Finite Element Analysis of Air Receiver Using ANSYS

Manish Kumar Gupta[1] Man Mohan[2]
[1,2]Christian College of Engineering and Technology, Bhilai, Chhattisgarh, India

Abstract— This paper includes finite element analysis of air receiver of 1.0 m3 capacity. Air receiver is used as a storage reservoir for compressed air. It is a type of pressure vessel which is subjected to high internal pressure. In the design of pressure vessel, safety is primary consideration due to potential impact of possible accidents. The literature review has indicated the growing interest in the different types of analysis of pressure vessel and its design. Hence, this research work focuses on finite element analysis of air receiver using Finite element analysis software ANSYS to ensure safety of equipment.

Keywords: Pressure Vessel, ASME Code, ANSYS, CREO

I. Introduction

The Finite Element Analysis (FEA) is a numerical method for solving problem of engineering and mathematical physics. It is useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained. Almost every industrial plant has some type of compressed air system. One of the most important components of compression system is air receiver which is used to store compressed air. The main function of air receiver is to equalize the pressure variation for the start and stop of system. Compressor should not run continuously. Air receiver stores and delivers air when the compressor is not running. Compressed air commonly store in a tank or pressure vessel (air receiver). Pressure vessels indeed are containers for the storage of compressible fluids. Such vessels can be extremely dangerous if not used properly because of the abundant stored energy created when fluids are compressed. A sudden release of this energy may have catastrophic explosive consequences.. Internal pressure failure can be understood as a vessel failing after stresses in part or a large portion exceeds the materials strength. Failure in Pressure vessel occurs due to improper selection of material, defects in material, incorrect design data, design method, shop testing, improper or insufficient fabrication process including welding. Fig 1 shows internal pressure failure.

Considering the safety of operation of pressure vessel, pressure vessels must be designed with great care and full compliance with the applicable codes and legislation.

Pressure vessels are subjected to various types of loadings which exert different types of stress intensities in the vessel components. So it becomes necessary to determine stresses in pressure vessel subjected to internal pressure. For that purpose

Finite element analysis of air receiver is carried out using ANSYS software. Modeling of air receiver is completed in CREO modeling software.

Fig. 1: Internal Pressure Failure

There are many engineering standards which give information on the design, construction and fittings of an air receiver like BS 470, BS EN 286, PD 5500, ASME etc;. ASME Sec. VIII div.1 is most widely used code for design & construction of pressure vessel. In this research work pressure vessel i.e.; air receiver is designed using ASME ‘Boilers and Pressure Vessel Code’. Air receiver consists of mainly cylindrical shell, heads for closure on both ends of shell, nozzles & reinforcements. Theoretical calculations for determination of stresses in shell & head are also performed with reference to ASME code.

II. Literature review

Eui so kim et al [1] researched on “Risk analysis of CNG composite pressure vessel via computer aided method and fractography”. Here, Cause or pressure vessel is investigated through formal inspection and engineering test procedure. After experiments results shows that Visual inspection does not advert the possibility of catastrophic failure. So it is necessary to develop monitoring system to check structural health of vessel. Bandarupalli Praneeth et al [2] carried out research on finite element analysis of pressure vessel and piping design. The stresses developed in solid layer pressure vessel and multilayer pressure vessel are analysed. Here, theoretical and ANSYS results are compared. Finally they conclude that theoretical calculated values are very close to that of the values obtained from ANSYS is suitable for multilayer pressure vessels. Multilayer pressure vessels are superior than solid layer pressure vessel. David Heckman [3] tested three dimensional, symmetric and axisymmetric models; the preliminary conclusion is that finite element analysis is an extremely powerful tool when employed correctly. Depending on the desired solutions, there are different methods that offers faster run times and less error. The two recommended methods included symmetric models using shell elements and axisymmetric models using solid elements. Yogesh Borse and Avadesh K. Sharma [4] present the finite element modeling and Analysis of Pressure vessels with different end connections i.e. Hemispherical, Ellipsoidal & Toro spherical. They describes its basic structure, stress characteristics and the engineering finite element modeling for analyzing, testing and validation of pressure vessels under high stress zones. Their results with the used loads and boundary conditions which remain same for all the analysis with different end connections shows that the end connection with hemispherical shape results in the least stresses when compared to other models not only at weld zone but also at the far end of the end-connection. V.N. Skopinsky [5] presented work on structural modeling and stress analysis of nozzle connections in ellipsoidal heads subjected to external loadings. Timoshenko shell theory and the finite element method are used. Results showed that it is necessary to pay more attention to the effective stresses in the shells in these loading cases. Drazan, Pejo, Franjo and Darko [6] considered influence of stresses resulting from weld misalignment in cylindrical shell circumferential weld joint on the shell integrity .The stresses estimated analytically by API recommended practice procedure and calculated numerically by using the finite element method

III. MODELING OF AIR RECEIVER

Geometry of air receiver is modeled in modeling software CATIA-V5. In ANSYS there is a provision for importing geometries from any modeling software. File format of geometry which is to be imported should be in IGES OR STEP format only. Therefore, geometry of air receiver is saved in IGES format in CATIA-V5. Then it is imported to design modeler of ANSYS Workbench 15.0.
Main Parts of air receiver are shown below:

1) Cylindrical shell

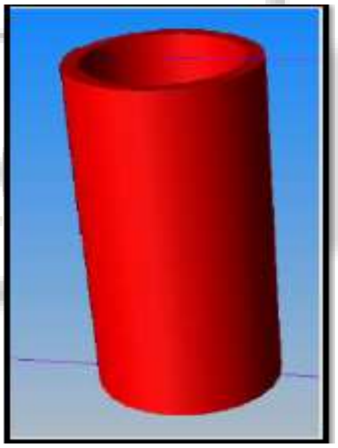

Fig. 2: Cylindrical Shell of Air Receiver

Dimensions:
1) Outer diameter (Do) = 870 mm
2) Inner diameter (Di) = 850 mm
3) Provided thickness of shell (t) = 10 mm Material: SA 36
4) Allowable stress (S) = 114.5 Mpa

2) 2:1 Ellipsoidal Head

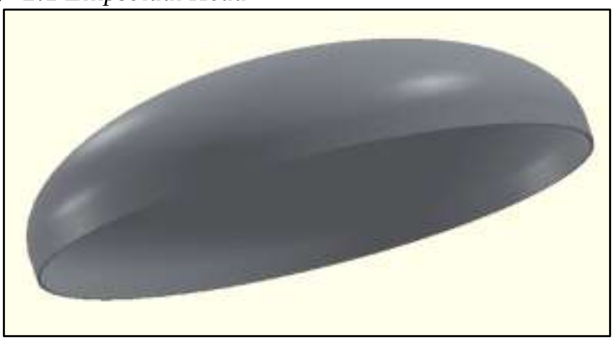

Fig. 3: Model of 2:1 Ellipsoidal head

Dimensions:
1) Inside diameter of the head skirt (D) = Shell I.D. + 2(C.A.) = 850+2 = 852 mm
2) Provided thickness of head after forming (t) = 10 mm
3) One-half of the length of the minor axis of the ellipsoidal head (h) = D/4 = 213 mm

Material: SA 36,
Allowable stress (S) = 114.5 Mpa,

3) Geometry of Air receiver:

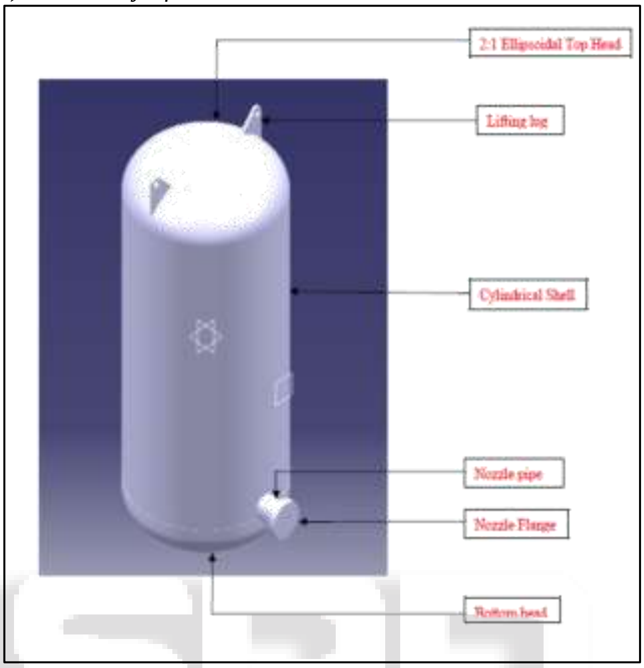

Fig. 4: Components & model of Air Receiver

IV. FINITE ELEMENT ANALYSIS OF AIR RECEIVER

1) Complete model of air receiver is shown in fig. no. 4.

2) Element Type Used:

Geometry created in ANSYS by joining nodes. As geometry created, element has to be defined. Solid 187 is used as a element in analysis.

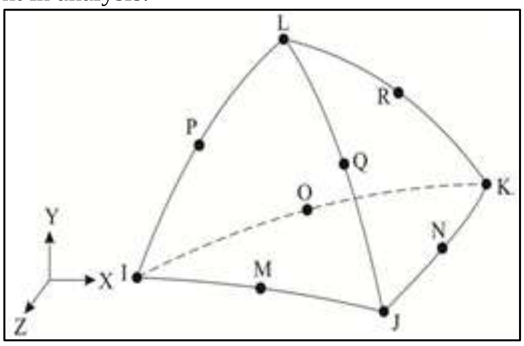

Fig. 5: Solid 187 Geometry

3) Meshing:

Meshing is an integral part of the computer-aided engineering simulation process. The mesh influences the accuracy, convergence and speed of the solution. Meshing is nothing but discretization of model into number of smaller units known as elements. In this analysis hex-dominant method is used for meshing. Fine meshing is used. Fine meshing gives better results than course or medium size meshing. The points at which elements are connected to each other are known as nodes. Element type used is Solid

187. It is a 2nd order 3-D 20-node solid element. Meshing is done using tetra mesh having element size 20 mm. Meshing of geometry is shown in fig 6.

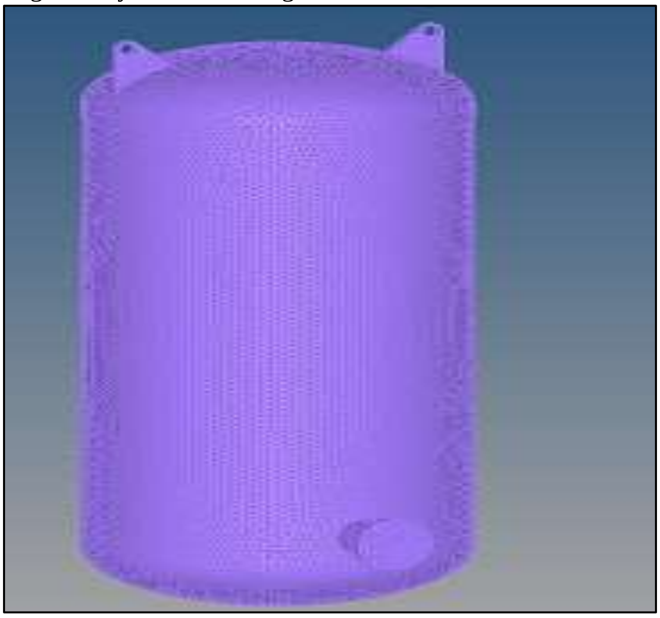

Fig. 6: Air receiver meshing in ANSYS Workbench 15.0

4) Loading:

Shell is subjected to internal pressure of 18.5 kg/cm2. Therefore, pressure of 18.5 kg/cm2 is applied to interior of shell.

Results of von mises stresses after applying load are discussed in results and discussions.

V. Results & Discussions

1) Complete Pressure Vessel is analyzed in a Finite Element Software ANSYS Workbench 15.0.

Finite element result after applying load is shown below:

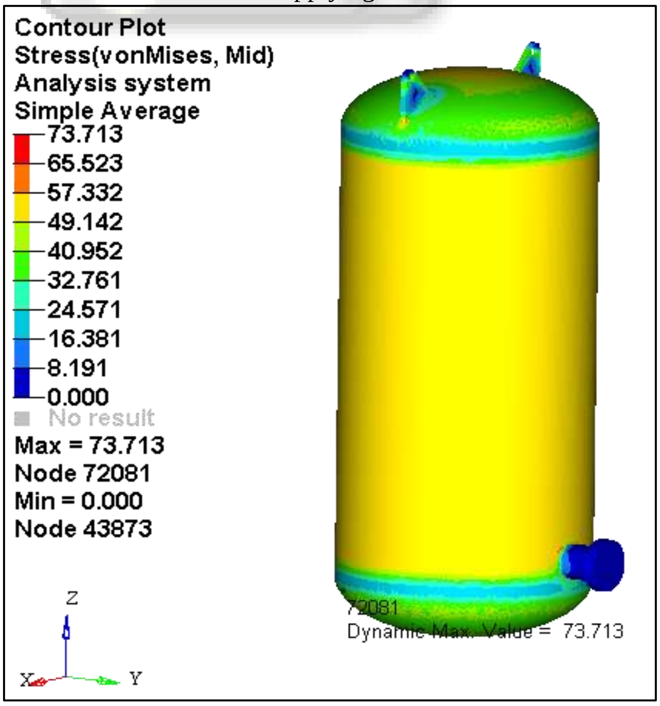

Fig. 6: Von mises stress result

From above figure it can be seen that maximum stress generated in vessel is 73 Mpa. And allowable stress for SA 36 material as per ASME code Sec II Part D is 114.5 Mpa. Therefore, stress values obtained after FEA are lesser than allowable limits of material and hence pressure vessel design is safe. Therefore this design can be used to manufacture the pressure vessel.

2) Validation of FEA Results:

In structural analysis of ANSYS maximum stress of 73 Mpa is generated at the centre of head. Hence stresses in head due to internal pressure are calculated analytically by taking reference from ASME code.

Stresses in head due to internal pressure:

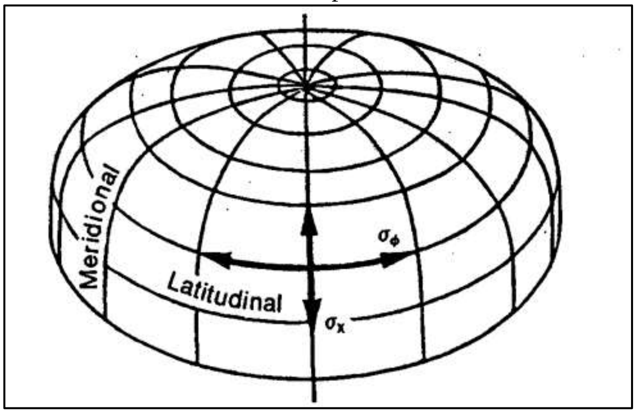

Fig. 7: Elliptical head

$\boldsymbol{\sigma_x} \rightarrow$ **Meridonial Stress**

$\boldsymbol{\sigma_\emptyset} \rightarrow$ **Latitudinal Stress**

At center of head,

$$\boldsymbol{\sigma_x} = \frac{\mathbf{P \times R^2}}{\mathbf{2 \times t \times h}}$$

$$\boldsymbol{\sigma_x} = \boldsymbol{\sigma_\emptyset}$$

Where,

R is inside radius of head = 425 mm,

h is depth of head = 263 mm, and

t is thickness of head = 10 mm.

After analytical calculations of stress generated in head due to internal pressure, it is found that maximum stress generated in head is 62.29 Mpa. Results of FE analysis are within 14.67% error limit when compared to analytical results. Percentage error in FEA results and analytical results occurs because of a variety of reasons such as assumptions in analytical calculation, approximations in FE formulations, selection of element in FE analysis, selection of mesh type etc.

VI. Conclusion

This study has demonstrated the usefulness using general finite element software in conducting finite element simulation and analysis for the design of a air receiver. A Finite element analysis was performed to determine maximum stress in air receiver exposed to internal pressure.

Overall conclusions based on present study are as below:

1) Pressure vessel is designed and analyzed for the given internal pressure.
2) Maximum stress induced due to pressure alone in air receiver is calculated theoretically using ASME code

formula and compared with the analysis values. The maximum percentage error after comparison is 14.67%.

3) Safe operating conditions for the vessel are verified with the help of FEA advanced techniques and ASME codes & standards.

REFERENCES

[1] Eui so kim, suing choi, "Risk analysis of CNG composite pressure vessel via computer aided method and fractography", Engineering Failure Analysis 27 (2013).

[2] Bandarupalli Praneeth, T.B.S.Rao, "Finite Element Analysis ofPressure Vessel and Piping Design", International Journal of Engineering Trends and Technology- Volume3Issue5- 2012.

[3] David Heckman "Finite Element Analysis of Pressure Vessels",MBARI -1998,pp(1-7)

[4] Yougesh Borse and Avadesh Sharma "Modeling of Pressure Vessels with Different End connections using Pro Mechanica", IJERA - 2012, pp(1493-1497)

[5] 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, 2006, vol.11, No.4, PP-965-979.

[6] DrazanKozak, FranjoMatejicek, DarkoDamjanovic, "Weld misalignment influence on structural integrity of Cylindrical Pressure Vessel", Structural integrity and life, Vol. 10, No 2 ,2010, Page no. 153-159.

[7] M. A. Porter, D. H. Martens, and C. S. Hsieh, "A comparison of finite element codes and recommended investigation methodology," ASME, vol. 359, pp. 241-246, 1997.

[8] M. Amin and S. Ahmed, "Finite element analysis of pressure vessel with flat metal ribbon wound construction under the effect of changing helical winding angle," Journal of Space Technology, vol. 1, no. 1, pp 34-39, 2011.

[9] A. K. Sohaimi, A. P. P. A. Majeed, and J. Mahmud, "Preliminary numerical analysis of a platform structure," Applied Mechanics and Materials, vol. 680, pp. 280-283, 2014.

[10] M. I. Z Abidin, J. Mahmud, M. J. A. Latif, and A. Jumahat, "Experimental and numerical investigation of SUP12 steel coil spring," Procedia Engineering, vol. 68, pp. 251-257, 2013.

[11] M. R. M. Sahri, C. Muhamad, M. J. A. Latif, and J. Mahmud, "Parametric study of low velocity impact using finite element analysis," Applied Mechanics and Materials, vol. 393, pp. 397-402, 2013.

[12] M. J. A. Latif, N. H. Hashim, R. Ramlan, J. Mahmud, A. Jumahat, and M. R. A. Kadir, "The effects of surface curvature on cartilage behaviour in indentation test: A finite element study," Procedia Engineering, Elsevier, vol. 68, pp. 109-115, 2013.

[13] N. F. A. Manan, M. H. M. Ramli, M. A. Ab Patar, C. Holt, S. Evans, M. Chizari, and J. Mahmud, "Determining hyperelastic parameters of human skin using 2D finite element modelling and simulation," in Proc. Symposium on Humanities, Science and Engineering, 2012, pp. 805-809.

[14] Rules for construction of pressure vessel ASME Sec vii Div 1 2001 Edition

[15] "Pressure vessel design manual'', by Dennis Moss 3rd Edition

[16] Rules for construction of pressure vessel ASME Sec vii Div 1 2001 Edition

