Table of Contents

What's New

Version 11 summary of new features:

- Add Auto mesh a set of curves and polys ....more
- Reset panel ID number button added.
- Add assembly marks now uses the panel ID numbers.
- Match mesh density includes the option to use highest or lowest mesh density.
- Invert panels selectable on both Nest and Output pages.
- Button names can be shown for training.
- Null layers now ignored - previously caused errors.

Version 10 summary of new features:

- Adjust mesh...New "Add adjacent marks" adds points and match marks where multiple meshes join a single mesh.
- Adjust mesh...New "Add extra marks" adds points and therefore match marks where multiple edges join.
- Arrange mesh....Tool now ignores any items other than meshes and points.
- Arrange mesh....Adds any selected items other than meshes and points to the "Delete old objects" list.
- Output....New option to invert panels.
- Utilities…. Draw mesh edges added to mesh utilities that can be used to fix a mesh edge during cable edge relaxation.
- Utilities…. marks and detail correctly preserved during panel split and panel split by hole.
- General....Units are now shown on all relevant text entry boxes and the seam table

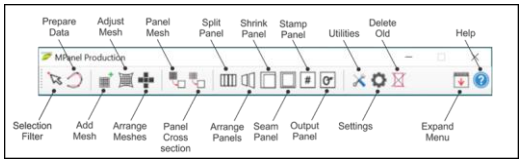
Version 9 summary of new features:

- Add mesh...New "edge filter" simplifies adding meshes by restricting selection to valid edges.
- Paneling....Make panel from 3 or 4 entities added to paneling.
- Paneling....Make panel from single entity added to paneling.
- Paneling.... Join Colored points joined… all the points join, irrespective of where they are in the panel.
- Paneling....Use 3Dpoly as cut line added to advanced paneling.
- Seams… Copy old panel edge is suppressed when seam allowance is zero or negative.
- Utilities…. Mirror mesh added to mesh utilities.
- Utilities…. Join meshes with differing edge point count added to mesh utilities.
- Utilities…. Divide curve into points tool added.

Version 8 summary of new features:

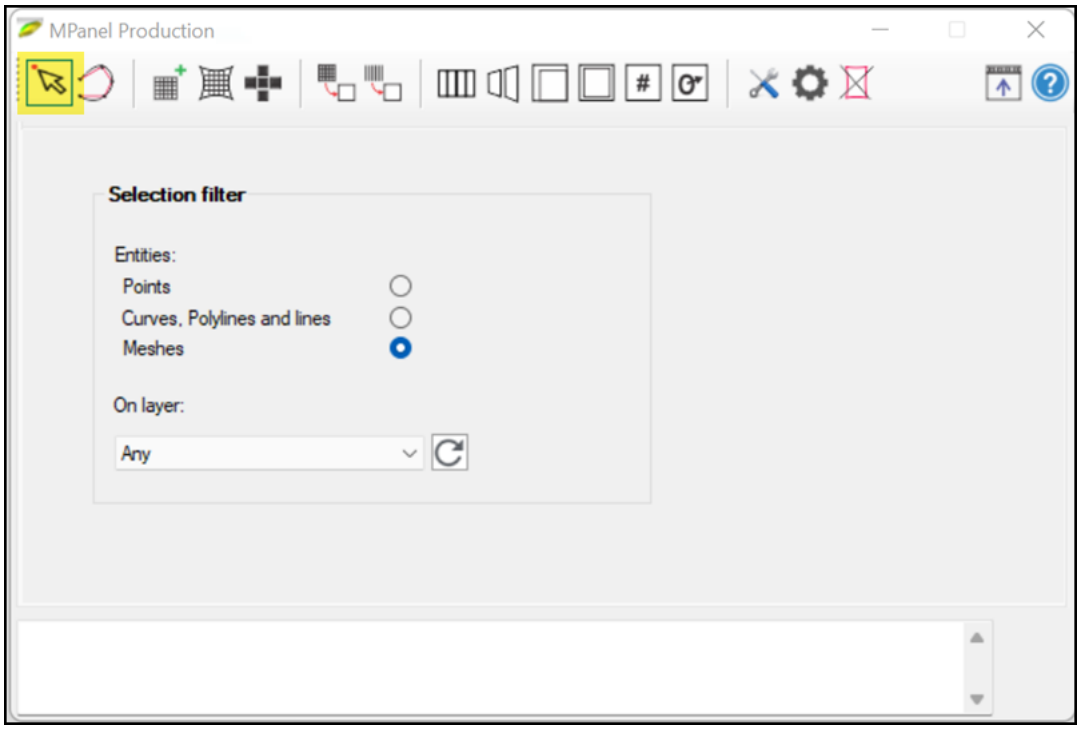
- New "Add mesh between intersecting curves" speeds up model building.
- New "Add hole in Panel" for clear fabric windows with simple split panel with hole to split the surrounding panel into parts.
- New "Selection filter" tool simplifies selection of meshes, points, curves, lines or polylines from your model so you can copy, move or delete them.
- New "Show seams and panel numbers on mesh" provides a new visualization of the model to help production with correct panel arrangement.
- Split panel by fabric width tool has been updated and improved.
- Push old objects to back of view to make it simpler to select new items.
- Panel numbers shown are the same as the MPanel panel ID numbers.

General



MPanel Production is a separate toolbar which works in Rhino to draw, shape and panel mesh shapes that represent fabric. It consists of a set of tools that interact with the Rhino drawing.

The program is arranged as a toolbar with options for each tool shown in the window below when the toolbar is maximized. The program can be minimized to show just the toolbar, or the option pages can be left open and the toolbar used to run the tools by clicking on the open/close toolbar option button. If "autohide to toolbar" is selected (settings) then the open/close toolbar buttons will not be shown.



Video tutorial: **Toolbar operation**

Installation

Make sure that Rhino is closed.
MPanel Production needs to install a Rhino plug in, and this can't happen if Rhino is running

Run the MPanel Production setup.
During the setup the Rhino installations will be detected, the plug in installed, and the MPanel Production files copied to your computer.

Running MPanel Production for the first time

Start Rhino
MPanel Production expects Rhino to be running when it starts.

Start MPanel Production
You will be prompted to enter the authorization code that was issued to you.
This will be checked on line, so internet access is needed at this point.

You will be prompted to select the units you normally work in. This will help MPanel production to show sensible default values.

MPanel Production Workflow

Usually a project will involve using each tab in turn, like a wizard with a workflow from left to right, to go through these steps:

- Boundaries or edges are created / adjusted / smoothed from measured data.
- A mesh, representing a fabric surface,is made to fit the boundaries or edges.
- Optionally the mesh is shaped by tensioning, inflating, or tightening cable edges.
- Meshes and any added points are copied from the model and then arranged so they are facing up
- Meshes are converted into flat panels
- The panels are shrunk or enlarged, seams are added, and a stamp with panel identification is added
- The panels are output in a format suitable for a cutter, plotter, or hand construction.

The text area at the bottom of the MPanel screen will give useful information as each tool runs.

When using a tool, it can be useful to see the "before and after" images. The "before" image is colored Red, and the new result is colored Black. After comparing them, you can use the "Delete Old" button to remove the old entities from the drawing. It is good practice to make a copy of the model at each step of the workflow either to the right or below before applying the next set of tools so you have a complete history of the project through each step of the process. This is particularly useful if you need to send a file to support with technical questions as we often will need to trace every step of the project in order to provide assistance.

Selection Filter

The Selection filter tool introduced in v8 of MPanel Production, simplifies the selection and extraction of meshes, points, curves, lines or polylines from your model so you can copy, move or delete them. To operate the filter, select your model in Rhino (including all elements such as meshes, lines, arcs, points etc., then use the filter to isolate the chosen items from your model, e.g. if you set the filter to "meshes" then after running the tool, only the meshes that were part of your model would remain selected. It is then a simple case to click back into Rhino (left click in Rhino header to make it active) and then using Rhino commands you could copy, move, or delete them.

During a normal project workflow you would use this procedure to copy your meshes form the model and then use the "Arrange Mesh" tool to arrange them in preparation for paneling.

The procedure is demonstrated in the following video example.

- **Selection Filter**

Points

After running the tool, only the points that were part of your model would remain selected and the number of points is shown in the information window.

Curves, Polylines and lines

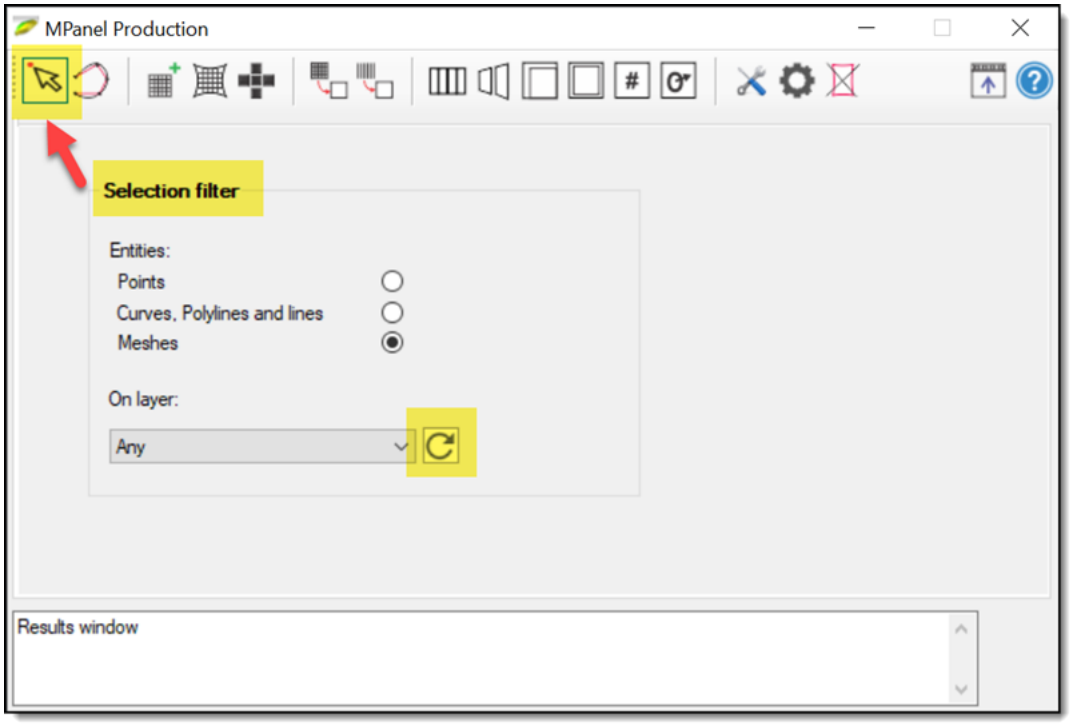
After running the tool, only the curves/polylines/lines that were part of your model would remain selected and the number of curves/polylines/lines shown in the information window.

Meshes

After running the tool, only the meshes that were part of your model would remain selected and the number of meshes is shown in the information window.

On Layer

The default setting is "Any" so filters will apply to all layers. By selecting a layer you can further refine the selection criteria to those items available on the selected layer. If you make any changes to layer names, renaming, adding etc then it is important to refresh the layer filter using the refresh button to the right of the layer selector. Alternatively, you can switch to another MPanel Production tool tab and then switch back to the Selection Filter tool - this will perform the same function to reset the layer list.



Video tutorial: **Selection Filter**

Prepare Data

Machine digitized data points often have some issues that prevent them being used directly as boundaries for modeling a fabric surface. They may be partial sets of points along a frame, or a set of points that includes several frame members, or there may be gaps in the data. And machine digitized data points usually have some jitter, or slight roughness, that can give problems during modeling. The tools on this page help to solve these issues and prepare the data for modeling. The raw data is usually in the form of polylines.

- **Modify polys**

Join points into poly

Takes a set of points and converts them into a single polyline. This is useful when the digitized data is in the form of separate point measurements.

Convert curve to poly

Converts a single curve, arc or line into a poly with the specified number of points. Specify a number of points that is sufficient to approximate the curves, which will often be a lower number than expected.

Convert several curves to a poly

Takes a set of curves and lines, and converts them into a single polyline, preserving the curve ends as nodes in the poly. This lets you create a polyline that captures corners where two curve meet. It is useful when the digitized data is in the form of arcs and lines. Specify a number of points that is sufficient to approximate the curves, which will often be a lower number than expected.

Renumber points in a poly

Takes a single polyline and allows for the number of points (vertices) to be changed.

Join Polys

Takes any number of selected polys and joins them together into a single poly. If there are small gaps between the ends of the polys there will be an option to ignore the gaps by using just one of the poly end points. This is useful when a frame member has been machine digitized into several polys, which need to be joined together to form a boundary edges for the fabric surface.

Split a Poly

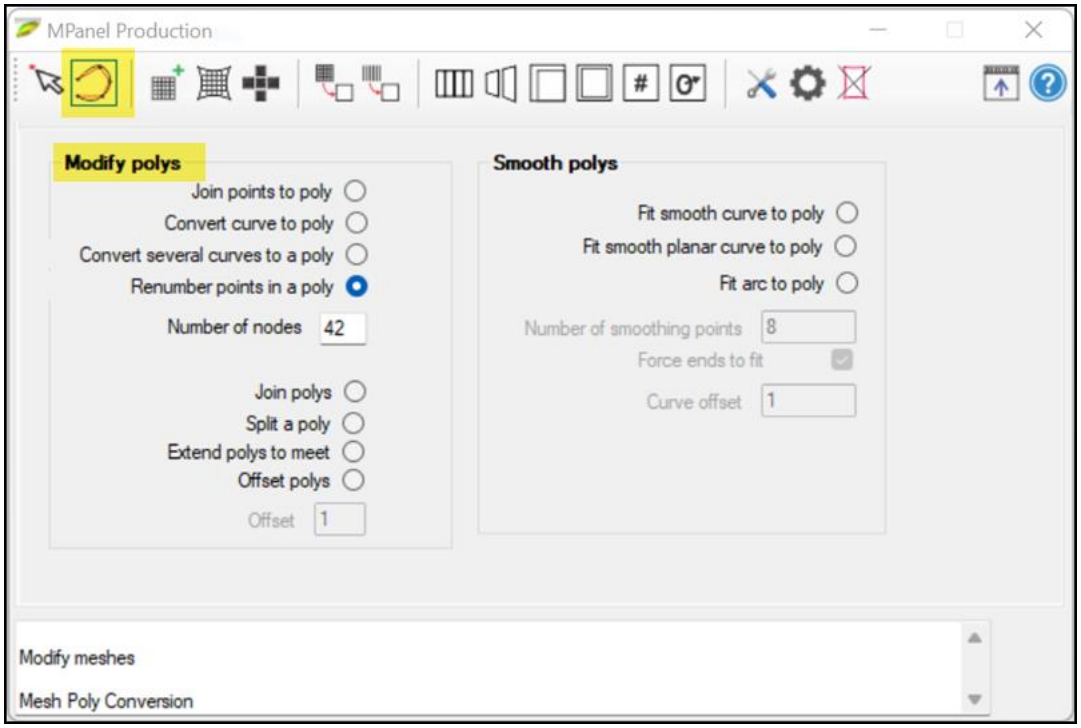
Takes a selected poly and asks for the split point to be selected on screen. The poly is then split at the nearest polyline node to the selected point. This is useful when several frame members has been machine digitized into a single poly, which then needs to be split to form distinct boundary edges for the fabric surface.

Extend 2 polys to meet

Takes 2 selected polys which have a gap between them, and extends each poly to join at an extended intersection. This is useful when two frame members join in an acute corner, and it wasn't possible to machine digitize all the way into the corner due to the measuring probe size.

Offset polys

Takes a poly, and if the poly is planar applies an offset to the poly. This is useful when a frame member is machine digitized along it's bottom edge and the fabric boundary will be the top edge. In this case the offset would be the frame diameter.



Video tutorial: **Make meshes with poly edges**

- **Smooth polys**

Fit smooth curve to a poly

Takes a poly that has some jitter, or slight roughness, and draws a smooth curve that best fits between the poly points. The smooth curve is similar to the curves produced by pipe bending machines. The curve is defined by a number of smoothing points, a higher number of smoothing points gives a better fit but with more waviness in the curves. In general the lowest number that gets a reasonable fit to the data points should be used, and 4 smoothing points are often sufficient. It may be useful to force the curve start and end points to match the poly start and end exactly, to ensure continuity with the next poly. The smooth curve can then be used as a boundary edge when building

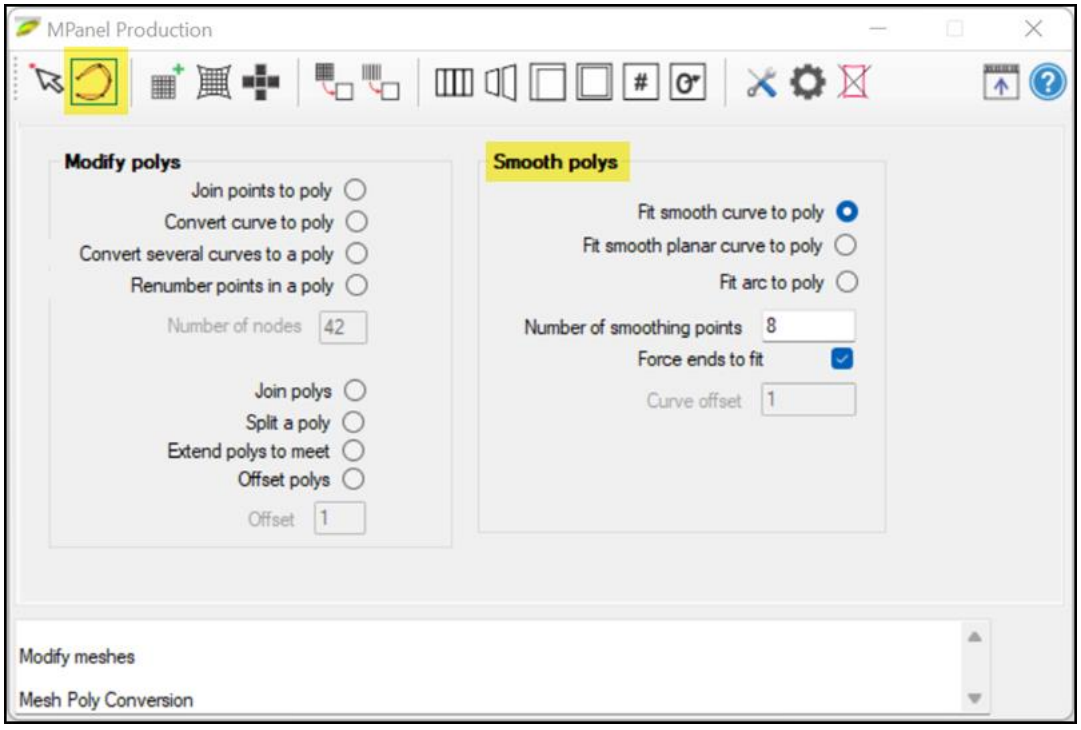
a mesh from edges.

Fit smooth planar curve to a poly

If it is known that the frame is planar (so the frame member could be cut out and then laid flat on the round) then restricting the smooth curve to be planar will give a better approximation to the true frame shape. Once a curve is planar it can have an offset specified. This is useful when a frame member is machine digitized along it's bottom edge and the fabric boundary will be the top edge. In this case the offset would be the frame diameter.

Fit arc to a poly

If it is known that the frame is a simple arc (perhaps produced by a single run through a pipe bending machine) then restricting the smooth curve to be an arc will give a better approximation to the true frame shape. Arcs are always planar, so an offset can be specified. This is useful when a frame member is machine digitized along it's bottom edge and the fabric boundary will be the top edge. In this case the offset would be the frame diameter.



Video tutorial: **Prepare data**

Add Meshes

This tool is used to add meshes (that represent fabric surfaces) to the drawing, and to shape the meshes.

In version 9, a new "edge filter" was Implemented which simplifies adding meshes by restricting edge selection to valid edges.

- **Add mesh between 2, 3, or 4 edges**

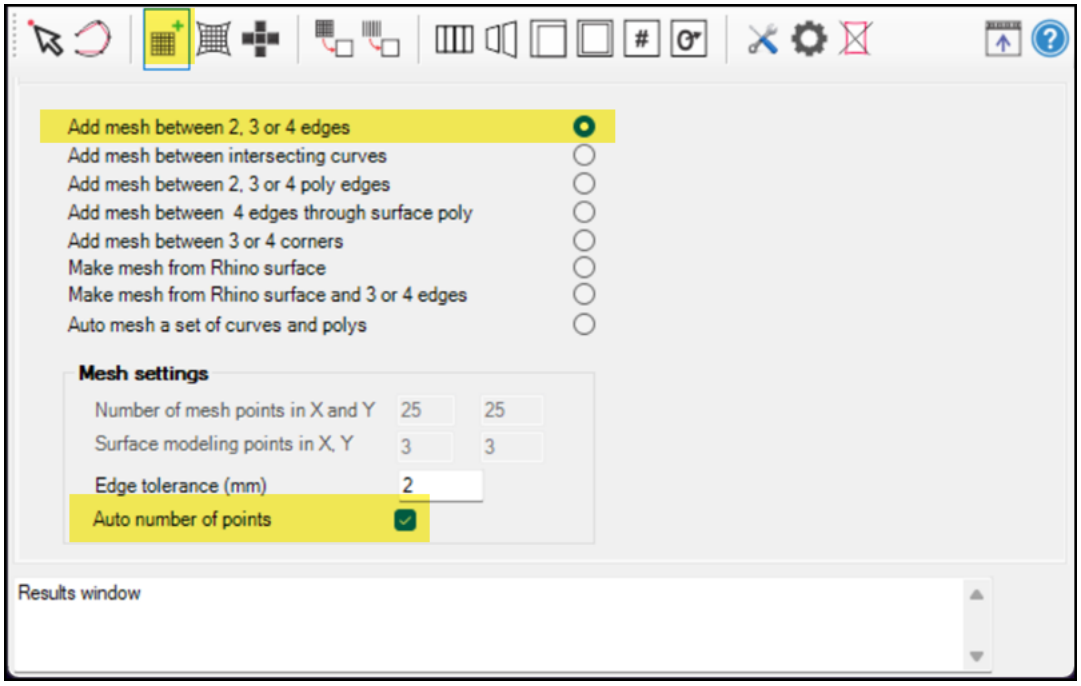
This draws a mesh that is bounded by any 2, 3 or 4 simple drawing entities. The drawing entities can be lines, polylines, curves, splines or arcs. The drawing entities can be 3d, in any orientation. The edges must meet at the corners.

Choose the number of points in the X and Y direction that will give sufficient accuracy in the finished drawing. Often 13 by 13 is sufficient. Although you can specify any number here, the numbers 13, 17, and 25 are mathematically good numbers that will make the paneling easier. Checking the "Auto number of points" option at the bottom will cause the program to automatically determine the X and Y mesh point count required to meet a specified edge tolerance. The program will try to place a mesh edge point at any sharp corners (more than 45degrees) in an edge, and will place enough additional points to capture the edge to the desired accuracy. Edge tolerance specifies the maximum distance between the mesh edge and the edge line/curve.

Using this tool means that the model will be built with meshes that have the lowest point count consistent with the specified accuracy. Keeping the point count low helps with program speed and drawing clarity.

Run the tool by clicking the "Add meshes" button, and a prompt will appear in the Rhino text window to select the edges, bottom edge first.
For a 3 sided mesh skip the top edge by clicking the Enter key on your keyboard.
For a 2 sided mesh skip the left and right edges by clicking the Enter key on your keyboard.

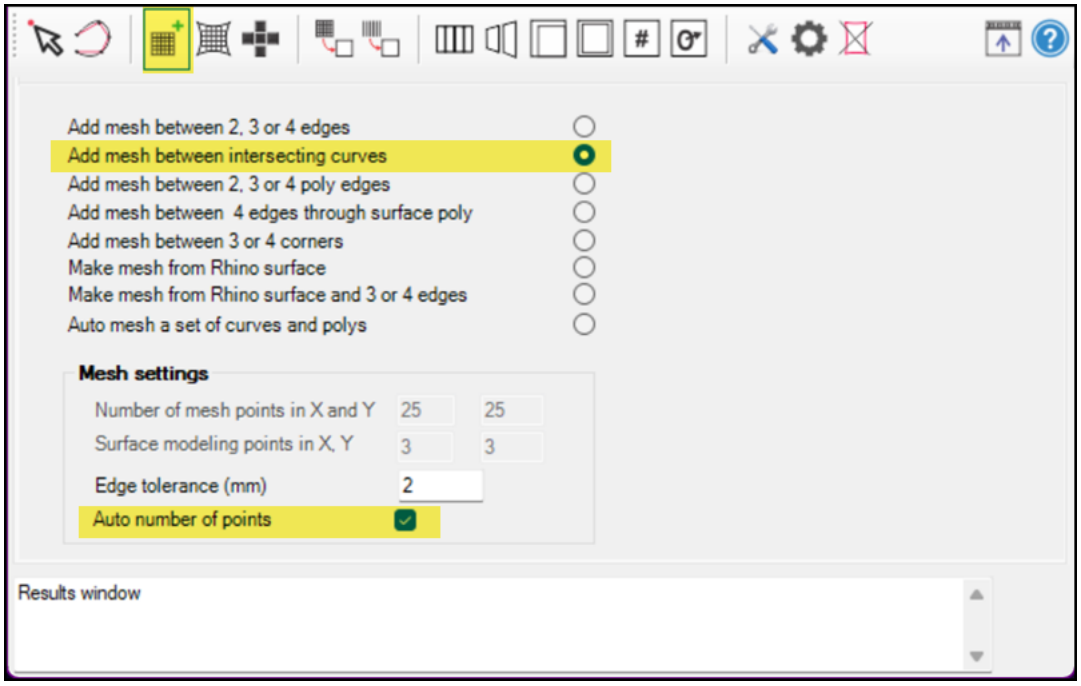
The mesh drawn will be a smooth shaped mesh that blends the edge gradient gradually. In some cases this mesh will be an adequate representation of the fabric surface.



Video tutorial: [Add mesh between edges]

- **Add mesh between intersecting curves**

This works in a similar manner to the previous tool, but the edges only need to intersect on the same plane thereby reducing the time required to split lines/curves to use the previous tool. Like the previous tool, you can choose the number of points used to build the mesh or you can select the "Auto number of points" option to simplify mesh building.

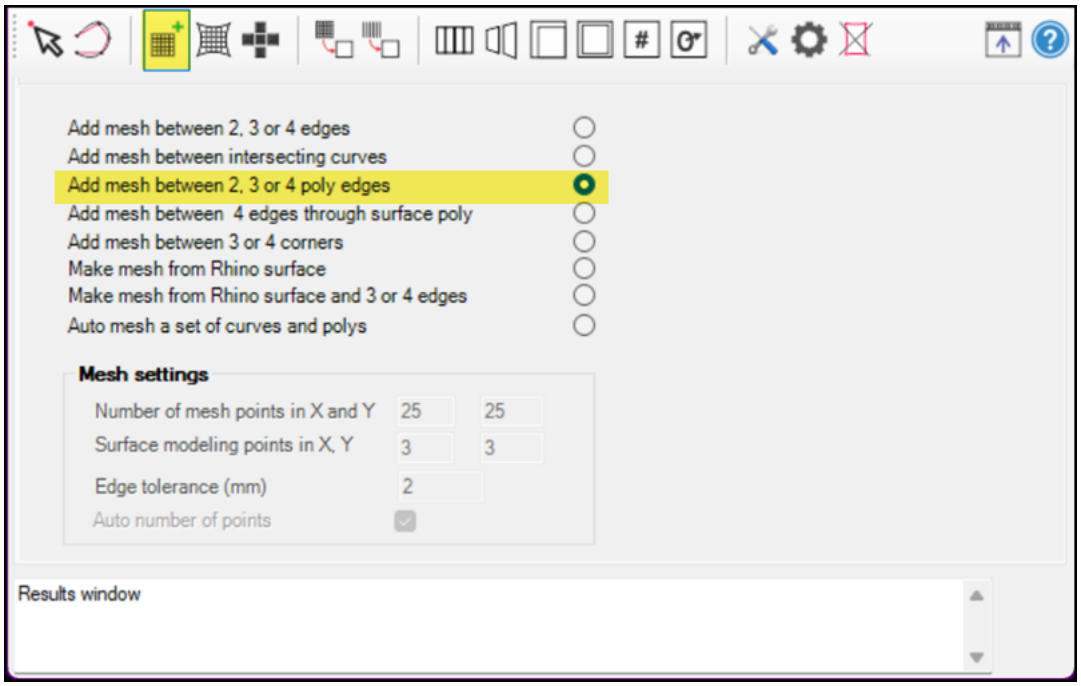


Video tutorial: **Add Mesh between intersecting curves**

Video tutorial: **Add Auto-Mesh between edges**

- **Add mesh between 2, 3 4 polys**

This draws a mesh that is bounded by any 2, 3 or 4 polys.. The mesh edge nodes will be directly based on the poly nodes. This preserves any corners in the edge polys. The number of mesh points in the X direction is determined by the number of points in the top and bottom polys, and the Y direction from the left and right edges. This means that the top and bottom polys must have the same number of points, and similarly for the left and right.



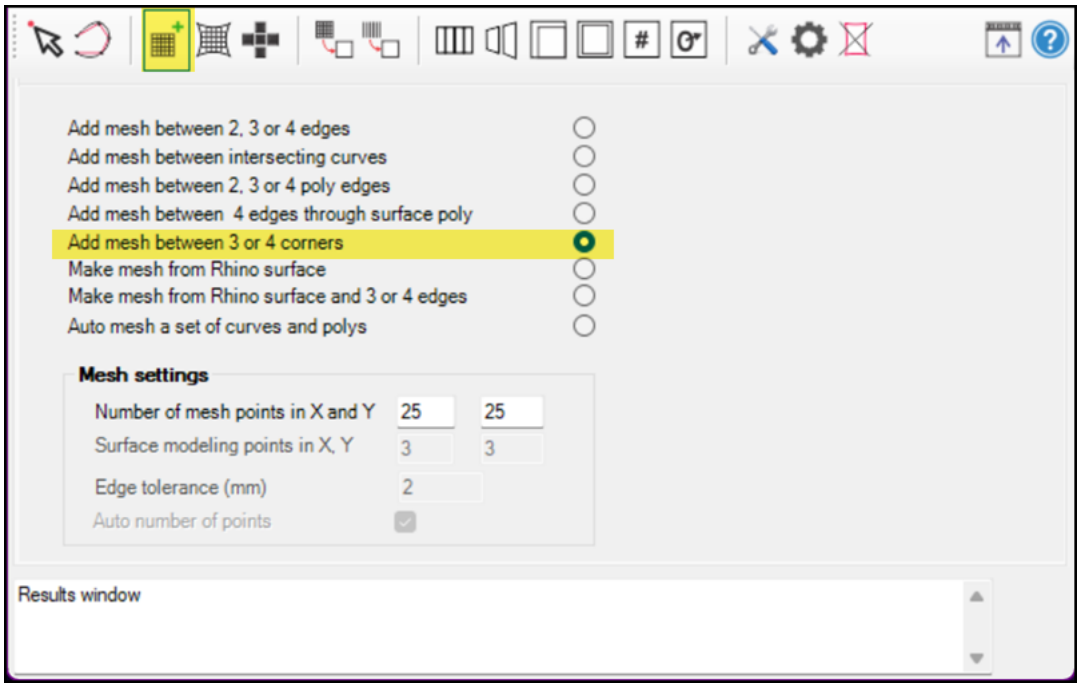
Video tutorial: **Make meshes with poly edges**

- **Add mesh between 3 or 4 corners**

This draws a mesh between 3 or 4 corners, with the mesh edges being straight. The corner points can be in 3d.

Choose the number of points in the X and Y direction that will give sufficient accuracy in the finished drawing. Often 13 by 13 is sufficient. Although you can specify any number here, the numbers 13, 17, and 25 are mathematically good numbers that will make the paneling easier.

Run the tool by clicking the "Add meshes" button, and a prompt will appear in the Rhino text window to select the corners, bottom left first.
For a 3 sided mesh skip the top left corner by clicking the Enter key on your keyboard.



Video tutorial: [Add mesh between corners]

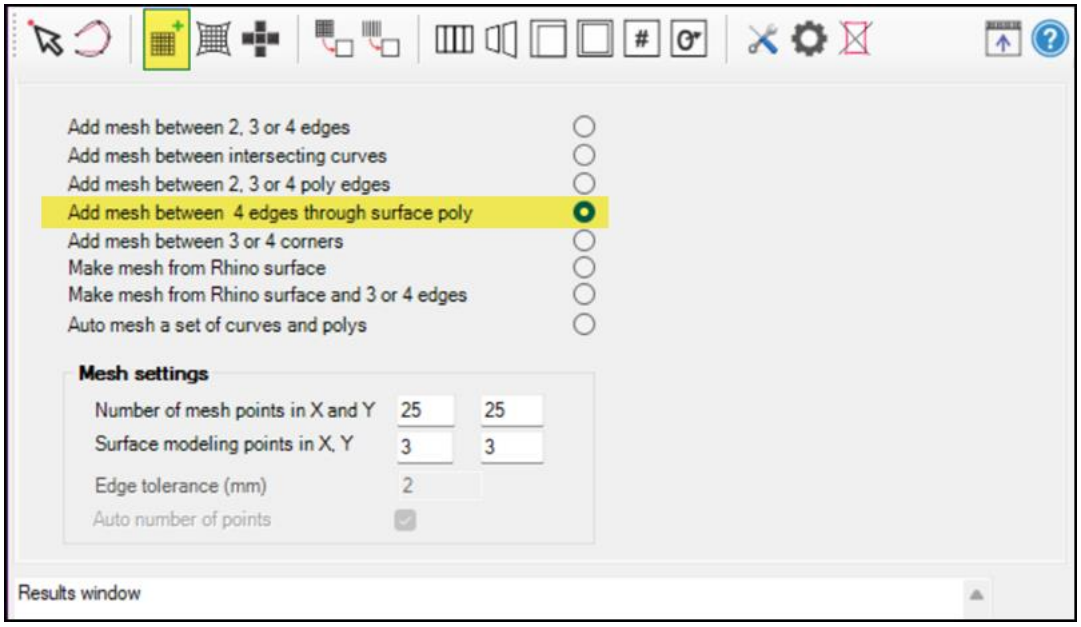
- **Add mesh between 4 edges through surface poly**

This is used when there is information available about the surface shape, often in the form of a "scribble poly" or surface poly" that has been traced over the surface by digitizing equipment. It will draw a mesh that is bounded by 4 sides, shaped to fit approximately through the points on the surface poly.

Choose the number of points in the X and Y direction that will give sufficient accuracy in the finished drawing. Often 13 by 13 is sufficient. Although you can specify any number here, the numbers 13, 17, and 25 are mathematically good numbers that will make the paneling easier.

Choose the number of surface modeling points. These are used to define the internal shape that the mesh, and a lower number gives a smoother mesh. Often a setting of 2 is sufficient to allow the surface to approximate to the surface poly.

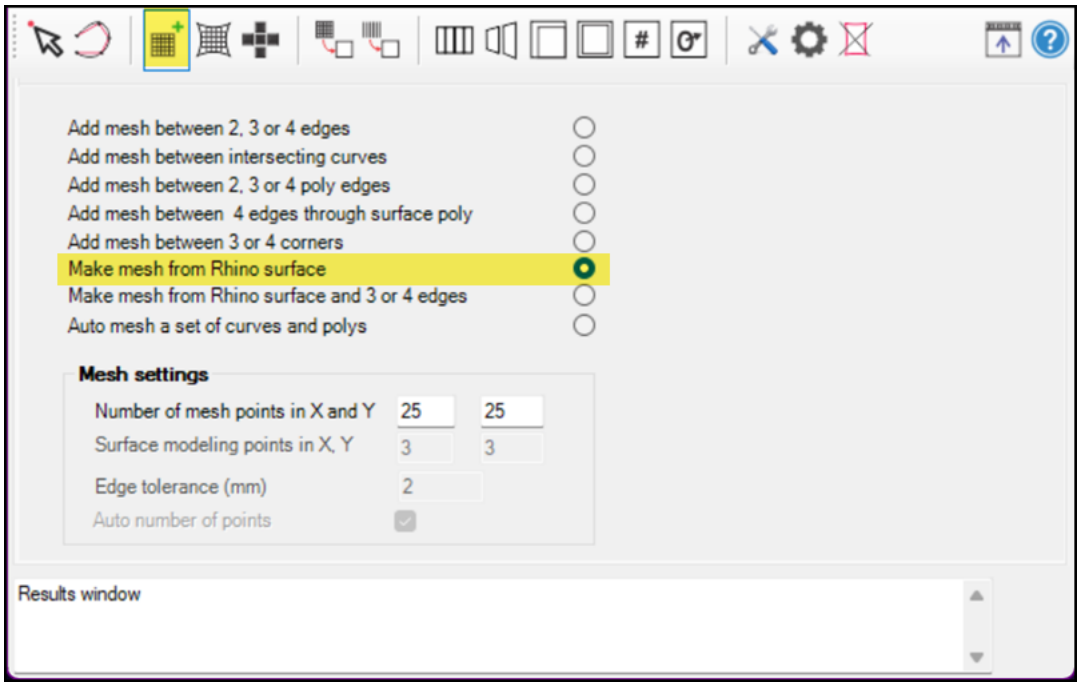
Run the tool by clicking the "Add meshes" button, and a prompt will appear in the Rhino text window to select the edges, bottom edge first. After the 4 edges have been selected you are prompted to select the surface poly.



Video tutorial: **Add mesh between edges and surface poly**

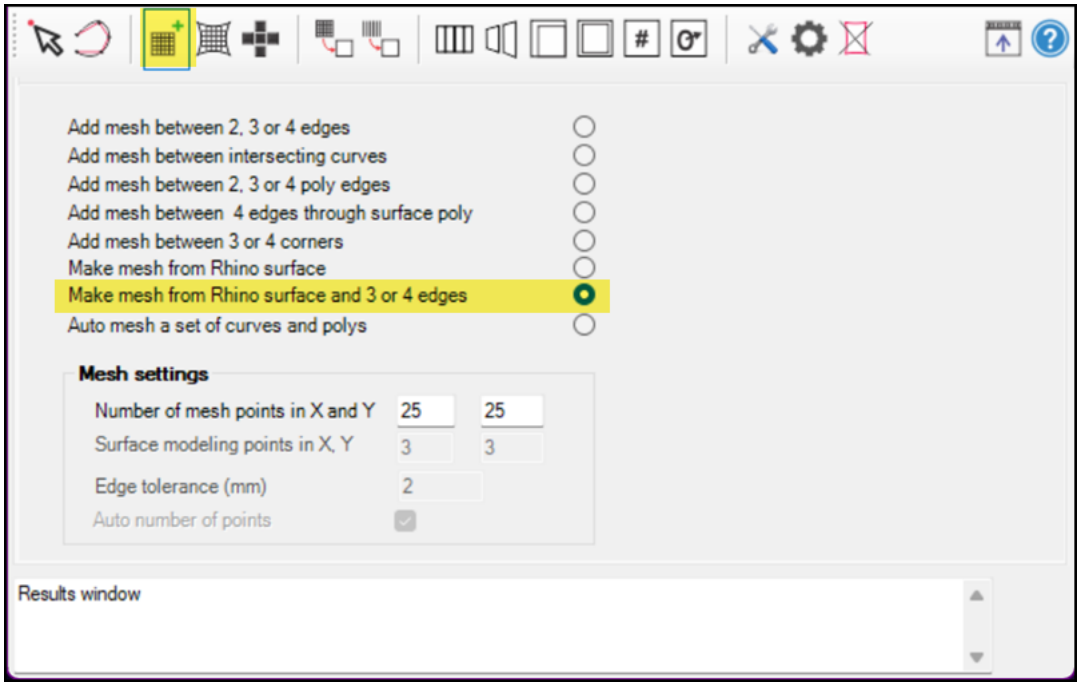
- **Make mesh from Rhino surface**

This tool will take a simple surface, and try to convert to a regular polygon mesh that can be used by MPanel. The surface needs to be a single surface, with 3 or 4 distinct edges.



- **Make mesh from Rhino surface and 3 or 4 edges**

This tool will take a more complicated surface and still allow a regular polygon mesh to be created. The mesh will be crated in the surface between 3 or 4 boundary edges, which will have to be created by hand drawing or combining / splitting existing edges.



- **Auto mesh a set of curves and polys**

This tool automatically draws multiple 3 or 4 sided meshes based on selected simple drawing entities. The drawing entities can be lines, polylines, curves, splines or arcs. The drawing entities can be 3d, in any orientation. The edges must meet at the corners. The tool will correctly add meshes with simple model geometry however may not do so when more complex model geometry is selected. In these cases, the normal add single mesh tool will need to be used.

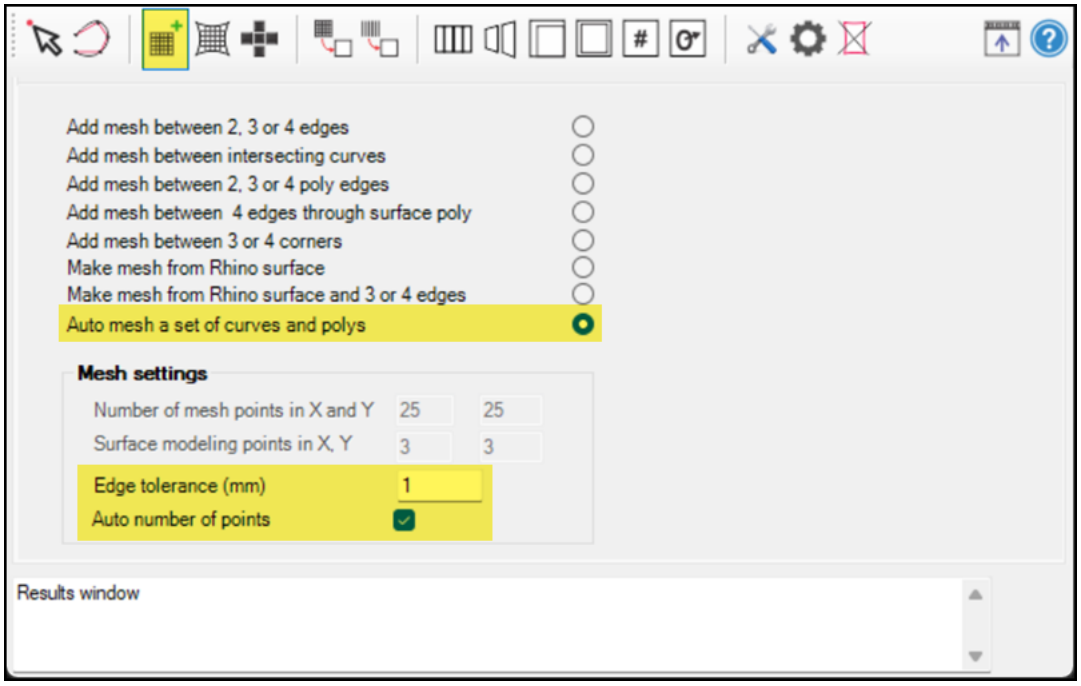
Choose the number of points in the X and Y direction that will give sufficient accuracy in the finished drawing. Often 13 by 13 is sufficient. Although you can specify any number here, the numbers 13, 17, and 25 are mathematically good numbers that will make the paneling easier. Checking the "Auto number of points" option at the bottom will cause the program to automatically determine the X and Y mesh point count required to meet a specified edge tolerance. The program will try to place a mesh edge point at any sharp corners (more than 45degrees) in an edge but may miss corners with lesser angles so it is essential to check these corners carefully and you may need to manually select a mesh point and move it to snap to the corner. Edge tolerance specifies the maximum distance between the mesh edge and the edge line/curve.

Using this tool means that the model will be built with meshes that have the lowest point count consistent with the specified accuracy. Keeping the point count low helps with program speed and drawing clarity.

Run the tool by first selecting the group of lines, curves, polys that make up the clean digital model then click the "Add meshes" button, and the meshes will be added automatically. If the model includes a closed poly e.g. the fender line around a boat then an unwanted mesh may be added. This can either be deleted or if the poly is split then the unwanted mesh will not be drawn.

The mesh drawn will be a smooth shaped mesh that blends the edge gradient gradually. In some

cases this mesh will be an adequate representation of the fabric surface.



Video tutorial: **Auto mesh tool**

Adjust meshes

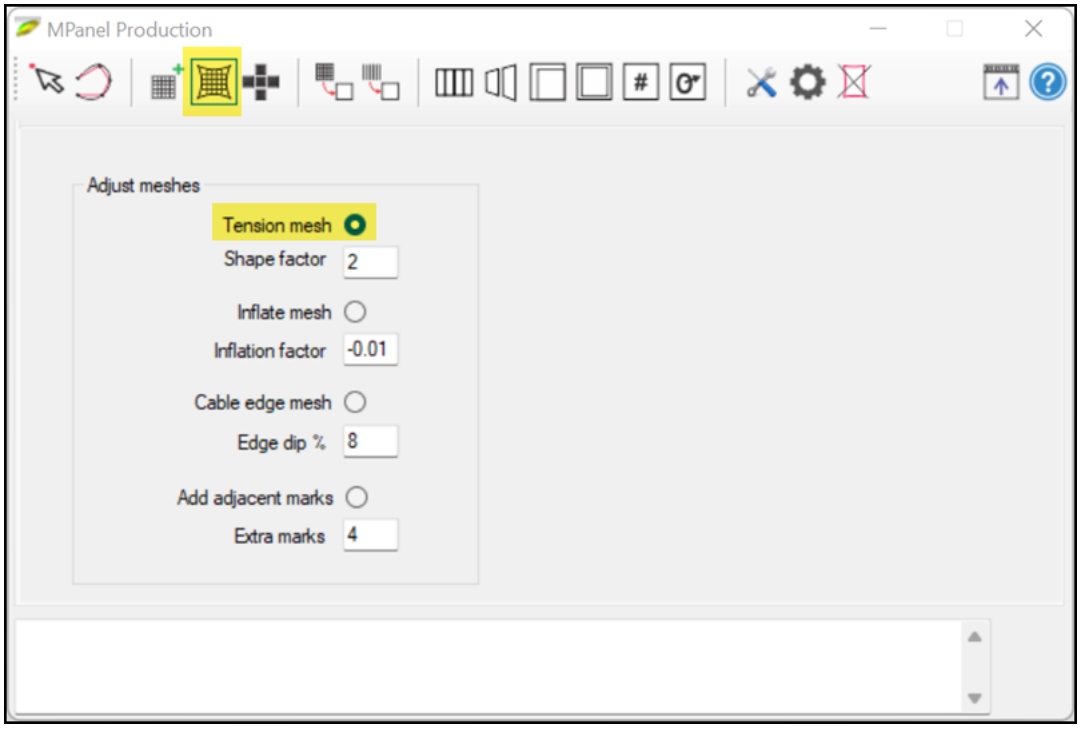
These tools adjust the mesh shape to better suit the end purpose.

- **Tension mesh**

If the mesh represents a fabric surface that is pulled tight, then the surface should usually be "saddle shaped". To shape the mesh like this click the "Tension mesh" button. This will move the mesh surface towards the correct saddle shape, using 100 iterations, or internal mesh adjustments. On some shapes you may need to repeat this process (by selecting the new mesh and running the tool again) until the final shape is achieved. The final shape is indicated by there being no further shape change of the mesh, or by the residual movement, reported in the text window, of less than 0.0001.

The shape of the saddle surface can be changed by setting the "Shape factor". This sets the ratio between the fabric stress in the warp direction and the fabric stress in the weft or fill direction. Although you can specify any number, most fabrics will wrinkle of the stress in one direction is more than 3 times the other stress.

Tensioning with a high density mesh can be slow, a cancel button will show during the process.



Video tutorial: **Change mesh shape by tension**

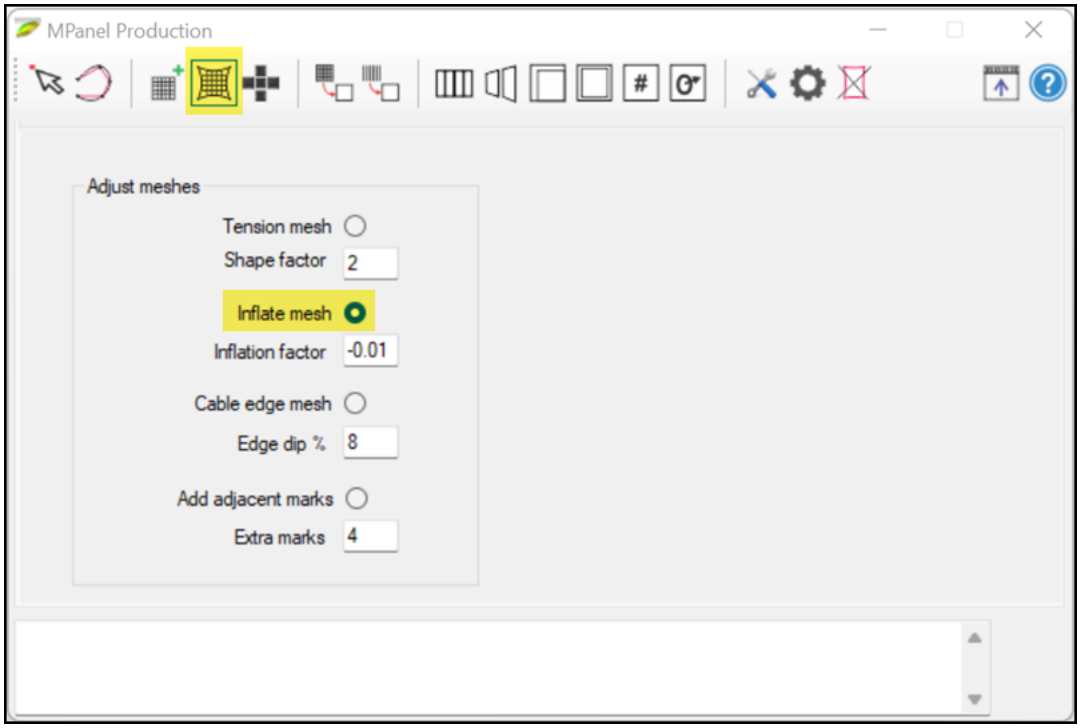
- **Inflate mesh**

If the mesh represents a fabric surface that is inflated, then the surface should usually be "cushion shaped. ". To shape the mesh like this click the "Inflate mesh" button. This will move the mesh surface

towards the correct inflated shape, using 100 iterations, or internal mesh adjustments. On some shapes you may need to repeat this process (be selecting the new mesh and running the tool again) until the final shape is achieved. The final shape is indicated by there being no further shape change of the mesh, or by the residual movement, reported in the text window, of less than 0.0001.

The amount of inflation is set with the "Inflation factor", and values like 0.3 are usually sufficient to create an inflated shape. Inflation is usually upwards. for downward inflation use a negative inflation factor. Large inflation factors will make the mesh unstable, and the model will "blow up"

Inflating with a high density mesh can be slow, a cancel button will show during the process.



Video tutorial: **Change mesh shape by infation**

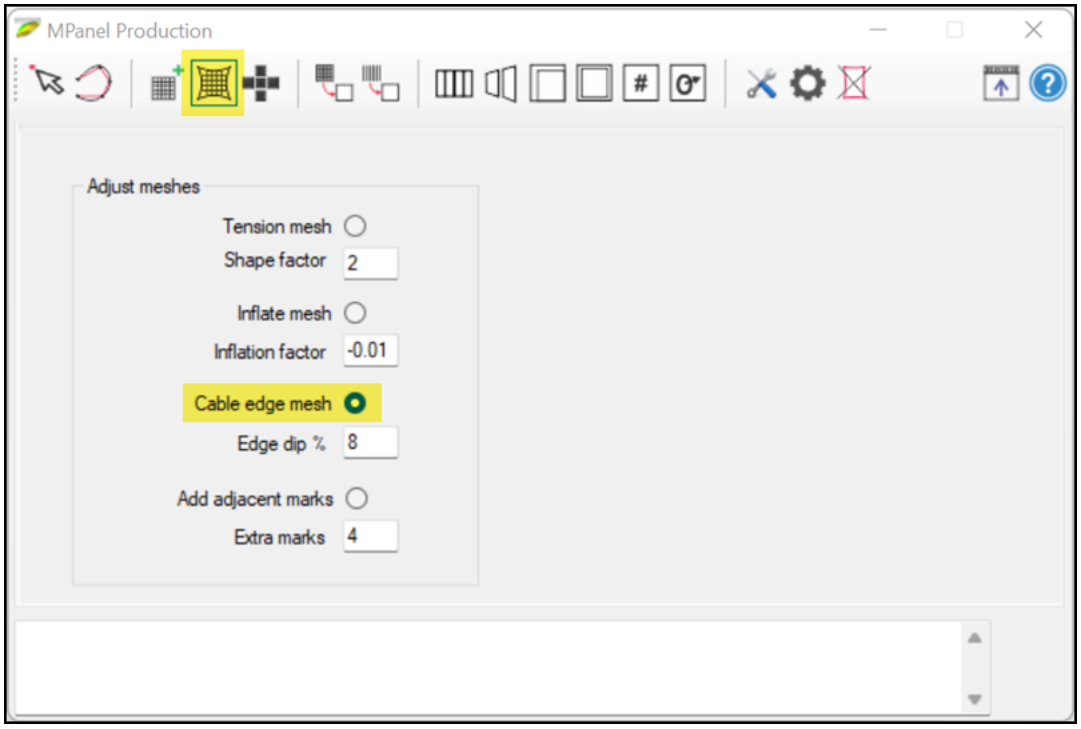
- **Cable edges**

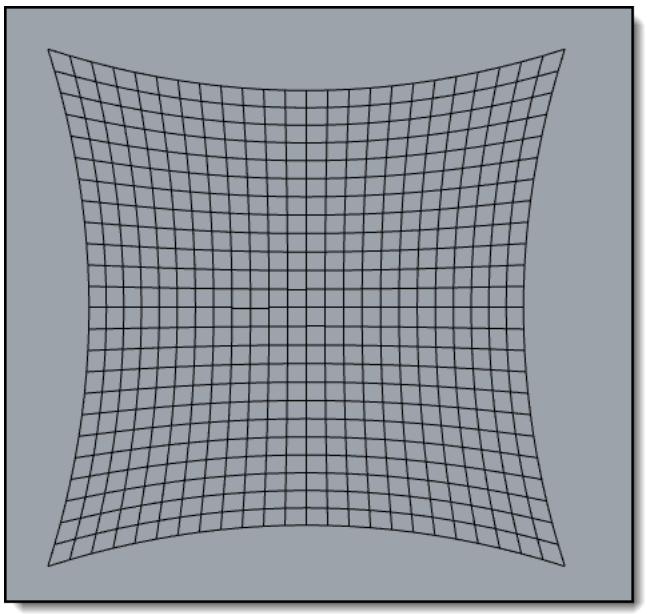
If the mesh represents a fabric surface (aka shade sail) attached to 3 or 4 corner points, with reinforced edges (webbing, rope, cable) then the edges should move inward to make a smoothly curved edge. The amount that the edges move inward is specified as a dip percentage of the edge length. A value of 8% is often used for cable edged shade sails.

On some shapes you may need to repeat this process (by selecting the new mesh and running the tool again) until the final shape is achieved. The final shape is indicated by there being no further shape change of the mesh, or by the residual movement, reported in the text window, of less than 0.0001.

In the case that one or more edges are fixed (e.g. keder edge) then a mesh edge poly should be added to the fixed edges (these can be produced using the Modify Meshes utility tool "Draw mesh edges"). The mesh and the fixed edge polys should be selected before running the Adjust meshes tool to relax the sail.

Cable edges with a high density mesh can be slow, a cancel button will show during the process.



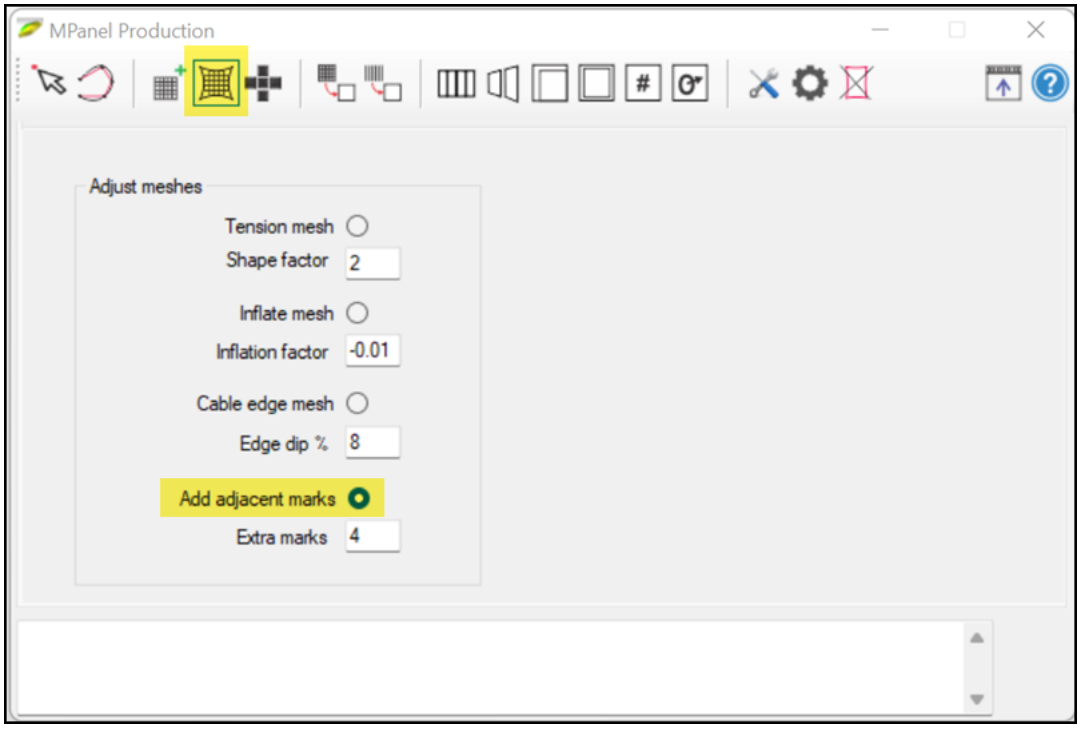


- **Adjacent marks**

It is useful during fabrication to identify with marks in the panel seam edge the location where more than one mesh joins with a larger mesh. To achieve this Production will add a point wherever multiple smaller meshes join to a larger mesh when the "Add adjacent marks" option is selected when running the Adjust meshes tool. When these meshes are paneled with the "Use points as panel marks" option selected then panel marks are added to the panels. When the panels are seamed these marks will show as alignment marks on the panel edges.

If a number is entered in the extra marks cell then this number of alignment marks will be added to all

panel seams.



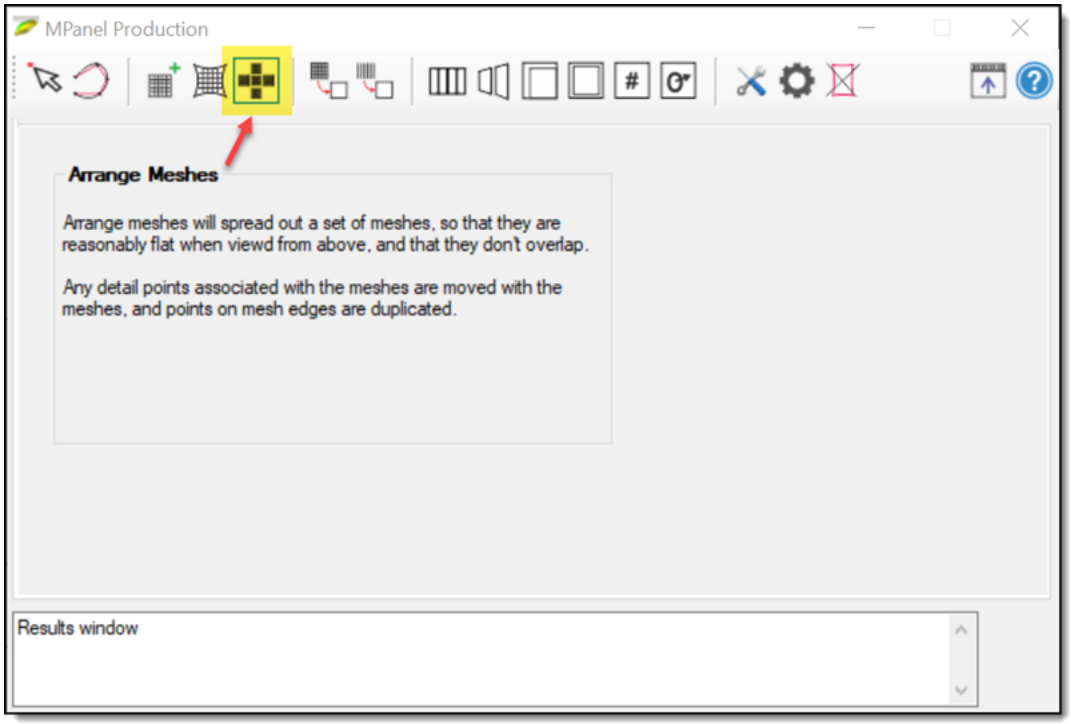
Vimeo Video: **Add adjacent marks**

Arrange meshes

Takes all of the model meshes and arranges them in an "exploded" fashion so that the meshes are oriented to be as flat as possible when viewed from the top. The position of points in the meshes is maintained and points are grouped with the meshes that contained them.

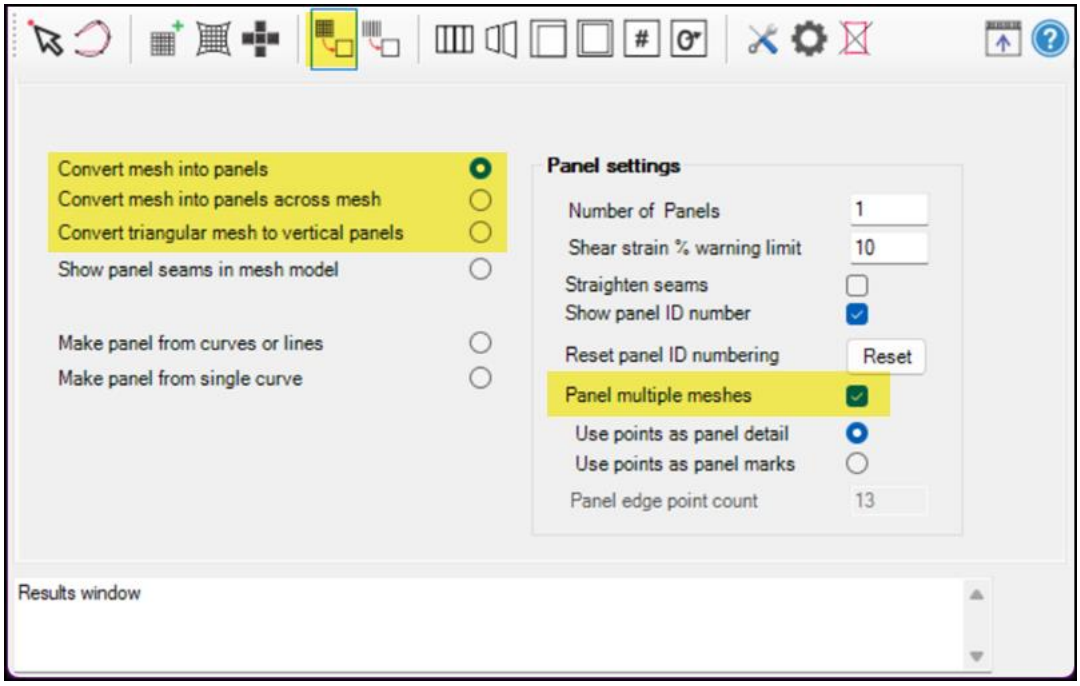
The arrange mesh tool will ignore any elements (e.g. curves, splines) other than meshes and points. It will however, add these other elements to the "Delete old object" list so they will be deleted when the Delete old button is pressed. For this reason, it is good practice to make copies of the model and meshes for each step in the workflow.

This can be used to make the meshes more easily visible before paneling, especially when some of the meshes are close to vertical, for example on a boat cover sides.

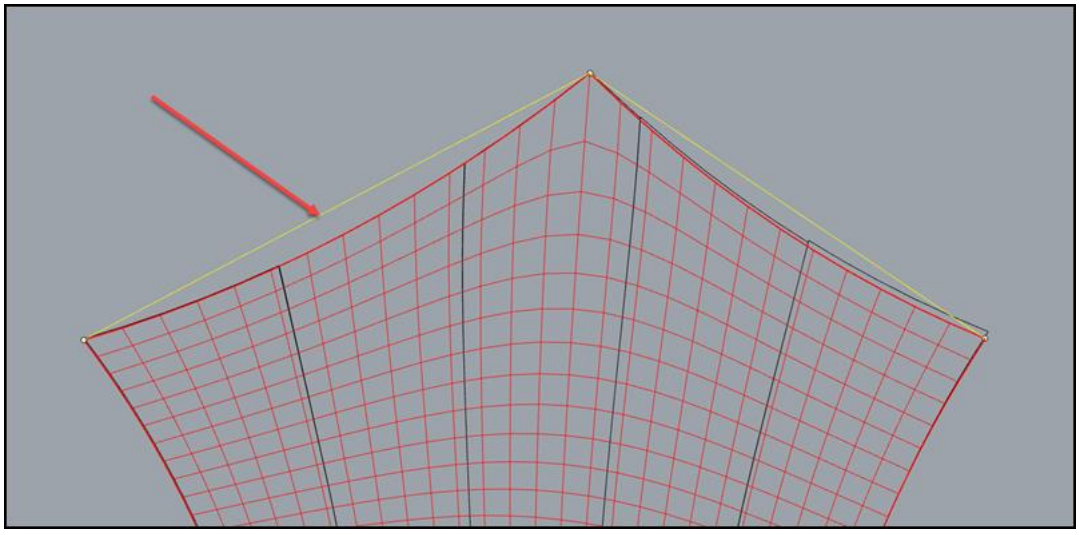


Video tutorial: **Arrange and panel multiple meshes**

Simple Paneling



Simple paneling converts the mesh directly into panels. Set the number of panels required, select the mesh, and run the tool by clicking the "Panel mesh" button. The panels will be drawn out directly under the part of the mesh that created them. The mesh can be split into more than one panel by specifying the number of panels however, if the mesh density is not evenly divisible by the number of panels mid connections of the mesh can be missed when paneled. To overcome this, a polyline can be drawn on the mesh (see arrow in below image) to pick up the mid connection. Then select the mesh and the polyline and run the tool to panel the mesh into multiple panels.



You can choose to panel multiple meshes at one time with the panel multiple meshes option checked. Each mesh is paneled individually, according to the panel settings. Although this is a time saving feature, it is sometimes better to panel meshes individually, to review each panel shape and shear strain separately.

In the Setting page you can choose for the panels to be the same color as the mesh. This can help to keep the panels organized and related to a particular mesh.

Panel across mesh

To panel "in the other direction", choose the option "Convert mesh into panels across the mesh"

Panel triangular mesh

If the mesh is triangular you can choose to panel with "vertical" seams.

Panel 2 sided mesh

In order to process panels (shrink, seam, stamp and output) they must have a left and right sides. In some circumstances (such as some bimini tops) meshes are created with a bottom and top edge only. When converting these meshes to panels MPanel will automatically panel across the mesh so the resultant panel will now have a left and right edge (no top and bottom) - a pop-up message advises this change.

Show seams in mesh model

Panel seams and numbers (if selected) can be shown superimposed on a mesh model using the "Show panel seams in mesh model" tool. A copy of the model meshes and panels is made and then with both the mesh and panels selected, run the tool (don't delete old as normal). See video below.

Panel from flat entities

Panels can also be created by selecting 3 or 4 flat entities (lines/curves) and choosing the option to "make panel from curves or lines".

Alternatively, panels can be created by selecting single flat entities (curves) and choosing the option to "make panel from single curve" which is useful for creating parts such as reinforcements etc. to be included in nesting plots.

Straighten panel edges/seams

Normally the panel edges (seams) will follow the mesh threads, this keeps the seams approximately evenly spaced. You can specify that the seams are straightened, and on simple shapes this will work OK. It is possible for a straightened seam to result in seams hitting the mesh edges, or in poorly spaced seams.

Shear strain

The panels will follow the mesh threads, but will have differently curved edges, so that when panels are sewn together the assembly makes an approximation to the curved surface. The remainder of the shaping to the curved surface has to be done by fabric distortion. Fabric distortion is only allowed by stretching / shrinking on the bias, and is called shear distortion.

So to minimize the amount of distortion in the fabric, you have to use a sufficient number of panels. You may have to make more panels than you would expect, to get most of the shaping done in the seams. The amount of shaping that can be done with fabric distortion (shear strain) varies with the fabric type, the amount of tension the fabric is under, and the tolerance for small wrinkles. Some approximate guidelines are:

Lycra 30%
Knitted HDPE (shade cloth) 10%
Canvas 5%
PVC coated polyester architectural fabric 1%

You can set a shear strain limit, and receive a warning message if the shear strain is too high during paneling.

Panel Numbers

During paneling a reference number is added to each panel. This reference number stays with the panel during the panel handling operations, but is automatically stripped from the final panel output stage. The Panel ID number can be reset back to 1 using the Panel ID "Reset" button. Caution should be taken when using the reset tool as the drawing may contain multiple panels with the same number which can be lead to confusion if the file is re-used in the future. We recommend if you reset the panel ID number that earlier panels which are being replaced should be deleted. The panel ID number will also be used when adding assembly marks in the stamp tool.

Points

Points drawn close to or on the mesh surface can be used in several ways to identify points in the panel that correspond to the 3D point in the mesh:

- Use points as panel detail

Points are transferred into the panel as small grey crosses that will typically be plotted on the panel surface. If a set of points are all colored with a simple color (Red, Yellow, Green, etc) then the points will be joined up and shown in the panel as a light grey polyline which could be used to indicate the location of a zipper or similar detail.

Vimeo Video: **Add Panel detail with points**

- Use points as panel marks

Points, which are usually near the mesh edges, are transferred into the panel as small orange crosses. During the seam process these crosses are converted to marks in the nearest panel seam.

If you need some points to be read as panel detail and some as panel marks, then Select the option "Use points as panel detail" and:

- Identify the points to be read as marks by coloring then Orange
- or identify the points to be read as marks by putting then on a layer called "Mark"

Video tutorial: **Create Panel simple**


Video tutorial: **Create Panel across mesh**


Video tutorial: **Arrange and panel multiple meshes**


Video tutorial: **Using panel detail and panel marks**


Video tutorial: **Panel detail from cross sections**


Video tutorial: **Show panel seams in mesh model**

Paneling with cross sections

Advanced paneling is used when simple paneling does not give the seams or panels that are required. Advanced paneling consists of 2 stages:

a) make cross sections that run across the mesh
b) convert the cross sections into panels.

- **Cross Sections**

To define a cross section, draw a line over the mesh, when viewed in Top view. Select the line and the mesh, and click the "Make cross sections" button. A polyline will be drawn that is the cross section of the mesh along that line.

Multiple cross sections can be made at once using multiple lines over the mesh. The lines must be going in the essentially the same direction.

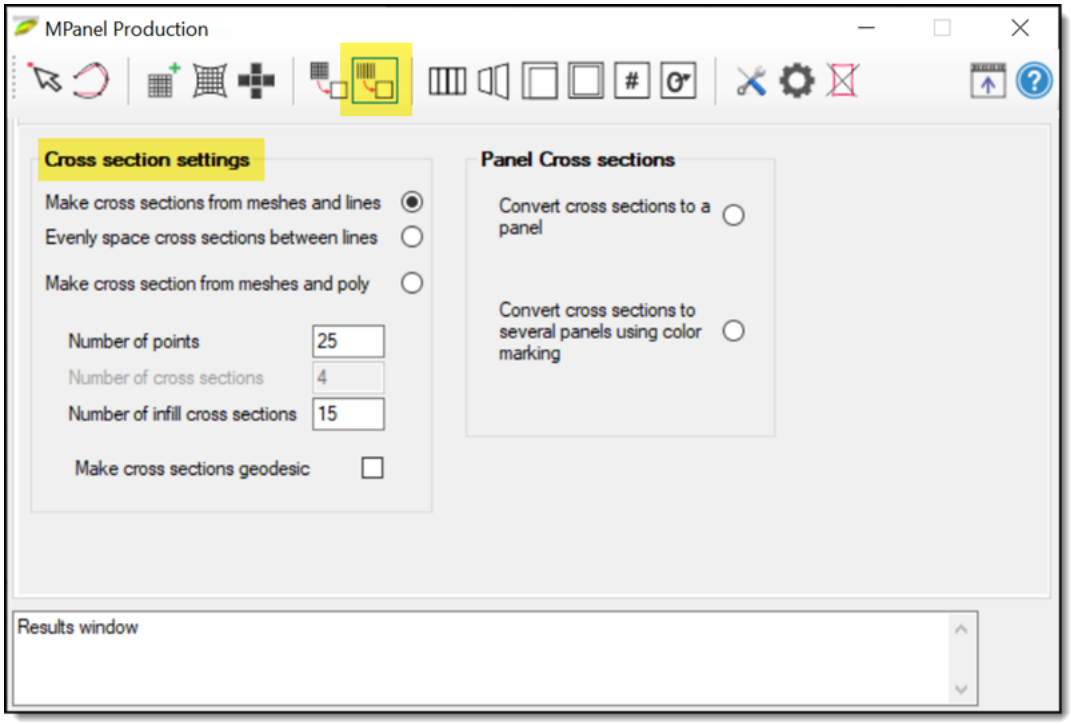
The number of points in the cross section should be similar to, or a bit more than, the number of points in the mesh. Often 20 points are used.

If a line goes corner to corner on a mesh edge, then the mesh edge will be used instead of the cross section.

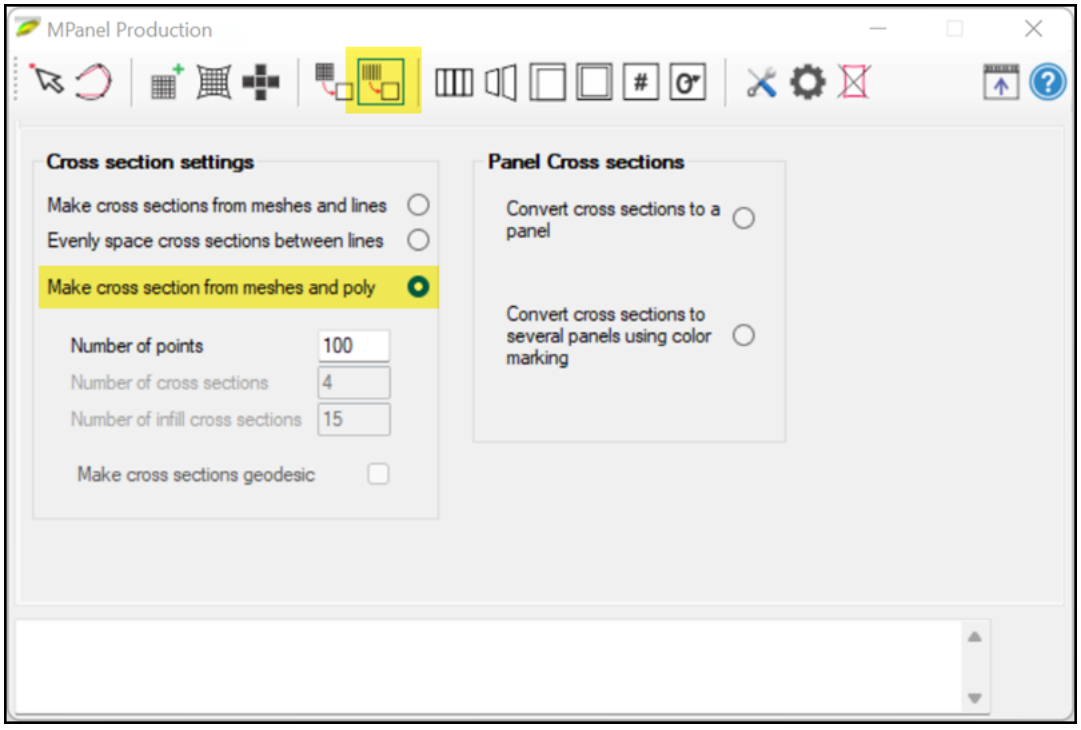
Often more cross sections are needed than the number of seams, to define the internal shape of the panel. These can be automatically created by specifying a number of infill cross sections.

Usually the seams from cross sections will be reasonably straight, as they are defined from a straight line. But they may have some curvature due to the 3d shape of the mesh. To straighten the seams more you can specify "Make the cross sections geodesic", which will alter each cross section to a curve on the mesh surface that will give a straighter seam.

Instead of drawing lines for each cross section, you can draw lines for the first and last cross sections, and then specify that the program draws more cross sections evenly spaced across the mesh.



In version 9, a new tool "make cross section from meshes and polyline" was added. This tool simplifies adding 3D details to the meshes and panels made from the meshes. Curves can be drawn on the mesh and converted to polylines which can be used to create a cross section following the mesh surface. Using the "Utilities" tool "Divide curve into points" the curve can be converted to points which if colored, can be added to panels as detail such as zipper lines etc. using points as panel detail.



Video tutorial: **Cross Section using polyline**

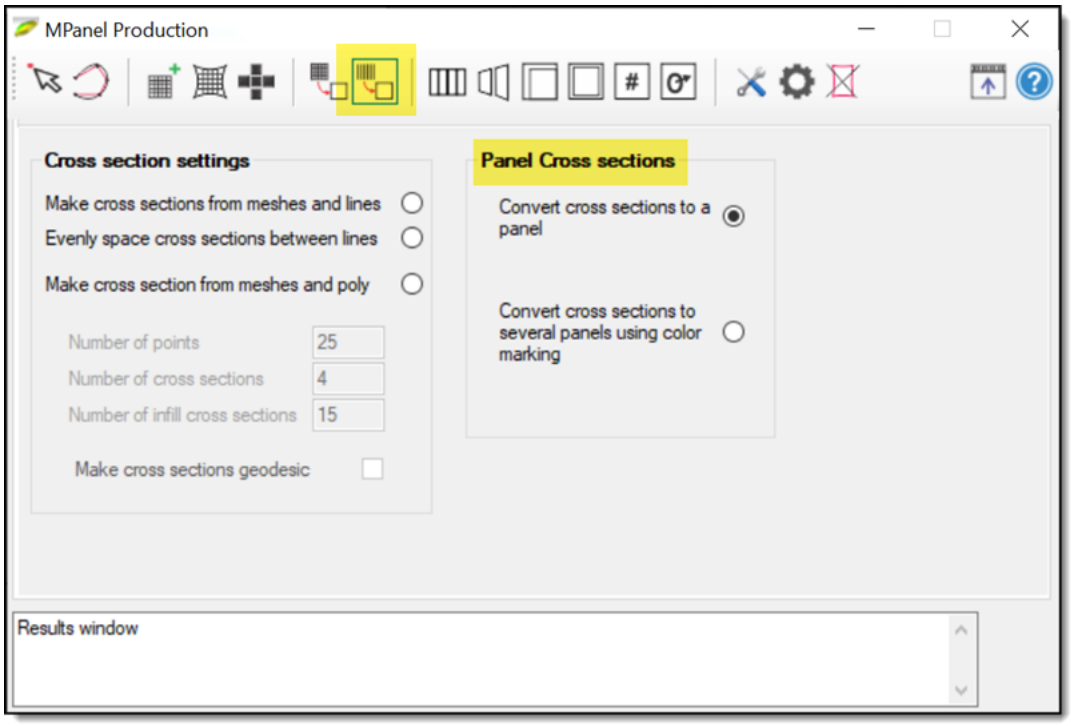
- **Make panels from cross sections**

Two or more cross sections (polylines) cab be converted to a panel by selecting them, an clicking on the button "Make Panels" . The polylines must be going in the essentially the same direction.

With more than two polylines used to make a panel, the panel will be shaped to pass through all of the polylines.

A number of panels can be made at once by coloring the polylines that represent seams, and choosing the option to "Convert cross sections to several panels using color marking". For color marking use simple colors, like Green, Blue, Cyan, or Magenta.

Cross sections created using infill cross sections are automatically color marked.



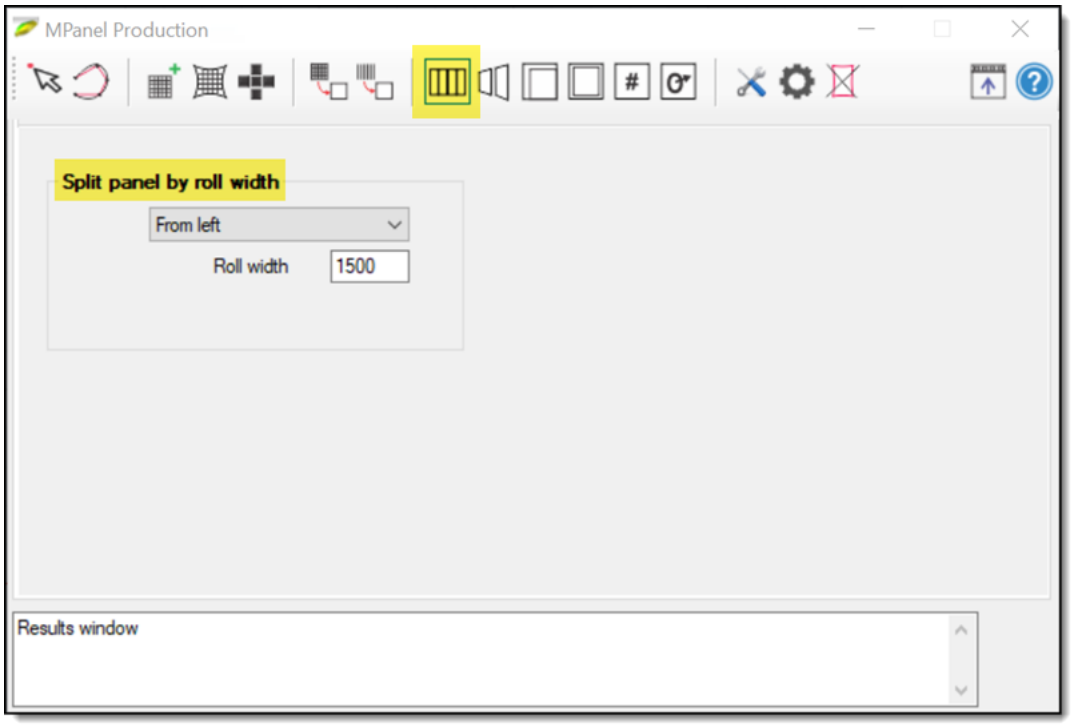
Video tutorial: **Create Panel using cross sections**

Video tutorial: **Create panel using evenly spaced cross sections**

Video tutorial: **Create panel with straight seams**

Split Panel

This takes a panel which is typically larger than the fabric roll width, and splits it into sub panels from a specified roll width. The specified roll width should include an allowance for material used in the seams. The panel can be split from the left edge, right edge, or from a center panel. The panel should be oriented on the screen so the bottom of the panel aligns with the bottom of the screen.



Video tutorial: **Split panels by fabric width**

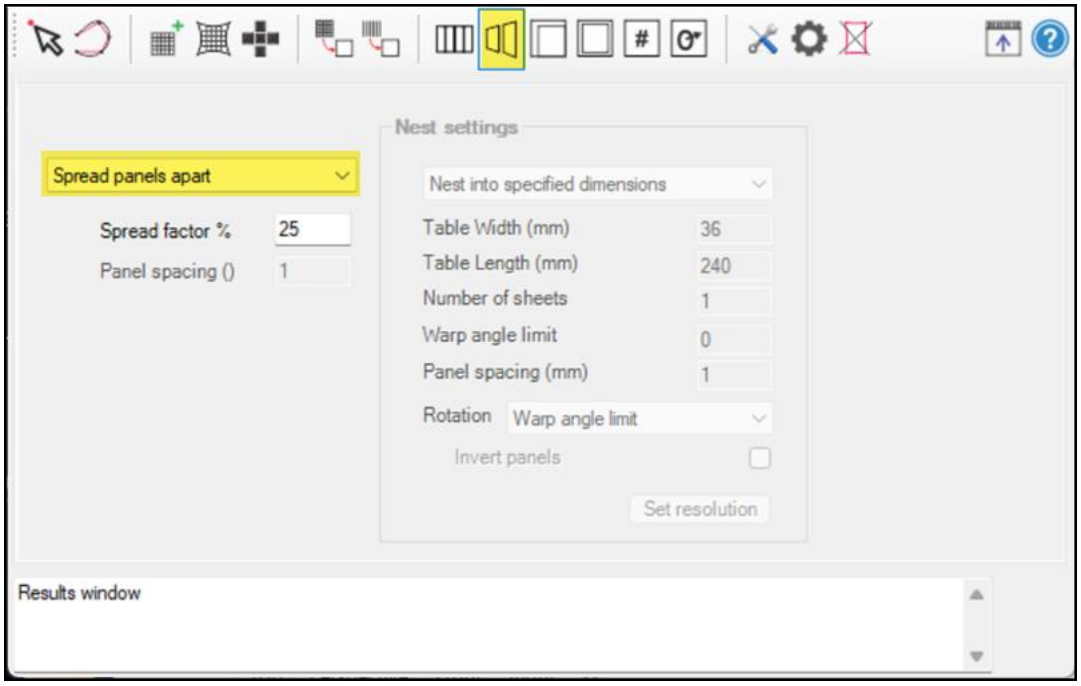
Arrange Panels

Arrange panels consists of tools that are used to arrange the panels on screen:

Spread panels apart
Arrange panels horizontally
Arrange panels vertically
Nest panels

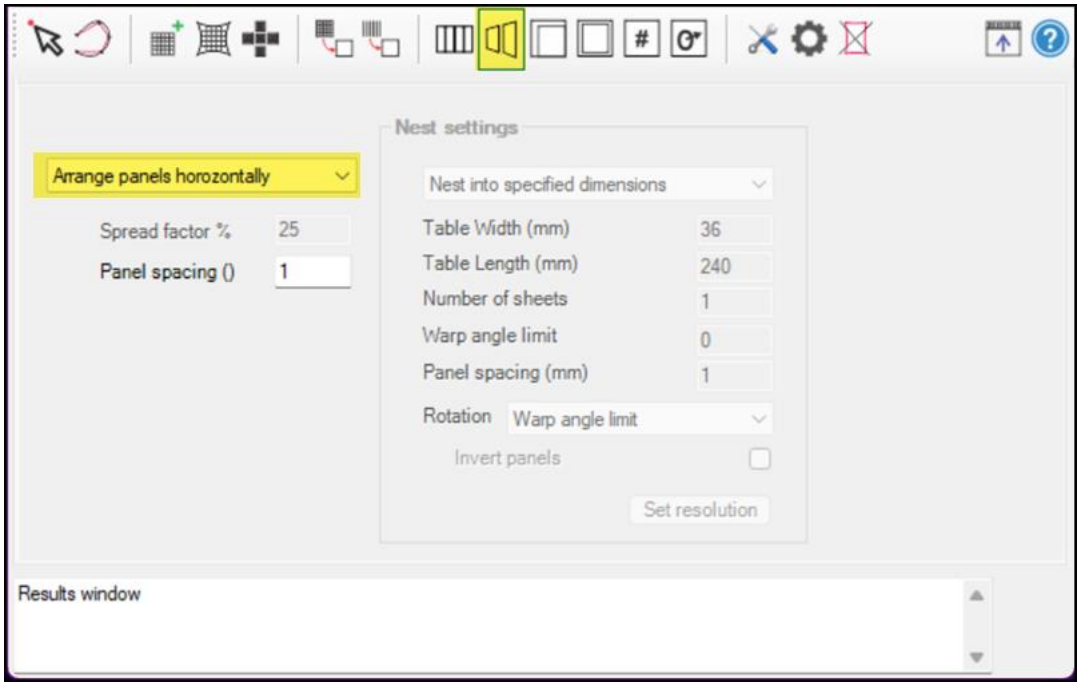
- **Spread panels apart**

This takes a set of panels and spreads them out by a specified percentage spread factor. This is useful if the panels are overlapping, or to get enough space to add seams without overlapping.



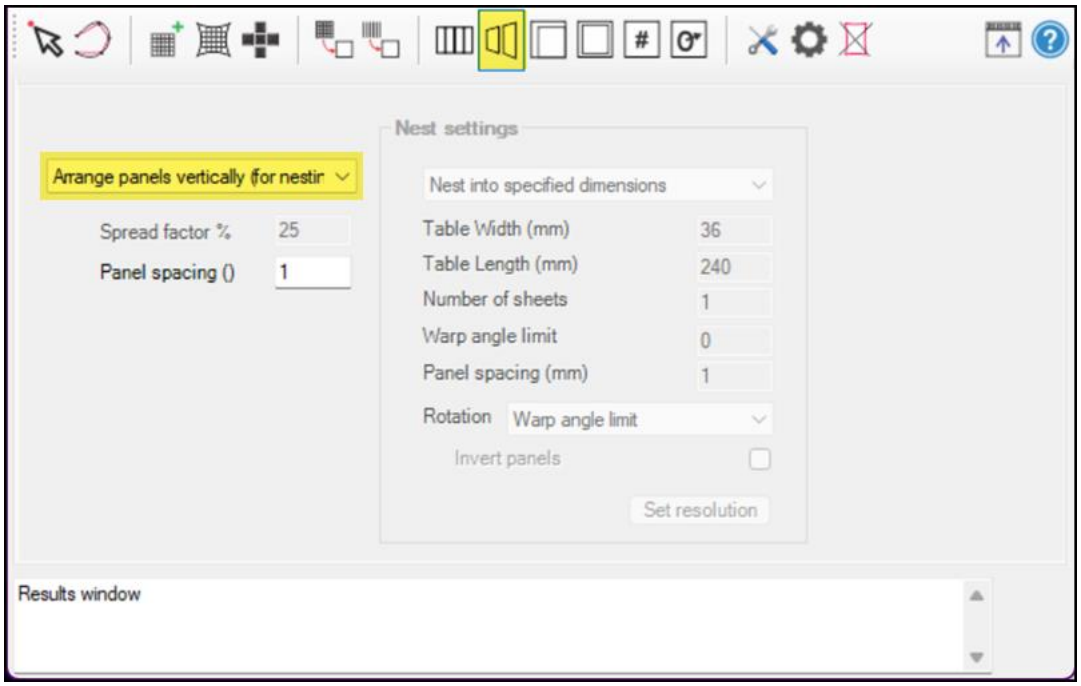
- **Arrange panels horizontally**

This takes a set of panels, orients each panel vertically, and arranges them in a horizontal row in the order of their internal ID number. To orient the panel the program will rotate each panel, but is not allowed to mirror a panel.



- **Arrange panels vertically**

This takes a set of panels, orients each panel horizontally, and arranges them in a vertical column in the order of their internal ID number. To orient the panel the program will rotate each panel, but is not allowed to mirror a panel. This is used a the starting position for nesting.



- **Nest Panels**

MPanel offers a tightly integrated interface to a powerful nesting library, supplied by Nesting Technologies SRL.
The nesting library is an optional add in to MPanel, and a separate license for it's use must be purchased from MPanel Software Solutions.
The nesting is described more fully here: Nesting

Nesting

MPanel offers a tightly integrated interface to a powerful nesting library, supplied by Nesting Technologies SRL.
The nesting function is an optional add in to MPanel, and a separate license for it's use must be purchased from MPanel Software Solutions.

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.

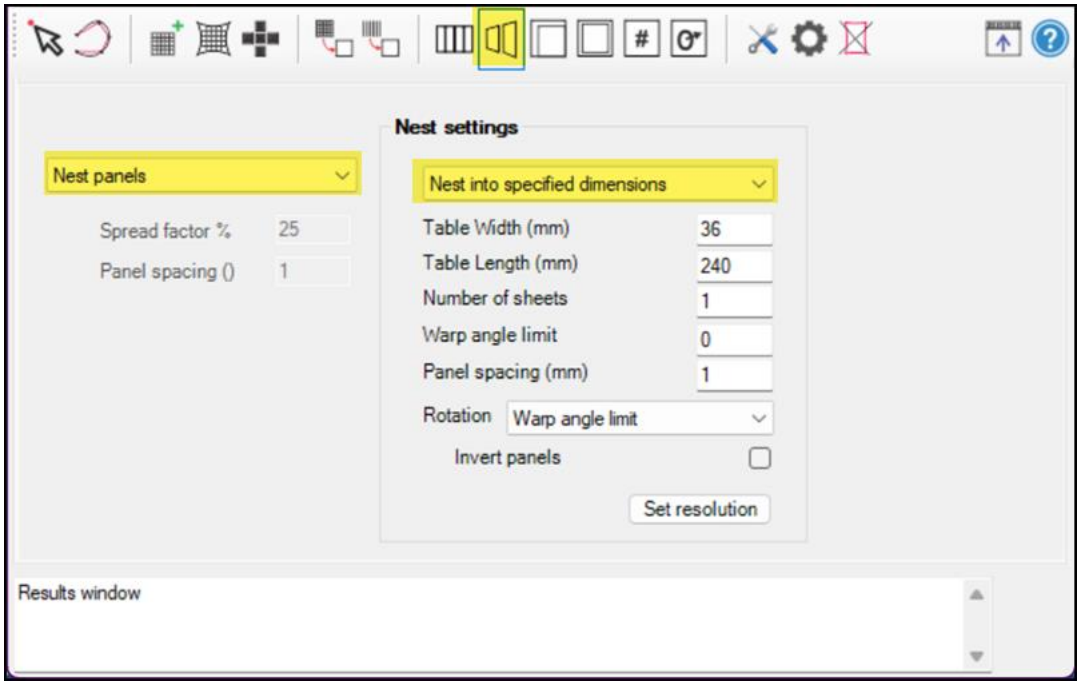
The number of sheets can be specified, and as many panels as can be nested onto those sheets will be used. Or you can specify a large number of sheets (100) and then all the panels will be nested.

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.
In addition you can also specify that the panel can be rotated by 180 degrees, and then by values up to the warp limit. This is the usual setting for fabrics that don't show a "nap" so they can be rotated by 180 with no visual effects.

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.

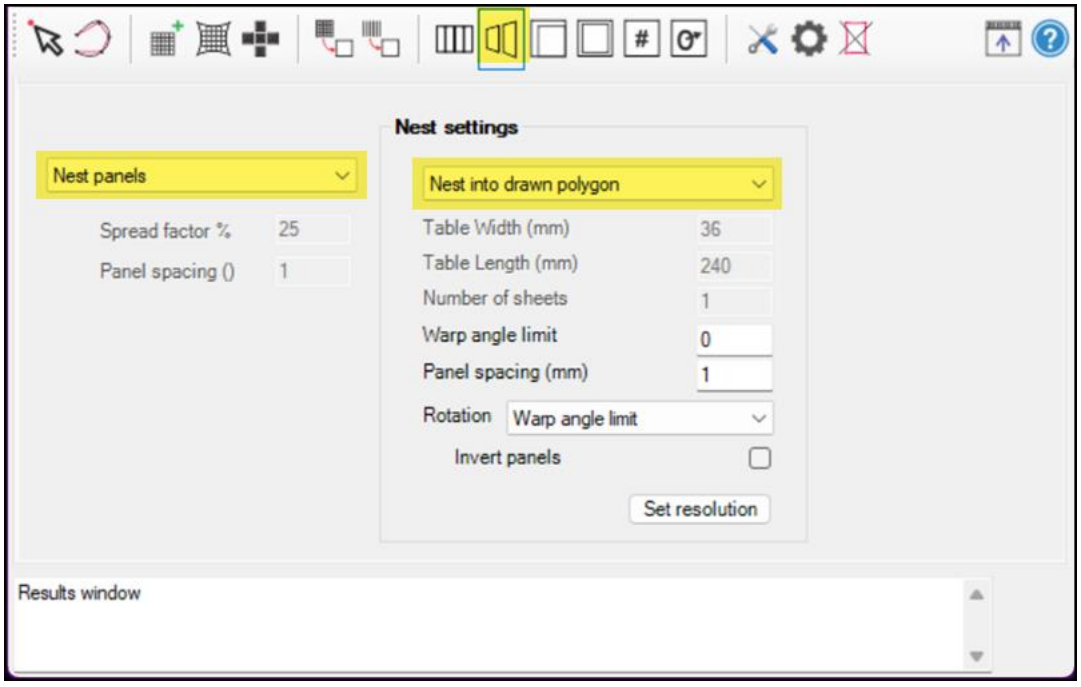


Nesting into a drawn polygon

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.



Invert Panels.
Panels can be inverted at the same time as being nested by selecting the invert panels option. Panels can also be inverted as art of the Output tool however, if panels were inverted during nesting the invert function during panel output is not allowed.

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%
Utilisation 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, -10, +20, -20 and the same angles 180 degrees around. A smaller angular resolution will take longer to nest. A reasonable value to use is 50% of the warp angle limit.

(If Allow180 rotation is selected then because the warp angle allows included the same angles 180 degrees around, there is no sense in specifying a warp angle limit greater than 90 degrees. To allow for full 360 degree rotation specify 90 degree warp angle limit, and a angular resolution of 90, 45, 22, or 10 degrees)

Optimization 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 Optimization level.

Nesting can be slow, a cancel button will show during the process.

Nesting License.

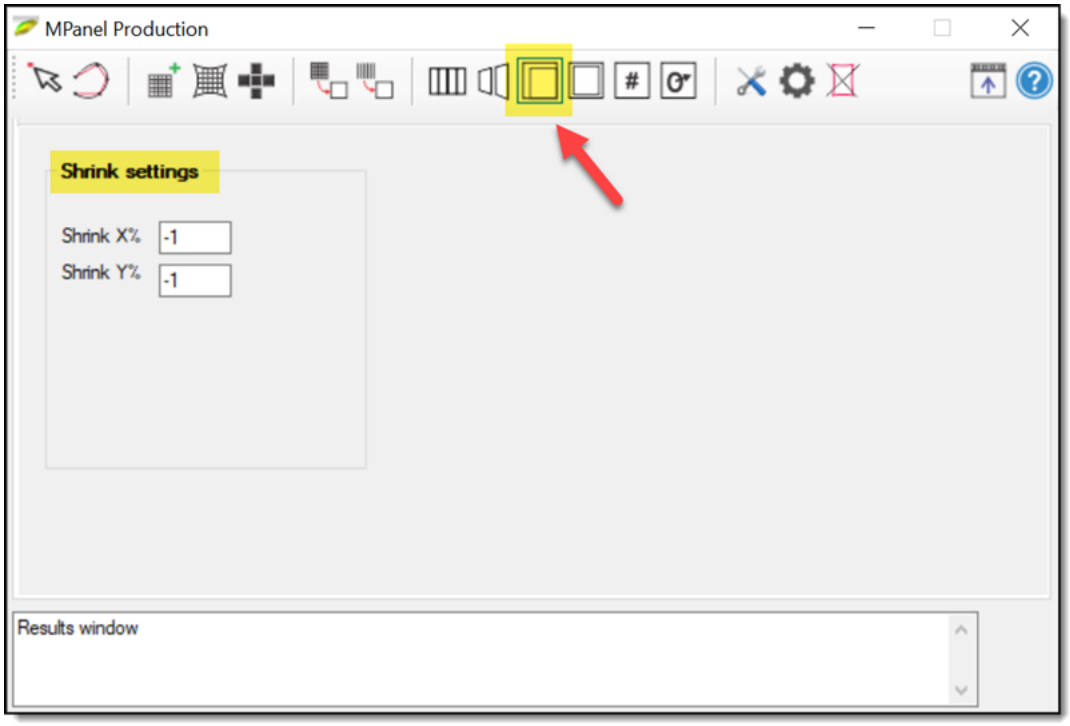
The nesting function requires an additional on the MPanel license. Once this has been added the first nest that is attempted will be interrupted to set the nest license on that computer.
To license the nesting, contact MPanel Software Solutions.

Shrink panels

Panels are shrunk so that when they are fitted to a frame the fabric is taught. To shrink some panels, select them and click the "Shrink panels" button. The amount of panel shrink needed depends on the stretch of the fabric, some approximate guidelines are:

Lycra 5%
Knitted HDPE (shade cloth) 3% Y 1% X
Canvas 1.5%
PVC coated polyester architectural fabric 0.5%

In some situations the fabric needs to fit loosely over a frame. Then you use a negative shrink the get panels that are a little too large.



Video tutorial: **Panel Handling - shrink**

Seam Panels

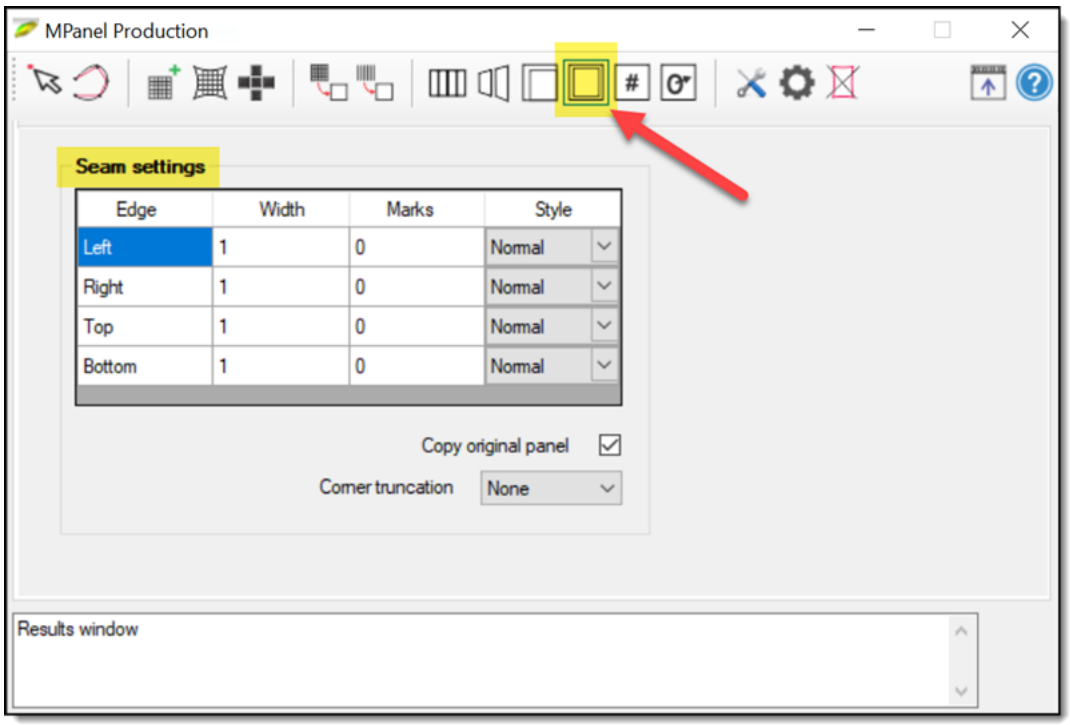
Usually a seam allowance is needed on each edge, to make a hem, attach to reinforcing, or attach to the next panel. To add a seam allowance some panels, select them and click the "Seam panels" button The width of the seam allowance is specified in the table. If no seam allowance is needed on an edge specify the width as 0 (zero).

When sewing panels together it can be useful to have "tick marks" that line up from one panel to the next. A number of marks can be specified along each edge.

If you want a copy of the original panel including with the panel (to provide a sew to or hem to line) then tick the box that specified this. If seam allowance is set to 0 or a negative value then the copy of the original edge will be suppressed.

The seam style should usually be set to "Normal". There is an option for "Mirror" seams, that makes a seam that, when folded back into it's correct position, will match the panel edges exactly. If mirror seams are required they are usually only specified on one or two sides of a panel, and never on adjacent sides.

Corner truncation can be specified, which can help with panels with acute corners. Simple truncation will truncate the corner with a straight line, If the panels will be sewn together back to back it is useful to truncate with an arc, as this will allow the comer to line up exactly when they are positioned for sewing.



Video tutorial: **Panel Handling - seams**

Video tutorial: **Corner truncation**

Video tutorial: **Mirror seams**

Stamp Panels

- **Panel stamp.... Add panel number**

A stamp can be added to each panel, to identity the panel during production. To stamp panels, select them and click the "Stamp panels" button. The stamp can be placed:
 a) In the seam allowance on the bottom edge
 b) In the panel along the center line
 c) In a fabric tag on the bottom edge
Note: If a panel is created from a 2-sided mesh it will have a left and right edge only so MPanel will automatically add the stamp to the right side of the panel.

The stamp consists of the text specified, where # is replaced with a number. The number can either be:
 a) the internal panel number, as shown in the middle of each panel.
 b) a number specified by the user, which is incremented for each panel.
The size of the stamp is controlled by specifying the text height. Note that Rhino templates can include text scaling which can make it difficult to control text height so it is generally good practice to set scaling to 0 - this is shown in the later portion of this short video:

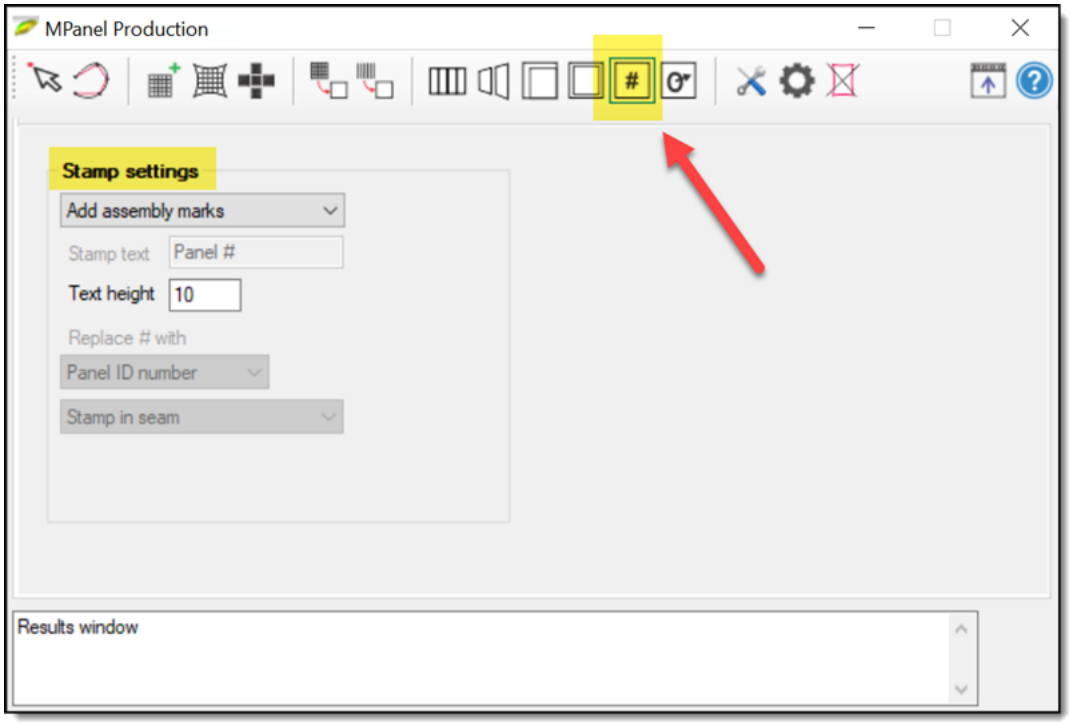
Video tutorial: **Panel stamp Vector font and text scaling**

- **Panel stamp.... Draw assembly lines**

With this tool a set of panels will have lines drawn between them to indicate which panels join together.
Lay panels are laid out approximately as they would be assembled, with a small gap between each panel. Then select the panel and run the tool. A group of assembly lines will be drawn between the panels.
With some layouts the lines will be incorrect, or some will be missing. Lines can be edited, deleted, or added as needed.

- **Panel stamp.... Add assembly marks**

With this tool a the assembly lines are replaced by assembly marks in the panel seams. The assembly marks consist on a letter "m" followed by the panel ID numbers. During panel assembly placing the "m" symbols back to back will get the correct alignment of the panels.
The size of the assembly marks is controlled by specifying the text height.



Video tutorial: **Panel handling - stamp**

Panel Output

Panel output changes the panel into a form that is suitable for your cutter, plotter, or manual cutting measurements. It strips out the panel number, but keeps the panel stamp if there is one. To output the panels select them, and click on the "Out panels" button. You can output the panel to:

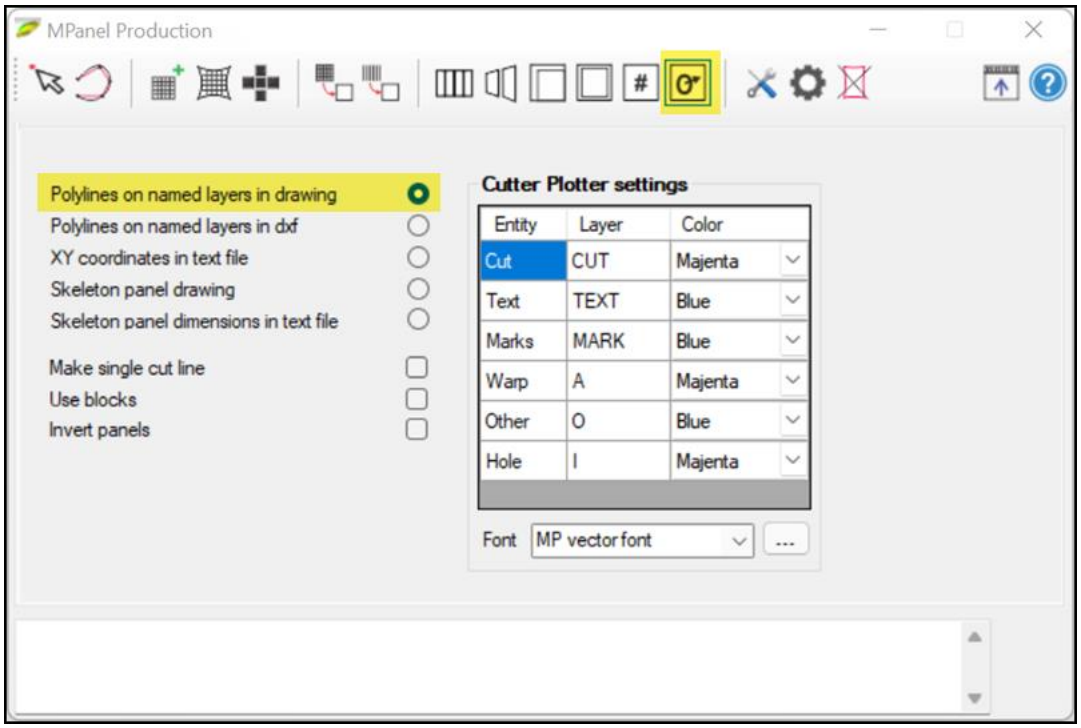
- **Polylines in the drawing**

The cut, text, marks and other lines are separated into specified layers and colors.
If your plotter / cutter uses layers to identify the cut curve, then the color can be left as "Bylayer" .
If your plotter / cutter uses colors to identify the cut curve, then specify the color in the table.

To suppress output of an entity, such as text, specify the layer as blank ("")

Optionally you can specify that the cut line is a single continuous polyline.
Optionally you can specify that each panel is output as a block (otherwise each panel is output as a group).
Optionally you can specify that each panel is inverted (printing on the bottom of the fabric).

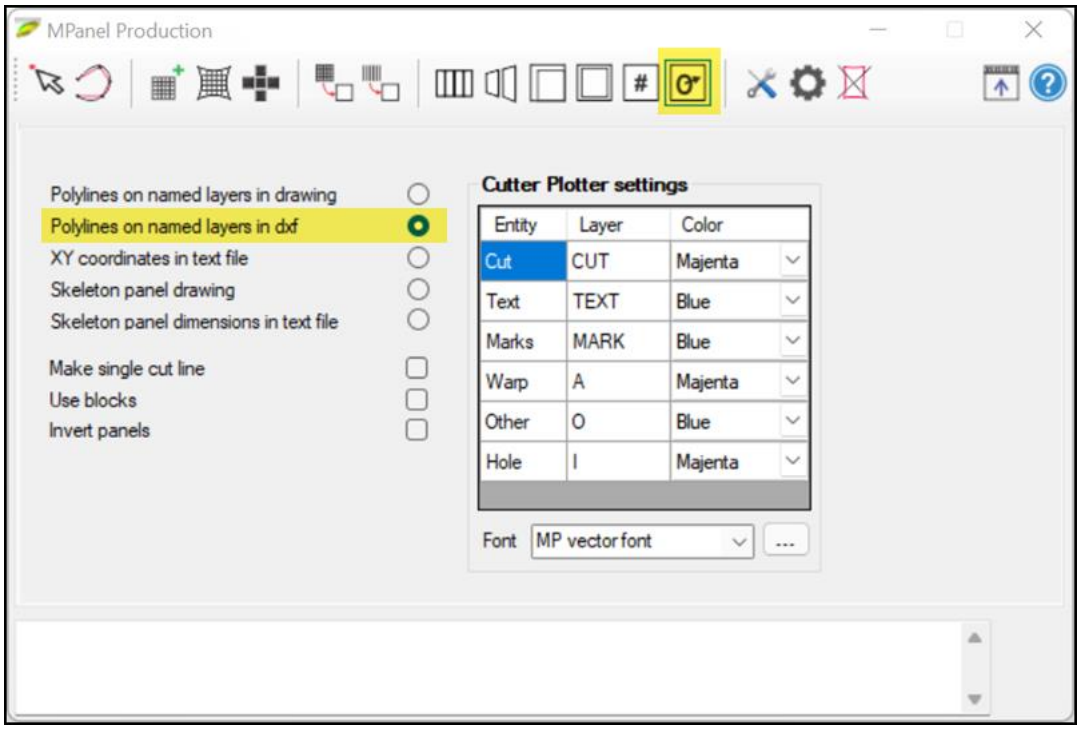
A font for the stamp text can be specified. Simple line fonts will plot faster than solid fonts.
Some cutters are slow when drawing text. They may draw faster if you specify "MP vector font". This substitutes polylines for the text outlines.
Once the Output panel has been drawn, it can be sent to the cutter or plotter using Rhino export options, in several different formats.



Video tutorial: **Panel output - normal**

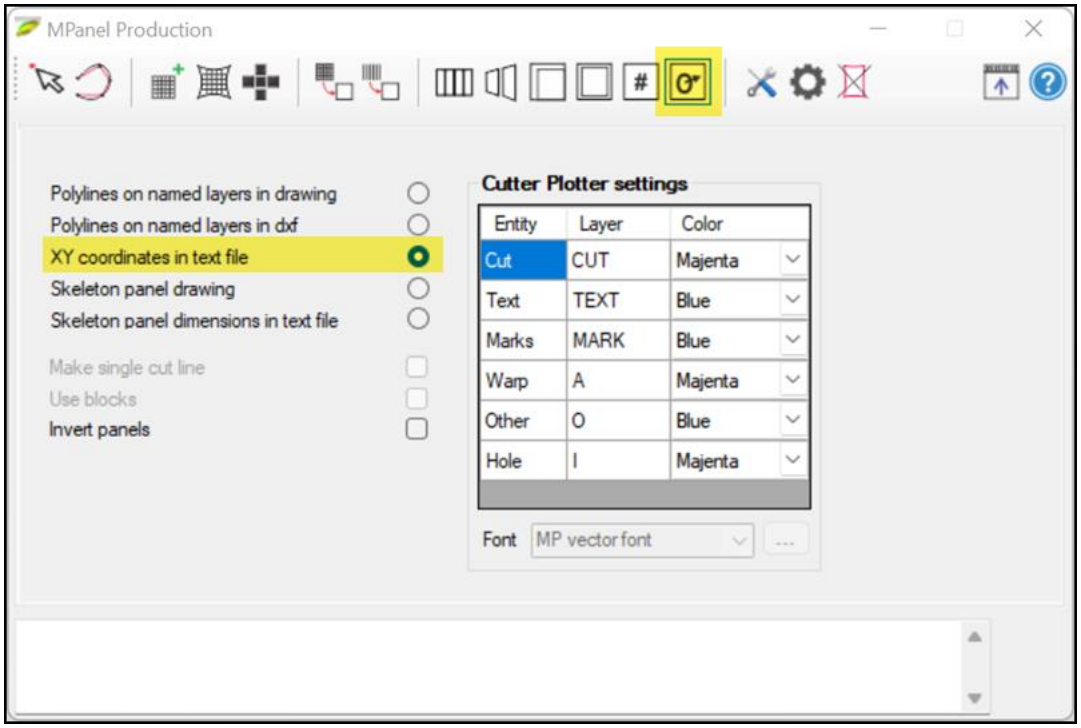
- **Polylines in DXF**

The same output as above, but sent directly to a dxf file. With a dxf it is usually best not to specify the font, so the cutter / plotter can use it's default font.



- **XY coordinates in text file.**

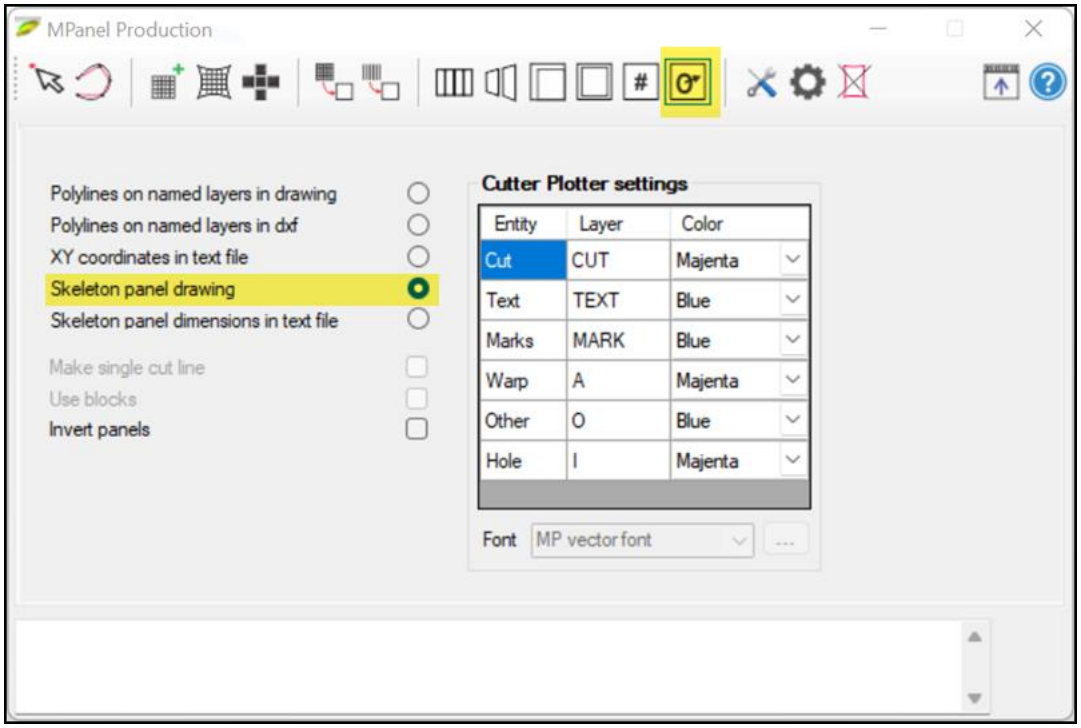
The bottom left corner of the panel is referenced as 0,0, and the polylines node XY coordinates are output as XY coordinates to a text file. These can then be plotted out manually.



Video tutorial: **Panel output - XY coordinates**

- **Skeleton panel in drawing**

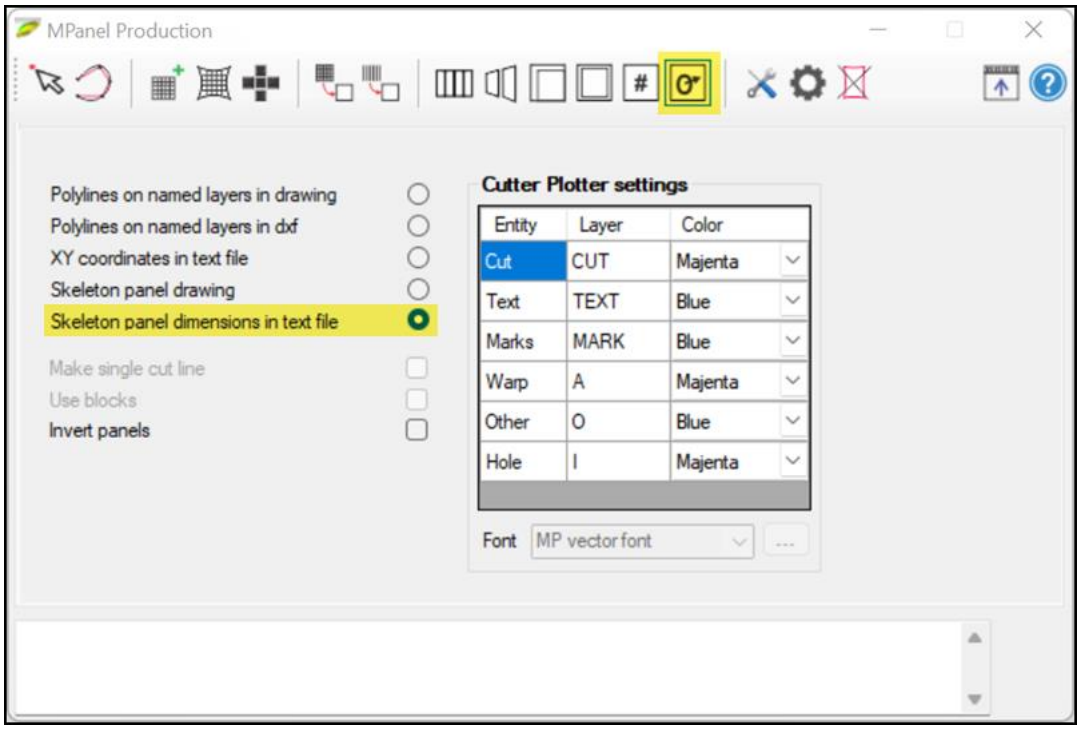
A skeleton drawing, which has corner to corner measurements and edge midpoint dips, is drawn on the screen. This used when the panel is laid out by hand, with a bendy straight edge. It is also used for checking panels that have been produced by other methods.



Video tutorial: **Panel output - skeleton**

- **Skeleton panel in text file.**

Contains all the skeleton measurements for each panel, in a text file.



Invert Panels.
Panels can be inverted at the same time as being output by selecting the invert panels option. Panels can also be inverted as art of the Nesting tool however, if panels were inverted during nesting the invert function during panel output is not allowed.

Utility Tools

Utility tools consist of 3 sets of tools that can help solve drawing problems.

Modify polylines
Modify Meshes
Modify Panels
Fix Rhino 5 entities

- **Modify polylines**

Modify polylines has these tools:

Join polys
Will join any number of polylines together. The direction of the joined polylines is determined by the direction of the first polyline selected. Small gaps between the polys will generate a warning, and can then be jumped over.

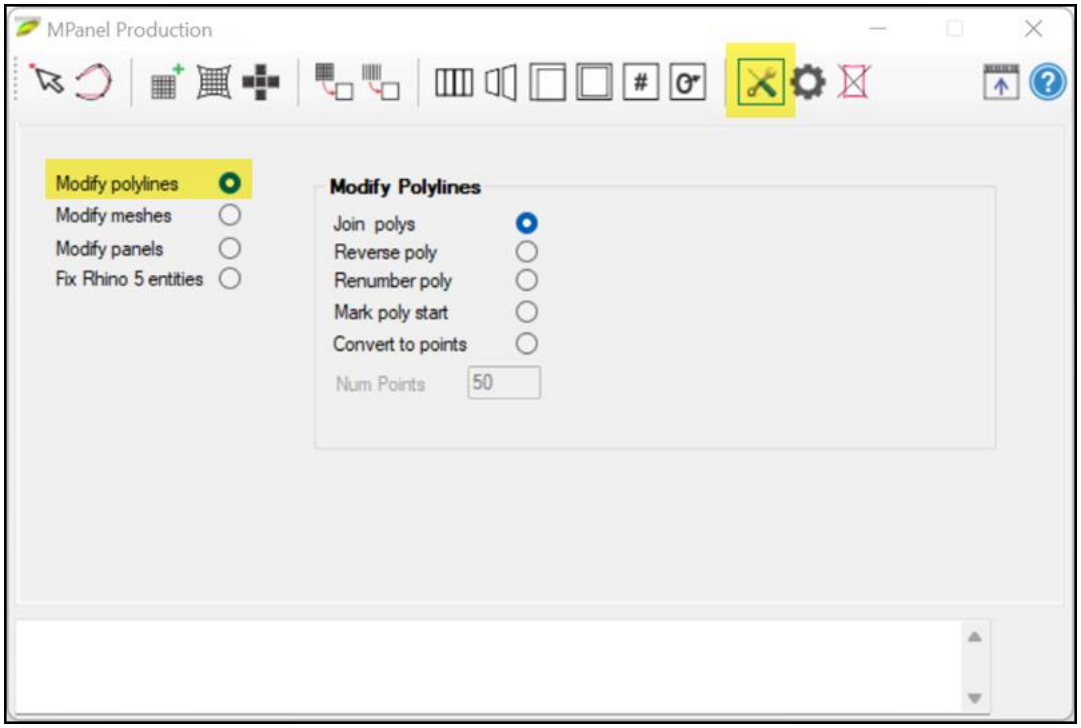
Reverse poly
Will reverse the direction of a polyline.

Renumber poly
Will redraw the selected polys with the specified number of points. This works well for a smoothly curved polyline. If the original polyline has sharp corners then the corners will be missed during the renumbering.

Join dots into poly
Will join together a number of points into the best fit poly, based on a simple rule. This is used when input data from a digitiser is presented as a point cloud, rather than a polyline.

Mark poly start.
Draws an arrow to mark the poly start. Can help with understanding paneling problems when some polys are reversed.

Convert poly to points.
Converts a polyline into the specified number of points which can be used to add details to panels.



Video tutorial: **Utilities - polylines**

Video tutorial: **Convert Poly to Points**

- **Modify Meshes**

Color mark edges.
Shows which are the top, bottom, left and right edges of a mesh. Helps in understanding how the mesh shape factor is applied, and what the default paneling direction will be.

Swap weft and warp.
Swaps the mesh orientation, so that the top-bottom becomes the left-right

Swap top and bottom
Changes the mesh orientation, so that the top and bottom are swapped.

Swap left and right
Changes the mesh orientation, so that the left and right are swapped.

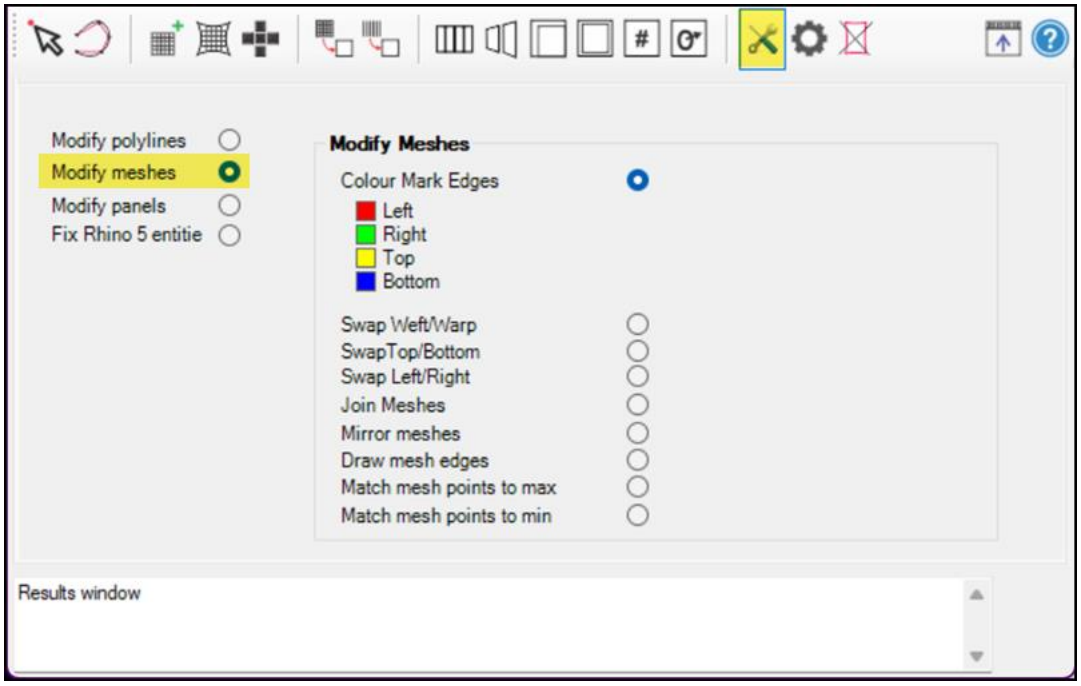
Join meshes
Join two meshes together, if their corners are joined and they have the same number of mesh threads on the joined edges. From version 9, 2 meshes with different number of mesh threads can be joined however, it should be noted mesh interpolation may cause small changes in the surface shape.

Mirror meshes
This takes selected meshes (and any curves/lines etc) and mirrors it/them, keeping the correct mesh orientation. Note… Rhino fails to mirror a mesh correctly with their inbuilt mirror tool.

Draw mesh edges
This takes selected meshes and adds an edge poly to each edge with the same number of points

(vertices) as the to match the mesh. This is useful when making shade sails using the cable edge option in Adjust meshes tool. In the case that one or more edges are fixed (e.g. keder edge) then a mesh edge poly should be added to the fixed edges. The mesh and the fixed edge polys should be selected before running the Adjust meshes tool to relax the sail.

Match mesh points
This takes selected joining meshes and matches the mesh density either the lowest or highest depending on which option is selected. To use this tool, select 2 or more joining meshes and run the tool and delete old.



Video tutorial: **Utilities - meshing**

- **Modify Panels**

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 detail is added to the panel in a Gray color. The details are correctly preserved through shrinking, seaming, stamping and output.

Add mark to panel
Add Mark to a Panel allows marks made from polylines to be grouped with the panel. Select the panel and the required detail, and run the tool. The detail is added to the panel in a Orange color. The mark will be converted to a seam mark during the seam process, with the seam mark added to the nearest panel edge, based on the middle node in the marked panel.

Add hole to panel for windows.
Add hole to panel tool simplifies the process of adding clear window panels to boat and spray covers and should be used in conjunction with the "Split panels with hole" tool (see below). The hole is specified by a 4 sided polyline drawn inside the panel using the same procedure as adding detail. Select the panel and the new polyline and run the Add detail tool to add the detail to the panel.

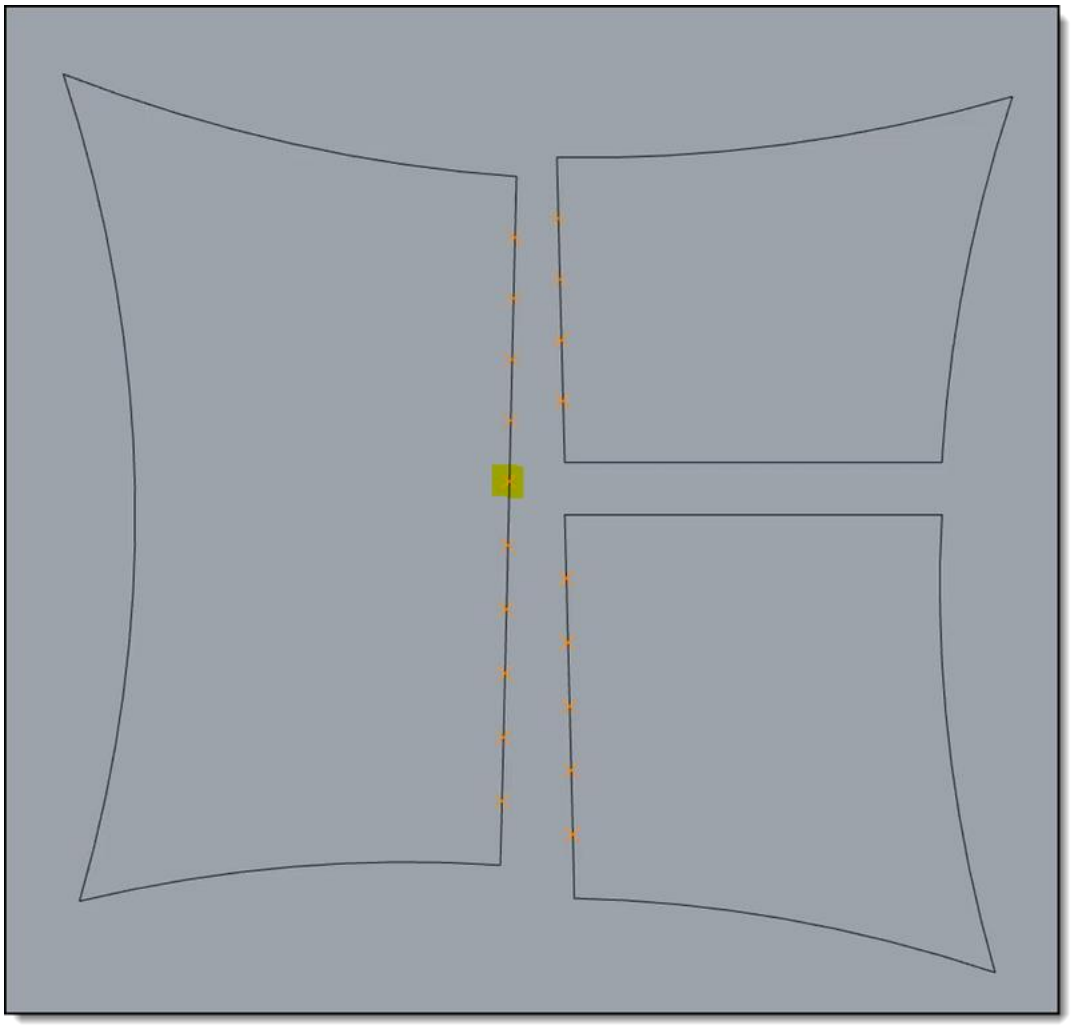
Video tutorial: **Window Hole in Panel**

Adjacent panel marks

When 2 panels join to a single panel along an edge, it is useful to have a mark on the single panel edge to show where the other 2 panels connect. To do this color the main single panel Green, position the other two panels in approximately the correct position, and run the tool. A mark will be added to the main panel.

In addition, if a mark spacing is specified then additional marks will be added to the panels that will line up correctly.

The procedure to operate this tool is shown in the following video.



Video tutorial: **Adjacent marks**

Color mark panel edges
This draws a set of colored polylines inside the panel, to identify which panel edges are top, bottom, left, right, using the same color code as the mesh edges (blue = bottom, yellow = top, green = right, red = left)

Make panel copies.
This makes a copy of the panel, as a new panel with a different panel number. This can be helpful when nesting several copies of the same panel.

Swap panel warp and weft
This makes a copy of the panel, with top_bottom swapped with left-right. This is useful if the original panel was created the wrong way around. The number of nodes on the panel edges are adjusted to meet panel rules.

Mirror panel
This makes a copy of the panel mirrored left-right.

Split panel
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. This tool can be used for splitting panels into colored segments, for artwork purposes. The number of nodes on the each panel edge are adjusted to meet panel rules

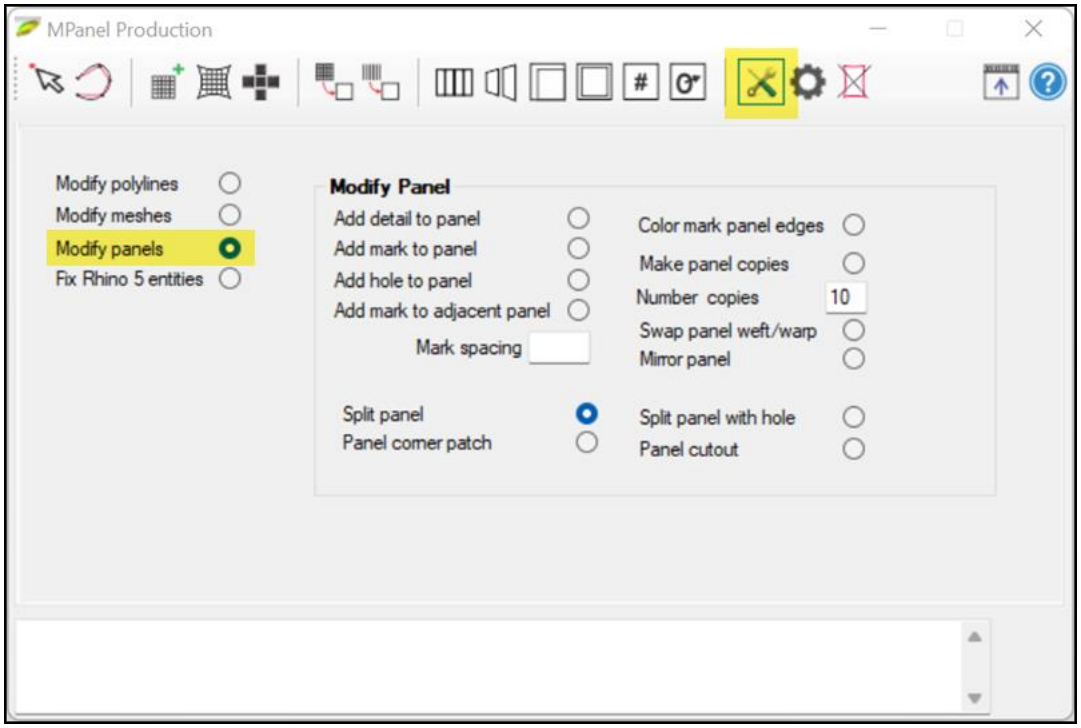
Panel corner patch
A new panel can be defined by a polyline that crosses a corner of the original panel. The new panel always has a left, right, and top edge. This tool can be used to make reinforcement patches in panel corners. The number of nodes on the each panel edge are adjusted to meet panel rules.

Split panel with hole
A panel with a hole added using the "Add hole to panel" tool (above) can be split into 5 parts defined by the polyline that represents the hole (window) which is one part, and the 4 smaller surrounding parts of the fabric frame. The new panels maintain the orientation of the original panels. The orientation of the 4 frame elements can be changed by swapping the panel weft/warp prior to splitting the panel.

Panel cutout
A panel can have a cutout added on any edge, or across any corner. The cutout is specified by a polyline drawn over the panel. Cutouts are used to allow for protrusions, covers, rope pathways, etc on the panel edges that we did not want to include in the modeling.

Because the cutout can be an arbitrary shape, and adds an arbitrary number of nodes to the panel edge, it is not possible to shrink or seam a panel with cutouts, so the cutout has to be applied after seaming. Cutouts can be applied before or after stamping. Multiple cutouts can be applied by using this tool several times.

If no separate cutout poly is supplied then this tool will try to use any panel detail polys as the cutout poly. The panel detail polys can be created from points in the fabric surface mesh, (described here) so this gives a method to transfer measured data points into panel cutouts.



Video tutorial: **Modify Panel**
Video tutorial: **Panel cutout**

- **Fix Rhino 5 entities**

When a MPanel drawing is made in Rhino 5, and then opened in Rhino 6, some entities may incorrectly have transparency applied. This tool forces the entities to be non transparent.

Settings

General setting that affect the overall operation of MPanel.

New Object Layer

Specifies the layer that new object will be drawn on, either the Rhino current layer or a named layer.

Selection Filter

Filters the selection set by a layer name, which can make it easier to select objects in a complex drawing.

New Object Color

Specifies the color of new objects that Mpanel draws as:

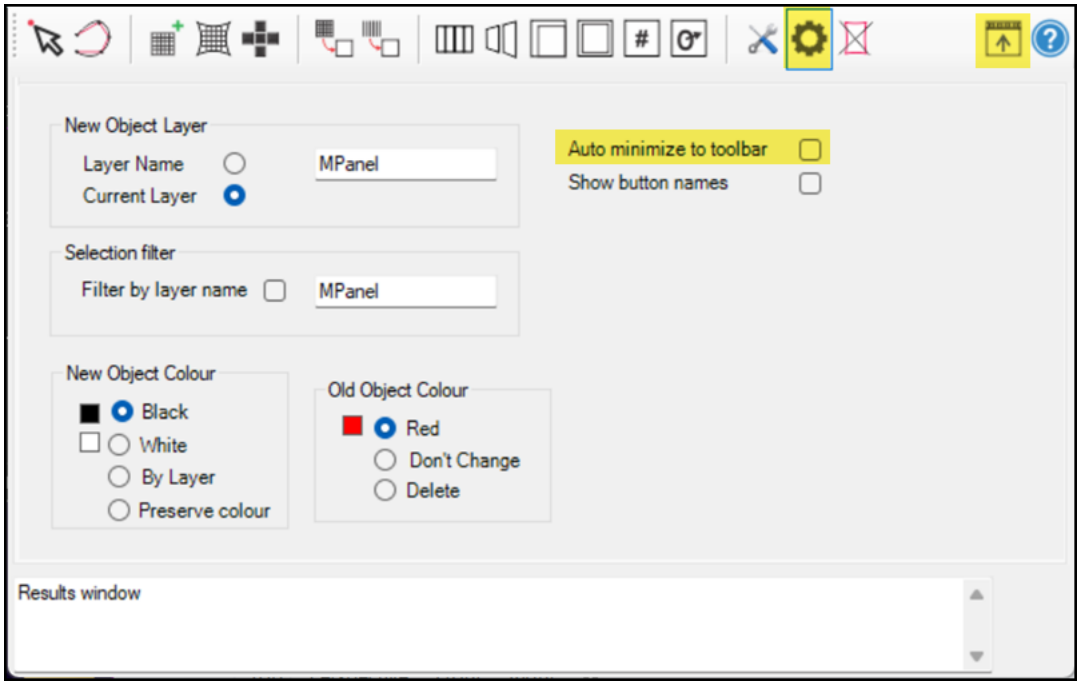
Black	
White	
By layer	(will show as the color specified for the layer the object is drawn on)
Preserve color	(keeps the same color as the object that it was derived from)

Old Object Color

Takes 2 selected polys which have a gap between them, and extends each poly to join at an extended intersection. This is useful when two frame members join in an acute corner, and it wasn't possible to machine digitize all the way into the corner due to the measuring probe size.

Auto minimize to toolbar

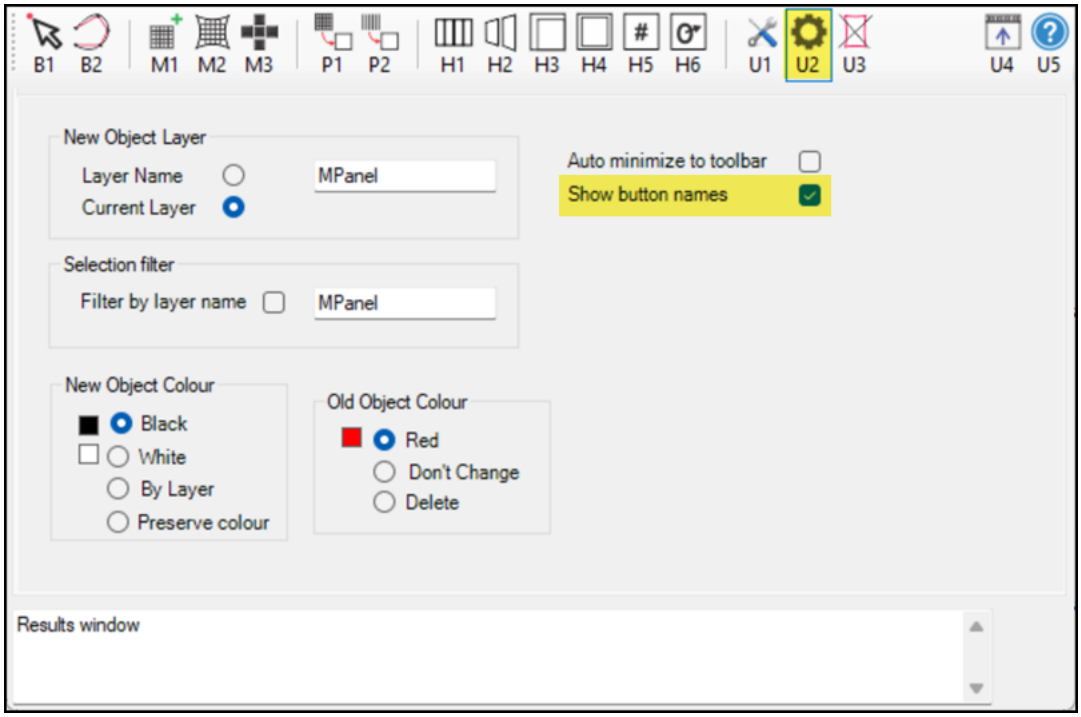
If selected the open/close toolbar buttons will not be shown and the options window will auto-hide when the cursor is moved away from the toolbar.



Video tutorial: **Settings**

Show button names

If selected the toolbar ribbon buttons will include a letter and number e.g. U1, U2 etc which will help identify buttons during training sessions.



Advanced

MPanel Production configuration settings are saved in a text file called "MPanelProd.ini" which is located in:

C:\Users\YOURUSERNAME\AppData\Roaming\MPanel (YOURUSERNAME should be replaced with your username name)

This windows directory is typically hidden by default so you will need to select "show hidden" in file explorer in order to access the file.

Move to a new computer

This file can be copied and moved if you are moving your MPanel software to a new computer.

Reset Units

The file can also be edited using a simple text editor (such as notepad) to reset measurement units:

At the first run of MPanel Production you will have selected the measurement units you wish to use. This selection is written to the ini file. If you decide to change units you would edit the ini file - locate the units heading (below example) then delete the setting and save the ini file. Next time you start MPanel Production you will once again be prompted to select units.

[UNITS]
ComboUnits=3

Reset Next Panel Number

The next panel number can be changed in Rhino, using Options… Document Properties… Document User Text. It is good practice to remove all previous panels if you reset the next panel number to avoid potential confusion when reviewing work in future.

Notes for advanced users:

DXF Output file version
The predefined DXF Output file version can also be tailored by editing the "MPanelProd.ini". To add this option to an existing ini file simply copy and paste the following 2 lines at the bottom of the file and save.

[DXF]
Version = 14

By default the DXF Output file version = 14.

Alternative values are:
version 14,15,18, 21, 24, 27 or -1 for legacy method (only use this if your plotter/cutter no longer operates as it did prior to updating MPanel to v25.1 or later).

If you prefer not to edit the MPanel Production User Setting.ini file a new version of the file with these options already included can be obtained from C:\ProgramData\MPanel Production\Archive - this can be copied to Documents\MPanel and replace old file.

Contact

Technical support is available to customers who:

- 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.

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