RESEARCH ARTICLE OPEN ACCESS

Study of Convergence of Results in Finite Element Analysis of a Plane Stress Bracket

Gowtham K L*, Shivashankar R. Srivatsa**
**(Department of Mechanical Engineering, B. M. S. College of Engineering, Bengaluru, India - 560 019*
*** (Department of Mechanical Engineering, B. M. S. College of Engineering, Bengaluru., India - 560 019*

ABSTRACT : *A major computational tool for solving engineering problems is Finite Element Analysis. This uses different meshing techniques to divide a complex problem into small elements and solving it using software which is coded with Finite Element Method algorithm. With an increase in the number of elements in modelling, the results tend to be more accurate. Convergence of results depends on nature of the problem and elements used in the analysis. This paper presents a comparison on the effect of using different two-dimensional and three-dimensional elements for the analysis of a plane stress problem using a commercial software. The variation structural response is discussed and compared to six different solid elements of varying mesh sizes. The study is intended for understanding the performance of various two-dimensional and three-dimensional elements in the analysis of plane stress problem using FEM.*

Keywords– *brick element, convergence, plane Stress, quadrilateral element, tetrahedral element, Von-Mises stress*

I. Introduction

The efficiency of a finite element analysis in any software is determined by the ease of modelling and analysis in the commercial packages. In the finite element analysis of a component, mesh size and the type of element used plays an important role in determining the results. This paper presents a brief discussion on the variation of the results and their convergence in the analysis of a stress concentration problem in case of a plane stress bracket.

Number of publications, which deal with the application, study and improvisation of finite elements has increased in the past few years. Razaqpur[1] presents review on the use of a 4-node, 12 degrees-of-freedom quadrilateral elements for analysis of thin plate. Yang.Y and Tang.X[2]constructs a mesh free Quad4 element using mean value coordinates. Eijo, A[3] develops a numerical method to model delamination in laminated plates using the 4-node quadrilateral QLRZ plate element based on the refined zigzag theory. Barrett.K.E[4] investigates the conditions under which it is possible to determine exact integrals for stiffness matrix entries for an eight-node isoparametric plane strain element. Robinson, John[5] defines shape parameters of quadrilateral and hexahedron using the coefficients of the basic shape functions. Naganarayana, B.P. and Prathap, G[6] suggests that a 27-node Lagrangian hexahedral element formulation for robust 3D modelling over the 8-node serendipity element. Cisloiu, R[7] solves small and large deformation problems using a stabilized mixed finite element formulation for 4-node tetrahedral elements. Nejati, Morteza[8] discusses the reproduction of the square root singularity in quarter-point tetrahedral (QPT) finite elements.

Addressing an efficient way of analysis is required to reduce the computational time required as the formulation is widely used in applications like spherical and cylindrical pressure vessels, support brackets and other structural applications.

II. Elements Used and Their Characteristics

2.1 PLANE182: A 4-node quadrilateral element

This is suitable for 2-D modelling of structures. This element can be used as an axisymmetric element or as a plane element (plane stress, plane strain or generalized plane strain). It has four nodes andat each node it has two degrees of freedom: translations in the x and y directions. The element has large deflection, large strain, plasticity, hyper elasticity and stress stiffening capabilities. Deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials can also be simulated as it has mixed formulation capability.[9]

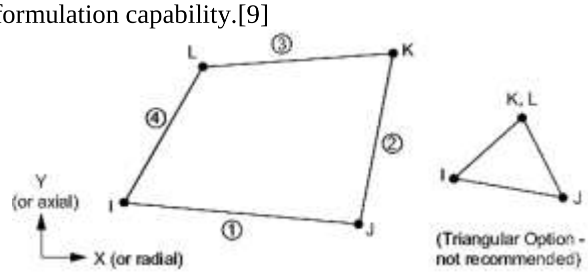

Figure.1: PLANE182 Geometry.

2.2 PLANE183: A 8-node quadrilateral element

This element is defined by 8 nodes or 6 nodes having two degrees of freedom at each node: translationsin the nodal x and y directions. The element may be used as a plane element (plane stress, plane strainand generalized plane strain) or as an axisymmetric element. The element has large deflection, large strain, plasticity, hyper elasticity, creep and stress stiffening capabilities. Deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials can also be simulated as it has mixed formulation capability. Initial state is supported[9]

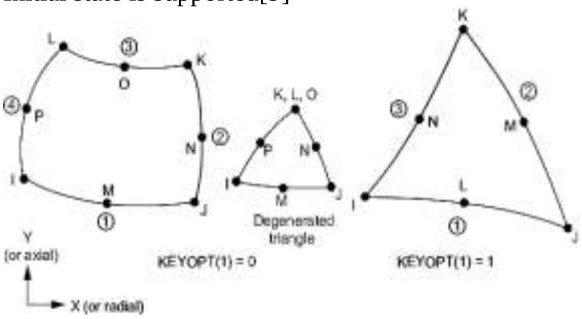

Figure.2: PLANE183 Geometry

2.3 SOLID185: A 8-node brick element

SOLID185 is used for 3-D modelling of structures. It is defined by eight nodes.Each nodes have three degrees of freedom: translations in the nodal x, y, and z directions. The element has large deflection, large strain, plasticity, hyper elasticity, creep and stress stiffening capabilities. Deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials can also be simulated as it has mixed formulation capability. This is available as a homogenous structural solid element and layered structural solid element.[9]

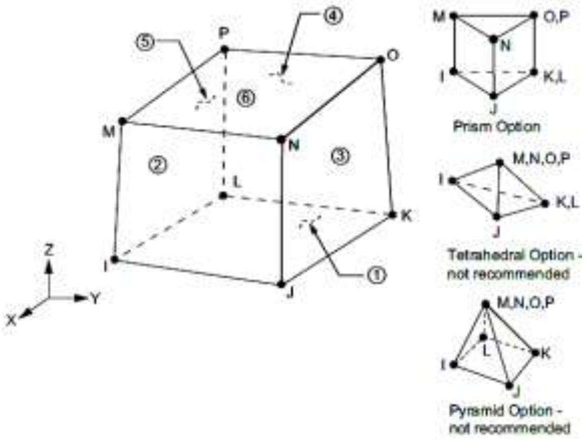

Figure.3: SOLID185 Homogeneous Structural Solid Geometry

2.4 SOLID186: A 20-node brick element

SOLID186 is a higher order 20-node 3D solid element and it exhibits quadratic displacement behaviour.The element is defined by 20 nodes having three degrees of freedom per node: translations in thenodal x, y, and z directions. The element has large deflection, large strain, plasticity, hyper elasticity, creep and stress stiffening capabilities. Deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials can also be simulated as it has mixed formulation capability. Initial state is supported.[9]

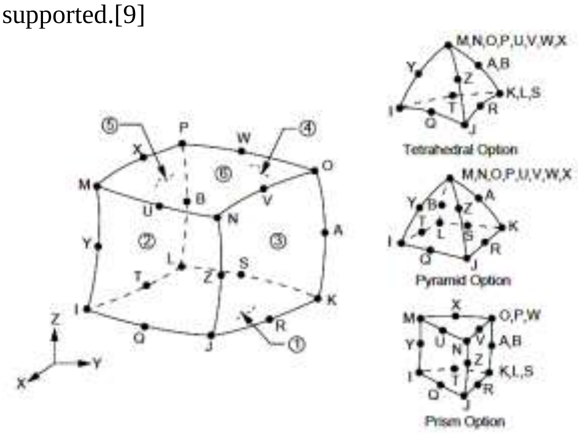

Figure.4: SOLID186 Homogeneous Structural Solid Geometry

2.5 SOLID285: A 4-node tetrahedral element

SOLID285 element is4-node mixed u-P, a lower-order 3-D element. The element has a linear displacementand hydrostatic pressure behaviour. Modelling irregular meshes and general materials can be done using his element (including incompressible materials).

The element is defined by four nodes having four degrees of freedom at each node: three translationsin the nodal x, y, and z directions, and one hydrostatic pressure (HDSP) for all materials exceptnearlyincompressible hyperelastic materials. Volume change rate is used at each node together with the three translation degrees of freedom instead of hydrostatic pressure for nearly incompressible materials. The element has large deflection, large strain, plasticity, hyper elasticity, creep and stress stiffening capabilities. It is capable of simulating deformations of nearly incompressible elastoplastic materials,nearly incompressible hyperelastic materials, and fully incompressible hyperelastic materials.[9]

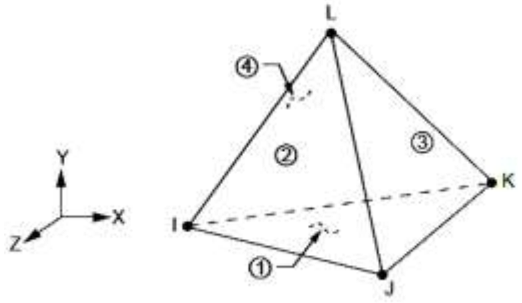

Figure.5: SOLID285 Geometry

2.6 SOLID187: A 10 node tetrahedral element

SOLID187 element is a higher order 3-D, 10-node element. This has a quadratic displacement behaviourand modellingof irregular meshes and general materials can be done using his element. The element is defined by 10 nodes having three degrees of freedom at each node: translations in thenodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, largedeflection, and large strain capabilities. . Deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials can also be simulated as it has mixed formulation capability.[9]

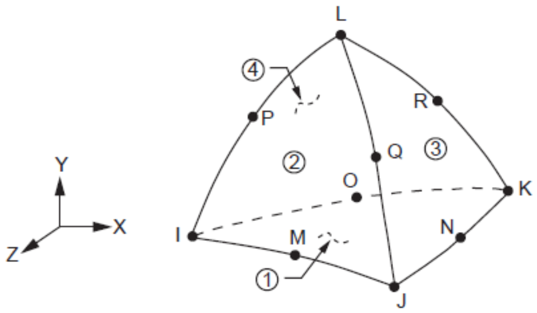

Figure.6: SOLID187 Geometry

III. Modelling and Analysis

The plate is modelled in Ansys Mechanical APDL according to the dimensions given in Fig.7. For the structural analysis using elements PLANE182 and PLANE183, a 2D planar model with thickness is considered. Whereas, for the analysis using solid elements, a 3D model of the same is considered to enforce the boundary conditions. Models are meshed with free mesh except for the brick elements where sweeping is used.

The boundary conditions are defined in Fig.8. A pressure of 1MPa is applied at one end keeping the other end fixed to induce tension in the component.

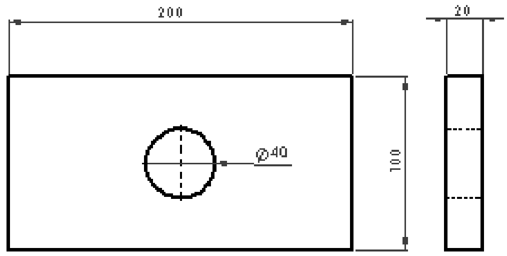

Figure.7: Geometric model

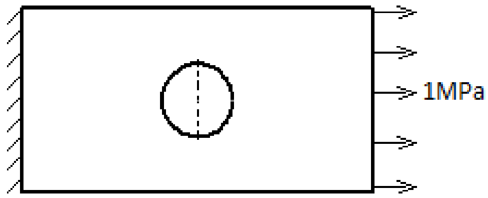

Figure.8: Boundary conditions

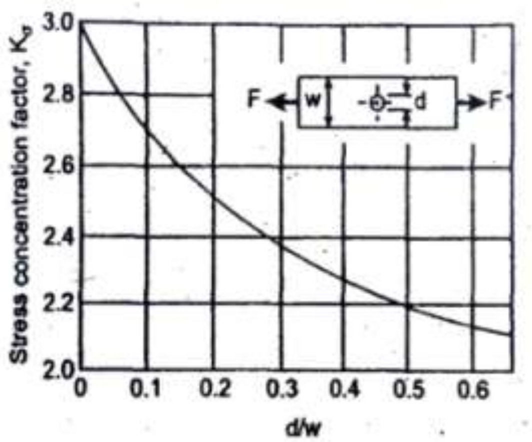

Figure.9: Stress concentration factor for a plate of finite width with a circular hole in tension.

$$K_\sigma = \frac{\sigma_{max}}{\sigma_{nom}} \quad (1)$$

$$\sigma_{nom} = \frac{F}{(w-d)h} \quad (2)$$

The stress concentration factor (K_σ) for the model is taken as 2.28 for a (d/w) ratio 0.4 from Fig.9 [10]

Hence the nominal stress from equation 2 is,

$$\sigma_{nom} = \frac{F}{(w-d)h} = \frac{2000}{(100-40)20} = 1.667 \text{ MPa}$$

The maximum stress developed,according to equation 2 is,

$$\sigma_{max} = 2.28 * 1.667 = 3.80076\text{MPa}$$

Fig.10 shows the finite element model of the plane stress bracket with PLANE183 element and a mesh element size of 5mm.

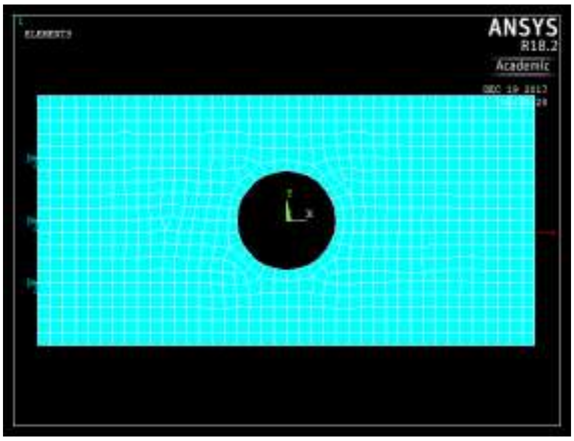

Figure10: Finite Element model.

A structural analysis is carried out to observe the variation of displacement and von Mises stress in all the listed elements for mesh size of 2mm, 3mm, 5mm, 10mm, 15mm, 20mm and 25mm.

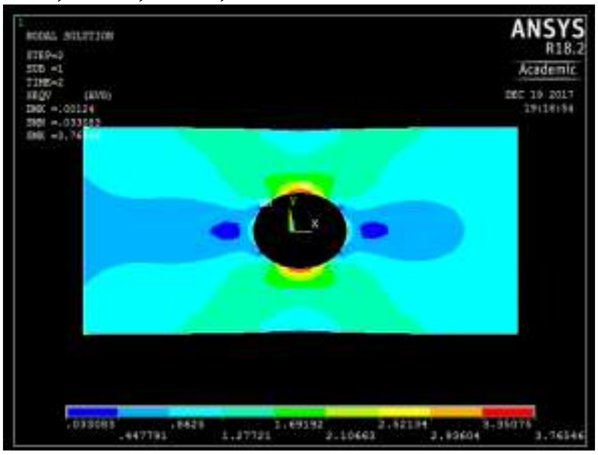

Figure.11: Distribution of von Mises stress

Fig.11 and Fig.12 shows the distribution of von Mises stress and displacementalso the location of maximum values of respective parameters in the specimen. The result shown is for the model with PLANE183 element and for a mesh element size of 5mm.

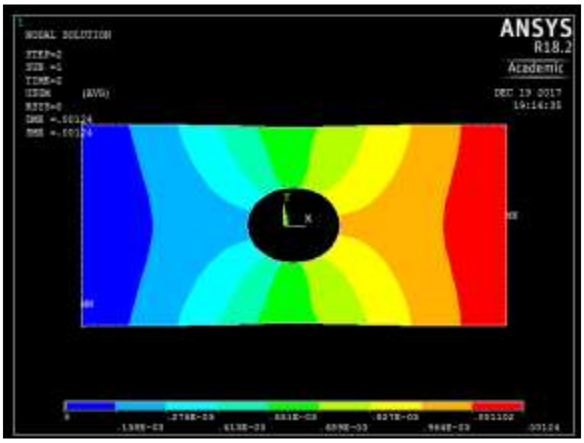

Figure.12: Displacement

IV. Discussion

The analysis results are shown in Table.1

Table.1 Analysis results

Table.1.1 PLANE182: A 4-node quadrilateral element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.001239	3.69704
3	0.001237	3.67476
5	0.001232	3.52173
10	0.001215	3.22583
15	0.001196	3.04703
20	0.001162	2.71594
25	0.001153	2.26218

Table.1.2 PLANE183: A 8-node quadrilateral element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.00124	3.75414
3	0.00124	3.76569
5	0.00124	3.76546
10	0.00124	3.68745
15	0.001234	3.5447
20	0.001235	3.62995
25	0.001237	3.57147

Table.1.3 SOLID185: A 8-node brick element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.001236	3.6599
3	0.001235	3.58496
5	0.001231	3.56214
10	0.001216	3.32887

15	0.001202	3.17431
20	0.00116	2.79115
25	0.001163	2.47368

Table.1.4 SOLID186: A 20-node brick element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.001238	3.77189
3	0.001238	3.77502
5	0.001238	3.80993
10	0.001237	3.74247
15	0.001236	3.72566
20	0.001232	3.5474
25	0.001234	3.5798

Table.1.5 SOLID285: A 4-node tetrahedral element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.001236	3.67025
3	0.001233	3.61177
5	0.001239	3.48981
10	0.001231	2.78483
15	0.001197	2.71004
20	0.001156	2.14832
25	0.001156	2.11592

Table.1.6 SOLID187: A 10-node tetrahedral element

Mesh Size	Displacement(mm)	von Mises Stress(MPa)
2	0.001238	3.79079
3	0.001238	3.79765
5	0.001238	3.79626
10	0.001236	3.82354
15	0.001234	3.76851
20	0.001228	3.65339
25	0.001228	3.70686

It can be noted that with the increase in the mesh size the results obtained are converging in to a more reliable value. The PLANE elements are 2D elements and have less nodes compared to the solid elements. This makes them more reliable to be used in the plane stress analysis as the elements can be used with a thickness being specified in the software. The SOLID elements have more nodes compared to the 2D elements. This makes the formulation more precise.

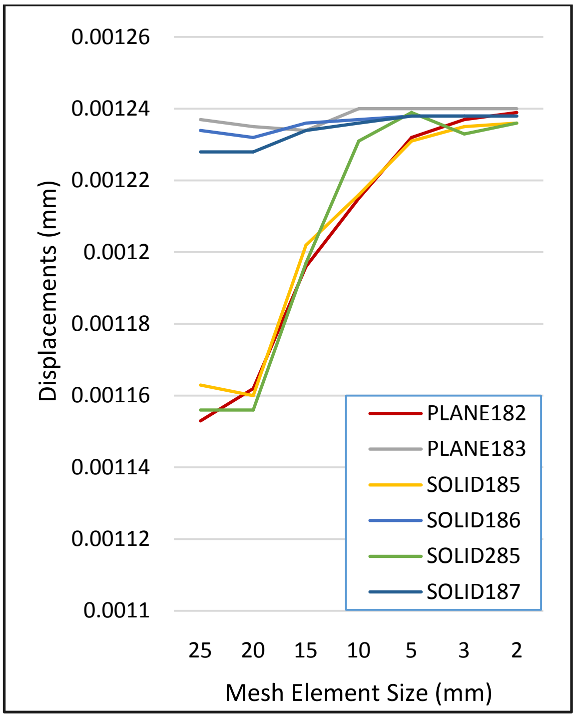

Figure.13: Variation of Displacements with Mesh element size

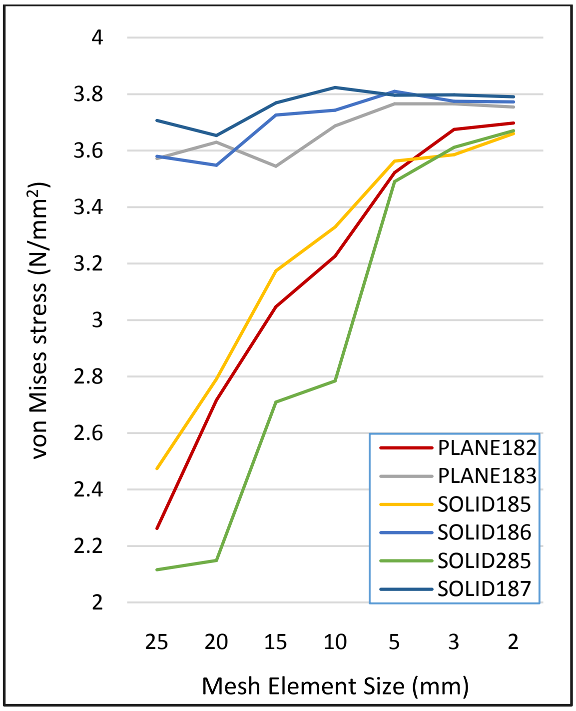

Figure.14: Variation of von Mises stress with mesh element size

From Fig.12 and Fig.13 which shows the variation of displacement and von Mises stresses respectively with mesh element size, we can note that, the elements, PLANE183, SOLID186 and SOLID187 gives nearly accurate results even at the lesser mesh sizes. Making them effective to use in the plane stress analysis. The element SOLID285 even though gives much deviated results with larger mesh sizes, the results are much closer to the converged value for a mesh element length of 10mm and below. Whereas the elements PLANE182 and SOLID185 which are lower order elements show more deviation from the required value.

V. CONCLUSION

A comparison between the results obtained by using different solid elements for a plane stress condition is presented. As it is a plane stress condition, the dimensions in the third direction are negligible compared to the in plane dimensions. The element PLANE183 fits into the problem perfectly as it is a 2D 8-node quadrilateral element along with SOLID186 and SOLID187 elements which seem to give more accurate results even with a coarse mesh. Even though the three-dimensional elements do not give more accurate results using coarse mesh, they perform as good as the plane elements in solving a problem of plane stress. Thus, when there is a complicated model involved in the analysis, one can always rely on the 3D elements for the ease of modelling and higher order elements which give results close to the exact solution with smaller mesh sizes.

REFERENCES

[1]. A. G. Razaqpur, M. Nofal, and A. Vasilescu, “An improved quadrilateral finite element for analysis of thin plates,” Finite Elem. Anal. Des., vol. 40, no. 1, pp. 1–23, Nov. 2003.
[2]. Y. Yang, X. Tang, and H. Zheng, “Construct ‘FE-Meshfree’ Quad4 using mean value coordinates,” Eng. Anal. Bound. Elem., vol. 59, pp. 78–88, Oct. 2015.
[3]. A. Eijo, E. Oñate, and S. Oller, “Delamination in laminated plates using the 4-noded quadrilateral QLRZ plate element based on the refined zigzag theory,” Compos. Struct., vol. 108, pp. 456–471, Feb. 2014.
[4]. K. E. Barrett, “Explicit eight-noded quadrilateral elements,” Finite Elem. Anal. Des., vol. 31, no. 3, pp. 209–222, Jan. 1999.
[5]. J. Robinson, “Quadrilateral and hexahedron shape parameters,” Finite Elem. Anal. Des., vol. 16, no. 1, pp. 43–52, Apr. 1994.
[6]. B. P. Naganarayana and G. Prathap, “Field-consistency analysis of 27-noded hexahedral elements for constrained media elasticity,” Finite Elem. Anal. Des., vol. 9, no. 2, pp. 149–168, Jun. 1991.
[7]. R. Cisloiu, M. Lovell, and J. Wang, “A stabilized mixed formulation for finite strain deformation for low-order tetrahedral solid elements,” Finite Elem. Anal. Des., vol. 44, no. 8, pp. 472–482, May 2008.
[8]. M. Nejati, A. Paluszny, and R. W. Zimmerman, “On the use of quarter-point tetrahedral finite elements in linear elastic fracture mechanics,” Eng. Fract. Mech., vol. 144, pp. 194–221, Aug. 2015.
[9]. T. D. Canonsburg, “ANSYS Mechanical APDL Command Reference,” vol. 15317, no. November, pp. 724–746, 2010.
[10]. K. Lingaiah, Machine design databook. McGraw-Hill, 2006.