Finite Element Analysis of Horizontal Tube Sheet Filters: A Review

Mrs. Rutuja Deshmukh[1], Mr. Sumant S. Patil[2], Mr. Sagar Gade[3]
Dept of Mechanical Engineering
[2, 3] AISSMS COE, Pune -01
[1]JSCOE, Hadapsar, Pune -28

Abstract- *Ideally Vertical Pressure Vessels are efficient for tube sheet applications, however in certain cases of space constraint, it is imperative to adjust and place a tube sheet horizontally. However this creates an uneven stress distribution. As the filter sheets start clogging, gravity effects ensure that the sheets at bottom start clogging more than the ones on the top. This in turn means that the delta pressure on the bottom side is more than on the top side. Hence, along with the axial push that the filter sheet will experience, it will also have slight bending component, and this combination will create a non-uniform stress profile on the assembly. The aim of this paper is to use FEA and determine limits on the non-uniform stress profiles for the operating conditions, so that as per ASME code alarms and safety equipment's can be designed according to the outputs.*
spring stiffness for different oils.

Keywords- Pressure Vessel Design, ASME code, FEA, Modal Analysis, etc.

I. INTRODUCTION

Filter Sheets are non standard components used in Oil & Natural gas Industry for filtration. The usual engineering practice is to extend the current available design with an increased factor of safety. In the pressure vessel high pressure rise takes place due to this pressure vessel has to withstand severe forces. So the selection of pressure vessel is most critical that is pressure vessel is the heart for storage of fluid.

Due to Coagulation in filter ,the internal pressure gradually increases with time. Due to this rise in internal pressure, the tube sheet is deflected for some time and it will break and the joint fails. Ignition temp of Natural gas is around 3000oC .The impurities which are present contaminated fluid having temperature near about 1100oC or even more. In Middle East Region, during operation due to high temperature it gets ignited and results in explosion of overall plant(or may be of whole industry) . To avoid this model of the tube sheet is require to improve as per AMSE code of design.. Finite element based modeling is the fine way to improve model of tube sheet assembly. Analysis required for Validation of FEA results and comparing with tested(actual) results. After validation we are having a strong base for modeling and simulation of proposed assembly.

The filtration process is widely used in various applications where the contaminants needs to be removed(filtered) from the working medium (which is contaminated) of the system. This project deals with the analysis and determination of safety limits on non-uniform stresses of the tube sheet which is widely used in the filters. The application of the filters which is considered in this project is of petro chemical industries. The specific application analyzed dealing with the filtering of natural gas immediately after it is mined.

II. PROBLEM SPECIFICATION

To determine limits on the non-uniform stress for the horizontal tubes which occurred due to clogging while in operation, so that alarms and safety equipment's designed according to the outputs consequently improve performance of new model .

III.MATERIAL SPECIFICATIONS

Table 1. Material Specification

Material	Young's Modulus (MPa)	Poisson's Ratio (MPa)	Yield Strenght (MPa)	Tensile Strength (MPa)
SAE 516 GR 70	1.78E+05	0.30	262	485

IV. OBJECTIVES

a) ASME design for Analysis
b) To use FEA and determine limits on non-uniform stress profiles for the operating conditions
c) Theoretical results Comparison with ANSYS results
d) Combination of loads
e) To analyze variation of stress due to horizontal arrangement.
f) To study the effect of variation of parameters on the performance of the system.

V. DESIGN CALCULATIONS

A. *Process Parameter*

Table 2. Process Parameters

Sr. No.	Parameter Description	Notations	Given Value
1	Internal Pressure	P_i	0.18 MPa
2	External Pressure	P_e	Atmospheric
3	Process Volume	V_p	900 m^3
4	Expected Stagnant Volume	Vs	Not Specified
5	Buffer Volume	V_b	Not Specified
6	Tube Porosity Volume	T_p	90 m^3
7	Tube Length	T_L	5 m
8	Radius of tube sheet	r	5 m
9	Tube Diameter	T_d	0.15 m

B. Calculated Parameters As Per ASME Section-VIII, Division-I

Table 3.Design of Shell (mm)

Outer Dia. (OD)	Inner Dia. (ID)	Thickness (t_s)	Main stock Cylindrical Length (L_0)	Buffer stock Cylindrical Length (L_1)	Total Length (L)
10024	10000	12	14563	3140	17703

Table 4.Design of Head / Dome (mm)

Outer Dia. (OD)	Thickness (t_h)	Dome Height (h_d)
10024	12	2500

Table 5.Design of Nozzle (mm)

Nozzle to Nozzle Distance (NTD)	Height of Nozzle (H)	Inner Dia. (ID)	X	Thickness (t_n)
13130	225	300	306	12

Table 6.Design of Reinforcement Pad (mm)

Inner Dia. (D_{ip})	Outer Dia. (D_{op})	Thickness (t_p)
324	563	12

Table 7.Design of Tube Sheet (mm)

Tube Diameter (T_d)	Tube Sheet Thickness (t_{ts})	P	a	No. of holes
150	361.3	165	142.89	2810

Table 8.Design of Saddle Support (mm)

Thickness of Saddle (t_s)	Height of Saddle (h_s)	Contact angle of saddle (Ø)
38	800	120°

VI. FINITE ELEMENT MODELING AND STRUCTURAL ANALYSIS

The finite element analysis is done using ANSYS software. The model is created using CAD software and is imported to ANSYS so that volumes are generated. Solid 187 type element is used in the analysis. This element is used to mesh volumes in ANSYS and the geometry is fully defined by the element nodes. Meshing is done in the ANSYS software and node connectivity between elements in different surfaces has been achieved as shown in figure 2.The mesh size was limited on the basis of computer ability to process the generated equations during solving. The discretized model is then applied with boundary conditions at the solution stage in ANSYS. A boundary condition for the model is the setting of a known value for a displacement or an associated load. To serve its purposeful function, structures are prevented from moving freely in space at certain points called supports. Since the pressure vessel is supported by means of saddle supports, we need to constrain only the required degrees of freedom at the supports. [2]

1. SOLID187 Element Description

SOLID187 element is a higher order 3-D and 10-node element. SOLID187 has a quadratic displacement behavior . It is well suited for modeling irregular meshes & shapes (such as those produced from various CAD/CAM modules). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions as shown in figure 1.

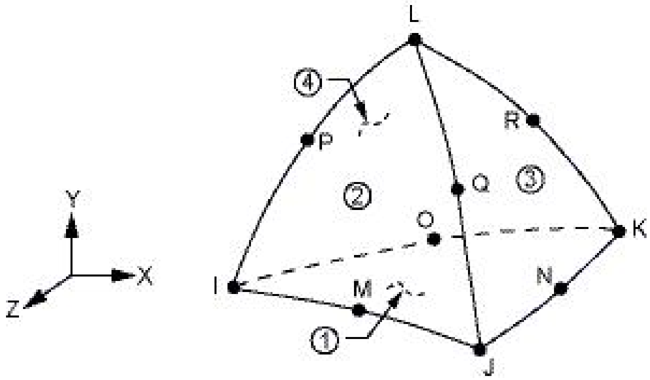


Fig. 1: 3-D 10-Node Tetrahedral Structural Solid [2]

2.Model of *Horizontal* Pressure Vessel

The CAD model of pressure vessel have been create as per ASME Code

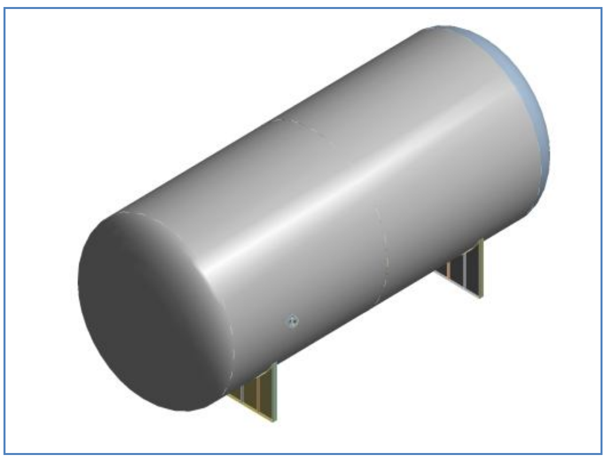


Fig. 2: CAD model of pressure vessel

3. Meshing

3.1 Meshing of whole vessel

All elements have been properly meshed with a fine meshing as shown below

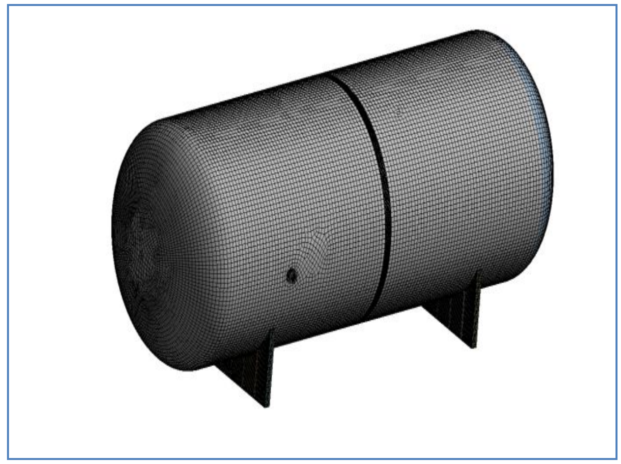
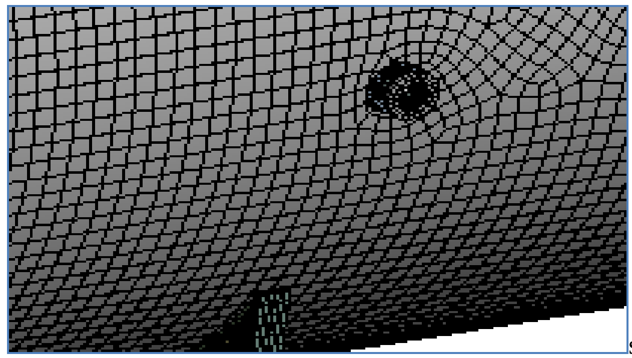


Fig. 3: Meshed Model of vessel

.3.2 Meshing of *Nozzle*



s

Fig. 4: Meshed Model of Nozzle

3.3 Meshing of saddle support

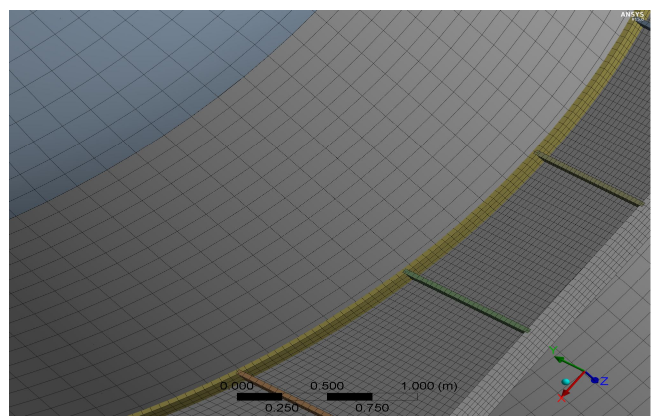


Fig. 5: Meshed Model of Saddle

4. Boundary Conditions for Horizontal Pressure Vessel:

The pressure vessel has two saddle supports at its bottom which helps to specify boundary condition i.e. two saddle supports are fixed to ground

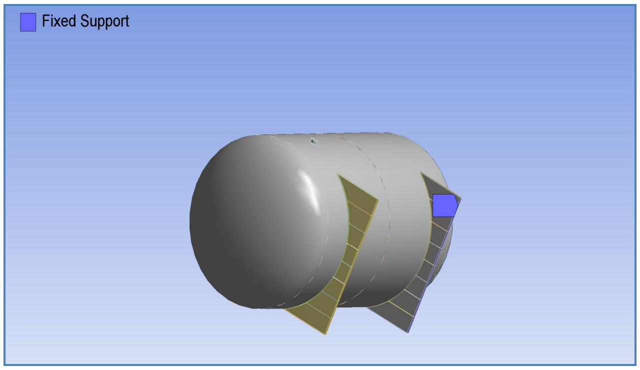


Fig. 6: Boundary conditions

5. Total Deformation of Horizontal Pressure Vessel In loading condition

Total Deformation has been derived from the static analysis of structure using ANSYS as shown in below figures

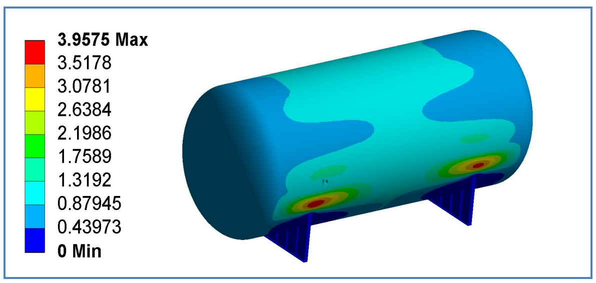


Fig. 7: Total deformation =3.9575 mm

6. Structural Static Analysis of Vessel

Equivalent (von-Mises) stresses has been derived from the static analysis of structure using ANSYS as shown in below figure

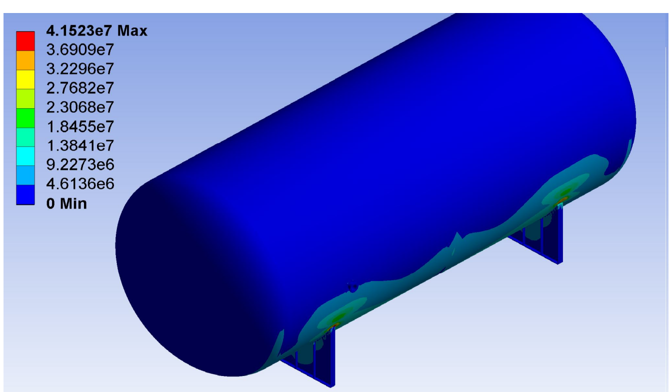


Fig. 8: Equivalent (Von-Mises) stresse =4.1523 Mpa

7. Modal Analysis of Vessel

Modal analysis is used to determine a structure's vibration characteristics — natural frequencies and mode shapes.

Modal analysis is performed as detailed below to determine 5 modes and their respective frequencies. Boundary Condition for Modal Analysis- The entire pressure vessel will be mounted through fixed saddle supports. The bottom surface of fixed saddle has fixed support boundary condition.[3]

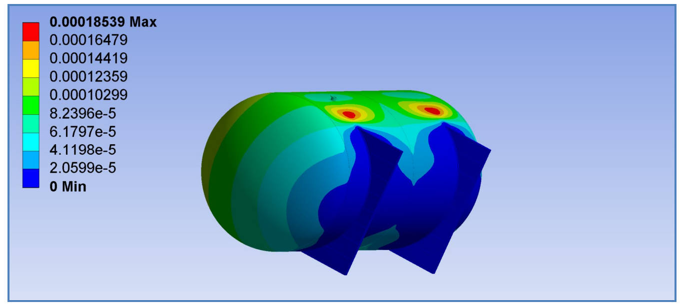


Fig. 9: Natural Frequency with 1st mode =4.5348Hz

Table 9: Modal Analysis using ANSYS

Sr.No.	Mode	Natural Frequency (Hz)
1	1	4.5348
2	2	7.7189
3	3	9.4814
4	4	9.527
5	5	10.32

8. Gravity Analysis of Vessel

Gravity Analysis of Vessel has been done to ensure safe working of vessel during its self weight.

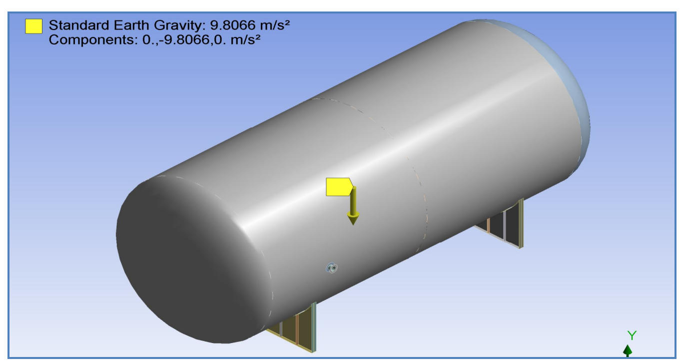


Fig. 10: Gravity Analysis of Vessel

VII. EXPERIMENTAL VALIDATION METHODOLOGY

1.FEA plots:

Finite Element Analysis will gives the values of Total deformation=3.9575 mm and comparing this value with Hydrostat deformation.

2.Hydro Test:

The tube sheet will be checking for the maximum deformation under the fluid pressure of 0.2457 Mpa (Hydrostat test pressure = 1.365 times maximum working pressure) and at standard working temp. To measure the deformation Strain gauge will be locating at the center of the filter tube sheet and the table of values of deformation obtained as well as Experimental and the percentage of error will gives required conclusion.

By comparing FEA Vs Experimental results we can do validation.

VIII. RESULT AND DISCUSSION

1. Pressure vessel has been designed as per ASME Section VIII, Division-I.
2. Model have been created and properly meshed.
3. Calculated parameters have forwarded to the client for testing and they will give required changes in parameters.
4. Again Model have been properly re-meshed according to changes in dimension with reduction in stress concentration.
5. Equivalent (von-Misses) stresses has been derived from the static analysis of structure using ANSYS which are safe in limit.

IX. CONCLUSION

1. Modal analysis have been correctly ensures proper contact details i.e face-to-face, end-to-end, surface-to-surface etc.
2. Gravity analysis is also done to ensure it's safe working during self-weight.
3. Experimental Vs FEA results with acceptable percentage error will gives validation for completion of the project.

REFERENCES

[1] Ms. Shweta A. Naik, "Thickness and Shape Optimization of Filter Sheet by Non-Linear FEA" Global Journal of Researches in Engineering Mechanical and Mechanics Engineering Volume 13 Issue 1 Version 1.0 Year 2013 Type: Double Blind Peer Reviewed International Research Journal Publisher: Global Journals Inc. (USA)Online ISSN: 2249-4596 Print ISSN:0975-5861

[2] Adithya M. M. ,M. M. Patnaik, "Finite Element Analysis of Horizontal Reactor Pressure Vessel Supported on Saddles" International Journal of Innovative Research in Science, Engineering and Technology Vol. 2, Issue 7, July 2013

[3] Londhe B. C., Bhane A. B. "Fatigue Finite Element Analysis of Pressure Vessel for Vertical Acceleration Loading" International Journal of Emerging Technology and Advanced Engineering Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 5, Issue 1, January 2015)

[4] B.S.Thakkar, S.A.Thakkar " Design of pressure vessel using ASME code, Section viii, division 1 "International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974

[5] Zaid Khan "Design and analysis the effect on large opening and structure stability of pressure vessels". International Journal of Engineering Trends and Technology (IJETT) – Volume 12 Number 8 - Jun 2014

[6] Shafique M.A. Khan "Stress distributions in a horizontal pressure vessel and the saddle supports". International Journal of Pressure Vessels and Piping 87 (2010) 239e244

[7] Prof. Vishal V. Saidpatil, Prof. Arun S. Thakare "Design & Weight Optimization of Pressure Vessel Due to Thickness Using Finite Element Analysis" International Journal of Emerging Engineering Research and Technology Volume 2, Issue 3, June 2014, PP 1-8 ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online).

[8] S. S. Pande, P. D. Darade, G. R. Gogate , "Transient analysis and fatigue life prediction of tube sheet", IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163, pISSN: 2321-7308

[9] Dr. Enrique Gomez, Mr. Roberto Ruiz 'ASME SECTION III Stress Analysis of a Heat ExchangerTubesheet With a Misdrilled Hole and Irregular or Thin Ligaments' [Conference] // ASME 2013 Pressure Vessel and Piping Conference. - Paris : ASME, 2013.

[10] Giglio M. 'Fatigue Analysis of Different types of Nozzles' [Journal] of International Journal of Pressure Vessels and Pipings.

[11] Liu Minshan 'Stress analysis of Ω-tubesheet in waste heat boiler' [Journal]. - [s.l.] : Journal of Pressure Equipment and Systems, 2006.

[12] Myung Jo Jhung Jong Chull Jo 'Equivalent Material Propeties of Perforated Plate with Triangular or Square Penetration Pattern For Dynamic Analysis' [Journal]. - [s.l.] : Nuclear Engineering and Technology, 2006. - Vol. 62.

[13] Nandagopan O. R. 'Non Linear Behaviour of Perforated Plate with Lining' [Journal]. - [s.l.] : Defense Science Journal, 2012. - Vol. 62.

[14] R. D. Patil Dr. Bimlesh Kumar 'An Approach to Finite Element Analysis of Boiler Tubesheet' [Journal]. - [s.l.] : American Journal of Engineering Research, 2013. - 08 : Vol. 02.

[15] 2010 ASME Boiler and Pressure Vessel Codes Section VIII Division II