CUSTOMER SUPPORT NOTE

Obtaining Slice Forces and Moments in Solid Meshed Models

Note Number:	**CSN/LUSAS/1031**

This support note is issued as a guideline only.

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

For a beam meshed model, forces and bending moments can be outputted directly. For a shell meshed model, resultant forces and moments per unit width can be integrated for a given slice to obtain equivalent beam forces and moments using Utilities>Slice Resultants Beams/Shells. Due to the element theory and degrees of freedom in use, bending moments cannot be “directly” outputted for solid continuum elements. Stresses can be integrated to obtain forces and moments for a slice created using Utilities>Section through 3D. This gives a cross-section slice through the three-dimensional solid meshed model on which the user can display results. Results for this slice through the solid model are calculated at pseudo-nodes, formed at the intersections of the slice with the element edges by linear interpolation of the nodal results. Stresses for the slice can then be integrated over the area of the cross-section to determine equivalent beam forces and moments.

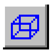
This Support Note explains how this facility can be used in the context of a simple example of a cantilever beam meshed with solid 3D continuum elements.

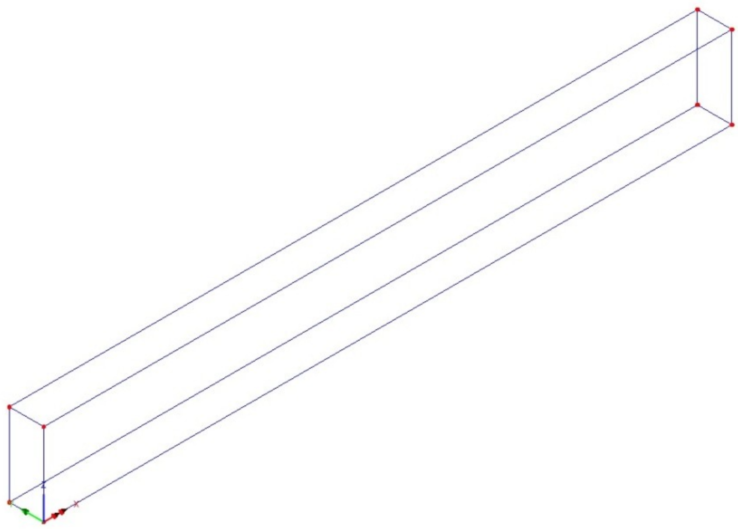
2. Description

2.1 Geometry - Volume Features

A cantilever beam of 5m length and a cross section of 0.25m x 0.60m is to be modelled.

Create a new model with units of kN, m, t, s, C and Z-axis assigned as the vertical one.

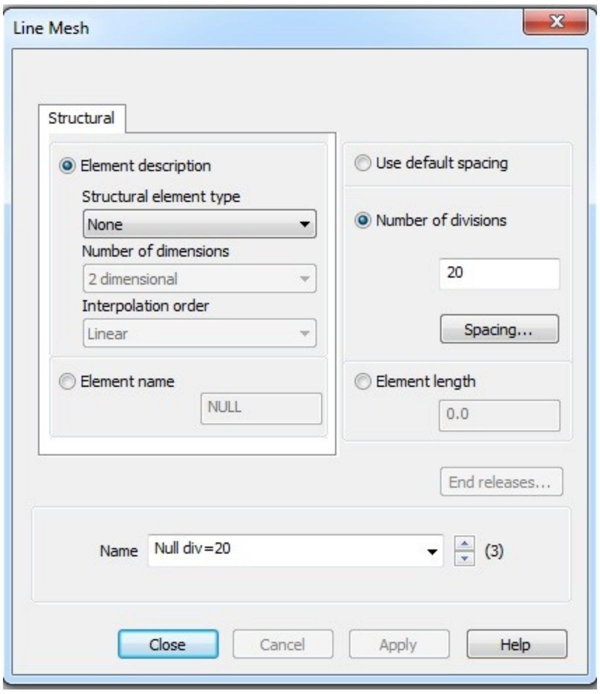
Select the  button to create a cuboid of length (X) 5, height (Y) 0.25 and depth (Z) 0.60.



2.2 Mesh attributes

The four lines along the longitudinal axis of the beam shall have 20 divisions i.e. 0.25m per element.

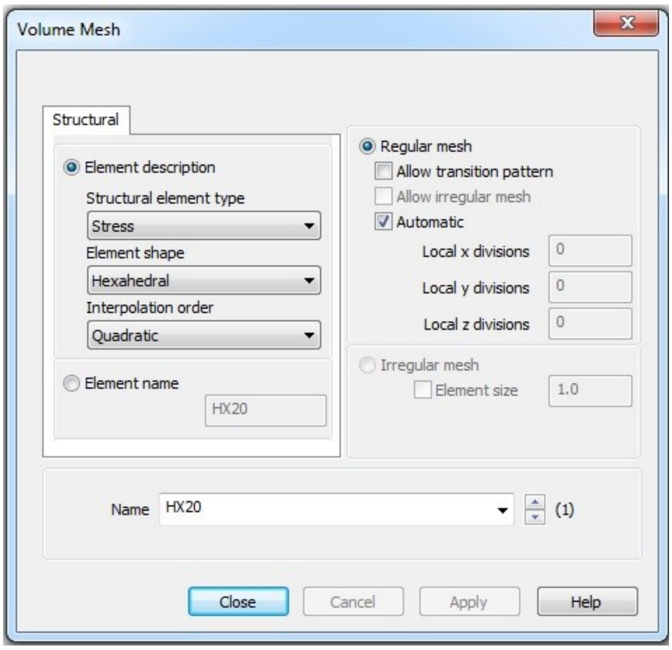
Attributes > Mesh > Line



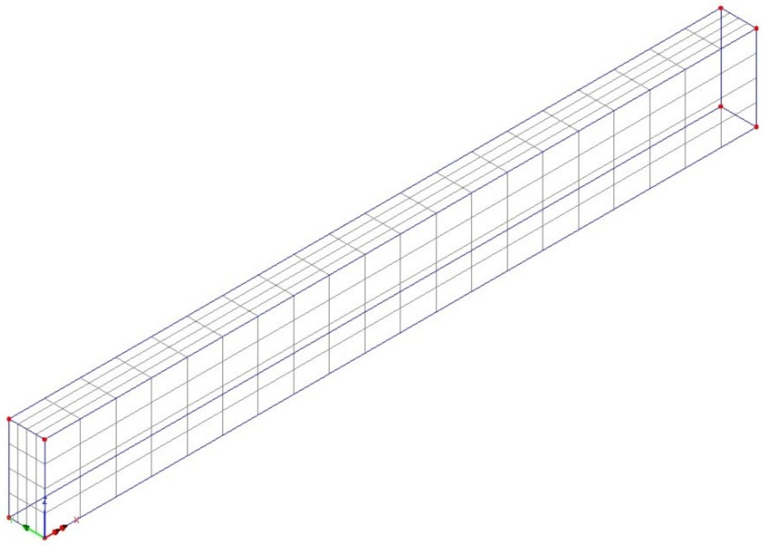
Assign this attribute to the aforementioned lines.

The beam is going to be modelled with stress hexahedral elements with a quadratic interpolation order (HX20).

Attributes > *Mesh* > *Volume*



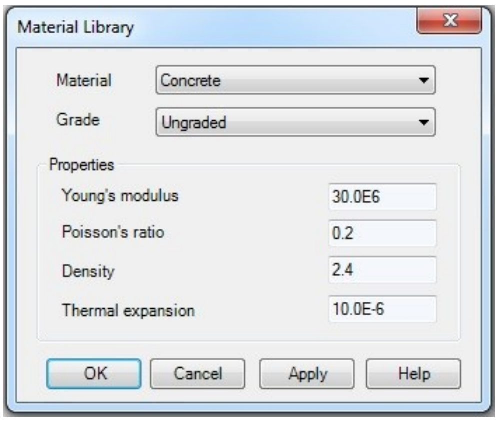
Select the whole model and assign the volume mesh 'HX20' from 'Attributes Treeview' by dragging and dropping the attribute into the Graphics Window.



2.3 Material attributes

An ungraded concrete material will be assigned on the whole model.

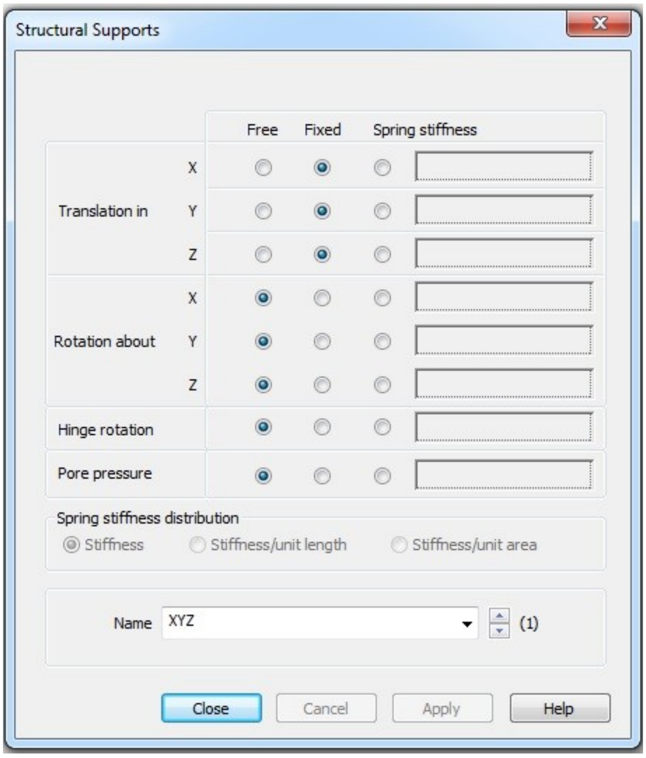
Attributes > Material > Material Library



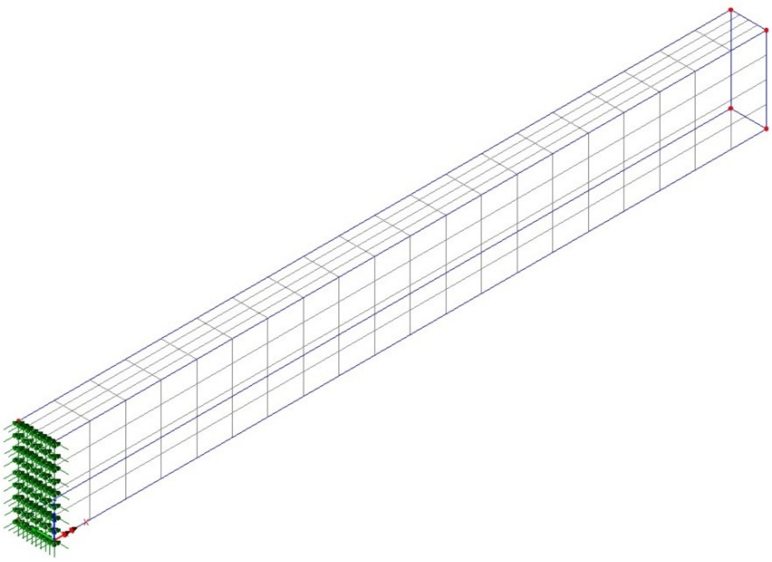
Select the whole model and then assign the material attribute 'Iso1' from 'Attributes Treeview' onto it by dragging and dropping. Ensure that 'Assign to volumes' is selected.

2.4 Support attributes

Support attributes will be assigned to one end surface of the beam.



Select the surface at the left end of the beam and drag and drop the support attribute onto the model.

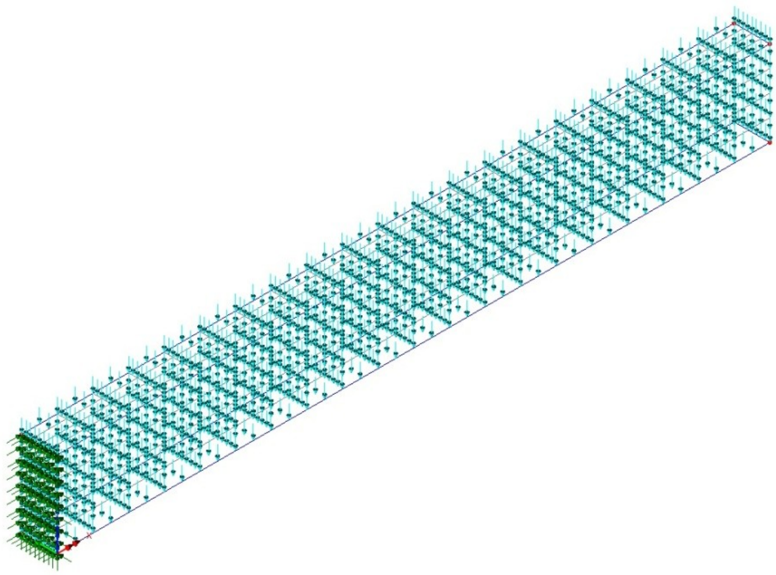


2.5 Loading attributes

Only one loadcase (self-weight) is going to be considered for the structure in this example. Define a linear acceleration of 9.81kN/m^2 in the global-Z axis.

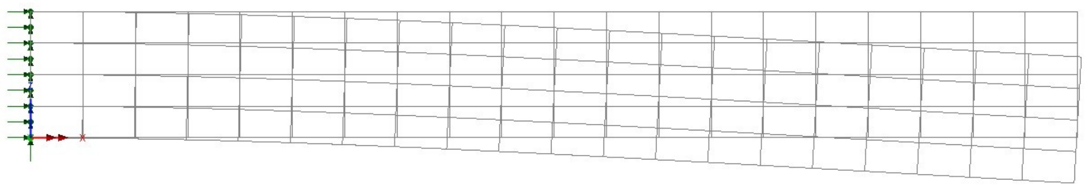
Attributes > *Loading* > *Body Force*

Drag and drop the attribute onto the model (assign to volumes only) and check the loading arrows are as shown below.



2.6 Checking the model

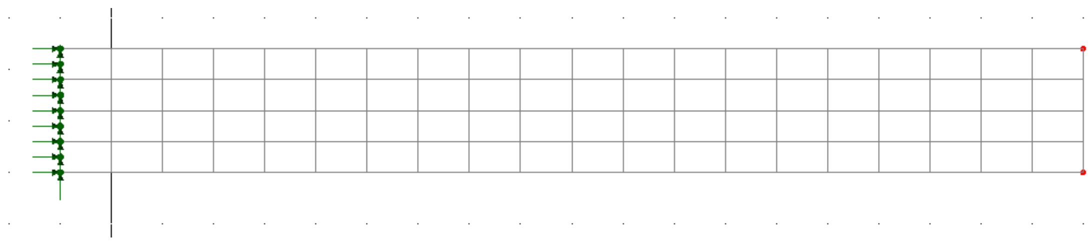
Turn off the load attribute visualisation and then run the model to check that the deflected shape is sensible.



2.7 Creating a (section through 3D) slice

Before creating a slice through the solid meshed model, ensure that the view of the model is orientated as desired bearing in mind that the model will be sliced in a plane perpendicular with the current view orientation of the model (into the screen). Also ensure that only the volumes (and their underlying volume mesh) that you wish to slice are showing results, or set as that only visible on screen.

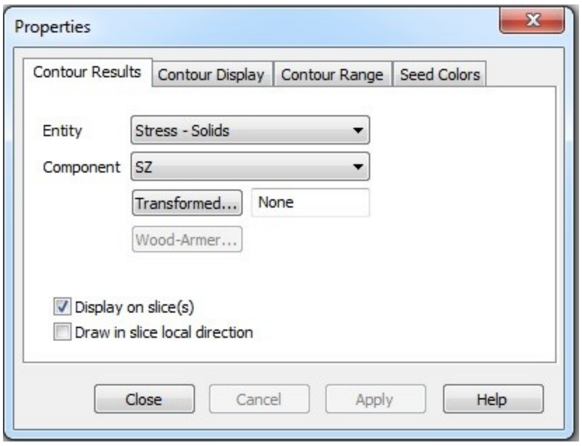
A section through the 3D solid continuum model is now created at X=0.25 by using **Utilities > Section through 3D**. In the dialog that appears, set the grid size equal to the elements length (i.e. 0.25m) and draw a line parallel to the global Z axis at .

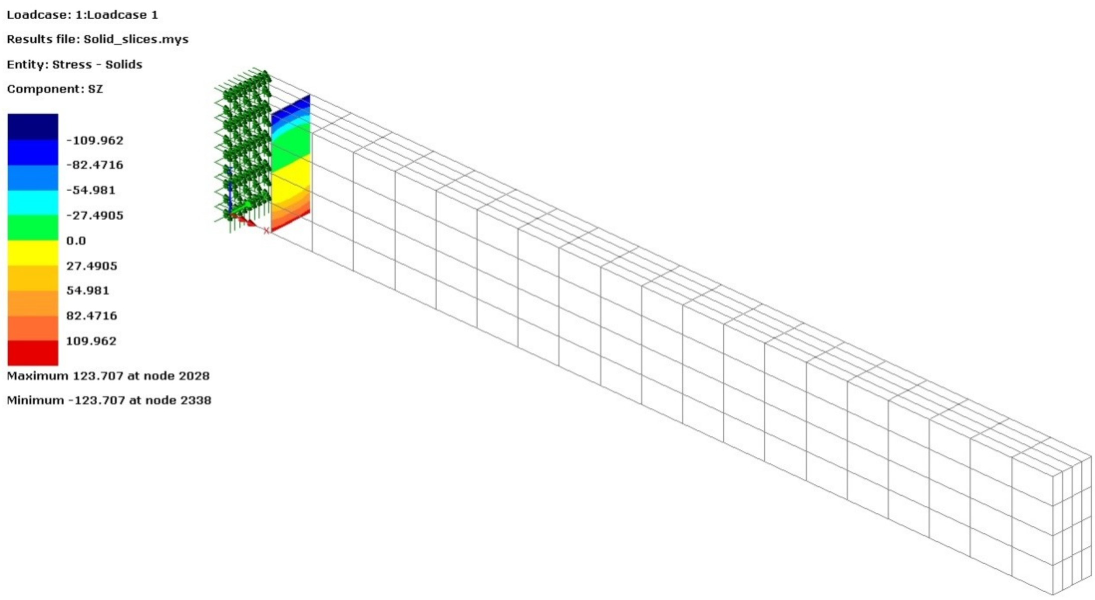


A parent group called “Slices” is automatically created, with a separate sequentially numbered group per slice created under it.

2.8 Creating contours

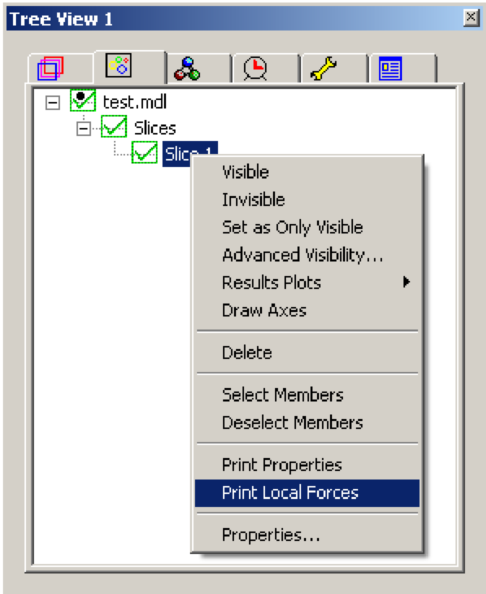
You can check the stress contours on the slice by right clicking on the slice group and choosing “Set as Only Visible” or you can view contours only on the slice by selecting “Display on slice(s)” in the Contour Properties Dialog.



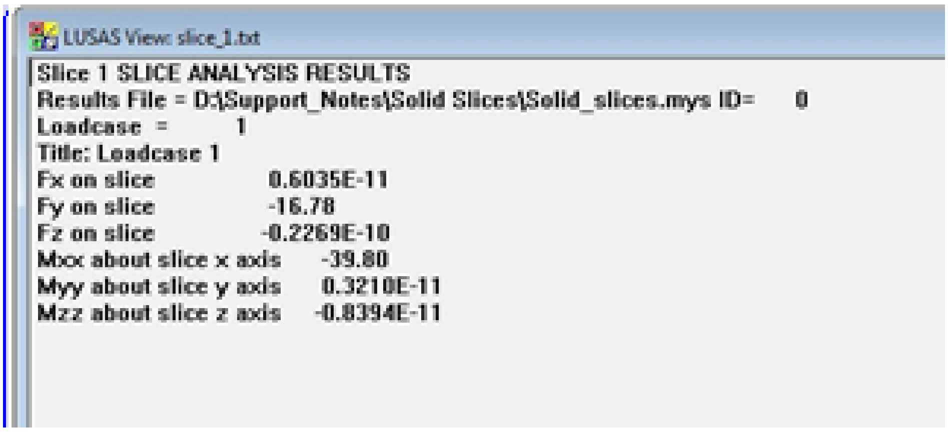


2.9 Printing forces

In the  Treeview, you can now right click on the slice group to select “Print Local Forces”.



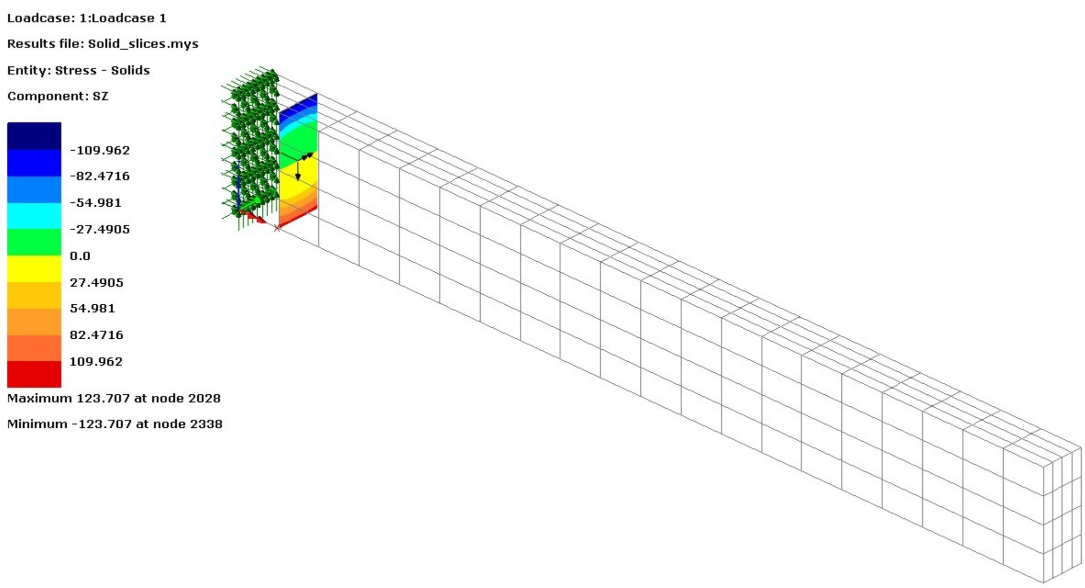
This will load a pop-up a window showing the forces and bending moments on the slice about the slice local axes. Results improve with the mesh density.



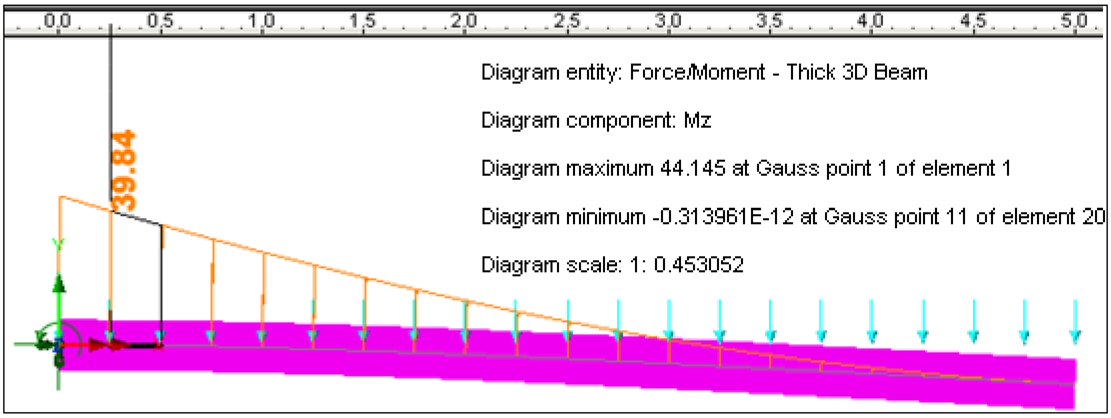
Slice 1 SLICE ANALYSIS RESULTS
Results File = D:\Support_Notes\Solid Slices\Solid_slices.mys ID= 0
Loadcase = 1
Title: Loadcase 1

Fx on slice	0.6035E-11
Fy on slice	-16.78
Fz on slice	-0.2269E-10
Mxx about slice x axis	-39.80
Myy about slice y axis	0.3210E-11
Mzz about slice z axis	-0.8394E-11

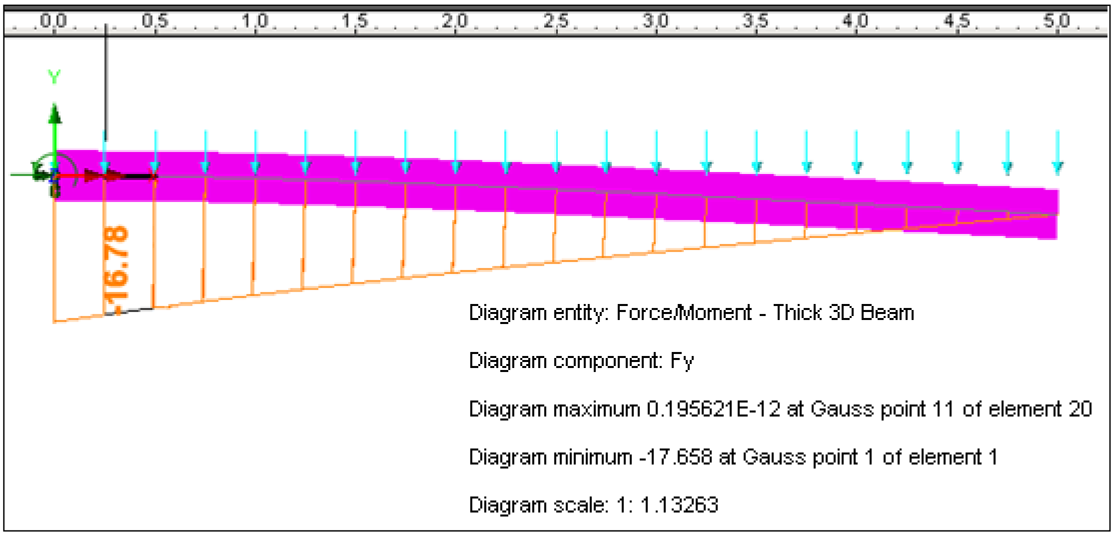
The slice axes can be displayed on the slice by right clicking on the slice group and selecting “Draw axes”.



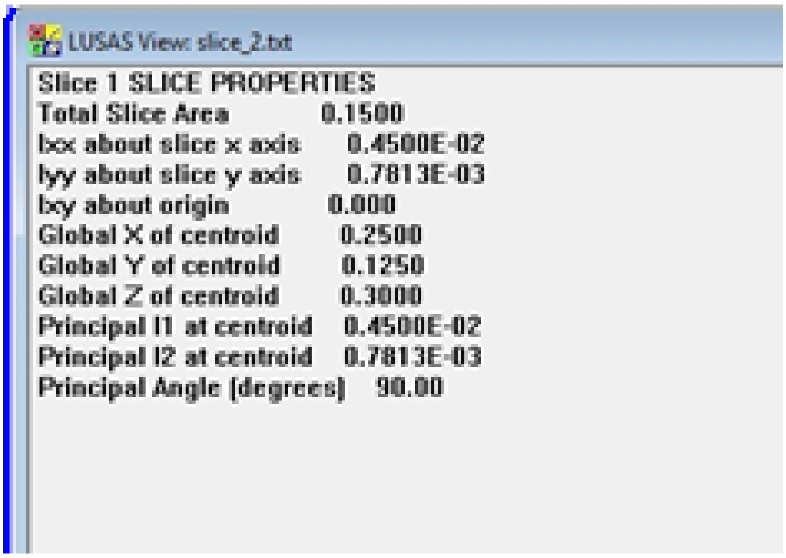
If we compare these slice results to that of an equivalent beam meshed model we can see that we get the same result for the moment to three significant figures at this location on the beam:



The result for the vertical shear force is also the same as that in the equivalent beam meshed model to four significant figures:



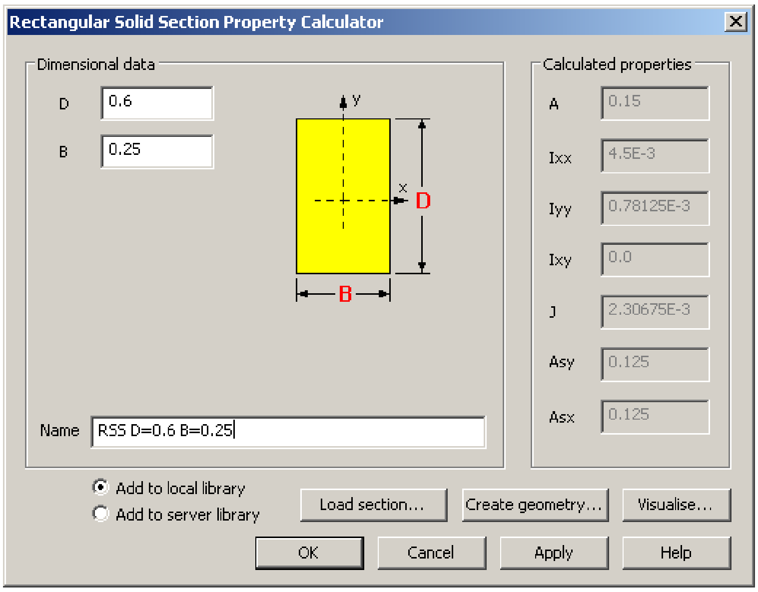
In addition, you can output the section properties of the slice (e.g. area, 2nd moment of area etc.) by selecting "Print Properties" from the context menu for the slice group.


LUSAS View: slice_2.txt

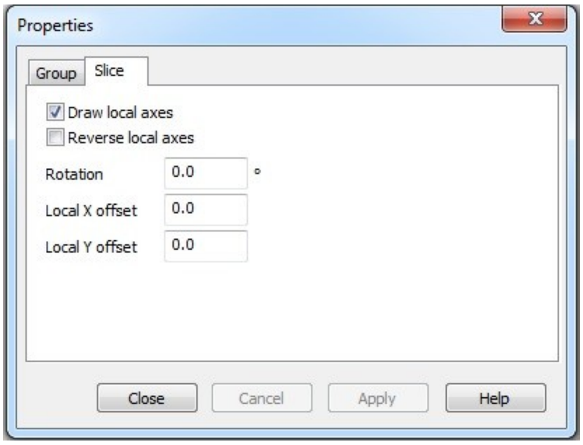
Slice 1 SLICE PROPERTIES

Total Slice Area	0.1500
Ixx about slice x axis	0.4500E-02
Iyy about slice y axis	0.7813E-03
Ixy about origin	0.000
Global X of centroid	0.2500
Global Y of centroid	0.1250
Global Z of centroid	0.3000
Principal I1 at centroid	0.4500E-02
Principal I2 at centroid	0.7813E-03
Principal Angle [degrees]	90.00

If we compare these section properties to those given by the Standard Section Property Calculator (**Utilities** menu) for this rectangular section (the output of which is shown below) we can see that we get the same values for the section properties calculated for the "Section through 3D":



In case you want the values about another location in the slice plane, you can move the slice local axes by right clicking on the slice group and selecting "Properties…" and choosing the "Slice" tab.



Note that this method will not slice line or surfaces meshed elements and these must be made invisible if present and sliced separately. For slices through beam and/or shell elements the **Slice Resultants Beams / Shells** facility should be used.