the mesh stress.

Cable loads

The cable tension is the total edge load along the cable length multiplied by the cable tension ratio for that edge. So a 5M long edge at 30 KG/M edge load with a cable tension ratio of 1 has a cable tension of 150 KG, or for a cable tension ratio of 1.5 the tension would be 225 kg.

Anchor loads

The anchor loads are determined by vectorially adding the loads for the cables from that point. The MPanel Info tool will perform this addition automatically, by selecting two polylines that join at a corner and specifying their tensions.

Reported base modelling tensions

After each relaxation the average width of fabric that each thread in the mesh is representing is reported in the text window, as the "model warp spacing" and "model weft spacing". These can be helpful in estimating stresses, and in determining values to be used with later FEA analysis.

For example if the fabric will be tensioned to 100 Kgf/m and the model warp spacing is 0.4 meters, the tension carried by each warp thread is 40 kgf.

The value calculated for the weft threads will have to be multiplied by the weft/warp stress ratio.

If the relaxation used the "constant stress" option, then this would be the average thread tension, adjusted according to the spacing of the threads.

When a cable link is used, the tension on the link is specified as a multiplier times the warp thread tension. So in this example if you had a cable link, with a tension setting of 5, the real cable link tension would be $5 * 40 = 200$ kgf.

If "even tension" is selected on the cable options, the theoretical base cable tension will be reported. This doesn't allow for the cable tension ratios so the value has to be multiplied by those ratios.

Demo of creating a Cone Tent

We are going to develop a simple cone tent, using the old method of building the model by hand. (You can use the Model Builder tool to do this for you, but it is useful to understand the base method). The system fixed points are drawn in as a 3D Polyline, shown here in blue. The top ring is also drawn as a 3D Polyline, with enough points to attach the meshes to.

The meshes are drawn in flat.

(In AutoCAD draw them using the Mesh command, the number of mesh divisions is set with nsurf and msurf)

(In Rhino draw them with the MeshPlane command, the number of mesh divisions is set with XFaces and YFaces)

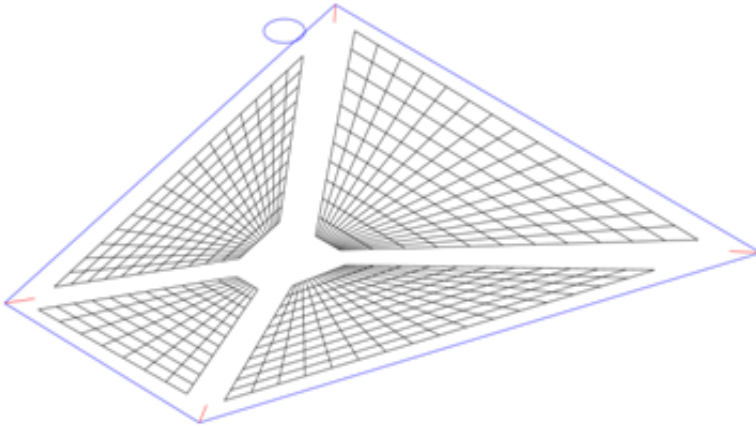
The meshes each have nsurf points in the weft direction and msurf points in the warp direction. In this model we will have the warp running up to the ring. Since we want the meshes to join together they must all have the same msurftab numbers. Often we use the same mesh in different places, by copying and rotating.

The number of points needed on the top ring is: $\text{NumPoints} = \text{Sum}(\text{nsurf} - 1)$

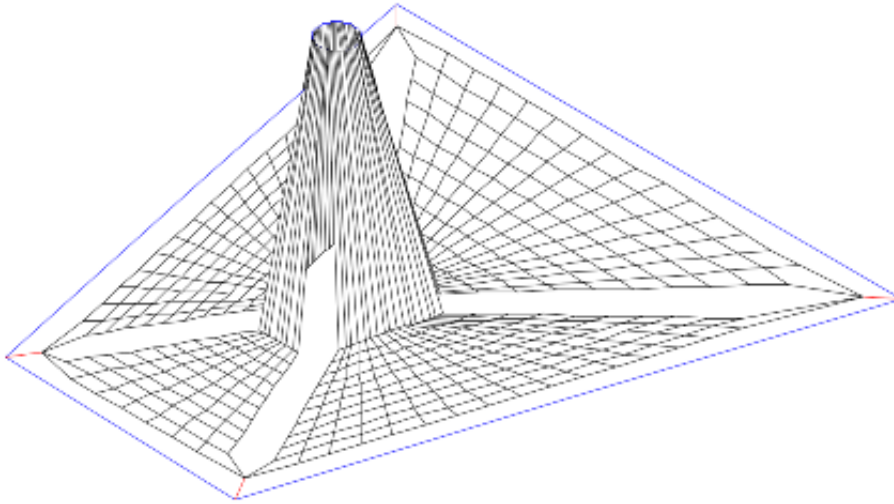
In this case $\text{NumPoints} = (8-1)+(17-1)+(17-1)+(16-1) = 54$

(In AutoCad, the Polygon tool, useful for drawing top rings, produces a 2D Polyline that MPanel will not use. You can convert it to a 3D Polyline using the MPanel tool "Convert Lines" set to the 3D Polyline option.)

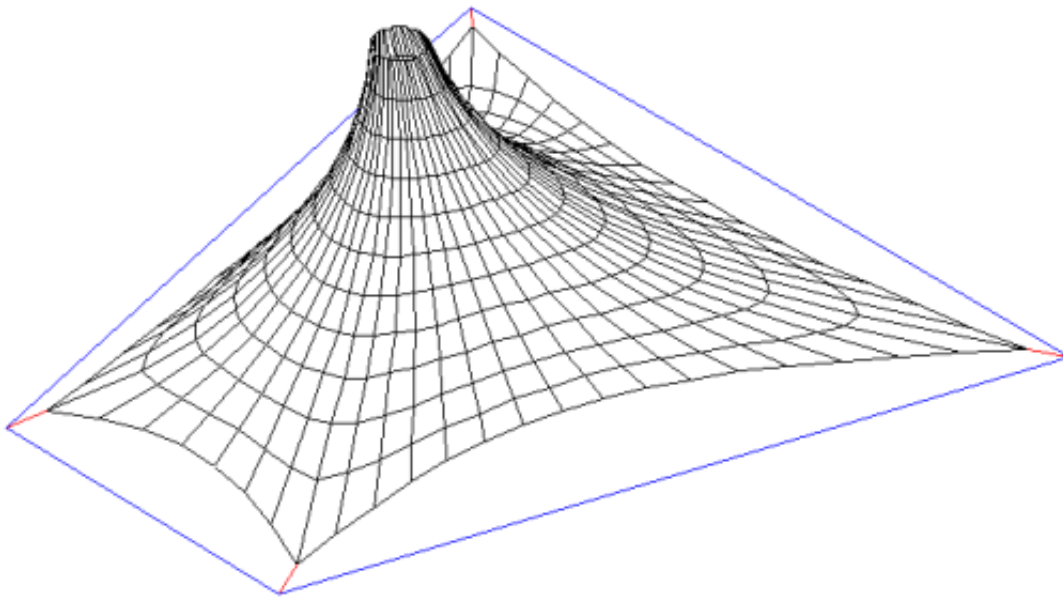
The guy lines or links, shown here in red, are drawn in flat using the Line command, to the correct length.



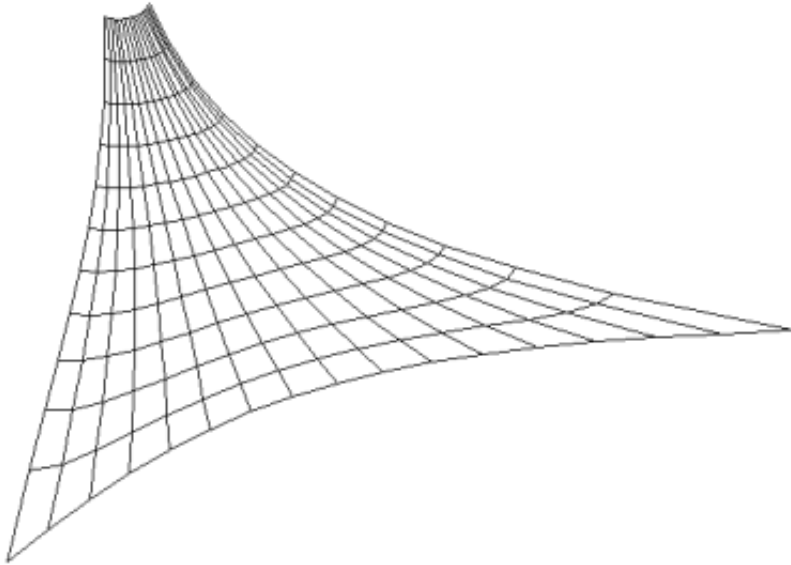
We now stitch the model together. This is done using OSnap. Meshes will typically join the adjacent mesh at their corners, sharing a system fixed point or link ends.



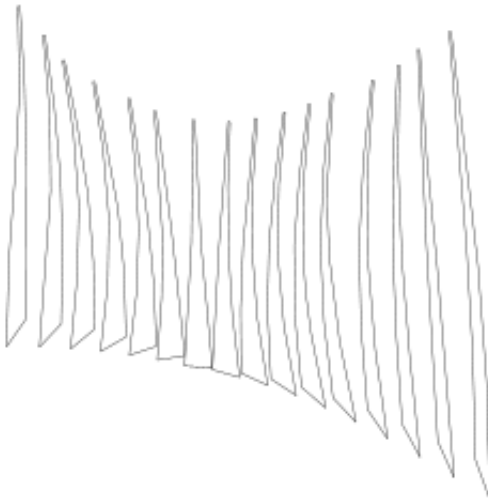
The model is now relaxed using the MPanel Joined Edge Mesh tool. Be sure to use enough iterations to ensure a fully relaxed model. The shape of the model can be controlled by setting the weft/warp stress ratio and the cable edge tensions.



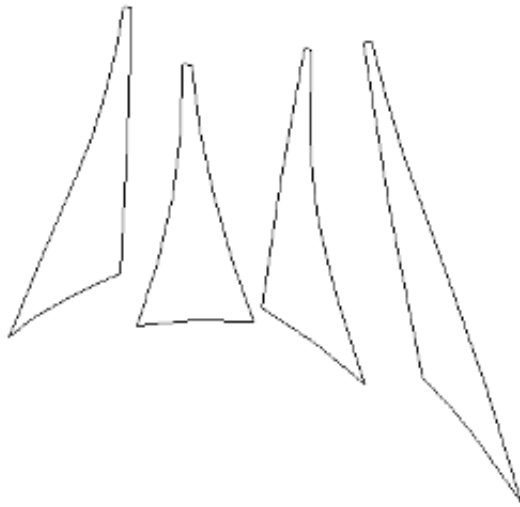
Although the meshes have all relaxed together, they remain as separate meshes. Here we select one mesh for further processing.



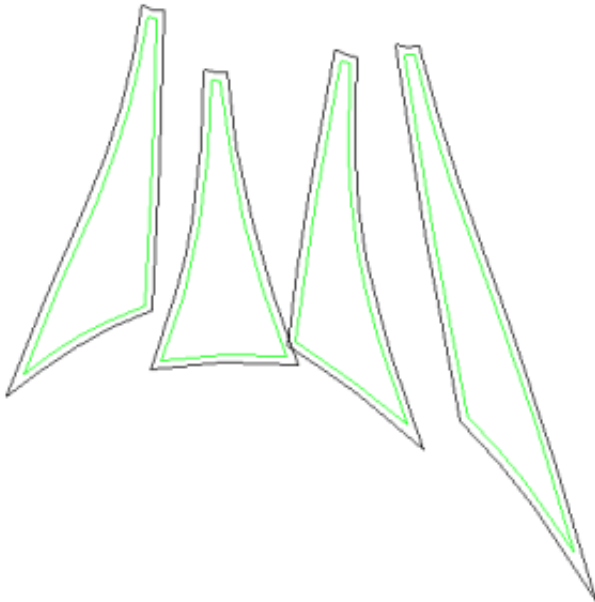
Using the MPanel tool "Mesh Panel" we can create all the panels needed to create the 3D mesh shape in the flat.



Adjacent panels can be merged together, creating larger panels with curvature on all 4 sides, by using the MPanel tool "Merge Panels"



The panels can have seam allowances, or pre stress compensation added to them, using the MPanel tool "Compensation" to produce a production ready template.



Internal Model

MPanel reads the drawing model, the extended data, and the current options, and creates an internal model that consists just of links and nodes. This model is then relaxed, and the results written back to the original model entities. The model can optionally be drawn out during the relaxation. It has no further use to MPanel once it is drawn out, but inspecting it can shed light onto the MPanel process.

Each link and node in the internal model has extended data, which can be displayed in the extended data tab. The data format is as follows:

Nodes

Node Number	Node identifying number
NodeFree / NodeFixed / NodeDead	NodeStatus
SoftSpace / HardSpace	How to space the nodes in the mesh
Guyed / GuyTrim	Status of guys on the node
Linklist	A list of links connected to the node. Negative means connected to the link end

Links

Link number	Link Identifying number
Rubber / Stiff / Pressure / Dead	Link type, normal link = rubber, a guy = stiff
LocY / LocX / LocXY / LocXX / LocYY	Link orientation, used for spacing in mesh.
Edge	Marks a cable edge
StNode	Node at link start
EndNode	Node at link end
Xten	Xtension multiplier
Yten	Ytension multiplier

During the relaxation the tension on each link is calculated as :

Link tension = Xten multiplier * base weft tension + Yten multiplier * base warp tension.

Base Modelling tensions

During the relaxation the base modelling tensions are recalculated to approximately get a stress of 1 on each edge in the fabric. If there are no meshes then the base weft and warp tensions are set to 1.

Comparison of CSA and legacy relaxation methods.

The CSA relaxation method was introduced in version 17 to help achieve a more even stress across the mesh surface.

Modelling a similar shape with CSA and legacy relaxation will need different parameter values for weft/warp stress ratio and cable tension ratios to achieve the same shape. This page describes those changes.

- **Even stress**

The CSA default relaxation method is for an even stress surface.

Optionally a surface where the stress varies with the warp thread spacing can be specified.

The legacy default relaxation method is for a surface with the stress varied with the thread spacing

Optionally an even stress surface could be specified, but required multiple relaxations.

- **Weft/warp stress ratio**

The CSA by default will make a more evenly stressed surface. This may require a different weft/warp stress ratio to achieve the desired shape.

In particular conics will require a weft/warp stress ratio that is more extreme than before.

If the conic shape produced is not visually appealing, the option for "variable warp stress" will make a surface which is similar to the legacy default shape.

- **Cable tension ratios**

The CSA cable tension calculation is based on the distance between cable attachment points, even if they span several meshes or just part of a mesh.

The legacy cable tension calculation is based on each mesh edge distance.

This means that the cable tension is now based on the model edge fixed points.

Previously a mesh edge that was supported at mid points exhibited a cable tension that was too high, and had to be corrected with a lower cable tension ratio. This is no longer necessary, a cable tension ratio that defines a dip/span ratio will give the same dip/span ratio when the edge has mid supports.

Previously a cable that spanned several meshes exhibited a cable tension that was too low, and had to be corrected with a higher cable tension ratio. This is no longer necessary, a cable tension that defines a dip/span ratio will give the same dip/span ratio when the cable spans several meshes.

CSA Internal Model Tension Calculations

Note: this description applies to the CSA method of relaxation, introduced in MPanel ver 17. Legacy methods of relaxation (used in Mpanel ver 16 and earlier) used a simpler method which is described elsewhere.

Thread tension

MPanel assigns a tension to each short thread in the mesh so that the thread can represent the membrane around it, and then during the relaxation each node is moved towards a force balance position with its attached threads.

The thread tension is calculated from the adjacent cell widths, so physically it corresponds to the resultant of the fabric stress, at a nominal fabric stress of 1, over the cell width in the thread directions.

The threads that represent the weft direction have their tension modified by the weft/warp stress ratio.

The thread tensions are updated about every 10 iterations during the relaxation, which is sufficient to keep the thread tensions following the changes in the mesh shape.

Variable warp stress option

If the option "Variable warp stress" is selected then the threads that represent the warp threads have their tension replaced by an "fixed warp thread tension". This has the effect of increasing the membrane stress in areas where the warp threads are close together, and reducing it where they are wider apart.

The "fixed warp thread tension" is calculated from threads that start or end on a cable, to ensure that the shape of those cables does not change significantly when selecting this option.

On conics this has the effect of increasing the overall warp tension, which would decrease the effective weft/warp tension ratio. To avoid this the weft thread tensions are all increased by the factor "fixed warp thread tension" / "average warp thread tension". This correction is not applied to models that have cables along the mesh warp edges (ie hypars, triangular sails) otherwise it would change the shape of the warp cables.

Cables

Threads which represent cables have their tension assigned once at the beginning of each relaxation run, and it is not changed during the relaxation. The tension is related to the distance between the fixed points at the cable ends. This measurement may span several meshes.

If the cables are connected to fixed points by guys the tension assigned is still calculated on the fixed point length, so the guys are ignored and the cable tension is based on a slightly longer length than the distance between the cable ends. (There can be a maximum of 2 guys in series connecting the cable ends to fixed points)

This method of fixing the cable tension at the beginning of the relaxation based on system fixed points means that the major forces in the model are not changing during the relaxation, which helps to keep the solution stable.

Other links

If a pressure is specified it is modelled as an additional thread at each free node, normal to the local surface, with a tension calculated from the area of the cells surrounding the node.

Stiff links (guys) have a modelling stiffness set such that a stiffness of 100 means a guy will extend by 1% under typical model loading.

Cable links have a tension assigned to them which is the average of the warp thread tension, modified by the cable link tension setting in the options.

Detail dxf specification.

The detail dxf consists of these entities:

A V formed by two lines of the same length. The start of the lines is at the corner of the V. The most clockwise of the lines is coloured blue, the other is green. The V is usually drawn in the "12 o'clock to 3 o'clock" position. This V is the reference that is scaled angularly and linearly to fit the V in the 3D drawing. These reference lines are not transferred to the 3D drawing

Any number of polylines that define the detail that you want to transfer to the 3d drawing.
(In Autocad these are polylines, not 3d polylines)
(In Rhino each polyline must have at least 3 points)

Any number of points. These will be transferred to the 3d drawing, and can be used as panel alignment marks.

The dxf should be saved as a dxf file, R14 or later. The file should be placed in this folder:
MyDocuments/Mpanel/Detail

The exact location of this folder will vary depending on the operating system. If the dxf file is created in Rhino, use the 2004-Natural style for dxf export.

Notes on non scaling polys:

If a polyline is to be "Not Scaled" then it should be coloured Magenta. Non scaling polylines are handled in this manner: The first point of the polyline is regarded as the reference point, and it's position is scaled angularly and linearly. The rest of the polyline is moved with the first point. So polyline representing a circle should have the first point at the circle centre.

If the entity orientation is important, say for a rectangular polyline, the second point on the polyline is used to define the part rotation during angular and linear scaling.

Notes on point sets.

If a set of points are to be joined together during panelling, they should be coloured the same simple colour, like Red, Yellow, Blue, Green.

If some points should not be joined together, they should be left un-coloured.