A REVIEW ON DESIGN AND ANALYSIS OF PRESSURE VESSEL

[1] Umbarkar Bhagyashri B. [2] Prof. Hredeya Mishra

[1]*Student, Dept. of Mechanical Engg.,Jaihind college of Engg. & Technology, Maharashtra, India.*
Um.bhagyashri@gmail.com
[2]*Associate Professor,Dept. of Mechanical Engg.,Jaihind college of Engg. & Technology, Maharashtra, India.*
[3]*HOD, Dept. of Mechanical Engg., Jaihind college of Engg. & Technology, Maharashtra, India.*

ABSTRACT

This paper present design and analysis of pressure vessel. Design of pressure vessel depends on its pressure and temperature. When pressure and temperature get changed every pressure vessel is new. In pressure vessel design safety is the main consideration. The structural integrity of mechanical components of pressure vessel requires a fatigue analysis including thermal and stress analysis. Pressure vessel parameter are designed in Pv Elite and checked according to ASME (American society of mechanical engineering) sec. viii Div.1.Fatigue analysis also done on modeled in Pv Elite software to improve the life of pressure vessel. Pv Elite helps engineer to comply their design and calculation strictly as per code.&According to ASME SEC VIII. DIV-2 Analysis of pressure vessel is carried out at different pressure and temperature conditions.

Keywords-*Pressure Vessel, Fatigue, Stress Concentration Factor, Fatigue Curve, Cumulative Usage Factor***.**

1. INTRODUCTION

The term pressure vessel referred to those reservoirs or containers, which are subjected to internal or external pressure. The pressure vessels are used to store fluids under pressure. The fluid being stored may undergo a change of state inside vessels as in case of steam boilers or it may combine with other reagents as in chemical plants. High pressure is developed in pressure vessel so pressure vessel has to withstand several forces developed due to internal pressure, so selection of pressure vessel is most critical. ASME Sec.VIII div.1 is most widely used code for design & construction of pressure vessel.Div.1 does not consider harmonic analysis. Div.1 consider biaxial state of stress combined in accordance with maximum stress theory. When pressure of operating fluid increases, increase in thickness of vessel. This increase in thickness beyond a certain value possess fabrication difficulties and stronger material for vessel construction. The material of pressure vessel may be brittle such as cast iron or ductile such as mild steel. 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. To obtain safety of pressure vessel and to design Pressure vessel the selection of code is important. Corrosion allowance is the main consideration in vessel design. Corrosion occurring over the life of the vessel. During service, pressure vessel may be subjected to cyclic or repeated stresses. Fatigue in pressure vessel occurs due to:

a) Fluctuation of pressure
b) Temperature transients,
c) Restriction of expansion or contraction during normal temperature variations,
d) Forced vibrations,
e) Variation in external load

Fig.1. The typical horizontal storage vessel design

2. LITERATURE REVIEW

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. Contact elements were tested to determine their usefulness in modeling the interaction between pressure vessel cylinder walls and end caps.

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.

A. J. Dureli (1973) presented work on the stresses concentration in a ribbed cylindrical shell with a reinforced circular hole subjected to internal pressure, by several experimental methods and the results obtained were compared with those corresponding to a non-reinforced hole in a ribbed and un-ribbed shell and also to a reinforced hole in an un-ribbed shell. From the result it was found that the maximum value of hoop stress, and longitudinal stress, in shells always occurred at the points $\theta = 0°$ and $\theta = 90°$, respectively, along the edge of the hole, θ being the angle measured clockwise from the longitudinal axis of the hole R.

C. Gwaltney (1973) compared theoretical and experimental stresses for spherical shells having single non-radial nozzles. The stress distributions for radial and non-radial nozzle geometry are analyzed. Stress distributions for the non-radial and the radial nozzle attachments are quite similar but the non-radial nozzle configuration gave the maximum normalized stress, both theoretical and experimental, for internal pressure and for axial loads on the nozzleas well as for pure bending moment loading in the plane of obliquity.

M.A. Guerrer, C. Betego´n, J. Belzunce [5] A finite element analysis (FEM) was used to calculate the behavior of a pressure vessel (PV) made of high strength steel (P500) subject to the design loads and assuming the existence of the ‘‘worst case’’ crack allowed by the European standards in order to demonstrate the safe use of these steels and the too conservative design rules currently applied by the PV manufacture codes. analysis was checked by the simulation of a Wide Plate Test. A good agreement was obtained with the experimental values determined using strain gauges and with the analytical KI expression available for this specific geometry. It was demonstrated that the presence of cracks on pressure vessels made of P500 high strength steel non detected during non-destructive tests, do not endanger the safety of the vessel, from the fracture mechanics point of view, since the maximum values of the stress intensity factor along the crack tip is always much lower than the room temperature fracture toughness of the material (coarse grain heat affected zone). That is why, although high strength P500 steel is excluded by EN 13445 Part 2, Annex B for the manufacture of pressure vessels, because it has a yield strength higher than 460MPa, its application can be fully successful and safe even under the worst allowed conditions, given way to significant reductions of wall thicknesses, weights and costs.

3. PROBLEM STATEMENT

3.1. Mechanical design for air receiver as per ASME Sec.VIII div.1

Air receiver is considered as a pressure vessel. Inthis 2000 liter Air Receiver Vessel is to be designed as per ASME secViii, Div-1.

Table.1.list of code

Sr no.	ASME code	Description
1	ASME SEC II	Material specification
2	ASME SEC V	Nondestructive examination
3	ASME SEC VII Div.1	Rules for construction of pressure vessel

Table.2.list of material of construction

Sr no.	Item	Moc
1	Shell	SA-516 Gr .70
2	Head	SA-516 Gr .70
3	RF Pad/ Pad plates	SA-516 Gr .70
4	Nozzle Neck.	SA-105
5	Base Plate, web Plate, Rib plate	SA-36

1. Shell Thickness Calculation

tr: = (P*R)/(S*E-0.6*P) per UG-27 (c)(1)

= (3.846*631.0000)/ (137.90*1.00-0.6*3.846)

= 17.9001 + 1.0000 = 18.9001 mm

Nominal Thickness:- 20 mm.

2.Dish end Thickness Calculation

Required Thickness due to Internal Pressure

tr: = (P*D*Kcor)/ (2*S*E-0.2*P) Appendix 1-4(c)

= (3.846*1262.0000*0.998)/ (2*137.90*1.00-0.2*3.846)

= 17.6125 + 1.0000 = 18.6125 mm.

Nominal Thickness:- 20 mm.

Table .3Material Properties for Analysis

Material	Design temperature (o C)	Elastic Modulus (MPa)
SA 516 Gr 70	75	$199.33 * 10^3$
SA 105	75	$198.33 * 10^3$
SA 36	75	$199.33 * 10^3$

3.2 Analysis of pressure vessel by Pv Elite software

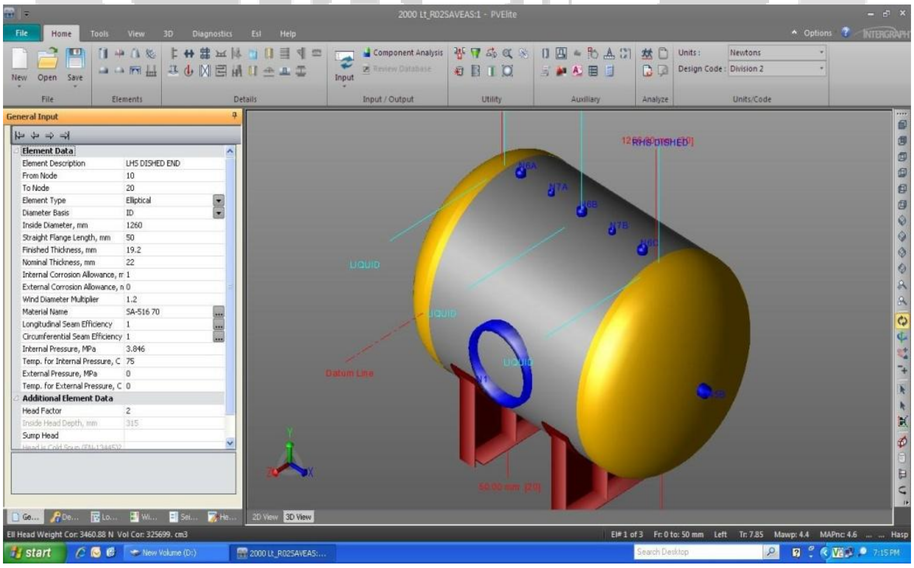

Fig.2. Vessel Geometry in PV Elite.

Table. 4Design Data

Relevant Code for Analysis	ASME, Sec.VIII, Div.2 Ed.2013
Design Pressure	3.846 MPa
Operating Pressure	3.5 MPa
Corrosion Allowance	1 mm
Design no of cycles for shutdown case (1.5-3.5 MPa)	< 50000 Cycles

Table
.5Pressure range details for no. of cycles

Case	Pressure 1	Pressure 2	Range	Number of cycles
1	15.00	35.00	20.00	50000.00

After running the analysis, it was observed that nozzle N6-C is subjected to maximum stress with No. of cycles without fatigue failure less than that as compared to other Nozzles i.e. N1, N2, N3, N4, N5, N6-A, N6-B, N6C-1”

ITEM: Main Component: SHELL Nozzle: N6C-1"
Nozzle installed in: A Cylindrical Shell

Input Values: Pressure in bars
Table .6 stress ranges

Longitudinal Plane			Transverse Plane	
Stress	Inside corner	Outside corner	Inside corner	Outside corner
sn	3.1000	1.2000	1.0000	2.1000
st	-.2000	1.0000	-.2000	2.6000
sr	-.0301	0.0000	-.0301	0.0000
s	3.3000	1.2000	1.2000	2.6000

Calculation for the First Pressure Range:
Compute Primary Membrane Stress [S]:
= P / (E * ln((2 * t + D) /(D)))
= 20.000/ (1.00*ln ((2*19.000+1262.000)/ (1262.000)))
= 67.4200 N./mm²
Sample calculation for the Intensified Stress Amplitude [Sa]:
= S * 3.3 / 2
= 67.420 * 3.3/2
= 111.2430 N./mm²
Stress Factor used to compute X [Y]:
= (Sa/Cus)(Efc/Et)
= (16.1/1)(28300000/28952368)
= 15.7703 ksi

$[X]:=(C1+C3*Y+C5*Y^2+C7*Y^3+C9*Y^4+C11*Y^5)/(1+C2*Y+C4*Y^2+C6*Y^3+C8*Y^4+C10*Y^5)$
= 5.4191

C Factors used in the above equation:

Table .7 values of factor C

C1 = 2.25451	C2= -464224	C3= -.831275	C4= -.0.863466E-01
C5= 0.202083	C6= -.694053E-02	C7= -.207973E-01	C8= -.0.201024E-03
C9=0.713772E-03	C10= -.0.00000	C11= - .0.00000	

From the table, EFc = 195128 N./mm²
Compute the Number of Cycles from Equation 3.F.1 [N]:
= 10^X
= $10^{5.419}$
= 262492 Cycles
Case 1 Peak Stress: Adjusted below per above Pressure Index

Table.8 Peak stresses

Longitudinal Plane			Transverse Plane	
Stress	Inside corner	Outside corner	Inside corner	Outside corner
Sn 33.710	104.501	40.452	33.710	70.791
St 33.710	-6.742	33.710	-6.742	87.646
Sr 33.710	-1.015	0.000	-1.015	0.000
Sint 33.710	111.243	40.452	40.452	87.646

Table 9. Result of N6C

Sr no.	Stress intensities	N cycles	Nmax cycles	Damage factor
1	111.243	50000	0.2625E+06	0.190

Total: Damage Factor:0.190

Fatigue Analysis Passed: Damage Factor < 1.00

Hence, Design is safe for pressure cycle 1.5 MPa to 3.5 MPa for designed 50000 number of Cycles.

1) Fatigue Analysis is said to be passed since Damage Factor <1.00

4. FINITE ELEMENT ANALYSIS OF PRESSURE VESSELS

Because of complicated shape of shell stress analysis by using photo –elasticity will also be difficult. Stress Analysis by finite element method is obviously best choice. Hence a finite element technique has been selected for analysis purpose. There are different types of commercial FEM software's available in market. ANSYS FEM software is one of the most popular commercial software is used for finite element analysis of vessel. The objective of analysis was to check fatigue life of 2000Ltr Air Receiver for Required Thickness due to Internal Pressure cyclic pressure service and impact loading service in accordance with ASME Section analysis is carried out. The study is conducted to determine the stress levels in the 20000 ltr. Air receiver to a sufficient level of accuracy. Hence the study is conducted using the following methodology.3D Model of 2000 ltr. Air receiver is created using pro-e.

Hence the Nominal Thickness:- 20 mm

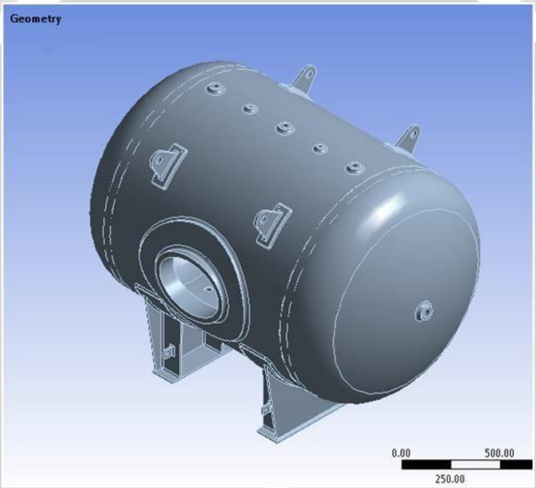

Fig. 3. Model of Air Receiver.

To achieve accuracy within satisfactory level, convergence study is conducted for 3.5MPa pressure case. Model is analyzed for variety of element sizes and a size is chosen wherein satisfactory accuracy is obtained having less computation time.

Model is analyzed for Cyclic Pressure service- 1.5 Mpa to 3.5 Mpa.

Design no of Cycles = 5000 nos.

The 3D geometry is meshed using Solid 187 having element size of 25mm. Total numbers of elements are 243916.

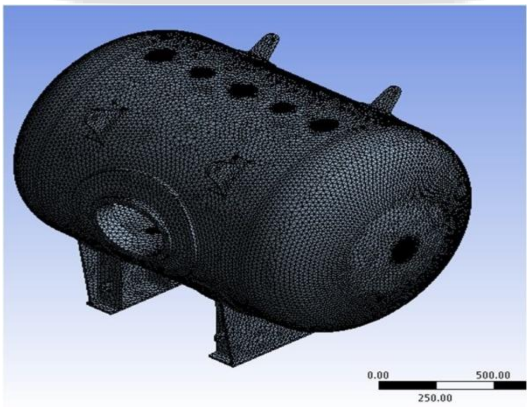

Fig.-4 Meshing of Equipment

5. CONCLUSION

1. Fatigue analysis will be carried out for entire equipment for specified regeneration cycles and we will found fatigue life more than required cycles.
2. Accordingly we conclude that all evaluation points for fatigue are within allowable limits specified by code. The maximum fatigue damage fraction observed which less than unity as required by code.

6. REFERENCES

1) ASME sec VIII, Div-1&2.Pressure vessel design code.
2) D. Moss pressure vessel design handbook.
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) ASME Sec II D-2010