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 all finite element analysis, a real structure is idealised to a (finite) number of 'elements' - hence the name. The type and arrangement of these elements will have a large influence on the results achieved. Poor selection of element type or a poor arrangement of elements can provide incorrect and unconservative results.

2. Description

Most finite element users are aware that they will get poor results if the mesh (of elements) is not fine enough, but it is difficult to quantify exactly how fine a mesh should be. To a large extent this is analysis-specific, but there are a few general considerations which should be applied to every analysis.

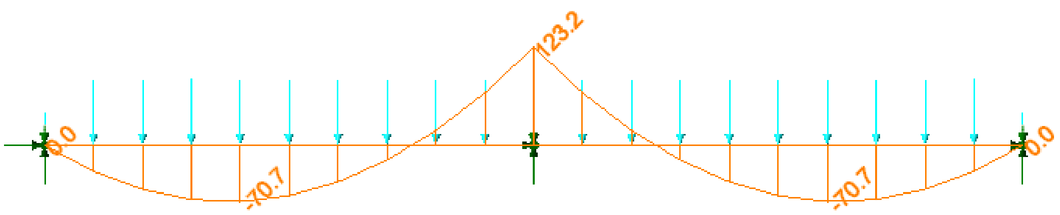
2.1 Mesh Type

There are many different element types available in LUSAS. Different geometric features (e.g. lines, surface or volumes) will require different element types, but there are many element options for each type of feature. Details of all of them can be found in the '**Element Reference Manual**' which is accessible through the Help system, but a summary table is provided on page 4 of this document.

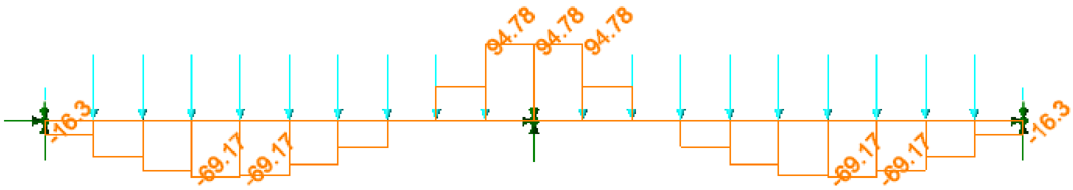
A basic knowledge of the features of the various element options is very useful when planning an analysis. For example, there are (at time of writing) 7 different types of beam element. Depending on the analysis being undertaken some of these might be more suitable than others. Some element types will accept certain types of material properties and/or geometric nonlinearity whereas others will not. If you are using more than one type of element in an analysis (for example beams and shells) compatibility between the specific beam element and the specific shell element must also be considered. Linear order elements should not be mixed with quadratic order elements if it can be avoided.

Some mesh types specify either 'thick' or 'thin' elements. The difference is that thick elements have transverse shear flexibility, meaning that they undergo shear deformations and as a result can output shear forces. Thin elements do not have shear flexibility, and as a result do not output shear forces. This difference means that thick mesh types are applicable to any model, but thin meshes can be more efficient for structures where out-of-plane shear effects are minimal. For example, shear in a steel I-beam is primarily carried by in-plane shear in the web rather than out-of-plane shear in the flanges, so thin shells could be used for modelling such a component.

Another important consideration when selecting a mesh type is the force variation across each element. The '3D thick beam' (BMS3) is probably the most commonly-used beam element and is very efficient because each individual element has a quadratic variation of bending moment across it. This means that the bending moment diagram across each element will form a nice smooth curve, resulting in good results even with relatively few BMS3 elements used:



Conversely, the ‘3D thick nonlinear beam’ (BTS3) has constant bending moment results across each element. This results in a ‘stepped’ bending moment diagram which can drastically underestimate moment results, especially at sharp changes such as hogging at supports. The diagram below shows how the hogging moment of 123kNm predicted by the BMS3 elements is underestimated by 23% when BTS3 is used.



It is therefore important to be aware of whether your chosen mesh has quadratic (very good), linear (good) or constant (poor) variation of results across each element. In LUSAS you can check this in the **Element Reference Manual** under the heading ‘**Notes on Use**’. For example, the entry for element type BMS3 states *“The force variations along the beam are constant axial force, constant torsion, linear shear forces and quadratic moments.”* Obviously an element with higher-order force variations is preferable.

All mesh types have an ‘interpolation order’ which is either linear or quadratic. Linear order elements only have nodes at the ends (or corners) of each element, whereas quadratic order elements also have ‘midside’ nodes half way along each edge of the element. Quadratic order elements usually have higher-order force variations than the linear equivalent and therefore fewer are required to achieve accurate results.

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

Category	Subcategory	Description, uses
Beams and bars		These mesh types are assigned to line features and are generally suitable for the analysis of structural members where the cross-section dimensions are much smaller than the length.
	Bars	This mesh type only carries axial force. It is typically used for cables (use a single division only to prevent lateral mechanisms due to the lack of bending resistance) or for rebar in reinforced concrete structures.
	Beams	Beams carry axial force plus torsion, bending moments and shear forces. There are various types of beam, the most generally-applicable of which is probably the ‘thick 3D beam’ (BMS3). These are very widely used for frame type structures and can also be connected to shell elements for beam-and-slab or stiffened plate structures.
Plates and shells		These mesh types are assigned to surface features and are generally suitable for the analysis of flat or curved structures where the thickness is much smaller than the plan dimensions.
	Plates	These 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	These also carry in-plane forces so are more versatile than plate elements. They can be used for 3D structures such as the walls of box girders or the plates of I-beams.
2D continuum		2D continuum meshes are assigned to 2D surface features, which must be modelled in the XY plane. The 2D model represents a section through a 3D structure.
	Plane stress	Plane stress elements only carry stress in the in-plane (XY plane) directions, but can experience strain in the out-of-plane direction. These assumptions make them suitable for analysis of planar structures of limited thickness. See example “**Linear Elastic Analysis of a Spanner**” in the **Examples Manual**.
	Plane strain	Plane strain elements only undergo strains in the in-plane directions, but can carry stress in the out-of-plane direction. These assumptions make them suitable for analyses where the surfaces represent a slice through a long structure. A typical use is for embankments or cuttings in geotechnical applications. See example “**Drained Nonlinear Analysis of a Retaining Wall**” in the **Application Examples Manual (Bridge, Civil & Structural)**.
	Axisymmetric	Similar to plane strain elements except that rather than a section through a long, straight structure, these elements are formulated to represent a section through a circular structure. Typical uses include the analysis of circular pressure vessels or mechanical components.
3D continuum		A 3D continuum mesh is used where it is necessary to represent the actual geometry of a structure in a finite element analysis. The number of elements required for this type of analysis usually makes it impractical and/or slow to use them for large-scale analyses (for example whole bridges or buildings).
	Solids	Solid meshes are typically used for detailed analysis of mechanical components which cannot reasonably be represented by any of the other types of mesh.

2.2 Mesh Density

As well as an appraisal based on the judgment of the engineer, it is recommended that all analyses are evaluated with a mesh refinement check. This involves changing the mesh density, re-solving and checking whether the results are significantly affected. If increasing the mesh density changes the results significantly, this suggests that the mesh is not fine enough. Once the results start to converge with increasing numbers of elements (decreasing element size) then you can have confidence that the mesh density is sufficient.

In particular, be aware that any elements with ‘constant’ force variations will require very fine meshes to achieve accurate results. Quadratic order elements will generally provide good results with a coarser mesh than the equivalent linear-order element.

It is always important to perform a mesh refinement study for Finite Element Analysis in any type and subject of analysis, with any software package. Please see the NAFEMS article: http://www.nafems.org/resources/knowledgebase/001/

Please see the example "**Simple Building Slab Design**" also in the **Application Examples Manual**, which has a **Discussion** section at the end of the example looking at the effects of mesh refinement in a mesh refinement study.

2.3 Mesh pattern

Once surfaces and volumes are introduced into an analysis, meshing becomes more complicated. As well as setting the number of mesh divisions, it is now necessary to ensure that the meshing pattern is reasonable. In the simplest terms, the following should be avoided:

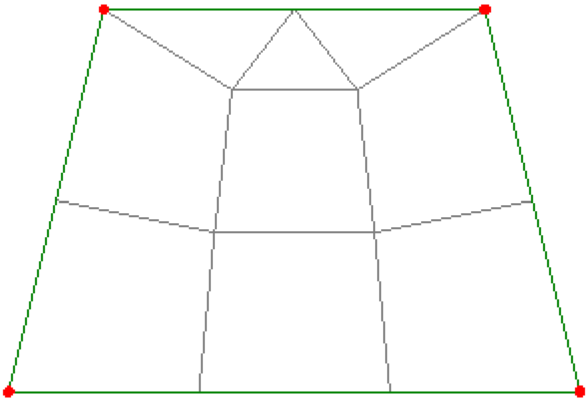
- Elongated (high ‘aspect ratio’) elements
- Elements with acute-angled corners

In other words, the ideal element shape is as near to square as possible. The reason for this is that results are calculated at a number of locations (called ‘Gauss’ points) within the surface or volume elements. Results are extrapolated from the Gauss points to the element ‘nodes’ from where they can be read by the user. In the case of highly elongated or angular elements, the distance between the Gauss points and the nodes can be large relative to the size of the element, and errors can be introduced in the extrapolation process.

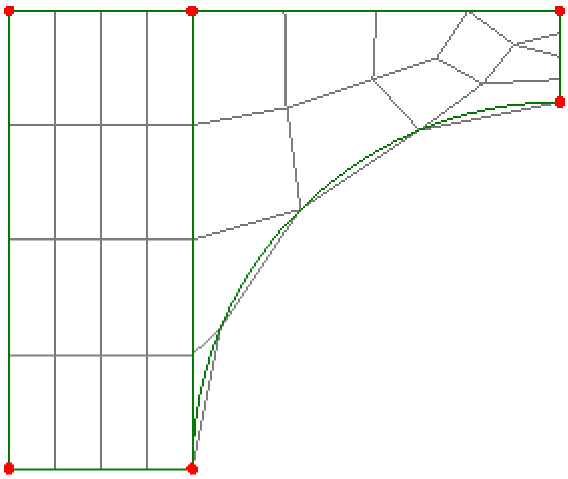
Meshing of surfaces and volumes is a complicated topic, but the golden rule is to keep the shape of the surfaces and volumes as simple as possible to allow a neat grid of elements to be used. Square elements are preferable to triangular elements for surface meshes, and hexahedral meshes are preferable to pentahedral or tetrahedral meshes for volumes.

Some examples of poor meshing and preferable alternatives are given on the following page.

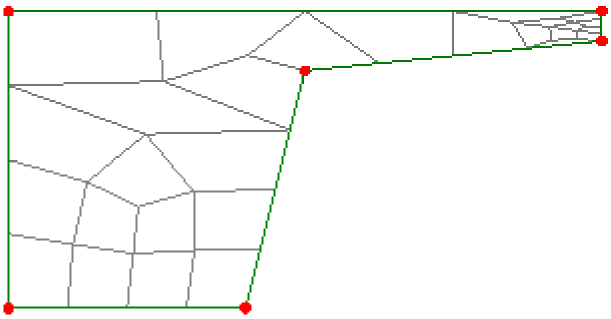
Poor Example



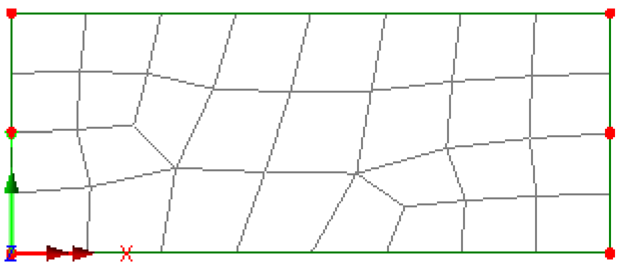
Triangular elements



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

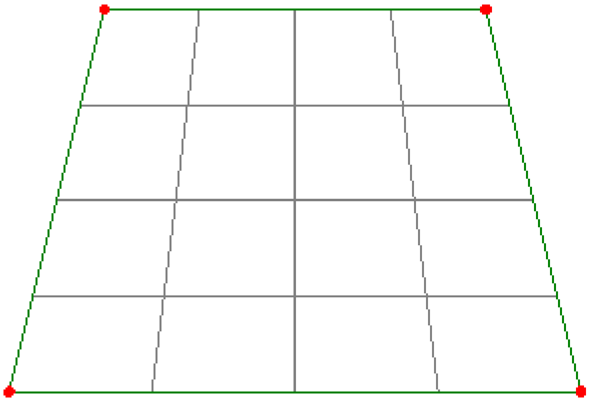


Standard 4 divisions per line results in poor mesh

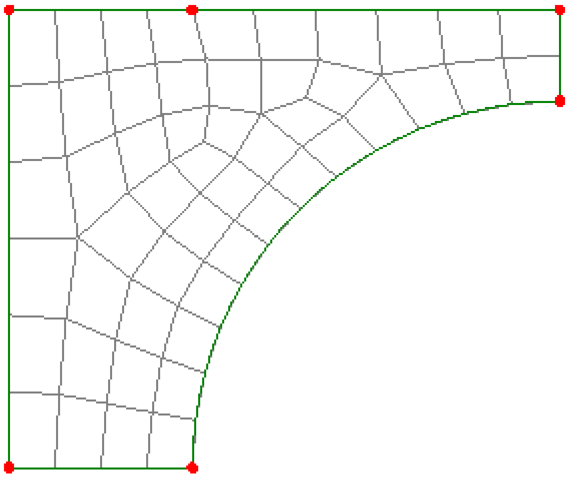


Six-sided surface with irregular mesh

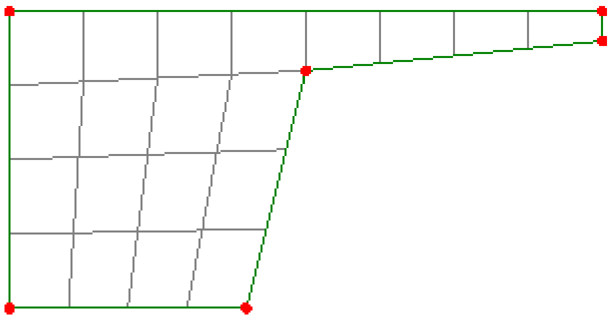
Preferable Alternative



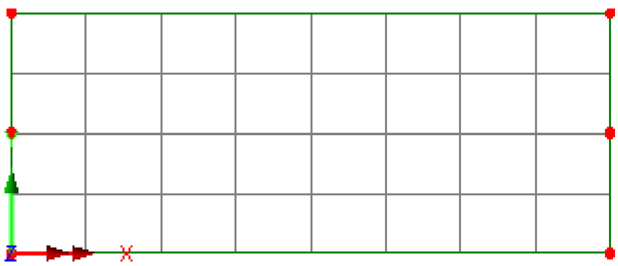
Grid with only quadrilateral elements



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



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



Made regular using ‘combined lines’

3. Summary

- Before starting the analysis, consider the options with regard to element type.
- If more than one type of element is used in the analysis, ensure that they are compatible.
- Ensure that the mesh density is sufficient using a mesh refinement check.
- Check the pattern of any surface or volume meshes to ensure the elements are as close to square as reasonably possible.

If in any doubt, please contact LUSAS Technical Support team (support@lusas.com) for advice specific to your type of analysis.