IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 10, December 2014.
www.ijiset.com
ISSN 2348 – 7968

Design, Analysis and Experimental Verification of Torispherical Head and Toriconical Bottom Pressure Vessel

Dr. M. M. Patil[1], Dr. Lokesh Bajpai[2]

[1]Mechanical Engineering Department, J. T. Mahajan College of Engineering Faizpur, Maharashtra State, 425524, India,

[2] Mechanical Engineering Department, SATI Vidisha, Madya Pradesh State, India,

Abstract

The significance of the title of this paper to highlight designing structure of the pressure vessel for static loading and its assessment using ANSYS and Experimental setup. This work is concerned with design of different pressure vessel elements such as shell, torispherical head, toriconical bottom, operating nozzle, its reinforcement, on standards and codes and evolution of all components analyzed by ANSYS and Experimental setup. This study addresses the problem of pressure vessel using both experimental and Finite Element Analysis (FEA) approach. When pressure vessels are designed using traditional methods they often suffer permanent volume expansion at the bottom end closure and become unstable when they are pressure tested experimentally. In this work, experimental investigations are carried out using hydrostatic pressure tests with water. In the case of numerical investigations, the FEA models are constructed using material SA240 Gr 316. The results obtained from both FEA models and experimental tests are compared which shows close agreement.

Keywords: *FEA, Torispherical Head, Pressure Vessel*

1. Introduction

The design of pressure vessels requires a careful study of many regions, the most critical of which is the knuckle region. Torispherical heads and Toriconical bottom with various knuckle radii (for each head height) have been investigated by researchers to establish an 'optimum' Torispherical head and Toriconical bottom. The effect of axial loads, external pressure and thermal loads on stiffened cylindrical shells has also been investigated in the past. Theoretical and numerical calculation of stress at stress concentration points of both torispherical and ellipsoidal heads have also been investigated in previous researches and a complete method for using reinforcement plates in these points has been addressed in ASME codes. No standard code or procedure has addressed the application of toriconical bottom on heads subjected to either external or internal pressure. On the other hand, reduction of even 1mm in the thickness of a head can sometimes have a great impact on the total costs of a head. For example, in the case of a design recommending the application of a 26mm plate for a head, because of the market limitation, the manufacturer may order a 30mm thickness plate. The application of a stiffening-ring could be an alternative solution for the application of a thinner plate, eg a 25mm plate instead of a 30 mm one.

Fig. 1 Torispherical head and toriconical bottom vessel.

Shell Design

If a thin circular cylinder is subjected to the action of radial forces uniformly distributed along its circumference, hoop stress will be produced throughout its thickness in tangential direction as given by,

$\sigma_H = Pr/t$, $\sigma_L = Pr/2t$

Where P = Internal Pressure, r = Radius of shell, t=Thickness σ_H = Hoop stress, σ_l = Longitudinal stress

Taking fabrication and inspection quality into account, ASME has suggested modified formula for finding thickness for given pressure.

i) Circumferential Stress (Longitudinal Joint)

$t = PR / (SE – 0.6 P)$ or $p = SET / (R + 0.6t)$

Design Data

Operating pressure = 0.45 N/mm2 (shell side)
Design pressure = 0.8 N/mm2
Inside diameter = 1230 mm
Shell material – SA 204 GR.316.

ISSN 2348 – 7968

Permissible stress, S = 85.83 N/mm2
Assume welding efficiency, E = 0.8
F.O.S = 6. (Repeated load applied gradually but not reversed)

Shell Thickness:

For circumferential stress

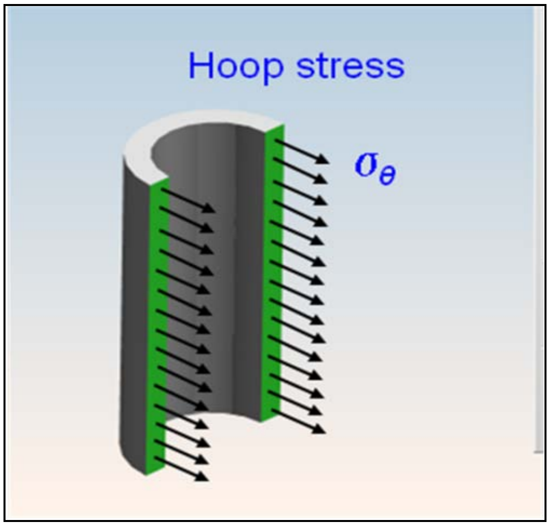

Fig. 2 Hoop stresses in the shell.

$\frac{D0}{Di} > 0.3$

Thickness = $\frac{P*di}{2fj-pi}$ = $\frac{0.8*1230}{2*85.83*0.8-0.8}$
=7.207mm

Thickness of Shell is considered 8 mm.
Height of shell is 2000mm.
Outer diameter of shell is 1246mm.

Design of Torispherical Head (Thickness Calculation)

Material: use SA240 Gr 316, E =1, Allowable Stress, S = 85.83 N/mm^2. Design Pressure = 8 Kgf/cm^2
According to UG 31 of ASME Sec VIII Div 1,

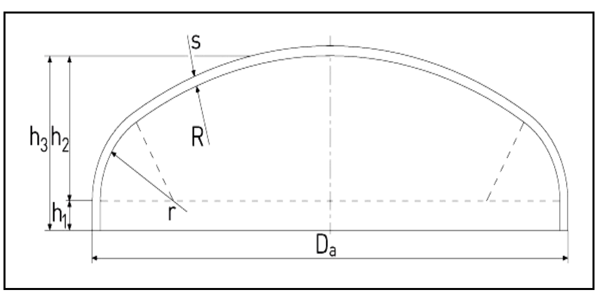

Fig. 3 Torispherical head.

The Stress intensification factor, W, is given

W=$1/4(3 + (\sqrt{\frac{Rc}{Ri}})$

W= $1/4(3 + (\sqrt{(\frac{1230}{123}})$ =1.54

Thickness of Head

$S = \frac{p*Rc*w}{2fj} = \frac{0.8*1230*1.54}{2*85.83*0.8}$ = 11.034mm.

Thickness of Torispherical head is considered 12 mm.
Where
P= Internal pressure
Rc = Crown Radius
Ri= Knuckle Radius
W= Stress intensification factor.
S=Thickness of Head.

H_1= 60mm

$H_2 = Di - \{[D_i - R_i]^2 * -[\left[\frac{D_i}{2} - R_i\right]^2\}^{0.5}$

H_2=$1230 - \{[1230 - 123]^2 - \left[\frac{1230}{2} - 123\right]^2\}^2$

$H_2 = 1230 - [1223449 - 242064]^{0.5}$

$H_2 = 238.34$ mm

Total Height (H) = $H_1 - H_2$
= 60+238.34
=298.34 mm.

Knuckle radius is 10% of Crown radius that is 123mm.

Toriconical Bottom (Thickness Calculation)

Material: use SA240 Gr 316, E =1, Allowable Stress, S = 70 N/mm^2. P = 4.5 Kgf/cm^2.

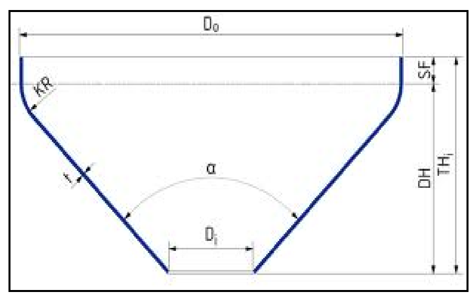

Fig. 4 Toriconical bottom.

Thickness = $\frac{(p*di)}{2*(f-06*p)cos\alpha}$

Thickness = $\frac{0.8*1230}{2*(85.83-0.6*0.8)\,cos\alpha}$

Thickness = $\frac{984}{82.74}$

Thickness =11.89 mm
Thickness of Toriconical bottom is 12mm

2. Finite Element Analysis of Pressure Vessel

Because of the complicated shape of the shell, stress analysis by using photo-elasticity will also be difficult. Stress analysis by finite element method is obviously the best choice. Hence finite element technique has been selected for the analysis purpose. There are different types of commercial FEM software available in the market. ANSYS FEM software is one of the most popular commercial software is used for the Finite element analysis of the pressure vessel. The material used for the construction of pressure vessel is SA240 GR.316 and its properties are as shown in Table 1.

Table 1: Material properties SA240 GR.316

Sr. no	Properties	Values
1	Modules of elasticity	2×10^5 N/mm^2
2	Poison's ratio	0.3
3	Yield stress	205 N/mm^2
4	Tensile stress	505 N/mm^2
5	Density of material	8000 Kg/m^3

Modeling is done on the solid works with the actual industrial design data and during modeling all inlet-outlet is considered for the particular problem, the model was prepared by using solid works graphic software as shown in Figure-5.

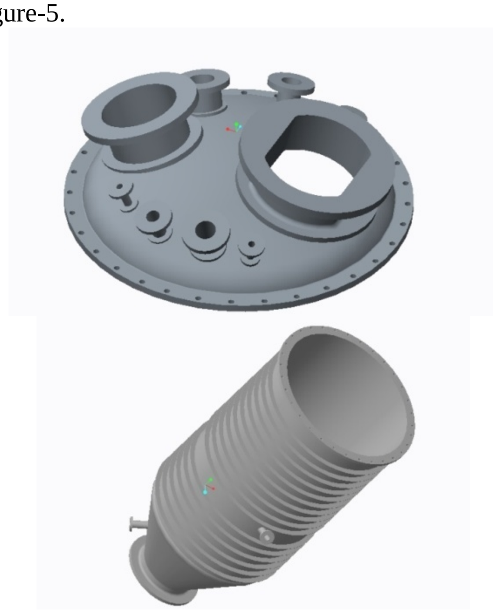

Fig. 5 Geometric model of head with shell and having toriconical bottom.

CASE-I Actual Industrial model analysis on ANSYS:
Analysis of the model is done on based on actual industrial model. In this model shell and head is connected with the flat flange. During the analysis operating pressure and operating condition is considered as per actual industrial model.
Testing Pressure -8 kgf/cm^2

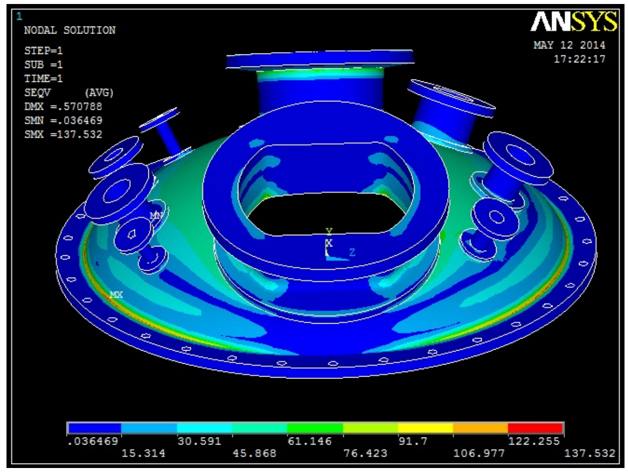

Fig. 6 Analysis of actual industrial pressure vessel head with the flat flange

Analysis of Shell and Bottom with flat flange:
Testing Pressure -8 kgf/cm^2

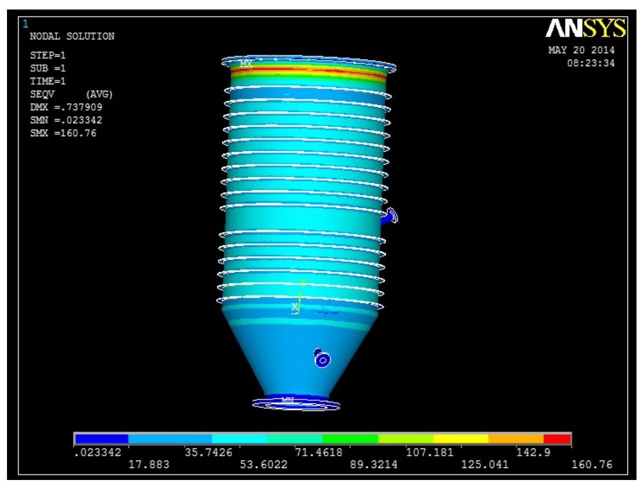

Fig. 7 Analysis of actual pressure vessel shell with the flat flange

CASE-II Modified Pressure Vessel Analysis on Ansys:
The modified model is similar to earlier pressure vessel used in industry except flange design. Flange design used for shell and head connection is changed from flat flange to hub flange. Testing Pressure -8 kgf/cm^2

ISSN 2348 – 7968

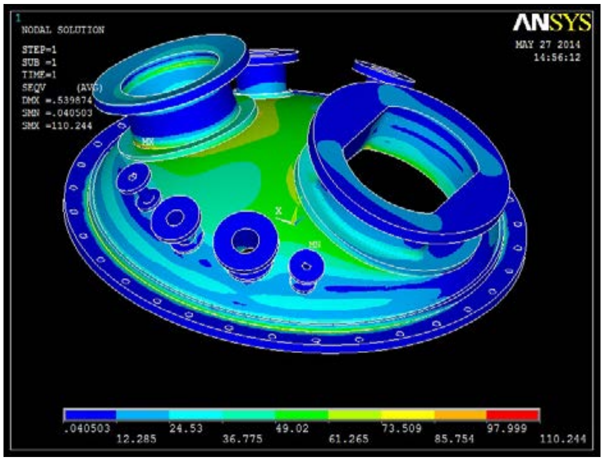

Fig. 8 Analysis of head on the Ansys for hub flange.

Analysis of Shell and Bottom with Hub flange:
Testing Pressure -8 kgf/cm^2
During the shell analysis all data is same as actual data but in place of flat flange hub flange is considered.

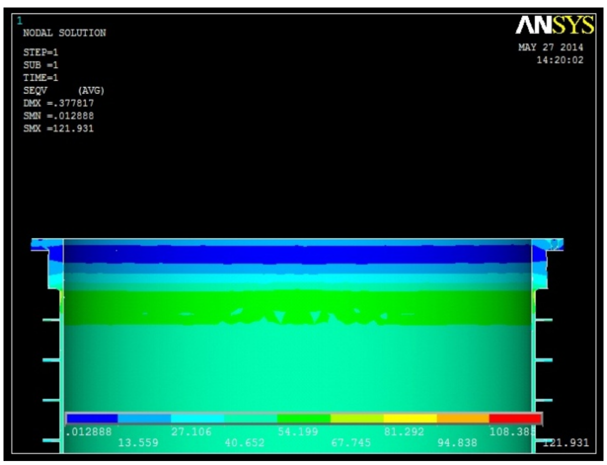

Fig. 9 Analysis of head on the Ansys for hub flange hub.

3. Experimental Analysis of Pressure Vessel

Electrical Resistance strain Gauges are metallic resistors that can be pasted onto the surface of a part on which you want to measure strain. When the part deforms the strain gauge deforms along with it. This deformation is reflected as a proportional change in the resistance of the strain gauge.

Figure-11 shows the actual arrangement of experimental setup in this setup actual pressure vessel is connected with the strain gauge measurement system and strain gauge system is connected with the strain gauges .During the analysis strain is measured in micron and that strain is converted in to stress with help of young modulus of the material.

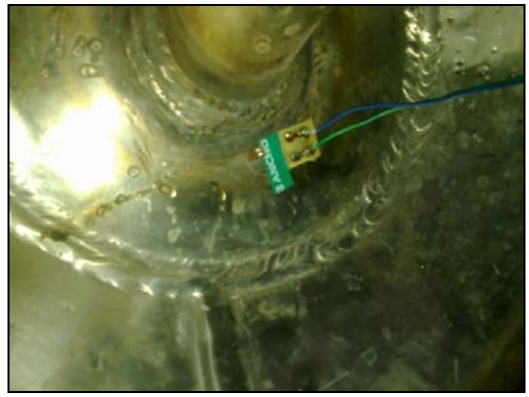

Fig. 10 Electrical strain gauge on shell.

Fig. 11 Strain measurement with pressure vessel.

Observations:
Following observations are found during operating conditions.
Circumferential Stresses in shell:
From mechanics of material laboratory:-
E- Young's Modulus – (for M.S. E= $2x10^5$ N/mm^2)
v- Poisons ratio = for M.S. v=0.3)

σ_c = E X e_h
= $2x10^5$ x 395 x 10^{-6}
=79 N/mm^2

Circumferential Stresses in Bottom:

σ_c = E X e_h
= $2x10^5$ x 248 x 10^{-6} =49.6 N/mm^2

Stresses in Head

$\sigma_{c=}$ E X e_h
=$2x10^5$ x 405 x 10^{-6}
=81 N/mm^2

4. Comparison of Stresses

Table 2: Comparison of FEM with experimental results

Sr. No.	Parts of P.V.	Pressure by FEM (N/mm^2)	Pressure by Strain Gauge (N/mm^2)
1	Shell	69.44	79
2	Toriconical Bottom	42.167	49.6
3	Torispherical head	65.070	81

Comparison of Maximum Stress near to flat flange and Hub flange:

Table 3: Comparison of stresses on different types of flange

Sr. No.	Stress	Max Von-Mises stress near hub flange	Max Von-Mises stress near flat flange	% Variation
1	Shell	121.931	160.760	18.9
2	Head	110.244	137.532	13.3

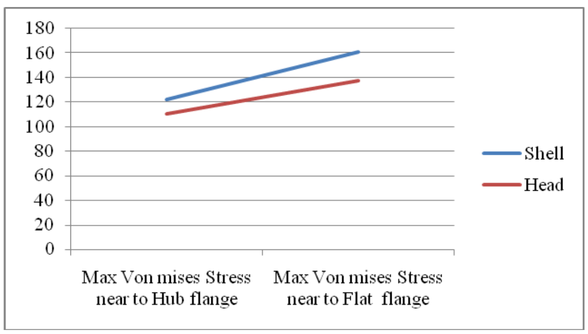

Fig. 12 Comparison of Von-Mises stress on different types of flange.

In ANSYS study found that different types of finite element behavior is different for the same operating condition or load condition and it is observed that Solid 92 Element gives close result at vessel shell and head. Strain gauge is mounted on the different point of the pressure vessel and with the help of strain gauge measurement system stress is calculated and stress found is compared with the ANSYS results.

CASE -I

In this case head and shell is connected with the flat flange and analysis done on the ANSYS and maximum stress is found near to the shell and head flange.

CASE- II

In this case head and shell is connected with hub flange and analysis is done on the ANSYS and it is found that maximum stress is reduced compared to case I

By comparing case –I and case –II it is observed that Hub flange gives more strength when compared to flat flange.

5. Conclusions

The basic objective of this work is design and analysis of the pressure vessel assembly using ANSYS. The flange and welding portion become a critical in design as it is subjected to a differential pressure The interference between the vessel and welding is a critical area and is analyzed in FEA to understand effects on Stress attributes of the Vessel. We can have an analytical design of regular pressure vessel with reference to ASME Boiler and Pressure Vessel Code, Section VIII, Div.1, 2, and 3. Accordingly stress is calculated at the different location of vessel in Experimental analysis and that stress is found very close with the ANSYS result from table 2. Industrial vessel model design is safe but maximum stress found near to the shell flange and head flange that is $165N/mm^2$ and 137 N/mm^2 respectively from table 3. Due to this maximum stress, some portion of shell and head weakens over the period of time. In modified model hub type flange used in vessel and found that maximum stress near to Head flange and shell flange is $110N/mm^2$ and $121.N/mm^2$ respectively from table 3. Comparing the flat flange and hub flange model on the ANSYS with the same loading and operating condition, we found 15 to 20 % reduction in stress.

References

[1] Amran Ayob,Faculty of Mechanical Engineering, “Stress Analysis Of Torispherical Shell With Radial Nozzle” Journal - The Institution of Engineers, Malaysia (Vol. 67, No. 3, September 2006)

[2] ASME codes -2003 -Section VIII, Division –I, Pressure Vessel codes,

[3] Bin Liang a,b,, N. Noda b,1, Shufen Zhang a, “Two Types Of Optimization Of A Cylindrical Shell Subjected To Lateral Pressure” B. Liang et al. / International Journal of Pressure Vessels and Piping 83 (2006) 477–482

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

[5] M.V.Joshi, Edition 1981, “Process Equipment Design”, Macmillan India Limited, New Dehli.

[6] S. Golabi A. Ghorbanpour arani and M. Tahvilian.“Experimental and numerical study of the effect of stiffeningrings on reducing the thickness of torispherical heads with pressure on their concave side”. Iranian Journal of Science & Technology, Transaction B, Engineering, Vol. 32, No. B4, pp 425-436 Printed in the Islamic Republic of Iran, 2008