

CUSTOMER SUPPORT NOTE

**Joints for releasing degrees of freedom
between surfaces**

Note Number: CSN/LUSAS/1017

This support note is issued as a guideline only.

1. Introduction

Interface joint elements can be used to connect surface elements (e.g. shells) without full fixity. This mesh is assigned to the two line features on the edge of the two separate surfaces (one line is designated "master" and one "slave"). The gap between these surfaces could be zero.

The approach outlined below will enable the generation of (for example) moment release between surfaces meshed with shell elements

2. Description

2.1 Geometry - Surface and Line Features

You must have a model, which includes two surface features arranged such that you can assign the joints between two parallel line features. You may need to temporarily create a gap between the two lines (e.g. copy a surface feature at an appropriate offset and delete the original surface).

The points and lines at the interface must be made "unmergable", so that they are not automatically merged by LUSAS Modeller if they share the same coordinate positions. You can do this by selecting the features and using the menu items:

Geometry > Point > Make unmergable

Geometry > Line > Make unmergable

Joint elements have no length in a stiffness matrix and so any length given in the model will introduce an inaccuracy in the length of any substructure features and will trigger a warning in the LUSAS Solver text output file (*.OUT). However, moving the features at each end of joint elements to be coincident can make it difficult to be sure of the orientation of the joints element axes (local x, y, z). Therefore it is prudent always to use a local coordinate dataset to control the axes.

2.2 Mesh attributes

Your surfaces should be meshed in an appropriate manner, usually using shell or plate elements.

The appropriate mesh to connect to all 6 degrees of freedom for thin shells (QSI4), thick shells (QTS4, QTS8) and 3D beam elements is "joint for beams 3D" (JSH4). When no rotational stiffness is required however, it may be more appropriate to use "joint no rotational stiffness" (JNT4). This is simpler than using JSH4 as the rotational stiffnesses do not need to be calculated.

It is prudent always to use a local coordinate dataset to control the element axes for joints. This should be defined before the joint mesh attribute is assigned. More information on local coordinates may be found in the help menu item:

Help Topics > Modeller user manual > Building a model > Attributes > Local coordinate systems

A JNT4 mesh (for no rotational stiffness at all) may be defined using the menu item:

Attributes > Mesh > Line > (drop down menus) Generic element type=Joint no rotational stiffness, Number of dimensions=3d

The mesh definition should include the number of divisions you want on the relevant side of the surface features you are joining.

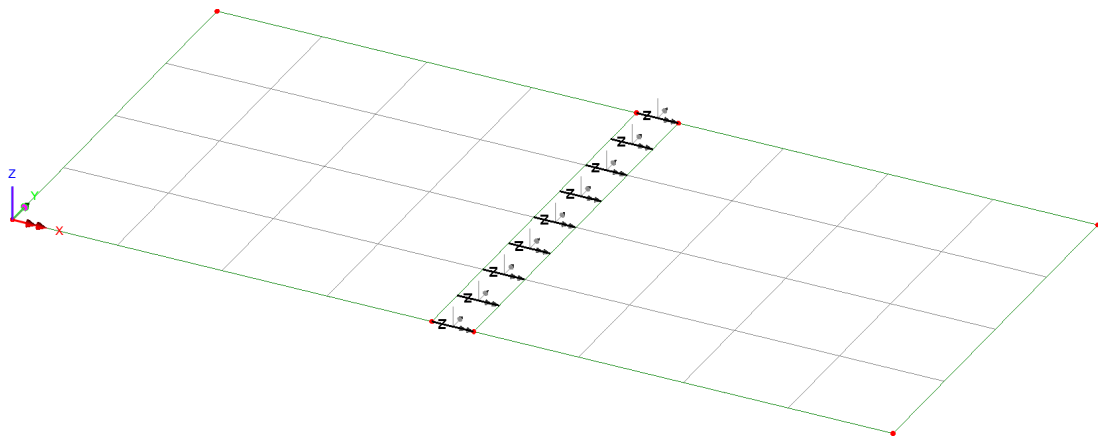
You may assign the mesh by selecting one line with your mouse and adding it to the selection memory (this will be the 'slave' of the joints). Then select the second line with your mouse and assign the joint mesh attribute by dragging and dropping it from the TreeView onto the model working pane (this second line will be the 'master'. This is as described in the help menu item:

Help Topics > Contents > Modeller User Manual > Building a Model > Attributes > Meshing and Element Selection > Joint/Interface Element Meshes

When you assign the mesh attribute you are able to specify a local coordinate dataset if one is defined (recommended).

You may check element axes using:

TreeView > Layers tab > Mesh > (check) Show element axes



2.3 Geometric attributes

As stated in the Element Reference Manual, Geometric properties are not required for the 3-degree-of-freedom (3-dof) JNT4 elements but are required for the 6-degree-of-freedom (6-dof) JSH4 elements. For JSH4 elements you should use 'Attributes->Geometric->Joint' to define a geometric eccentricity, which can of course just be zero.

2.4 Material attributes

Your surfaces should have an appropriate material attribute assigned (e.g. concrete, steel etc).

The joint material attribute used in your model controls how the joint will behave when subjected to loading or displacements etc. Materials appropriate to common uses of joints are described below; full information on all available joint materials is given in the help menu:

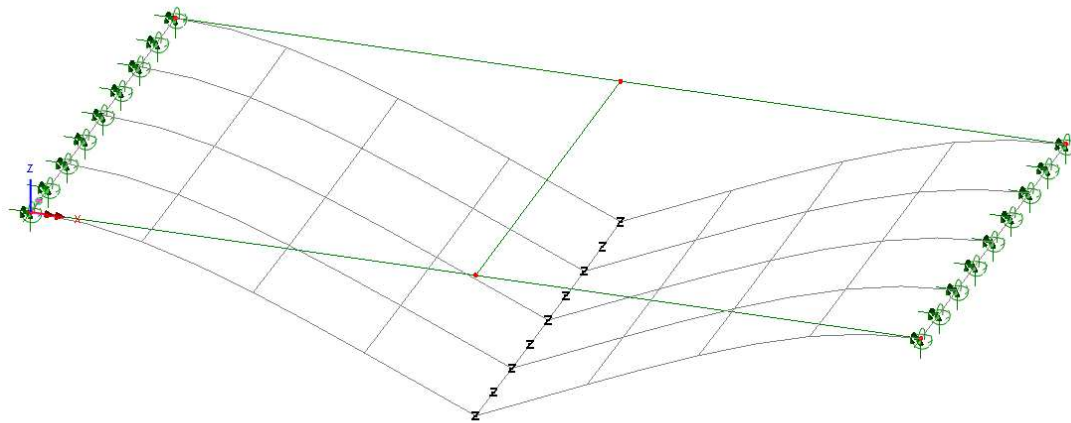
Help topics > Theory manual > Chapter 4 Constitutive models > 4.10 Joint Models

For a simple hinge JNT4 joint you may use:

Attributes > Material > Joint > (drop down lists) Joint type=Spring stiffnesses, Freedoms=3

The number of "Freedoms" specified in the material should match the DOF in joint mesh attribute selected. You must enter a stiffness value for each DOF. For JNT4, Freedom=3 should be used: the DOF are, in order, local x, y, z. For JSH4, Freedom=6 is appropriate: the DOF are, in order, local x, y, z, THx, THy, THz.

The image below demonstrates the use of joint elements to create a hinge connection between two shell surfaces.



For more information on the calculation of spring stiffnesses, please see the following page on the password protected LUSAS User Area:

http://www.lusas.com/protected/instruct/joint_spring_stiffness.html (The Username and Password for accessing the User Area are 'lusas' and 'powerfulfe' respectively)