Design and Finite Element Analysis for Pressure Vessel Nozzles

H.A. Khande, A.B. Rathod, A.A. Khot, Amaan Afroz, R.R. Mahajan
Mechanical Department of Skncoe , Savitri Bai Phule Pune University

Abstract:- This paper presents the stress analysis of nozzle and shell junction of a pressure vessel. The ASME Boiler and Pressure Vessel Code (BPVC) standards are used for the design and fabrication of boilers and pressure vessels. ASME section viii division 1 follows design-by-formula approach while division 2 contains a set of alternative rules based on design-by-analysis approach. Div.2 has procedure for the use of Finite Element Analysis (FEA) to determine the expected stresses that may develop during operation. A solid model, pressure vessel having nozzle is created by using Design Modeler of ANSYS program. For given boundary and loading conditions, the stress developed is analyzed using mechanical workbench of ANSYS software. After analysis, it is found that maximum localized stress arises at the nozzle to shell interface near the junction area. The results obtained shows that the nozzle design is safe for the design loading conditions.

Keywords— Pressure vessel; FEA; nozzle; stress analysis; ASME BPV; ansys.

1. INTRODUCTION

A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. The failure of pressure vessel is very dangerous and sometimes heavy loss of life, health and property. Pressure vessel nozzles are required for inlet and outlet purposes. Husain developed a simplified formula of stress concentration factors for pressure vessel nozzle junction. The author explained that the value of SCF (Stress concentration factor) was depended on not only the vessel stresses but also geometric configuration of juncture. Smetankin and Skopinsky worked on the structural modeling and stress analysis of nozzle connections in ellipsoidal head of pressure vessel. They considered external loadings and applied Timoshenko shell theory. They recommended that internal pressure and external loading both should be consider for complete and accurate stress analysis of nozzle connections of a pressure vessel. Qadir studied stress concentration factor (SCF) of pressure vessel nozzle connections subjected to internal pressure and found wall thinning effect. Observations shown that increased in the diameter ratio d/D increased the SCF value for a specific diameter-thickness ratio D/T. It also found that for a specific d/D ratio, the SCF value increased as the D/T ratio was increased. Yu Sun et al. presented work on strain linearization for structural strain by using maximum principle structural strain criterion. Chapuliot and Marie [11] used elastic-plastic fracture mechanics assessment of nozzle corners subjected to thermal shock loading. Dong et al.[4] presented a new structural strain method to extend the early structural stress based master S-N curve method to low cycle fatigue regime in which plastic deformation can be significant while an elastic core was still present. Mukhtar and Husain focused design and stress analysis of cylindrical pressure vessels intersected by small-diameter nozzles. Author used solid elements (based on theory of elasticity) in modeling the cylindrical vessels with small-diameter nozzles. Porter et al.explained the formulations of different elements and selection criteria for the elements. There should be 2.5(rt)^0.5 distance between two discontinuities. Finer mesh will give better results but also time for analysis will increase. A great care should be taking while applying boundary conditions because boundary performed elastic-plastic analysis and found that the local stress distribution near opening was maximum. There is high probability of progressive ductile fracture in this zone due to stress concentration.

The connection region of vessel shell and nozzle can become the weakest location. Hence, a detailed analysis is required. The ASME Boiler and Pressure Vessel Code Section VIII Division 2 provides standard regarding this type of analysis. In this paper, Finite Element Analysis is used to determine the stress distribution and possible failure location for pressure vessel and nozzle connection as per ASME VIII Division 2. This type of analysis will allow a pressure-vessel designer to understand how the vessel will fail, and creates the opportunity to design in safety features into the pressure vessel and its surrounding containment component.

2.DESIGN PARAMETER

Code		ASME SEC VIII DIV 1
CONTENTS		AQUEOUS CAUSTIC,LPG
SOUR SERVICE		YES
OPERATING TEMPERATURE(TOP/BOTTOM)	C	45/45
OPERATING PRESSURE(TOP/BOTTOM)	Kg/cm^2gm	18.2/19.5
DESIGN TEMPERATURE	C	70/-27
DESIGN PRESSURE INTERNAL(TOP/BOTTOM)	Kg/cm^2gm	42/42
DESIGN PRESSURE EXTERNAL	Kg/cm^2gm	NIL
HYDROTEST PRESSURE AT TOP(VERICAL)	Kg/cm^2gm	55.42
HYDROTEST PRESSURE AT TOP(HORIZANTAL)		56.78
SPECIFIC GRAVITY		.525
SHELL THICKNESS	mm	35
Corrosion allowance	mm	3.00/0.00
HEAD THICKNESS	mm	34/34
ORIENTATION		vertical
TYPE OF HEADS		ELLIPTICAL
VESSEL VOLUME	mm	21.71
LIQUID VOLUME	mm	24.26
SEISMIC DESIGN CODE		IS-1893 PART 4 SPECTRUM METHOD
STEAM OUT CONDITION		3KG/CM2C AND FV 175 C
FIRE PROOFING TYPE		CEMENTITIOUS DENSE CONCRETE (2400KG/M3)
WIND DESIGN CODE		IS-875 PART 3,1987 (REAFFIRM 2003)
PAINTING/CLEANING		NOTE -17

3.Materials of Construction

Table 1.2- Materials Construction of :

Components	**Material Grade**
Shell	ASTM A 516 GR 60 N
Nozzle	ASTM A 105 N

1.3 Material Properties for Analysis

Material	Design Temperature (°C)	Elastic Modulus (MPa)	Allowable Stress (MPa)	Yield Strength (MPa)	Density (Kg/m3)	Poisson's Ratio
ASTM A 516 GR 60 N	70°C	199.6E03	118	207	7750	0.30
ASTM A 105 N		199.6E03	138	232.14	7750	0.30

3.PROBLEM STATEMENT

Welding Research Council (WRC) Bulletin 107 & 297 are two of the most widely used method to calculate the stress at the connection of attachments and nozzles on the vessels. WRC-107 is for attachments and nozzles on spherical shell and attachments on cylindrical shell.WRC-297 was published as a supplement to WRC-107, and it is invaluable to calculate the stress at the nozzle connections on cylindrical shells.Even though those two bulletins are widely used to design attachments and nozzle connections, many engineers have a question about the accuracy of the resultant of the bulletins.To resolve the question, three dimensional finite element (FE) Model analyses were performed and the resultant stress intensities were compared with the stress intensity calculated with WRC-297.The other object of this study is to suggest methodology to used the stress intensity calculated at perpendicular line as described in WRC-297 for calculating stress intensity at diagonal line.

4.METHODOLOGY

The study is conducted to determine the stress level for T2, V1 and SV1 Nozzles**.** Hence the study is conducted using the following methodology
Group1 (80 NB nozzles)
As the nozzles in the group contains the same dimensions hence the single nozzle is consider for the analysis from group 1.

- 3D CAD model of T2,V1 and SV1 Nozzle are generated with the help of drawings provided.
- FEA Analysis of T2,V1 and SV1 Nozzle will be carried out separately since these nozzles are falling outside the scope of WRC-107 & WRC-297.
- FEA Analysis of **T2,V1 and SV1 Nozzles** will be carried out for in following load case
  - **Load Case 1** – Design Pressure
  - **Load Case 2** – Steam Out
  - **Load Case 3** – FV Condition
- Results are validated as per ASME Section VIII Div. 2 Part 5.

5.FINITE ELEMENT MODEL

4.1 Geometry of T2,V1 & SV1 nozzle- 3D CAD model of T2,V1 & SV1 Nozzle

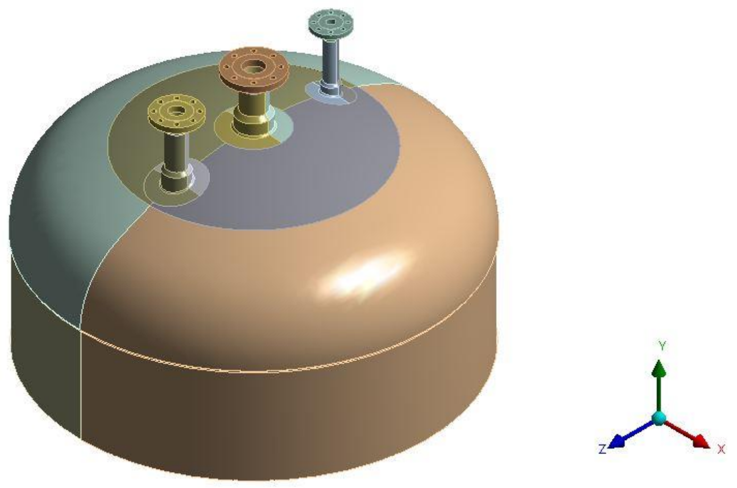


Figure 2.3 – 3D CAD model of T2,V1 & SV1 Nozzle

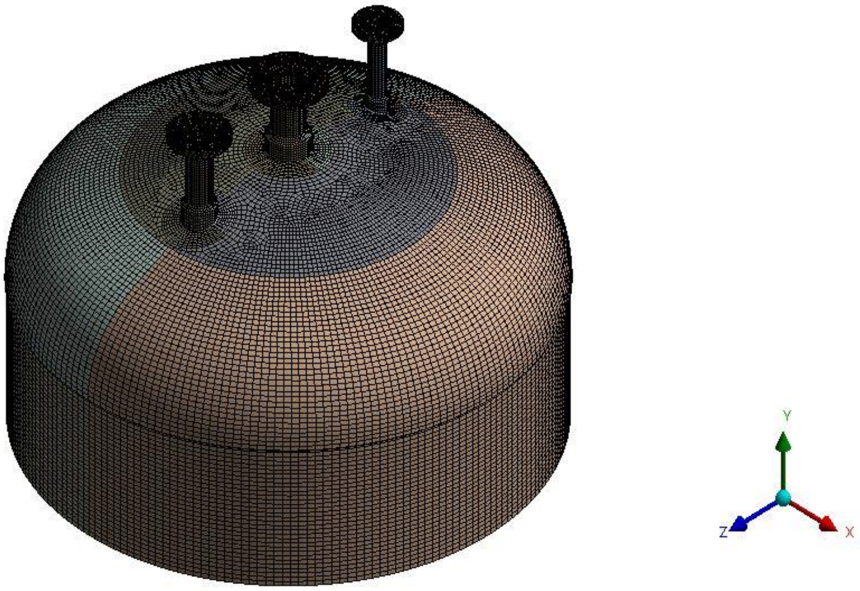
5.2 Finite Element Model:

Finite Element Model is based on 3D CAD model of Nozzles
Solid-186 element is used for analysis.
Total numbers of Elements in T2,V1 and SV1 nozzle Model = 133090

Minimum 2 to 3 elements are maintain across the thickness of each component in order to achieve accurate results at all critical locations.
Overall mesh quality checks with their acceptable limits and achieved values are shown below in Table 2.1

Quality Check	Acceptable Value	Achieved Value
Aspect Ratio	< 5	4.40
Jacobian Ratio	> 0.5	0.97
Skewness	< 0.70	0.65
Element Quality	> 0.1	0.87

Table 2.1- Mesh quality parameters
Meshed FEA model of T1 and F2 nozzle model



6.CONCLUSION AND FUTURE SCOPE

This paper outlines the Design by Analysis methodologies offered in ASME Section VIII Division 2 for satisfying protection against plastic collapse including elastic stress analysis.

We should apply a smaller mesh element size to all shell and nozzle junction areas to capture stress concentration accurately.

Pressure vessel stress behaviors are studied under various loading conditions including internal pressure as well external loading. Analyzed Load cases are the simulation for actual loading environment under which a pressure vessel is subjected during their service life. It found that maximum stress concentration occurs at the junction of Pressure Vessel shell and the nozzle.

Along with the modeling, analysis and verification a discussion on how to perform the code verifications are presented, shows the design is safe for design loading conditions.

The work, design analysis for fatigue and cyclic loading, nozzle optimization with different material, experimental test can do as future scope.

6.REFERENCES

[1] Husain J. Al-Gahtani, Simplified Formulation of Stress Concentration Factors for Spherical Pressure Vessel–Cylindrical Nozzle Juncture. ASME Journal of Pressure Vessel Technology,Vol. 138/031201, June 2016, pp. 1-9.

[2] Smetankin, A.B. and Smetankin, V.N., Modeling and stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under external loading. Int. J. of Applied Mechanics and Engineering, Vol.11(4), 2006, pp. 965-979.

[3] Quider M., SCF analysis of a pressurized vessel-nozzle intersection with wall thinning damage. International Journal of Pressure Vessels and Piping, Vol. 86, 2009, pp. 541-549,

[4] Dong P., Pei X. , Xing S., M.H. Kim, A structural strain method for low-cycle fatigue evaluation of welded components. International Journal of Pressure Vessels and Piping, Vol. 119, 2014, pp. 39-51.

[5] Michael A. Porter, Pedro Marcal and Dannis H. Martens, On using Finite Element Analysis for Pressure Vessel Design. ASME, Pressure vessel and piping division (Publication) PVP,Vol. 338, 1999.

[6] Laczek S., Load capacity of a thick-walled cylinder with a radial hole. International Journal of Pressure Vessels and Piping, Vol. 87, 2010, pp. 433-439.

[7] ASME Boiler and Pressure Vessel Code, Section VIII, Division 1, Division 2.

[8] ASME Boiler and Pressure Vessel Code, Section II-Part D-2015.

[9] ANSYS Release 15.0, ANSYS Structural Analysis Guide & Theory Reference Manual.
Yu Sun, Cheng-Hong Duan, Ming-Wan Lu, Strain linearization for structural strain evaluation and maximum equivalent structural strain criterion. Int. j. of pressure vessels and piping, Vol. 146, 2016, pp.

[10]Pressure Vessel Handbook by Eugene F.Megyesy. (11th Edition).

[11]Pressure Vessel Design Manual by Dennis Moss(3rd Edition).

[12] ASME Boiler and Pressure Vessel Code Section VIII (2017 Edition).