International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 11, November-2014

Optimization of Hemispherical Pressure Vessel Chamber by using Finite Element Analysis

A. Radha[1]
M.Tech student
Department of Mechanical Engineering
Vijay Rural Engineering College
Nizamabad, Telangana.

G. Bheemanna[2]
Assistant Professor
Department of Mechanical Engineering
Vijay Rural Engineering College
Nizamabad, Telangana

Abstract: **A pressure vessel is a container used to store substances at more than 15 psi, it can withstand greater than normal amounts of pressure without bursting. Those are used to contain a multitude of substances including air, water, chemicals, nitrogen, and fuel. They are used in paper and pulp, energy, chemical industries, food and beverage. The objective of present work is to design a pressure vessel whose sole purpose is to withstand the pressure of the substance stored in it. Modeling is done in Pro-E and is imported to Ansys for further analysis. The dimensions of the pressure vessel have been arrived by analytical calculations. The shell is analyzed for the Von-Mises and the shear stresses induced in the plate, finally the analytical calculations and the values obtained by analysis are compared. The Von-Mises stress induced in the bolt material is 573.04MPa, which is less than the maximum stress i.e. 680MPa. Hence it is found that the design is safe. The shear stress induced in the plate is 30MPa, which is less than the allowable stress in the plate material**

Keywords: cylindrical shell, Bolts, Pro/E, ANASYS

I. INTRODUCTION

pressure vessel is subjected to internal fluid pressure of 75 bar. This pressure vessel is used to test the components, which are used in submarine applications. This cylinder pressure chamber of mild steel is subjected to internal fluid pressure of 75bar, The thickness of the shell wall (t),The diameter of bolt and number of bolts required for flange and pressure chamber assembly, The thickness of flange for the pressure chamber and hemispherical cover plate, Thicknesses of the end cover plates and The thickness of the gasket in between the flange and cylindrical chamber are determined in order to sustain the internal fluid pressure The present work is carried out to analyze a pressure vessel which can withstand to a pressure of substance more then normal pressure stored in it. David Heckman[1] et.al (1998) presents For this pressure vessel, the model had a sharp corner, where in the actual pressure vessel there is a small radius which reduces the stress. data must be verified with hand calculations to confirm that solutions are relatively accurate. Where results are questionable, such as in the final contact element model, one must understand just what the finite element model is modeling and how well this approximates the actual subject. Farhad Nabhani[2] et.al discussing about the three main factors for the development of stresses in pressure vessels. These are thickness, nozzle positions and the joints of the enclosure heads. These were are tested together to finally reduce the stresses for all types of pressure vessels. Siva Krishna raparlal[3] et.al They compared both the solid wall and multilevel pressure vessel to overcome the disadvantage behind solid wall by decreasing the overall weight and cost and of the component. Here we are comparing the strees variations in a very detailed manner through mathematical calculations. Shaik Abdul Lathuef[4] et.al They are discussing about the Boiler and Pressure Vessel (B&PV) Code and the use of international pressure vessel codes such as EN13445. the main discussion was taken place regarding the thickness of the shell by replacing the minimum thickness 8mm for safe design conditions Bandarupalli Praneeth[5] et.al finite element analysis of pressure vessel and piping design are discussed here. Features of multilayered high pressure vessels, their advantages over mono block vessel are discussed. The theoretical values and ANSYS values are compared for both solid wall and multilayer pressure vessels over the mono block vessels.

II. METHODOLOGY

To achieve the above objective the following methodology has been adopted in the present work. A pressure vessel which will withstand a pressure of 75bar has been theoretically designed Modeling of the pressure vessel is done using pro-E The model is imported to Ansys and analysis is preformed as follows. Shell element is chosen & a real constant of 3mm is added which is the thickness of the shell material properties are added. Meshing is done, finally the boundary conditions are applied & it is solved. After solution the results are viewed in general postprocessor. Then the results from the analytical method & the Ansys are compared. The shear stress induced in the plate is less than allowable stress in the plate material. Hence the design is safe.

2.1 Problem Description

Material = mild steel, Internal pressure = 75 bar

Internal diameter =400 mm, Height of the cylinder = 400mm

Young's modulus (E) = 207GPa =207000N/mm^2

Poisson's ratio (μ) = 0.3 Yield strength (σ_y) =400MPa

2.2 Modeling of pressure vessel

The Pro-E provides the power of parametric design. With parameters, we define the model according to the size, shape and positional relationship of its parts.

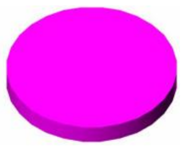

Figure: 2.1

The figure 2.1 shows the circular flat plate, the thickness of the circular flat plate obtained by the analytical calculation is 33.54mm. But, it approximated to 34mm, as this is the standard thickness available

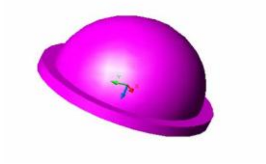

Figure: 2.2

The figure 2.2 shows the Hemispherical cover plate, the thickness of the cover plate obtained by analytical calculation is 3.75mm. But, it is approximated to 4mm, as this is the standard thickness available.

Figure 2.3

The figure 2.3 shows Hemispherical cover plate circular flat plate flange shell wall the thickness of the cylindrical pressure vessel, the result obtained by analytical calculations is 7.5mm. But, it approximated to 8mm, as this is the standard thickness available.

2.3 Analysis of modeled pressure vessel

ANSYS is a general purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis(both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems. Basically there are four stages which are to be executed while solving any simulation problem by using ANSYS. They are as follows.

Preferences:

In this stage, the option structural is selected out of structural, thermal, Ansys fluid because the problem to be solved belongs to structural category.

Preprocessor:

In this stage, the following steps are executed.

Initially, geometry of required size and shape is build or imported from other CAD files. Depending upon the geometry, element shape is selected i.e.., for the current problem 'SHELL-93' is selected. Inputting the real constants such as thickness of the shell (t=8mm), thickness of the circular flat plate (=34mm), thickness of the flange (=10mm), thickness of the hemispherical cover plate (=4mm). Defining the element material and its properties. For the present problem Isotropic material (Mild steel) is selected whose material properties are Poisson's ratio (μ=0.3), Young's modulus (E=207GPa).

Solution

In this stage the type of analysis is selected. Again in which either 'STATIC' or 'DYNAMIC' option is selected.

For E.g. Structural, Thermal, etc.

For the current problem the option structural along with static is selected because the pressure vessel is designed for static pressure load. A pressure of 7.5bar is applied on the inner walls of the pressure vessel and degrees of freedom are arrested wherever required. Finally the option 'solve current LS' is executed.

General postprocessor:

In this stage, results are viewed and plotted.

Time history postprocessor:

By executing this option, graphs are plotted.

For e.g. Displacement vs. time, etc are plotted.

2.3.1finite element analysis of pressure vessel

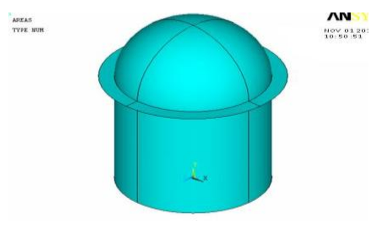

Fig: 2.4 Finite element model

The process of dividing the entire body into several smaller units is called 'meshing,. The geometric model which is created in ANSYS is divided into smaller elements to form the meshed model. Here in this problem we use shell 93 element is used.

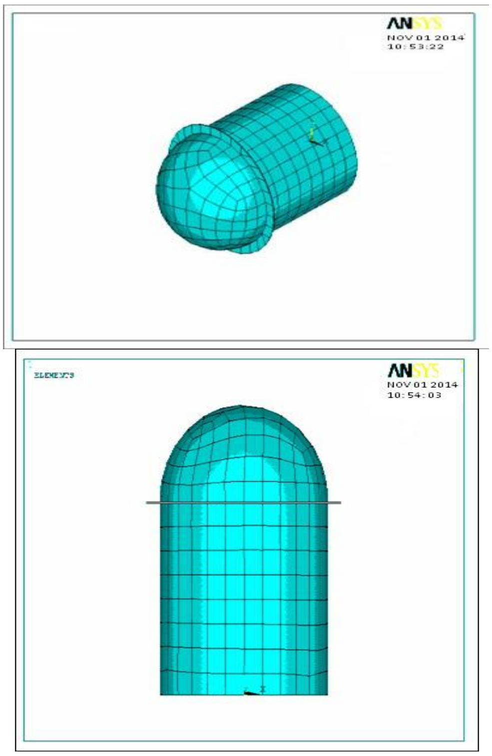

Fig: 2.5 Meshing of finite element model

2.3.2 Von-Mises stress variation

I have plotted the stress distribution of pressure vessel .The maximum Von- Mises stresses are found to be 194.105MPa.The stress distribution of full expansion of model is shown in figure

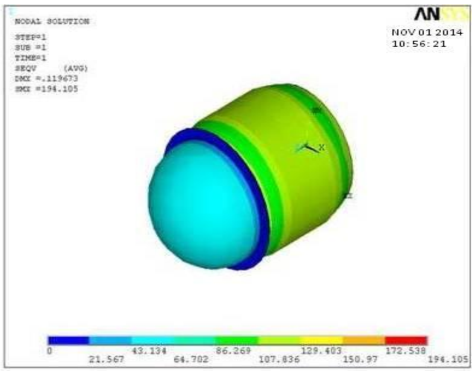

Fig:2.6 deformed+undeformed shape

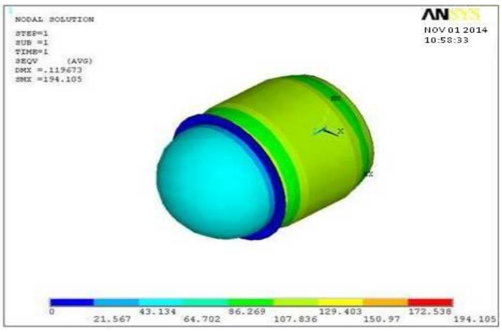

Fig:2.7deformed shape

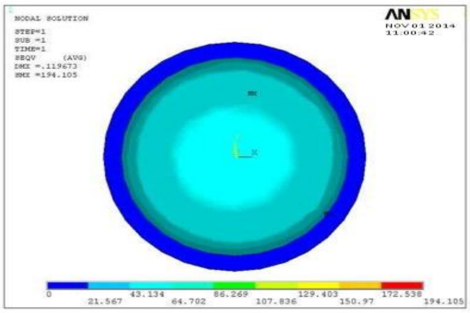

Fig:2.8 Top view of Von- Mises stress variation deformed shape

III. RESULTS AND DISCUSSION

The results of static analysis of pressure vessel by using Ansys, the discussions and the results are In static analysis, the pressure 75bar(load) is applied in the pressure Vessel from that the minimum deflection is found to be 0.119673mm. Finally the Von-Mises stress produced by using FINITE ELEMENT METHOD in the pressure test chamber is 194.105MPa, which is approximately equal to the allowable stress (200MPa) for the material of the pressure vessel. Hence by reducing the thickness of pressure test chamber i.e. shell wall we can obtain stress less than allowable stress of material in interns reduces the weight of the pressure vessel and economical

PARTS	ANALAYTICAL THICKNESS IN MM	APROXIMATION THICKNESS IN MM	VON MISES STRESS IN MPa
PRESSURE TEST CHAMBER (SHELL WALL)	7.5	8	194.105
COVER PLATE	33.5	34	30
BOLT	18	C40	573.04

IV. CONCLUSION

The following conclusions have been drawn from the present work.

The thickness of the shell wall, obtained by analytical calculation is 7.5mm, but it is approximated to 8mm, as this is the standard thickness available.

The thickness of the base cover plate, which is circular in shape, is obtained as 34mm.

The shear stress induced in the plate is 30MPa, which is less than the allowable stress in the plate material. Hence the design of plate is safe.

The thickness of the flange by analytical calculations is found to be 10mm

The thickness of the hemispherical cover plate, which is at the top of the pressure vessel, obtained by analytical calculation is 3.75mm. But it is approximated to 4mm,as this is the standard thickness available

The diameter of the bolt of C40 material is calculated by analytical method and obtained as 18mm by assuming the number of bolts to be 12 and considering the 'SILICONE' gasket material in between the flanges.

The shear stress induced in the bolt material is 573.04MPa.Therefore the stress produced in the bolt is less than the maximum stress i.e. 680MPa. Hence the design of bolt is safe.

The Von- Mises stress produced by using FINITE ELEMENT METHOD in the pressure test chamber is 194.105MPa,which is approximately equal to the allowable stress for the material of the pressure vessel.

REFERENCES

1. "Finite Element Analysis of Pressure Vessels" David Heckman, University of California, Davis Mentor: Gene Massion, Mark Greise Summer 1998
2. "Reduction of Stresses in Cylindrical Pressure Vessels Using Finite Element Analysis" Farhad Nabhani, Temilade Ladokun and Vahid Askari Teesside University, School of Science and Engineering, Middlesbrough, TS1 3BA, UK
3. "Design and analysis of multilayer high pressure vessels" department of mechanical engineering qis college of engineering &technology ongole, andhra Pradesh
4. Design and structural analysis of pressure vessel due to change of nozzle location and shell thickness" shaik abdul lathuef department of mechanical engineering, qis college of engineering and technology, ongole, Andhra pradesh.
5. "Finite Element Analysis of Pressure Vessel and Piping Design"Bandarupalli Praneeth, Dept. of M.E, Nimra Institute of Science & Technology, Vijayawada, A.P., India.