Design and Structural Analysis of Spherical Pressure Vessel used to Operate a Pneumatic Jack Hammer

Rahul S. Kandure[1], Gaurav V. Lahoti[2]
[1]M. Tech in Mechanical Engineering, D.I.E.M.S. Aurangabad, India.
[2]M. Tech in CAD/CAM, S.G.G.S.I.E&T, Nanded, India.

***Abstract* - This paper presents on a design and analysis of spherical pressure vessel used to operate a pneumatic jack hammer. The pressure vessel is a type of vessel used to store liquids, vapors and gases under a pressure different from the atmospheric pressure. Different shapes of pressure vessels exist but most generally cylindrical and spherical shapes are used. We are focused on spherical pressure vessel because strength of spherical pressure vessel is double the strength of the cylindrical pressure vessel with same wall thickness and same inside pressure acted on wall. High pressure increase is developed in the pressure vessel and pressure vessel has to survive severe forces. We are designing spherical pressure vessel by using SOLIDWORKS and the analysis is complete by using ANSYS. Mainly we are focused on the analysis of von mises stress and total deformation of the spherical pressure vessel.**

Key Words: — Pressure Vessel, SOLIDWORKS, ANSYS.

1) INTRODUCTION

The pressure vessel is a type of vessel used to store liquids, vapors and gases under a pressure different from the atmospheric pressure. Different shapes of pressure vessels exist but most generally cylindrical and spherical shapes are used. We are focused on spherical pressure vessel because strength of spherical pressure vessel is double the strength of the cylindrical pressure vessel with same wall thickness and same inside pressure. Pressure vessels find varied applications in process and chemical industries, in space and ocean depths, thermal and nuclear power plants and fluid supply systems in industries. The failure of pressure vessel may result in loss of life, health hazards and damage of property. Due to practical requirements, pressure vessels are often equipped with openings of various shapes, sizes and positions.

Spherical pressure vessel has different components such as sphere, supporting ring and legs, viewing windows and nozzles. Pressure Vessels have openings to provide accommodations to manholes, and nozzles. Openings vary in size from small drain nozzles to full vessel size openings. The openings cannot be avoided because of various piping or measuring gauge attachments.

Classifications of pressure vessels:-

a) According to the thickness provided to the pressure vessel:-

1. Thin shell: If ratio of the thickness to diameter of shell is less than 1/20th, then the shell is said to be thin shell. Thin pressure vessels can withstand only internal fluid pressure, but they cannot withstand external fluid pressure.

2. Thick shell: If ratio of the thickness to diameter of shell is more than 1/20th, then the shell is said to be thick shell. Thick pressure vessels can survive both internal and external fluid pressure.

b) According to the internal fluid pressure:-

1. Thin shell: If the permissible stress (σ_t) is more than 6 times of internal fluid pressure (p), then the shell is supposed to be thin shell.

2. Thick shell: If the permissible stress (σ_t) is less than 6 times of internal fluid pressure (p), then the shell is supposed to be thin shell.

2. LITERATURE REVIEW

In this section research papers are discussed related to the present work. Published papers are highlight in this section.

A. *David Heckman[5]*

Finite element analysis is an extremely powerful tool for pressure vessel analysis when used correctly. Tested models were run with errors ranging from seven to nearly zero per cent error and could be run in a relatively short time. However, even with such results the operator still is essential to be well-informed of not just how to run the finite element analysis, but also how to read the results. 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. For this pressure vessel, the model had a sharp corner, where in the real pressure vessel there is a small radius which decreases the stress. For pressure vessels finite element analysis provides an extra tool for use in analysis. However, it must be compared to other available data, not taken as being correct just because it looks right. Used with this understanding, finite element analysis offers great vision into the intricate interactions found in pressure vessel design.

B. *Farhad Nabhani[6]*

Three main features are seen to contribute extensively to the improvement of stresses in pressure vessels. These are thickness, nozzle positions and the joints of the enclosure

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 09, September-2019

heads. From the model design cases used in this research, it could be seen that as the thickness of pressure vessel rises, the stresses falls, however this is not a viable solution due to cost. Nozzles though are safety relief devices and important component of pressure vessels comes with its own drawbacks of growing weak areas and stress concentration. However, this was moderated by use of high alloy reinforcement pads as applied in the design case two and three of this work. The high strength reinforcement pad used has a chemical composition of titanium 0.4 to 1.20%; hollow disc shaped with rectangular section can also reduce the stresses concentration around the nozzle. Finally, the joints of enclosure heads either welded or bolted were recognized as areas with the highest concentration of stresses i.e. with peak stress. Addition of 254 mm skirt length at the end of enclosure heads afford extra room for the stresses to grow slowly in the wall of the head areas, thus making the pressure vessels more resistant to the loadings.

C. Siva Krishna Raparla[7]

The hypothetical values and ANSYS values are matched for both solid wall and multilayer pressure vessels. 26.02% is material saving by using multi-layered vessels in the place of solid walled vessel. These reductions not only the overall weight of the component but also the cost of the material required to manufacture the pressure vessel. This is one of the main aspects of designer to keep the weight and cost as low as possible. The Stress variation from inner side to outer side of the multi-layered pressure vessel is around 12.5%, in which as to that of solid wall vessel is 17.35%. This means that the stress distribution is uniform when compared to that of solid wall vessel. Minimization of stress concentration is another most important aspect of the designer. It also shows that the material is utilized most effectively in the fabrication of shell. Theoretical calculated values by using different formulas are very close to that of the values obtained from ANSYS analysis. This indicates that ANSYS analysis is suitable for multilayer pressure vessels. Owing to the advantages of the multi layered pressure vessels over the conventional mono block pressure vessels, it is concluded that multi layered pressure vessels are superior for high pressures and high temperature operating conditions.

3. PROBLEM STATEMENT

"To design and analyse a spherical pressure vessel used to store the compressed air of 1000 litre capacity working at a pressure of 15 bar for operating a pneumatic jack hammer."

Explanation: Pneumatic jack hammers are operated at high pressures therefore generally reciprocating compressors are employed. As we know the discharge of reciprocating compressor is pulsating, which unable to run the jack hammer. Hence the discharge of reciprocating compressor needs to be store in a pressure vessel which will ensure the continuous supply of compressed air to jack hammer.

4. MATERIAL SELECTION

A number of materials have been used pressure vessel fabrication. The selection of material is based on the appropriateness of the design requirement. The selection of materials of the shell is going to takings into account the suitability of the materials with the maximum operating pressure and fabrication process. The Grade 25 cast iron is grey cast iron in the as-fabricated condition. Grade 25 is the ASTM designation for this material. The material assigned to the different components of the spherical pressure vessel is shown in table-1. And its properties are shown in table-2.

Table -1: Material assignment

Sphere	ASTM grade 25 Gray CI
Supporting ring and legs	ASTM grade 25 Gray CI
Nozzles	ASTM grade 25 Gray CI
Viewing window	ASTM grade 25 Gray CI

Table -2: Material properties

Ultimate tensile strength	179 Mpa
Modulus of elasticity	79-102 Gpa
Ultimate compressive strength	669 Mpa
Poissons ratio	0.29
Shear modulus	32-41 Gpa
Shear Strength	220 Mpa

4. DESIGN OF SPHERICAL PRESSURE VESSEL

The study of stresses in the spherical pressure vessel is based on the following assumptions:-

1) The stress along the radial direction will be small and negligible.
2) Shear stress across a cross section is negligible.
3) Plane sections remains plane.
4) R/t ratio ≥ 10 with thickness (t) being uniform and constant.
5) The theoretical pressure is gauge pressure.
6) Material properties are linearly isotropic and homogeneous.
7) Stress distribution throughout the wall thickness will not vary.
8) Working fluid has negligible weight.

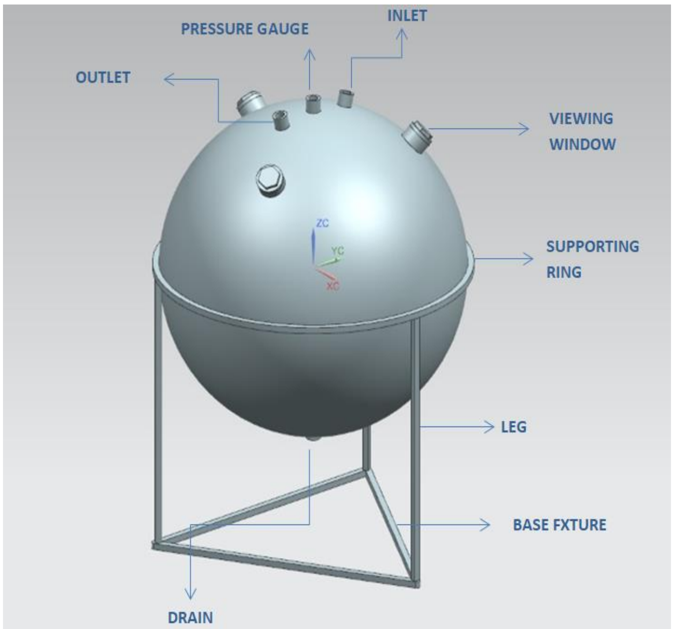

Fig -1: CAD model of Spherical pressure vessel

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 09, September-2019

4.1 Design of Sphere:-

To decide stresses in the spherical vessel let us cut through the sphere on a vertical diameter plane and segregate half of the shell and its fluid content as a single free body. Acting on this free body is the tensile stress σ in the wall of the vessel and the fluid pressure p.

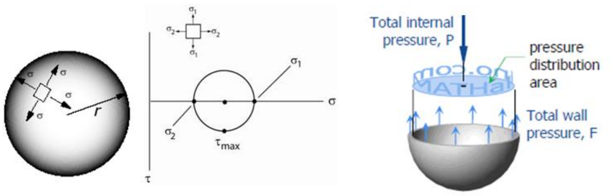

Fig -2: Pressure distribution in sphere vessel

The pressure that acts horizontally against plane circular area uniform and gives a resultant pressure force of $P = p\pi r^2$. The stress is uniform around the circumference and is uniformly distributed crossly the thickness of wall. The resultant horizontal force is σ (2πrm) t Where rm = r + t/2 but rm ~ r for thin walls.

As the evident from symmetry of a spectrochemical shell that we will obtain the same equation regardless of direction of cut through the center. The wall of a spherical pressure vessel is subjected to uniform tensile stresses in all direction. As the two stresses are equal Mohr's circle for in plane transformations reduces to a point as shown,

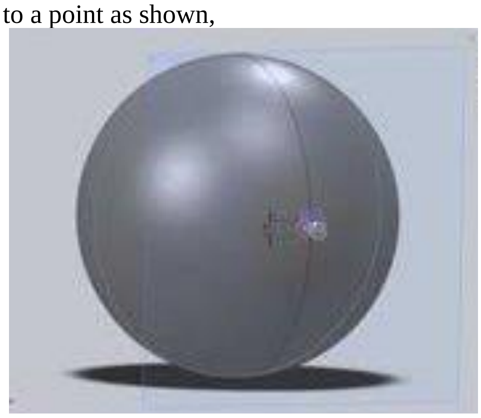

Fig -3: CAD model of sphere

Volume of sphere = $4/3\ \pi r^3$

Requirement = 1000 lit = 1 m^3.

Therefore, 1 = $4/3\ \pi r^3$ which gives r = 0.62 m.

Hence Internal diameter will be, d = 2× r

d= 2 × 620 = 1240 mm.

For safety, we take design pressure 1.5 times of working pressure.

Thickness of wall = ((p × d)/ (4×σ×η-p)) +c

= ((2 × 1240)/ (4×60×0.75-2)) +1.5

= 15 mm.

Where,

p = Internal pressure in bar,

d = Internal diameter in mm,

σ = Allowable stress of material in Mpa,

η = Circumferential weld efficiency for spot radiography,

c = Corrosion allowance in mm,

D = External diameter.

D = Internal diameter + 2 times thickness

= 1240 + 30 = 1270 mm

4.2 Design of Supporting ring and leg:-

Weight of sphere = Volume × Density

= $4/3\ \pi\ (R^3 - r^3) \times 7200$

= 5700 N

Cross section of leg is hollow square 25mm ×25mm with 1 `mm of thickness.

Therefore,

Cross sectional area = $25^2 - 23^2 = 96\ mm^2$.

Total area of 3 leg = 3 × 96 = 288 mm^2.

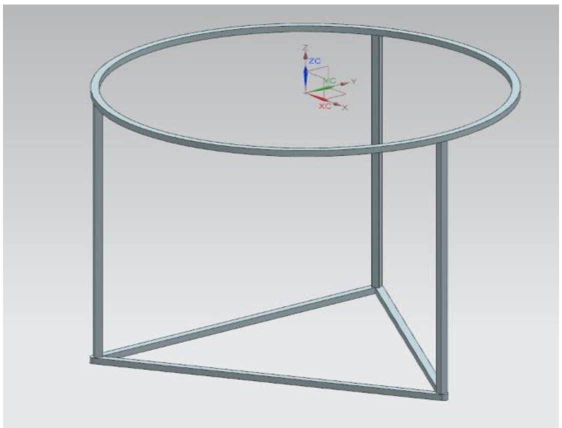

Fig -4: CAD model of supporting ring and legs

Strength check = Weight / Total area

= 5700 / 288 = 19.79 N/ mm^2.

Where,

R = External radius of sphere,

r = Internal radius of sphere.

Induced stress is less than the allowable stress, hence design is safe.

5. ANALYSIS OF SPHERICAL PRESSURE VESSEL

The CAD model of geometry is created in a Solid works and analysis is carried out in ansys workbench software.

Solidworks is cad software which is working on the sketch base modeler. It also supports other Computer Aided Engineering (CAE). It consists of solid modeler and utilizes a parametric feature approach for the design and assemblies of the components. Any model creation is start with 2D sketch; hence it is called sketch base modeler. The sketch contains of different entities such as points, lines, curves, conics (except the hyperbola), and splines. Dimensions are provided to the sketch to describe the scope and position of the geometry. Relations are used to describe features which include tangency, parallelism, perpendicularity, and concentricity. In an

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 09, September-2019

assembly, the analog to sketch relations are mates. Just as sketch relations explain conditions consisting of tangency, parallelism, and concentricity with recognize to sketch geometry, assembly mates define equivalence relations with admiration of different parts or components, permitting the easy creation of assemblies.

FEA contains of a computer model of a material or design that is loaded and examined for precise results. Scientifically, the structure is to be studied is divided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is expected to be resolved by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are advanced in expressions of the unknown nodal displacements. From this, the equations of equilibrium are collected in a matrix from which may be easily being automated and solved on a computer. After putting on the proper boundary conditions, the nodal displacements are determined via resolving the matrix stiffness equation. Once the nodal displacements are recognized, element stresses and strains may be calculated. Within all of these modelling systems, the Engineer can enclosure numerous algorithms which may additionally make the system perform linearly or non-linearly. Linear structures are significantly much less complex and usually forget about many weaknesses of model loading and behaviour.
Nonlinear systems can account for more realistic behaviour such as plastic deformation, changing loads etc. and is capable of testing a component all the way to failure. The following are the five basic steps involved in an FEA analysis:
1. Discretization of the domain
2. Applications of field/boundary conditions
3. Assembling the system equations
4. Solution for the system equations
5. Review of results.

The geometry required for validation by Finite analysis is created with the same major dimensions as specified in the experimental results that are the geometry having inner diameter (d) with 1240 mm and thickness (t) of the wall as 15 mm.

The meshing of spherical pressure vessel is proximity and curvature type to obtain good result of stress and deformation on a curvature portion. To obtain a very precise and accurate result we are carry fine mesh. In a spherical pressure vessel, all parts are stationary and are joined by welding. So, the analysis point of view, we are selecting the static structural model to obtain a stress and deformation of a sphere and supporting rings and legs. In an analysis set up of sphere, we are providing internal pressure inside the wall of sphere and provide fixed support to the ring. in the similar way, the weight of the sphere is applying on a ring in -Z direction and bottom of the legs are fixed.

5.1 Total Deformation of Sphere:-
In elastic deformation a material changes shape when a stress is applied to it but goes back to its original state when the stress is removed. The total deformation produce in the sphere is due to the internal pressure acted inside the wall of sphere. The total deformation of the sphere is easily visualizing in the fig. 6. The deformation of sphere is equal in all side i.e. 0.20052mm, except the sphere and ring contact.

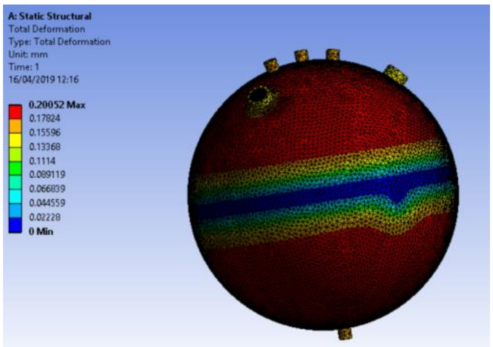

Fig -6: Total Deformation of Sphere

5.2 Equivalent stress Analysis of Sphere:-
The stresses produce in the sphere due to the internal pressure of the fluid is easily visualized as shown in fig. 7. The equivalent (von- Mises) stress produce in the sphere is 197.11 Mpa. We can easily show that the stress produces in the material provide on sphere is totally under control. The maximum stress is producing near the ring and sphere contact, whereas there are very small variations of stress in the remaining portion of sphere.

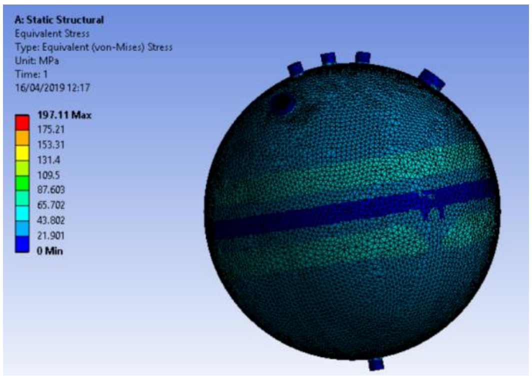

Fig -7: Equivalent stress Analysis of Sphere

5.3 Total Deformation of Supporting Ring & Leg
The total deformation produce in the Supporting Ring & Leg is due to the weight of the sphere. The total deformation of the Supporting Ring & Leg is shown in the fig. 8. Maximum deformation is 2.356 mm.

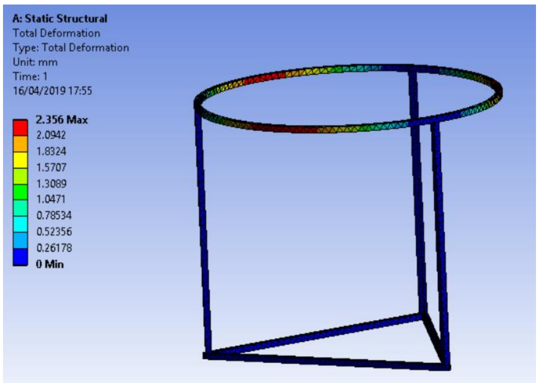

Fig -8: Total Deformation of Supporting Ring & Leg

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 09, September-2019

5.4 Equivalent stress Analysis of Supporting Ring & Leg:-

The stresses produce in the Supporting Ring & Leg due to the weight of the sphere is easily visualized as shown in fig. 9. The maximum principal stress produce in the Supporting Ring & Leg is 96.628 Mpa. The maximum stress is producing in contact region of Supporting Ring & Leg.

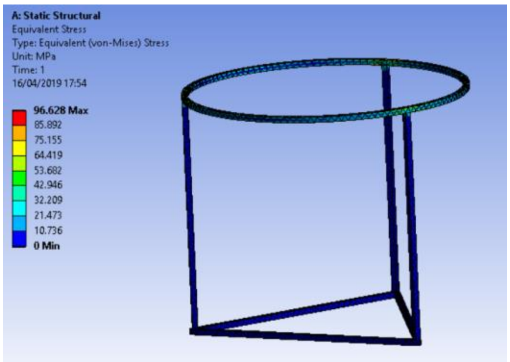

Fig -9: Equivalent stress Analysis of Supporting Ring & Leg

6. CONCLUSION

In this project, the radial and hoop stresses of a spherical pressure vessel are analysed by using ANSYS software the obtain results are found to be a safe. The spherical pressure vessel has a great advantage is that they have less surface area to volume ratio as compared to other shape of the pressure vessel. This means that, the heat transfer rate from the surroundings of the liquid inside the sphere will be lower as compared to the cylindrical or rectangular pressure vessels. It is possible to apply proposed procedure to optimize wall thickness over different zones of huge spherical pressure vessels.

7. REFERENCE

[1] NAfkar, M. N. Kamari, A. Paykani, “Design and analysis of spherical pressure vessel using finite element method”, World journal of modelling and simulation 10(2) (2014) 126-135

[2] H. S. Lee, J. H. Yoon, J. S. Park, “A study on failure characteristics of spherical pressure vessels”, Journal of material processing technology 164-165 (2005) 882-888

[3] Code for Unfired pressure vessel IS-2815-1969 prepared by Bureau of Indian Standards, New Delhi, Eighth receipt Aug.98, UDC 66.023:621.642

[4] Mechanical System Design - Anup Goel, Technical publications, fourth edition.

[5] David Heckman, “Finite Element Analysis of Pressure Vessels” University of California, Davis Mentor: Gene Massion, Mark Greise Summer 1998.

[6] Farhad Nabhani, Temilade Ladokun and Vahid Askari, “Reduction of Stresses in Cylindrical Pressure Vessels Using Finite Element Analysis” Teesside University, School of Science and Engineering, Middlesbrough, TS1 3BA, UK.

[7] Raparla, Siva Krishna, and T. Seshaiah. "Design and analysis of multilayer high pressure vessels." International Journal of Engineering Research and Application 2.1 (2012): 355-361.

[8] M. Jadav Hyder, M Asif, “Optimization of Location and Size of Opening in a Pressure Vessel Cylinder Using ANSYS”. Engineering Failure Analysis .Pp. 1-19, 2008.

[9] Joship Kacmarcik, Nedelijko Vukojevic, “Comparison of Design Method For Opening In Cylindrical Shells Under Internal Pressure Reinforced By Flush (Set-On) Nozzles”. 2011

[10] V.N. Skopinsky and A.B. Smetankin, “Modeling and Stress Analysis of Nozzle Connections in Ellipsoidal Heads of Pressure Vessels under External Loading.” Int. J. Of Applied Mechanics And Engineering, Vol.11, No.4, Pp.965-979, 2006

[11] Mechanics of solids by ROGER .T. FENNER.

[12] Machine Design by SHIGLEY.