CUSTOMER SUPPORT NOTE

Connectivity - How to Spot Problems and Fix Them

This support note is issued as a guideline only.

Table of Contents

1. Introduction

Connectivity problems are a common type of modelling error in finite element analyses, often leading to unintended structural behaviour in a model and causing warnings and errors in the solution regarding the condition of the stiffness matrix.

In finite element analysis, elements are principally connected by sharing common nodes at their boundaries or corners. Each matching nodal degree of freedom for compatible elements at a shared node is rigidly linked. Whilst there are other valid methods of connecting elements (such as slidelines and constraint equations), this technical note looks at the different methods of checking a model for fundamental connectivity in terms of shared nodes between adjacent elements.

LUSAS is a feature based Modeller; meaning that we define the geometry of the structure first and then allow LUSAS to create a mesh of elements and nodes within the enveloped form that the geometry defines. Therefore to ensure connectivity of the underlying mesh, the following principles apply to the geometry features:

* For volumes to be fully connected they must share a common surface
* For surfaces to be fully connected they must share a common line
* For lines to be fully connected they must share a common point

Connectivity checks can therefore be done at the geometry level as well as at the mesh level.

2. Description

2.1 Checking Connectivity

In this section different tools and methods for checking connectivity are detailed.

Please also see the following section in the Help:

> **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 checked quickly by hovering the cursor over the selected item in the graphics window and a datatip will appear. For a node for example, a list of elements connected at that node will be given as well as its coordinates. For a point, the higher order lines will be listed along with the coordinates and any assignments. To check that two adjacent surfaces are connected at a shared line, the line can be selected and the datatip can be checked as listing the two surfaces as being defined by the line.

2.1.2 Selected Items and Cyclable Items

Selected Items and **Cyclable Items** windows can be used to interrogate regions of geometry and mesh selected. To check the items that are in the current selection, a window can be displayed showing the **Selected Items**. This can be accessed via:

> **View** menu by selecting **Browse Selection...** (or **Selection Panels** and then **Selection** in V16 onwards)

A window will appear on the screen which will list all currently selected items (as shown in the image below).
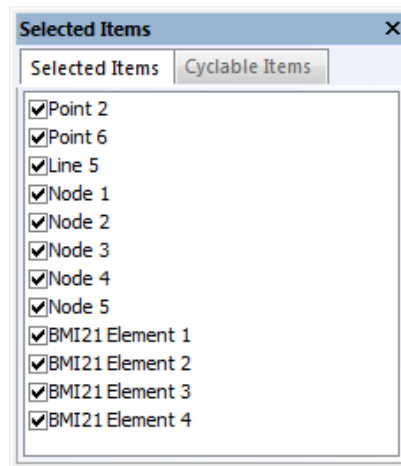
Figure 1: Selected Items

This tool can be useful to:

- Check mesh connectivity by box selecting an area that should contain one node/point (below left) and to check that there is not more than one (below right)
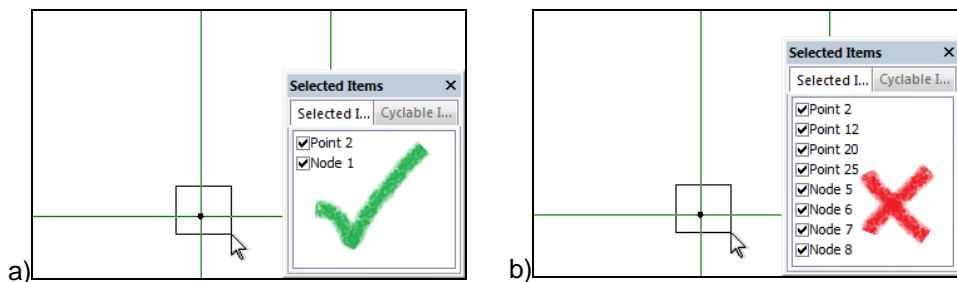

Figure 2: Selected Items Example: Single Point Shared between Four Intersecting Lines.

(a*) Elements connected sharing a single node. (b) Four coincident unmerged points and consequently four adjacent, but disconnected elements each with a separate node.*

- Check imported geometry by box selecting an area where there should be only one line or surface for example and checking that there is not more than one
- Select an element and check its type
- Approximately select an area and remove unwanted feature types or a few specific items from the selection

This window can be 'docked' in the Modeller window frame if it is to be made permanently visible.

Similarly a **Cyclable Items** window is also available. This can be accessed from the **View** menu by selecting **Browse Cyclable Items...** (or **Selection Panels** and then **Cyclable Items** in V16 onwards). This allows the user to see a list of items that could be added to the selection that are in close proximity to those already selected. This window can also be 'docked' in the window frame. To quickly dock or undock these windows in the Modeller window frame, double-click on their border.

2.1.3 Viewing the Mesh as an Outline Plot

Turning off the **Geometry** layer and other layers and then viewing the **Mesh** layer as an **Outline only** set in the **Mesh** layer **Properties** can help to identify disconnected surface or volume mesh.

To get to the **Properties** of the **Mesh** layer, right-click on **Mesh** layer in the **Layers** tab of the **Tree View** and choose **Properties** in the context menu, or simply double-click on the **Mesh** layer to go straight to its **Properties**.

The mesh outline plot will show lines at the mesh boundaries, splits, creases, joins and other significant changes of angle, as well as the locations of bar or beam elements. The example below shows some geometry shaded solid and then mesh outline plots are shown for the case where the connectivity is good and for a case where there are some coincident unmerged lines and hence a split in the mesh.
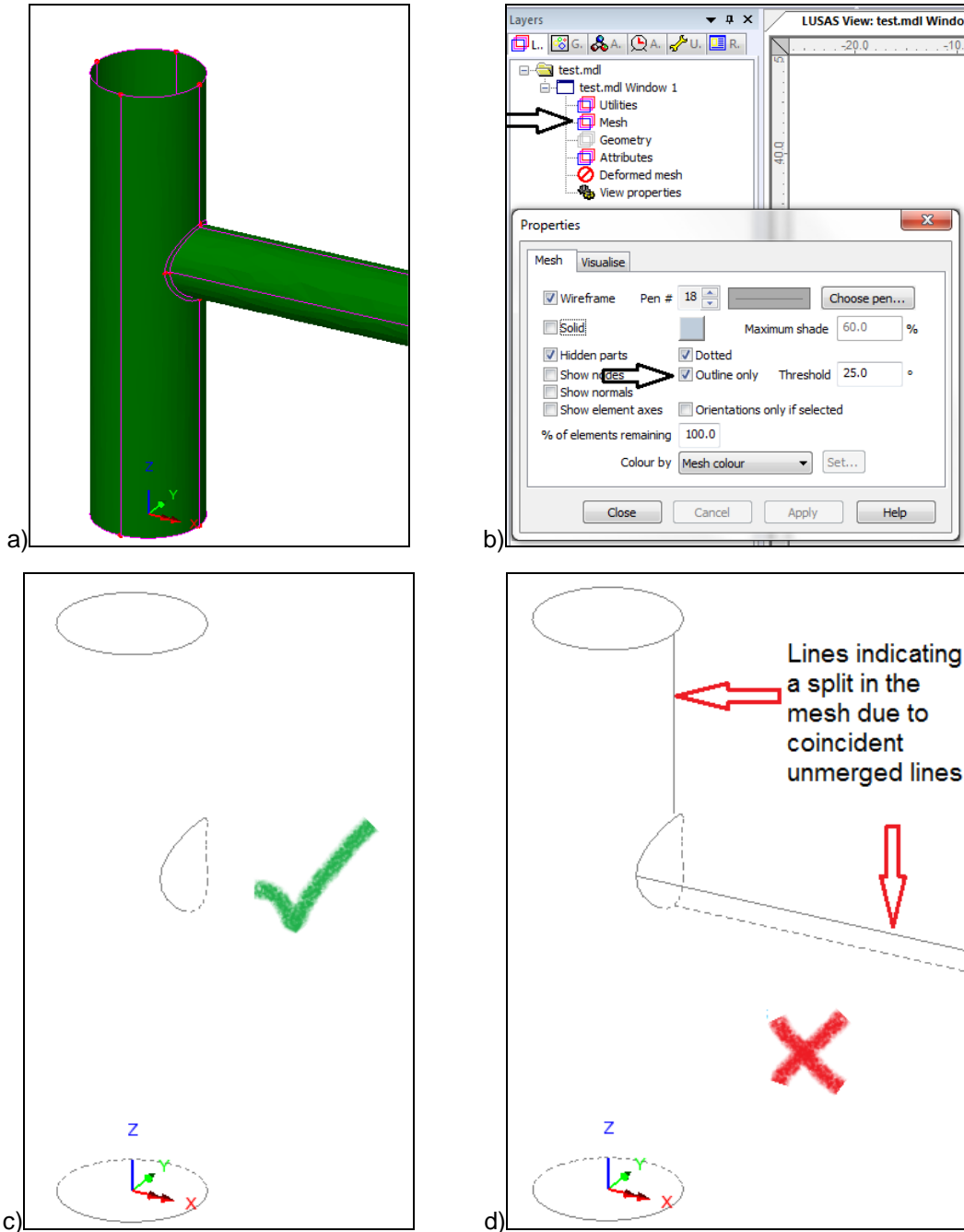
Figure 3: Mesh Outline Plot Example

(a) Geometry coloured "Solid". (b) Mesh layer Properties for a mesh outline plot. (c) Mesh outline plot showing good connectivity. (d) Further mesh outline plot revealing disconnected elements along cylinder seams

2.1.4 Check the Deformed Mesh

A simple analysis of the model can be run as an initial check, for example a linear static analysis for self weight and then the **Deformed Mesh** can be inspected with a specified factor of deformation.

> **View** menu **> Drawing Layer > Deformed Mesh**

If parts of the model are totally disconnected and unsupported, they will be displayed evidently having been displaced from the model under free body motion (see the image below).

Scale: 1: 52.3955
Zoom: 110.714
Eye: (-0.556453, 0.301623, 0.774198)
Linear/dynamic analysis
Analysis: Analysis 1
Loadcase: 1:Loadcase 1
Results file: plate girder~Analysis 1.mys
Maximum displacement 27.4115E6 at node 777
Deformation exaggeration: 22.9373E-9
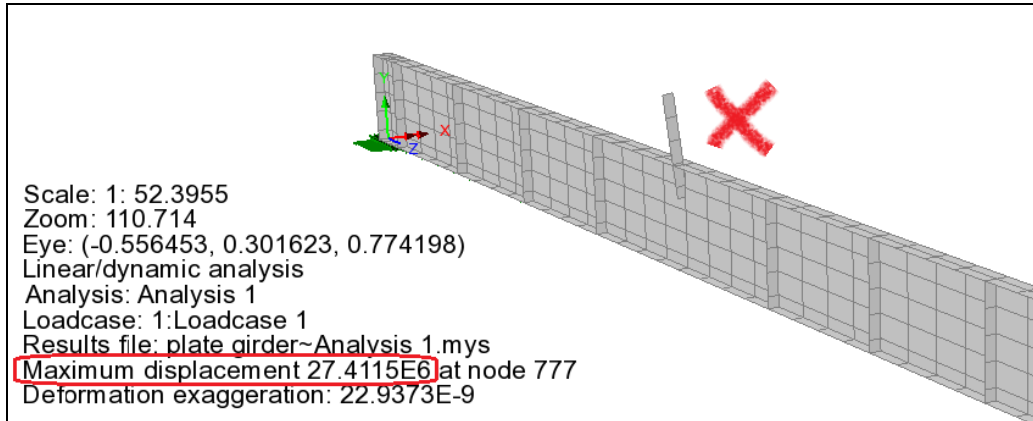
Figure 4 Deformed Mesh Check:

A disconnected stiffener behaves like a free body with excessive displacement in a simple linear static analysis.

For partially connected parts of the model, the **Deformed Mesh** viewed for a specified magnitude will reveal splits in the mesh or unexpected deformations due to connectivity problems.

Figure 5 Deformed Mesh Check:

The web of a plate girder deforms through a stiffener indicating that that stiffeners on each side are only connected to the flanges, but not to the web, nor to each other.

2.1.5 Geometry Layer – Colour by Connectivity

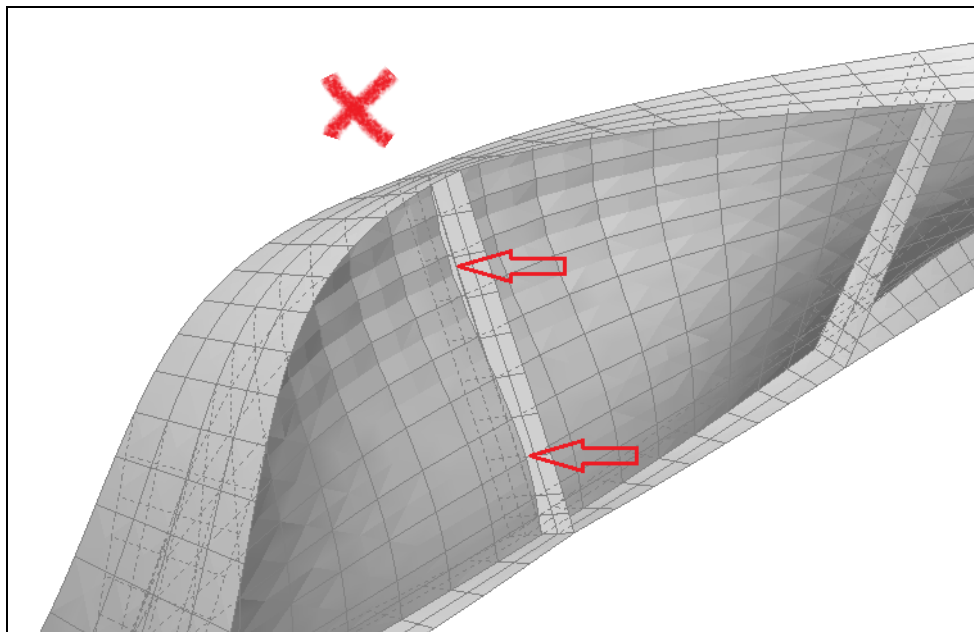In the **Properties** of the **Geometry** layer there is a setting to **Colour by "Line/Surface Connectivity"** which will show the following information:

i) How many lines are connected to each point by colour coding the points ("**Line/Surface Connectivity"** options)

ii) How many surfaces are connected to each line by colour coding the lines with the same key ("**Line Connectivity"** option)

iii) How many volumes are connected to each surface by colour coding the surfaces ("**Surface Connectivity"** option)

Please see the following references for further information:

> **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**

The example shown in Figure 6 below is a continuous floor slab and beams. The surface geometry has been split to achieve internal points where support from columns below will be modelled. The geometry has been set to **Colour by** and **"Line Connectivity"** with one example of good connectivity (b) and one example of disconnectivity (c) given. In the good example all perimeter lines are coloured green, which the key indicates to mean that they define just one surface, which is correct. All internal lines are shared between a pair of adjacent surfaces and are therefore blue because those lines define 2 connected surfaces. In (c), some of the lines that are intended to be internal and shared between adjacent surfaces are in fact green meaning they define only one surface, like the perimeter lines, in other words, lines of the adjacent surfaces are coincident and unmerged.
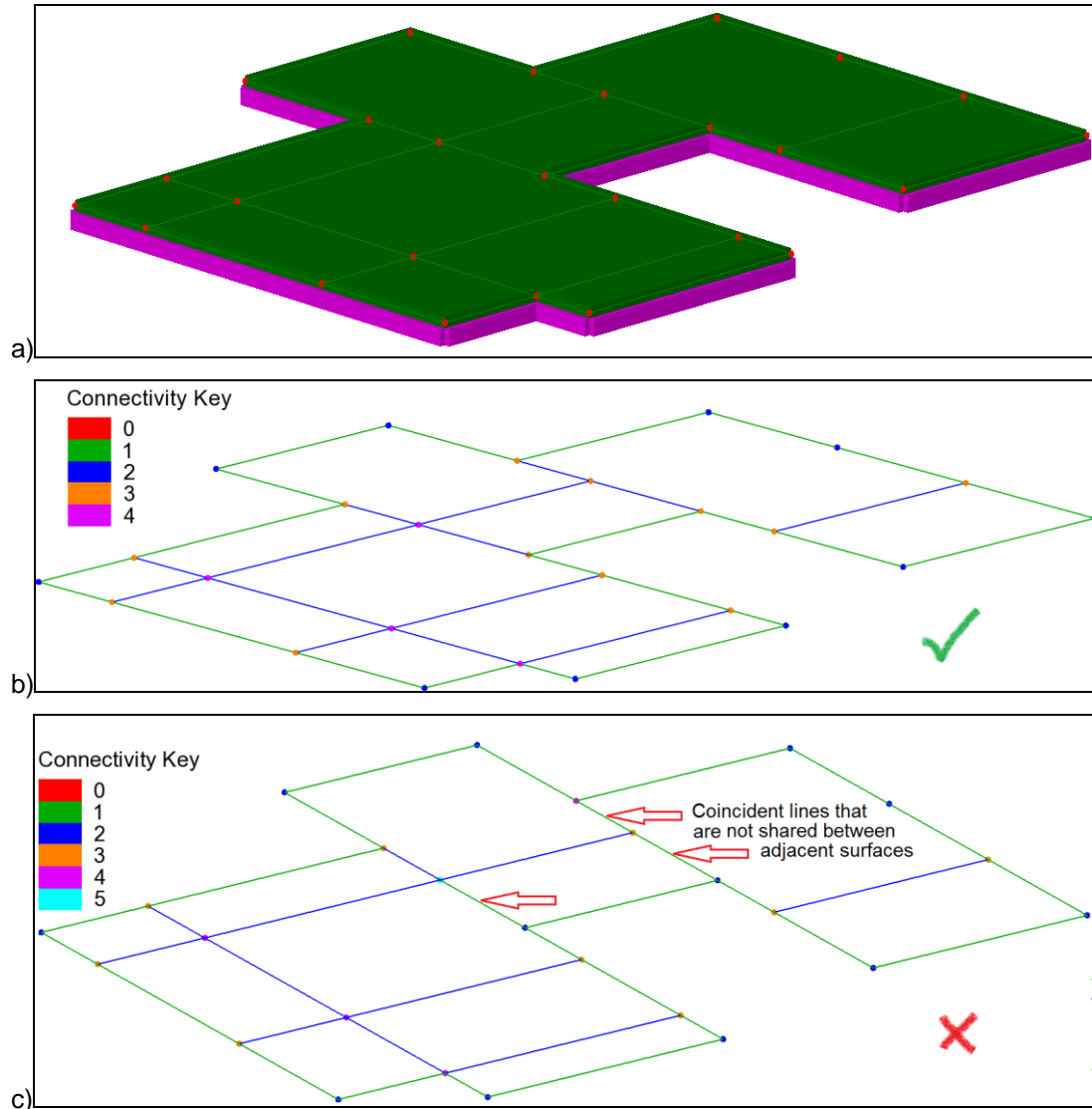
Figure 6 Colour by Connectivity Example: A Floor Slab

a) 'Fleshed' geometry showing slab and beams. b) Geometry layer set to "Colour by" "Line Connectivity" - example with good connections. c) Same example but with connectivity problems due to coincident unmerged lines.

2.1.6 Mesh Layer – Colour by Connectivity

Similarly, the mesh can be coloured to check connectivity. This is set via the **Properties** of the **Mesh** layer using the **Colour by** option set to **"Connectivity"**. This will show how many elements are connected to each node (by colour coding the nodes) and how many elements are connected to each element edge (by colour coding the element edges with the same key).

2.1.7 Using a Labels Layer

The **Labels** layer can be useful for identifying any duplicated or overlapping geometry. For example where two points occupy the same location rather than one.

The example shown in Figure 7 below takes the same problem geometry as shown in Figure 6 c) above and shows lines labeled. We can see that lines 63 and 64 are two longer lines that overlap lines 39 and 40 respectively. This is clear because of the positioning of these labels, which is set at a certain parametric distance along each line.
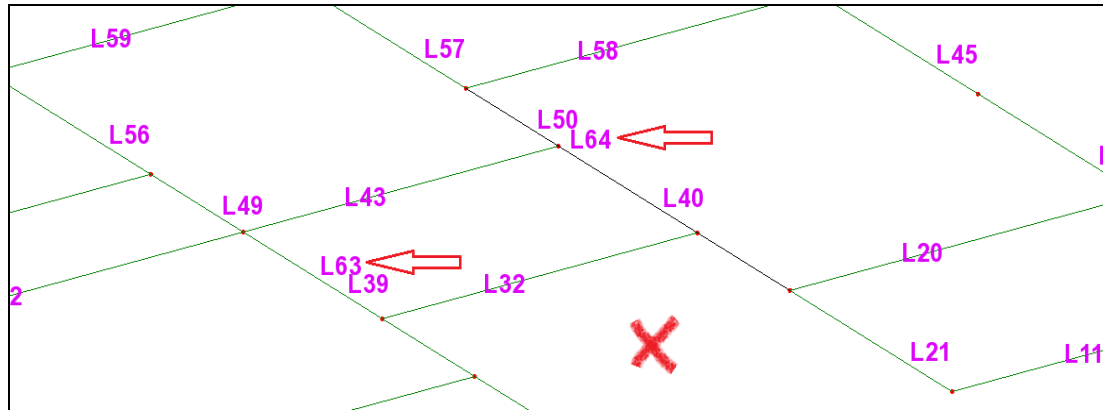

Figure 7 Using a Labels Layer – Coincident/Overlapping Lines Revealed

2.1.8 Checking the Distance between Points

Coincident points within a small region of the model may be identified by selecting a small number of points and then by going to:

Geometry menu **> Point > Distance Between Points**

This will give the distances between each possible pair of points for the selection, which may help in finding the two closest points for a small number of selected points. The output in the text output window could be copied and pasted into Excel and manipulated for anything more than 6 selected points (15 pairs) at a time to help identify which points are closest.

2.1.9 Feature Properties – Checking a Geometry Feature's Hierarchy

The **Properties** of a geometry feature, such as a line for example, can be accessed by selecting a single feature and then right-click and choose **Properties** from the context menu. The **Hierarchy** tab can be viewed to establish a line's lower order points and higher order surfaces. The lowered order defining features and higher order dependent features can be checked as being as intended and also they can be selected using the **Jump to** option (see below).
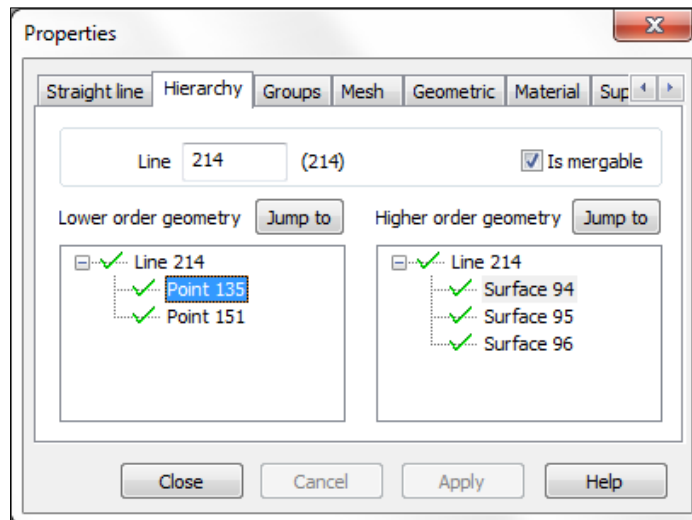
Figure 8 Geometry Feature Properties - Checking the Hierarchy

2.1.10 Using Advanced Selection

Advanced Selection enables the current selection to be filtered for a particular feature number, element name, stress model or geometric or material attribute.

> **Edit** menu > **Advanced Selection**

The **Advanced Selection** dialog may also be activated from the graphics window context menu (by right-clicking in blank space). For checking connectivity of two coincident points, for example, one point could be selected and **Advanced Selection** used to select higher order features (options: **Current selection**, **Add to selection** and **Also apply to higher order**) to see what each point is connected to.

2.1.11 Using Visibility

Features may be made visible by:

- Selecting **All Visible** from the context menu activated from the graphics window (right-click in blank space>**All Visible**)
- Making a group visible from the **Groups** tab of the **Tree View** (right-click on a group>**Visible** or **Set as Only Visible**)
- Making features assigned with a selected attribute visible from an attribute's context menu in the **Attributes** tab of the **Tree View** (right-click on a attribute>**Visible** or **Set as Only Visible**)
- Using the **Advanced Visibility** dialog activated from the graphics window context menu (right-click in blank space **> Advanced Visibility**)

The **Advanced Visibility** dialog allows fine control on the visible items by controlling the visibility of higher order and lower order features. This allows all lines attached to a point to be made visible or all surfaces connected to a line to be made invisible.

Looking at the connectivity problem, illustrated in Figure 6 c), the line 64 could be set as that only visible on the screen and then selected along with its lines and points. **Advanced Visibility** can then be used (as shown in a)) to make higher and lowered order features visible as shown in b). Note that only surface 20, to the right is defined by line 64. Surface 16 is made visible along with its line 50 which partially overlays line 64, because lines 50 and 64 share point 73. We can also see that there is no point where there should be at the end of line 40. Really line 50 should be shared between these two surfaces.
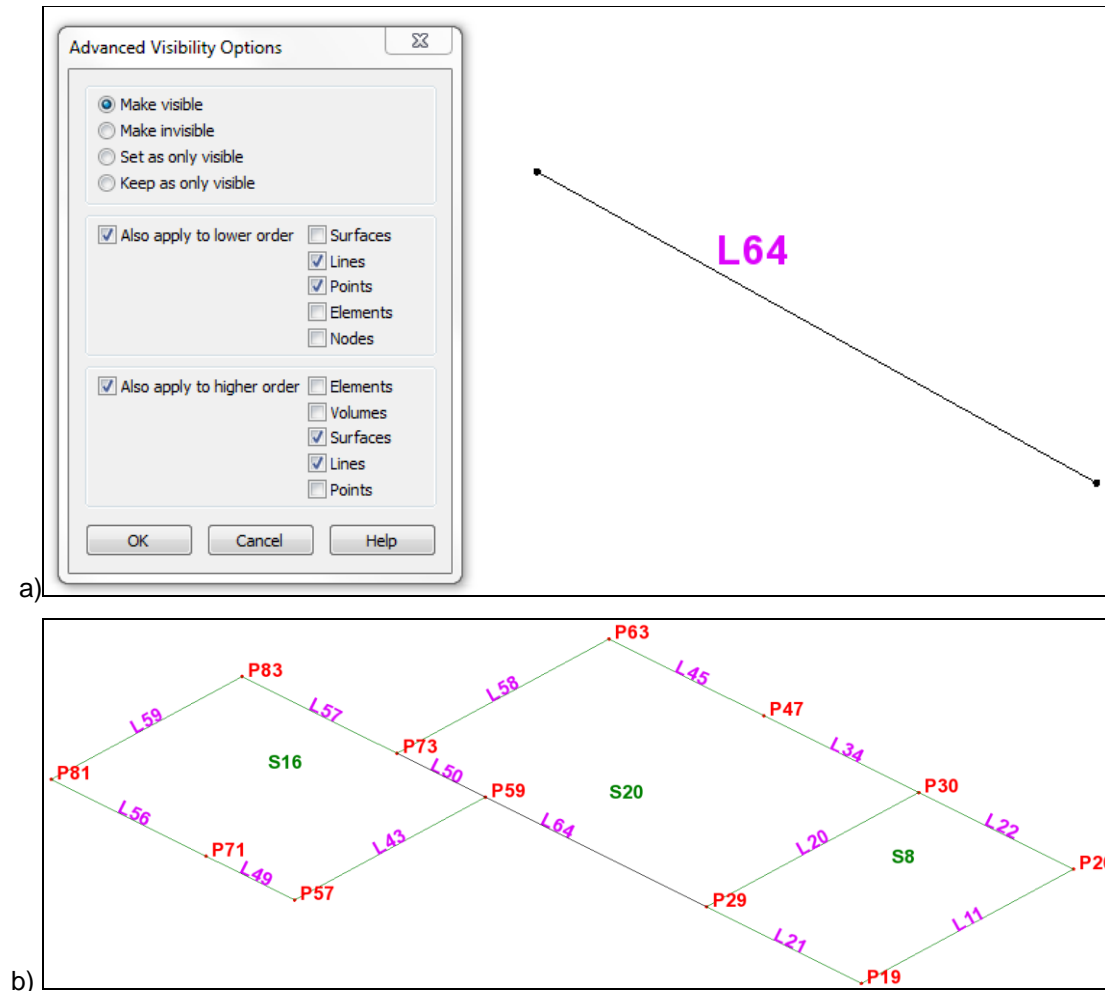
Figure 9 Using Advanced Visibility to Check Geometry Hierarchy Example

a) Line 64 and its end points 73 and 29 are selected and set as that only visible first and then Advanced visibility is used. b) Higher order lines and surfaces of line 64 and its end points made visible.

2.2 Correcting Connectivity Problems

Having used the methods given in the previous section to check connectivity, this section now moves on to look at ways of resolving the connectivity problems found.

2.2.1 Merging Coincident Features

Coincident features will merge automatically by default or can be made to merge by selecting them and going to:

> **Geometry** menu **> Point/Line/Surface > Make Mergable** first, and then **Merge**

The settings for the conditions under which features may merge can be changed via **File** menu **> Model Properties>Geometry**. The **Merge Action** options are as follows:

- **"Off"** - Where no merging is carried out.

- **"Exact"** - 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.

- "I**gnore assignment"** - Assignments where features are merged even if assignments differ. The assignments of the feature merged out will be transferred to the feature retained unless the retained feature already has that particular assignment.

Please see the following reference for further information:

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

To ensure that the assignments intended for the geometry are correct and remain after merging, it is recommended to allow merging with the merge action set to "**Exact**" and to assign attributes to all intended geometry. Geometry can also be constructed and merged as necessary *prior* to the assignment of attributes as a means of maintaining clarity during model construction and avoiding anything not merging due to differing assignments.

2.2.2 CAD Export Settings

Model geometry as well as other model data can be imported into Modeller via **File** menu **> Import** in the form of CAD files (*.igs, *.dxf, *.step, *.stp, *.stl), BIM/BrIM files (*.ifc) for V16 onwards, and a number of proprietary formats for other FEA packages (*.e2k,*.mct,*.mgt,*.s2k),

Please see the following reference for details of the file types that can be imported into LUSAS as well as explanation of the advanced options that are available when importing.

> **Help** menu **> Help Topics > Contents > Modeller Reference Manual > Chapter 3 - File Types > Interface Files**

When geometry form an IGES file is imported for example, it is possible that some lines defining edges of surfaces may not be merged together, causing two touching surfaces to not be joined together by a single common line. The lines may not be identical due to the tolerance that they were created and exported with, which is set in the CAD package that created them. An IGES file defines surfaces, each with separate lines. If these lines are not within the merge tolerance of Modeller, they are not merged on import. It may not then be possible to merge them post import due to the inability to edit the imported surface mapping generated in the CAD package (see further discussion below). The required precision of

geometry defined for an FEA analysis is much finer than that generally required for CAD modelling. Consequently CAD packages tend to have a much coarser default modelling tolerance than an FEA package. In the CAD package used, the modelling tolerance should be set prior to creating and exporting geometry. The tolerance set should be as tight as is reasonable, but ideally comparable to the default merge tolerance in Modeller of 1.0E-6 which can be accessed via:

File menu **> Model Properties... > Geometry** tab

If a CAD model is created with a tolerance of say 0.01, coincident lines defining a common surface edge may be a distance, say 0.001 apart. On importing these lines into LUSAS with a default merge tolerance of 1.0E-6, these lines will not be merged. This particularly affects polylines as the curve is approximated within the tolerance set. Tightening the tolerance in the CAD package prior to geometry creation to 1.0E-6 will ensure that the same tolerance is used to define such lines as required for merging in Modeller.

CAD software can also create surface mapping (nurbs surfaces). LUSAS can import such surfaces and their mapping, but cannot edit this mapping, as it is not equipped with the geometry creation tools of a CAD package. CAD software should be used to create and edit such surfaces. The geometry creation tools available in LUSAS may be considered sufficient to create the geometry required without the need for creating a CAD model for import, depending on the model geometry required.

Cylindrical or spherical surfaces created by sweeping a circle in a CAD package may create a closed surface with one 'seam'. On importing such a sphere or cylinder the surface may not be mapped correctly as it may be unclear which side of the 'seam' mapping should be referenced to. This should be avoided as follows:

- Some CAD packages allow an option to be set on exporting an IGES file to "Split closed surfaces" (this includes Rhino for example). This will divide the surface into two, creating an extra 'seam'. This option should also be used with other closed surfaces that are perhaps more complex than a cylinder or sphere and that are less easily split in two in the CAD package.

- Splitting cylinders by sweeping/extruding two semi-circles instead to create two 'seams' in the CAD package. Split a sphere by sweeping a semi-circle by 90 degrees in four times to create 4 seams. For example a cylindrical volume could be created by, extruding two semi-circular surfaces to create two volumes, each being half of the cylinder.

2.2.3 Ensuring Elements Sharing Nodes have Compatible Nodal Freedoms

Please see the following page on our user area for further details:

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

Also see the following reference which summarises the nodal freedoms of each element (as well as other properties):

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

2.2.4 Splitting Geometry

Where there are overlapping lines, the simple solution is often to split a line at a point. To do this, the line and a point part way along it are selected and then the following menu item is used to split the line into two at that point:

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

For example, in Figure 9 b) line 64 is overlapped by line 50. Line 64 can be split at point 59. This will create a second line between points 59 and 73 that will be perfectly coincident with line 50 and thus they can be merged such that one line (line 50) is then shared between surfaces 16 and 20. This is shown below in Figure 10. Line 64 is deleted and a new shorter line 65, between points 29 and 59 is created.
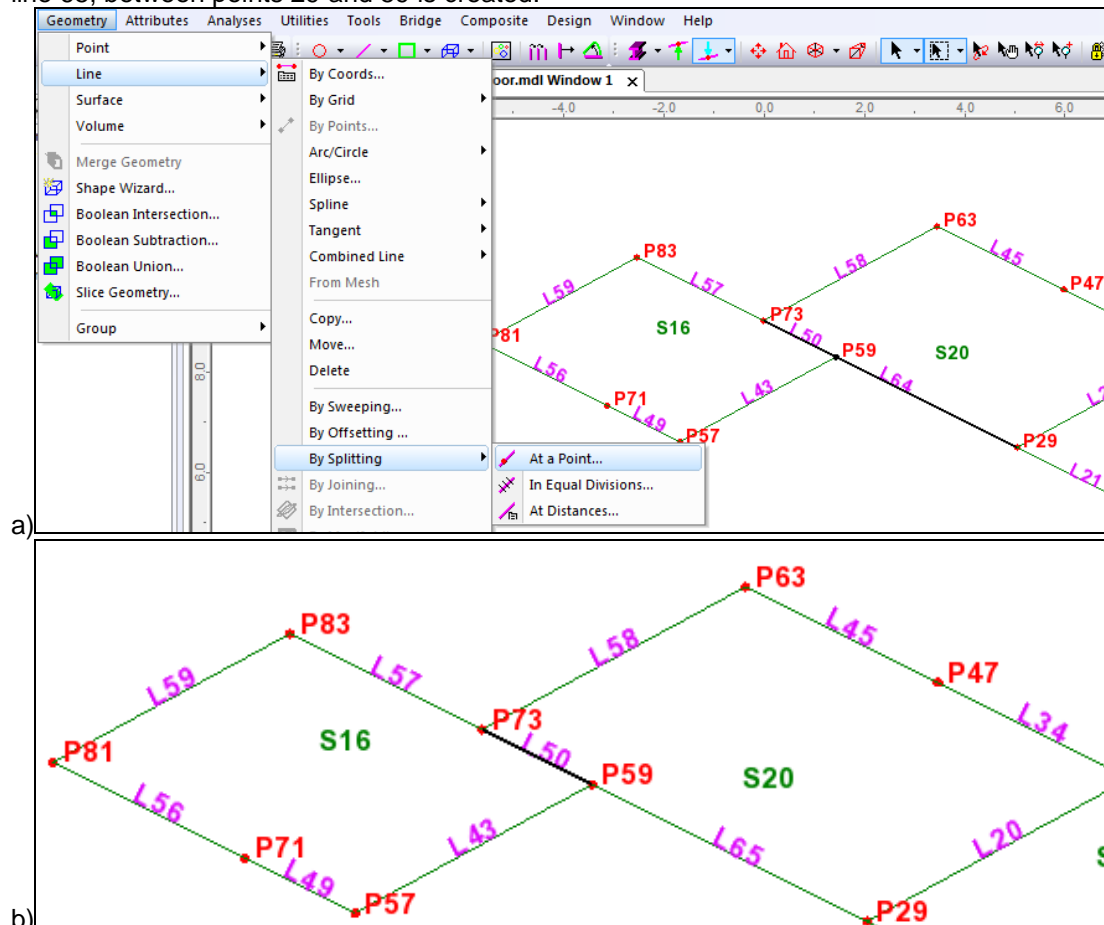

Figure 10 Geometry>Line>By Splitting>At a Point Example

a) Line 64 and point 59 are selected and then the line is split. b) a new shorter line 65 is created between point 29 and 59 and line 50 is now shared between surfaces 16 and 20.

In this example, line 65 will also need to be split at a point in order to be connected to the other adjacent surface which was shown in Figure 7.

Similarly, a surface can be split at a line. To do this, a surface and a line (which completely spans across the surface) are selected and then the following menu item is used to split the surface into two at that line:

Geometry menu > **Surface** > **By Splitting** > **By Line…**

Also see the following reference for further useful geometry operations that can be used to split lines, surfaces or volumes:

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 connectivity issues found, the analyst may judge it to be more practical and take less time to redefine geometry in Modeller to resolve connectivity issues, rather than to try and fix them. With regard to redefining geometry or creating it from scratch, the following recommendations are given:

- **Complete the import, manipulation and any editing of geometry before making any assignments (including mesh) if possible** – this is because the default conditions under which features may merge is if they have identical assignments. It may be desired to test meshing by assigning a mesh, but then it is advised to deassign it again before manipulating the geometry with operations such as splitting surfaces, just to ensure good connectivity is achieved.

- **Try to create all surfaces such that they are defined by 3 or 4 lines only** – such surfaces are termed regular surfaces and can be meshed with a grid like shell mesh. These surfaces can be easily manipulated and changed and they facilitate the achievement of a good mesh as well as making mesh refinement easier. A non-planar surface of 5 lines or more cannot be edited by moving the lines or points that define it because the associated change to the internal shape/surface map (for example a complex curved shape NURBS surface from a CAD import) is indeterminate.

- **Plan the geometry required in terms of the features to be modelled, the assignments to be made, results to be extracted and the areas where the mesh may need to be controlled/refined** – it is better to take time to plan out and create the required geometry upfront than it is to spend time editing a nearly finished model later. Drawing surfaces such that the required lines and points are in the locations needed for connections between members, load application, supports, interrogation of results, mesh refinement, application of constraint equations or contact modelling and so on is better than trying to split up surfaces later on to achieve this retrospectively and will reduce potential modelling errors like connectivity problems or incorrect assignments. Remember that for surfaces to be fully connected, they must share common lines, and for lines to be fully connected they must share common points. Fully connected geometry ensures a fully connected mesh with adjacent elements sharing common nodes.

- **Use the Move, Copy and Sweep commands** – Connectivity can be easily ensured if a point is swept to create a line and a line swept to create a surface and then a surface copied carefully (or further lines swept) to create more surfaces and surfaces swept to create volumes. This is because this process minimises the risk of tolerance errors creeping in, minimises the creation of coincident features and reduces the chances of features failing to merge. Please see the example referenced below which demonstrates this recommended approach to building up geometry.

  Help menu **> Help Topics > Contents > Application Examples Manual (Bridge, Civil & Structural) > Buckling Analysis of a Plate Girder**

Note: Combined lines cannot be deleted directly when they are already part of a surface definition; the surface would need to be deleted first. When redefining surfaces of more than 4 lines, there may be combined lines which should also be deleted during this process, particularly if the surface is to be replaced by two or more regular surfaces defined with 3 or 4 lines only. Please see the following reference for more information**.**

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

The use of scripting in LUSAS is very helpful for automating repetitive tasks and a number of example scripts are provided to download from our User Area:

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

For example, one of the scripts provided on this page will automate the redefining of multiple surfaces from a large selection of lines "**Create Multiple Surfaces from a Large Selection of Lines**". Also the scripts "**Check and Detect Short Lines**" and "**Select Nearby Features"** can be helpful when dealing with connectivity issues.