Journal of the Nigerian Association of Mathematical Physics
Volume 30, **(May,** ***2015*****), pp305 – 314**
© J. of NAMP

Modelling Convergence of Finite Element Analysis of Cantilever Beam

S. O. Osuji and S. A. Adegbemileke

Department of Civil Engineering, Faculty of Engineering,
University of Benin, Benin City, Nigeria.

Abstract

Convergence studies are carried out by investigating the convergence of numerical results as the number of elements is increased. If convergence is not obtained, the engineer using the finite element method has absolutely no indication whether the results are indicative of a meaningful approximation to the correct solution. There are two major methods of mesh refinement; h-refinement and p-refinement.

The cantilever beam plate was modelled using Abaqus/CAE 6.12-1, a finite element analysis tool. The geometry consists of a 300 x 100 mm beam section, spanning 3m and fixed at one end. A load of 1kN was applied at the free end. Also the model was meshed using 2D plane stress linear and quadratic quadrilaterals elements (CPS4R and CPS8), triangular elements (CPS3 and CPS6) and refined. For the linear quadrilateral element, a total of 20, 40,160 and 2560 elements were used for the coarse, medium, fine and very fine mesh respectively. Total numbers of 33, 63, 205 and 2737 nodes were generated accordingly.

The maximum bending stresses and shear stresses occurred at the fixed end. Exact stress and maximum displacement value at the mid-top fibre and free end of the beam was 100 N/mm^2 and-19.5122 mm respectively. Simulated results at these points were analysed using the four element types at different mesh refinement levels. The study shows that linear FE converges slower compared to quadratic elements. Also a finer mesh is required to predict accurate stresses than is needed to calculate accurate displacements.

Keywords: Cantilever beam, finite element, Abaqus, convergence, stress, strain.

1.0 Introduction

Finite element methods have been used for many years with success in the analysis of complex structures and many aerospace structures are designed on the basis of these analysis. In spite of such widespread use, not enough is known of the theoretical accuracy and convergence properties of these models when used to represent a structure. Accuracy studies are usually based on numerical solutions to restricted problems for comparison with known results. Convergence studies are carried out by investigating the convergence of numerical results as the number of elements is increased [1].

Solution accuracy is judged in terms of convergence as the element “mesh” is refined [2]. There are two major methods of mesh refinement. In the first, known as h-refinement, it involves increasing the number of elements used to model a given domain, consequently, reducing individual element size. In the second method, p-refinement, element size is unchanged but the order of the polynomials used as interpolation functions is increased. Also it is possible to combine efficiently the h- and p-refinements and this is called the hp-refinement [3].

Generally, local mesh refinements should be such that very small elements are not placed adjacent to very large ones [4]. A good model involves a fine mesh in areas of stress concentration and a coarse mesh in the remainder of the model. To produce a good mesh, a stress distribution should be known beforehand. However, stresses are not known before meshing but there is need for a good understanding of the stress pattern before analysis.

The need for convergence during regular mesh refinement is rather clear. If convergence is not obtained, the engineer using the finite element method has absolutely no indication whether the results are indicative of a meaningful approximation to the correct solution.

Corresponding author: S.O. Osuji, E-mail: sylvester.osuji@uniben.edu, Tel.: +2348030726214

2.0 Types of Finite Elements

Bhavikatti [6] classified elements based on shapes as: (i) One-dimensional elements (ii) Two dimensional elements (iii) Axi-symmetric elements and (iv) Three-dimensional elements. In two-dimensional analysis the simplest element shapes are obviously a triangle and a rectangle, defined by three or four nodes respectively. We need two dimensional elements to solve two dimensional problems. Common two dimensional problems in stress analysis are plane stress, plane strain and plate problems [6].

Abaqus provides several different types of two-dimensional elements. For structural applications these include plane stress elements and plane strain elements. Plane stress elements can be used when the thickness of a body or domain is small relative to its lateral (in-plane) dimensions. The stresses are functions of planar coordinates alone, and the out-of-plane normal and shear stresses are equal to zero. Plane stress elements must be defined in the X–Y plane, and all loading and deformation are also restricted to this plane. This modelling method generally applies to thin, flat bodies [7].

Size of elements influences the convergence of the solution directly and hence it has to be chosen with care. If the size of elements is small, the final solution is expected to be more accurate. However, we have to remember that the use of elements of smaller size will also mean more computational time. Another characteristic related to the size of elements, which affects the finite element solution, is the aspect ratio of elements. Elements with an aspect ratio of nearly unity generally yield best results. However, the use of large number of elements involves large number of degrees of freedom and we may not be able to store the resulting matrices in the available computer memory [8].

2.1 Shape Function

In the finite element analysis aim is to find the field variables at nodal points by rigorous analysis, assuming at any point inside the element basic variable is a function of values at nodal points of the element. This function which relates the field variable at any point within the element to the field variables of nodal points is called shape function. This is also called as interpolation function and approximating function. In two dimensional stress analysis in which basic field variable is displacement,

$$u = \sum N_i u_i \;; \qquad v = \sum N_i v_i \tag{1}$$

where summation is over the number of nodes of the element.

2.2 Choices of Mesh

Reddy [4] suggested the following guidelines for generation of finite element mesh:

1. The mesh should represent the geometry of the computational domain and load representation accurately.
2. The mesh should be such that large gradients in the solution are adequately represented.
3. The mesh should not contain elements with unacceptable geometries, especially in regions of large gradients.

Within the above guidelines, the mesh used can be coarse (i.e., have few elements) or refined (i.e. have many elements), and may consist of or more orders and types of elements (e.g., linear and quadratic, triangular and quadrilateral). A judicious choice of element order and type could save computational cost while giving accurate results. It should be noted that the choice of elements and mesh is problem-dependent.

2.3 Convergence Requirements

The solution to an analysis is greatly affected by convergence. A solution that has not converged could provide very inaccurate results. In finite element analysis, solution accuracy is judged in terms of convergence as the element "mesh" is refined.

For a general field problem in which the field variable of interest is expressed on an element basis in the discretized form

$$\emptyset^{(e)}(x,y,z) = \sum_{i=1}^{M} N_i(x,y,z)\emptyset_i \tag{2}$$

Where *M* is the number of element degrees of freedom, the interpolation functions must satisfy two primary conditions to ensure convergence during mesh refinement: the compatibility and completeness requirements [9].

2.4 Cantilever Beam

A cantilever is a beam anchored at only one end. The beam carries the load to the support where it is forced against by a moment and shear stress [10]. Cantilever construction allows for overhanging structures without external bracing. Cantilevers are widely found in construction, notably in cantilever bridges and balconies. In cantilever bridges, the cantilevers are usually built as pairs, with each cantilever used to support one end of a central section. They are also used as parking canopies, traffic light stand, and billboards. Another use of the cantilever is in fixed-wing aircraft design, pioneered by Hugo Junkers in 1915 [11].

3.0 Methodology

3.1 Finite Element Method

The Finite Element Method (FEM) is a key technology in the modelling and simulation of advanced engineering systems. The behaviour of a phenomenon in an engineering system depends upon the geometry or domain of the system, the property of the material or medium, and the boundary, initial and loading conditions.

3.1.1 Abaqus [7]

Abaqus is a product of Dassault Systèmes Simulia Corp., Providence, RI, USA. It is an indispensable software package due to its abilities of dealing with nonlinear problems. An Abaqus model is composed of several different components that together describe the physical problem to be analysed and the results to be obtained. At a minimum the analysis model consists of the following information: discretized geometry, element section properties, material data, loads and boundary conditions, analysis type, and output requests.

3.2 Modelling of the Geometry

The cantilever beam was modelled as a rectangular section of 300 x 100 mm and a span of 3000mm using the Part Module and then partitioned mid-way so as to have nodes and obtain results on the neutral axis. The beam was represented as a 2D solid, where one coordinate (z-axis) was removed (Figure 1).

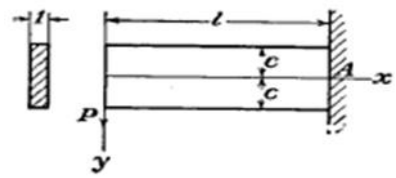

Figure1: Idealized cantilever beam model

3.2.1 Material and Section Properties

The next step in creating the model involves defining and assigning material and section properties to the part. Each region of a deformable body must refer to a section property, which includes the material definition. The material used was steel with Young's modulus of $205 kN/mm^2$ and the Poisson's ratio of 0.3 to set the structural behaviour of the part. As these material parameters were chosen, the section parameters have to be assigned to the part. A unit thickness was used for the plane stress analysis.

3.2.2 Load and Boundary Condition

A point load of 1kN was applied to the free end of the beam and fixed at other end. The self-weight of the beam was neglected in this analysis for simplicity.

3.3 Analysis and Meshing Procedure

The beam of unit sectional width was analysed using plane stress analysis. The vertical displacements and horizontal stresses at any point on the beam plate were manually calculated using the expressions [12].

$$v = \frac{vPxy^2}{2EI} + \frac{Px^3}{6EI} - \frac{Pl^2x}{2EI} + \frac{Pl^3}{3EI} \quad (3)$$

$$\sigma_x = -\frac{3}{2}\frac{P}{c^3}xy, \quad (4)$$

The beam was modelled using:

- **CPS4R**: A 4-node bilinear plane stress quadrilateral, reduced integration, hourglass control element and remeshed using *h*-refinement;
- **CPS8**: An 8-node biquadratic plane stress quadrilateral and remeshed using h*p*-refinement;
- **CPS3**: A 3-node linear plane stress triangle and remeshed using *h*-refinement;
- **CPS6**: A 6-node quadratic plane stress triangle and remeshed using *hp*-refinement.

An initial element size h (300mm) was used in the first coarse mesh, subsequent medium (normal) and fine meshes had element sizes h / λ and h / λ^2, where λ is the scale factor; typically $\lambda \approx 2$.

The influence of the mesh density on two particular results was considered from this model:

i. The displacement of a region of the model.
ii. The bending stress on a region of the model.

Coarse meshes are often adequate to predict trends and to compare how different concepts behave relative to each other. In general, refinement of the mesh shall be made mainly in regions where accurate results are required. A finer mesh is required to predict accurate stresses than is needed to calculate accurate displacements.

4.0 Result and Discussion

Figure 2 below shows the cantilever beam meshed with a 4-noded linear quadrilateral element. A total of 20, 40,160 and 2560 elements were used for the coarse, medium, fine and very fine mesh respectively. Total number of 33, 63, 205 and 2737 nodes were generated accordingly.

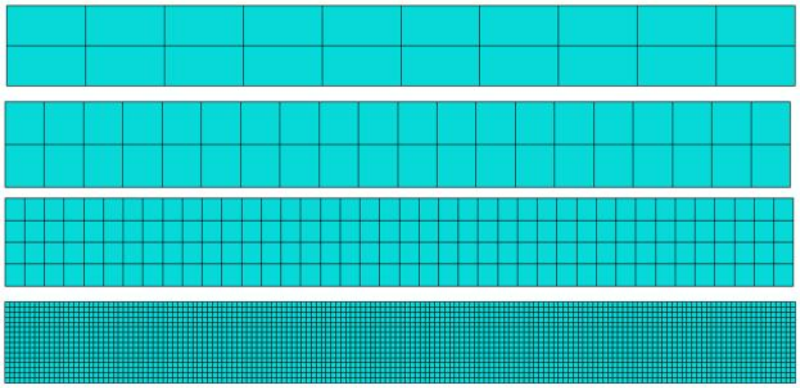

Figure 2: Coarse, medium, fine and very fine mesh using 4-noded linear quadrilateral elements (*h*-refinement)

The horizontal stress, shear stress, maximum and minimum principal stress distributions are shown in Figures 3a- 3d for the quadratic quadrilateral element mesh. The maximum horizontal stresses (±206.0 N/mm^2, tension and compression) occurred at the fixed end of the beam. Positive stresses, i.e tensile stresses, occurred above the neutral axis while the negative, i.e. compressive stresses, occurred below the neutral axis of the beam. Minimum values of horizontal stresses occurred around the neutral axis. Maximum shear stress (21.1 N/mm^2) was also recorded at the fixed end of the beam. The maximum and minimum principal stresses occurred around the top and bottom fibre towards the support as shown in Figures 3c and 3d.

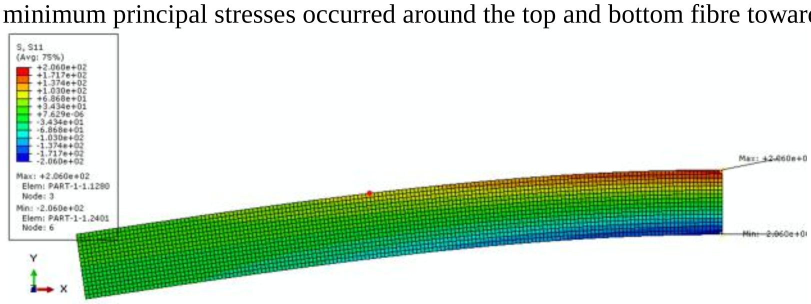

Fig 3a: Horizontal stress distribution (S_{11})

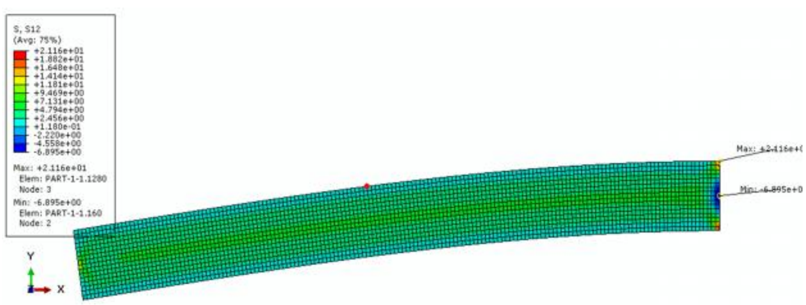

Fig 3b: Shear stress distribution (S_{12})

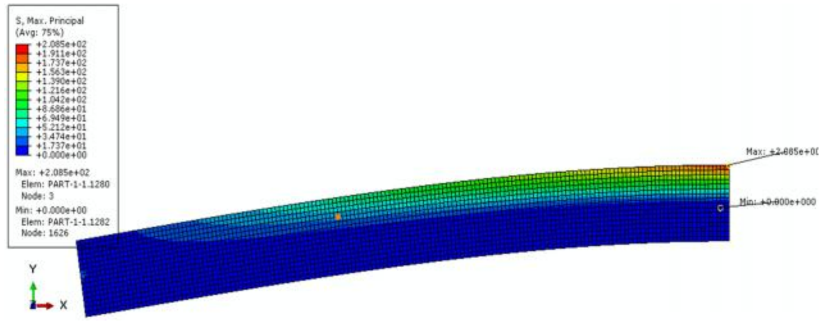

Fig 3c: Maximum Principal stress (σ_1) distribution

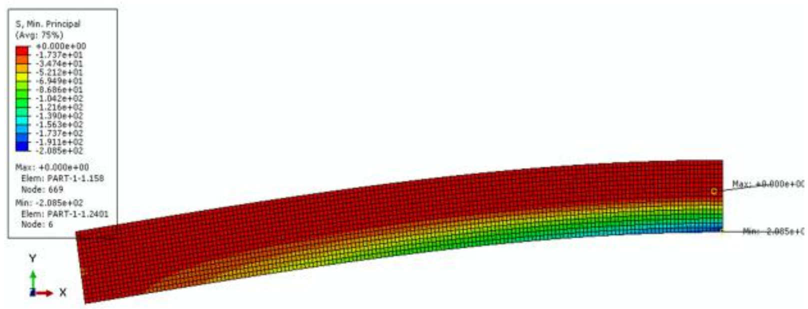

Figure 3d: Minimum Princippal stress (σ_2) distribution

Table 1 shows the analytic and simulated result using the linear quadrilateral element, CPS4R. The stress at mid-point of the top fibre (coordinate 1500, 150) and the maximum displacement at the free end along neutral axis (coordinate 0, 0) was studied for convergence. The exact stress and maximum displacement values as shown in the table are 100N/mm^2 and -19.5122mm respectively. The negative sign shows downward displacement. The simulated stress and displacement values ranges from 66.5068 to -94.1166 N/mm^2 and 26.0492 to -19.7231mm respectively from coarse meshing to very fine meshing. Also the percentage error (i.e. deviation from the exact solution) for stress and displacement simulations ranges from 33.49 to 5.88 % and 33.50 to 1.08 % respectively.

Table 1: Analytic and simulated solution using linear quadrilateral element, CPS4R

MODEL	**σ (N/mm2) EXACT**	**σ (N/mm2) SIMULATED**	**Δσ (N/mm2)**	**ERROR (%)**	**δmax(mm) EXACT**	**δmax (mm) SIMULATED**	**Δδmax (mm)**	**ERROR (%)**
Coarse mesh	100	66.5068		33.49	-19.5122	-26.0492		33.50
Medium mesh	100	66.6026	0.0958	33.40	-19.5122	-26.1461	-0.0969	34.00
Fine mesh	100	79.9846	13.3820	20.02	-19.5122	-20.9424	5.2037	7.33
Very fine mesh	100	94.1166	14.1320	5.88	-19.5122	-19.7231	1.2193	1.08

Using the second element type i.e. the 8-noded quadratic quadrilateral, and at the points of interest stated earlier, the simulated stress values yielded the exact result. However the maximum displacement values ranges from -19.5852 to -19.6382mm, and maximum percentage error deviation of 0.65% as shown in Table A.1 (see Appendix A).

The 3-noded linear triangular element mesh is shown in Figure 4a. The coarse, medium, fine and very fine meshes consist of a total of 40, 80, 320 and 5120 elements respectively. And a total of 33, 63, 205 and 2737 nodes were generated for the meshes accordingly. Also the 6-noded quadratic triangular element mesh is shown in Figure 4b. Though the numbers of elements are the same for every mesh as in the 3-noded triangular element, the numbers of nodes differ. A total of 105, 205, 729 and 10593 nodes were generated for the coarse, medium, fine and very fine mesh respectively.

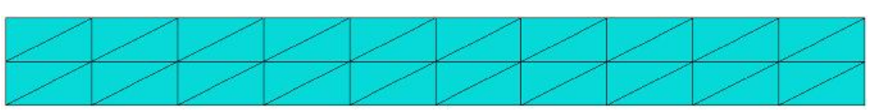

Figure 4a: Meshed model using the 3-noded linear triangular elements

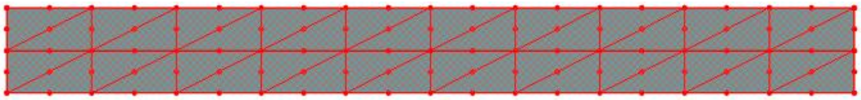

Figure 4b: Meshed model using the 6-noded quadratic triangular elements

The simulated stress and max displacement values at the stipulated points are presented in Table A.2 (see Appendix A); the values range from 25.1622 to 94.6026N/mm^2 and -7.2411 to -19.3669mm respectively. The percentage error ranges from 74.84 to 5.40% and 62.89 to 0.74% on stress and max displacement respectively. Also the max displacement result shows quick convergence compared to stress values.

The refinement procedure for the 6-noded quadratic triangular element shows close results to the exact solution for stress and max displacement values. Also the max displacement result shows divergence of result, though very negligible (max percentage error of 0.64%).

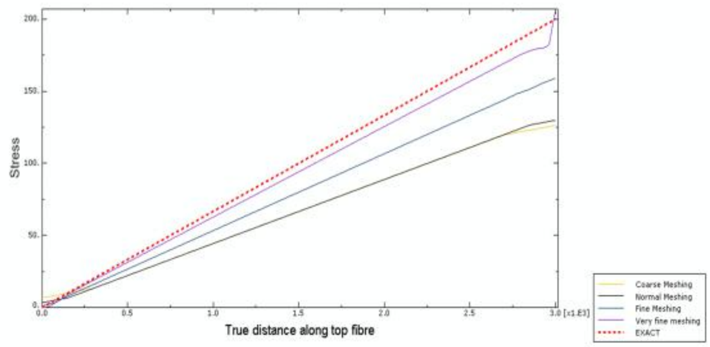

Figure 5a: Stress distribution along top fibre for analytic and simulated solutions using 4-noded linear quadrilateral element (*h*-refinement).

Stress distribution along the top surface of the beam was also determined analytically and using the FE method for the four element types at different levels of refinement. The results were plotted as shown in Figures 5a- 5d.

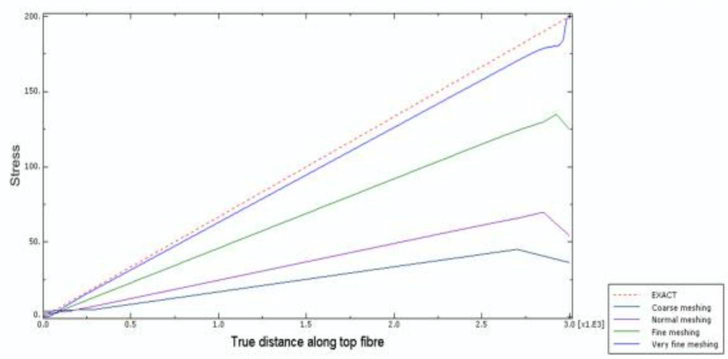

Figure 5b: Stress distribution along top fibre for analytic and simulated solutions using 3-noded linear triangular element (*h*-refinement).

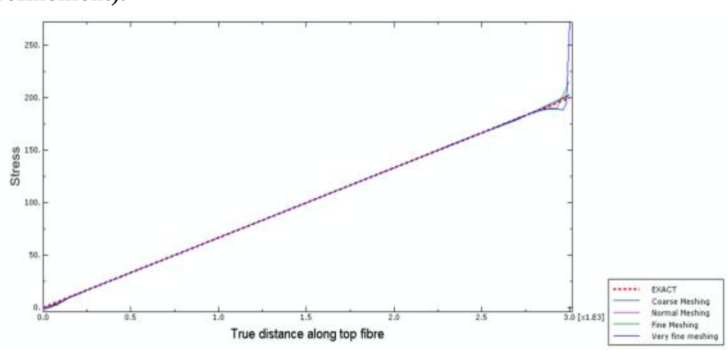

Figure 5c: Stress distribution along top fibre for analytic and simulated solutions using 8-noded quadratic quadrilateral element (*hp*-refinement).

Also the displacement along the neutral axis was determined analytically and using the FE method for the four different element types and levels of refinement. The results are shown in Figures 5a - 5d.

The convergence results of stresses at the mid-top fibre presented earlier show that the linear elements produce unsatisfactory solution when used in finite element analysis without proper refinement of the mesh. The linear quadrilateral elements produced better solution than the linear triangular elements, especially on a coarse mesh. This is seen by the stress value of 66.6026 N/mm^2 for the medium mesh of the linear quadrilateral element, CPS4R (see Table 1); closer to the exact solution (100 N/mm^2) than that using the linear triangular element, CPS3 (36.7649 N/mm^2, see Table A.2). Figures 5a, 5b, 6a and 6c show this observation very clearly in the stress distribution along the top fibre and maximum displacement values along the neutral axis of the beam.

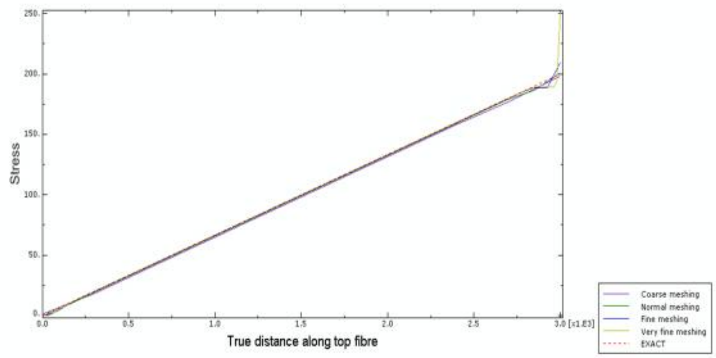

Figure 5d: Stress distribution along top fibre for analytic and simulated solutions using 6-noded quadratic triangular element (*hp*-refinement).

Also the maximum displacements result shows quick convergence than in the stress computation. This means that a finer mesh is required to predict accurate stresses than is needed to calculate accurate displacements as also discovered by Cugnoni [13].

Ergatoudis et al. [14] stated that both the triangular and rectangular elements present the lowest possible forms of approximation and are of limited accuracy. This was observed especially in the coarsely and medially refined meshes. However, further refinement has shown to produce better and very close result.

Using the quadratic elements in the meshing, the displacements result tends to match up with the exact solution even with the coarse mesh. This shows the accuracy of quadratic elements in predicting displacement values. The convergence test shows that using *hp*-refinement to determine displacement solution leads to divergence of the result. However this divergence is very negligible even at a very fine meshing (a maximum of 0.65% from the exact displacement solution). *P*-refinement would however be sufficient for a considerable choice of mesh size. More so, the stress values converge very fast with the quadratic element types.

The divergence in the displacement result using the quadratic elements, (though negligible, max of 0.65%) shows that quadratic elements do not necessarily result in accurate result. This was argued by Kurowski [5] that models made with any order may or may not be accurate. It depends on how well the real stress distribution is represented with the given mesh. Higher order elements converge faster and models usually need fewer of them, however, by themselves, they are not any more or less accurate than lower order elements.

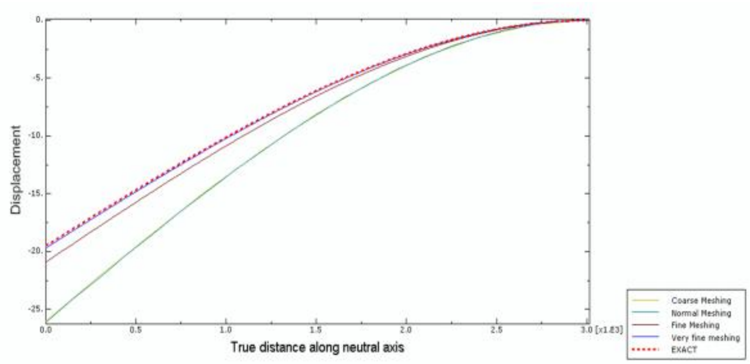

Figure 6a: Displacement along neutral axis for analytic and FEM using 4-noded linear quadrilateral element (*h*-refinement).

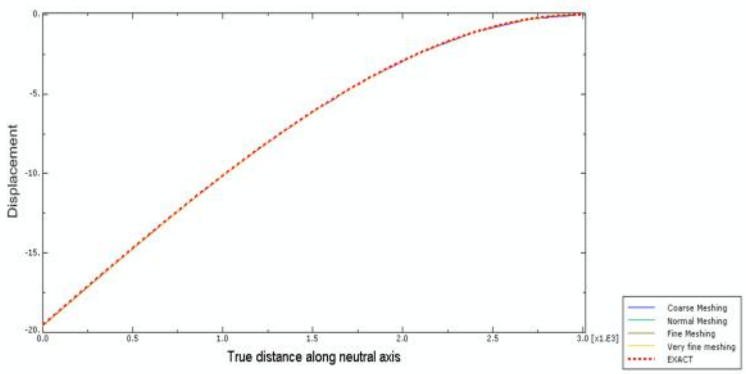

Figure 6b: Displacement along neutral axis for analytic and FEM using 8-noded quadratic quadrilateral element (*hp*-refinement).

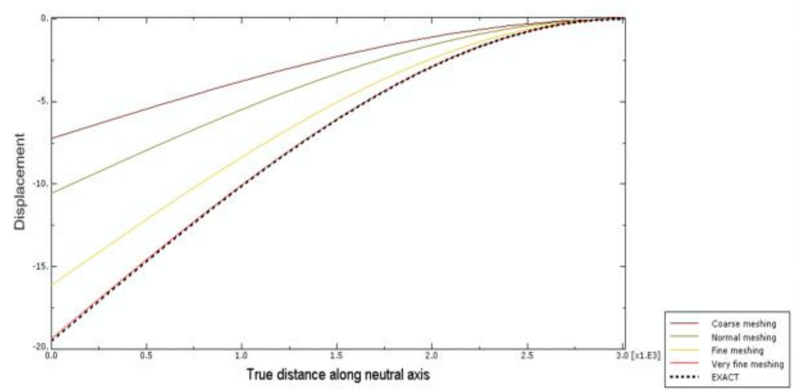

Figure 6c: Displacement along neutral axis for analytic and FEM using 3-noded linear triangular element.

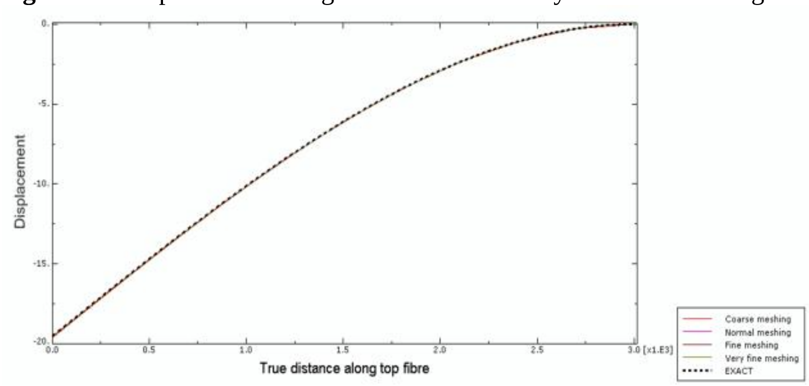

Figure 6d: Displacement along neutral axis for analytic and FEM using 6-noded quadratic triangular element.

The accurate prediction of both stress and displacement values along the path (especially using the quadratic element) as shown in Figures 5and 6 shows the reliability of the Abaqus software in FE solutions.

5.0 Conclusion and Recommendation

The need for convergence during regular mesh refinement is rather clear. If convergence is not obtained, the engineer using the finite element method has absolutely no indication whether the results are indicative of a meaningful approximation to the correct solution. A good model involves a fine mesh in areas of stress concentration and a coarse mesh in the remainder of the model. To produce a good mesh, there is need for a good understanding of the stress pattern before analysis.

The study shows that linear FE converges slower compared to quadratic elements. Also a finer mesh is required to predict accurate stresses than is needed to calculate accurate displacements.

The finite element method of analysis is a powerful tool for numerical analysis, however, the governing mathematics are tedious and complex. Also the availability of veritable commercial software such as Abaqus makes it possible to test and judge solutions.

The following recommendations may be adopted from the convergence study,

1. Choose quadratic elements for stress analysis
2. Use linear elements, but with adequate mesh refinement. However, the computer memory and time will determine the level of refinement.
3. By default use reduced integration. For very thin structures, reduced int. is preferred.
4. Build your own experience by doing convergence tests

Appendix A

Table A.1 Analytic and simulated solution using quadratic quadrilateral element, CPS4R

MODEL	σ (N/mm2) EXACT SOLUTION	σ (N/mm2) SIMULATED	Δσ (N/mm2)	ERROR (%)	δmax (mm) EXACT SOLUTION	δmax (mm) SIMULATED	Δδ (mm)	ERROR (%)
Coarse mesh	100	99.9989		0.0011	-19.5122	-19.5852		0.37
Medium mesh	100	100.0000	0.00	0.0000	-19.5122	-19.6068	-0.0216	0.48
Fine mesh	100	100.0000	0.00	0.0000	-19.5122	-19.6262	-0.0194	0.58
Very fine mesh	100	100.0000	0.00	0.0000	-19.5122	-19.6382	-0.0120	0.65

Table A.2 Analytic and simulated solution using linear triangular element, CPS3

MODEL	σ (N/mm2) EXACT SOLUTION	σ (N/mm2) SIMULATED	Δσ (N/mm2)	ERROR (%)	δmax (mm) EXACT SOLUTION	δmax (mm) SIMULATED	Δδ (mm)	ERROR (%)
Coarse mesh	100	25.1622		74.8378	-19.5122	-7.2411		-62.89
Medium mesh	100	36.7649	11.6027	63.2351	-19.5122	-10.5678	-3.3267	-45.84
Fine mesh	100	68.9529	32.1880	31.0471	-19.5122	-16.1349	-5.5671	-17.31
Very fine mesh	100	94.6026	25.6497	5.3974	-19.5122	-19.3669	-3.2320	-0.74

Table A.3 Analytic and simulated solution using quadratic triangular element, CPS6

MODEL	σ (N/mm2) EXACT SOLUTION	σ (N/mm2) SIMULATED	Δσ (N/mm2)	ERROR (%)	δmax (mm) EXACT SOLTION	δmax (mm) SIMULATED	Δδ (mm)	ERROR (%)
Coarse mesh	100	98.1506		1.8494	-19.5122	-19.5458		0.17
Medium mesh	100	99.139	0.9884	0.8610	-19.5122	-19.5966	-0.0508	0.43
Fine mesh	100	99.7842	0.6452	0.2158	-19.5122	-19.6234	-0.0268	0.57
Very fine mesh	100	99.9865	0.2023	0.0135	-19.5122	-19.6373	-0.0139	0.64

6.0 References

[1] Clough R.W. and Tocher J.L. (1965), “Finite Element Stiffness Matrices for Analysis of Plate Bending.”Proceedings of Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-66-80, 1965.
[2] Joseph E.W., Robert E.F. and Nancy J.C (1968), “Accuracy and Convergence of Finite Element Approximations.” Proceedings of Second Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-68-150, 1968.
[3] Zienkiewicz O.C. and Taylor R.L. (2000), “The Finite Element Method, Volume 1: The Basis”.
[4] Reddy J.N. (2006), “An Introduction to Finite Element Method”, MacGraw-Hill, 3rd edition, 2006
[5] Kurowski P. (1994), “Avoiding Pitfalls in Finite Element Analysis.” Machine Design, Vol. 66 Issue 21, p78
[6] Bhavikatti S.S. (2005), “Finite Element Analysis.” New Age International Publishing Limited, 2005
[7] Abaqus Element Library, retrieved from Abaqus Analysis User’s Manual, Abaqus 6.12, Dassault Systèmes Simulia Corp., Providence, RI, USA, 2015.

[8] Barkanov E. (2001), “Introduction to the Finite Element Method.” Institute of Materials and Structures, Faculty of Civil Engineering, Riga Technical University, 2001

[9] Hutton D.V. (2004), “Fundamentals of Finite Element Analysis.” McGraw-Hill, 1st edition, 2004.

[10] Hool, George A.; Johnson, Nathan C. (1920). "Elements of Structural Theory - Definitions". *Handbook of Building Construction* vol. 1 (1st ed.). New York: McGraw-Hill. p. 2.

[11] Wikipedia (2015), “Cantilever”, retrieved from http://en.wikipedia.org/wiki/Cantilever, 20/02/2015

[12] Timoshenko S. and Goodier J.N. (1951), “Theory of Elasticity”, McGraw-Hill Book Company Inc, 1951, pp

[13] Cugnoni J. (2012), “Convergence & Choice of Finite Element Discretization”, LMAF/EPFL

[14] Ergatoudis I., Irons B.M. and Zienkiewicz O.C. (1968), “Curved, Isoparametric, “Quadrilateral” Elements for Finite Element Analysis.” Int’l Journal on Solids Structures, Vol. 4, pp. 31-42, 1986.