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

Navnath V. Avhad¹, Vijay G. Bhamre²

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...
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](image)

### 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](image)

### 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.
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.

4.5 Equivalent Stress Plot:

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

4.6 Normal Stress

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

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,
\[ = 1.458 \times 10^6 \text{ Pa} \quad \text{(i)} \]

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,
\[ = 1.4526 \times 10^6 \text{ Pa} \quad \text{(ii)} \]

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 uncomonly, 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