Computer Modelling of Structures

Session 4 – Special Case Study

Modelling Plane Stress for a Simply Supported Beam using Finite Element Analysis

Lab Group 7

Matthew Kennefick

Adrian Doherty

Michael Wynne

Thomas McGahon

Table of Contents

INTRODUCTION:

The objective of this report is to develop a plane stress model of a simply supported beam using the finite element analysis (FEA) programme LUSAS and to show how the accuracy of the solution is affected by changing the mesh density. The report will show how the finite element model converges towards or diverges from the results from a theoretical model as the mesh density increases or decreases. Different elements such as triangular and quadrilateral elements with only corner nodes will be used in the FEA and their effects on convergence will be studied.

BACKGROUND THEORY

Plane Stress

When an element of material is in a state of plane stress in the plane of a set of orthogonal axes x and y, only the x and y faces of the element are subjected to stresses (Fig 1). All the stresses either normal or shear acting on the material act parallel to the x and y axes. If the material is at the free surface of a body and the z axis is perpendicular to the x and y axes, out of the page in this case, then the z face of the material is in the plane of the free surface. If there are no external loads applied then the z face is stress free. This form of stress is common, as it exists at the surface of any body under stress as long as there are no loads applied to the surface. Plane stress allows 3- dimensional problems to be reduced to 2 – dimensional problems. This can simplify the modelling of structures and the computational effort is greatly reduced.

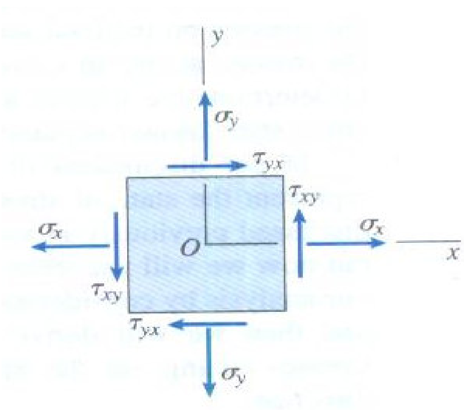


Figure 1. Plane Stress Element

Source: Gere, Mechanics of Materials, 5th Edition

Finite Element Analysis

The finite element method was discovered by Turner, Clough, Martin and Topp and was first published in a paper in 1956. The authors discovered that structures of irregular shape could be broken up into simpler geometrical elements and by deriving the load displacement equations in matrix form they could combine the effects of each element in a relatively simple manner. FEA can accurately determine the response of a model made up of finite elements when subjected to loading. In FEA you develop a model that is only an idealisation of the real structure. Only in a few exceptions does a FEA provide exact results for example in the case of a static analysis of a simple truss. However, with proper modelling accurate results can be obtained. The difference between results obtained from FEA and the exact solution is due to discretisation error. This error occurs when the model has been defined and is then split up into finite elements. The error arises from two sources. Firstly error due to assumptions made on the behaviour of the elements. For instance a plane stress triangular element (Fig 2. a) with constant stress over its area has 6 degrees of freedom, if mid-side nodes are now added to the triangular element the number of degrees of freedom rises to 12, with each node having 2 degrees of freedom. Due to this 'higher order' functions are used to define the behaviour of the element and a more accurate result is obtained. The second source of error is due to mesh density. The mesh density is the number of elements per unit area. It is usually the case that when the mesh density is increased the result converges on the exact solution.

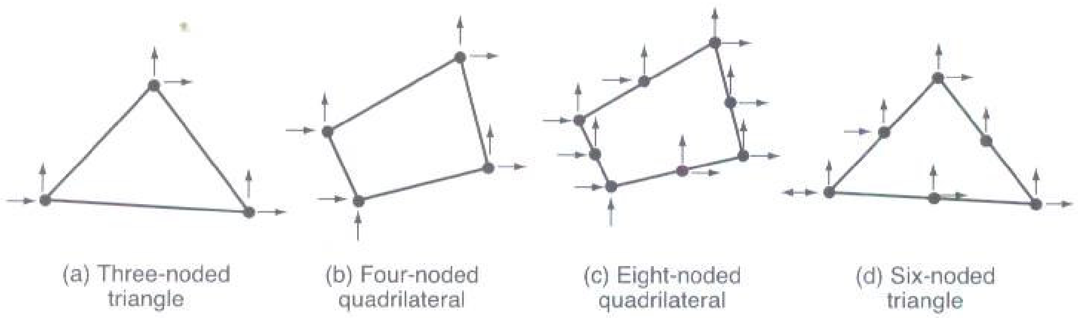


Figure 2. Plane Stress Elements Showing Degrees of Freedom

Source: *Macleod, Modern Structural Analysis*

This study demonstrates how the accuracy of a solution is affected by mesh refinement. The conclusions to be drawn from this study have a degree of general relevance to modelling with other types of element, such as plate bending, and volume elements.

Two plane elements types are included in the convergence analysis. The names used for the runs are from the LUSAS analysis programme.

- TPM3 – Linear triangular elements with only corner nodes
- QPM4 – linear quadrilateral elements with only corner nodes

All of these elements have two degrees of freedom per node.

Features of the behaviour of these elements include the following:

- The elements are all isoparametric, based on functions that define both the shape of the element and the displacements. Such elements are *conforming* in that the displacement functions ensure boundary compatibility between the elements. This means that:
  - i) they give a lower bound to an influence coefficient – i.e. the deformation at a single point load on a system of elements will always underestimate the true deformation as compared with an exact solution to the governing differential equations
  - ii) in general they tend to overestimate stiffness
- The loading for the system is not a single point load but the arrangement is such that the deformation in the line of the load is very likely to be underestimated by a mesh of any of two elements

In this context the order of the element is the number of terms in the displacement function used to define the element. For the elements considered here, the number of terms in the displacement function is equal to the number of degrees of freedom assigned to each element. A higher-order element will therefore tend to give more accurate results.

INVESTIGATION OF CONVERAGENCE:

Bending Theory Model:

Using bending theory defined in Gere (2001) a reference solution was produced for the max Bending Stress (Normal Stress) for the mid-span of a 5m long simply supported rectangular beam. The size of the beam 500mm deep x 100mm wide was chosen to best show up the stress distribution. It would not be considered a normal rectangular beam. The calculations can be seen in Appendix 1. This value is represented by the red line on the graphs below.

Finite Model:

The finite element model was made up of a surface element divided up using both a Quadrilateral (Quad) and Triangular (Tri) meshes. The full procedure on creating the model can be seen in Appendix 2

Meshes:

The meshes used for the results given in the graphs below are presented in the table below. The total number of degrees of freedom in the mesh is the order of the solution of the simultaneous equations and is therefore a measure of the computing effort needed to achieve a solution.

The calculation of the degrees of freedom is demonstrated using the 10 x 5 mesh.

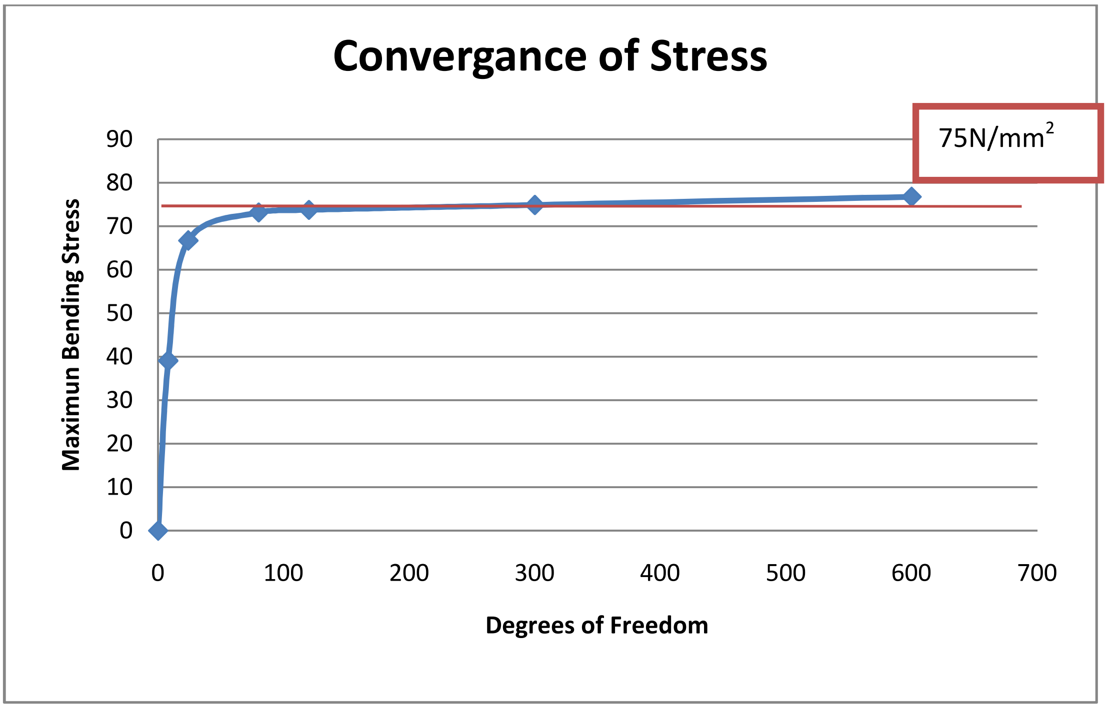
- With Quad (QPM4) elements there are 10x6 nodes, giving 60 active nodes for this mesh with two degrees of freedom per node.

  Dof = number of active nodes x freedoms per node = 60 x 2 = 120

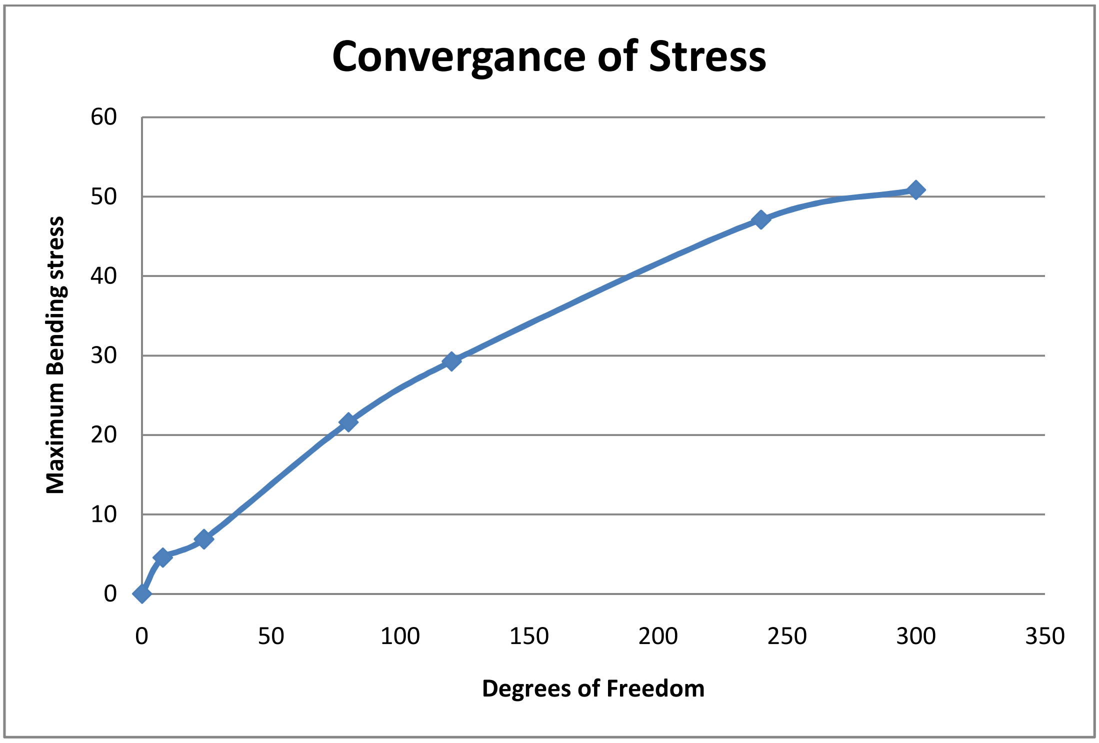
- With Tri (TPM3) elements, each rectangular element is divided into two triangles by a diagonal line. The number of elements is doubled, but the number of degrees of freedom is the same as with QPM4 elements.

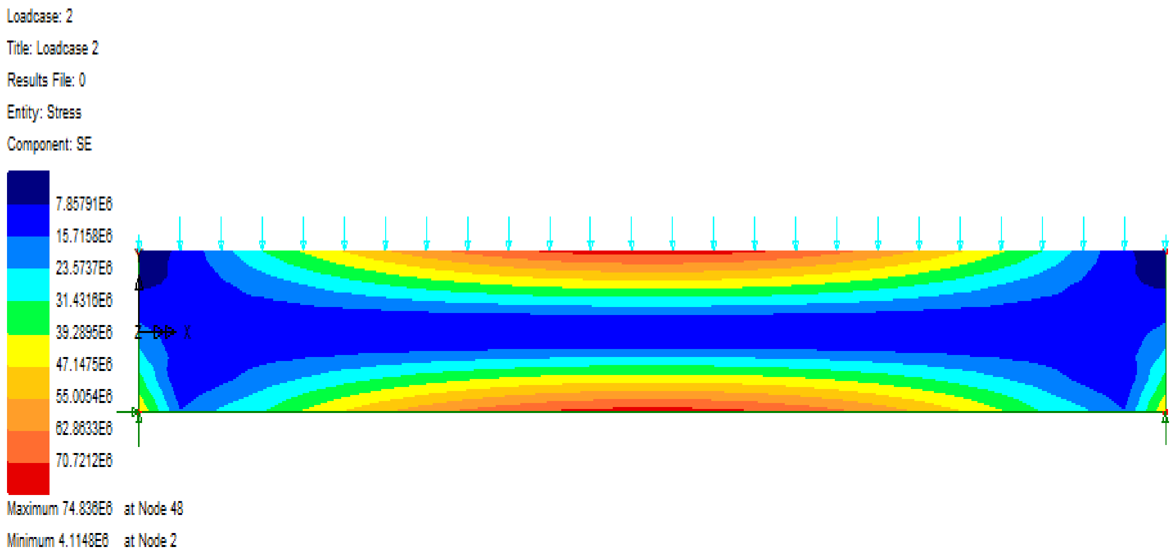
RESULTS:

Quad		
Mesh	Degrees of Freedom	Max Bending stress N/mm^2
2x1	8	39.05
4x2	24	66.68
8x4	80	73.12
10x5	120	73.71
25x5	300	74.83
50x5	600	76.71

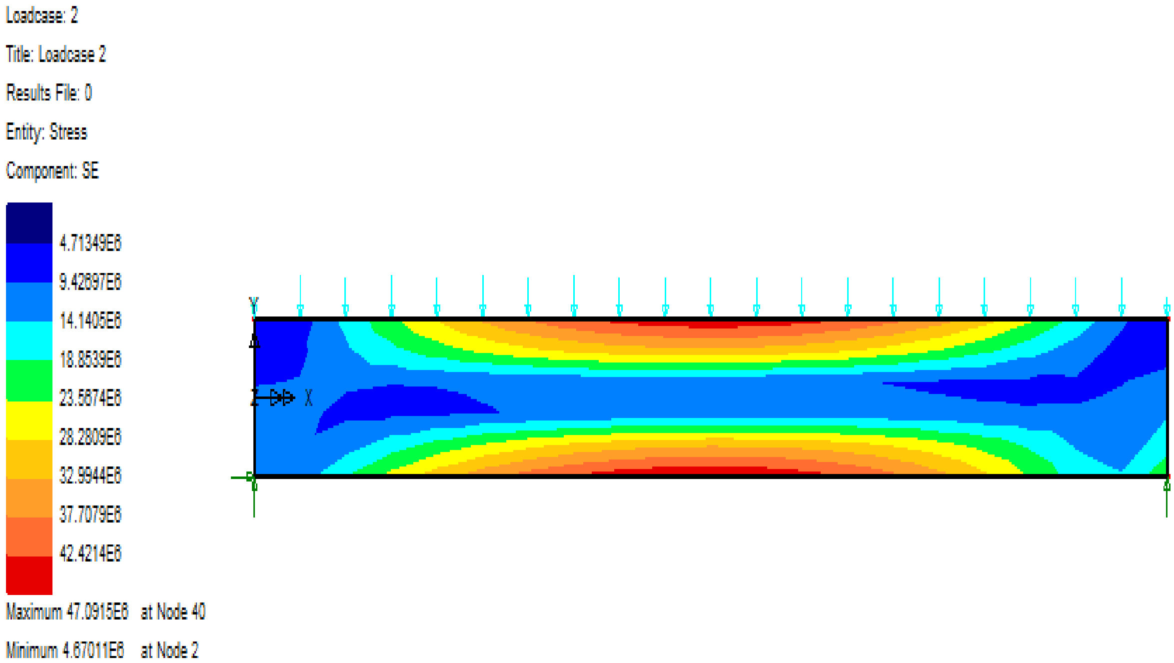


Tri		
Mesh	Degrees of Freedom	Max Bending stress N/mm^2
2x1	8	4.55
4x2	24	6.86
8x4	80	21.58
10x5	120	29.25
20x5	240	47.09
25x5	300	50.83





Quad 25x5 mesh



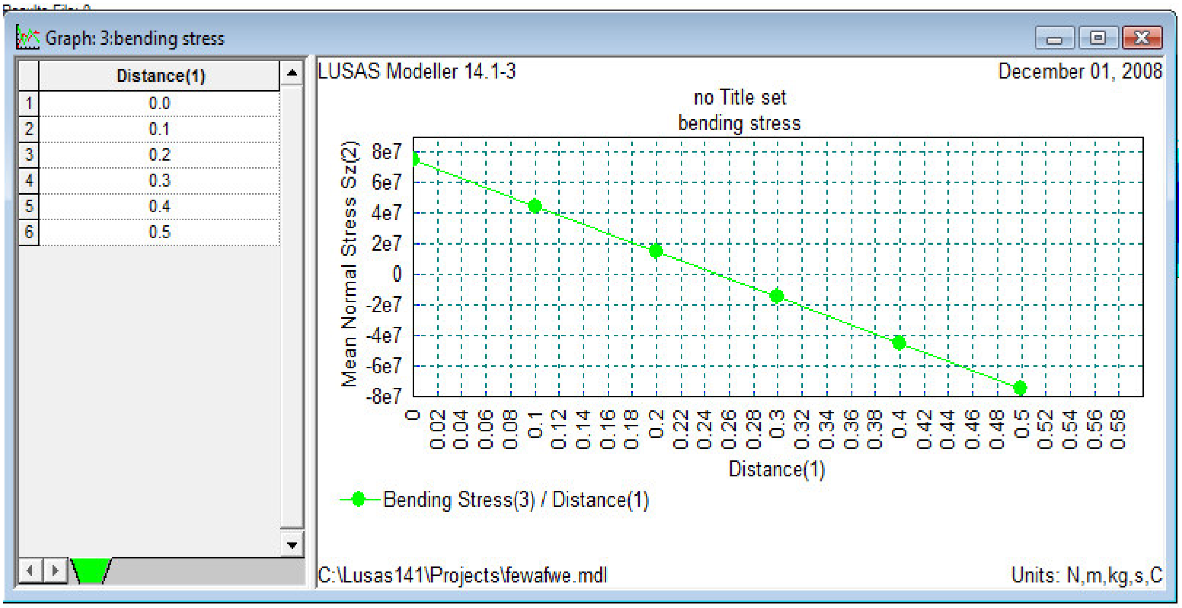
Tri 25x5 mesh.

Reference Appendix 3 for more screen shots of different meshes.

CONCLUSION:

Both models for the Bending Stress converge on the reference solution of 75N/mm^2 (Red Line). The Quad mesh is a lot more accurate at the same amount of Degrees of Freedom than the Tri mesh. The curve flattens after the 8x4 mesh giving a result close to the reference solution. To obtain an accurate result for the Tri mesh a considerably higher mesh density would have to be used. As LUSAS used is a training program there were limitations on the mesh density. This meant a Tri mesh giving the correct answer could not be found.

It is possible to obtain a graph of the bending stresses present through a cross section of the beam. Using this graph it is possible to obtain an accurate value for the bending stress at any cross section.



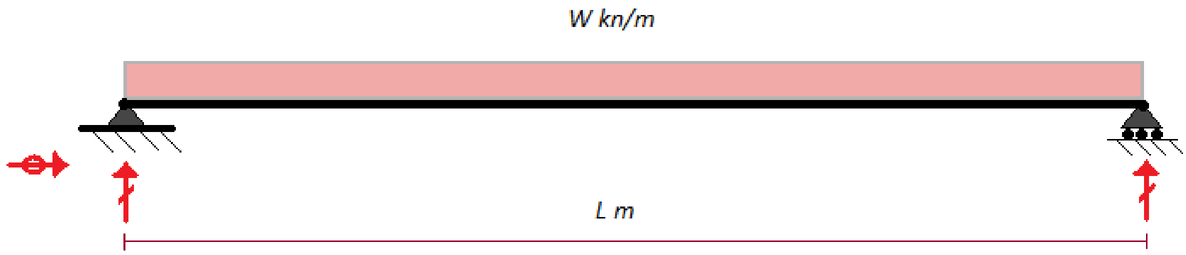
Recommendations:

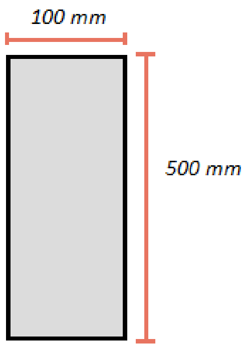
It is not best to use the highest mesh density as it can be seen that the highest density used gives a bigger stress than the reference solution. The best method is to choose a mesh density at which the convergence curve starts to flatten out.

In this analysis lower order elements were used. It can be seen that using a single Quad element give better results than splitting the area into two triangles. Therefore Tri mesh should only be used to fill triangular areas. A surface element also tends to give better results when a Quad mesh is used.

APPENDIX 1

Theoretical analysis of member:





$$\sigma_{max} = \frac{M_{max}}{Z}$$

$$M_{max} = \frac{W \times (L/2)^2}{2}$$

$$Z = \frac{bh^2}{6}$$

$$M_{max} = \frac{100 \times (2.5)^2}{2}$$

$$Z = \frac{100 \times 500^2}{6}$$

$$M_{max} = 312.5\ kNm$$

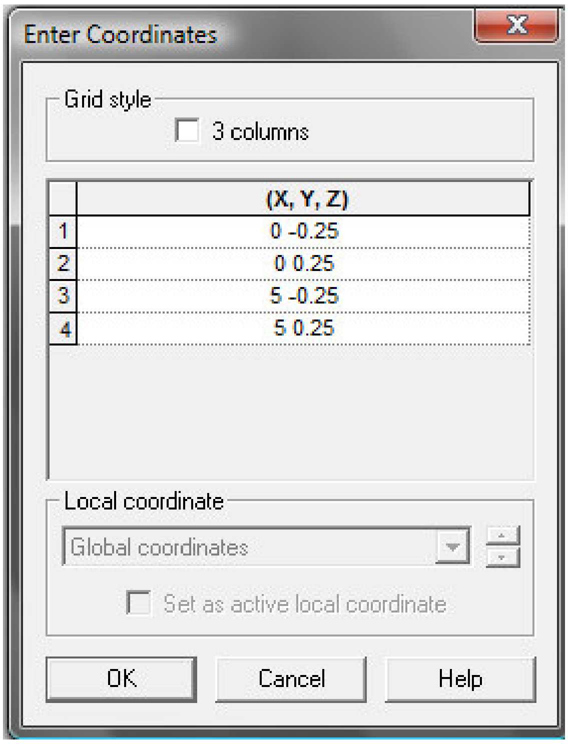
$$Z = 4166666.667\ mm^3$$

$$\sigma_{max} = \frac{312.5 \times 10^6}{4166666.667}$$

$$\boldsymbol{\sigma_{max} = 75\ \ N/mm^2}$$

APPENDIX 2
Process for generating and analysing model

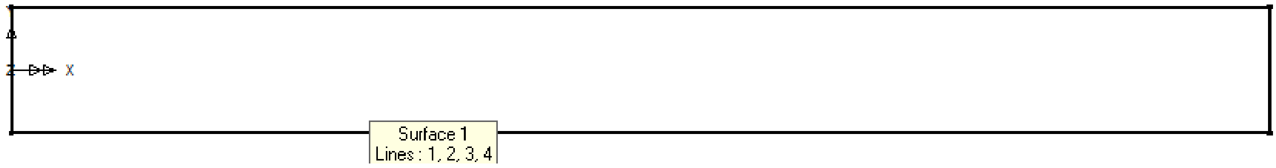
Place four nodes using Cartesian polar coordinates to represent the required dimensions in each axis (x, y, z).



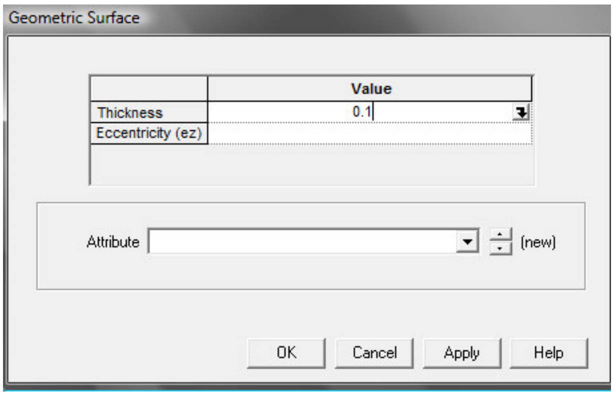
The four coordinates appear as red points on screen as shown below.

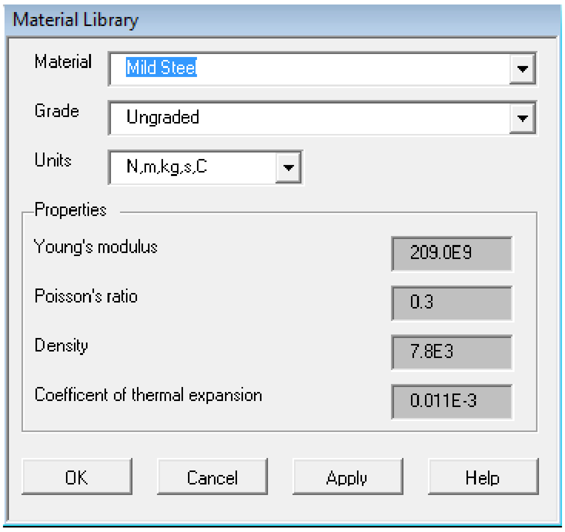
Next the four points must be connected by lines in a clockwise direction.

The area bound by the lines must now be defined as a surface.

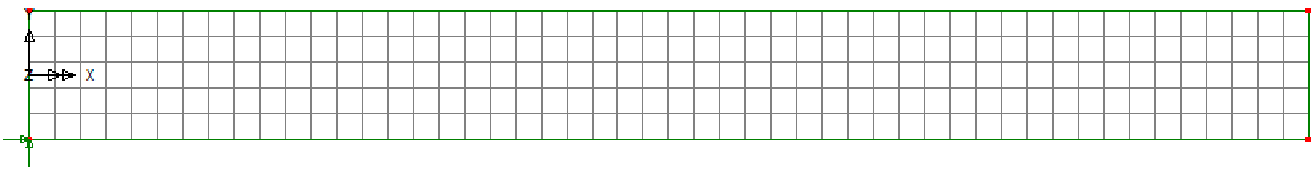


The surface must be given a thickness and as it is required to act as a particular material it should be given that materials properties.

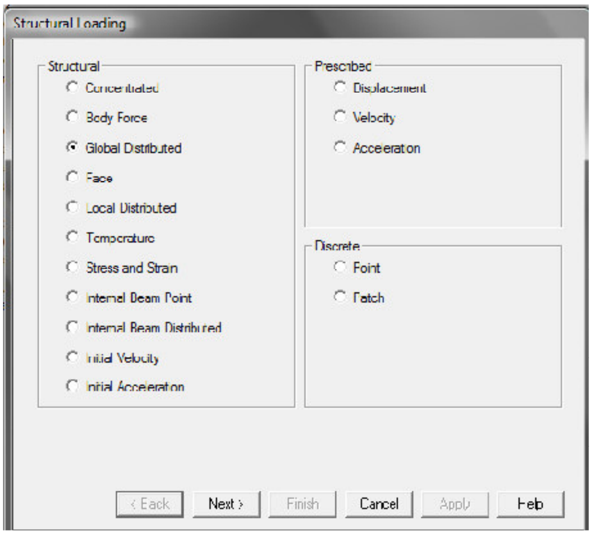


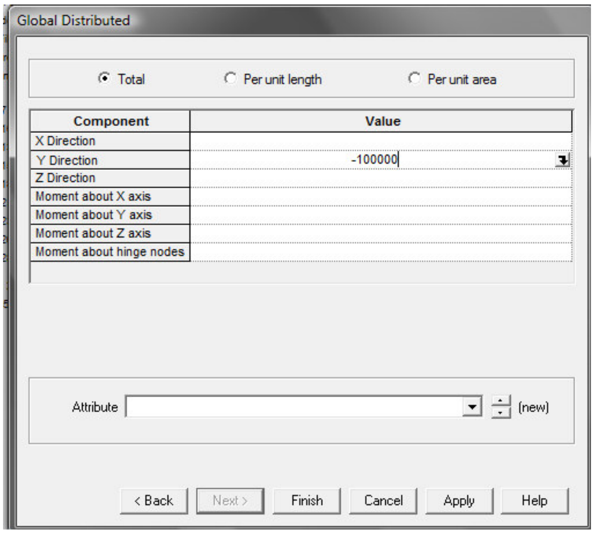


The member can now be meshed, in this case a Quad (square 4 node) has been chosen and as can be seen it appears over the materials surface.

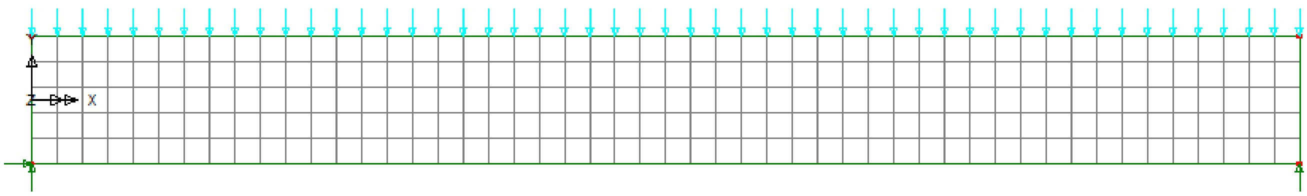


The required load must now be applied to the member. As it is a UDL select 'Global Distribution'. The value of the load must also be chosen, with care being paid to the direction of the applied load (positive upwards and negative downwards) the axis to which the loads are being applied to should also be selected.

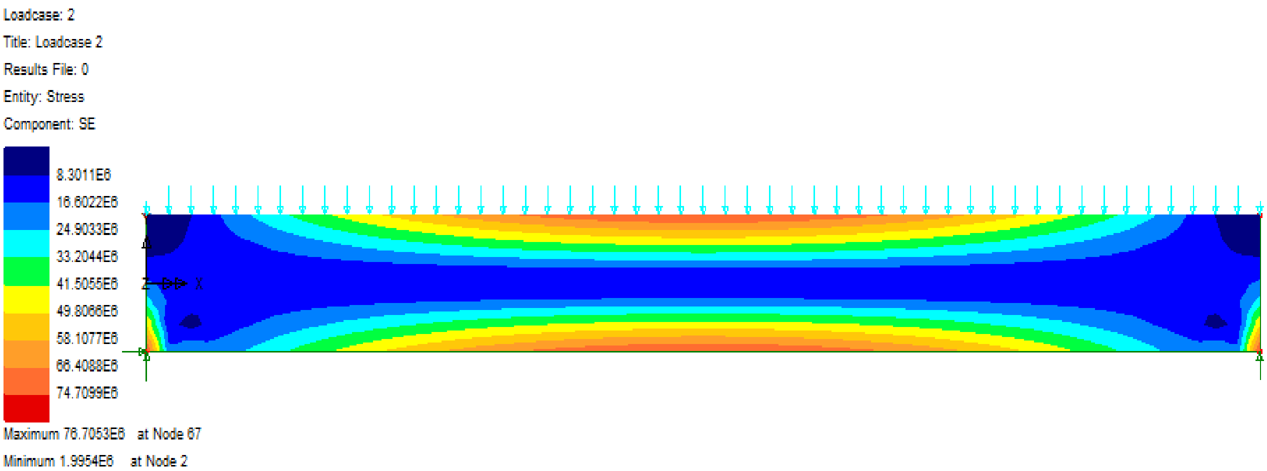




The member should appear as below



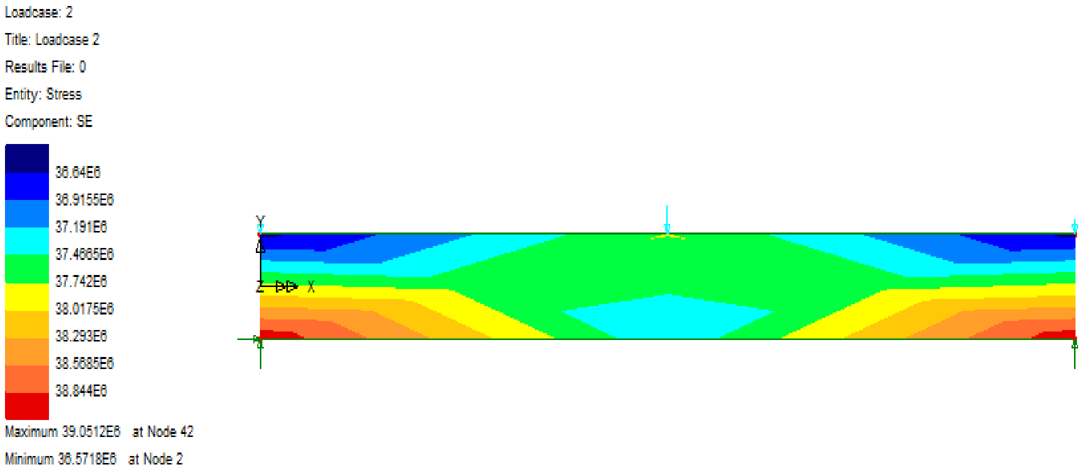
Running the analysis on the 50x5 quad mesh generates the following results.



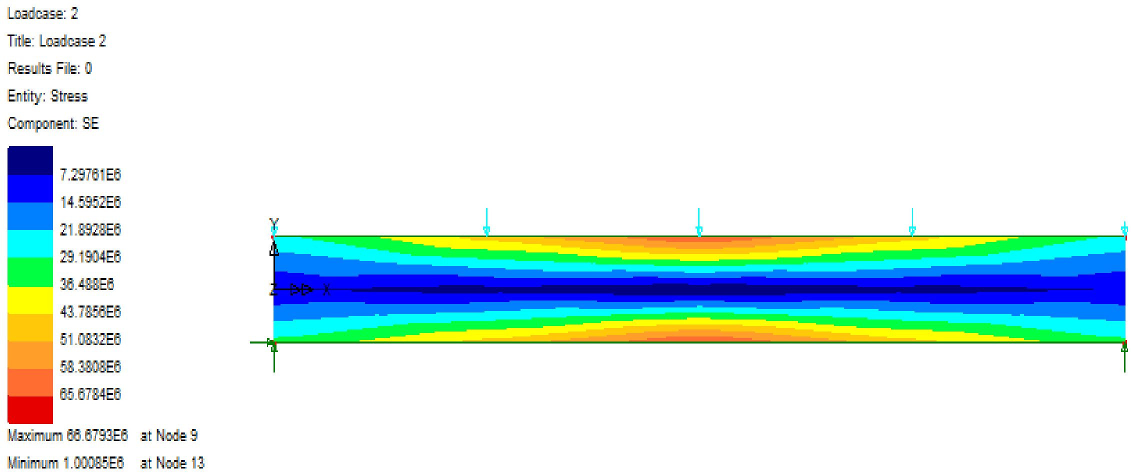
It can be seen how the maximum stress of 76.7 N/mm2 compares to the actual value of 75 N/mm^2. This is a percentage error of 2.2% and as such quite accurate and acceptable.

APPENDIX 3

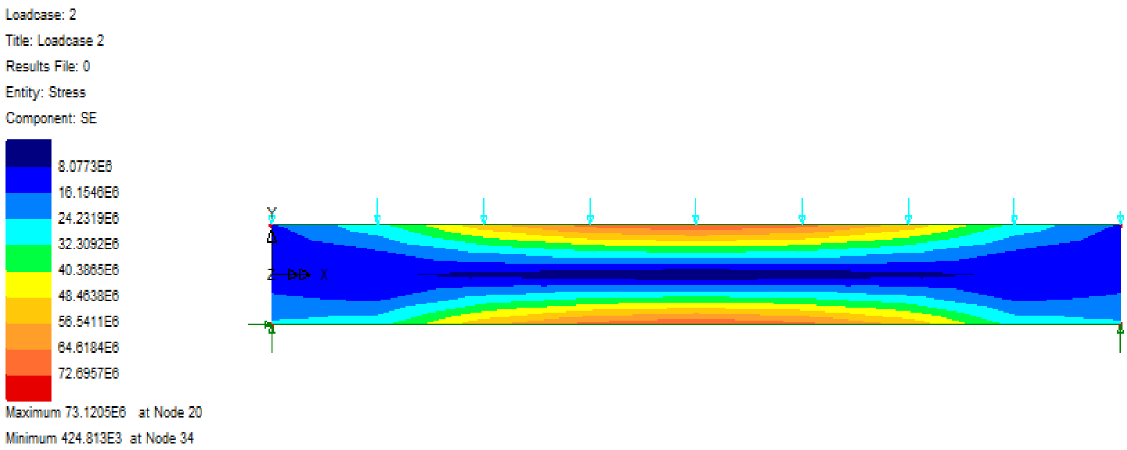
The following pictures are screen shots taken of the same member but with varying mesh sizes and shapes. This allows for a comparison to take place and the optimum mesh size and shape to be decided upon. It should be noted that the student edition of Lusas is quite constricting on the number of applicable nodes.



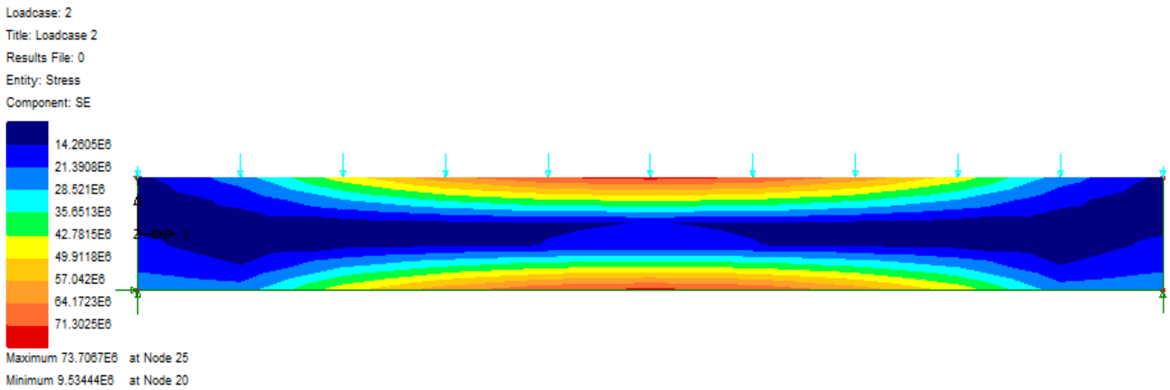
Quad 2x1



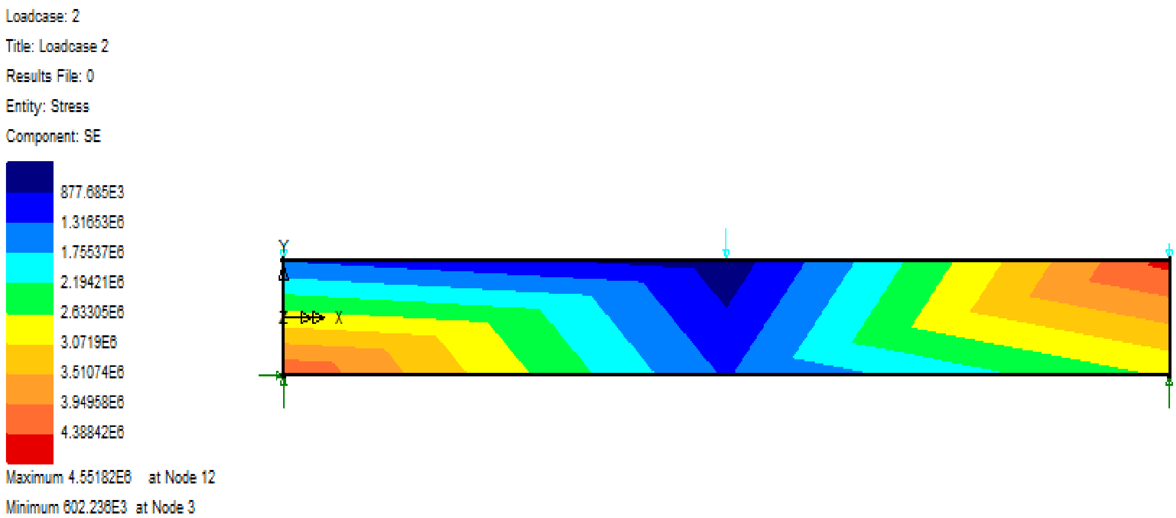
Quad 4x2



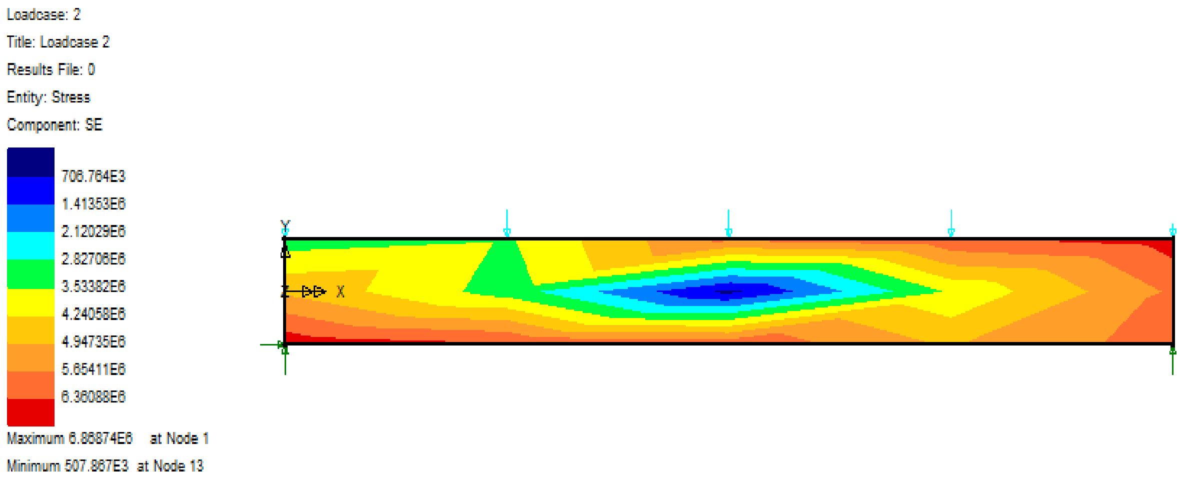
Quad 8x4



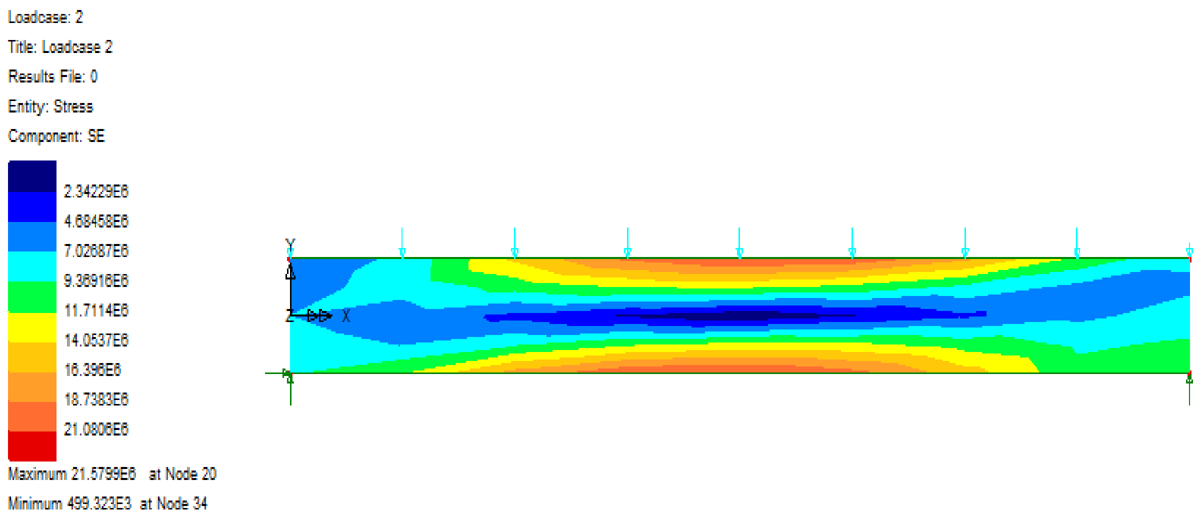
Quad 10x5



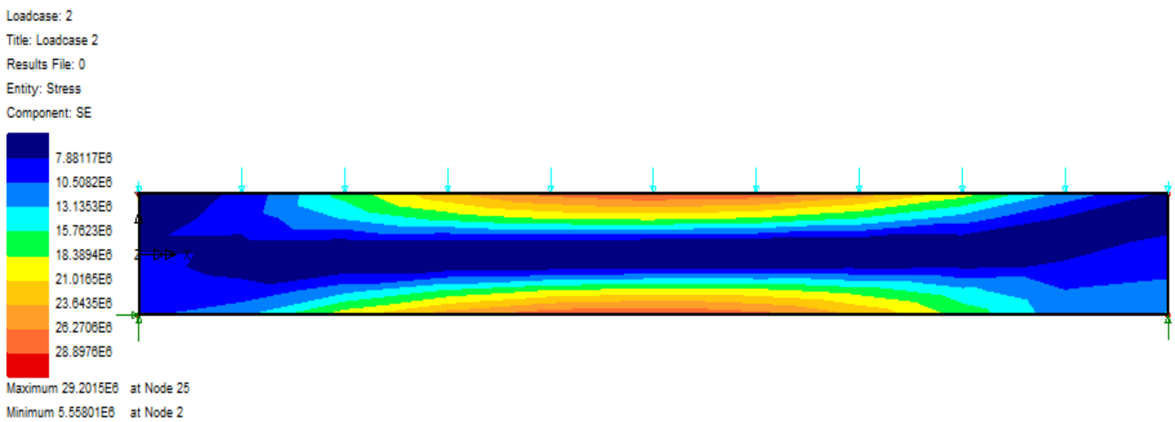
Tri 2x1



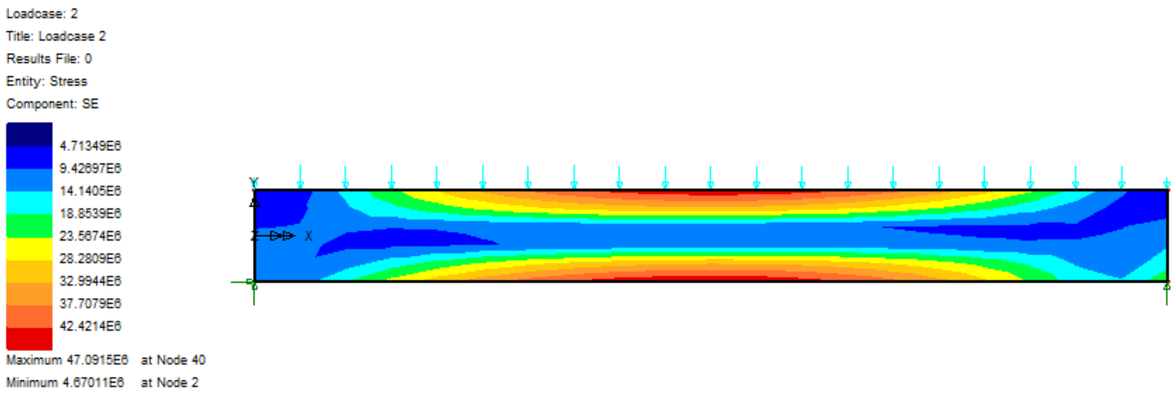
Tri 4x2



Tri 8x4



Tri 10x5


Loadcase: 2
Title: Loadcase 2
Results File: 0
Entity: Stress
Component: SE
4.71349E6
9.42697E6
14.1405E6
18.8539E6
23.5674E6
28.2809E6
32.9944E6
37.7079E6
42.4214E6
Maximum 47.0915E6 at Node 40
Minimum 4.67011E6 at Node 2

Tri 20x5

REFERENCES:

- MacLeod, I.A. (2005) Modern structural analysis Modelling process and guidance. London: Thomas Telford Publishing.
- Spyrakos, C.C. (1994) Finite Element Modelling in Engineering Practice. West Virginia: West Virginia University Press.
- Smith, J.W. (1998) Vibrations of Structures Applications in civil engineering design. London: Chapman and Hall.
- Gere, J.M. (2001) Mechanic of Materials Fifth SI Edition. Cheltenham: Nelson Thornes Ltd.