CUSTOMER SUPPORT NOTE

A Brief Guide to Meshing

Note Number:	**CSN/LUSAS/1025**

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

In finite element analysis, a real structure is idealised into a finite number of elements – hence the name. The type and arrangement of these elements have a significant impact on the results. Choosing the wrong element type or arranging the elements poorly can lead to incorrect and unconservative results.

2. Description

Most finite element users know that an insufficiently fine mesh will yield poor results. However, it is challenging to determine exactly how fine a mesh should be. While this is largely analysis-specific, there are a few general considerations that should be applied to every analysis.

2.1 Mesh Type

LUSAS offers a wide variety of element types to accommodate different geometric features such as lines, surfaces, and volumes. Each feature type has several element options available. Comprehensive details about all the element types can be found in the **Element Reference Manual**, accessible through the *Help* system. A summary table is provided on page 4 of this document.

A basic understanding of the features of various element options is crucial when setting up an analysis. For instance, there are currently more than ten different types of beam elements available. Depending on the type of analysis, some may be more suitable than others. Certain element types can handle specific material properties and geometric nonlinearities, while others cannot. When using multiple element types in an analysis, such as beams and shells, it is important to consider the compatibility between them. Additional considerations should be made to ensure that the mix of elements used in the model produces sufficiently accurate results. For instance, it is advisable to avoid mixing linear order elements with quadratic order elements to achieve more accurate results.

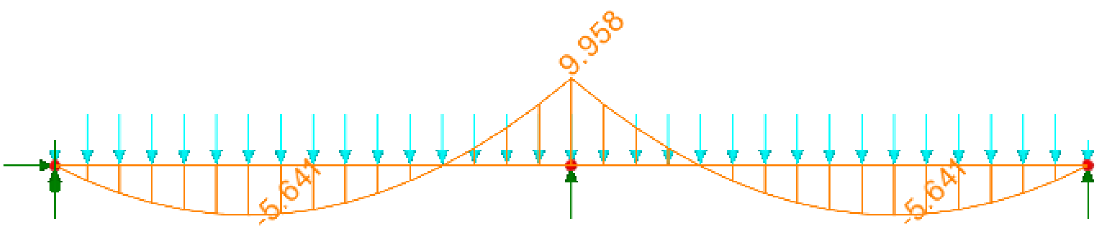
Here is a summary of some important information:

Thick and thin elements:

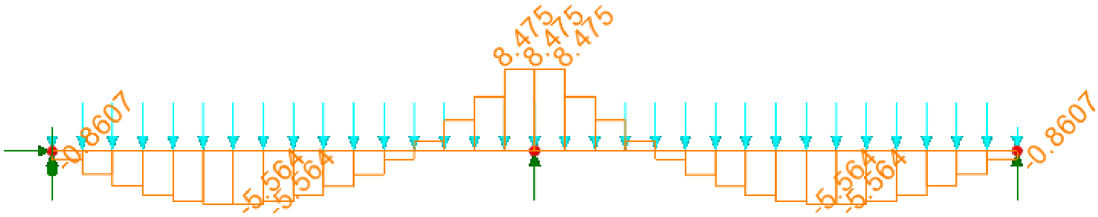
The distinction lies in their transverse shear flexibility: thick elements allow for shear deformations, whereas thin elements lack shear flexibility. This difference makes thick mesh types applicable to any model, while thin meshes can be more efficient for structures where out-of-plane shear effects are minimal. For example, in a steel I-beam modelled using surface elements, 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.

Force variation across element:

Another important factor when selecting a mesh type is the force/moment variation across each element. Some elements perform better than others in this regard. One of the most used beam elements is the 3D thick beam with linear interpolation order (BMI21), known for its efficiency. BMI21 elements effectively model quadratic variation in bending moment, resulting in a smooth and continuous bending moment diagram. This allows for accurate results even with a relatively small number of BMI21 elements.



In contrast, the 3D thick nonlinear beam (BTS3) – now retired but used here for comparison – yields constant bending moment results. This results in a “stepped” bending moment diagram, which can notably underestimate moment values.



Comparing the results presented above, it becomes clear that using BMI21 elements produces a more accurate bending moment diagram. The analytical bending moment at the intermediate support is 10 kNm, which closely matches the result obtained using BMI21 elements. The result obtained using BTS3 elements is 15% lower than the analytically predicted value.

Similarly, when using the same number of elements, QSI4 thin shell elements capture the variation in out-of-plane bending more effectively than QTS4 thick shell elements, which generally produce an almost constant out-of-plane bending moment per element.

Therefore, it is important to determine whether the variation of results across each element is suitable for the intended purposes of the analysis.

Linear and quadratic interpolation order:

All mesh types have an interpolation order that is either linear or quadratic. Linear order elements only have nodes at the ends (or corners) of each element, while quadratic order elements additionally include midside nodes halfway along each edge. Quadratic order elements typically exhibit higher-order force variations compared to their linear counterparts, requiring fewer elements to achieve accurate results.

A summary of the structural mesh types available in LUSAS (excluding thermal types) is presented on the following page.

Category	Subcategory	Description, uses
Beams and bars		These mesh types are assigned to line features and are generally appropriate for the analysis of structural members where the cross-sectional dimensions are much smaller compared to the length.
	Bars	Bars carry only axial forces. They are typically used for cables (with a single division to prevent mechanisms) or for rebar in reinforced concrete structures.
	Beams	Beams carry axial force, torsion, bending moments, and shear forces. Among the various types of beams, the 3D thick beam (BMI21 and BMI31) is likely the most versatile. It is widely used for modelling frame-type structures and can also be connected to shell elements for modelling beam-and-slab or stiffened plate structures.
Plates and shells		These mesh types are assigned to surfaces and are generally suitable for the analysis of flat or curved structures where the thickness is much smaller than the other two dimensions (length and width).
	Plates	Plates only carry out-of-plane forces and moments (i.e. no in-plane membrane forces). They can be used for the analysis of simple flat structures such as concrete slabs.
	Shells	Shells also carry in-plane forces, making them more versatile than plate elements. They can be used for modelling 3D structures such as the walls of box girders or the flanges and webs of I-beams.
2D continuum		2D continuum meshes are assigned to 2D surfaces, which are modelled in the XY plane. A 2D model represents a cross-section of a 3D structure.
	Plane stress	Plane stress elements carry stress exclusively in the in-plane (XY plane) directions but may experience strain in the out-of-plane direction. These properties make them suitable for analysing planar structures with relatively small thickness. Refer to the example **Linear Elastic Analysis of a Spanner** in the LUSAS worked examples for further details.
	Plane strain	Plane strain elements undergo strains exclusively in the in-plane directions but can experience non-zero stress in the out-of-plane direction. They are suitable for analyses where the surfaces represent a slice through a long structure. A typical application is for embankments or cuttings in geotechnical engineering. Many worked examples demonstrating geotechnical applications using plane strain elements have been added to the LUSAS worked examples; refer to them for further information.
	Axisymmetric	Axisymmetric elements are a special type of two-dimensional elements particularly useful when symmetry with respect to geometry and loading exists about an axis of the body being analysed. They represent a 2D slice of a 3D revolved solid about the axis of symmetry. They are commonly applied in the analysis of cylindrical storage tanks, pressure vessels or mechanical components.
3D continuum		A 3D continuum mesh is utilised when it is essential to accurately represent the geometry of a structure in finite element analysis. However, the high number of elements typically needed for such analyses often makes it impractical or slow to use for large-scale structures, such as entire bridges or buildings.
	Solids	Solid meshes are typically employed for detailed analysis of structural or mechanical components that cannot be adequately represented by other types of meshes.

2.2 Mesh Density

It is advisable to evaluate all analyses with a mesh refinement check. It is crucial to conduct a mesh refinement study for finite element analysis across all types and subjects of analysis, using any software package.

This process entails adjusting the mesh density, re-solving the analysis, and assessing whether the results show significant changes. If increasing the mesh density significantly alters the results, it indicates that the mesh may be too coarse. Once the results begin to converge as the number of elements increases (and element size decreases), it provides confidence that the mesh density is adequate.

It is worth noting that:

- Elements with constant force variations require very fine meshes to achieve more accurate results.
- Quadratic elements (quadratic interpolation order) typically yield more accurate results with a coarser mesh compared to linear elements (linear interpolation order).

2.3 Mesh pattern

Besides setting the number of mesh divisions, it's essential to ensure that the meshing pattern is effective. To achieve this, avoid the following:

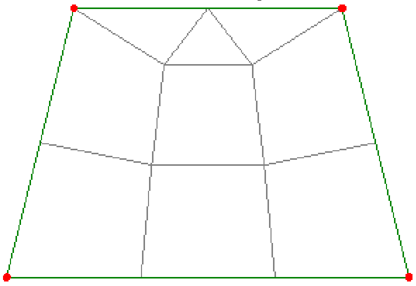
- Elongated (high “aspect ratio”) elements.
- Elements with acute-angled corners.

The ideal element shape is as close to square as possible. This is important because results are calculated at locations, called Gauss points, within the surface or volume elements. These results are then extrapolated from the Gauss points to the element nodes. In highly elongated or angular elements, the distance between the Gauss points and the nodes can be large relative to the element size, introducing errors/inaccuracies in the extrapolation process.

Meshing surfaces and volumes is a complex topic, but the fundamental principle is to maintain simple shapes for surfaces and volumes to allow for a neat grid of elements. Square elements are generally preferable to triangular elements for surface meshes, while hexahedral meshes are favoured over pentahedral or tetrahedral meshes for volumes.

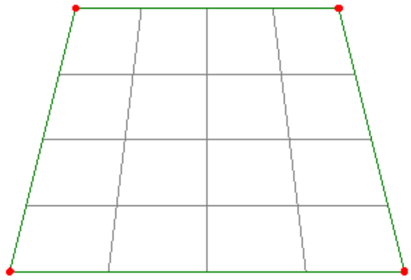
Examples of poor meshing practices and their preferable alternatives are provided on the following page.

Poor Example

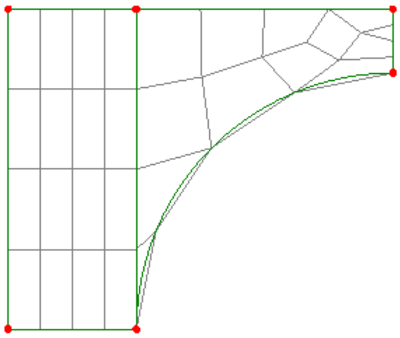


Mesh pattern using both triangular and quadrilateral elements

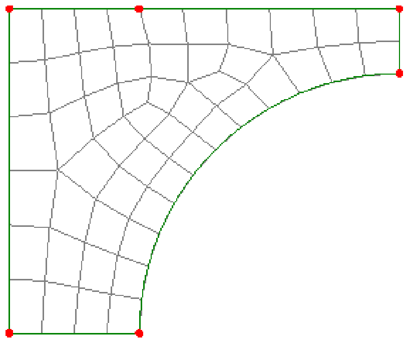
Preferable Alternative



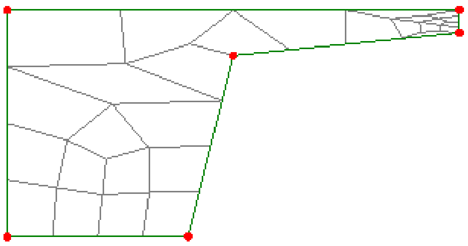
Mesh pattern using quadrilateral elements only



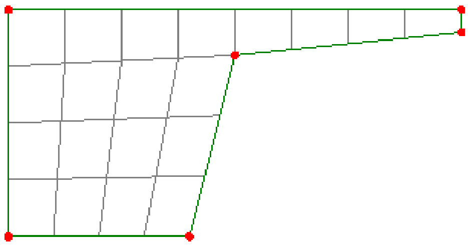
The surface on the right has elongated triangular elements with very acute corners



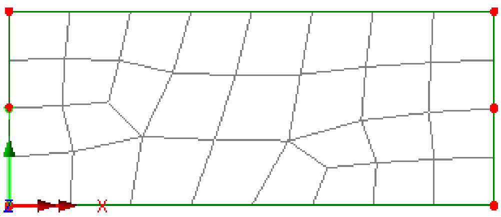
A single surface meshed with quadrilateral elements having a better aspect ratio and fewer very acute corners



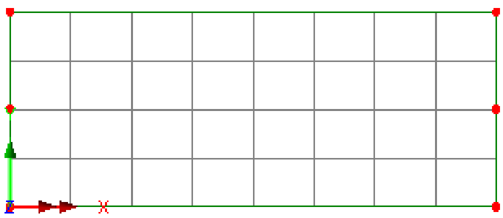
Mesh pattern using the standard 4 divisions per line



Specifying appropriate numbers of divisions per line



Six-sided surface with irregular mesh



Using “combined lines” to achieve a regular mesh

3. Summary

Analysis preparation guidelines:

1. **Element type selection:**
   - Evaluate the various element types to determine which are most suitable for your model.
   - Ensure compatibility when using more than one type of element.
2. **Mesh density verification:**
   - Perform a mesh refinement check to ensure that the mesh density is adequate.
3. **Mesh pattern inspection:**
   - Check the pattern of any surface or volume meshes to ensure the elements are as close to square as reasonably possible.

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.