CUSTOMER SUPPORT NOTE

Using Surface Elements in FE Analysis

Note Number:	**CSN/LUSAS/1027**

This support note is issued as a guideline only.

Table of Contents

1. Introduction

This support note provides guidelines on the choice of surface element type, mesh density, application and the visualisation and interpretation of the results.

2. Description

2.1 Choice of mesh type

The two mesh options for the modelling of structural slabs and steel plates are '**shell**' and '**plate**' elements. Each of these gives the option of either '**thick**' or '**thin**' formulation, '**quadrilateral**' or '**triangular**' element shape and '**linear**' or '**quadratic**' interpolation order (quadratic is only available with the **Plus** software option).

2.1.1 Shell Vs Plate Elements

The difference between shell and plate elements is that shells carry in-plane (membrane) as well as out-of-plane (bending/shear) forces. The result of this is that plate elements are only suitable for flat (two-dimensional) models with loads applied vertically. If there is any 'height' to the modelling, shell elements must be used, even if the height is only an eccentricity of the surface geometry. Shells can therefore be used in a greater number of situations, but plates are computationally less demanding due to reduced number of freedoms in comparison with shells.

Some examples of application:

Plates
- Flat slabs supported directly

Shells
- Flat slabs supported on other geometry (e.g. columns)
- Slabs curved or angled in elevation (e.g. crossfall)
- Stiffened slabs (e.g. under-slung beams)
- Walls
- Box structures (e.g. box culverts, box girders)
- Steel sections modelled as their component plates

2.1.2   Thick Vs Thin Elements

Shells and plates are each available as either 'thick' or 'thin'. Thick shells and plates are formulated to include shear deformation and also will provide shear results. Thin shells and plates do not include shear deformation and will not output shear results.

Although the thick or thin terms are not specifically intended to represent 'thick' or 'thin' structural components, thick elements are generally more suitable for structural features such as concrete slabs where shear strains are likely to constitute a more significant proportion of the deformation. Thin elements are better suited to structural components in which the response is mainly due to flexural or in-plane strains, such as the steel plates making up an I-Beam. In most cases thick elements can be used for both cases.

In a slab with a span-to-depth ratio of 10, shear deformations contribute approximately 3% of the deflection. As the span-to-depth ratio increases, a smaller percentage of the total deflection becomes due to shear deformation. If shear results are needed from your analysis, then thick elements must be used.

2.1.3   Triangular Vs Quadrilateral

A quadrilateral mesh will generally give better results than a triangular mesh due to the higher order variation of forces and moments across the elements. This can be verified from the Notes section of the LUSAS Element Reference Manual.

Triangles should be used sparingly in small areas of a model only if they result in a better meshing pattern due to the unusual shape of a surface. Triangular elements may sometimes be used as transition elements between quadrilateral elements of different size.

The mesh in a model can be adjusted using line divisions (see the example in Appendix A) to remove triangles wherever possible. Using element size is another good method for adjusting a surface mesh density.

2.1.4   Linear Vs Quadratic

Linear thick shell elements (QTS4) do not simulate the change in bending moment across a structure as well as linear thin shell elements (QSI4), quadratic thick shell elements (QTS8) or plate elements. Therefore if QTS4 elements are used, a more refined mesh will be needed compared to the other elements listed above. The use of quadratic interpolation order is generally recommended if it is available. Quadratic interpolation elements can be used with TLO (Traffic Load Optimisation, for example used for Eurocode) but cannot be used with Autoloader so for the latter case linear elements must be used.

Summary

Sensible default mesh choices for bridge analysis are thick shell – quadrilateral – quadratic (QTS8) for concrete slabs and walls, and thin shell – quadrilateral – quadratic (QSI8) for linear steel plates.

However, care will need to be taken when shells are connected to beams – compatibility between these elements should be accounted for.

2.2 Mesh Application

The results achieved from a surface element analysis not only depend on the type of elements chosen, but also on the size, shape and distribution of the elements. A fine mesh (smaller elements) will better approximate the model, but will be slower to run than a coarse mesh (larger elements). The mesh density can be controlled in several ways:

1. By changing the default number of mesh divisions. This is altered by clicking '**File > Model Properties > Meshing (Tab) > Default Line Divisions**'. By default this number is set to 4. This will set the number of divisions for any line which has not been altered by either of the methods explained below.

2. By setting a size for the surface mesh elements (see Figure 1). This will override the default mesh divisions.
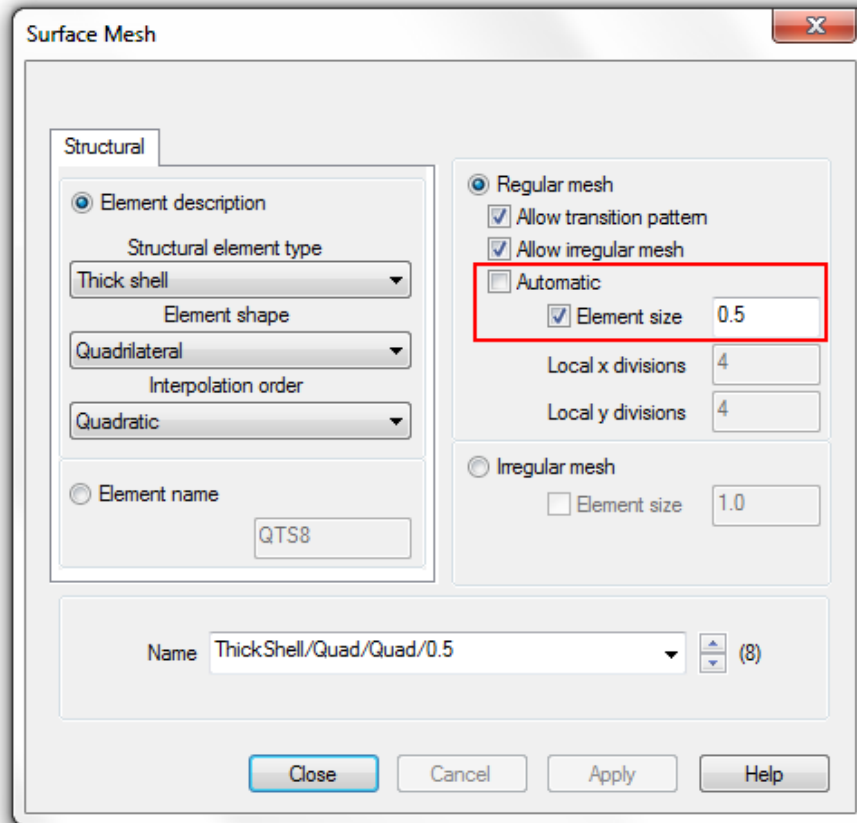
Figure 1 - Controlling density by specifying element size

3. By applying NULL line meshes (line meshes with no mesh type selected) to the edge lines of a surface. The number of divisions of the NULL mesh will determine how many elements that side of the surface has. Element spacing can also be controlled by clicking the 'Spacing' button in the line mesh properties if a non-constant spacing is required along any side of the surface. The NULL mesh will override either of the previous two methods.

Once the mesh has been applied to the model, it is important to check its shape. The mesh can be 'improved by adjusting the spacing as described above. Ideally, the elements should have aspect ratios as close to 1 as possible, and certainly less than 10 (i.e. approximate squares are preferable to elongated rectangles). Small-angled (sharp) element corners should also be avoided where possible.

If the number of mesh divisions on opposite sides of a surface is equal, then a grid (regular mesh) may be achieved. Otherwise, transition patterns (or an irregular mesh) will be generated. Surfaces with more than four sides can be meshed with a regular mesh by making 'combined lines' from some of the perimeter lines as shown in Figure 2. Note that to

achieve a regular mesh on the image in the right, a combined line has been defined using the two bottom lines of the surface.
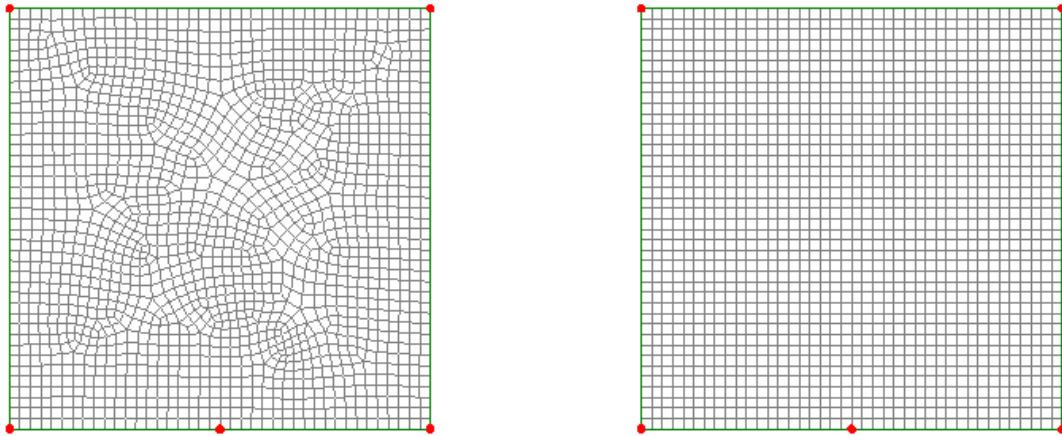

Figure 2 – 5-sided surfaces meshed with irregular mesh (left) and regular mesh using combined line (right)

It is also important to ensure that the surface z-axes are consistent. Otherwise, the local shell results will vary from positive to negative between surfaces. Surface axes can be viewed by double-clicking the geometry layer and selecting 'view surface axes'. Inconsistent surface axes can be flipped over by clicking 'Geometry > Surface > Reverse'. Alternatively, all surfaces can be made consistent by selecting one correctly-oriented surface, then using 'CTRL-A' to select the entire model and clicking 'Geometry > Surface > Cycle relative'.
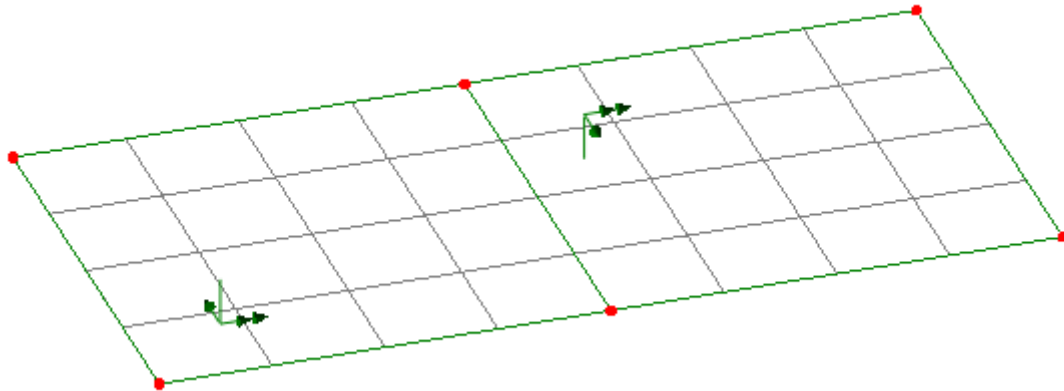

Figure 3 - Incorrectly oriented surface axes

2.3 Viewing the results

It must be noted that in plate and shell elements, forces and moments are provided 'per unit width'. On section 2.3.2 of this document a tool which allows obtaining resultant values - similar to the standard beam convention - is presented. It should also be kept in mind that shell elements forces and moments are expressed in the elements' local axes. Note that

shell stress Top/Middle/Bottom are in the global directions. These points are discussed below.

2.3.1   Contour Plots

Contour plots are generally the easiest way to check results for surfaces. They provide a quick and clear visual representation of the distribution of forces and moments in the component. However, they can be misleading if not properly understood.

By default contour plots are 'smoothed'. This averages results at each node and interpolates between them to provide a clear, tidy appearance. By averaging, however, peak values can be reduced, leading to non-conservative values being presented. This is especially pronounced when too coarse a mesh has been used. It is recommended that values for design are taken from nodal unaveraged results, i.e. the unsmoothed contour plot.

Figure 4 shows the same results plotted smoothed and unsmoothed. A contour plot like the one on the right is a clear indicator that the mesh is too coarse and a rerun with a better mesh is required, but this can be hidden by the smoothing.
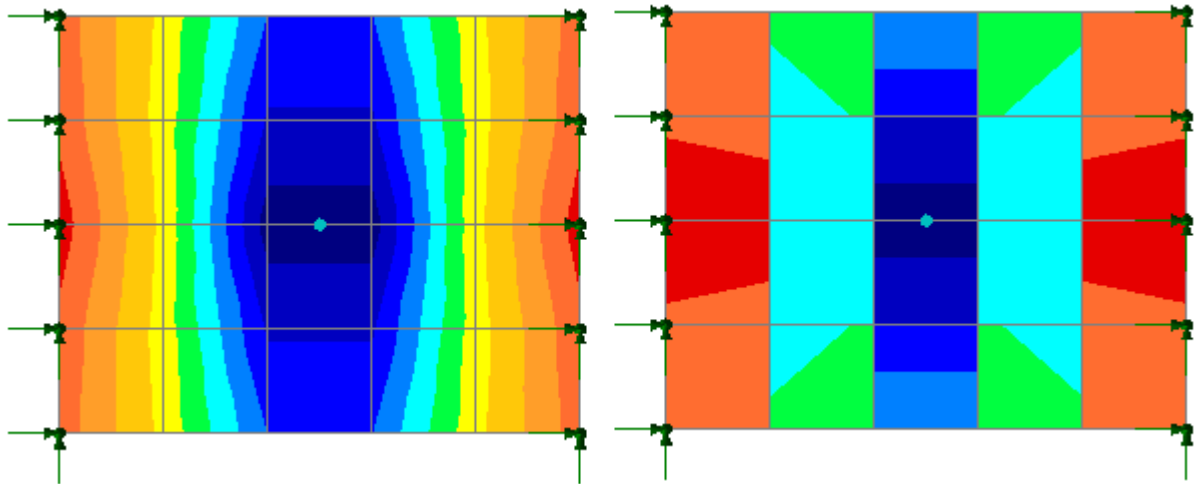

Figure 4 - The same results plotted smoothed (left) and unsmoothed (right)

Figure 5 shows the importance of mesh refinement. The two structures are identical apart from the mesh.
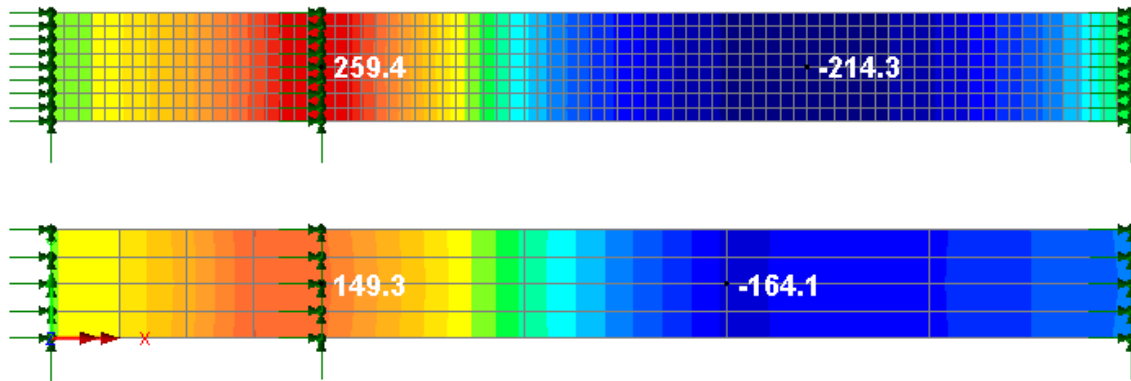
Figure 5 - Different peak bending results from identical structures, showing the importance of mesh refinement

Another important aspect to shell forces and moments is to understand that these are presented relative to the mesh element axes. The mesh element axes can be checked by double-clicking the mesh layer and ticking the 'show element axes' box. Unless the mesh is a regular, rectangular grid, the element axes will not match the surface axes. Figure 6 - **Example of a surface where none of the element axes (grey) match the surfaces axes (green)Figure 6**shows an example of a mesh which despite having a good shape, would provide the wrong results if the default contour plot was used. If the element axes are not aligned, the results will need to be transformed. This is done when setting up the results plots. For a surface in XY plane if results relative to the global axes are required, an angle of 0° on the XY plane can be specified. Otherwise, it will be necessary to set up a local coordinate set and transform results relative to that, as it is shown on Figure 6.1.
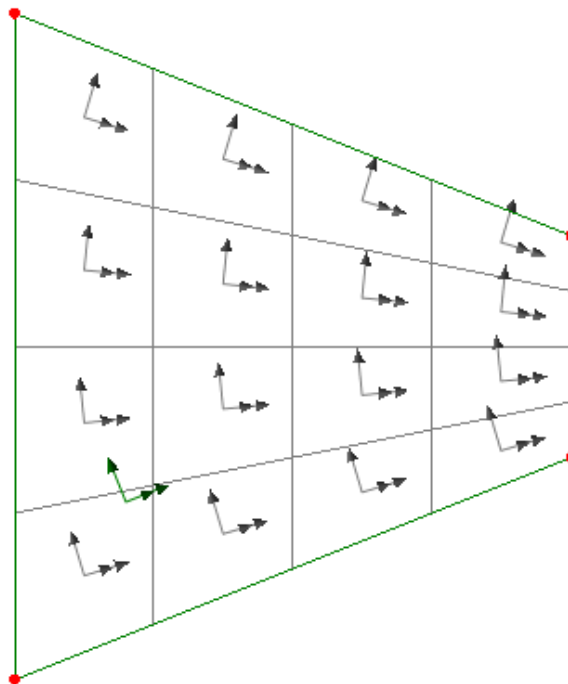

Figure 6 - Example of a surface where none of the element axes (grey) match the surfaces axes (green)
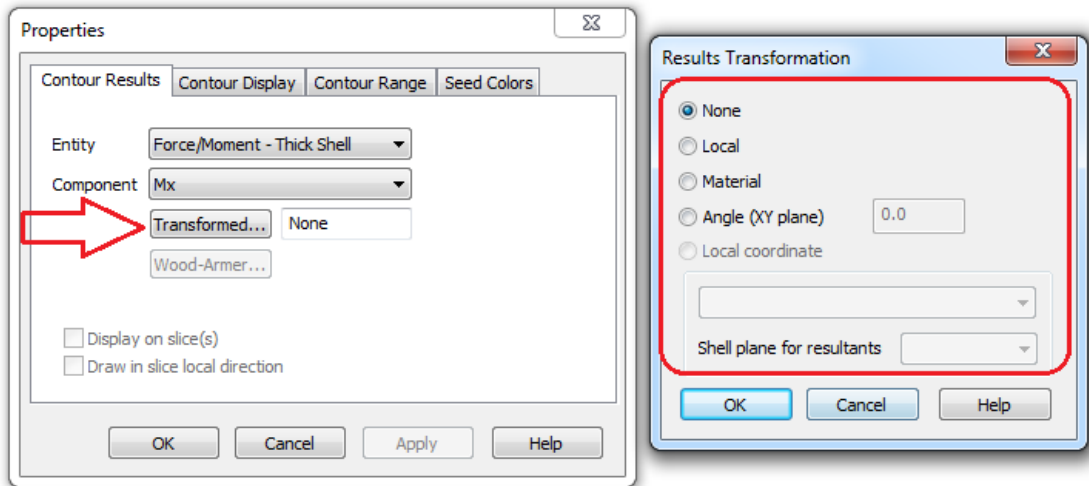
Figure 6.1 – Results Transformation

An easy way to set up local coordinates for a generalised surface is to select three of the points which define that surface and then go to 'Attributes > Local Coordinate'. In the local coordinate options box under the heading 'local coordinate generated from selection' will now be available. If 'Cartesian matrix – points A, B, C' is selected and then 'Use' is clicked, a local coordinate set will be defined, centred on the first point, with the x-axis pointing towards the second point and the y-axis lying on the plane of the three points. Therefore, by selecting the points in the correct order it will be possible to define the new reference coordinates on which the results will be able to be transformed.

Figure 7 to Figure 9 show the importance of transforming results when the element axes are not aligned to the surface axes. The example uses quadratic, triangular thick shell elements but the same problem can occur with irregular quadrilateral meshes.
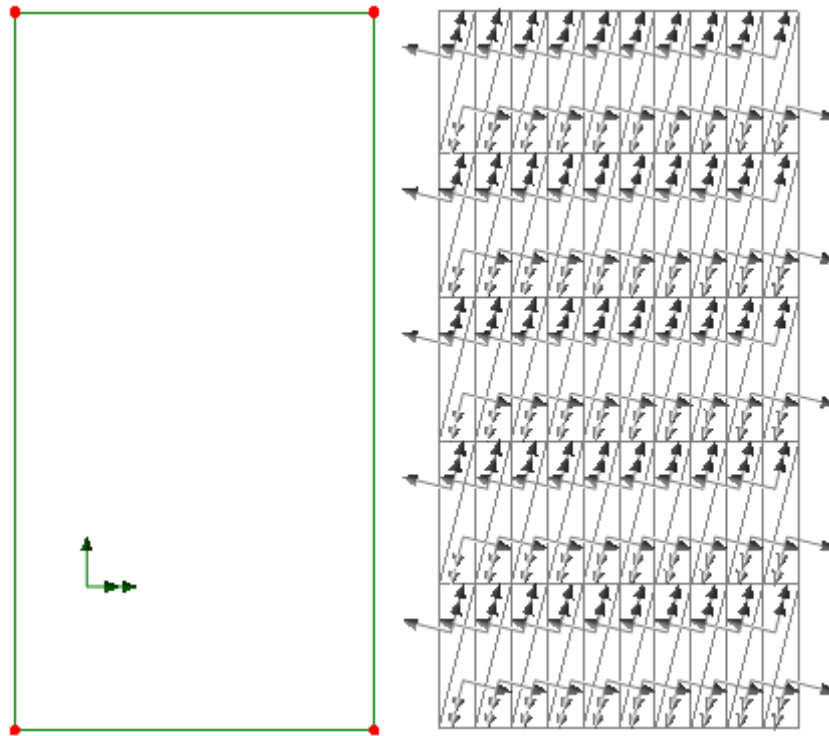
Figure 7 - Model geometry showing surface axes (left), and mesh showing element axes (right). Note that the element axes are rotated by approximately ±90° relative to the surface axes.
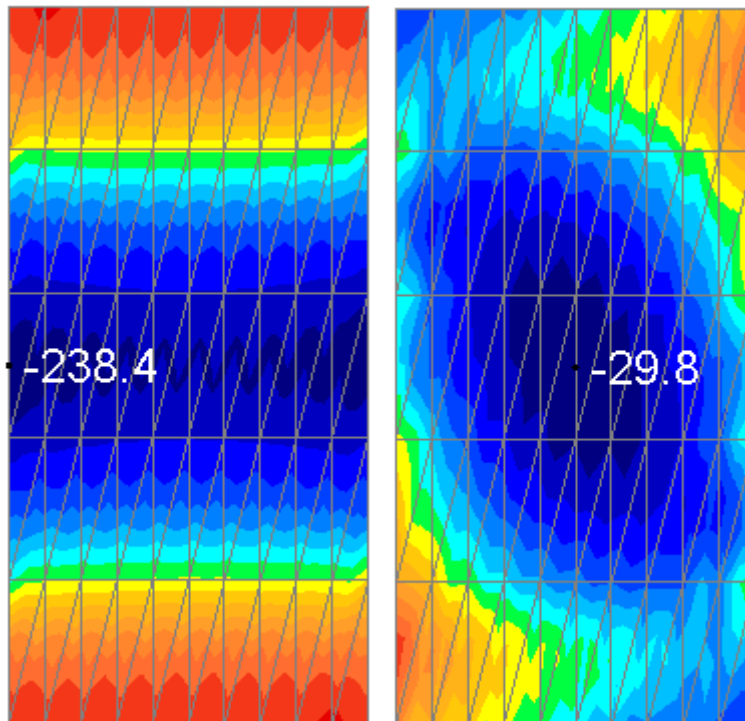

Figure 8 - Bending (My) results - transformed (relative to surface axes, left), and untransformed (relative to element axes, right). The transformed results are correct and the difference in peak moment demonstrates the importance of paying attention to element orientation. The smoothing of the contour plot means that it is not immediately obvious that the right hand results are incorrect.
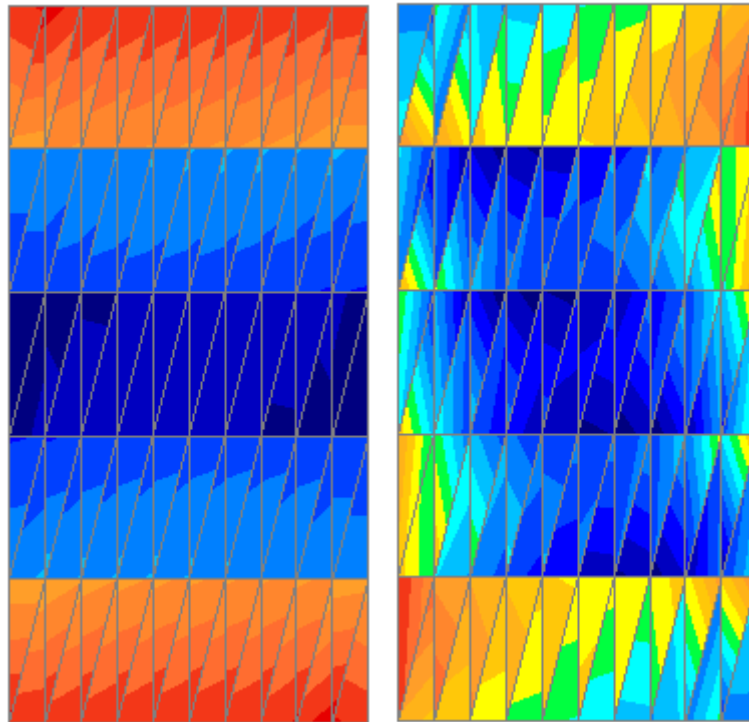
Figure 9 - The same results with contour smoothing turned off. The fragmented pattern of the right-hand diagram is a clear indicator that the results need transforming. Please note that the mesh would still need to be finer (and preferably rectangular) to provide results accurate enough for design.

2.3.2 Taking Slices

The **slice resultants from beams and shells** facility allows the computation of the equivalent beam forces and moments for beam and shell models. The slicing facility will include results across all **visible** parts of the model, for example with a slab stiffened with downstand beams the slicing facility will give results across the whole compound section, rather than just the surface feature. This can drastically simplify results processing in composite structures. For more details on this method see the example "**Section Slicing of a 3D Shell Structure**" in LUSAS Application Example Manual.

Slices are also useful for averaging peak stresses in contour plots. Shell elements in particular are prone to developing high localised shear forces at surface edges. In some cases the peak shear forces will even tend to infinity as the mesh size tends to zero. By taking a slice over a certain width at the slab edge then scaling the resultant force back up to equivalent 1m width, a more reasonable shear force for design can be determined ("The Assessment of Concrete Bridges" by the Concrete Bridges Development Group suggests that a width of 2 to 3 times the slab depth is considered reasonable by most engineers).

The example in Figure 10 shows shear results (Sy) for a 500mm thick slab subjected to gravity and a patch load at midspan, modelled with a fine regular mesh. The peak shear force is given as 716.4kN/m, but by averaging the peak value by taking a slice of twice the depth of the slab (1m) a more reasonable design value of 205.3kN/m is achieved. Note that

if using the edge line of the surface to define the slice path, a slice width of 2m must be specified to slice 1m into the actual slab.
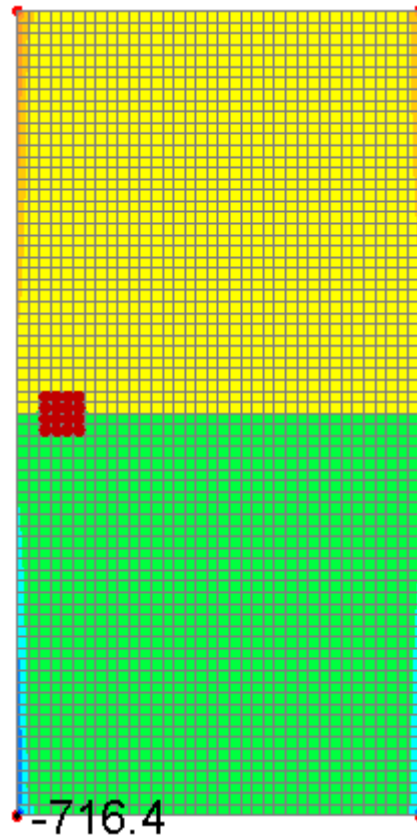

Figure 10 – Slicing to reduce peak shear example

Summary

It is critical to check element aspect ratio, shape, size and orientation. If the elements are not regularly oriented, results must be transformed relative to the global axes or the surface itself.

Appendix A - Examples of Poor Meshes with Preferable Alternatives

Poor Example	Preferable Alternative

Triangular elements

Grid with only quadrilateral elements

Right-hand surface has elongated triangular elements with highly acute corners

Single surface meshed with quadrilateral elements of better aspect ratio and fewer acute corners

Standard 4 divisions per line results in poor mesh

Giving division numbers approximately proportional to line length results in neater mesh

Six-sided surface with irregular mesh

Made regular using combined lines