EFFECT OF ELEMENT TYPES ON THE RESULT OF FE ANALYSIS

Mabrouk M. AlGAMIL[1,*], Khalifa M. Ahmed[2] , Hassan Guffa [3]

[1] Mech. Engg. Dept., Faculty of Engineering, Sebratha University-Libya
*e-mail: malgamil57@gmail.com
[2] Mech. Engg. Dept., Faculty of Engineering, Zawia University-Libya
e-mail: khalifa_online381@yahoo.com
[3] Mech. Engg. Dept., Faculty of Engineering, Sebratha University-Libya
e-mail: hassanalgoffa@gmail.com

Abstract :

From the very beginning of any analysis tasks the analyst ask him self what type of elements that is suitable for this task ,which element that gives an accurate result .Modern finite element packages have libraries that contain a vast number of elements ranging from simple 1D link element to 3D brick element . These elements increase the capability of finite element (FE) programs to model and solve complex engineering problems. Not only do these elements provide improvement, validity in accuracy of the results but also brought about new challenges which include evaluation of numerical errors of results, setup and execution time as well as large computer memory capacity. The outcome of the analysis is very much dependent of the type of element chosen. The aim of this paper is to investigate the factors influencing the selection of elements in FEA by considering the effects of different types of elements on the results of FEA. A cylindrical pressure vessel is considered for this analysis as an example to demonstrate the importance of element selection .Three different models of the pressure vessel was constructed using ANSYS elements 183 , element 272 and axi har 25 element. The results of these models were compared with the analytical calculation . The FEA results of these models showed good agreement with the theoretical calculation for model meshed with element 183 . Error with element CPT213 and axi har 25 ranged from 12 % to 17% respectively, these might be due to non ax symmetric loading feature of these elements.

Key wards : pressure vessel , element selection , F.E , ANSYS , element type .

1- INTRODUCTION

Cylindrical or spherical pressure vessels (e.g., hydraulic cylinders, gun barrels, pipes, boilers and tanks) are commonly used in industry to carry both liquids and gases under pressure. When the pressure vessel is exposed to this pressure, the material comprising the vessel is subjected to pressure loading, and hence stresses, from all directions. The normal stresses resulting from this pressure are functions of the radius of the element under consideration, the shape of the pressure vessel (i.e., open ended cylinder, closed end cylinder, or sphere) as well as the applied pressure. Two types of analysis are commonly applied to pressure vessels. The most common method is based on a simple mechanics approach and is applicable to “thin wall” pressure vessels which by definition have a ratio of inner radius, r, to wall thickness, t, of $r/t \geq 10$. The second method is based on elasticity solution and is always applicable regardless of the r/t ratio and can be referred to as the solution for “thick wall” pressure vessels.

2- LITERATURE REVIEW

Drazan Kozak et.al [1] made numerical analysis on cylindrical pressure vessel with changeable head geometry i.e.semi-elliptical and hemispherical heads with three types of elements: SOLID 95, PLANE 183 and SHELL 181. It is concluded that in both cases of pressure vessel heads, using of PLANE 183 element presents the best approach, because of minimal number of elements for meshing, shortest calculation time, insight into the stress distribution per plate thickness and obtained results which are closest to the analytical ones. This type of axisymmetric element could b e recommended in such cases, when the total symmetry of model is considered. Also analysis of cylindrical pressure vessel with different head type is performed in purpose of comparison of values of maximal equivalent stresses. It is concluded that smaller values of

equivalent stresses are appearing in pressure vessel with hemispherical heads, and equivalent stress distribution is advantageous too in that case of head geometry.

Porter & Martens, 1996 (2), the authors demonstrated that five different FEA software codes produced comparable results in the analysis of a typical thin wall nozzle-to-shell junction where the indicated stresses remained below the material yield point. Where the indicated stresses were above yield, considerable divergence was noted. In order to explore the stress distribution patterns that may have caused the divergences, this paper presents a nonlinear (elastic-plastic, material nonlinearly only)analysis of the same nozzle. The results are compared with the results from the previous linear analysis. The results are discussed with respect to an evaluation procedure for Shell/Plate element. Element investigations presented in a paper by Porter, et al,1999 (2).

In 1997 Ibrahim begovic [3], addressed the various approaches for finite rotations element formulation and the related complex issues involved while focusing on the stress resultant geometrically exact shell theory. In 1997, MacNeal [4] presented an historical review which describes the evolution of the linear shell elements. He divided his paper to several sections where each containing the evolution of a specific type of formulation e.g. Kirchhoff type elements, lower order Mindlin type elements and high order elements. It may be worth noting that he pointed out that the first membrane element was formulated in 1956 by Turner et al. [5] and the first plate bending element was formulated in 1961 . In 1990, Yang et al. [6] presented a broad review of 287 publications which have contributed mainly to the development of the linear thin shell finite element field. Their review included: flat element modeling for shells, axisymmetric shell elements, curved shell elements, degenerated shell elements and some extensions and application of shells element. As far as flat elements are concerned the following advantages were stressed: they are simple to formulate, it is easy to input data to describe the geometry, they are easy to mix with other types of elements and are capable of modeling rigid body motion without inducing strains. As for disadvantages: they exclude coupling of membrane and bending within the element (this is true only for the linear analysis case), they have difficulties to treat nodes where all elements meeting at a point are coplanar, 'discontinuity' of bending moments and the approximation of the geometry

In 1989 Wempner [7] presented a review paper on mechanics and finite element of shells. The paper address several fundamental issues regarding the formulation of shells finite element e.g. shear and membrane locking, reduced and selected integration, hybrid and mixed formulation, Kirchhoff's assumption, flat triangular and quadrilateral element and the patch test. The geometrical nonlinearities behavior of the shell structures is addresses too yet, the differences between the nonlinear shell element formulations are not highlight there. Erez etal[8] Presented a state of the art review on geometrically nonlinear analysis of shell structures that is limited to the co-rotational approach and to flat triangular shell finite elements. These shell elements are built up from flat triangular membranes and plates.

3- Aim of the paper

The aim of this present work is to illustrate the importance of element selection as to obtain the proper finite element result. As the following

- To investigate the factors influencing the selection of elements in FEA by considering the effects of different types of elements on the results of FEA
- Selection of proper element leads to proper model representation and construction
- Proper element selection leads to accurate FE results
-

4- Geometry of the pressure vessel

The cylindrical pressure vessel used in the analysis is presented in figure (1) below .

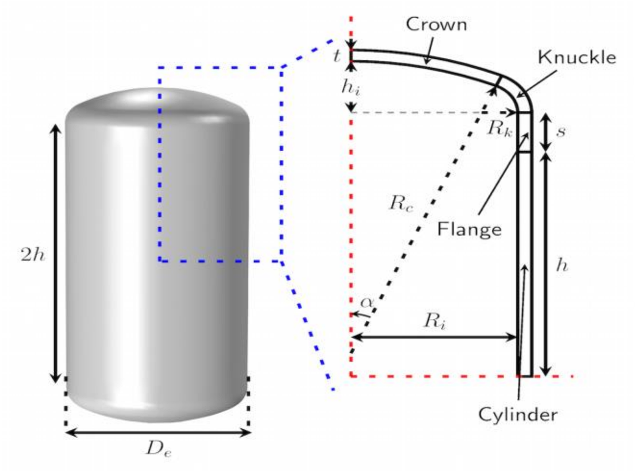


Figure (1) Geometry of pressure vessel

Mechanical properties of tank materials are given in table 1 below

Table No. (1) Mechanical properties of p.v. material

Material Type	Pressure Vessel Dimension	Material Thickness { mm }	Density {Kg/m^3 }	Yield stress [Mpa }	Ultimate stress { Mpa }	Modules of Elasticity { GPA }
Mild Steel	Length = 2000 mm Thickness = 5 mm Outside diameter =1500 mm Inside Diameter =1490 mm Pressure = 60 bar	5	7750	1800	2450	213

5- F. E. MODELING OF PRESSURE VESSEL

In most or all finite element packages uses the following sequences in modeling and analysis ,

They are :.

a- Preprocessor (build finite element model ,mesh , apply loads and constraints)
b- Finite element solver (assemble and solve system of equations)

c- Post Processing (sort and display results)

5.1 Numerical Mesh

Once the model of pressure vessel is created , it is necessary to select the proper element type suitable for the analysis in order to achieve correct results , the vessel divide into a finite number of regions called elements . The network of elements obtained is called a mesh. The computations are then performed, by solving the constitutive equations that describe the relationship between the forces and the displacements in the materials.
In this analysis three different elements are used in modeling as to display how important the adequate selection of elements are . elements used are :.
PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).This element is defined by 8 nodes or 6 nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element. This element has plasticity, hyperelasticity, creep, stress stiffening, large deflection,

and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. Geometry of element 183 is shown in figure no. 2 ,while Figure no.3 illustrates The mesh of the vessel with this element .

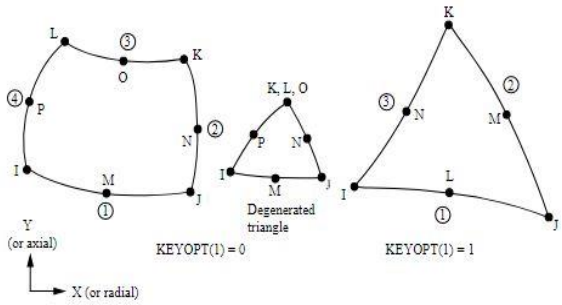


Figure 2 element 183 geometry

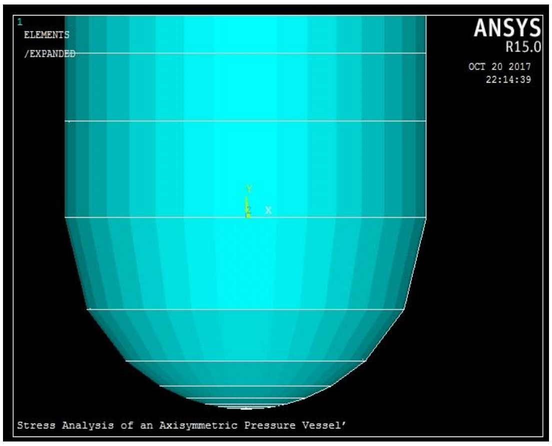


Figure 3 mesh of p.v with element 183

Element CPT213 is a higher-order 2-D eight-node coupled pore-pressure mechanical solid element. The element has quadratic displacement behavior and is well suited to modeling curved boundaries. The element is defined by eight nodes having three degrees of freedom at each corner node . The translations in the nodal x and y directions , one pore-pressure degree of freedom and two degrees of freedom at the midside nodes .
CPT213 can be used as a plane strain or axisymmetric element. The element has stress stiffening, large deflection, and large strain capabilities. CPT213 element geometry and element mesh are given in figures 4 and 5 respectively .

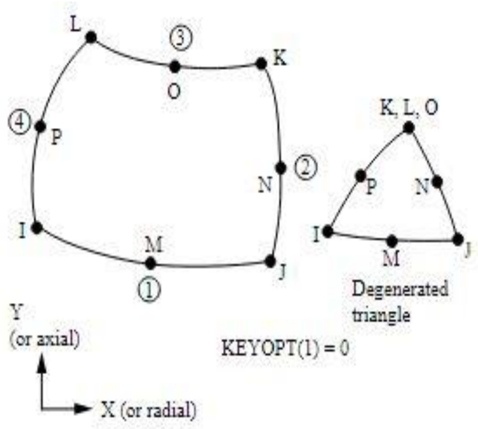


Figure 4 Element CPT213 geometry

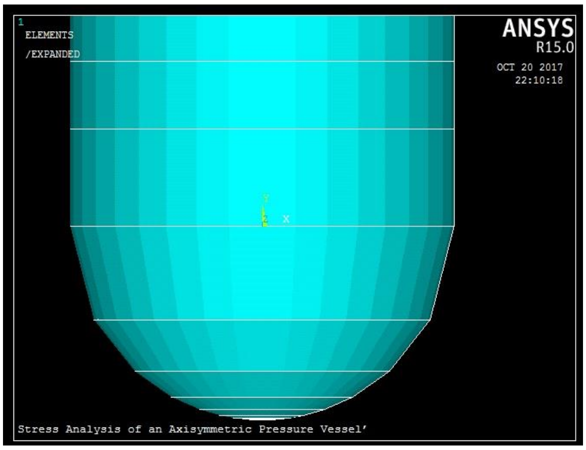
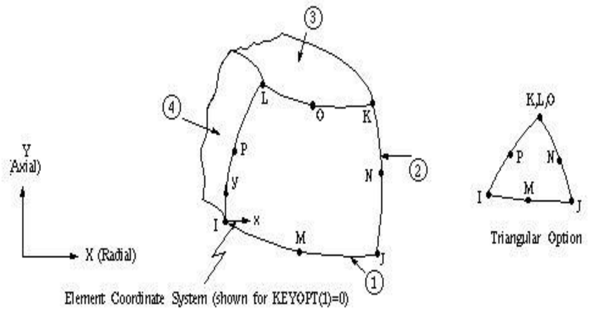


Figure 5 mesh of p.v. with element CPT213

PLANE25 is used for two-dimensional modeling of axisymmetric structures with nonaxisymmetric loading. Examples of such loading are bending, shear, or torsion. The element is defined by four nodes having three degrees of freedom per node: translations in the nodal x, y,
and z direction. For unrotated nodal coordinates, these directions correspond to the radial, axial, and tangential directions, respectively.



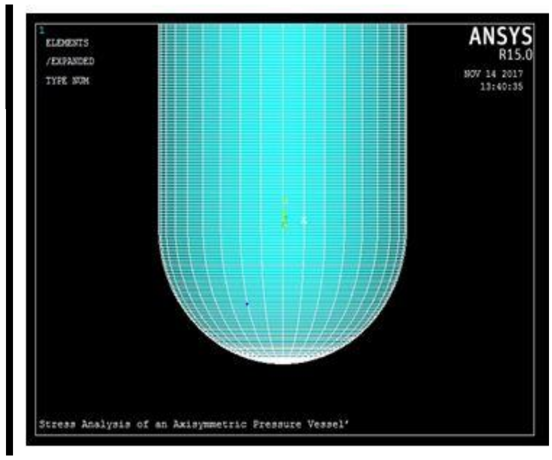
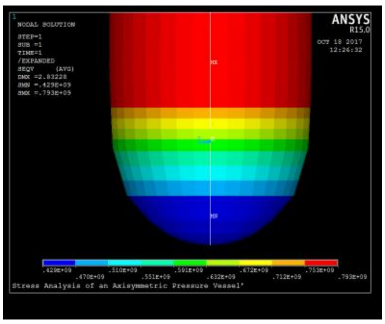
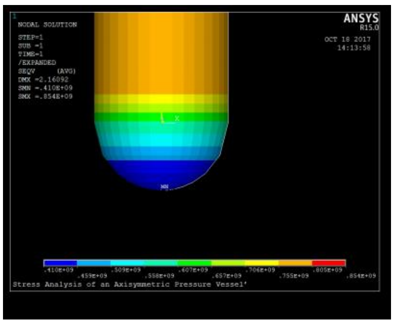


Figure 6 axi har noded25 Figure 7 mesh of p.v. with element axi har25

6- FEA Results

Once the geometry of the object to be analyzed is defined, the first task is to select the type of element that is to be employed. For most pressure vessel analyses, the element selection is made from three categories of elements: axi symmetric solid elements, shell/plate elements and 3-D brick elements. Although nearly all problems can be solved using 3-D brick elements, the other two types offer significant reductions in the solution time and effort where they are applicable. Often, this reduction in solution effort is significant enough to make the use of FE analysis feasible where it might not be with 3-D brick elements .
Figure 8 shows von misses stress for element 183 , the stress is 793Mpa ,while figure 9 shows von misses stress for element CPT213 which is 854Mpa and figure 10 presents Von Mises for axi harmonic which is 1090Mpa





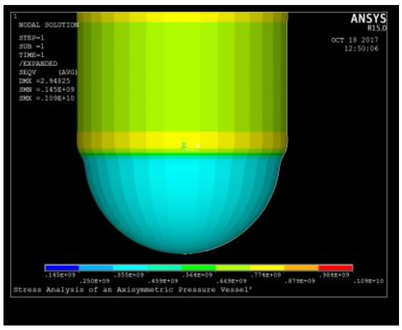


Figure 8 Von Misses stress for element 183

Figure 9 Von Mises stress for element cpt213

Figure 10 Von Misses stress for element axi harmonic

Comparison of numerically calculated vessel parameter with analytical values are given in table no2 .

Table No.2 Comparison of Numerical and analytical values

	NUMERICAL " ANSYS "			ANALYTICAL
Parameter	Element 183	Element CPT213	axi harmonic element	Calculated
Von Mises stress (Mpa)	793	854	1090	897
Displacement (mm)	3.07	2.33	2.94	4.21

Plot esults here

For stress in x and y directions

7- Discussion :

The choice of elements for FEA, therefore, depends largely on the geometry of the structure. Not all structures can be modeled using 1D element or 2D element. 1D element is used for long and slender symmetrical structure with uniform cross section. 2D element is used for plate or shell like structure while 3D element is used for structure with complex geometry which cannot be simplified for analysis. Since the form of FEA result is very much influenced by the
type of elements, the desired analysis result becomes one of the deciding factor for the selection of elements in the early stage of FE modeling. If a detail analysis in which the stress
distribution due to the applied load is required, then 2D element or 3D element are of better choices. 1D element, however, can still be used for rough and quick estimation of
overall factor of safety for the engineering structure. Lastly, the choice of element for FEA also depends on the analysis time and memory capacity of the computer available.
Since all three FE models produced reliable FEA results, the execution time and computer memory becomes the deciding factor. It is always better to carry out the analysis in the
shortest amount of time with the smallest computer memory required so that the engineering problem can be solved effectively and efficiently. In this case, modeling with 3D
elements is the best choice followed by 2D elements .
In addition to the above mentioned consideration the analyst should consider the following points too which are
Element Order. When using most FEA packages ('H' method), the analyst selects the element order, but with some packages, ie PTC Mechanica 'Structure' ('P' method) the selection of the element order may be left to the software. Where the user is specifying the element order, it is important that an appropriate combination of mesh density and element order be chosen, as otherwise results will be of low accuracy. This means that great care is needed in areas of rapidly changing stresses, eg notches etc

Element size

FEA analysis is carried out on an assembly of discrete elements, not on a continuous structure. The smaller the element size (the finer the mesh), the smaller the discritisation error, but computation time increases.

Connecting different types of element

Where two element edges are coincident, the function describing the displacements of the two edges must be the same. This can create difficulties when thin sections of components are connected to thick sections, e.g: cooling fins on a cylinder. The body of the cylinder may well be meshed with solid tetrahedra but for efficiency the fins will probably need to be meshed with thin plates. Appropriate linking will have to be done to ensure the model is a reasonably representing the component.

8 Conclusion

As can be seen from stress and displacement results obtained using above mentioned elements , the results might differ if different element types are used for modeling of prescribed pressure vessel . . The difference is due to the different characteristics inherent by different elements. Therefore, it is crucial to understand the characteristic of the elements in order to optimize modeling to achieve accurate and reliable FEA results. With the advantages and limitations of these elements in mind,

Few factors are to be considered when deciding on the types of elements to be used in FEA, which are the geometry of the structure, desired analysis results, analysis time frame as
well as the capability of the computer , and a lot of common sense .

References

[1] Drazan Kozak1, Ivan Samardzic2, Antun Stoic, Zeljko Ivandic,
Darko Damjanovic, Stress Analyses of Cylindrical Vessel with

Changeable Head Geometry, BULETIN ştiinţific, Seria C,
Volume XXIII, ISSN 1224-3264.

[2] Michael A. Porter, Dennis H. Martens, Comparison of Linear and
Nonlinear FE Analysis of a Typical Vessel Nozzle,199.

[3] A. Ibrahim begovic (1997). Stress resultant geometrically exact shell theory for finite rotations and its finite element implementation. Appl. Mech. Rev., **50**(4), 199–226.

[4] R.H. MacNeal (1997). Perspective on finite elements for shell analysis. AIAA, Paper 97-1139, 2104–2113.

[5] M.J. Turner, R.W. Clough, H.C. Martin and L.J. Topp (1956). Stiffness and deflection analysis of complex structures. J. Aeronaut. Sci., **23**, 805–824.

[6] H.T.Y. Yang, S. Saigal and D.G. Liaw (1990). Advances of thin shell finite elements and some applications-version I. Computers and Structures, **35**, 481–504.

[7] G. Wempner (1989). Mechanics and finite element of shells. Applied Mechanics Review, ASME, **42**, 129–142.

[8] T.J.R. Hughes and E. Hinton (eds.)(1986). Finite element methods for plate and shell structures. Element Technology, Vol. 1, Pineridge Press International, Swansea.