International Journal for Ignited Minds (IJIMIINDS)

Determination of Burst Pressure of Spherical Shell

Balaji J[a] & Shivappa H A[b]

[a]PG Student, Dept. of Mechanical Engineering ,Dr. Ambedkar Institute of Technology, Karnataka, India.
[b]Asst. Prof, Dept. of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Karnataka, India.

ABSTRACT

It is important for every engineer to analyze and design the pressure vessel that will provide safety, durability and serviceability to the company. Accomplishing this task will require knowledge of parameters that affecting the pressure vessel due to varying loads, pressure and thickness of shell element. The most important one is that the given geometry of pressure vessel must be analyzed to assured it meet the design standards and design of pressure vessel is required to meet an acceptable stresses. Now a day, wide number of fire accidents are reported in news because of failure of these pressure vessels. Many standards like ASME, API provide guidelines for the design of the pressure vessels with safety factors. At the same time, it is equally important to know the burst pressure of a pressure vessel. In this perspective, current study would like to address the procedure to determine the burst pressure and burst failure locations of spherical pressure vessel using FEA, Commercial finite element software ANSYS is used. A detailed non-linear FEA analysis will be carried out for this purpose. Further, design iterations will be carried out to optimize the performance of spherical shells under same capacity.

Keywords- *Non-Linear Structural Analysis, Spherical Pressure Vessels, Material Non-Linearity, Burst Pressure, Internal Pressure.*

1. INTRODUCTION

Pressure vessels are closed structures which is used to store liquids and gases in industry and transportation under pressure. Pressure vessel can be of different shape and size depending upon the capacity of the container and pressure of the liquid or gases that it is going to contain/store. Example tanks, pipes, pressurized cabins, etc. When pressure vessel are under the service loads (pressure), the material of the pressure vessel experience the pressure loads or service loads and stresses are distributed all the way through. The normal stresses resulting from this internal pressure is a function of the radius of the element under consideration, the shape of the pressure vessel (i.e., open and close end cylinder, or sphere) as well as the applied pressure. They also attached with the components of aerospace and marine vehicles such as rocket, balloon skins and submarine hulls. In some cases, pressure vessel are extremely pressurized beyond certain limits without causing bursting of the vessel.

1.1 Classification of Pressure Vessel

Based on Shape Theoretically, pressure vessels are often nearly any form, however shapes product of sections of spheres, cylinders, and cones are sometimes utilized. A typical style could be a cylinder with finish caps known as heads. Head shapes are often either dish-shaped (torispherical) or hemispherical. Additional difficult shapes have traditionally been a lot of tougher to research for safe operation and are sometimes much more troublesome to construct.

Pressure vessel are classified into type based on the shape

1. Cylindrical Pressure Vessel.
2. Spherical Pressure Vessel.

1.1.1 Cylindrical Pressure Vessel

Cylinders area are wide used for storage to their being more cost-effective to supply than spheres. However, cylinders are weak compared to spheres because of weak point at their end. The weak points is decreased by hemispherical or rounded ends are mounted. If the whole cylinder is manufactured from thicker material than a comparable spherical vessel of similar capacity, storage pressure can be similar to that of a sphere.

A sphere, theoretically would be the best shape of a pressure vessel. Unhappily, a spherical shape is tough to manufacture, therefore more expensive, so most pressure vessels are cylindrical with 2:1 semi-elliptical heads or end caps on each end. Smaller pressure vessels are assembled from a pipe and two covers. A disadvantage of these vessels is that greater breadths are more expensive. Consider a cylindrical pressure with wall thickness “t’ and inner radius ‘r’. A gauge pressure ‘p’ exists within the vessel by the working fluid (gas or liquid). For an element sufficiently removed from the ends of the cylinder and oriented as shown in Fig 1.1 & 1.2, two types of normal stresses are generated: hoop ‘σh’ and axial ‘σa’ that both exhibit tension of the material. Cylindrical Pressure Vessel can be further differentiated based on the installation.

1. Horizontal Pressure Vessel (Refer Fig 1).
2. Vertical Pressure Vessel (Refer Fig 2).

Fig 1: Horizontal Pressure Vessel.

Fig 2: Vertical Pressure Vessel.

1.1.2 Spherical Pressure Vessel

This type of vessel is preferred for storage of high pressure fluids. A sphere is a very strong structure. The even distribution of stresses on the sphere's surfaces, both internally and externally, generally means that there are no weak points. Spheres however, are much more costly to manufacture than cylindrical vessels.

Storage Spheres need ancillary equipment similar to tank storage - e.g. Access manholes, Pressure / Vacuum vent that is set to prevent venting loss from boiling and breathing loss from daily temperature or barometric pressure changes, Access ladders, Earthing points, etc. An advantage of spherical storage vessels is, that they have a smaller surface area per unit volume than any other shape of vessel. This means, that the quantity of heat transferred from warmer surroundings to the liquid in the sphere, will be less than that for cylindrical or rectangular storage vessels. When pressure vessels have walls that are thin in comparison to their radii and length. In the case of thin walled pressure vessels of spherical shape the ratio of radius ‘r’ to wall thickness‘t’ is greater than 10. The theoretical ideal shape for a vessel that resists internal pressure is the sphere. To determine the stresses in a spherical vessel, let us consider the cu section through the sphere on a vertical plane and remove half of the shell and its fluid is considered as a single free body. Tensile stress σ will be acting on this free body in the wall of the vessel and the fluid pressure p. The pressure that acts horizontally against the plane circular area is uniform and gives a resultant pressure force of: $P = p\,\pi\,r^2$

Where,

P is the gage or internal pressure (above the pressure acting in the outside of the vessel).

The stress will be uniform around the circumference and it is uniformly distributed across the thickness t (because the wall is thin).

Fig 3: Spherical Pressure Vessel.

2. LITERATURE SURVEY

Amir Afkar a, et al [1], they published the paper on spherical pressure vessel facing simultaneous thermal and pressure loadings under transient loading along with the numeric analytical method using finite element analysis. They also give a new numerical analytical approach for calculating the transient stress and displacement. They designed and conducted FE analysis of spherical pressure vessel using ANSYS. And the von-Mises yield criterion has been used to determine conclusions and the obtained FE results are compared with analytical results.

Prof. Vishal V, et al [2], they presented paper by carry out a detailed design & analysis of Pressure vessel that are used in boiler and condensers for optimum thickness under temperature and dynamic loadings using Finite element analysis method. The designing work is carried out in modelling software called CATIA and the finite elements analysis is carried out FE Software called Ansys. The loads and design is based on the approach of pressure vessel code provided and approved by ASME codes. They studied the behavior of the pressure vessel for different thickness for service load of 5 bar and suggested the suitable thickness by limiting the maximum shear stress.

Tushar Kanti Acharya [3], made a report, on the thick spherical pressure vessel under the creep loading conditions made up of an isotropic and homogeneous material subjected to internal pressure. Spherical vessels can be used to fluid pressure and hence are mainly used in power plants and in petro-chemical industry. Creep of any pressure vessel is common over its usage as they are subjected to high pressure. The basic work of design engineering is to predict the creep analysis and identifying the behavior of creep over its running period is important. The considered strains energy are assumed to be higher which demands us to use strain theory for evaluating the mechanical stresses and strain rates along with the creep strains. They have used the general theory basically developed by N.S. Bhatnagar and V.K. Aryaduring the year 1973 using Norton's law of creep along with this the infinitesimal strain as discussed by Finnie and Heller in the year 1959 is also considered for comparison. To achieve this, FEM approach has been used by using the ANSYS software. The available analysis and numerical method will help the designers in the prediction of correct results (i.e. stress, strain rates and creep strains).

Josip Sertic, et al [4], he along with his fellow colleague published a paper on spherical pressure vessel using analytical equations for calculating the meridian and circumferential forces of a pressure vessel filled with 80% of liquid propylene and 20% empty tank. This analysis results provided as the foundation for the equivalent stress distribution calculation in the pressure vessel and the graphs are plotted for the upper and lower part of the pressure vessel. Analytical results have been closely proved by finite element analyses which are available in the market. The results comparison showed an excellent correlation between the two and concluded that the calculated equivalent stress field by finite elements method could help by assessing critical stress state in joints between the pressure vessel body and its legs.

OludeleAdeyefa, et al [5], this work expands on a prior work done which utilized global coordinate, where countless were expected to shape a union of results for shop fabricated round weight vessels. In this work zone directions were utilized. Any activity that prompts a powerlessness with respect to a structure to work as proposed is known as failure. This exploration, accordingly, examines stresses created in shop manufactured carbon steel circular stockpiling vessels utilizing limited component approach as the analytical tool. 3-D limited component demonstrating utilizing 3-node shallow triangular component with five degrees of opportunity at every node is utilized. These five degrees of opportunity are the key nodal degrees of flexibility without the 6th in-plane turn. The subsequent conditions from limited component examination are coded utilizing FORTRAN 90 computer program. Spherical pressure vessels are subjected to different internal pressures under its service, while its displacements, strains (elastic and plastic) and the respective maximum von-Mises stresses are tracked. The observed maximum von-Mises stresses are further compared with the yield strength for the given shell plate material. Using prescribed factor of safety, within range internal pressures with the respective wall thicknesses for spherical pressure vessels are determined. The FE analysis carried out can be used to predict maximum induced stresses, strains, and displacement of spherical pressure vessels by using readily available yield criteria such as von-Mises yield criteria and also the obtained results obtained are validated for analytical method and noticed that there was no significant difference with values that are obtained through the analytical method.

K.S.J.Prakash, et al [6], A pressure vessel is a kind of holder which is utilized to store fluids or gasses under a weight not quite the same as the encompassing weight. Distinctive states of pressure vessels exist yet most by and large tube shaped and spherical shapes are utilized. Circular vessels are hypothetically 2 times more grounded than barrel shaped ones however because of the assembling challenges, tube shaped ones are by and large favored in the business. For the most part the weight vessels are flimsy walled yet here we are producing multi layered divider

3. METHODOLOGY AND OBJECTIVE

3.1 Methodology

To complete the project successfully we have to fallow a suitable methodology. Once such methodology which is carried out in this project is as shown below fig 3.1.

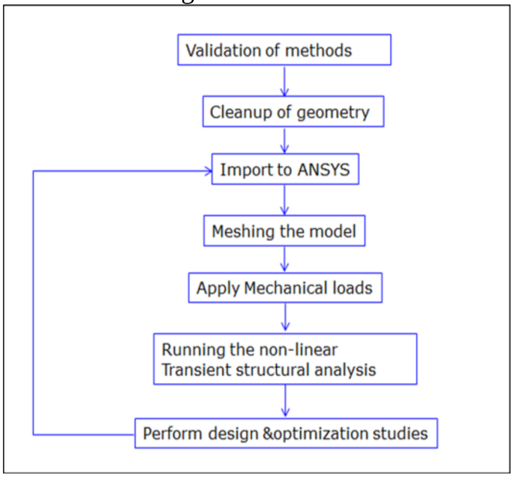


Fig 4: Project Methodology.

3.2 Objective

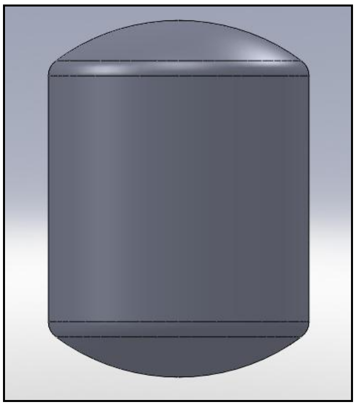
The main objective of this work is as fallows,

- To validate the paper and adopt the same boundary conditions and methodology from it.
- To find the burst pressure for the current design.
- Finally, to carry out new design iterations with different geometry optimization.

4. RESULTS AND DISCUSSIONS

4.1 Geometry

As the geometry and boundary conditions (internal pressure) are symmetry about the axis, we can consider the axisymmetric model and this helps in not only reducing meshing time but also faster computational time. The geometry that is considered for FE analysis is as shown in the below fig.



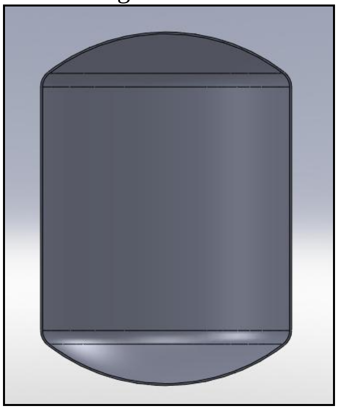


Fig 5: 60ltrs Front View. Fig 6: Section View.

The geometry required for validation by FE analysis is created with the same major dimensions as specified in the experimental results that is the geometry having inner diameter (d)with 310mm and thickness(t) of the wall as 2.5mm and the young's modulus material used is about E=104GPa.

Fig 7: Axisymmetric Geometry.

4.2 Boundary Condition

Pressure vessel with uniform internal pressure is symmetric about the axis. Hence, it be considered as axisymmetric model. The Fig 7 shows the boundary conditions imposed on the model in FE Analysis using symmetric boundary conditions. Uniform internal pressure is applied over the period of time which is possible by using transient conditions where the time is increased constantly along with uniform working pressure. Initial we have started the analysis by imposing initial pressure of 1bar.

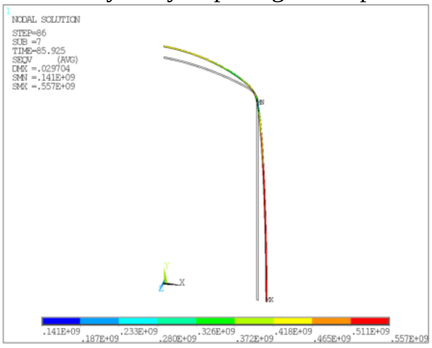


Fig 8: Maximum stress Plot at 85.925bar.

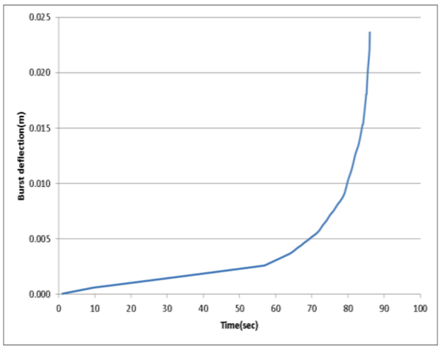


Fig 9: Rad Displacement vs. Time.

From the above fig, it is clearly noticed that the maximum pressure that can withstand by the cylinder is below 86bar after. The cylinder is failing due to plastic deformation at 86sec. We can also observe the maximum von-Mises stress induced in the model is 565Mpa for 85.925bar pressure and is noticed at the middle of the cylinder. Above graph gives the radial deformation over time. It can be observed that elastic behavior can be seen up to 60bar. After 60bar, huge increase in deformation can be noted for every 1bar increase in pressure because of the plasticity behavior. Hence middle of the cylinder location is going fail the component at 85.925bar.

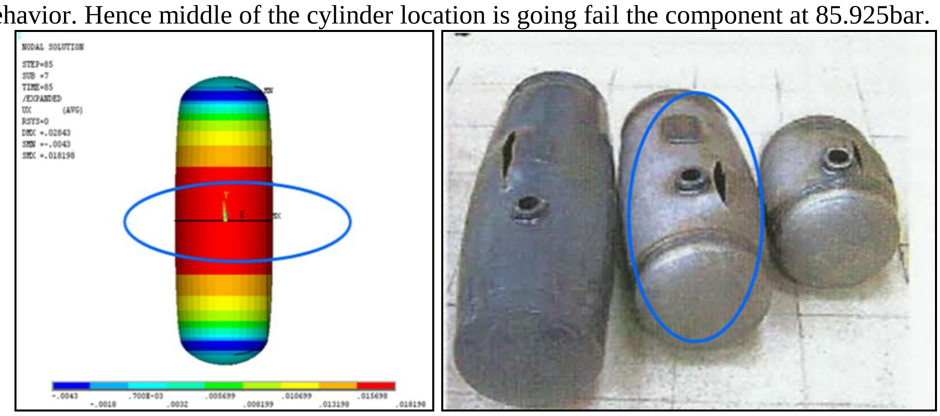


Fig 10: Failure location of Experiment results with FEA results.

Tank Capacity	Nominal thickness(mm)	Burst pressure(bar)		%error
		Experiment	Simulation	
60lt	2.5	85.2	85.925	0.8

Fig 11: FEA and Experimental Results Comparison for 60lts Cylinder.

This chapter deliberates about the design iterations performed. The fundamental idea is to extend the performance of the vessel with a design modification. It is explicitly concluded from the previous chapters that the FEA methodologies are correct so, similar methodology and boundary conditions are used for these iterations as well. Three iterations are performed as part of optimization study of vessel.

- Iteration 01 with Double Wall Thickness.
- Iteration 02 vessel with Radial Stiffeners.
- Iteration 03 vessel with circumferential stiffeners.

4.3 Iteration-02:- Vessel with Radial Stiffeners

In this iteration the thickness of the vessel is kept constant & three stiffeners are added to the vessel. Fig 12 shows the stiffeners which is highlighted in a different color. Fig 14 gives the finite element model of this geometry, 5 divisions are maintained over this stiffener to capture the stress & deformation effects accurately.

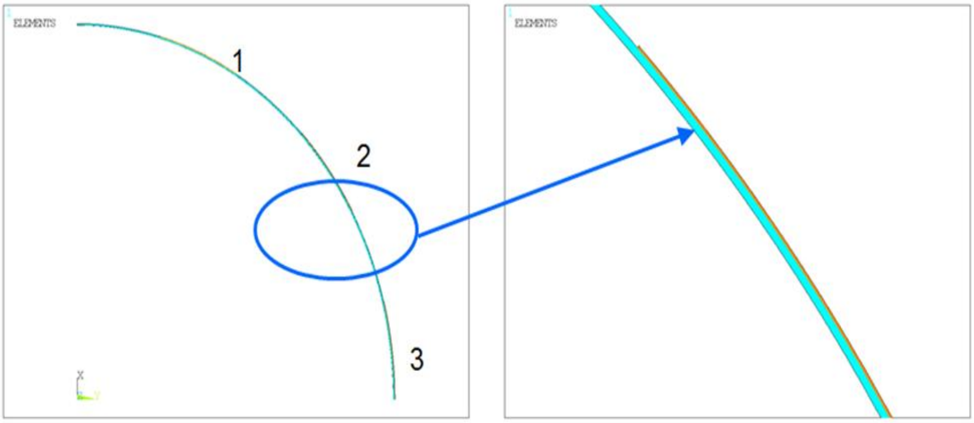


Fig 12: Geometry with Radial Stiffener. Fig 12: Zoom in View of Stiffener.

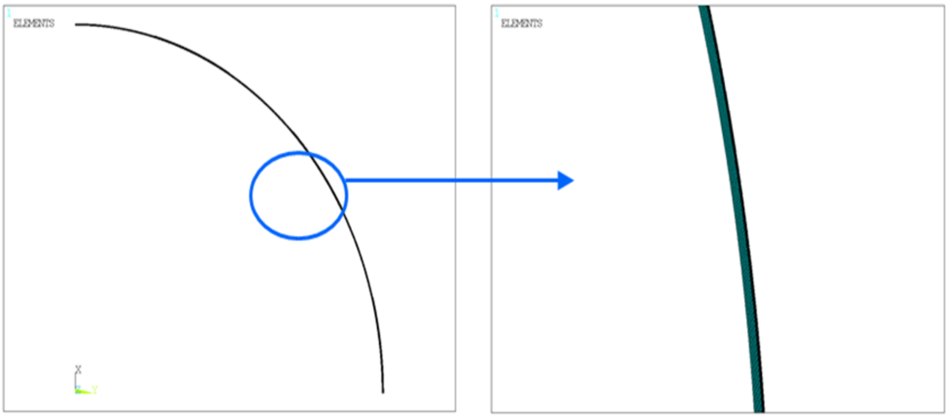


Fig 16: FEA Model of Geometry with Radial Stiffener.

The objective of this study is estimate the burst pressure of this vessel. At the instant of burst pressure, solution in ANSYS solution will not converge. But, solution can be captured till the point of burst pressure. For the current design iteration, convergence error in solution occurred at 40sec. Means Burst pressure is 40bar. Refer below error message from obtained from ANSYS results output file.

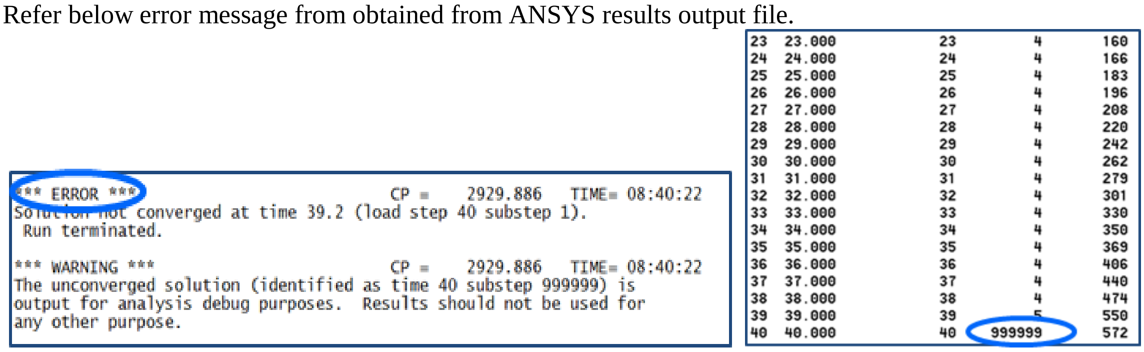


Fig 17: ANSYS Output file.

Structural results at 39bar & 40bar is given in the below fig 18 & 19. It is clear that the solution is diverging for 40bar which is clear indication of burst phenomenon.

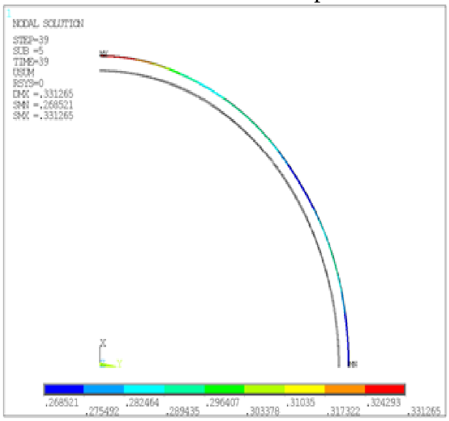


Fig 18: Maximum Displacement plot at 39bar.

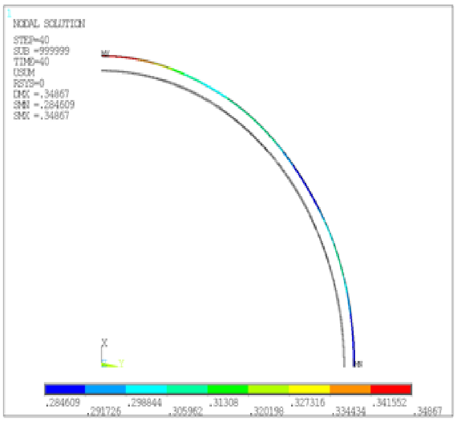


Fig 19: Maximum Displacement plot at 40bar.

At the time of 39sec, for the burst pressure of 39bar, below is the equivalent stress contour. From the plot the Maximum von-Misses stress is found to be 580MPa & is observed besides the rib due to the bending behavior of vessel at the corner of stiffener.

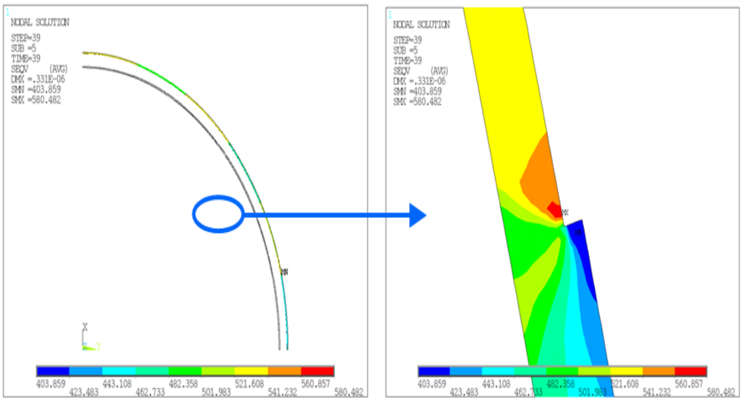


Fig 20: von-Mises Stress plot at 39bar.

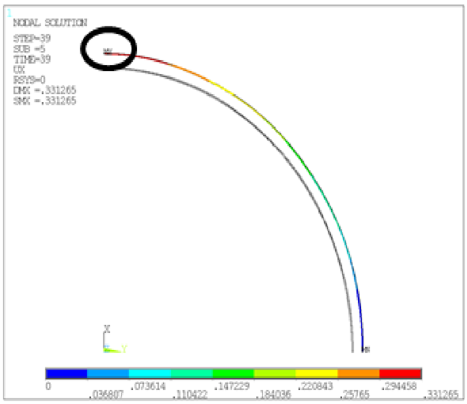


Fig 21: Radial deformation (UX) at 39bar.

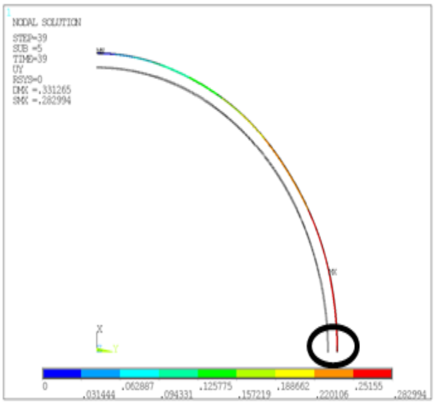


Fig 22: Axial deformation (UY) at 40bar.

From the above FE result, it is seen that max deformation occurs at both ends of the sphere which is sensible. The below graph give the radial & axial deformation over time at the highlighted locations of fig 21 & fig 22. It can be observed that elastic behavior can be seen up to 24bar. After 24bar, huge increase in deformation can be noted for every 1bar increase in pressure because of the plasticity behavior. Highlighted location in the above fig is going to fail the component at 40bar.

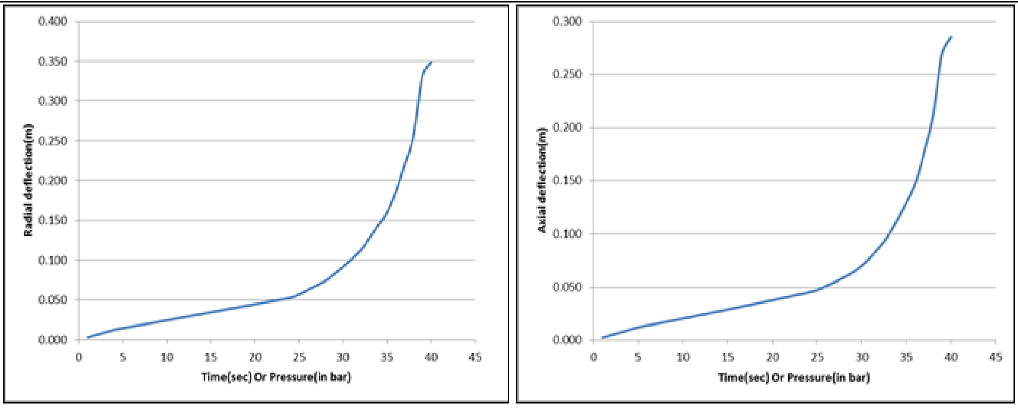


Fig 23: Burst Deflection vs. Time Graph.

5. CONCLUSIONS

From this project with the help of FE Analysis following conclusion can be drawn.

- Given spherical pressure vessel is analyzed for its burst pressure.
- A nonlinear transient structural analysis is necessary for estimating the burst pressure of vessel.
- This analysis method is validated with experimental data.
- Further, it is found that burst pressure of given pressure vessel is 35bar. Both elastic-plastic deformations can be seen from the time vs deformation plot.
- Three different optimization studies are performed to improve the performance of the vessel in the perspective of burst pressure.
- Doubling the thickness has doubled the burst pressure.
- Circumferential stiffeners have very little influence on burst pressure.
- Radial stiffeners have improved the burst pressure by 15%. With less cost, this is the best method to improve the burst pressure characteristics of the vessel.

REFERENCES

1. Amir Afkar, et al., "*Design and analysis of a spherical pressure vessel using finite element method*", World Journal of Modelling and Simulation, Vol-10 (2014), No. 2, pp126-135.
2. Prof. Vishal V, et al., "*Design & Weight Optimization of Pressure Vessel Due to Thickness Using Finite Element Analysis*", IJEERT, Vol-2, Issue 3, June 2014, PP 1-8.
3. Tushar Kanti Acharya, "*Creep Analysis of a Thick Walled Spherical Pressure Vessel Considering Large Strain*", Rensselaer Polytechnic Institute
4. Josip Sertic, et al, "*Stress and Strain Analysis of the Spherical Pressure Vessel: Analytical and Numerical Approach*", Advanced Technologies for Developing Countries, June 28-30, 2006.
5. OludeleAdeyefa, et al, "*Finite Element Modeling of Shop Built Spherical Pressure Vessels*", scientific research, Engineering, 2013, 5, 537-542.
6. OludeleAdeyefa, et al, "*Industrial Spherical pressure vessel design & analysis using FEA*", IJCER, Vol-04, Issue-10, October– 2014.
7. Oludele ADEYEFA, et al, "*Finite Element Modeling of Shell Thickness Requirement Due to Local Loads on Spherical Pressure Vessels*", IJSER, Vol-5, Issue-3, March-2014.
8. Manish M, et al, "*Finite Element Analysis of Elliptical Pressure Vessels*", AJER, Volume-02, Issue-12, pp-343-349.