CUSTOMER SUPPORT NOTE

Connectivity – How to Spot Problems and Fix Them

Note Number:	**CSN/LUSAS/1036**

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

Connectivity problems are a common source of modelling errors in finite element analysis (FEA). These issues can lead to unintended structural behaviour, and often result in warnings or errors during the solution process, particularly related to the condition of the stiffness matrix.

In FEA, elements are typically connected by sharing nodes at their boundaries or corners. For compatible elements, each matching nodal degree of freedom at a shared node is rigidly linked. While other connection techniques – such as slidelines or constraint equations – are useful in specific scenarios, this technical note focuses on verifying basic connectivity based on shared nodes between adjacent elements.

LUSAS is a feature-based Modeller, which means that the geometry of a structure is defined first. LUSAS then generates a mesh of elements and nodes within the geometric envelope. To ensure proper connectivity of the resulting mesh, the following geometric principles must be followed:

- Volumes must share a common surface to be fully connected.
- Surfaces must share a common line to be fully connected.
- Lines must share a common point to be fully connected.

Connectivity can therefore be verified at both the geometry level and the mesh level.

2. Assessing connectivity

2.1 Checking connectivity

This section outlines various tools and methods available for checking connectivity. For additional information, refer to the following section in the Help documentation:

Help menu > **Help Topics** > **Contents** > **Modeller Reference Manual** > **Chapter 7 – Analyses** > **Pre-Analysis Checks**

2.1.1 Checking higher order features with datatips

Higher-order features of a selected item can be quickly viewed by hovering the cursor over it in the graphics window, which will display a datatip. Examples:

1) For a node, the datatip shows the coordinates and a list of elements connected to that node.
2) For a point, it displays the coordinates, any assignments, and the higher-order lines associated with it.
3) To verify that two adjacent surfaces are connected along a shared line, select the line and check the datatip. It should list both surfaces as being defined by that line.

2.1.2 Selected items and cyclable items

To view the items currently selected in the model, you can open the Selected Items window. This should be visible by default, but if not, it can be accessed by navigating to:

View menu > Selection Panels > Selection

A window will appear on the screen which will list all currently selected items (Figure 1).

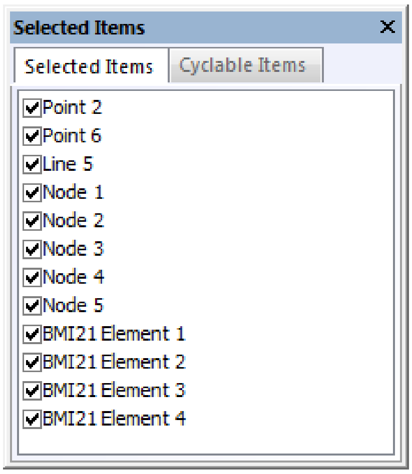


Figure 1 – Selected items

This tool can be useful for several tasks, including:

- **Checking mesh connectivity** – For example, you can box-select an area that should contain a single node or point to verify that no duplicates exist.
- **Verifying imported geometry** – Box-select areas where only one line or surface should exist to ensure no duplicates are present.
- **Identifying element types** – Select an element and view its type in the selection list.
- **Refining selections** – Roughly select an area, then deselect unwanted feature types or specific items to fine-tune your selection.

This window can be docked within the Modeller window frame if you want it to remain permanently visible.

Figure 2 demonstrates how the *Selected Items* window can be used to assess connectivity at the intersection of four lines. In Figure 2 a), a single point is present at the intersection, indicating that all four lines are properly connected through one node. In Figure 2 b), however, there are four coincident, unmerged points at the same location. This means the lines are not connected, and when the model is meshed, it will result in multiple coincident nodes, which can cause issues in the analysis.

a) b)

Figure 2 – a) A single point at the intersection of four lines and b) Four coincident, unmerged points at the intersection of four lines.

Similarly, a *Cyclable Items* window is also available. It can be accessed via the View menu (View menu > Selection Panels > Cyclable Items). This window displays a list of items that are in close proximity to those already selected and could potentially be added to the selection.

Like other panels, the *Cyclable Items* window can be docked within the Modeller window frame. To quickly dock or undock it, simply double-click on the window's border.

2.1.3 Viewing mesh as outline plot

Turning off the Geometry and other layers, and enabling the *Outline only* option in the *Mesh layer* properties, can help visually identify disconnected surface or volume mesh. To access the *Mesh layer* properties, either right-click on the Mesh layer in the Layers tab of the Tree View and select Properties from the context menu, or simply double-click the Mesh layer to open its properties directly (Figure 3).

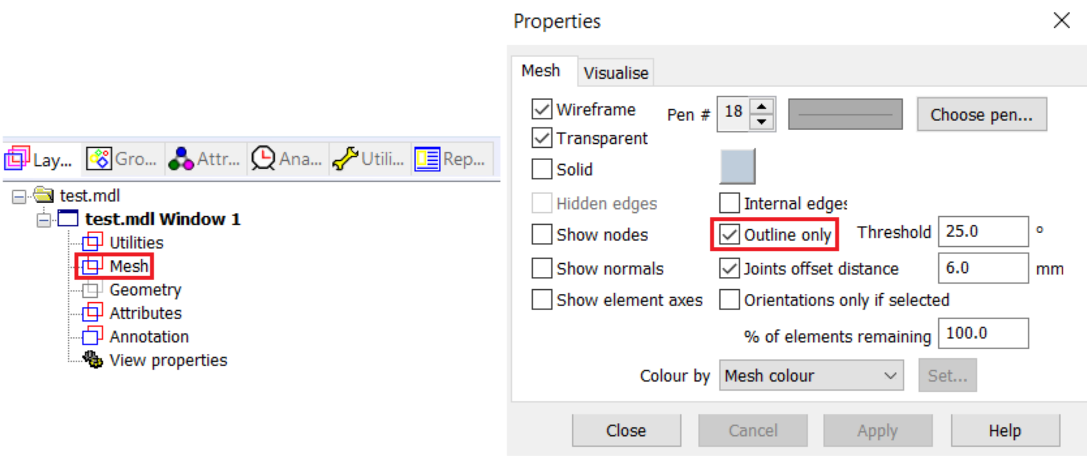
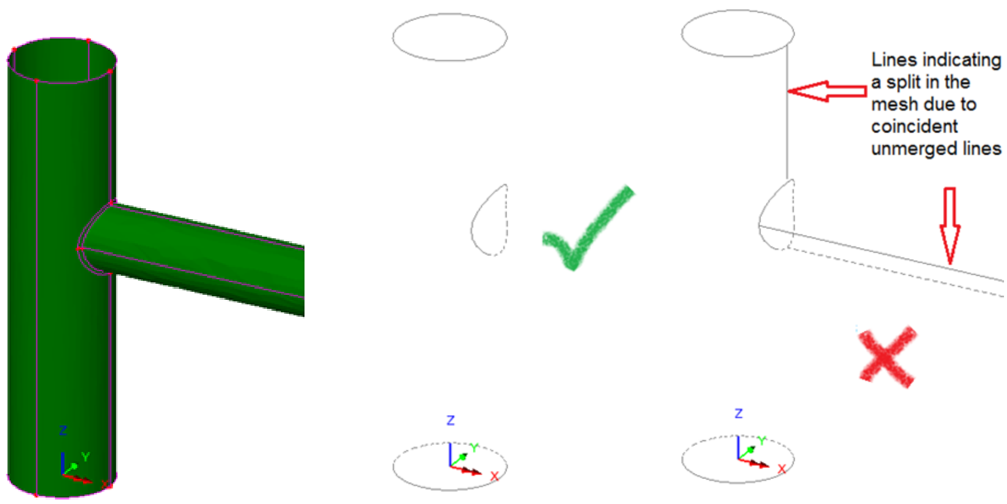


Figure 3 – Mesh layer properties – Outline only option.

The mesh outline plot displays lines at mesh boundaries, splits, creases, joins, and other significant changes in angle, as well as the positions of bar or beam elements. In the example below (Figure 4), the geometry is first shown shaded in solid. This is followed by two mesh outline plots: one illustrating proper connectivity, and another highlighting issues where coincident, unmerged lines result in a split in the mesh.



a) b) c)

Figure 4 – Mesh outline plot example

2.1.4 Check the deformed mesh

A simple analysis, such as a linear static analysis under self-weight, can be performed as an initial check of the model. After running the analysis, the deformed mesh can be viewed with a specified deformation scale factor to visually assess the model's behaviour.

To view the deformed mesh:

View menu > Drawing Layer > Deformed Mesh

If any parts of the model are completely disconnected or lack support, they will appear noticeably displaced due to free body motion, clearly indicating issues with connectivity or boundary conditions.

Figure 5 displays the deformed shape of an I-girder, clearly indicating that one of the stiffeners is not adequately connected to the girder. This insufficient connection results in excessive displacements and an unrealistic deformation, highlighting the need to revise the model and correct the connectivity issue.

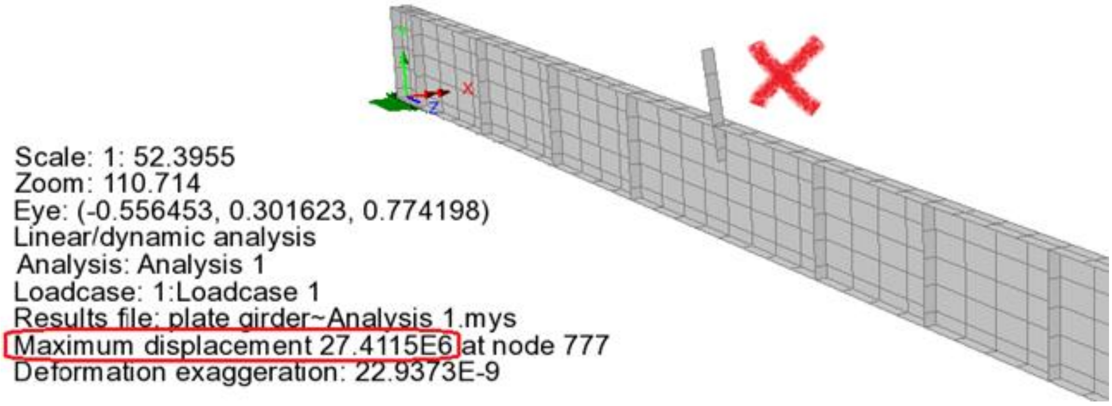


Figure 5 – Deformed mesh check – Disconnected stiffener.

For partially connected components, viewing the deformed mesh at a specified deformation scale can reveal mesh discontinuities or unexpected deformations caused by connectivity issues. Figure 6 illustrates a case where a stiffener is connected only to the flanges of the I-girder but remains disconnected from its web.

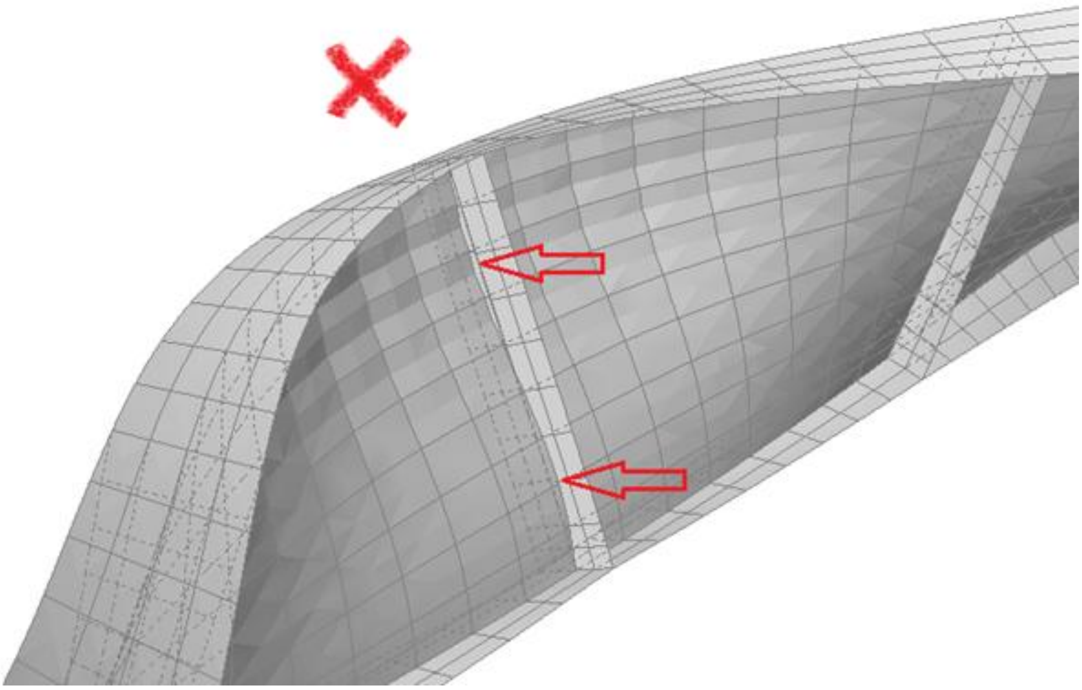


Figure 6 – Deformed mesh check – Stiffener not connected to the web.

2.1.5 Geometry layer – Colour by connectivity

In the *Properties* dialog of the *Geometry* layer, there is a setting called "Colour by Line/Surface Connectivity" that visually highlights connectivity issues through colour coding. This feature provides the following options:

- Line Connectivity – Colours the points based on the number of lines connected to each point, and colours the lines according to the number of surfaces connected to each line.
- Surface Connectivity – Colours the surfaces according to the number of volumes connected to each surface.

For further information, please consult the following references:

Help menu > Help Topics > Contents > Modeller Reference Manual>Chapter 2 - Using Modeller > Using Layers

Help menu > Help Topics > Contents > Modeller Reference Manual>Chapter 4 - Model Geometry > Visualising Geometry

Figure 7 shows an example of a continuous floor slab with beams. The surface geometry has been split to create internal points where column supports below will be modelled. The geometry is displayed with the *Colour by "Line Connectivity"* setting enabled, illustrating two cases: one with good connectivity (b) and one with connectivity issues (c).

In the good connectivity example (b), all perimeter lines are coloured green, indicating – according to the key – that each line defines only one surface, which is correct. The internal lines are coloured blue because they are shared by two adjacent surfaces, reflecting proper connectivity. In contrast, example (c) shows some lines intended to be internal and shared between adjacent surfaces coloured green instead of blue. This means these lines define only a single surface, like the perimeter lines. In other words, the adjacent surfaces have coincident but unmerged lines, indicating a connectivity problem.

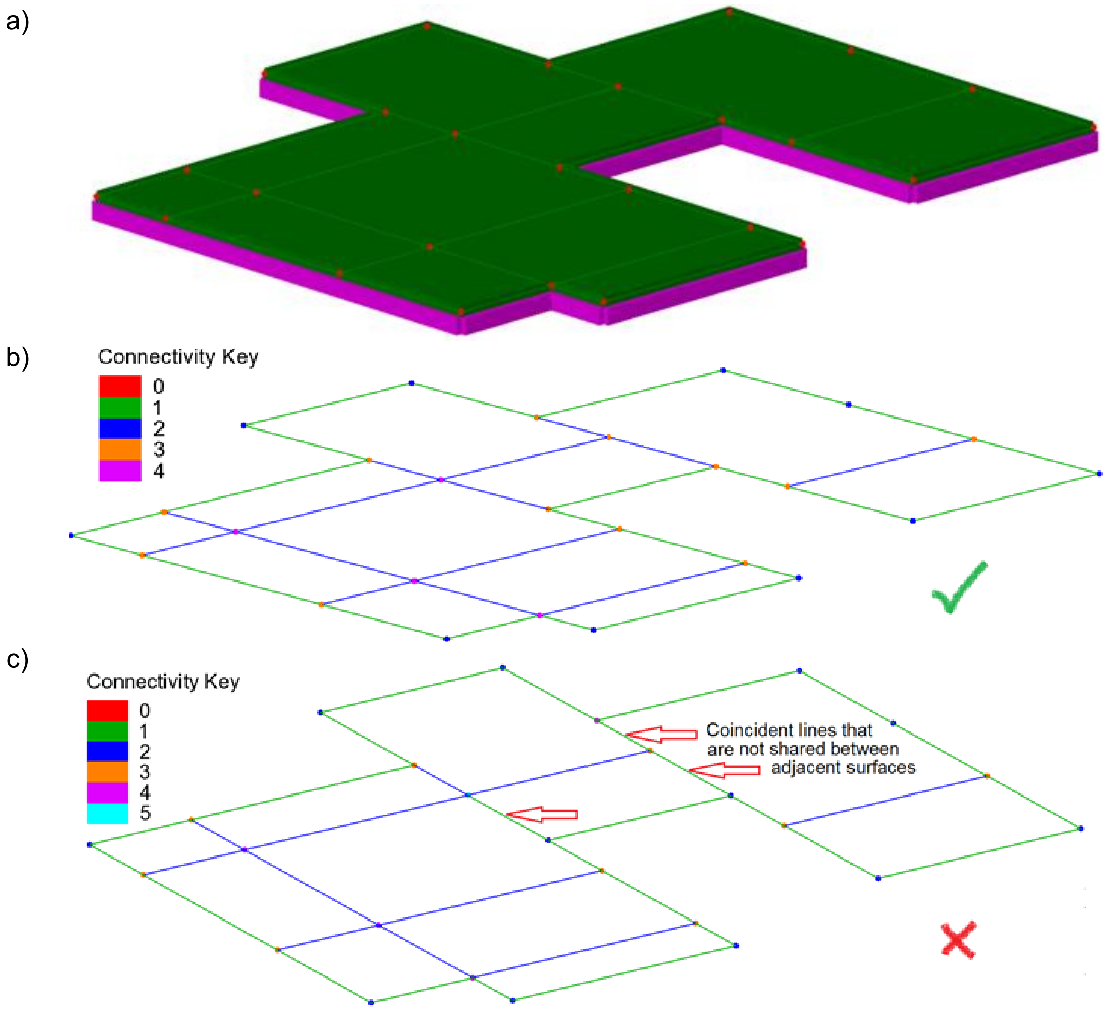


Figure 7 – Colour by line connectivity – Floor slab with beams.

2.1.6 Mesh layer – Colour by connectivity

Similarly, the mesh can be colour-coded to assess connectivity. This can be done by adjusting the *Colour by* setting in the *Properties* dialog of the *Mesh* layer to *Edge Connectivity*. This setting displays how many elements are connected to each node (by colouring the nodes accordingly) and how many elements are connected along each element edge (by applying a corresponding colour code to the edges).

2.1.7 Using the Labels layer

The *Labels* layer is helpful for identifying overlapping geometry – for example, when two points occupy the exact same location instead of just one.

In the example shown in Figure 8, the same problematic geometry from Figure 7 is displayed with labelled lines. It becomes evident that lines 63 and 64, which are longer, overlap with lines 39 and 40, respectively. This overlap is clearly indicated by the positioning of the labels, which are placed at a specific parametric distance along each line.

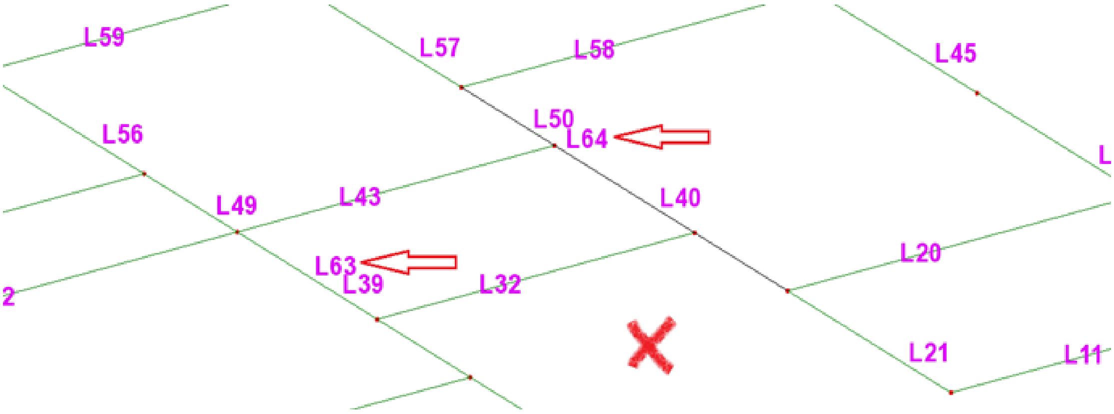


Figure 8 – Using the Labels layer – Coincident/overlapping lines revealed.

2.1.8 Checking the distance between points

To identify coincident points within a small region of the model, select a group of points and then go to:

Geometry menu > Point > Distance Between Points

This function calculates the distances between all pairs of selected points, making it easier to find the two closest points when working with a small selection. For more than six points (resulting in 15 or more pairs), you can copy the output from the text output window into Excel to analyse and identify the closest points more efficiently.

2.1.9 Feature properties – Checking a geometry feature's hierarchy

The properties of a geometry feature – such as a point, line, or surface – can be accessed by selecting the feature of interest, then right-clicking and choosing *Hierarchy* from the context menu.

Within the Hierarchy tab, you can view both the feature's lower-order defining features and higher-order dependent features. This helps confirm that the relationships are as intended. Figure 9 shows the *Hierarchy* tab for a line. This tab lists the lower-order features (points) that define the line, along with the higher-order features (surfaces) that are dependent on it.

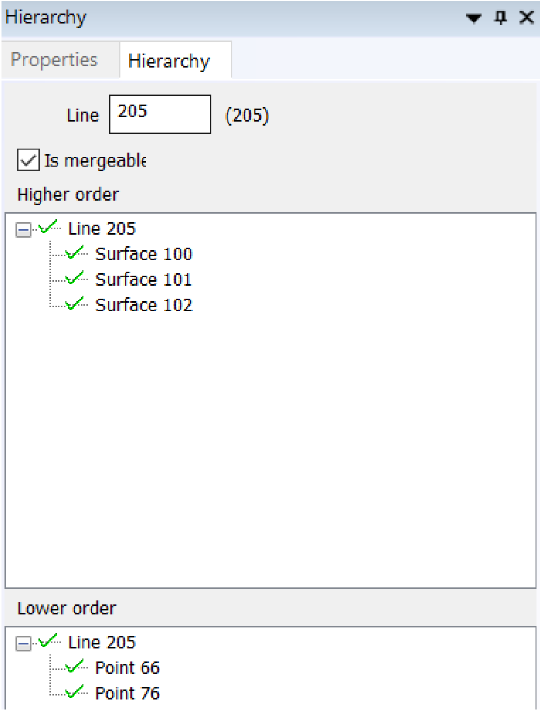


Figure 9 – Geometry feature hierarchy

2.1.10 Using Advanced Selection

Advanced Selection allows the current selection to be filtered by specific criteria such as feature number, element name, stress model, or geometric/material attributes. It can be accessed via Edit menu > Advanced Selection, or by right-clicking in a blank area of the graphics window and selecting it from the context menu.

This tool is particularly useful for tasks like checking the connectivity of coincident points. For example, after selecting one point, you can use *Advanced Selection* with the options *Current selection*, *Add to selection*, and *Also apply to higher order* to identify which higher-order features (e.g., lines or surfaces) are connected to each point.

2.1.11 Using Visibility

Features can be made visible by:

- Selecting *All Visible* from the graphics window context menu (right-click in blank space > All Visible).
- Making a group visible via the *Groups* tab in the Treeview (right-click on a group > Visible or Set as Only Visible).
- Displaying features assigned with a specific attribute from the attribute's context menu in the *Attributes* tab of the Treeview (right-click on an attribute > Visible or Set as Only Visible).
- Using the *Advanced Visibility* dialog, accessible from the graphics window context menu (right-click in blank space > Advanced Visibility).

The *Advanced Visibility* dialog provides fine control over which features are visible by managing the visibility of both higher-order and lower-order features. For example, it allows all lines connected to a point to be shown or all surfaces linked to a line to be hidden.

Let's now examine the connectivity issue shown in Figure 8. Line 64 can be isolated by setting it as the only visible feature on screen. Once isolated, it can be selected along with its end points (points 73 and 29). By using the *Advanced Visibility* tool, both higher-order and lower-order features can be displayed, as illustrated in Figure 10.

Only surface 20 is explicitly defined by line 64. However, surface 16 is also shown, along with its defining line 50, which partially overlaps line 64. This overlap occurs because lines 50 and 64 share point 73. In a correctly connected model, line 50 should be shared by both surfaces 16 and 20.

a)

b)

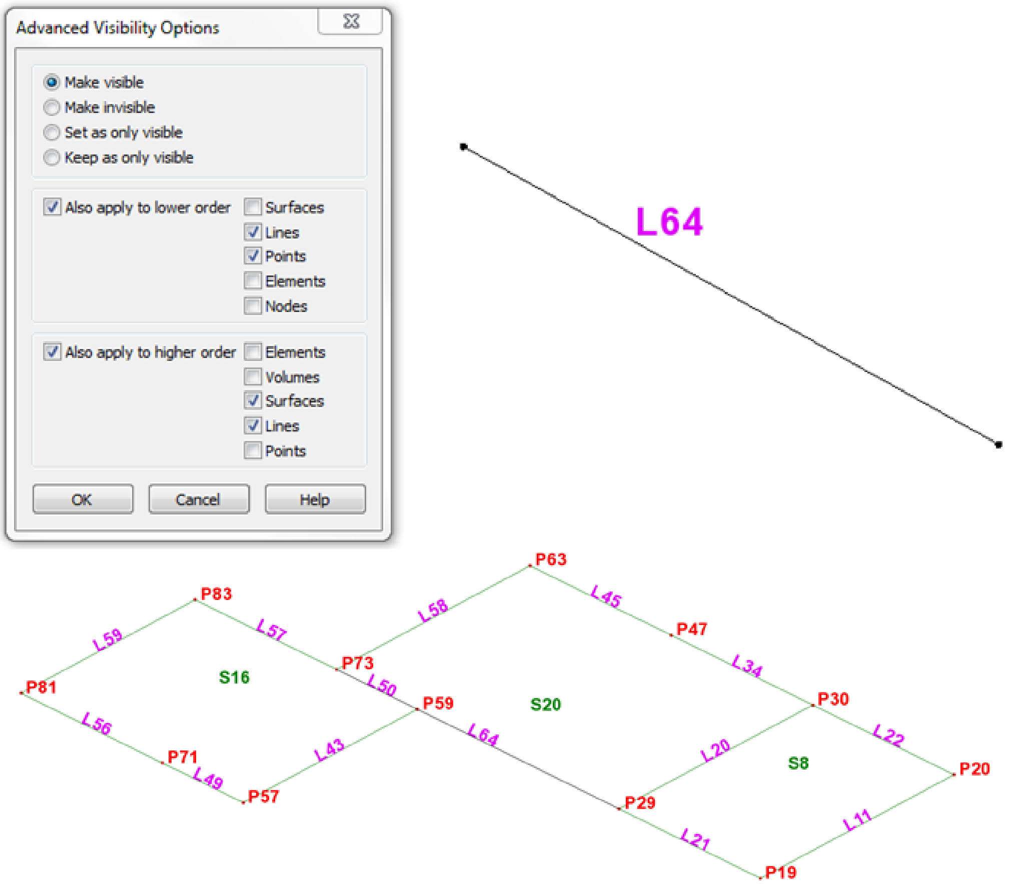


Figure 10 – Using Advanced Visibility to check connectivity.

2.2 Correcting connectivity problems

This section outlines approaches for resolving the connectivity issues identified using the methods described earlier in this technical note.

2.2.1 Merging coincident features

By default, coincident features will merge automatically. Alternatively, you can merge them manually by selecting the features and navigating to:

Geometry menu > Point/Line/Surface > Make Mergeable, followed by Merge.

You can control how features merge by adjusting the *Merge* settings in:

File menu > Model Properties > Geometry.

The available *Merge Action* options are:

- **Off** – Where no merging is carried out.
- **Exact (default)** – Where features are merged only if all assignments are identical. This is the default.
- **Wild** – Where features are merged if feature assignments of the same type for both features match. The assignments of both features are retained where the assignment type is unique to one feature.
- **Ignore assignment** – Ignores the assignments when deciding if two objects should merge (this is the opposite of "Exact" where the two objects must have the same

assignments to merge). The assignments of the feature merged out will be transferred to the feature retained unless the retained feature already has that particular assignment.

For more information, refer to:

Help menu > Help Topics > Contents > Modeller Reference Manual > Chapter 4 - Model Geometry > Merging and Unmerging Features

To ensure that geometry retains the correct assignments after merging, it is advisable to:

- Allow merging with the Merge Action set to "Exact".
- Assign attributes to all intended geometry after merging.
- Alternatively, construct and merge geometry before assigning attributes. This approach helps maintain clarity during model construction and avoids unintentional merging issues caused by differing assignments.

2.2.2 CAD export settings

Model geometry and other associated data can be imported into Modeller via the File > Import menu. Supported file formats include:

- CAD files: *.igs, *.dxf, *.step, *.stp, *.stl
- BIM/BrIM files: *.ifc
- FEA proprietary formats: *.e2k, *.mct, *.mgt, *.s2k

For a complete list of supported file types and an explanation of the advanced import options, refer to the LUSAS documentation:

Help menu > Help Topics > Contents > Modeller Reference Manual > Chapter 3 - File Types/Importing/Exporting > Importing/Exporting Data > Interface Files

When importing geometry from an IGES file, it is possible that some lines defining the edges of surfaces may not be merged. This can result in adjacent surfaces not sharing a common edge, even though they appear to touch. This issue typically arises because the lines are not identical – often due to the modelling tolerance used when the geometry was created and exported from the CAD package.

IGES files define surfaces individually, each with their own boundary lines. If these lines are not within the merge tolerance defined in Modeller, they will remain unmerged upon import. In many cases, it is not possible to merge them afterward, because the surface mapping generated in the CAD system is locked and cannot be modified within Modeller (see further discussion below).

It’s important to note that the level of precision required for finite element analysis (FEA) is significantly greater than that required for typical CAD modelling. CAD systems generally apply a coarser default modelling tolerance, which may not be suitable for FEA purposes. To ensure successful import and accurate geometry merging, you should set the modelling tolerance in the CAD package before creating and exporting the geometry. This tolerance should be as tight as reasonably possible – ideally matching or closely approximating the default Modeller merge tolerance of 1.0E-6, which can be accessed via:

File menu > Model Properties... > Geometry tab

If a CAD model is created using a modelling tolerance of, for example, 0.01, then lines that are intended to be coincident – such as those defining the shared edge of two surfaces – may actually be offset by a distance such as 0.001. When this geometry is imported into LUSAS, which uses a default merge tolerance of 1.0E-6, such lines will not be merged because they fall outside the permitted threshold. This issue is especially pronounced with polylines, where curves are approximated based on the tolerance used during their creation. To avoid such discrepancies, it is recommended to set a tighter modelling tolerance—ideally 1.0E-6—in the CAD software prior to creating the geometry. Doing so ensures the exported geometry meets the precision requirements needed for successful merging in Modeller.

CAD software can also define surface mapping using NURBS surfaces. While LUSAS is capable of importing these surfaces and their associated mapping, it cannot modify them. This limitation exists because Modeller does not include the full range of surface editing tools available in CAD packages. Therefore, any adjustments to NURBS geometry or surface mapping must be made within the original CAD environment.

Depending on the complexity of the required model, the geometry creation tools provided within LUSAS may be entirely sufficient, potentially eliminating the need to use a CAD package for geometry creation and import.

Cylindrical or spherical surfaces created in a CAD package by sweeping a circle can result in a closed surface with a single 'seam'. When such geometry is imported into LUSAS, the surface mapping may not be interpreted correctly, as the location of the seam can cause ambiguity in determining which side the mapping should reference. This can lead to incorrect surface representations.

To avoid these issues, the following approaches are recommended:

- Enable "Split Closed Surfaces" on IGES Export: Many CAD packages (e.g., Rhino) provide an option to split closed surfaces during IGES file export. Enabling this option divides the surface into two parts, introducing an additional seam that helps resolve mapping ambiguities during import. This is particularly useful not only for basic shapes like cylinders and spheres, but also for more complex closed surfaces that are difficult to split manually in the CAD environment.
- Manually Split Surfaces in CAD Prior to Export:
    - For cylinders, instead of sweeping a full circle, create the surface by sweeping two semi-circles, which introduces two seams. For example, a cylindrical volume could be defined by extruding two semi-circular profiles to form two separate half-cylinder volumes, which can then be imported with improved surface mapping reliability.
    - For spheres, perform four 90-degree sweeps of a semi-circle to generate four seams.

2.2.3 Ensuring elements sharing nodes have compatible nodal freedoms

For guidance on ensuring that elements sharing nodes have compatible nodal freedoms, refer to the following resource:

http://www.lusas.com/protected/theory/connecting_elements.html

Additionally, a detailed summary of the nodal freedoms and other properties for each element available in LUSAS can be found in the software documentation under:

Help menu > Help Topics > Contents > Element Reference Manual > Element Summary Tables

2.2.4 Splitting geometry

In certain cases, splitting geometry can help resolve connectivity issues caused by overlapping features. A common example is when two lines overlap: the issue can often be resolved by splitting one of the lines at the point of overlap. To do this, select the line and a point located along its length, then use the following menu option to divide the line into two segments at that point:

Geometry menu > Line > By Splitting > At a Point…

As shown in Figure 11a, lines 64 and 50 overlap. To resolve this, line 64 can be split at point 59, creating a new line between points 59 and 73. This new line will be exactly coincident with line 50, allowing the two to be merged. Once merged, line 50 serves as the shared edge between surfaces 16 and 20. The updated geometry is illustrated in Figure 11b. In this configuration, line 64 is deleted, and a shorter line – line 65 – is created between points 29 and 59. In many cases, additional adjustments may be necessary to ensure that newly created features are fully integrated with the existing geometry.

a)

b)

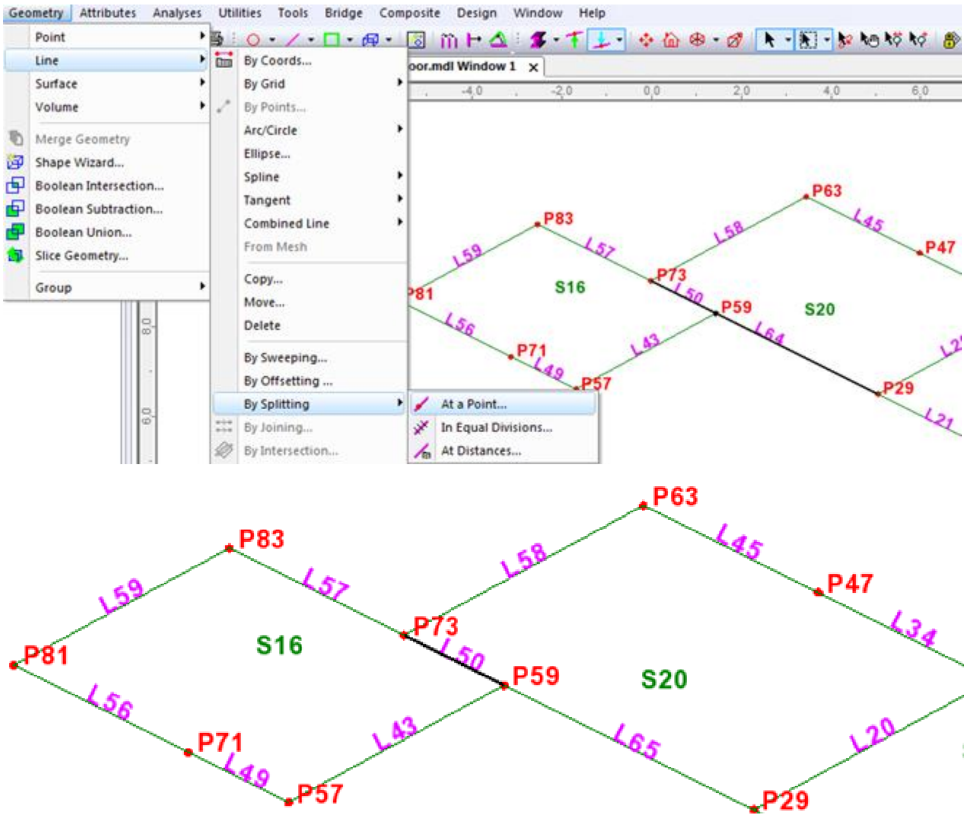


Figure 11 – Using the By Splitting option to resolve connectivity issues caused by overlapping lines.

Similarly, a surface can be split using a line. To do this, select both the surface and a line that fully spans across it, then use the following menu option to divide the surface into two along that line:

Geometry menu > Surface > By Splitting > By Line…

For additional tools and techniques related to splitting lines, surfaces, or volumes, refer to the documentation:

Help menu > Help Topics > Contents > Modeller Reference Manual > Chapter 4 – Model Geometry > Points/Lines/Surfaces/Volumes

2.2.5 Redefining geometry

Depending on the complexity of the model and the extent of the connectivity issues identified, the analyst may determine that redefining the geometry in Modeller is a more practical and time-efficient solution than attempting to fix the issues directly. When redefining geometry or creating it from scratch, the following recommendations should be considered:

- **Complete all geometry import, manipulation, and editing before making any assignments (including mesh), where possible**. This is because features are only merged by default if they share identical assignments. While it may be useful to test meshing during this stage, it is recommended to deassign the mesh before performing further geometry operations – such as splitting surfaces – to help ensure proper connectivity.
- **Where possible, define all surfaces using only 3 or 4 lines**. Such surfaces are known as regular surfaces and are well-suited for grid-like shell meshing. They are easier to manipulate, modify, and refine, making them ideal for achieving a high-quality mesh. In contrast, non-planar surfaces defined by 5 or more lines – often encountered in CAD imports as complex NURBS surfaces – cannot be easily edited by moving their defining lines or points, as the internal surface shape becomes indeterminate.

- **Carefully plan the geometry based on the features to be modelled, the necessary assignments, the desired results, and areas where mesh control or refinement may be required**. Investing time upfront to create the correct geometry is far more efficient than making extensive edits later in the modelling process. Surfaces should be drawn so that key lines and points are positioned where needed – for example, at member intersections, load and support locations, result extraction points, mesh refinement zones, or where constraint equations or contact definitions will be applied. Creating geometry with these considerations in mind reduces the need for retrospective splitting and helps prevent common modelling issues such as connectivity errors or incorrect assignments. Keep in mind that for surfaces to be fully connected, they must share common lines; for lines to be fully connected, they must share common points. Fully connected geometry ensures that the resulting mesh is continuous, with adjacent elements sharing common nodes.
- **Use the Move, Copy, and Sweep commands to build geometry progressively**. Connectivity is more reliably achieved when geometry is created through controlled operations – for example, sweeping a point to create a line, sweeping a line to create a surface, and then copying or sweeping that surface to form additional surfaces or volumes. This method helps maintain geometric precision, reduces the likelihood of tolerance-related errors, and minimises the creation of coincident or non-merging features.

Note: Combined lines cannot be deleted directly if they are already part of a surface definition – the associated surface must be deleted first. When redefining surfaces composed of more than four lines, any combined lines may also need to be removed, especially if the surface is to be replaced with two or more regular surfaces defined by only 3 or 4 lines. For more details, refer to:

Help menu > Help Topics > Contents > Modeller Reference Manual > Chapter 4 – Model Geometry > Combined Lines

Using scripting in LUSAS is highly effective for automating repetitive modelling tasks. A collection of example scripts is available for download from our User Area:

http://www.lusas.com/protected/download/scripts.html

For instance, the script "**Create Multiple Surfaces from a Large Selection of Lines**" automates the process of redefining multiple surfaces efficiently. Other useful scripts include "**Check and Detect Short Lines"** and **"Select Nearby Features**", both of which can assist with identifying and resolving connectivity issues.

3. Discussion

Connectivity problems are a common source of modelling errors in finite element analysis (FEA). These issues can lead to unintended structural behaviour, and often result in warnings or errors during the solution process, particularly related to the condition of the stiffness matrix.

As one might expect, investing time upfront to create accurate and well-structured geometry is far more efficient than making extensive edits later in the modelling process. However, it is still essential to check all models to ensure that no connectivity issues are present. Section 2.1 introduces tools and techniques for identifying such issues, while Section 2.2 outlines methods for resolving them.

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.