CUSTOMER SUPPORT NOTE

Using Surface Elements in FE Analysis

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

This support note is issued as a guideline only.

LUSAS

Forge House, 66 High Street, Kingston upon Thames, Surrey, KT1 1HN, UK
Tel: +44 (0)20 8541 1999 Fax: +44 (0)20 8549 9399
Email: info@lusas.com www.lusas.com

Table of Contents

1. Introduction

This support note focuses on surface element types, offering guidelines for their application in finite element models, and providing options for effective visualisation and interpretation of results.

2. Description

2.1 Surface element types

Structural components should be modelled using surface elements only when their thickness is relatively small compared to their in-plane dimensions – typically less than one-tenth of the smallest in-plane dimension. There are two mesh options for modelling structural slabs and steel plates: shell and plate elements. Each option allows the selection of a thick or thin formulation, quadrilateral or triangular element shape, and linear or quadratic interpolation order (note that quadratic interpolation is available only with the Plus software option).

2.1.1 Shell vs Plate elements

Before outlining the differences between plate and shell elements, it is important to clarify the terminology used in finite element analysis. In this context, the term "plate" refers to a specific type of element designed for particular applications, such as those discussed below. This definition differs from the broader use of "plate" in structural engineering. In other words, not all structural plates are appropriately modelled using plate elements.

Shell elements differ from plate elements in that they can handle both in-plane (membrane) and out-of-plane (bending/shear) forces. Plate elements, on the other hand, are limited to only out-of-plane forces, making them suitable solely for flat, two-dimensional models with vertically applied loads. For three-dimensional models, shell elements must be employed. Therefore, shell elements offer greater versatility in a wider range of scenarios, while plate elements are computationally less demanding due to their reduced degrees of freedom compared to shells.

Application Examples:
Plate Elements:

- Flat slabs supported directly.

Shell Elements:

- Flat slabs/plating supported by other structures (e.g. columns).
- Slabs/plating with curvature or angled elevation (e.g. crossfall).
- Stiffened slabs/plating (e.g. under-slung beams).
- Walls.
- Box structures (e.g. box culverts, box girders).
- Steel sections (e.g. I-sections).

2.1.2 Thick vs Thin elements

The terms "thick" and "thin" do not specifically refer to the actual thickness of structural components. The distinction lies in their transverse shear flexibility: thick elements allow for shear deformations and can output shear forces, whereas thin elements lack shear flexibility and do not output shear forces. This difference makes thick mesh types applicable to any model, while thin meshes can be more efficient for structures where out-of-plane shear deformations are minimal.

For example, in a steel I-beam, shear is primarily carried by in-plane shear in the web rather than out-of-plane shear in the flanges, making thin shells suitable for modelling such components. However, thick shells could also be considered in modelling these components. Concrete slabs, where shear strains are expected to contribute significantly to the deformation, can be modelled using thick shell elements.

2.1.3 Triangular vs Quadrilateral

A quadrilateral mesh typically provides better results than a triangular mesh due to its ability to accommodate higher-order variations of forces and moments across the elements (refer to the LUSAS Element Reference Manual).

Triangular elements should be used sparingly, mainly in small regions where their geometry facilitates efficient meshing. They can also serve as transition elements between quadrilateral elements of different sizes.

The mesh can be refined by adjusting line divisions to minimise the use of triangles. Another effective way to control surface mesh density is by using the *Element size* option.

2.1.4 Linear vs Quadratic

Linear thick shell elements (QTS4) do not capture the variation in out-of-plane bending moments across a structure as effectively as linear thin shell elements (QSI4), quadratic thick shell elements (QTS8), or plate elements. Therefore, when using QTS4 elements, a more refined mesh is necessary compared to the other elements mentioned. The use of quadratic interpolation is generally recommended, if available.

It is worth noting that elements with quadratic interpolation are better suited for modelling curved structures. While linear interpolation elements can also be used, achieving comparable accuracy and efficiency typically requires a higher number of elements.

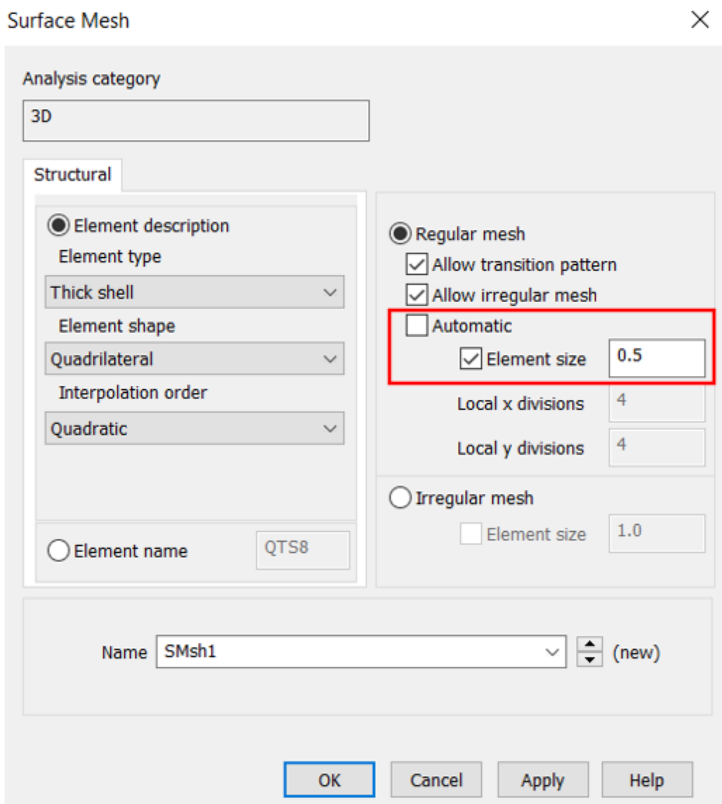
2.1.5 Summary

For bridge analysis, the recommended default mesh consists of thick shell, quadrilateral, quadratic elements (QTS8) for concrete slabs, walls, and steel plates. When modelling steel plates in linear elastic analysis, an efficient alternative is to use thin shell, quadrilateral, linear elements (QSI4), which reduce computational effort without significantly compromising accuracy. However, special attention should be given when connecting shells to beams, as compatibility between these elements must be considered.

2.2 Mesh application

The results of a finite element analysis depend not only on the type of elements selected but also on the size, shape, and distribution of the elements. A finer mesh (smaller elements) generally yields more accurate results, but it will require more computational time compared to a coarser mesh (larger elements). Mesh density can be controlled in several ways:

1. By adjusting the default number of mesh divisions. This can be changed by navigating to File > Model Properties > Meshing (Tab) > Default Line Divisions. By default, this number is set to 4, which determines the number of divisions for any line that has not been modified using the methods described below.
2. By specifying a size for the surface mesh elements. This will override the default mesh divisions.



3. By applying NULL line meshes (line meshes with no element type selected) to the edge lines of a surface. The number of divisions in the NULL mesh will determine the number of elements along that edge. Additionally, element spacing can be adjusted by clicking the *Spacing* button, allowing for non-uniform spacing along the surface edge. The NULL mesh will take precedence over the previous two methods.

Once the mesh has been applied to the model, it is essential to assess its quality. The mesh can be refined using the methods previously outlined. Ideally, elements should have aspect ratios close to 1, and no greater than 10 – meaning square elements are preferred over elongated rectangles. Additionally, sharp angles at element corners should be avoided whenever possible.

If the mesh divisions on opposite sides of a surface are equal, a regular grid mesh will be generated. Otherwise, the result will be an irregular mesh or transition pattern. For surfaces with more than four sides, a regular mesh can be achieved by creating “combined lines” from selected perimeter lines. For instance, in the image to the right (Figure 1), a combined line is defined using the two bottom perimeter lines of the surface to create a regular mesh.

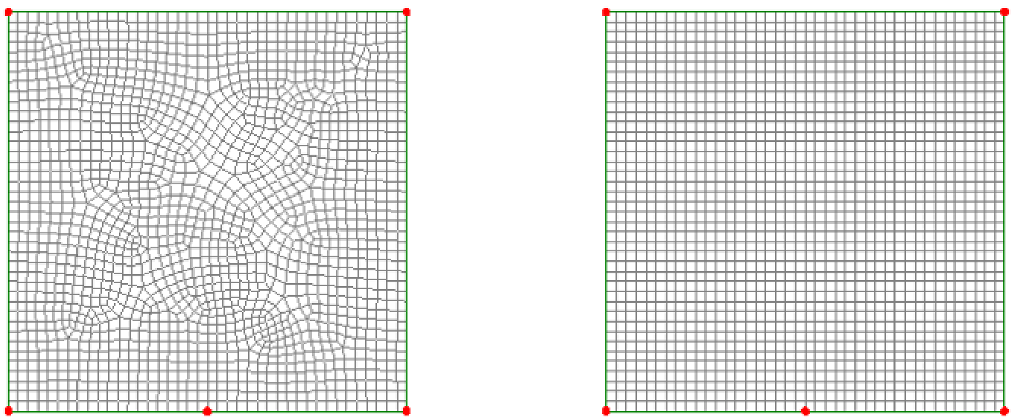


Figure 1 - Five-sided surfaces meshed with an irregular mesh (left) and a regular mesh using a combined line (right).

Ensuring consistency in the surface z-axes is crucial, as any discrepancies can cause local shell results to alternate between positive and negative values across surfaces. To view the surface axes, double-click the *Geometry* layer and select *Surface Axes*. If the axes are inconsistent (see Figure 2), they can be corrected by selecting Geometry > Surface > Reverse. Alternatively, to align all surfaces, select one correctly-oriented surface, press “CTRL-A” to select the entire model, and then click Geometry > Surface > Cycle Relative.

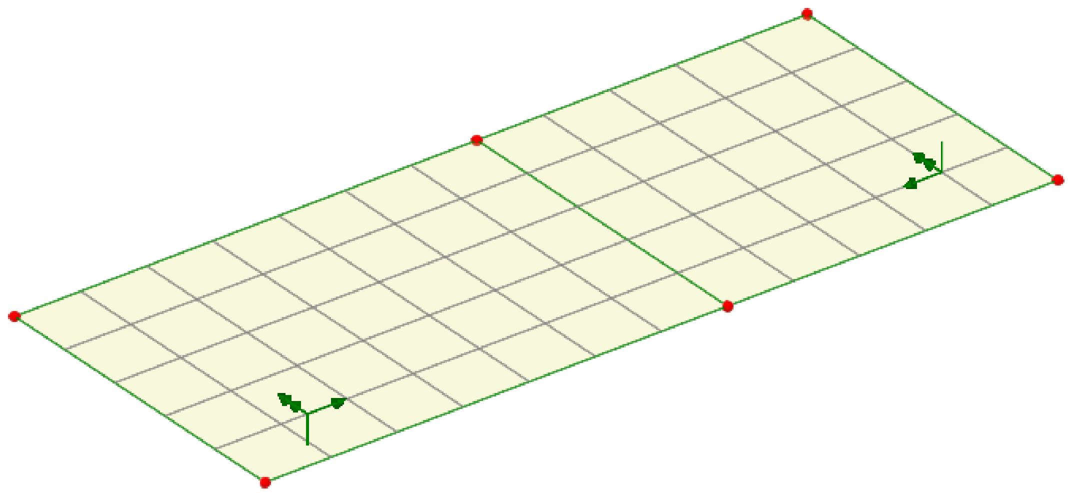


Figure 2 - Inconsistent surface axes.

2.3 Viewing results

It is important to note that in plate and shell elements, forces and moments are defined “per unit width”. Section 2.3.2 of this document introduces a tool for calculating resultant values, which enables the conversion of results from a complex shell model into an equivalent beam analogy. Shell element forces and moments are expressed in the element's local coordinate system (x, y, z), while shell stresses (Top/Middle/Bottom) are given in the global coordinate system (X, Y, Z). These topics are discussed in detail below.

2.3.1 Contour plots

Contour plots are one of the most effective ways to visualise surface results, providing a clear representation of the distribution of forces, moments, displacements, and other quantities of interest. However, they can be misleading if not properly interpreted.

By default, averaged nodal contours are displayed while respecting discontinuities caused by geometry orientation, geometric assignments, and material assignments. The default averaging settings can be adjusted in the *Model Properties - Options (Averaging)* dialog. However, averaging may sometimes smooth out peak values, potentially leading to non-conservative results – especially when a coarse mesh is used.

Unaveraged nodal contours, on the other hand, display results on an element-by-element basis, revealing inter-element discontinuities. This is particularly useful for detecting potential mesh discretisation errors and displaying results across geometric or material discontinuities.

Figure 3 presents the results of a slab subjected to a concentrated load at the centre, using the *Unaveraged nodal* option (left) and the *Averaged nodal* option (right). The unaveraged results (left) clearly reveal that the mesh is too coarse, indicating the need for a rerun with a refined mesh. In contrast, the averaged results (right) provide a smoother plot, which could give the misleading impression that no mesh refinement is necessary.

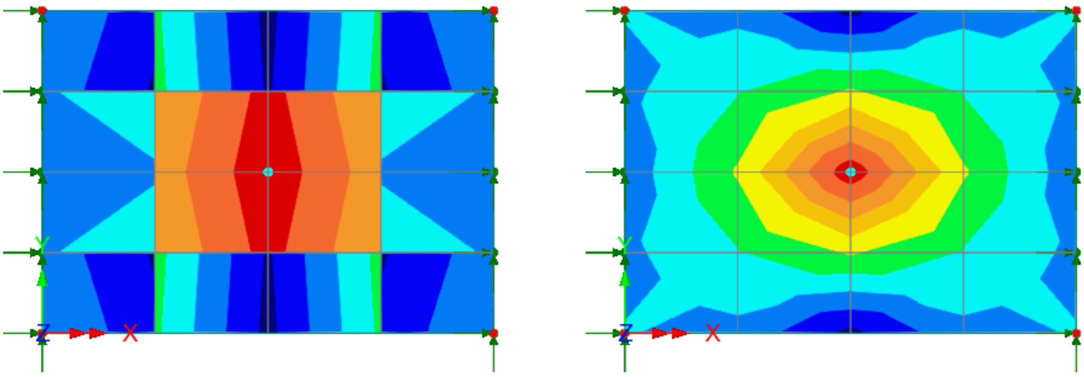


Figure 3 – Coarse mesh: Unaveraged results (left) and averaged results (right).

A finer mesh can reduce the differences between averaged and unaveraged results, as shown in Figure 4.

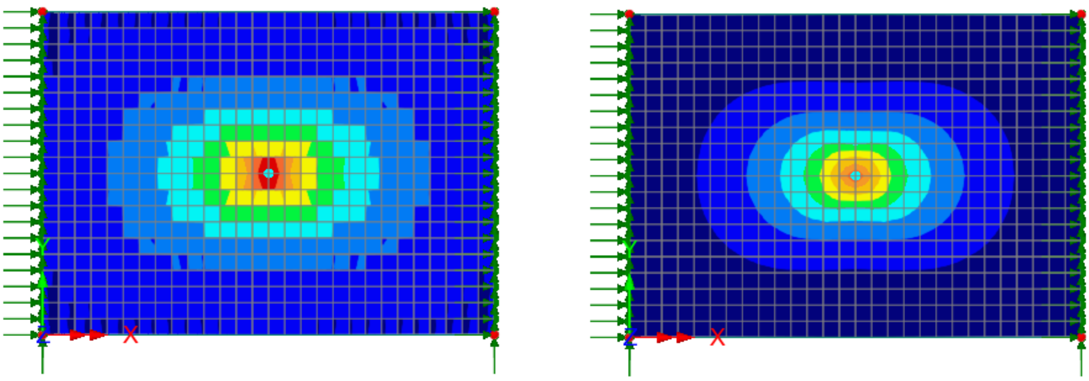


Figure 4 – Dense mesh: Unaveraged results (left) and averaged results (right).

It is also important to note that shell forces and moments are presented relative to the element axes. These axes can be viewed by double-clicking the mesh layer and selecting the *Show Element Axes* option. Unless the mesh is a regular rectangular grid, the element's x-y axes will not align with the surface's x-y axes, although the element's z-axis will always align with the surface's z-axis. Figure 5 illustrates an example of a mesh that would yield results based on different orientations if the default option (*No transformation applied*) were used. When the element axes are misaligned, the results must be transformed, which can be done using the *Results transformation* dialog (see Figure 6).

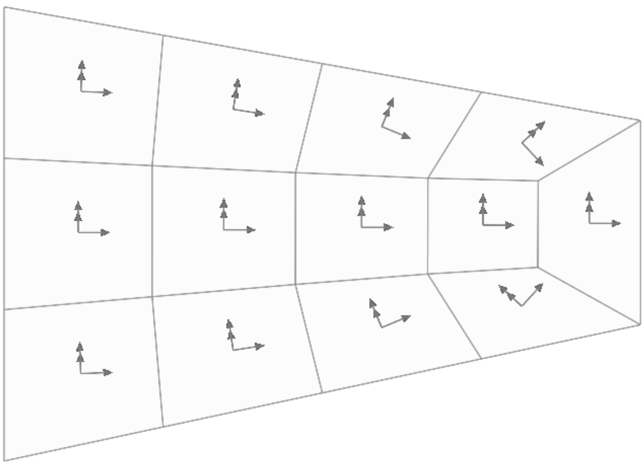


Figure 5 – Example of a surface where the element x-y axes vary.

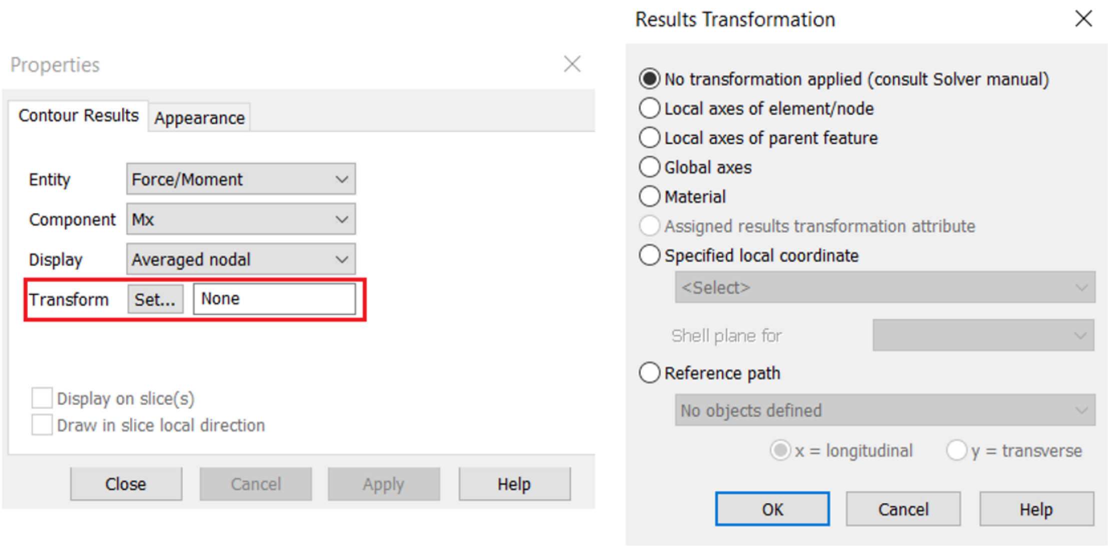


Figure 6 – Results transformation.

Figure 7 shows a slab modelled using triangular elements. Due to inconsistent element axis orientations, results obtained without transformation are not suitable for the design or assessment of the slab. In this case, transforming the results relative to the global axes is a convenient way to ensure consistency. Alternatively, the results can be transformed relative to another preferred coordinate system, with the available options shown in Figure 6.

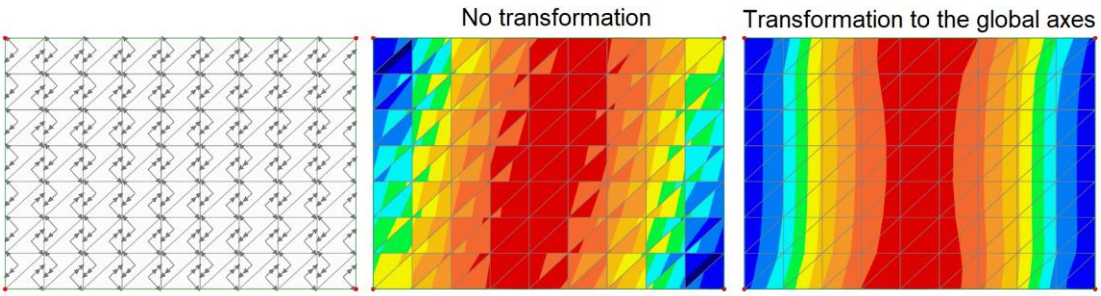


Figure 7 – Difference between untransformed and transformed results.

2.3.2 Slice resultants

The *Slice Resultants Beams/Shells* facility computes equivalent beam forces and moments for beam and shell models. It converts results from a complex shell model into an equivalent beam analogy, enabling checks against design codes. This is particularly beneficial in the design of composite structures.

Figure 8 depicts a composite girder model, where both the concrete slab and metallic girder are modelled using shell elements. The bending moments at specified locations of interest can be extracted using the *Slice Resultants Beams/Shells* facility. In some cases, plotting these bending moments in a diagram can help verify whether the model behaves as expected.

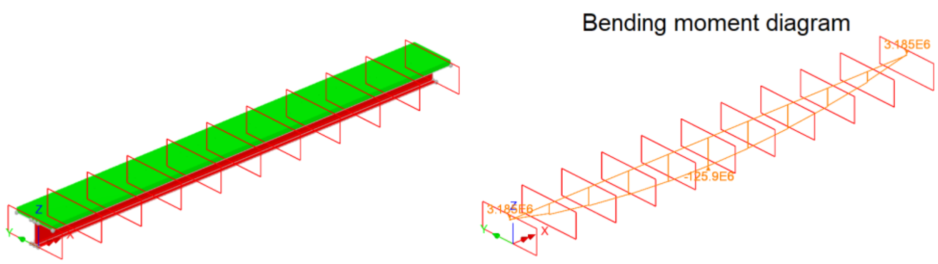


Figure 8 – Shell element model: Bending moment diagram created using the Slice Resultants Beams/Shells facility.

Additionally, this facility helps to average high localised forces, which are often developed along surface edges in shell elements. These forces may not always be realistic. By taking a slice over a specified width and scaling the resulting force to an equivalent 1m width, a more reasonable force can be obtained for design purposes. *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.

3. Summary

Using surface elements:

1. **Element type choice:**
    - For bridge analysis, the recommended default mesh consists of thick shell, quadrilateral, quadratic elements (QTS8) for concrete slabs, walls, and steel plates.
    - When modelling steel plates in linear elastic analysis, an efficient alternative is to use thin shell, quadrilateral, linear elements (QSI4), which reduce computational effort without significantly compromising accuracy.
    - Special attention should be given when connecting shells to beams, as compatibility between these elements must be considered.
2. **Element characteristics:**
    - Element aspect ratio, element size, and shape: Ensure these are appropriate for the intended analysis (refer to CSN/LUSAS/1025).
    - Element axis orientation: If orientations are inconsistent, results must be transformed for accurate interpretation.
3. **Averaged and unaveraged results:**
    - Unaveraged results can highlight potential issues with mesh density.
    - Averaging may smooth out peak values, potentially leading to non-conservative results – especially when a coarse mesh is used.
4. **Slice Resultants Beams/Shells facility:**
    - Converts results from a complex shell model into an equivalent beam analogy, facilitating design code checks.
    - Particularly useful in the design of composite structures.

If you have any doubts or require specific advice for your type of analysis, please contact the LUSAS Technical Support team at support@lusas.com.