International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 02 Issue: 05 | Aug-2015 www.irjet.net p-ISSN: 2395-0072

Investigating Stress Level through FEA of Nozzles of Carbon Drain Vessel by ASME Sect-VIII

Navnath V. Avhad[1], Vijay G. Bhamre[2]

[1] *Assistant Professor, Department of Mechanical Engineering, SNDCOE&RC, Yeola, Maharashtra, India*
[2] *Assistant Professor, Department of Mechanical Engineering, SNDCOE&RC, Yeola, Maharashtra, India*

Abstract - ***The objective of analysis is to check stress levels in the Nozzle N1, N4, N12 & N13 of Hydro Carbon Vessel. If geometrical parameters of nozzle connections significantly vary even in one pressure vessel, nozzles cause geometric discontinuity of the vessel wall. Therefore a stress concentration is created around the opening and junction may fail due to these high stresses. If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity, sometimes this cause generation of a couple & lead to a structural imbalance. The single Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Therefore, it needs to analyze in FEA to understand effects of nozzle on Stress attributes of the vessel. The study is conducted to determine the stress levels in the nozzle N1, N4, N12 & N13 to a sufficient level of accuracy. Model is analyzed with given conditions i.e. for internal design pressure. FEA analysis is performed according to ASME Section VIII, division 2, part 5.2.***

Key Words: *FEA of Nozzles, Hydro Carbon vessel, ASME Sec-VIII Div-2.*

1. INTRODUCTION

Nozzles or opening are necessary in the pressure vessels to satisfy certain requirements such as inlet or outlet connection, manholes, vents & drains etc. If geometrical parameters of nozzle connections significantly vary even in one pressure vessel, nozzles cause geometric discontinuity of the vessel wall. Therefore a stress concentration is created around the opening and junction may fail due to these high stresses. If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity, sometimes this cause generation of a couple & lead to a structural imbalance. The single Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Therefore, it needs to analyse in FEA to understand effects of nozzle on Stress attributes of the vessel.

The internal pressure is primary loading used in the structure analysis for determination of main vessel-nozzle connections. The effect of external forces and moments applied to nozzle should also be taken into consideration in addition to the stresses caused by the internal pressure. External loading usually imposed by a piping system attached to the nozzle & Values of the loads & moments are calculated by an analysis of piping system. Many works including analytical, experimental & numerical investigations have been devoted to the stress analysis of nozzle connections in pressure vessels, subjected to different external loadings. The codes do not provide any methodology to calculate the extended and magnitude of these high stresses but suggest a procedure to design the junction.

Therefore, there is need to carry out a detailed FE analysis of the junction to calculate stresses at the junction & both in the vessel & in the nozzle.

2. PROBLEM STATEMENT & OBJECTIVES

Thorough the study of 2-D drawing is performed, to check stress levels in the Nozzle N1, N4, N12 & N13 of Hydro Carbon Vessel. If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity, sometimes this cause generation of a couple & lead to a structural imbalance. The single Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Therefore, it needs to analyze in FEA to understand effects of nozzle on Stress attributes of the vessel. This work as the main goal of this study was to investigate whether the real world experience can be modeled in Finite Element Analysis.

2.1 Necessity

To prevent stress related vessel rupture failure, it is necessary to identify the main factors that contribute extensively to stress development in pressure vessels and how they can be mitigated. This work presents critical design analysis of stress development using 3D CAD models of pressure vessels assembly and finite element

engineering simulation of various stress and deformation tests at high specified pressure.

2.2 Objective of Work

1. Checking stress levels in the Nozzle N1, N4, N12 and N13 of Hydro Carbon Vessel.
2. Create a 3D model of nozzle & carried out study for given condition of vessel.
3. Model of nozzle are analysed with given conditions i.e. for internal design pressure.
4. FEA analysis is performed according to ASME Section VIII, Division 2, Part 5.2.

3. ANALYTICAL CALCULATIONS

Hoop stress in shell away from discontinuity can be calculated by using Lame's equation. The calculations for stress due to pressure are as follows,

$$\sigma_\phi = \frac{P * {R_i}^2}{({R_o}^2 - {R_i}^2)}\left(1 + \frac{{R_o}^2}{{R_i}^2}\right)$$

Where,

Ro = Outer radius of shell, mm.
R i = Inner radius of shell, mm.
P = Working pressure = 0.35 MPa

3.1 For Nozzle N1:

The calculations for stress due to pressure & other boundary conditions in Nozzle is as,

Outer Diameter = 168.28 mm
Thickness = 18.263 mm
Ro = 84.14 mm
Ri = 65.877 mm
P = 0.35 N/mm^2

$$\sigma_\phi = \frac{0.35 * 10^6 * 65.877^2}{(84.14^2 - 65.877^2)}\left(1 + \frac{84.14^2}{65.877^2}\right)$$

After solving this we get,

σ_ϕ = 1.458 * 10^6 Pa

σ_ϕ = 1.458 MPa

4. FEA VALIDATION

Because of complicated shapes stress analysis by using photo elasticity will also be difficult. Stress analysis by FEM is obviously best choice. Hence finite element technique has been selected for analysis purpose. FEM by Ansys software is one of the most popular commercial software used for finite element analysis. The objective of analysis is to check stress level in nozzles of pressure vessel in accordance with ASME section VIII Div-2. The study is conducted to determine stress levels in nozzles to a sufficient level of accuracy.

Hence the study is conducted by using the following methodology. A 3D model of pressure vessel with nozzles is created by using solid work CAD software.

4.1 2D Model of Vessel with Nozzle Positioning

The 2D model of vessel with Nozzle positioning is shown in figure below. Consider nozzle N1 for analysis,

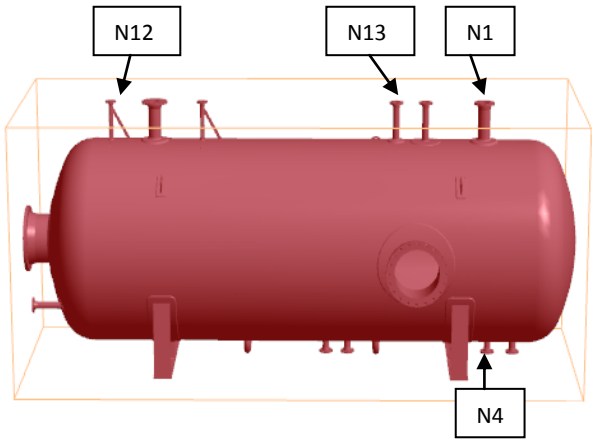

Fig -1: 2D model of vessel with Nozzle positioning

4.2 3D Model

The 3D model of carbon drain vessel is created in CAD and imported in ansys software as shown below,

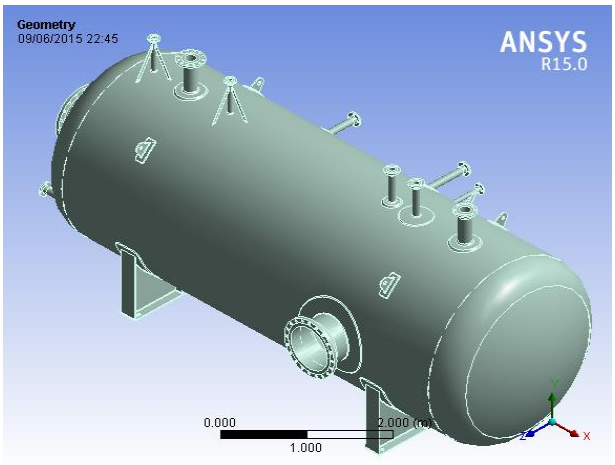

Fig -2: 3D Model

4.3 Meshing Model

Meshing is nothing but descretization of object into the small parts called as element. Following figure shows meshed model of drain vessel.

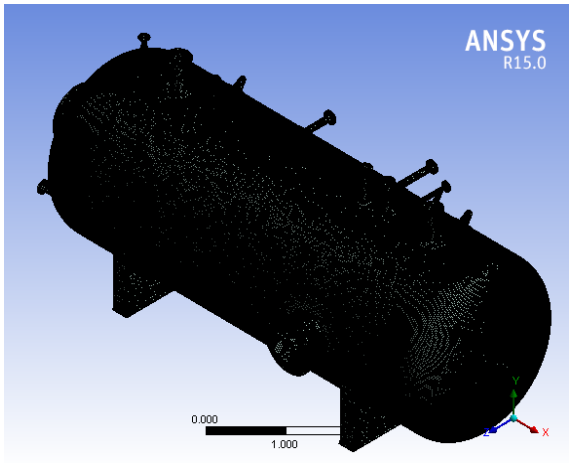

Fig -3: Meshing Model

4.4 Boundary Conditions:

Boundary conditions are applied on pressure vessel and nozzles for analysing the stress level in vessel. Operating Pressure, Force and Moments applied on nozzle N1 and results are evaluated as follows,

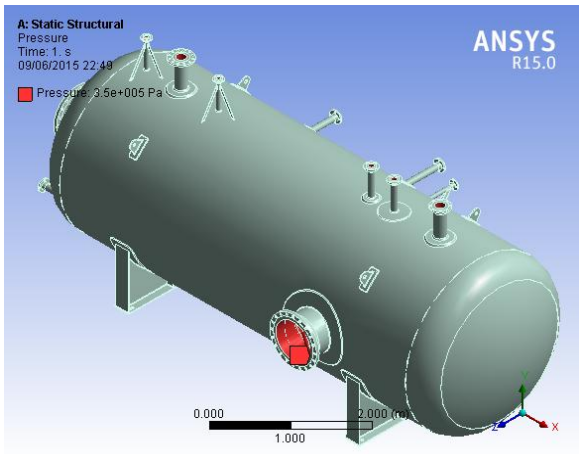

Fig -4: Operating Internal Pressure plot

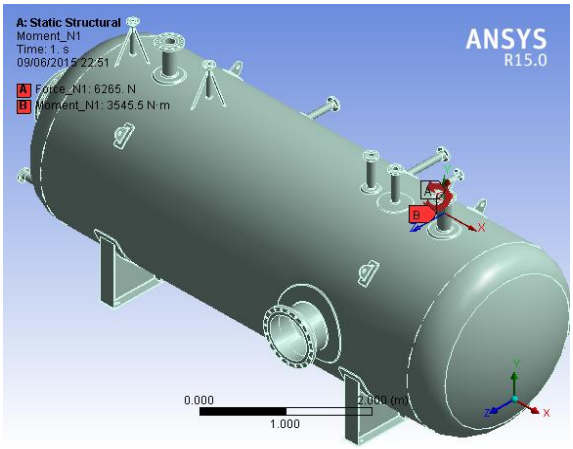

Fig -5: Force & Moment Plot for N1

4.5 Equivalent Stress Plot:

After evaluating the shell through FEA, the equivalent stress in the shell is as follows,

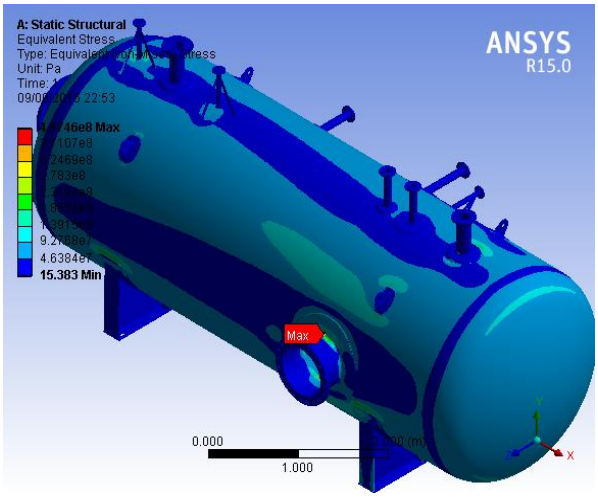

Fig -6: Equivalent Stress Plot

4.6 Normal Stress

Normal stress plot for nozzle N1 is as shown in figure as follows.

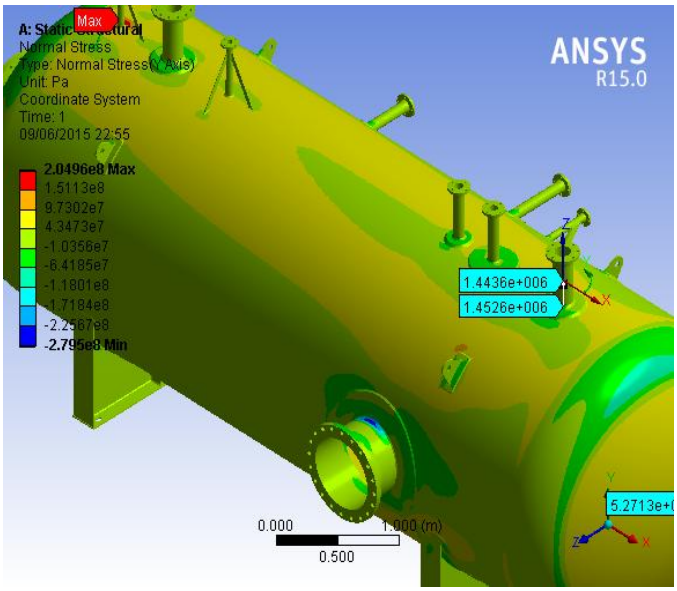

Fig -7: Normal Stress Plot

5. RESULT AND DISCUSSION

Analytically, by using mathematical formulae we can calculate the stress level in nozzle of pressure vessel. The mathematical results are then compared or validated with FE analysis results. Thus, Finite element analysis is used for evaluating the stress with more accuracy.

5.1 Analytical Result

Hoop stress in nozzle N1 away from discontinuity is calculated analytically,

$$= 1.458 * 10^6 \text{ Pa} \ldots\ldots\ldots\text{(i)}$$

5.2 Finite Element Analysis Result

Normal stress in nozzle N1 away from discontinuity is evaluated by finite element analysis (Ansys software) as shown in figure,

$$= 1.4526 * 10^6 \text{ Pa} \ldots\ldots\ldots\text{(ii)}$$

The above results are regarding with nozzle N1, the same procedure we can adopt for evaluating the stress level at nozzle N4, N12 & N13. Analytically, the stresses in the nozzle section can be calculated by using mathematical formulae and results then validated through Finite Element Analysis.

6. CONCLUSION

In this way investigation of stress level in nozzle is carried out as per the ASME section code & it can be concluded that Finite Element analysis is required to match the results that of hand calculations. This study is not exhaustive and conducted to try doing Finite Element analysis of a Pressure Vessel Nozzle section using 3D modeling and post-processing for analysis nozzle with pressure vessels according to codes puts high requirements on the analysis method. Not uncommonly, the mechanical design of a pressure vessel is verified by a third party inspection organization to make sure that it fulfills the requirement of the code. It is therefore of vital importance that the model is set up properly in order to get acceptable results.

REFERENCES

[1] **Dharmit Thakore, "**Finite Element Analysis of Pressure Vessel and Nozzle Junction" Revision: 0,Date: 29 March 2013.

[2] **Erik Bjarkby,** "Parameterized model for stress analysis of nozzles", 2011

[3] **Pravin Narale , Prof. P. S. Kachare,** "Structural Analysis of Nozzle Attachment on Pressure Vessel Design" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue4, July-August 2012, pp.1353-1358.

[4] **Lijing Wen, Chao Guo, Tieping Li, Chunming Zhang,** "Stress Analysis for Reactor Coolant Pump Nozzle of Nuclear Reactor Pressure Vessel" Journal of Applied Mathematics and Physics, 2013, 1, 62-64 Published Online November 2013.

[5] **A. Hardik B. Nayak and B. R. R. Trivedi, "**Stress Analysis of Reactor Nozzle to Head Junction" Institute Of Technology, Nirma University, Ahmedabad – 382 481, 08-10 December, 2011.

[6] **A Alper Ozalp**, "Numerical analysis of choked converging nozzle flows with surface roughness and heat flux conditions" Sadhana Vol. 31, Part 1, February 2006, pp. 31–46.

[7] **M. Giglio,** "Fatigue analysis of different types of pressure vessel nozzle" International Journal of Pressure Vessels and Piping 80, www.elsevier.com/locate/ijpvp (2003) 1–8.

[8] **Sanket S. Chaudhari & D.N. Jadhav**, "A Suggested Stress Analysis Procedure For Nozzle To Head Shell Element Model – A Case Study", International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME) ISSN : 2319 – 3182, Volume-1, Issue-2, 2012.

[9] **Shyam R Gupta, Ashish Desai**, "Optimize Nozzle Location For Minimization Of Strees In Pressure Vessel", IJIRST–International Journal for Innovative Research in Science & Technology| Vol. 1, Issue 1, ISSN(online): 2349-6010, June 2014.

[10] **M. A. Nalawade, S. H. Dwivedi, etl.,** "Analysis of STHE to Identify Low Stress Area for Locating Nozzle Opening" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) ISSN(e) : 2278-1684, ISSN(p) : 2320–334X, PP : 44-49.