"Structural Analysis of Inclined Pressure vessel Using FEM"

Imran M.Jamadar[a], Prof.R.M.Tayade[b], Mr.Vinay Patil[c]

[a]PG Student Mechanical Engineering Department, Veermata Jijabai Technological Institute(VJTI), Matunga,Mumbai, Maharashtra, Mumbai,India

[b]Prof.R.M.Tayade, Mechanical Engineering Department, Veermata Jijabai Technological Institute(VJTI), Matunga, Mumbai ,Maharashtra,India

[c]Mr.Vinay Patil, FEA Consultant, VAFTSY CAE, Hadapsar, Pune, Maharashtra,India

Abstract: Inclined pressure vessel (IPV) study using finite element analysis using ANSYS to find out stresses in the vessel for its structural stability is done in this paper. Inclined pressure vessel is used for production of nitrous oxide by ammonium nitrate pyrolysis reaction by passing the steam at around 200^0C and 1.37895 Mpa over the ammonium nitrate contained in the cylindrical vessel. Here design challenge is inclined nature of vessel as, ASME code enables design of Horizontal or a Vertical vessel but there is no provision for an Inclined Vessel in it.

Keywords-IPV, stress analysis, structural stability.

1. Introduction

Vessels, tanks, and pipelines that carry, store, or receive fluids are called pressure vessels. A pressure vessel is defined as a container with a pressure differential between inside and outside. 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 radio- active materials. Because of such hazards it is imperative that the design be such that no leakage can occur.

Designing, thus, involves estimation of stresses and deformations of the components at different critical points of a component for the specified loads and boundary conditions, so as to satisfy operational constraints .*Design* is associated with the calculation of dimensions of a component to withstand the applied loads and perform the desired function. *Analysis* is associated with the estimation of displacements or stresses in a component of assumed dimensions so that adequacy of assumed dimensions is validated.

2. Material selection

We are doing analysis of inclined pressure vessel for which ASME codes are not applicable. To overcome this firstly we have done analysis of horizontal vessel and its results are utilized to do further inclined vessel analysis. This section discusses the primary factors that influence material selection for pressure vessels and the maximum allowable material stresses specified by the ASME Code [9]. The mechanical design of a pressure vessel can proceed only after the materials have been specified. The ASME Code does not state what materials must be used in each application. It specifies what materials may be used for ASME Code vessels, plus rules and limitations on their use. But, it is up to the end user to specify the appropriate materials for each application considering various material selection factors in conjunction with ASME Code requirements. Accordingly, structural steel is selected as a material for vessel the properties of which are tabulated.

3. Modeling

The detailed 3D modeling of pressure vessel was done using CATIA V5R17.

4. Meshing

The accuracy of the FE model is highly dependent on the mesh employed, especially if higher order (cubic, quadratic etc.) elements are not used. In general, a finer mesh will produce more accurate results than a coarser mesh. At some point, one reaches a point of diminishing returns, where the increased mesh density fails to produce a significant change in the results. At this point the mesh is said to be "converged." This process of refining the mesh and evaluating the results is normally referred to as a "mesh convergence" study or analysis. Although many FE codes contain "error estimates" of one sort or another, mesh convergence remains the most reliable means of judging model accuracy. Coarse meshes almost always under-report the stresses in a model. It is not uncommon to have maximum reported stresses on the order of less than 50% of the converged stresses on a coarsely meshed model. Thus, without consideration of mesh convergence, gross errors in stress estimates are quite possible. If higher order elements are used, good results can be obtained with fewer elements. Either mesh convergence analysis or a reliable error estimate is absolutely necessary to quantify the analysis results. Typically, an increase of less than 5% in the stress levels after a doubling of mesh density or an "error estimate" of less than 0.05 will ensure that the indicated stresses are within 5-10% of the "converged" values [2]. Some FE codes employ an adaptive process to automatically refine the mesh and/or increase the order of the elements to reach the desired degree of accuracy. When available, these processes can save a lot of manual effort. They do not, however, completely relieve the engineer of the need to check the results.

5. Boundary conditions

One of the most significant sources of errors in FE modeling is the inaccurate (or inappropriate) modeling of the loads and restraints on a model. For example, fully fixing (restraining) all of the nodes on the end of a pressure vessel do not represent the same condition as does a normal head. With any normal head, the radial stiffness is not infinite, thus radial expansion under pressure loading will occur. This cannot be the case when the node is fixed. The symmetrical boundary condition always restrains translations normal to the plane of symmetry and the rotations in the plane of symmetry. In this case, we would have restrained translation in the X direction and the rotations about the Y and Z axes. Forces and moments may be applied to a model in a number of different ways. The particular method chosen is often a function of the pre-processor used to generate the model. Some pre-processors will automatically distribute a force or moment along a line or surface, such as the end of the nozzle. Even though we are using a modern computer-based tool, the sum of the forces and moments about each axis must remain equal to zero when the body is at rest.

6. Stress Analysis

Stress analysis is the determination of the relationship between external and internal forces applied to a vessel and the corresponding stress. The emphasis is not how to do stress analysis in particular, but rather how to analyze vessels and their component parts in an effort to arrive at an economical and safe design-the difference being that we analyze stresses where necessary to determine thickness of material and sizes of members. It is not necessary to find every stress but rather to know the governing stresses and how they relate to the vessel or its respective parts, attachments, and supports [4].

The starting place for stress analysis is to determine all design conditions for a given problem and then determine all the related external forces. We must then relate these external forces to the vessel parts which must resist them to find the corresponding stresses. By isolating the causes (loadings), the effects (stress) can be more accurately determined.

The designer must also be keenly aware of the types of loads and how they relate to the vessel as a whole. Are the effects long or short term? Do they apply to a localized portion of the vessel or are they uniform throughout? How these stresses are interpreted and combined, what significance they have to the overall safety of the vessel, and what allowable stresses are applied will be determined by three things:

1. The strength failure theory utilized.

2. The types and categories of loadings.

3. The hazard the stress represents to the vessel

The basic intent of design by analysis is to determine the stress conditions in a pressure vessel under load for each load condition for the vessel's entire operational life. In order to meet the intent and specifications of Section VIII, Division 2, the designer must carry out a sufficiently detailed stress analysis of the vessel to show compliance with the stress limitations imposed by the Code for the material from which the vessel is to be fabricated. The designer may specify any manner by which to determine the stresses in the vessel under load, so long as use of the methodologies employed.

This investigation primarily deals with the probable causes of in-service damage of IPV with approximate estimation of stresses. The design temperature and pressure of vessel 149^0C and 1.3789Mpa, respectively. There were four numbers of openings, Viz.entry and exit of steam, Exit of Nitrous oxide and drain. The vessel thickness was around 9.65mm. Stress analysis was carried out by finite element method using ANSYS 13.0 code.

Structural Elements Used for the Analysis: The higher order hexahedron element is used for meshing [8].

Boundary Constraints (figure 4): The whole vessel is supported on two saddle supports. The upper saddle was fixed while to the lower saddle cylindrical support was provided. Vessel was analyzed using ANSYS code on the basis of strength for internal pressure 1.38 Mpa, plus Self Weight considering portions of the drum of 1275.55 mm length with inside diameter of drum 304.8 mm, thickness of the shell 9.65mm. Considering the symmetry of the unit, one half of the unit was analyzed for stress calculations. The finite element model of the component under investigation is shown in Fig. 1

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 3, May - 2012

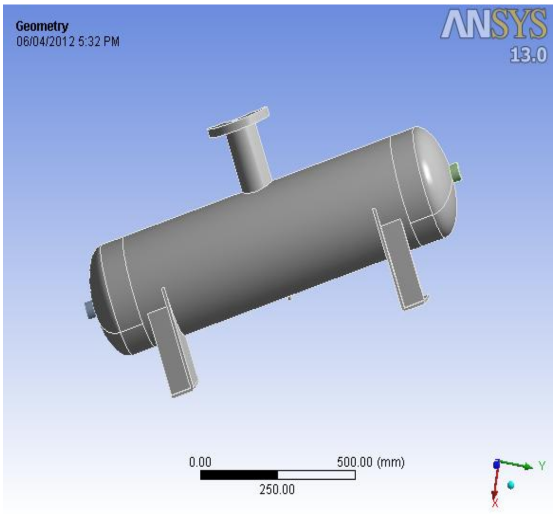

Fig.1 3DModel

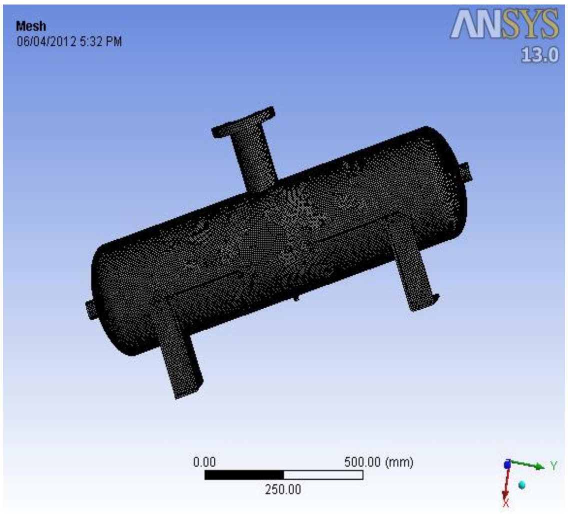

Fig.2 Meshing

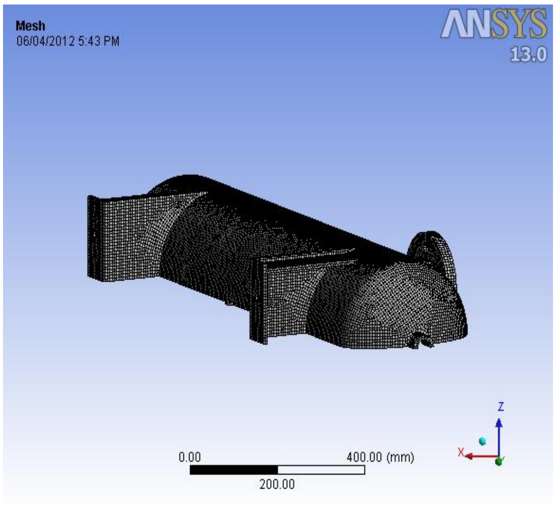

Fig.3 Axis-symmetric model

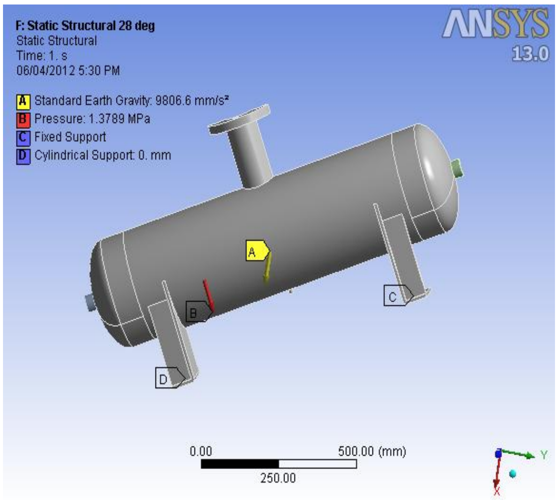

Fig.4Boundry Conditions

Material	Structural steel
Modulus of Elasticity in Mpa	2e5
Poisson's Ratio	0.3
Density Kg/m^3	7850
Tensile Yield Strength Mpa	250
Tensile Ultimate Steel Mpa	460
Allowable Hoop Stress Mpa	23.13
Allowable Equivalent Stress By PV Elite Software Code,Mpa	117

Table.1 Material Properties

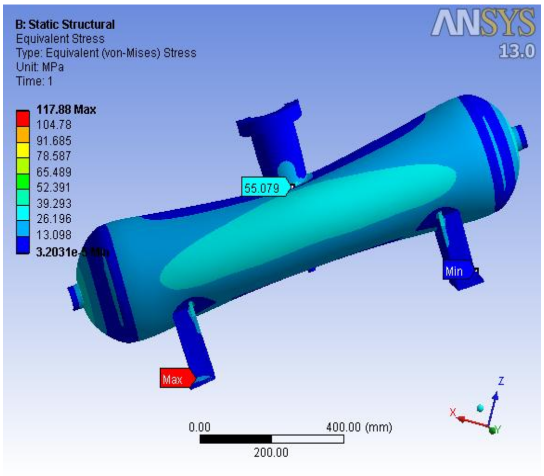

Fig.5 Equivalent Von-Mises Stress Plot

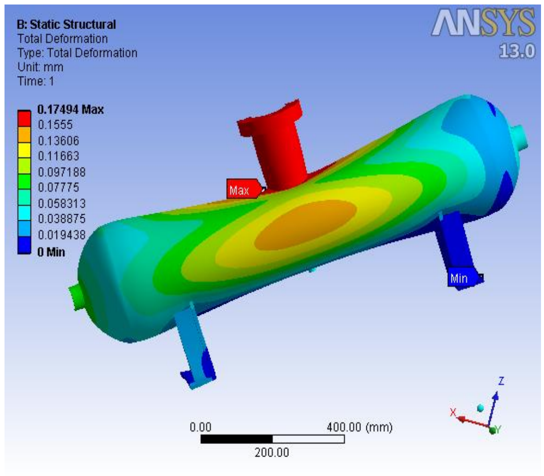

Fig.6 Total Deformation Plot

Angle	Nodes	Elements	Stress(Equivalent Von-Mises)Mpa	Total Deformation(mm)	Linearised Equivalent Membrane stress across vessel thickness
0	512863	214566	116.07	0.17523	32.741
4	514642	216241	116.10	0.17520	32.615
8	512896	215012	116.6	0.17516	32.577
12	521004	228791	116.96	0.17514	32.845
16	512635	213650	117.05	0.17512	32.954
20	532145	246531	117.16	0.17501	33.126
24	541236	250169	117.32	0.17499	33.469
28	532147	246545	117.50	0.17497	33.611
32	512347	203651	117.88	0.17494	33.805

Table.2 Stress and deformation values at various angles

Graphs:

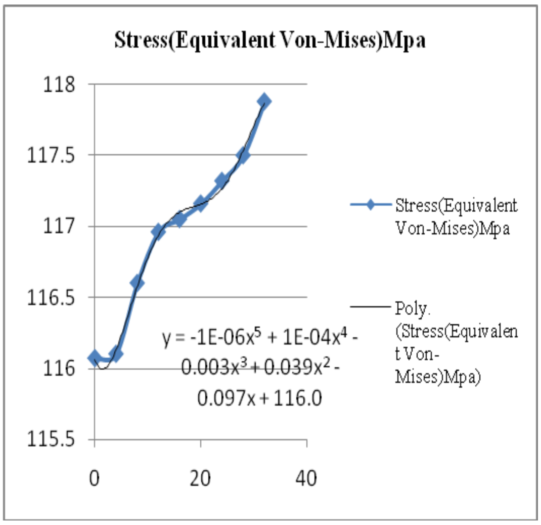

Fig.7 Inclination V/s Equivalent Stress

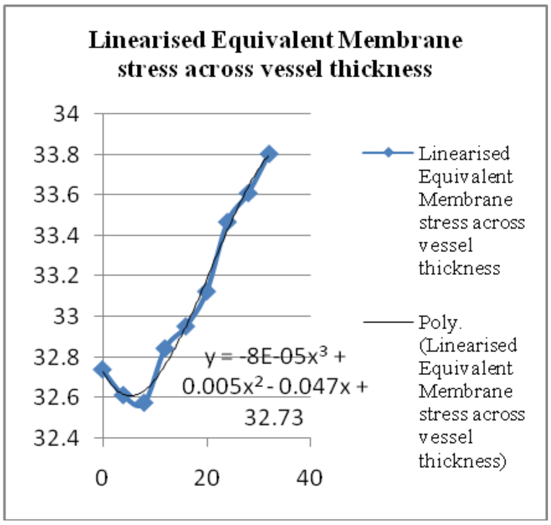

Fig.8 Inclination V/s Linearised Stress

7.Discussion and Conclusions

From above analysis it was observed that maximum equivalent stress observed in the vessel around the cylindrical support was about 117.88Mpa for vessel angle 32^0, it is exceeding allowable limits but it is less than yield strength and ultimate tensile strength .so failure will not be observed in this condition but to provide sufficient factor of safety and reduce stress below allowable limits some modification in this area are necessary. The deformation was observed to be maximum around nozzle vessel intersection 0.17494mm for 32^0 inclinations. The deformation was observed to be minimum around fixed support. Partially superimposed weld heat affected zones of nozzle weld caused localized adverse non liner condition and high load that facilitated initiation and propagation of cracks. From fig.8 and Fig..9 it is concluded that the equivalent Von-Mises Stress and linearised membrane stress along vessel thickness are increasing with inclination of vessel.

References

[1] CHEN Ru-xun, Structure analysis for filament-wound cylinder pressure vessel. Journal of Solid Rocket Technology. Vol.27 No.2, 2004, 105-107(3).

[2] Jaroslav Mackerle Finite Elements in the analysis of pressure vessel and piping.,1996 Elsevier publicationPp279-339

[3] Heckman, David: Finite element analysis of pressure vessels, MBARI 1998 Pp 1-7

[4] Dong-xia Liu et.al /Nonlinear Finite Element Analysis of Mechanical Characteristics on CFRP Composite Pressure Vessels IOP Conf. Series: Materials Science and Engineering 10 (2010) 012098

[5] P. V. Marcal, Elastic-plastic analysis of pressure vessel components Paper presented at 1st Pressure Vessel and Piping Conference Use of the Computer in Pressure Vessel Analysis ASME Computer Seminar Dallas, Texas, September 20, 1968

[6] J.L. Otegui et.al/Common root causes of recent failures of flanges in pressure vessels subjected to dynamic loads Elsevier -Engineering Failure Analysis 16 (2009) Pp1825–1836

[7] Young, W. (1989). Roark's Formulas for Stress & Strain. McGraw-Hill, New York, New York.

[8] **ANSYS (2004),** "Structural Analysis Guide", ANSYS Inc., Canonsburg – USA.

[9] ASME Boiler and Pressure Vessel Code. Am Soc Mech Eng; 1999