Finite Element Modeling For The Stress, Buckling And Modal Analysis Of A Cylindrical Pressure Vessel With Torispherical Enclosure

Jeevan. T. P[#1], Divya H. V[#2]

[#1] Lecturer, Department of Mechanical Engineering,
Malnad College of Engineering, Hassan, Karnataka, India

[#2] Lecturer, Department of Mechanical Engineering,
Malnad College of Engineering, Hassan, Karnataka, India

Abstract:

Pressure vessels are being widely employed worldwide as means to carry, store or receive fluids. The pressure differential is dangerous and many fatal accidents have occurred in the history of their development and operation. Effect of cylindrical pressure vessel with torispherical enclosure on the stress, buckling and vibrational characteristics subjected to an internal pressure is investigated by Finite Element Method (FEM). The two dimensional static stress analysis is performed for different vessel thickness to analyze the stresses and deflections in the vessel walls due to the internal pressure. Eigenvalue buckling analysis was performed for different vessel thickness and different knuckle radius to determine critical buckling pressure and modal analysis is performed to determine the vibration characteristics (natural frequencies and mode shapes) of a vessel. Finite Element Analysis was performed using software programme ANSYS.

Based on the results obtained from static stress analysis, it is found that thickness of the vessel plays an important role in withstanding the applied internal pressure. Influence of thickness and knuckle radius of the vessel on buckling pressure are studied. Comparing the rate of increase in buckling pressure in both cases, the variation of buckling pressure is more with the influence of thickness compared to influence of Knuckle radius. Thus the buckling pressure is more sensitive to the thickness of the vessel. The vibrational characteristics like natural frequency and mode shapes are presented. The results of stress analysis obtained by Finite Element Method are in good agreement with results obtained by classical approach.

Keywords: *Torispherical, Knuckle radius.*

1. Introduction:

The motivation for this study lies on the fact that Pressure vessels are being widely employed worldwide as means to carry, store, or receive fluids. The pressure differential is dangerous and many fatal accidents have occurred in the history of their development and operation. Therefore, their design, manufacture, and operation are regulated by engineering authorities backed up by laws. For these reasons, the definition of a pressure vessel varies from country to country, but involves parameters such as maximum safe operating pressure and temperature, which is very important. The analysis of pressure vessel is of critical importance in nuclear industries.

Pressure vessels often have a combination of high pressures together with high temperature, and in some cases flammable fluids or highly radioactive materials. Because of such hazards it is imperative that the design be such that no leakage can occur. In addition these vessels have to be designed carefully to cope with the operating temperature and pressure [10].

Torispherical heads are mainly used in pressure vessels to store fluids at high pressure in technical applications. Stress occurs at the junction between the cylinder and the dome (head) due to change of geometry. In certain advanced technical applications (rockets used in space craft launches), torispherical heads are employed in order to limit the length of the vehicle. Geometry of Torispherical head is shown in Figure 1.

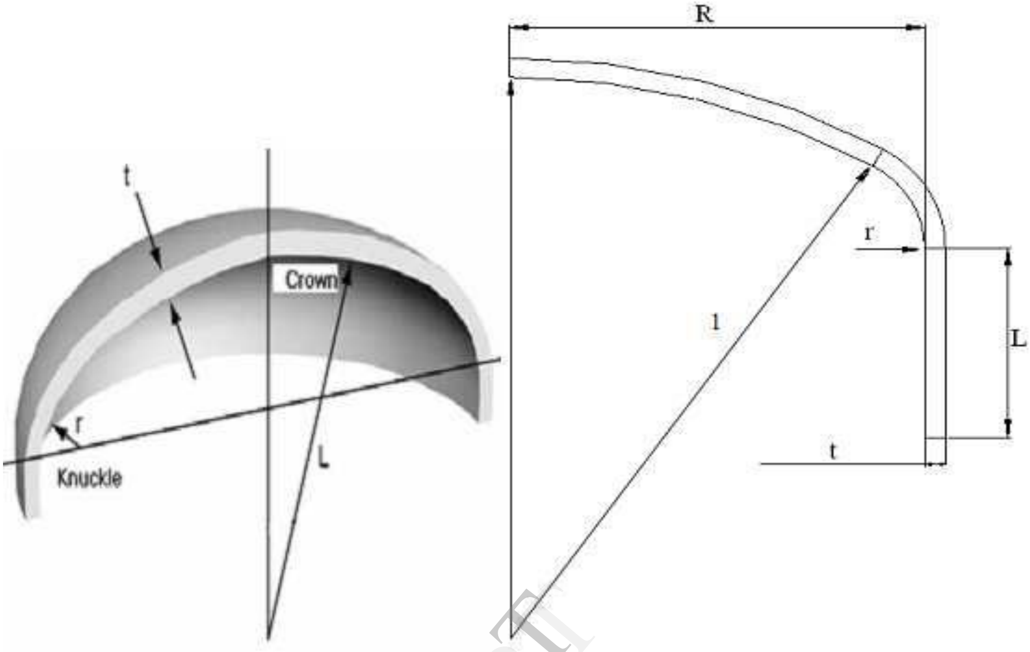

Fig. 1 Geometry of Torispherical head

2. Literature Review

Cylinders with torispherical ends have been the subject of numerous investigations over the last 80 years. The first theoretical work on sphere-cylinder intersections was initiated by Galletly [1] who utilized the theory of two intersecting thin shells. Galletly [1] cautioned that torispherical shells designed according to the ASME code could lead to failure during testing because the use of membrane shell theory would give poor results when applied to the toroidal region of the shell. Several numerical analysis study were made using the finite element method to investigate the design sensitivity of thin torispherical end pressure vessels, such as studies by John Meacall [2] in1963, A.Tafreshi [3] in 1977,Amran Ayob [4] in 2006, V.N. Skopinsky and A.B. Smetankin [5] in 2006, M.R.Khoshravan and A.Rahmani [6] in 2009. Theories of thin-walled structures applied on pressure vessels were reviewed by Teng et al [7]. Results of numerical evaluation of buckling by using linear and non-linear theories of thin-walled shells of revolution have been presented. Teng et al [8] have introduced a numerical model, aided by the method of Eigenmode-affine, in the non-linear analysis of elastic shells. As the shells are sensitive to initial geometrical imperfections, predicting of their buckling resistance would be precise if those imperfections are taken into account. In torispherical heads by increasing the ratio of knuckle radius per vessel diameter (r/D), dimension of spherical part decreases. Thus, the spherical part as part of the head becomes weaker and in a defined r/D a notable fall in buckling resistance is occurred [9].

3. Finite Element Model Development

Finite element method with large deflection analysis was performed using commercial Ansys software. A two dimensional finite element model for Static Analysis was generated using Ansys. Since the vessel is axially symmetric about its central axis, as shown in Figure 3.1(a). An axisymmetric analysis will be performed using two-dimensional, 8-node quadrilateral element Plane 82 with the axisymmetric option activated. In addition, the vessel is symmetric about a plane through the center of the cylinder. Thus, only a quarter section of the vessel is modeled as shown in the Figure 3.1(c) obtained from half symmetrical cross section as shown in Figure 3.1(b). The Finite Element Model under boundary conditions are shown in Figure 3.2

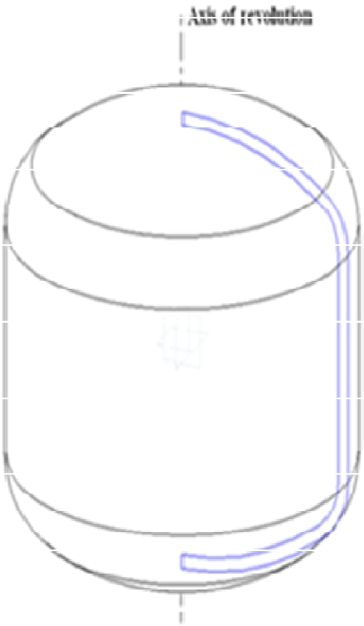

Fig. 3.1(a) SOR is obtained by cross rotating a cross-section about an axis of revolution

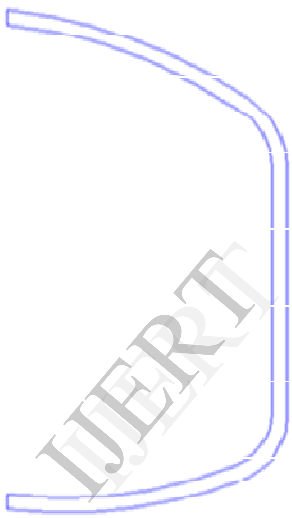

Fig. 3.1(b) Half symmetrical cross section

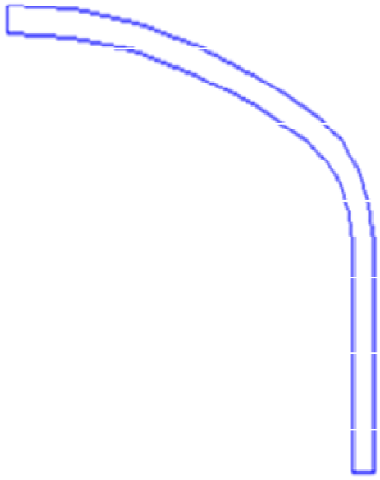

Fig. 3.1(c) Quarter symmetrical section

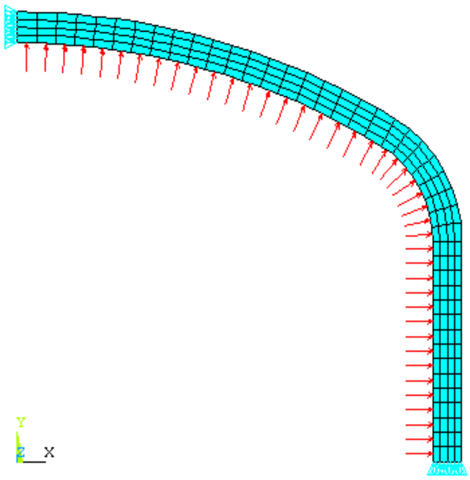

Fig. 3.2 Finite Element Model with Boundary Conditions for Static Analysis

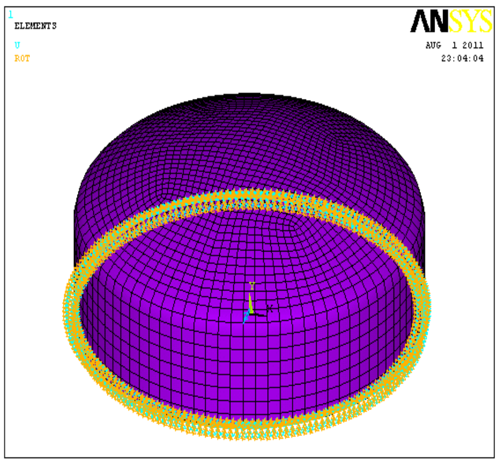

Fig. 3.3 Three Dimensional FE Model with Boundary conditions for buckling Analysis

A Three dimensional finite element model was generated using Ansys. For studying of buckling of pressure vessel with torispherical head, we modeled intersection of cylinder head. The influence of welding and forming on material property were neglected while the effect of welding can be accounted for by modifying the yield stresses. The model was meshed with shell 93 element. SHELL93 is particularly well suited to model curved shells. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y and z- axes. The element has plasticity, stress stiffening, large deflection, and large strain capabilities. The material of the intersections was assumed to have typical properties of steel: an elastic modulus of $2*10^5$ *MPa*; a Poisson's ratio of 0.26, and yield stress of 250 *MPa*, and exhibit an elastic-perfectly plastic behavior. The Finite Element Model under boundary conditions is shown in Figure 3.3

4. Results And Discussion

4.1 Parametric study for Static Analysis: The static stress analysis of 2D cylindrical pressure vessel with torispherical enclosure subjected to an internal pressure of 1Mpa is performed. The parametric study is carried out for different thickness of vessel and the obtained results are presented in this section. Deformation of the vessel,Von Mises Stress plot,axisymmetric quarter expansion of vessel is shown in Figure 4.1 and 4.2

Case 1: Vessel with thickness 10mm

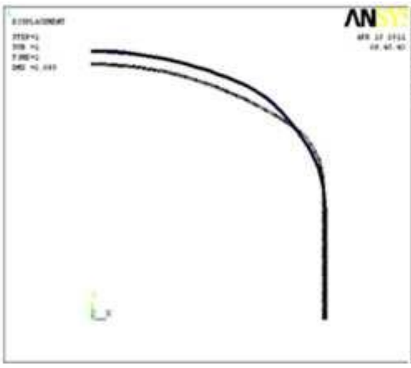

Fig. 4.1(a) Deformed and undeformed shape of vessel

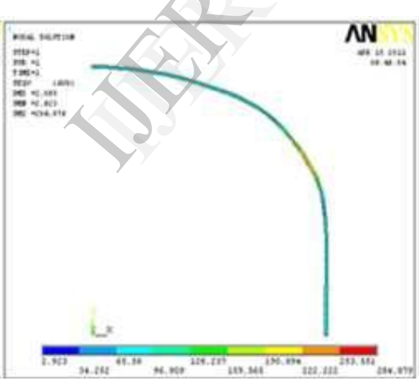

Fig. 4.1(b) Von mises stress

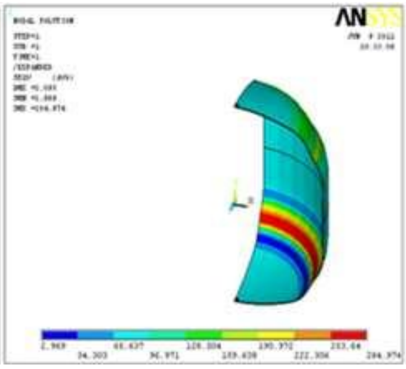

Fig. 4.1(c) Axisymmetric expansion of vessel

Case 2: Vessel with thickness 20mm

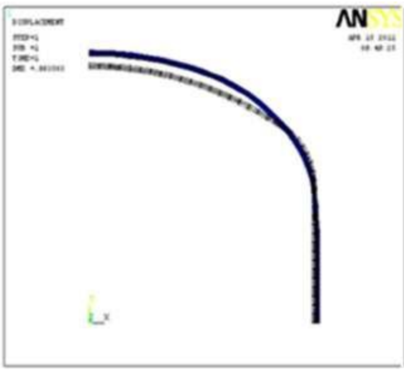

Fig. 4.2(a) Deformed and undeformed shape of vessel with thickness 20mm

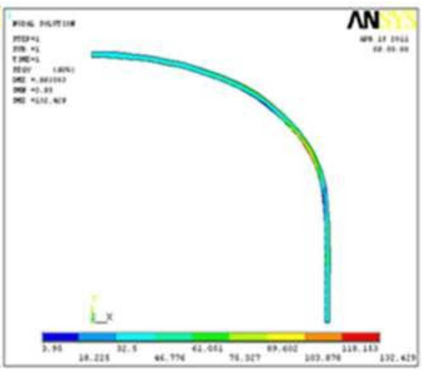

Fig. 4.2(b) Von mises stress

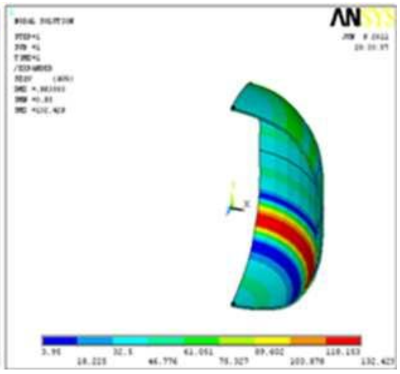

Fig. 4.2(c) Axisymmetric expansion of vessel

Maximum Stress on the vessel is seen in the knuckle region and hence it is considered as the critical region of the vessel. The von mises stress in vessel with thickness 10 mm which is about 284.875 N/mm^2 is greater than the yield strength i.e. 250 N/mm^2 . Hence the vessel with thickness 10mm cannot withstand the applied internal pressure of 1Mpa. The von mises stress in vessel with thickness 20 mm which is about 132.429 N/mm^2 is well within the yield strength. Hence the vessel with thickness 20mm can withstand the applied internal pressure of 1Mpa and thus it is recommended.

It can be observed that the deformation of vessel does not take place uniformly even though the uniform pressure is applied. It is clearly seen that the toroidal region of the vessel deforms in opposite direction to the cylindrical and spherical region. Thus it offers more resistance compared to other types of heads where the expansion takes place uniformly in single direction.

4.2. Variation of stress along the thickness of the vessel

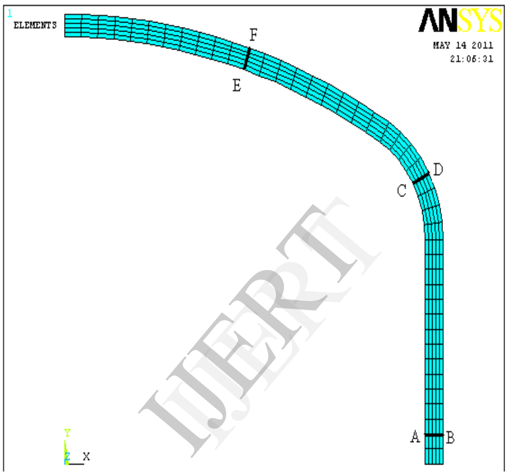

Fig. 4.3 FE Model with three different paths to show variation of stress

Three different paths on the finite element model are selected to determine the variation of stress along the thickness of the vessel. Path AB is selected in the cylindrical segment; this path is selected far away from the toroidal segment, to prevent the geometrical imperfections caused in toroidal region. Path CD and EF are selected in toroidal and spherical segments respectively.

The variation of stress along the thickness of the vessel for different vessel thickness in X- direction (radial stress), Y-direction (meridional stress) and Z-direction (hoop stress) for cylindrical segment, Toroidal Segment, Spherical Segment is shown in the Figure 4.4,4.5 and 4.6 respectively.

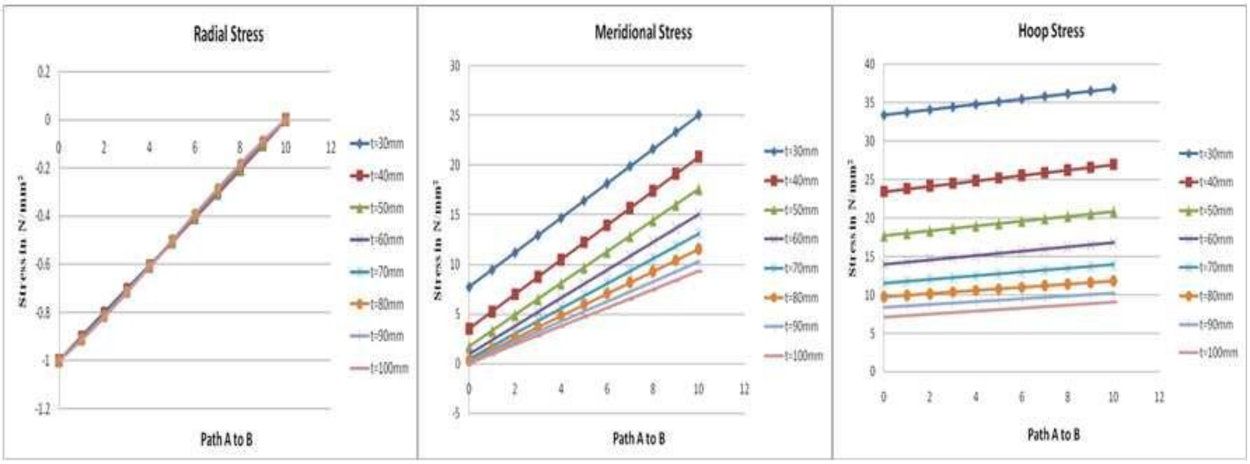

Fig. 4.4 Variation of Stress along the Thickness of the Vessel in Cylindrical Segment

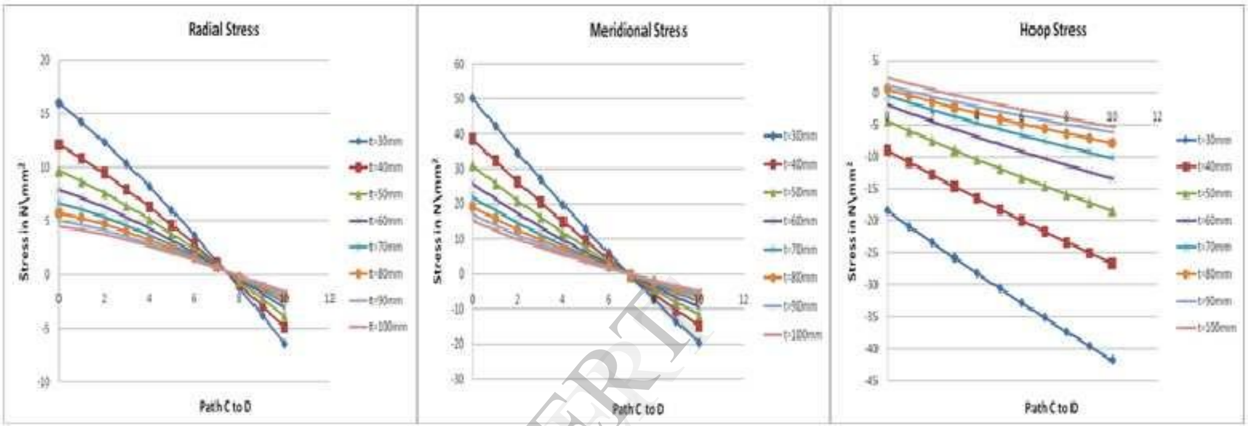

Fig. 4.5 Variation of Stress along the Thickness of the Vessel in Toroidal Segment

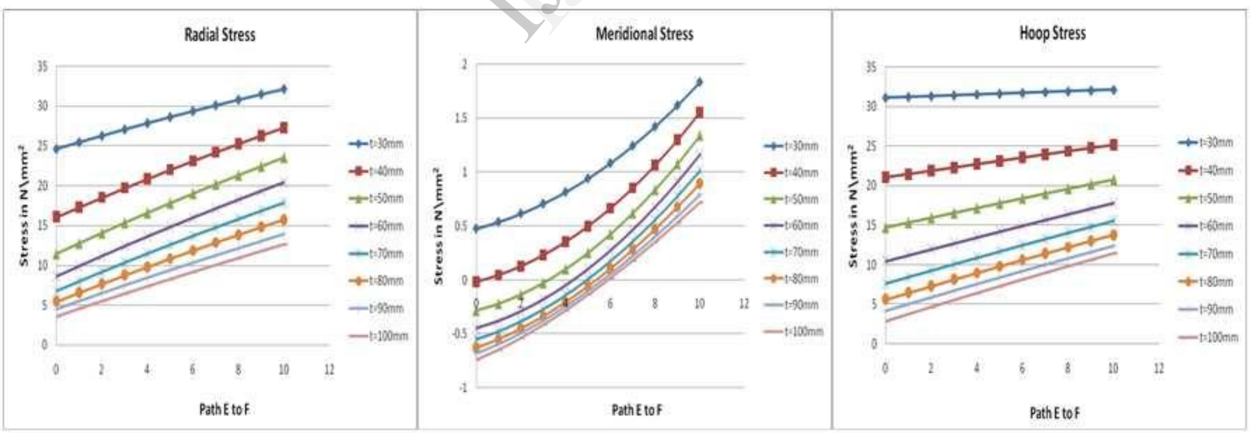

Fig. 4.6 Variation of Stress along the Thickness of the Vessel in Spherical Segment

A comparison of the results of stress analysis by classical approach and FEA approach using ANSYS software for a cylindrical pressure vessel with torispherical end closure for a different thickness of the vessel is as shown in the Figure 4.7 and 4.8.

The results of stress analysis obtained by classical approach are in good agreement with results obtained by Finite Element Method. Here the Radial stress effects are not taken into consideration because they are negligible in thin vessels.

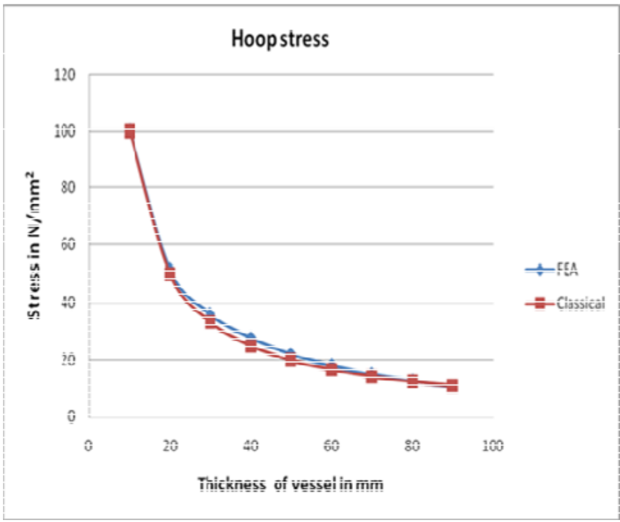

Fig. 4.7 Comparision of results obtained by FEA and classical approach for hoop stress

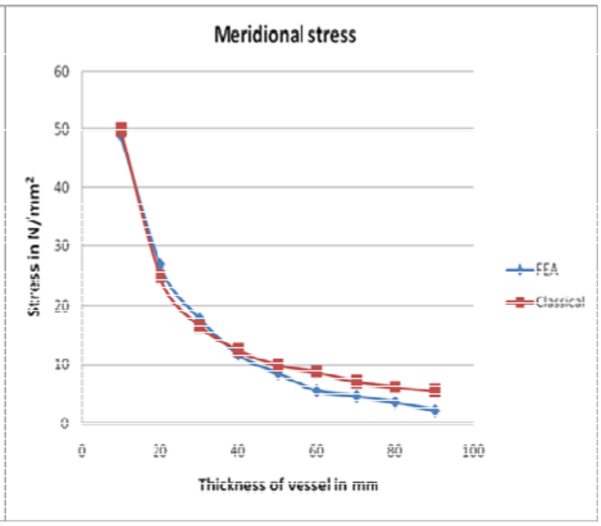

Fig. 4.8 Comparision of results obtained by FEA and classical approach for meridional stress

4.3 Parametric study for Buckling Analysis:

The Eigenvalue buckling analysis of 3D cylindrical pressure vessel with torispherical enclosure subjected to an internal pressure of 1Mpa is performed. The parametric study is performed to determine the critical buckling pressure of the vessel. The effect of geometrical parameters such as thickness and knuckle radius on the critical buckling pressure is presented.

4.3.1 Influence of Thickness on Buckling Pressure

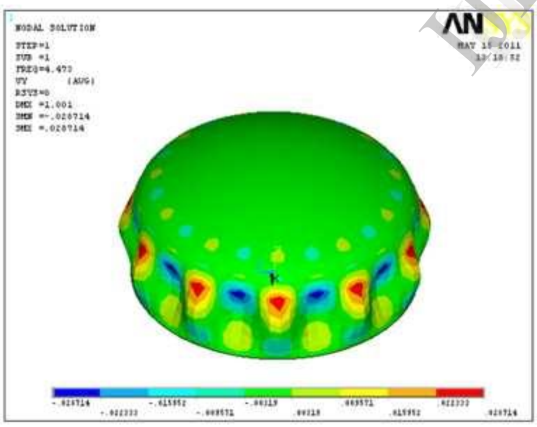

Fig.4.9 (a) Buckled shape of the vessel with the thickness 10mm

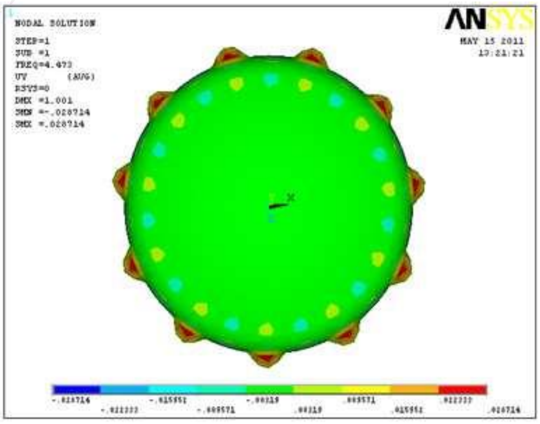

Fig.4.9 (b) Top view of buckled shape of thickness 10mm

Buckling shape of the vessel having thickness 10 mm subjected to an internal pressure of 1 Mpa is shown in Figure 4.9 (a), 4.9 (b). By the method of eigenvalue buckling analysis, the critical buckling pressure at which the vessel undergoes buckling is found to be 4.473 Mpa. The vessel buckling is seen in knuckle region because of geometric discontinuity. The results of parametric study are shown in table 4.1

Table 4.1 Results of Influence of Thickness on Buckling Pressure

SL.No.	Thickness of the vessel, in mm	Critical Buckling pressure, in Mpa
1	10	4.473
2	8	2.516
3	6	1.212
4	4	0.4377
5	2	0.07789
6	1	0.014

4.3.2 Influence of Knuckle radius on Buckling Pressure

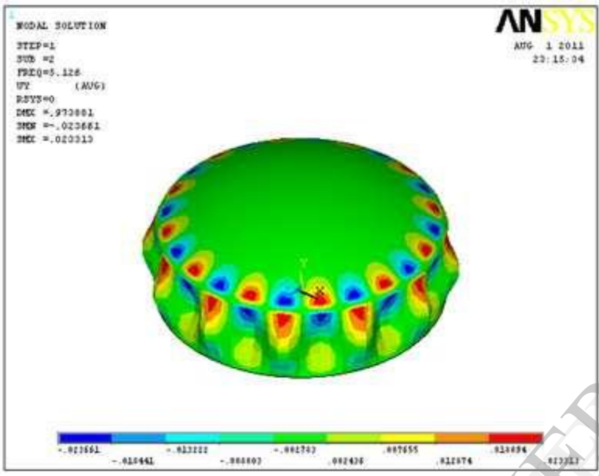

Fig.4.10 (a) Buckled shape of the vessel with Knuckle radius 25 mm

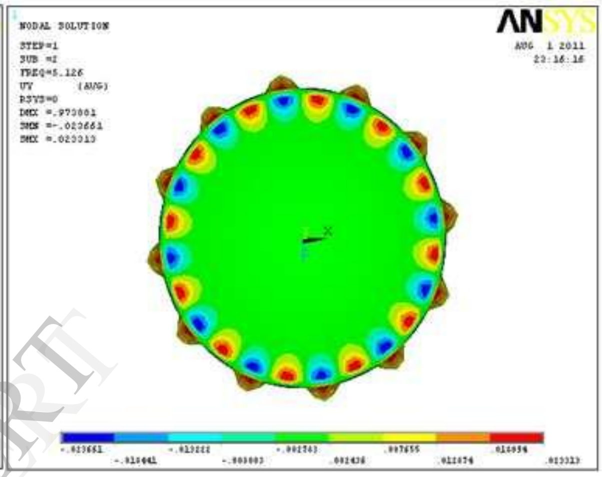

Fig.4.10 (b) Top view of buckled shape of vessel with Knuckle radius 25mm

Buckling shape of the vessel having Knuckle radius 25mm subjected to an internal pressure of 1 Mpa is shown in Figure 4.10 (a), 4.10 (b). By the method of eigenvalue buckling analysis, the critical buckling pressure at which the vessel undergoes buckling is found to be 5.126 Mpa. The vessel buckling is seen in knuckle region because of geometric discontinuity. The results of parametric study are shown in table 4.2.

Table 4.2 Results of Influence of Knuckle radius on Buckling Pressure

SL.No.	Knuckle radius of the vessel, in mm	Critical Buckling pressure, in Mpa
1	25	5.126
2	75	4.855
3	125	4.67
4	175	4.559
5	225	4.495

Figure 4.11 shows the influence of thickness on buckling pressure. It is observed that the increase of thickness leads to increase of Buckling Pressure. Figure 4.12 shows the influence of knuckle radius on buckling pressure. It is observed that the increase of

knuckle radius leads to decrease of Buckling Pressure. Comparing the rate of increase in buckling pressure in both cases (i.e. for different thickness and different knuckle radius of the vessel), the variation of pressure is more with the influence of thickness compared to influence of knuckle radius. The slope of the curves in Figure 4.11 is large compared to those of curves in Figure 4.12, thus the buckling pressure is more sensitive to the thickness of the vessel.

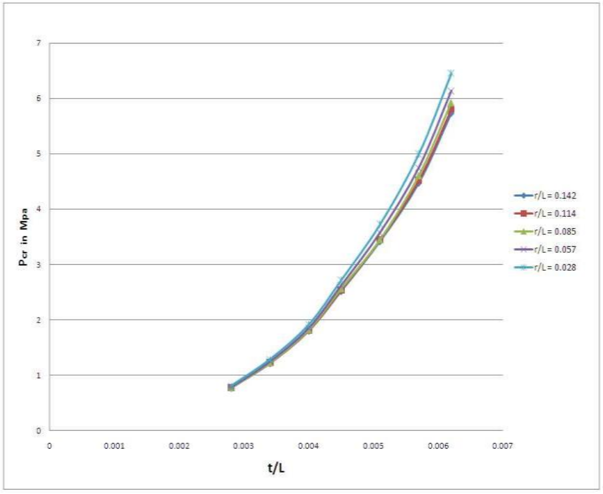

Fig. 4.11 Influence of thickness on buckling pressure

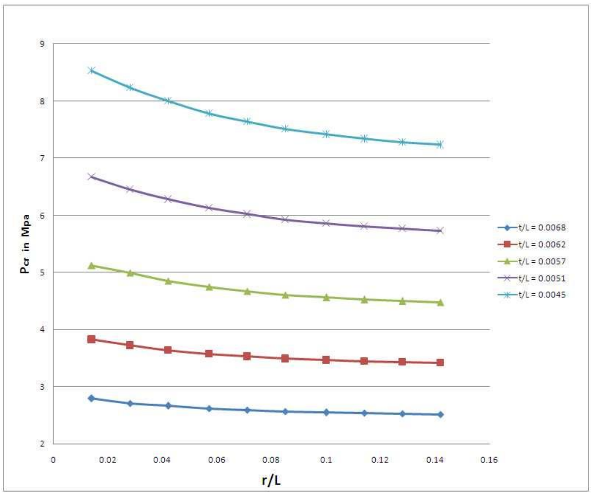

Fig 4.12 Influence of knuckle radius on buckling pressure

5. Conclusion

Finite Element Analysis was performed on a cylindrical pressure vessel with torispherical enclosure subjected to internal pressure. Based on the results obtained from static stress analysis, thickness of the vessel plays an important role in withstanding the applied internal pressure. It is also observed that high stresses occurred where there is geometric discontinuity, i.e. at the cylinder and head intersection. It is observed that the deformation of vessel does not take place uniformly even though the uniform pressure is applied. It is observed that the toroidal region of the vessel deforms in opposite direction to the cylindrical and spherical region thus it offers more resistance. The results of stress analysis obtained by Finite Element Method are in good agreement with results obtained by classical approach. Based on the parametric study performed to determine the critical buckling pressure of the vessel, buckling pressure is influenced by the thickness of the vessel. Higher the thicknesses of the vessel better the buckling resistance. Buckling pressure is also influenced by the knuckle radius, lesser the knuckle radius better the buckling resistance. Comparing the rate of increase in buckling pressure in both cases, the variation of buckling pressure is more with the influence of thickness compared to influence of Knuckle radius. Thus the buckling pressure is more sensitive to the thickness.

References

[1] G. D. Galletly and D. Bushnell, "Stress and buckling of internally pressurized, elastic-plastic torispherical vessel heads- comparisons of test and theory", Journal of pressure vessel technology, 1977, pp. 39-53

[2] John Meacall, "Stability of thin torispherical shells under uniform internal pressure", Basic research in physical sciences, Army materials research agency, 1963.

[3] A. Tafreshi, "Numerical analysis of thin torispherical end closures" Journal of pressure vessel and piping, 1997, pp. 77-80.

[4] Amran Ayob, "Stress Analysis of Torispherical Shell with Radial Nozzle", Faculty of Mechanical Engineering, University of Teknologi Malaysia, Skudai, Johor. Journal of the institution of engineers, Malaysia, Volume 67, 2006, pp.59-64

[5] V.N. Skopinsky and A.B. Smetankin, "Modeling and stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under external loading", International journal of applied mechanics and engineering, 2006, Volume 11, No.4, pp. 965-979.

[6] M.R.Khoshravan and A.Rahmani, "Numerical analysis of the effect of torispherical head on the buckling of pressure vessels", 7th International conference on modern practice in stress and vibration analysis, Journal of physics 2009.

[7] Teng, J.G. and Hong, T., 1998, "Non-linear thin shell theories for numerical buckling predictions", Journal of Thin-Walled Structures, 31, pp. 89-115.

[8] Teng, J.G., and Song, C.Y., 2001, "Numerical models for nonlinear analysis of elastic shells with eigenmode-affine imperfections", International journal of Solids and Structures, 38, pp. 3263-3280

[9] Blachut, J., 1998, "Buckling of sharp knuckle torispheres under external pressure", Thin walled Structure, 30, No. 1-4, pp 55-77.

[10] Somnath Chattopadhyay, "Pressure Vessel Design and Practice" CRC Press 2005.