Stress Analysis of Pressure Vessel Nozzle using FEA

Rajendra Bahadur
M.Tech, Mechanical Engineering Department
National Institute of Technology, Kurukshetra
Kurukshetra-136119, India

Vinod Kumar Mittal, Surjit Angra
Faculty of Mechanical Engineering Department
National Institute of Technology, Kurukshetra
Kurukshetra-136119, India

***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.

I. 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 [1] 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 [2] 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 [3] 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. [10] 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 [12] 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. [5] 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 conditions are main source of errors in FEA. Laczek [6] 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 [7] 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.

II. GEOMETRIC PARAMETERS OF SOLID MODEL

Main geometric parameters of pressure vessel shell and nozzle are shown in Table.I and Table.II. Table.III presents design parameters of pressure vessel. A 3D solid model of pressure vessel with a nozzle is created by using design modeler of ANSYS software, which is shown in fig. 1.

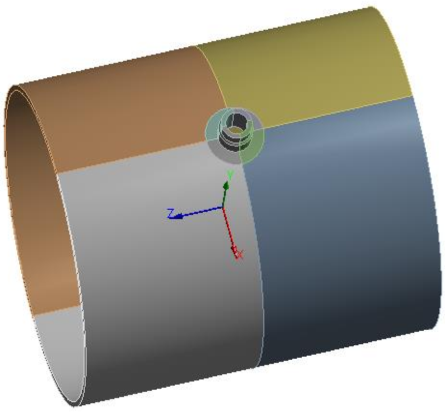


Fig. 1. Solid Model of Pressure Vessel

TABLE I. SHELL GEOMETRY PARAMETERS

Structures Parameters	Size(mm)
Internal diameter of the shell	4000
Thickness of shell wall	74
Length of shell	5000
Corrosion allowance of shell	0

TABLE II. NOZZLE GEOMETRY PARAMETERS

Structures Parameters	Size(mm)
Type of nozzle	Sefreinforced set in nozzle
Internal diameter of the nozzle	300
Thickness of nozzle wall	60
Nozzle projection outside the shell	326
Thickness of hub	85
Height of hub	240
Height of beveled transition	25

TABLE III. PRESSURE VESSEL DESIGN PARAMETERS

Design Parameters	Value
Design Code	ASME B&PV Code Section VIII Div2
Design Pressure	5 MPa
Design Temperature	300°C

III. MATERIAL OF CONSTRUCTION

Pressure vessel's design temperature is 300˚C. Pressure vessel shell and outlet nozzle materials are SA 542 Type D Cl. 4a, SA 182 F22V respectively. According to ASME BPVC Code Sec II [8], construction material properties at design temperature (300˚C) are shown in Table IV.

TABLE IV. MATERIAL PROPERTIES

Component	Material	Tensile Strength U (MPa)	Yield Strength Y (MPa)	Poisson's Ratio	Max. Allowable Stress S (MPa)
Shell	SA 542 Type D Cl. 4a	586	413	0.3	163
Nozzle	SA 182 F22V	586	413	0.3	163

IV. FINITE ELEMENT ANALYSIS

A. *Meshing*

Meshing is done for solid model using eight node brick element to ensure the optimum mesh size of FEA model for proper convergence and exact numerical results. Fine meshing is done at junction area. Meshed model of the solid is shown in fig. 2. In the meshed model total number of elements are 55,785 and total number of nodes are 2,58,407. With this meshing technique, the meshed model of pressure vessel with nozzle gives results that are more accurate.

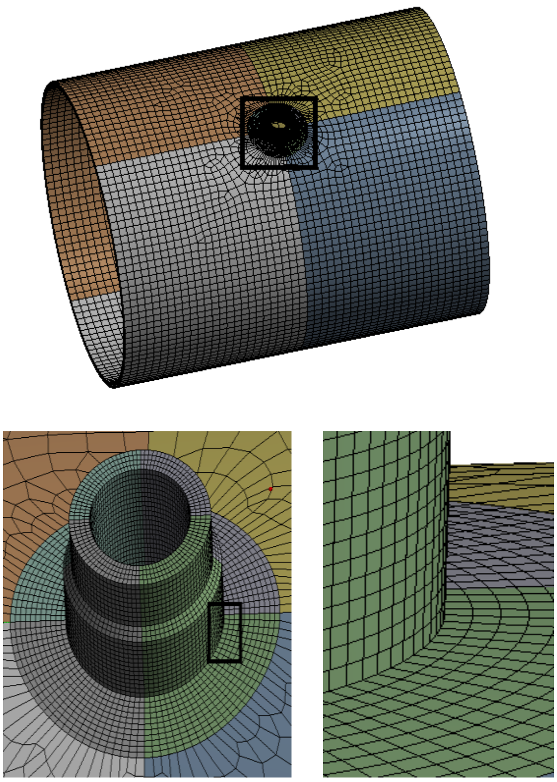


Fig. 2. Mesh Model

B. *Boundary Conditions*

Fixed boundary condition A is applied at the left face of modeled portion of the shell to avoid rigid body motion. Refer Fig. 3.

C. *Loading Conditions*

The internal pressure D (P=5Mpa) is applied on the inner surface of all parts; Longitudinal stress E (5.21 MPa) and F

(66.35 MPa) are applied on the faces of nozzle and Shell respectively as shown in fig.3.

External loads developed due to piping attachment are applied at end face of nozzle as shown in fig.3. Applied External loads are given in Table.V.

TABLE V. EXTERNAL LOADS

Load	Load Component	Load in Model
Moment, M (N-mm)	Longitudinal, M_L	M_X = 3.6×107
	Circumferential, M_C	M_Z = 1.5×107
	Tangential, M_T	M_Y = 2.5×107
Force, F (N)	Axial, P_A	F_Y = 5.0×104
	Longitudinal, V_L	F_Z = 8.0×104
	Circumferential, V_C	F_X = 0.8×104

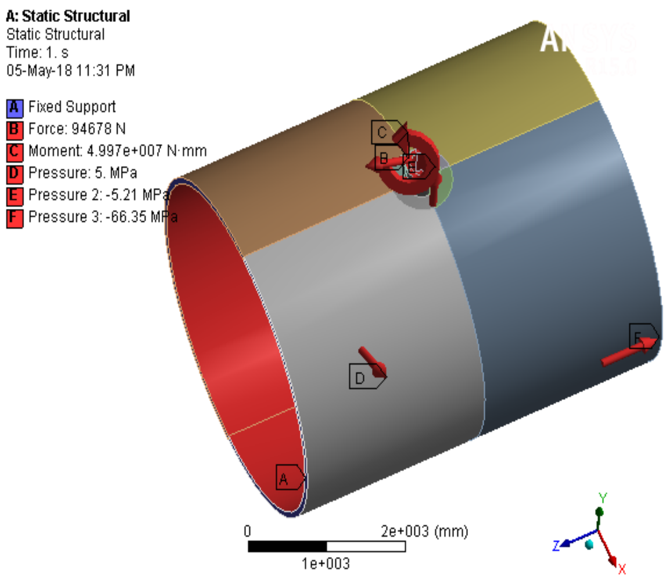


Fig. 3. Boundary and Loading Conditions

The solid model stress behavior is analyzed for a range of internal pressure 0 to 5.5 along with external applied loads. Applied loading conditions for various loading can be seen in Table VI.

TABLE VI. LOAD CASES ANALYZED

Load Case	P (MPa)	M_L (N-mm)	M_C (N-mm)	M_T (N-mm)	P_A (N)	V_L (N)	V_C (N)
1	0	3.6×10^7	1.5×10^7	2.5×10^7	5.0×10^4	8.0×10^4	0.8×110^4
2	4	3.6×10^7	1.5×10^7	2.5×10^7	5.0×10^4	8.0×10^4	0.8×10^4
3	4.5	3.6×10^7	1.5×10^7	2.5×10^7	5.0×10^4	8.0×10^4	0.8×10^4
4	5	3.6×10^7	1.5×10^7	2.5×10^7	5.0×10^4	8.0×10^4	0.8×10^4
5	5.5	3.6×10^7	1.5×10^7	2.5×10^7	5.0×10^4	8.0×10^4	0.8×10^4

V. STRESS CLASSIFICATION AND LINEARIZATION

As the ASME limits [7] are developed to prevent some classic failure modes besides the Primary and Secondary classification, the stresses should be linearized to obtain the generalized (P_m) or localized (P_L) membrane component, the bending (P_b) and the Peak (Q) stress. This linearization should be done along a cross section of the equipment but in the discontinuities. In such a case the stress linearization is done along a line, the SCL – Stress Classification Line. The location of SCL line in a plane of 3D model can be seen in fig.4.

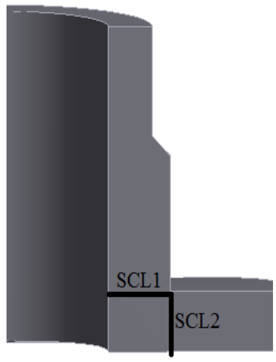
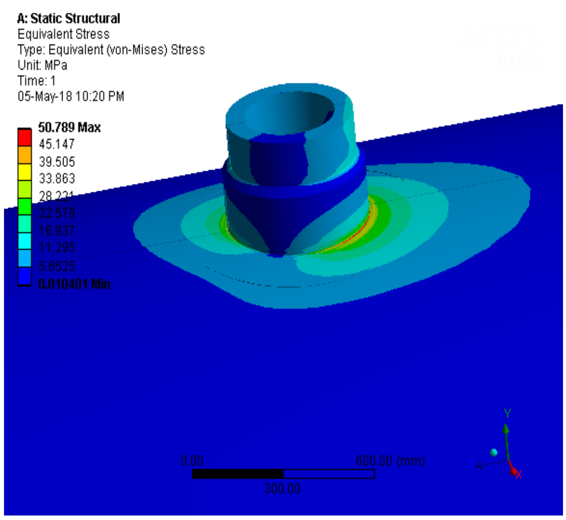
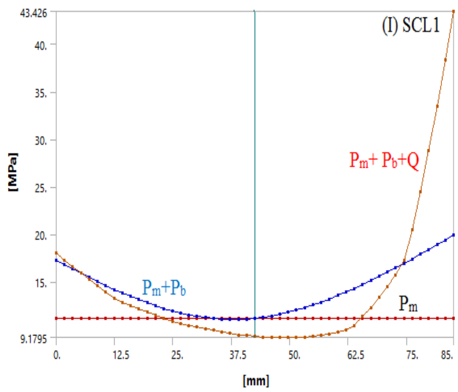


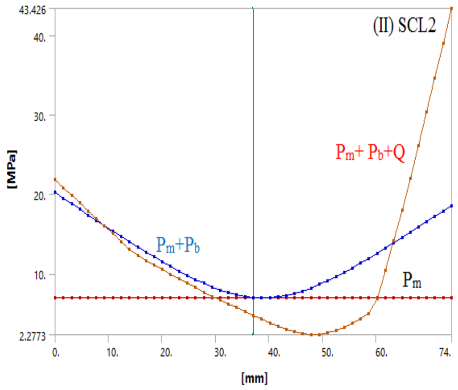
Fig. 4. Location of SCL Line

VI. RESULT AND DISCUSSION



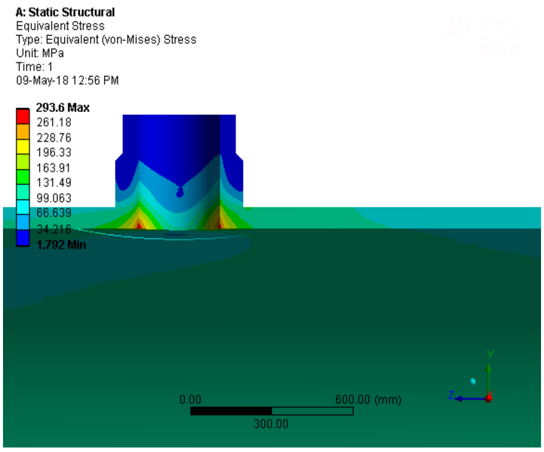
(a)



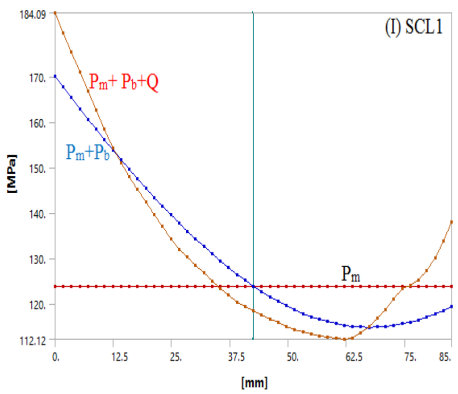


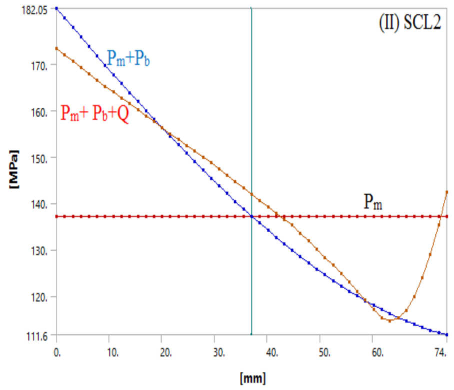
(b)

Fig. 5. Load case 1 (a) Equivalent stress (b) Stress Linearization Plot (I) for SCL1 (II) for SCL2



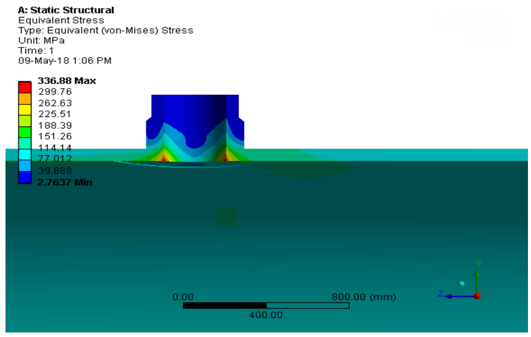
(a)



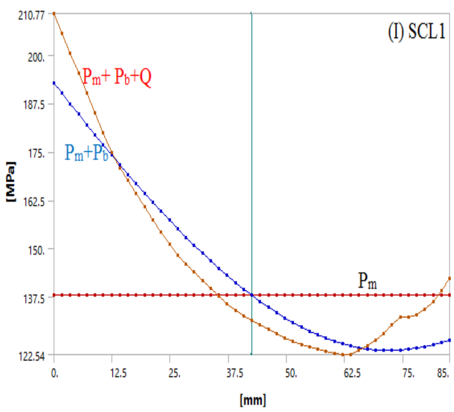


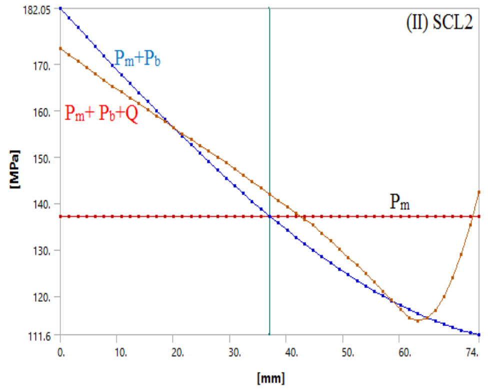
(b)

Fig. 6. Load case 2 (a) Equivalent stress (b) Stress Linearization Plot (I) for SCL1 (II) for SCL2



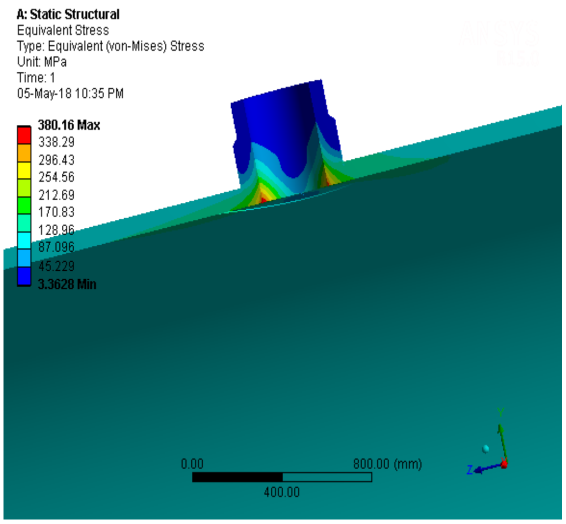
(a)



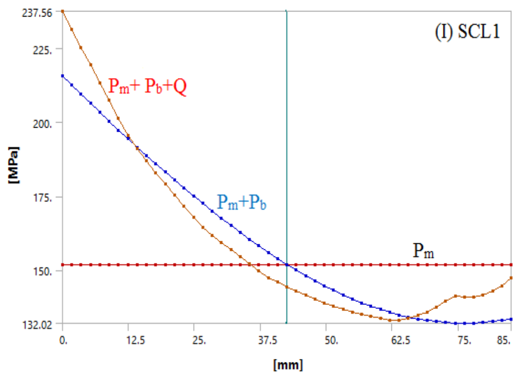


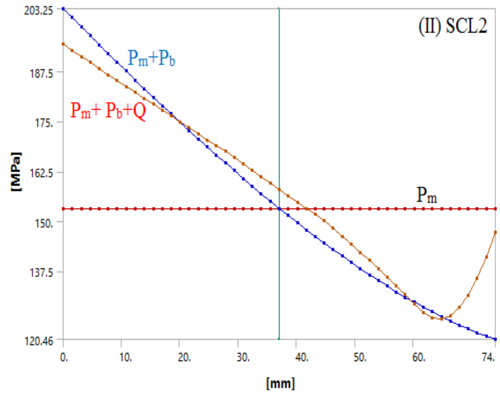
(b)

Fig. 7. Load case 3 (a) Equivalent stress (b) Stress Linearization Plot (I) for SCL1 (II) for SCL2



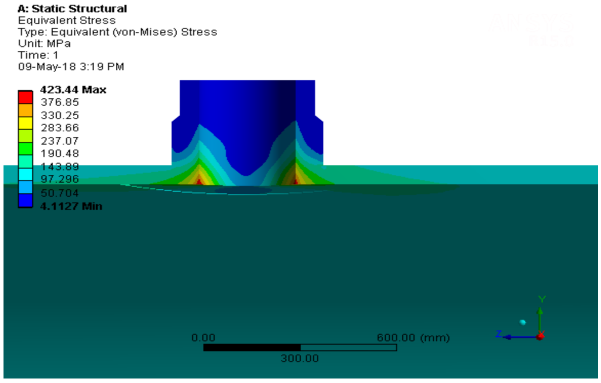
(a)



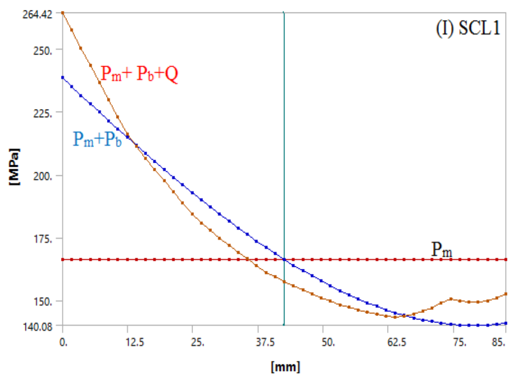


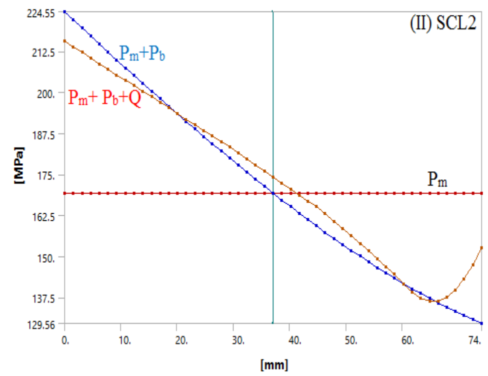
(b)

Fig. 8. Load case 4 (a) Equivalent stress (b) Stress Linearization Plot (I) for SCL1 (II) for SCL2



(a)





(b)

Fig. 9. Load case 5 (a) Equivalent stress (b) Stress Linearization Plot (I) for SCL1 (II) for SCL2

A. Validation of numerical simulation model

Model validation analysis was performed for the pressure vessel when subjected to internal pressure only. Theoretical value of hoop stress in the shell away from discontinuity (Pd_i /2T) was 135.13 MPa. Where d_i=Internal Diameter of shell, P=Design internal pressure, T=Shell thickness. The stresses in tangential direction (x) of ANSYS model in shell away from discontinuity was 132.14 Mpa. The comparison results between both methods were shown that FEA results were reliable and valid in this research with only 2.12% difference.

The calculated equivalent stress intensity distributions are shown in fig.5 to 9 section (a) under various loading condition. Load case 1(P=0) was the simulation for external loading environment under which a pressure vessel is subjected during their service life. Due to external loading outer surface of nozzle becomes critical at the junction. When pressure vessel is subjected to internal loading along with external loading, internal portion of nozzle becomes critical at the junction. It is observed that shell and nozzle junction area is subjected to maximum stress and leads to high probability of failure.

ANSYS program had stress linearization tool to obtain the stress component as membrane stress P_m and bending stress P_b distribution along any selected section. The Stress linearization results along SCL1 and SCL2 are shown in fig.5 to 9 section (b) under various loading condition. Stress P_m is constant and P_b varies along the material thickness.

B. Limits and verifications

According to ASME BPVC code [7], the allowable values for each type of stress (P_m, P_m+P_b, P+Q, P_m+P_b+Q) are derived from the basic allowable stress S at the working temperature. To stay within the scope of this work and to be coherent with the applied loading conditions, the stresses P_m, P_m+P_b and P_m+P_b+Q were verified. Where, Q=Peak Stress

The limits for verification are:

$P_m \leq S$ = 163 MPa;

$P_m + P_b \leq 1.5S$ = 244.5MPa and,

$P_m+P_b+Q \leq 3S$ = 489MPa

ASME code limit verification and the stress linearization results are tabulated in Table VII.

TABLE VII. STRESS LINEARIZATION RESULT FOR LOAD CASES ANALYZED AND LIMITS VERIFICATION

Stres-s	SCL	Load Case 1	2	3	4	5	Limit Verify
Pm	SCL1	11.18	123.81	137.83	128.92	166.24	Pm ≤S = 163 MPa
	SCL2	6.9414	121.21	137.22	152.34	169.26	
Pm + Pb	SCL1	19.903	170.17	192.91	185.37	238.55	Pm+ Pb ≤1.5Sm = 244.5MPa
	SCL2	17.886	160.97	182.05	210.28	224.55	
Pm + Pb +Q	SCL1	43.426	184.09	216.77	207.32	264.42	Pm+Pb+Q ≤3S = 489MPa
	SCL2	38.968	152.34	173.35	207.57	215.67	
Result		**Pass**	**Pass**	**Pass**	**Pass**	**Fail**	

The ASME code limits verification for analyzed loading condition shows that design is safe for internal design pressure ranges from 0 to 5MPa along with external loading.

VII. 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.

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.

[10] 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. 179-187,

[11] S. Chapuliot, S. Marie, Elastic-plastic fracture mechanics assessment of nozzle corners submitted to thermal shock loading. International journal of pressure vessels and piping, Vol. 147, 2016, pp. 55-63.

[12] Faisal M. Mukhtar, Husain J. Al-Gahtani, Design-focused stress analysis of cylindrical pressure vessels intersected by small-diameter nozzle. ASME Journal of pressure vessel technology, Vol. 139, 2017, pp.1-11.