ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Design Investigation into the Stress at the Base of a Nozzle of a Pressure Vessel

Devaraju U.R[1], Manjunath K[2], Mahesh T.S[3]
[1]M.Tech Student, Dept. of Mechanical Engineering, Govt. Engineering College, Hassan
[2]Asst.Professor, Dept. of Mechanical Engineering, Govt. Engineering College, Hassan
[3]Asso.Professor, Dept. of Mechanical Engineering, Govt. Engineering College, Hassan
devarajuhassan@gmail.com

Abstract

A pressure vessel typically consists of large cylindrical and / or spherical containers with nozzles through which the reactants flow in and out. . While plain cylindrical or spherical containers can be analyzed for internal pressure using thin/ thick cylinder formulae, the ones with nozzles are difficult to analyze. This is in view of complicated stress concentrations that arise at the interface of the nozzle and pressure vessel junction. The calculations have become complicated because of forces that arise at the free end of the nozzle. The forces include those of piping, wind forces, earth quake forces in addition to the internal pressure. In spite of these, strict adherence to safety codes is to be followed. ASME, Section VIII specifies the stress limits to be adhered to. One of the criterions is the stress intensity, which is not possible to compute by simple analytical procedures. FEM can be used for computing the deformation and stress at the nozzle-vessel junction in the structure and also at all other points on the pressure vessel. Quite often the geometric models are imported from CAD files for mesh generation with tetrahedral elements. Engineers generally use shell elements or tetrahedral elements while modeling the reactor vessels. But, precise estimation of stress intensity is not possible with these elements for a structure with nozzles. A method is developed for a precise structured modeling and for estimating the stress intensities at the junction of nozzles and pressure vessels. The structure of a reactor vessel of diameter 1900mm and length of 3600mm with a nozzle will be modeled and analyzed. The pressure load is 7 MPa. Stresses have been estimated. The induced stresses are compared with allowable stress. Based on the induced stress in the Pressure vessel three additional design variants have been studied to bring the stresses within allowable limits.

Keywords: Pressure vessel, finite element analysis, stress intensity, nozzle.

Introduction

A pressure vessel is defined as a container with a pressure differential between inside and outside. The pressure vessels are used to store fluids under pressure. The inside pressure is usually higher than the outside, except for some isolated situations. The fluid inside the vessel may undergo a change in state as in the case of steam boilers, or may combine with other reagents as in the case of a chemical reactor. Pressure vessels often have a combination of high pressures together with high temperatures, 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. Pressure vessels are used in a number of industries for example, the power generation industry for fossil and nuclear power, the petrochemical industry for storing and processing crude petroleum oil in tank farms as well as storing gasoline in service stations, and the chemical industry (in chemical reactors) to name but a few. Their use has expanded throughout the world. Pressure vessels and tanks are, in fact, essential to the chemical, petroleum, petrochemical and nuclear industries.

The high pressure rise is developed in the pressure vessel and pressure vessel has to withstand severe forces. So the selection of pressure vessel is most critical. That's why we can say that pressure vessel is the heart for storage of fluid. Pressure vessel must pass series of Hydrostatic tests. These tests examine the ability of the structure to withstand various pressures to see if protective zone around the operator station remains intact in an overturn. The structure is to be designed, fabricated, fitted and checked as per ASME standard [1].

The prediction of stress intensity precisely in the shell nozzle junction under various loading conditions. An efficient finite element modeling for shell nozzle junction has been presented in which shell elements are employed to idealize the whole region. These results are used for the analysis of leak before break concept [2]. The design of pressure vessel by using PVElite gives accurate analysis result and also reduces time [3]. An integrated shape optimization study of pressure vessels is conducted considering a model of the entire pressure vessel [4]. The finite element analysis (FEA) a practical tool in the study of

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

pressure vessels, especially in determining stresses in local areas such as penetrations [5]. Pressure vessels are leak tight containers. They invariably have a flanged bolted connection at least on one end. Bolts apparently take tensile loads while the cylinder is subjected to internal pressure. However the flange has comparatively higher thickness than that of shell and has geometric discontinuity taking place at the flange-shell junction. Due to this the flange will be subjected to a shear force and a bending moment which are in turn resisted by the shell part [6]. FEA is a very powerful tool used to determine burst strength of pressure vessel. Axis-symmetric FEA is carried out to accurately predict the burst strength of a thin cylindrical pressure vessel [7]. The Finite element analysis of a composite hydrogen storage vessel based on unit load method along with complete structural analysis and evaluation of fatigue lifetime were conducted using ABAQUS package [9]. The Objective of the Inclined Pressure Vessel is to have large scale production of Nitrous Oxide. The rate of the reaction and its temperature is controlled by the inclination of the vessel [10].

The existing pressure vessel has a wall thickness of 25mm and the internal pressure of 7MPa. The nozzle forces exerted in pressure vessel are 10060N, 18010N, and 3020N in FX, FY, and FZ direction respectively. And the induced moments of forces are 120110000Nm, 16100000Nm, and 60000Nm in MX, MY, and MZ direction respectively. These forces induce cracks in pressure vessel, in this context it is necessary to compute the required wall thickness of pressure vessel to withstand the forces induced in pressure vessel. So that the induced stresses are within allowable limits.

Material Property

The material of the main vessel and nozzle is Q235-A (low carbon steel). This material has an elastic tensile modulus of 2.01x105 MPa which is used throughout the analysis. Average values of the yield strength σ_y and ultimate strength σ_u for material Q235-A are 339.4MPa and 472 MPa, respectively. Poisson's ratio is taken as 0.3 for this material [8].

Methodology

Finite element static, non-linear analysis of the model vessel has been performed using ANSYS software. The figure 1, 5, 9 and 13 shows the 3D model of the pressure vessel for cases 1, 2, 3 and 4 respectively. The figures 2, 6, 10, and 14, show the meshed models of the pressure vessel for cases 1, 2, 3, and 4, respectively. The size of the elements in the vicinity of the shell or nozzle was made sufficiently small to ensure that accurate results are obtained in this area. The boundary condition used in the analysis is that, the top the cylindrical shell are fixed by applying the displacement on areas DOFs to be constrained in angular rotation is constrained and radial direction UX is constrained in top section of the cylinder. And the bottom portion is fixed by applying the displacement on areas DOFs to be constrained in UY direction. At the bottom nozzle appropriate boundary conditions are given to prevent rigid body motion of the vessel as shown in figures 3, 7, 11, and 15, for cases 1, 2, 3, and 4, respectively. The material is assumed to be isotropic.

Pressure load is applied throughout the inner surface of the model, nozzle and end plates. The internal pressure of 7 MPa is applied throughout the inner surface of the pressure vessel. The equivalent pressure is applied at the top spherical sectional area of the cylindrical vessel and nozzle surface area respectively. The concentrated induced nozzle loads are applied at the end of the nozzle so as to develop the equivalent moment. The figure 4, 8, 12, and 16, shows the load distribution in the pressure vessel for cases 1, 2, 3, and 4, respectively. The nozzle loads are given in below table 1.1.

Table 1.1 Load distributions in pressure vessel and nozzle

Case	Pressure (MPa)	Equivalent pressure Peq1 (MPa)	Equivalent pressure Peq2 (MPa)	FX (N)	FY (N)	FZ (N)	MX (Nm)	MY (Nm)	MZ (Nm)
1	7	131.2	4.9	10060	18010	3020	120110000	16100000	60000
2	7	64.79	4.9	10060	18010	3020	120110000	16100000	60000
3	7	42.64	4.9	10060	18010	3020	120110000	16100000	60000
4	7	31.58	4.9	10060	18010	3020	120110000	16100000	60000

A. Case 1 – 25mm Thickness

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

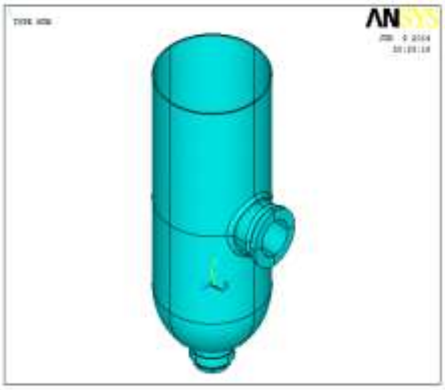

Fig 1 pressure vessel with spherical dish and nozzle

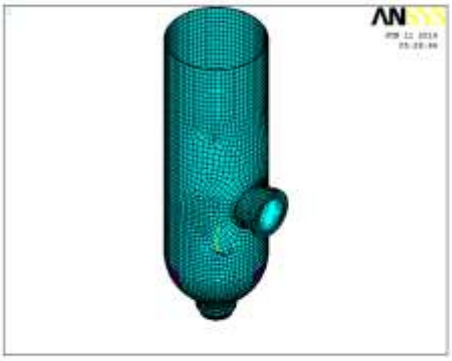

Fig.2 Meshed Model of Pressure Vessel

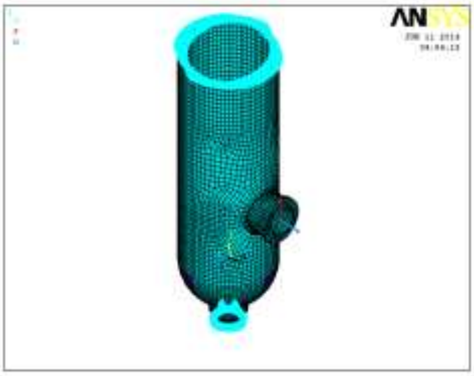

Fig. 3 Boundary Conditions of Pressure Vessel.

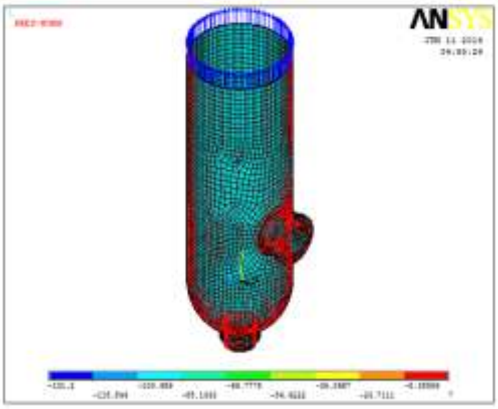

Fig.4 Loading in Pressure Vessel

B. Case 2 – 50mm Thickness

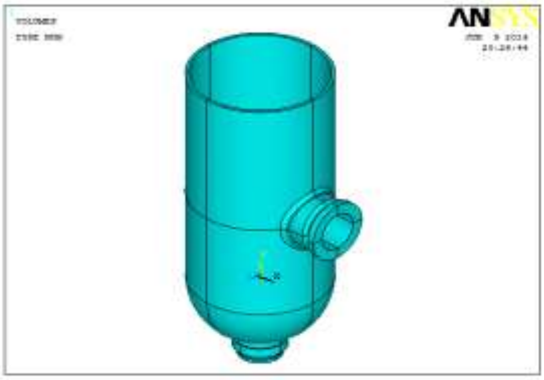

Fig 5 pressure vessel with spherical dish and nozzle

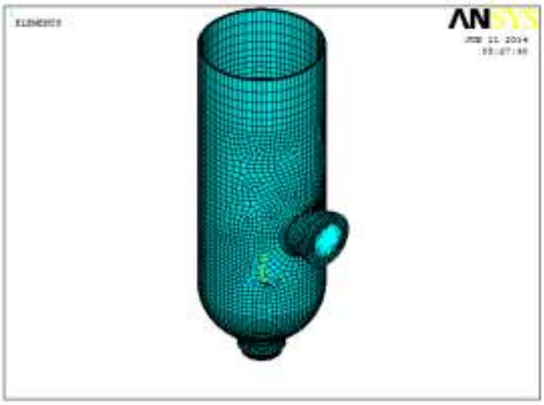

Fig.6 Meshed Model of Pressure Vessel

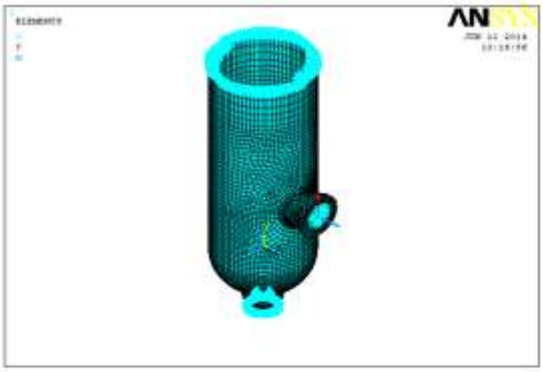

Fig. 7 Boundary Conditions of Pressure Vessel.

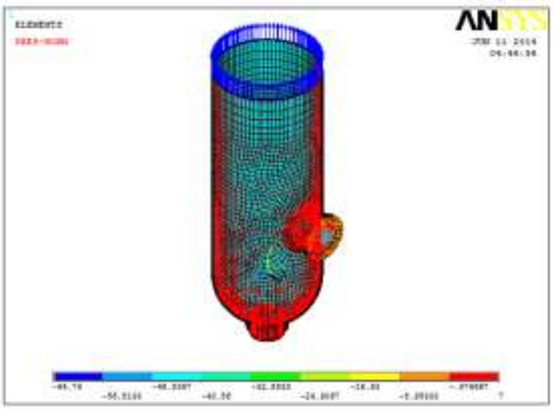

Fig.8 Loading in Pressure Vessel

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

C. Case 3 – 75mm Thickness

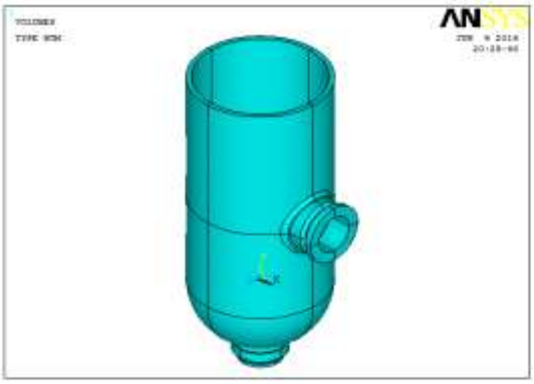

Fig 9 pressure vessel with spherical dish and nozzle

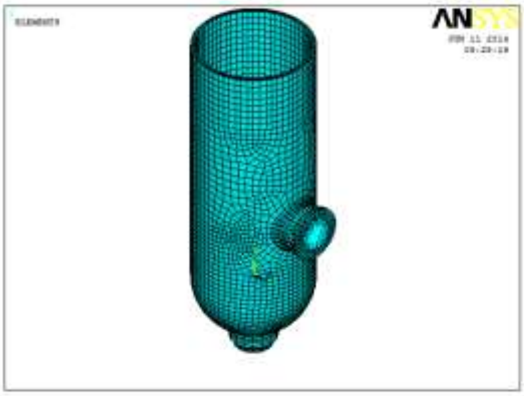

Fig.10 Meshed Model of Pressure Vessel

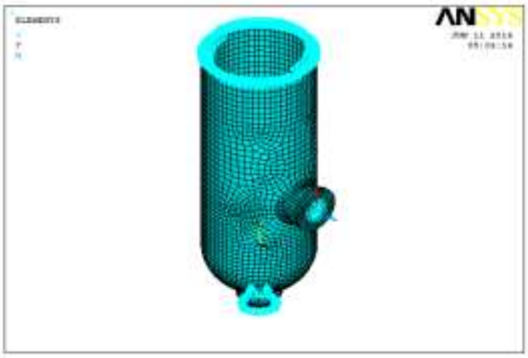

Fig. 11 Boundary Conditions of Pressure Vessel.

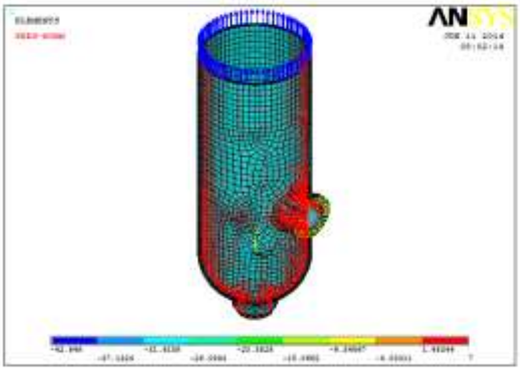

Fig.12 Loading in Pressure Vessel

D. Case 4 – 100mm Thickness

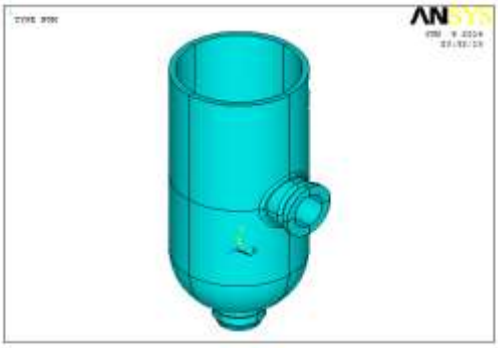

Fig 13 pressure vessel with spherical dish and nozzle

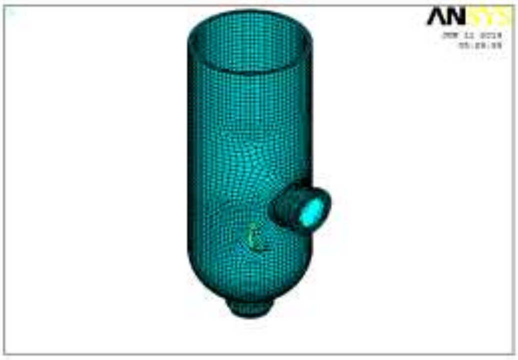

Fig.14 Meshed Model of Pressure Vessel

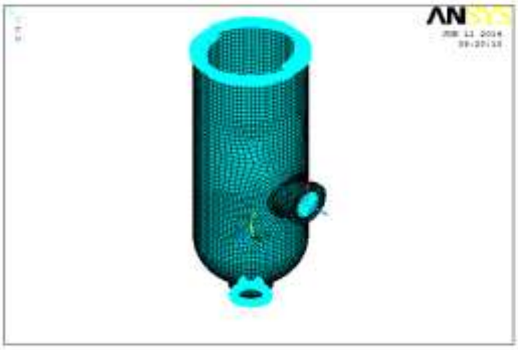

Fig. 15 Boundary Conditions of Pressure Vessel

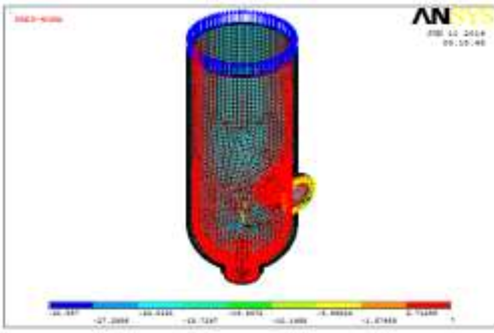

Fig.16 Loading in Pressure Vessel

Results and Discussion

The shell nozzle junction model is analyzed with load cases as given in table 1.1. The stress intensity obtained from ANSYS V 14 are compared with theoretical stress values and are tabulated in table 1.2, in this load cases pressure is

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

applied throughout the inner surface of the model and the moment is applied at the end of the nozzle. From this graphs, the maximum stress occurs at the junction of cylindrical shell and the nozzle of both ends of half symmetry shell nozzle junction. The stress value is gradually decreasing through the circumference and increasing again.

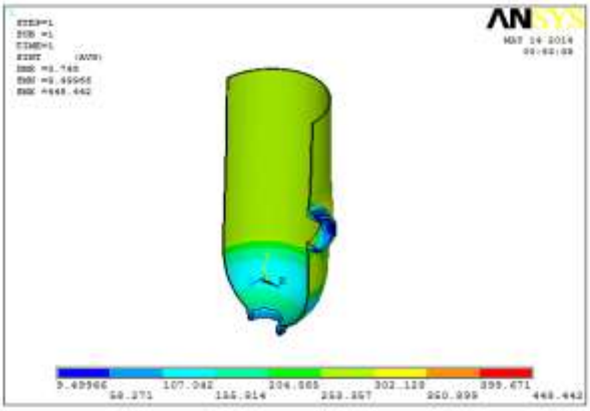

Fig. 17 Stress Intensity distribution in pressure vessel for case 1

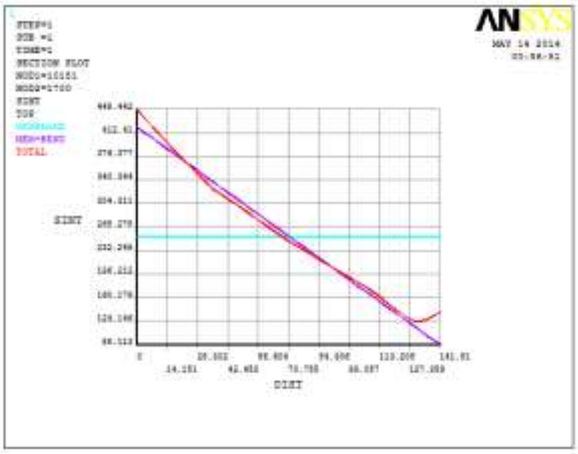

Fig. 18 Linearized Stress Intensity Plot for case 1

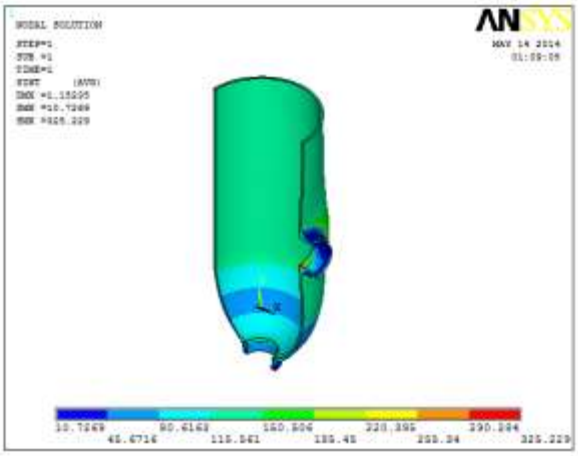

Fig. 19 Stress Intensity distribution in pressure vessel for case 2

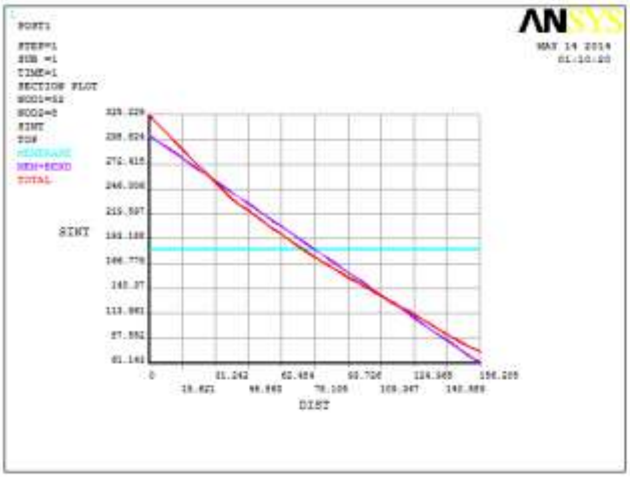

Fig. 20 Linearized Stress Intensity Plot for case 2

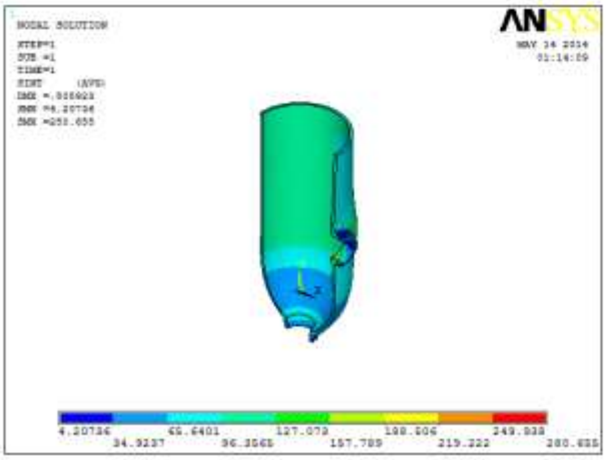

Fig. 21 Stress Intensity distribution in pressure vessel for case 3

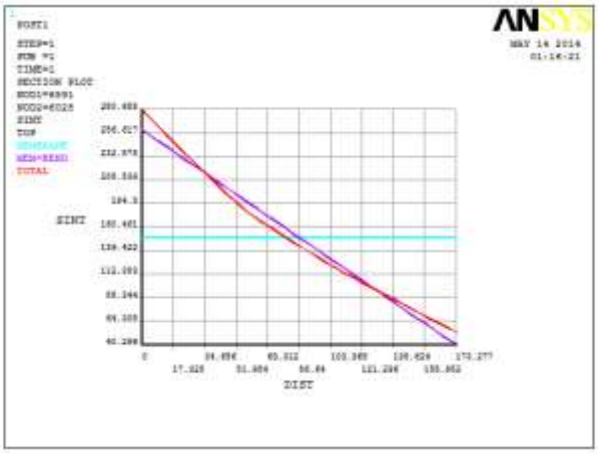

Fig. 22 Linearized Stress Intensity Plot for case 3

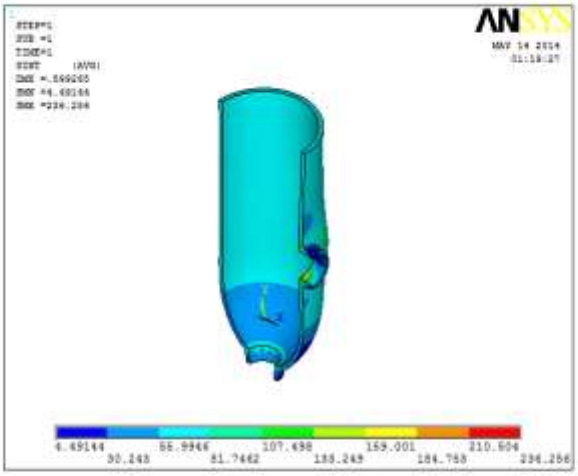

Fig. 23 Stress Intensity distribution in pressure vessel for case 4

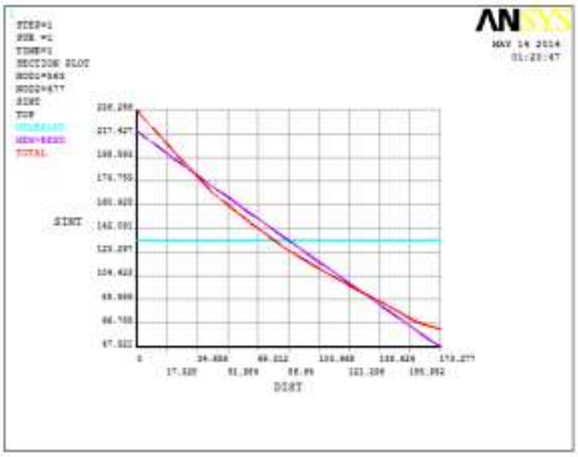

Fig. 24 Linearized Stress Intensity Plot for case 4

The stress intensity plot for case 1, 2, 3, and 4 is shown in figure 17, 19, 21, and 23 respectively. The linearized stress intensity values are plotted in the form of graph by taking SINT (stress intensity) in y-axis and the DIST (Specifies the viewing distance for magnifications and perspective) around the circumference in x-axis as shown in figure 18, 20, 22, and 24 respectively. In case1 the stress intensity plot shows that induced stress in this pressure vessel are higher than allowable limits as per ASME code Section VIII. So that pressure vessel is failed to withstand the forces induced in pressure vessel. As results the crack is formed at the nozzle and vessel intersection. In case2 the stress induced in the vessel are lower than the allowable limits as per ASME code. So these vessels can ability to withstand the forces induced in pressure vessel. In case 3 and 4 the induced stresses are the one of the good acceptable value such as within allowable limits but the weight of the pressure vessel is high and cost ineffective. So that in case2 the 50 mm is the required critical wall thickness of pressure vessel to withstand the forces induced in pressure vessel. So that the induced stresses are within allowable limits.
Variation of stresses with respect to the thickness of the pressure vessel are given below table 1.2

Table 1.2 Thickness Optimization Results

Cases	Thickness (mm)	Analytical Membrane stress (MPa)	Membrane stress (MPa)	Membrane + bending stress(MPa)	Stress at nozzle–vessel intersection (MPa)
1	25	269.54	269.012	422.28	448.44
2	50	136.58	136.58	302.83	325.22
3	75	92.29	94	255.88	280.65
4	100	70.17	71.96	218.88	236.25

Conclusions

Stress analysis plays a very important role in the design of pressure vessel, since it ensures the safe design standards. In this research work, static stress analyses are carried out for low carbon steel pressure vessel using ANSYS V 14.

From the research work, it is observed that:

The maximum stress values are obtained from the analysis for the above load case. From result of analysis it can be observed that the maximum stress occurs at the junction of pull out region and the nozzle and the maximum displacement occurs in the middle of pullout region. High stress concentration is developed at this location due to abrupt change in the geometry and consequent change in stress flow. In this location crack will form.

The stresses in the vessel thickness are increasing with reduction of thickness. For case1 25 mm thickness the induced stress is more than allowable value 339.4 MPa. Here, membrane and bending stresses are within allowable limits for cases 2, 3, and 4 Considered. But the equivalent stress intensity at nozzle-vessel intersection is increasing abruptly as thickness is reducing. Particularly at 25mm thickness the vessel will fail at intersection because stress is higher than allowable limits. So slight modification is made in the original design i.e. increasing thickness from 25 mm to 50 mm. for higher thickness the stresses are within allowable limits but the material content and cost is high. Hence not recommended. The results of the ANSYS were compared with analytical values which are in good agreement.

Scope for Future Study and Research

- The vibration analyses of the pressure vessel is carried out
- Analyses of the pressure vessel carried out under earth quake load
- Thermal analysis carried out to study the effect of fluid temperature.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 1.852

References

[1] B.S.Thakkar and S.A.Thakkar, "Design of pressure vessel using ASME code Section VII, division 1". IJAERS Vol. I Issue II, 2012 PP.228-234.

[2] Mani N, Thanigaiyarasu G, Chellapandi P. "Stress Analysis of Steam Generator Shell Nozzle Junction" International Journal of Engineering Science and Technology Vol. 2 Issue 7, 2010, PP.2690-2698.

[3] R.M. Tayade, Binesh P vyas, Ankit D Kumbhani, "Design of vertical pressure vessel using PVElite software". IJERT Vol. 2 Issue 3, 2013 PP. 78-81.

[4] R.C. Carbonari a, P.A. Muñoz-Rojas b, E.Q. Andrade c, and G.H. Paulino, "Design of pressure vessels using shape optimization an integrated approach'' International Journal of Pressure Vessels and Piping, Vol. 88, 2011, PP. 198-212.

[5] Heckman, David, "Finite element analysis of pressure" vessels,http://www.mbari.org/education/internship/98interns/98internpapers/98heckman.html MBARI 1998

[6] Sunil A. Patil "Finite element analysis of optimized compound cylinder" Journal of Mechanical Engineering Research. Vol. 5, 2013 PP. 90-96.

[7] Alfred Franklin V. "Stress analysis of bolted connection in space vehicles" International Journal on Design and Manufacturing Technologies, Vol.2, 2008 PP. 26-31.

[8] Nidhi Dwivedi, Veerendra Kumar, "Burst Pressure Prediction of Pressure Vessel using FEA" IJERT Vol. 1 Issue 7, 2012 PP. 1-6.

[9] Javad Marzbanrad, Amin Paykani, "Finite element analysis of composite high-pressure hydrogen storage Vessels" JMESCN, Vol.1 Issue 4 2013 PP.63-74.

[10] I.M.Jamadar, S.M.Patil, S.S.Chavan, "Thickness optimization of inclined pressure vessel Using non linear finite element analysis using design By analysis approach" IJMET, Volume 3, Issue 3, 2012, PP. 682-689

[11] Pravin Narale, Prof. P. S. Kachare, "Structural Analysis of Nozzle Attachment on Pressure Vessel Design" IJERA Vol. 2, Issue4, 2012, PP.1353-1358

[12] F. Y. M Wan, "Membrane And Bending Stresses In Shallow Spherical Shells'' Pergamon Press Ltd. Printed In Great Britain Int.J. Solid Structure, vol. 3, 1967, PP. 353 - 366.

[13] S al laham, "stress intensity factor and limit load Handbook'' Vol. 9, Issue 2, 1999, PP. 98-316.

[14] Shaik Abdul Lathuef and K.Chandra Sekhar, "Design and Structural Analysis of Pressure Vessel Due To Change of Nozzle Location and Shell Thickness'' IJAERS, Vol. I, Issue I, 2012, PP. 218-221.

Author Bibliography

<table>
<tr><td></td><td>Devaraju U.R
Mr. Devaraju U.R, has obtained his B.E. from Government Engineering College, Hassan and presently pursing M.Tech inGovernment Engineering College, Hassan
Email: devarajuhassan@gmail.com</td></tr>
<tr><td></td><td>Dr. Manjunath K
BE, M.Tech. - Assistant Professor
Dr. Manjunath K., obtained his B.E. from MCE, Hassan from Mysore University in1997, M.Tech. From PESCE, Mandya, VTU in 2003. Ph.D from VTU in 2012. He has 15 years of teaching experience. His areas of interest include CAD//CAE/CIM.
Ph: 98860-66740
Email: hassanmanju@gmail.com</td></tr>
<tr><td></td><td>Mahesh T.S.
BE, ME. - Associate Professor

Mr. Mahesh T S., obtained B.E. from SIT Tumkur in 1992, ME from UVCE Bangalore University in 2005 and Pursuing Ph.D. at VTU Belgaum. Before joining GECH, he served at KIT, Tiptur from 1997-2010. His areas of interest are Design Optimization, Mechanical Vibrations, FEA and Smart Materials. He has 01 national publication to his credit.
Ph: 94486-70213
Email: maheshkit@yahoo.co.in</td></tr>
</table>