International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 10, Issue 8 (August 2014), PP.20-25

A Study on the Performance of Elements in FEM

D. K. Pandey[1], H.S.Parihar[2], P.Pal[3]
[1]*GLA University Mathura, Uttar Pradesh 281406.*
[2]*GLA University Mathura, Uttar Pradesh 281406.*
[3]*MNNIT Allahabad, Uttar Pradesh 211004.*

Abstract:- This paper deals with the study of the performance of elements in FEM on plate structures. The behaviour of FSDT plates under transverse loading condition subjected to different boundary conditions is observed. The computed results are presented for different type of elements like 3 noded 5 DOF CST element, 6 noded 5 DOF LST element, 4 noded 6 DOF plate bending element, 8 noded 6 DOF Serendipity element and 9 noded 5 DOF Lagrangian elements. The performance of the considered elements is compared with the reported results published in literature. The obtained results are also verified with the analytical solutions by Timoshenko & Kringer [1] to achieve its precision and the range of applicability.

Keywords: - Elements; Plate Structure; ANSYS 12.0; ABAQUS CAE; FEM..

I. INTRODUCTION

The finite-element method is regarded as an accurate and versatile tool for structural analysis. The plate-bending problem is one of the first problems where finite element was applied at early 1960s. The various difficulties encountered at that time and subsequently could not be overcome satisfactorily and this has made the research in this area still alive in present days. At the beginning, attempts have been made to develop thin-plate elements based on Kirchhoff's hypothesis where the difficulties were mostly concerned with the satisfaction of inter-elemental continuity requirement for transverse displacement w. It requires C^1 continuity of transverse displacement (w) i.e., w and its derivatives should be continuous at the common edges between two elements. Again it has been established that the normal slope cannot be made continuous with the usual nodal unknowns w, θ_x and θ_y. In this context a number of nonconforming elements have been developed, which do not satisfy the continuity requirement of normal slope. Although these elements violate the above continuity requirement but the performance of these elements are found to be satisfactory in many cases.

The simplest element available for the analysis of plane elasticity problem is three noded triangular element. This element is most versatile which can fit into any irregular boundary whereas the curved boundaries may be approximated by a series of straight lines. For structures having regular shaped boundaries, four noded rectangular elements can be effectively employed for the analysis. This element is also referred as bilinear element. A rectangular plate bending element with total twelve degrees of freedom is one of the oldest element for the analysis of plates. A higher order eight noded element can be used for moderately thin and thick plates and also for modelling of curved boundaries. Kocak et al. [2] presented the analysis of thick plates by considering eight noded C0finite elements with transverse shear and normal strain and nonlinear in-plane displacement distribution with respect to the plate thickness. Belounar et al. [3] presented with the development of a new four noded rectangular finite element for the linear analysis of plate bending with transverse shear effect. Pal and Ray [4] studied the progressive failure analysis of laminated composite plates under transverse static loading. The laminate composite plate had been modeled using eight noded iso-parametric plate bending elements with FSDT. Sheikha and Chakrabarti [5] taken six noded triangular elements based on Reddy's higher-order shear deformation plate theory. Bhar et al. (6) presented the significance of using higher-order shear deformation theory (HSDT) over the first-order shear deformation theory (FSDT). Hua et al. (7) presented a FEM model based on a simple higher-order plate theory, which could satisfy the zero transverse shear strain condition on the top and bottom surfaces of plates. Sudhakar et al. (8) introduced a degenerate shell element, using higher order shear deformation theory taking the piezoelectric effects into account. An eight-noded element was used to derive global coupled electro elastic behaviour of the overall structure. Dumir et al. (9) presented for axi-symmetric post buckling response of polar orthotropic laminated moderately thick circular and annular plates subjected to uniformly distributed in plane radial compressive load at the outer edge.

The overall observation that emerges from the aforementioned discussions is that the performance of various types of elements in FEM have perhaps not been addressed significantly in open literature so far, to the best of the knowledge of the present investigator. This very observation generates the due motivation on the performance of FE model using various types of elements.

II. MATHEMATICAL FORMULATION

Isoperimetric element is used in the FE formulation. The geometry of the element and the displacement *W* and the rotations θ_x and θ_y, respectively are expressed by the shape functions.

The geometry of the element is given,

$$x = \sum_{i=1}^{n} N_i x_i \qquad y = \sum_{i=1}^{n} N_i y_i$$

the variation of displacements within an element is expressed in terms of the nodal values,

$$u = \sum_{i=1}^{n} N_i u_i \qquad \theta_x = \sum_{i=1}^{n} N_i \theta_{xi}$$

$$v = \sum_{i=1}^{n} N_i v_i \qquad \theta_y = \sum_{i=1}^{n} N_i \theta_{yi}$$

$$w = \sum_{i=1}^{n} N_i w_i \qquad \theta_z = \sum_{i=1}^{n} N_i \theta_{zi}$$

where the N_i is the shape function, Strain-Nodal displacement relationship yields

$$\{\varepsilon\} = \sum_{i=1}^{n} [B]_i \{X_r\}_e$$

Stress-Strain relationship yields

$\{\sigma\} = [D][B]\{X\}_e$ where, D is the rigidity matrix.

The element stiffness matrix is given as follows

$$[K] = \int_{-1}^{1}\int_{-1}^{1} t[B]^T [D][B][J]\, d\varepsilon\, d\eta$$

where $[J]$ = Jacobian matrix and is given as

$$[J] = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} \end{bmatrix}$$

for plate of size (a×b), the Jacobian matrix is given as

$$[J] = \begin{bmatrix} \dfrac{a}{2} & 0 \\ 0 & \dfrac{b}{2} \end{bmatrix}$$

element loading matrix is given as

$$\{Q\}_e = \int_{-1}^{1}\int_{-1}^{1} t[N]^T q[J] d\xi d\eta$$

the displacement vector can be calculated from a set of linear algebraic equations

$$\{Q\} = [K]\{X\}.$$

III. RESULTS AND DISCUSSIONS

The behaviour of simply supported and clamped isotropic and composite plates under transverse uniformly and concentrated load have been observed. The plate has been modelled using ABAQUS CAE (Ab) and ANSYS 9.0 (An) softwares. The obtained results are compared with the computed results obtained from the developed computer codes (Cc). The performance of various elements in terms of percentage variation with the reported results and their relative computational time is studied in the following examples.

Example-1

Deflection of simply supported isotropic plate subjected to uniformly distributed transverse load with different types of elements has been analyzed in this example. The same problem was solved analytically by Timoshenko & Krieger [1]. For the present study the different types of elements considered are:

Shell 63(4 nodes 6 DOF) for ANSYS 9.0
Shell 63(8 nodes 6 DOF) for ANSYS 9.0
3 noded membrane shell for ABAQUS CAE
4 noded membrane shell for ABAQUS CAE
6 noded(5DOF) for ABAQUS CAE
8 noded(6DOF) for ABAQUS CAE
8 noded (5 DOF) and 9 noded (5 DOF) for computer programming
Geometric and material properties considered are:
Size of plate = 1m×1m
Thickness of plate = 0.025 m
Modulus of elasticity (E) = 200 GPa
Load of intensity = 1 N/m2

The obtained results are compared with the analytical results published by Timoshenko & Krieger [1] and are presented in Fig.1. It has been observed that the obtained results for 3 noded triangular & 4 noded quadrilateral elements are converging at higher mesh size. The obtained results from ABAQUS and ANSYS are almost same for the same degrees of freedom. However, by increasing the number of nodes the deflection is increased, hence for higher order elements the variation is more but they are converging at lower mesh size. The results obtained from computer codes showing convergence at lower mesh size and also showing lesser variation with analytical results. For simply supported plate, 8 & 9 noded plate bending element may be used due to its convergence at lower mesh size i.e. higher noded elements are more cost effective. In simply supported plate, changing the element type, the variation of results is significant.

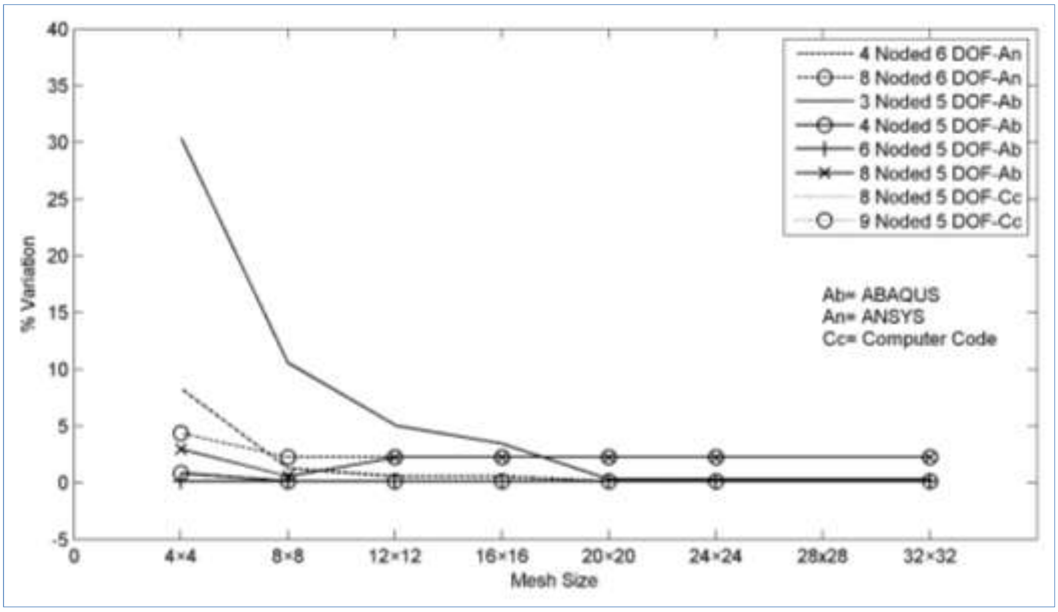

Figure 1: convergence study on the deflection of SSSS plate under UDL condition

Example 2

The performance of elements for clamped isotropic plate has been observed in this example. The geometric and material properties, loading conditions and the type of elements used are same as considered in previous example. The obtained results are compared with the analytical results reported by Timoshenko & Krieger [1] and are shown in Fig.2. It is observed that the results variation for different elements are not too much effective in case of clamped plates, 3 noded element showing convergence at higher mesh size, higher noded elements may be used as they are converging at lower mesh sizes hence these elements are cost effective.

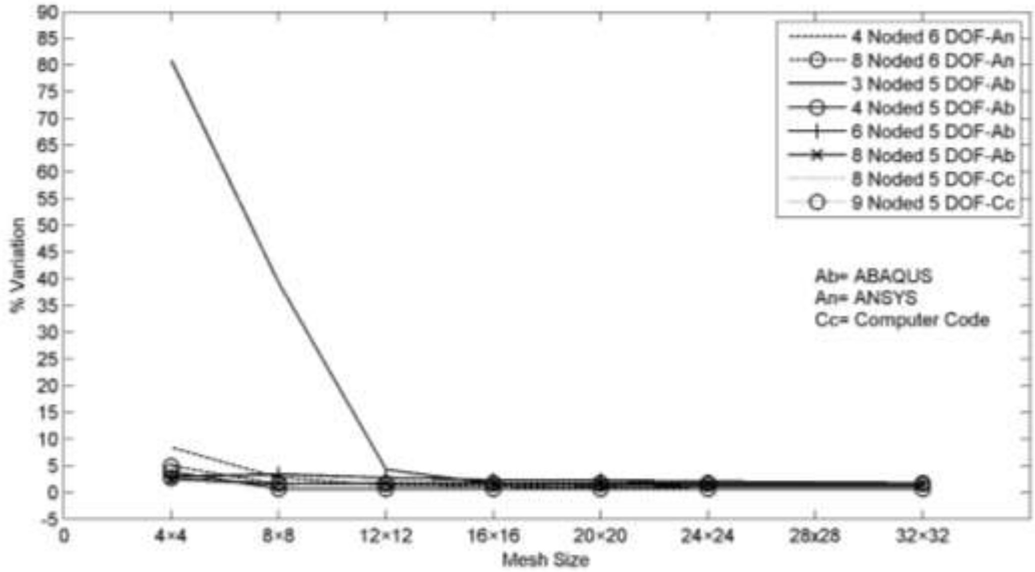

Figure 2: convergence study on the deflection of CCCC plate under UDL condition

Example 3

The performance of elements for simply supported isotropic plate subjected to concentrated load of 1.0 N acting at the center of the plate has been studied in this example. The geometric and material properties and the type of elements remain same considered as in Example 1. The obtained results are compared with the analytical results reported by Timoshenko & Krieger [1] and are shown in Fig.3. It is also observed that 4 noded quadrilateral element showing convergence at higher mesh size, the obtained results from ABAQUS and ANSYS are almost same for same degrees of freedoms, in simply supported plate, changing the element types, the variation of results is significant. For simply supported plate, 8 & 9 noded plate bending element may be used due to its convergence at lower mesh size i.e. higher noded elements are more cost effective. Higher noded elements are showing more variation with analytical results.

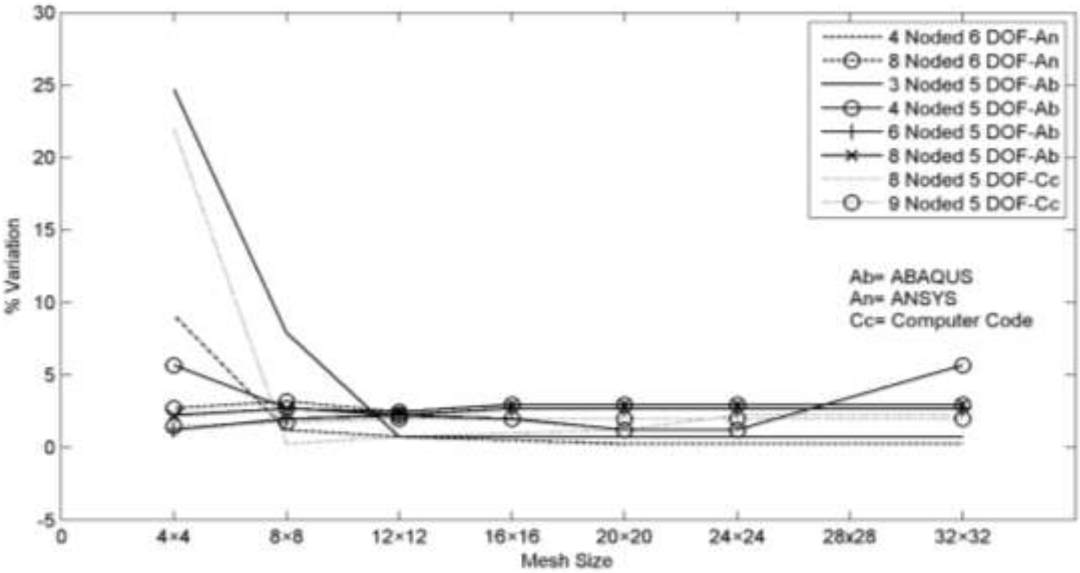

Figure 3: convergence study on the deflection of SSSS plate under point load condition

Example 4

The performance of elements for clamped isotropic plate is observed in this example. The geometric and material properties, loading conditions and the type of elements remain same considered as in example 1. The obtained results are compared with the analytical results reported by Timoshenko & Krieger [1] and are shown in Fig.4. It is observed that, variation on elements is not too much effective in case of clamped plates, 3 noded element showing convergence at higher mesh size, quadrilateral elements (4 noded, 8 noded and 9 noded) may be used as they are converging at lower mesh sizes hence these elements are cost effective.

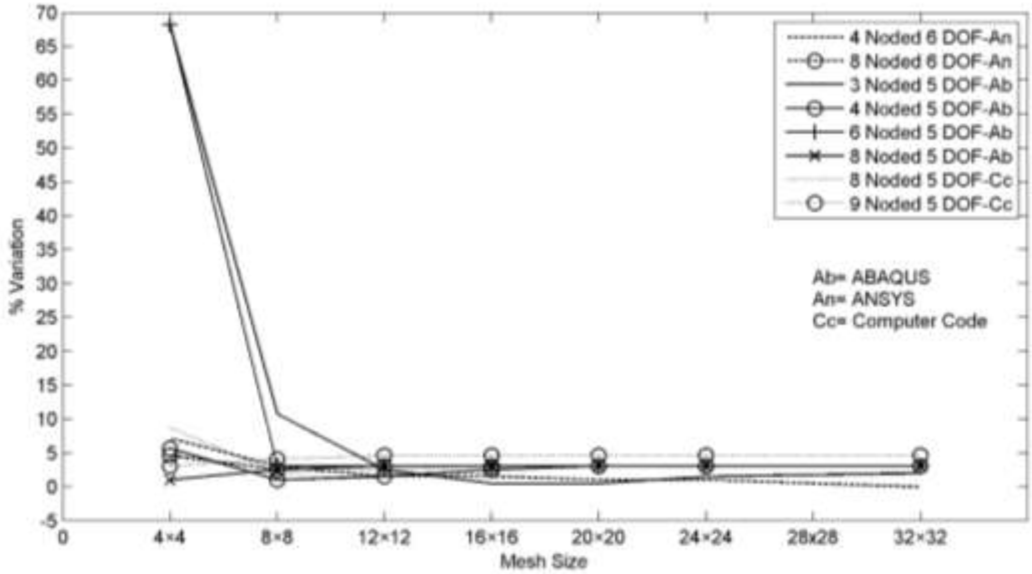

Figure 3: convergence study on the deflection of CCCC plate under point load condition

Example 5

The efficiency of the various types of element depends on the computational cost and accuracy. However, the computational time and accuracy are influenced by some parameters. The most important parameter, called shape functions, which affect both the cost and accuracy. In this example the computational cost has been compared in between the results obtained from ANSYS 9.0, ABAQUS CAE and Computer codes. Computational cost is the different between initial and final time to generate the global stiffness matrix, mass matrix and load vector and finally the solution of algebraic equations. Fig. 5 shows the computational cost versus the mesh size. The figure is plotted from the solution of an isotropic plate problem in the computation of maximum displacement. Various boundary conditions and loading conditions with varying mesh size are considered for the analysis and it is observed that required time is increasing with the increase in mesh size. In case of ABAQUS and ANSYS, required time is not significant within the lower mesh sizes, it may be significant on higher mesh sizes and as well as for laminated composite plates. It is also observed that required time in ABAQUS is more as compared with ANSYS. It is obvious that the computational time depends on the properties of personal computer. The results obtained here are in a computer of i3 series of 2.40 GHz CPU with 2 GB RAM.

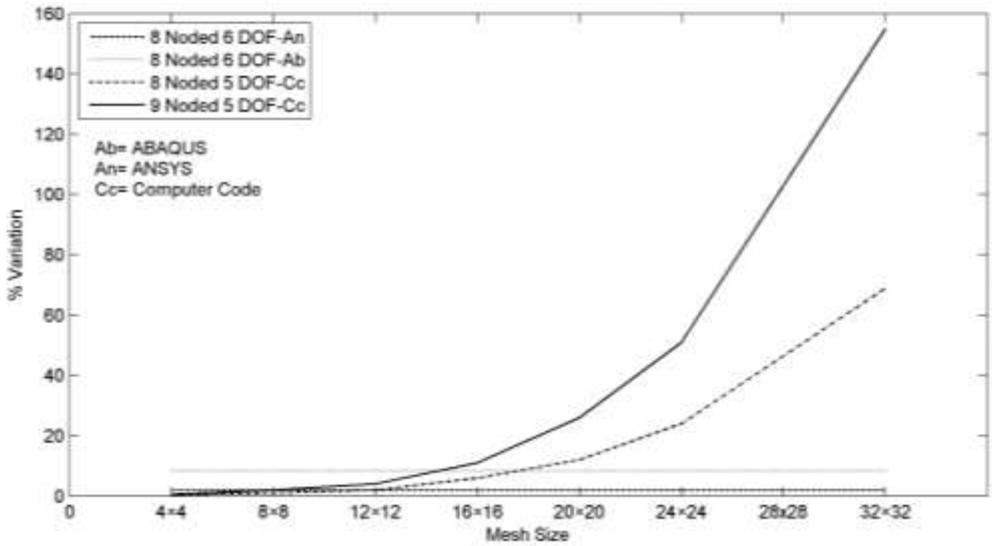

Figure 4: Computational time on isotropic plate

IV. CONCLUSIONS

The present investigation is focused on the case study of isotropic plates with a number of varying elements. Computer codes have been developed using FORTAN software and the numerical examples are presented herein to establish the efficacy of the present study. Based on the investigations carried out, the following conclusions are drawn,

1. Various types of elements having different number of nodes are used in present investigation. The elements can be used to represent any shape of regular or irregular boundaries.
2. For the same problem three different approaches are considered in the present study. Whereas ABAQUS CAE is friendlier to the investigators as the various types of elements are available in the software in comprising to ANSYS 9.0. The obtained results from the developed computer codes are used to verify the performance of elements having higher nodes.
3. For the modeling of plate elements, the required time is less by ANSYS 9.0 in comparison to ABAQUS CAE.
4. It is well known that three noded and four noded elements are converging on higher mesh sizes hence they are not cost effective. Whereas higher noded elements are converging at lower mesh sizes hence they are cost effective.
5. In higher noded elements the deflection is found to be more than lower noded elements, due to more number of nodes.
6. For clamped plate the variation of results due to varying the elements is not significant.
7. For the simply supported plate the lower noded elements may be used. For clamped plate higher noded elements may be preferred

REFERENCES

[1] Timoshenko SP, Winowsky-Krieger S. Theory of plates and shells. 2nd ed.New York: McGraw-Hill; 1959.

[2] Kocak S, Hassis H. A new rectangular finite element based on the strain approach for plate bending. Engineering Structures 2003; 23:131-9.

[3] Belounar L, Guenfoud M. A new rectangular finite element based on the strain approach for plate bending. Thin-Walled Structures 2005; 43:47-17.

[4] Pal P,Ray C. Progressive Failure Analysis of Laminated Composite Plates by Finite Element Method. Journal of Reinforced Plastics and Composites 2002; 21:1505-9.

[5] Sheikh AH, Chakrabarti A. A new plate bending element based on higher-order shear.deformation theory for the analysis of composite plates. Finite Elements in Analysis and Design 2003; 39:883-20.

[6] BharA, PhoenixSS,Satsangi SK. Finite element analysis of laminated composite stiffenedplates using FSDT and HSDT: A comparative perspective. Composite Structures. 2010; 92:312–321.

[7] HuaN,Fukunaga H,Kameyamab M, Aramaki Y, Chang FK.Vibrationanalysis of delaminated composite beams and plates using a higher-order finite element. International Journal of Mechanical Sciences.2002; 44:1479–1503.

[8] KulkarniSA, Bajoria KM.Finite element modeling of smart plates/shells using higher order shear deformation theory. Composite Structures.2003; 62:41–50.

[9] Dumir PC, Dube GP, Mallick A. A new rectangular finite element based on the strain approach for plate bending. Composite Structures.2001; 51:311-318.

[10] Mukhopadhyay M, Sheik AH. Matrix and Finite Element Analyses of Structures. New Delhi: Ane Books Pvt. Ltd.; 2008.