Structural Analysis of Pressure Vessel and Nozzle under Internal Pressure

Priyanka S. Jadhav*, R. K. Patil

PG Student (M.E.), Department of Mechanical Engineering, SVERI's College of Engineering, Pandharpur, India
Assistant Professor, Department of Mechanical Engineering, SVERI's College of Engineering, Pandharpur, India

ABSTRACT

Due to functional requirement, Nozzle are require for pressure vessel. If these nozzle present on peak of the dish end do not disturb the symmetry of the vessel. However sometimes process requires that nozzles to be placed on the periphery of the pressure vessel. These nozzles disturb the symmetry of the vessel.[10]

Geometrical parameters of nozzle connections may significantly vary even in one pressure vessel. These nozzles cause geometric discontinuity of the vessel wall. So a stress concentration is created around the opening. The junction may fail due to these high stresses. Hence a detailed analysis is required.[8]

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.[4]

The presence of nozzle is going to hinder the radial expansion of the structure. Here we shall consider the one end of the nozzle fixed to an assembly. One Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Then to see what will be the effect on system stresses in that case.[7]

So that it need to analyzed in FEA & compare that value with analytical equation to understand effects of nozzle on Stress attributes of the vessel [4].

KEYWORDS: Junction failure,Normal Stress,Periphery of pressure vessel.

INTRODUCTION

Pressure vessels are widely applied in many branches of industry such as chemical and petroleum machine-building, nuclear and power engineering, gas, oil and oil-refining industries, aerospace techniques, etc. As the name implies these are important components of processing equipment. Nozzles or opening are necessary in the pressure vessels to satisfy certain requirements such as inlet or outlet connection, manholes, vents & drains etc. Welded nozzles connecting a pressure vessel to piping can be placed both on the cylindrical shell & heads of the vessel.Geometrical parameters of nozzle connections may significantly vary even in one pressure vessel. These nozzles cause geometric discontinuity of the vessel wall. So a stress concentration is created around the opening. The junction may fail due to these high stresses. Hence a detailed analysis is required. One of the parts of overall structural analysis for nozzle connections is the stress analysis of two intersecting shells. Due to different loadings applied to these structures, a local stress state of nozzle connection characterized by high stress concentration occurs in intersection region. Internal pressure is primary loading used in the structure analysis for determination of main vessel-nozzle connections. However the effect of external forces and moments applied to nozzle should be taken into consideration in addition to the stresses caused by the internal pressure. External loading usually are imposed by a piping system attached to the nozzle. Values of the loads & moments are calculated by an analysis of piping system .[10]

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 suggest a procedure to design the junction, but do not provide any methodology to calculate the extended and magnitude of these high stresses. So, there is need to carry out a detailed finite element analysis of the junction to calculate stresses at the junction & both in the vessel & in the nozzle. ANSYS package is used as a finite element tool .[6]

SCOPE FOR WORK

1) If nozzles are placed on the periphery of a pressure vessel, they disturb the axis symmetry of the system and cause eccentricity.
2)Sometimes this eccentricity can cause generation of a couple which can lead to a structural imbalance.
3) In a radial nozzle, the presence of nozzle is going to hinder the radial expansion of the structure. Here we shall consider the one end of the nozzle fixed to an assembly.
4) One Nozzle is going to cause an eccentricity, what will happen if there are more nozzles. Then to see what will be the effect on system stresses in that case.

NEED OF FEA:

1) To prevent stress related vessel rupture and catastrophic 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 cylindrical pressure vessels assembly and finite element engineering simulation of various stress and deformation tests at high temperature and pressure.
2) ASME code gives formulation to design pressure vessel with nozzles, however the Nozzles are at separate locations, such as dish end, side etc. However ASME code does not have clear guidelines for having Nozzles at the same height and how to counter the effect of the stresses.
3) Quench Nozzles specifically were patented in 2003, and hence they are a recent technology and this further means that standardized testing data is not available for them. Hence the only option is to use Design by Analysis approach prescribed by ASME.

OBJECTIVE:

1) Finite element analysis of the junction *to* calculate stresses at the junction of pressure vessel and nozzle is performed using ANSYS software. Analysis is done for to see effect of presence of nozzle on pressure vessel design.
2) Analysis is conducted for pressure vessel with presence of peripheral nozzle to determine the stress conditions with a peripheral nozzle & to determine the stress distribution in pressure vessel and nozzle.
3) To conduct structural analysis in ANSYS for pressure vessel by increasing number of nozzles.
4) To investigate the value of stress in the Nozzle N-13 of liquid Carbon drain Pressure Vessel.

ANALYTICAL FORMULA

The calculations for hoop stress in shell away from discontinuity due to internal pressure are as follows,

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

Where,

Ro = Outer radius of shell, mm.
Ri = Inner radius of shell, mm.
Pi = Internal pressure = 0.35 Mpa

FOR NOZZLE N13:

The calculations for stress due to internal pressure in Nozzle N13 is as,

Outer Diameter = 88.90 mm
Thickness = 11.125 mm
Ro = 44.45 mm
Ri = 33.325 mm

$$\sigma_\phi = \frac{0.35 * 33.325^{\,2}}{(44.45^{\,2} - 33.325^{\,2})}\left(1 + \frac{44.45^{\,2}}{33.325^{\,2}}\right)$$

After solving,

σ_ϕ= 1.2484 * 10^6 Pa
$\boldsymbol{\sigma_\phi}$ **= 1.2484 Mpa**

INPUT FEA

Material of construction:

Nozzle	SA 516 Gr. 70 N
Shell	SA 106 Gr. B

Table No. 3.1

DESIGN DATA

Design Code	ASME Sec-VIII Div-2
Internal Pressure	3.5 bar
External Pressure	Nil
Corrosion Allowance	0
Environmental Temp.	22 ^{0} C

Table No. 3.2

NOZZLE GEOMETRY:

Nozzle No.	**Nozzle Dimensions**		
	O.D(mm)	**I.D.(mm)**	**Thickness (mm)**
N-1	168.28	131.75	18.26
N-4	88.90	66.65	11.12
N-12	60.33	42.85	8.73
N-13	88.90	66.65	11.12

Table No. 3.3

MATERIAL DATA:

Property	**Shell**	**Nozzle**
Material	SA 516 Gr. 70 N	SA 106 Gr. B
Tensile Strength(Mpa)	460	415
Yield Stength (Mpa)	250	240
Young's Modulus (Mpa)	0.1996	0.1996
Poisson's Ratio	0.3	0.3
Density (kg/m^3)	7750	7750
Allowable Stress	153.33	138.33

Table No. 3.4

FEA ANALYSIS

In finite element analysis first solid work drawing is to be draw then model is to be imported into ANSYS model.

2D MODEL OF VESSEL

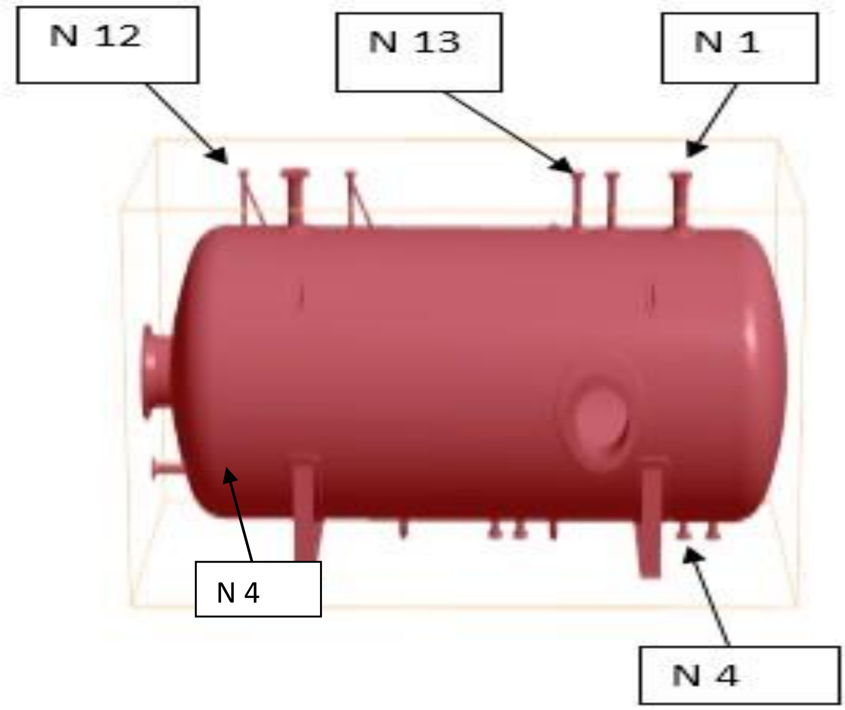

Fig. 1: Solid Model of Vessel with Nozzle

FEA 3D MODEL:

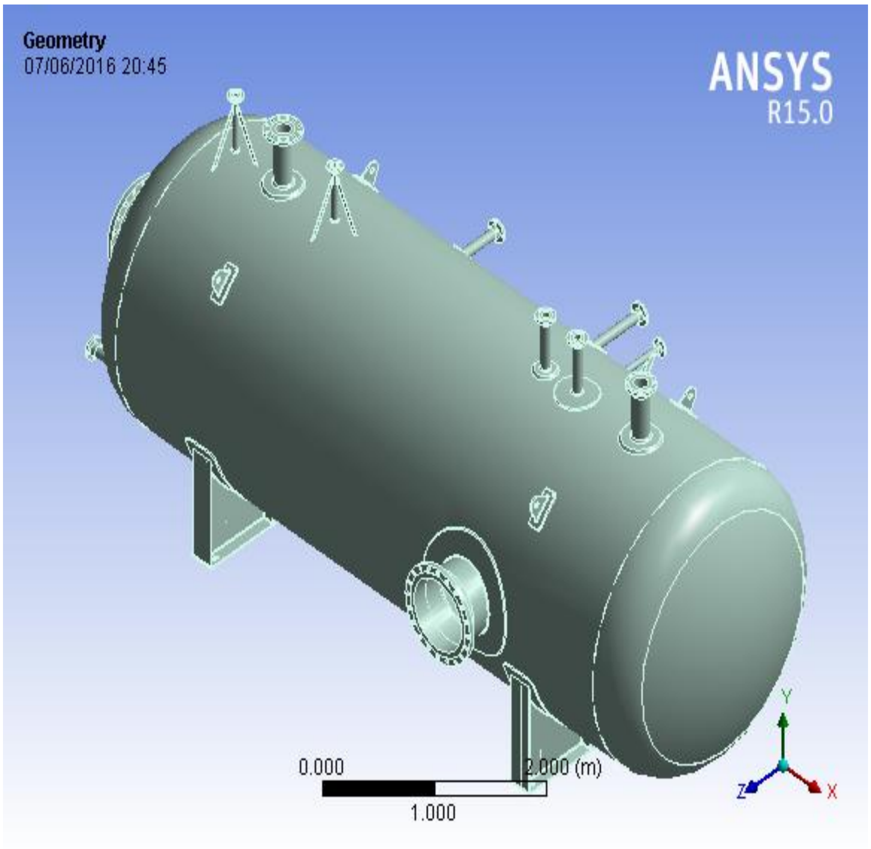

Fig. 2: 3D ANSYS MODEL

NORMAL STRESS PLOT:

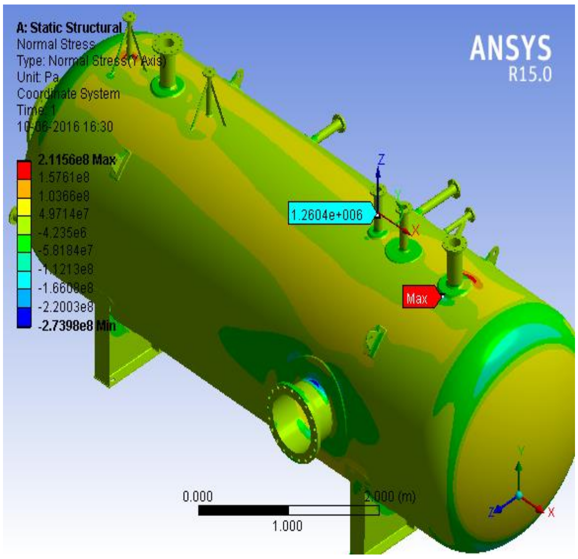

Fig. 3: Normal Stress Plot

RESULT & DISCUSSION

With the help of Analytical equation we can easily calculate the exact value of stress created in nozzle of a pressure vessel. With the help of numerical equation,mathematical results of a pressure vessel are compared with FEA analysis results. Thus, FEA Analysis is used for evaluating the stress level in the nozzle of pressure vessel with more accuracy.

Analytical Result

Circumferential stress or Hoop stress in Nozzle N-13 away from discontinuity is calculated as,

$$\sigma_{\phi} = 1.2484 \times 10^{6} \text{ Pa}$$

FEA Analysis Result

Normal stress in nozzle (N-13) away from discontinuity is evaluated by FEA Analysis with the help of Analysis software as shown in figure,

$$\sigma_{\phi} = 1.2604 \times 10^{6} \text{ Pa}$$

CONCLUSION

In this way a case study is to be performed on nozzle fitted on pressure vessel for stress analysis to be carried out as per the ASME code & from this analysis it is concluded that Analytical calculations & FEA Analysis results are approximately same. The purpose of analysis is to find out accurate& reliable results with the help of analysis software & to find out different Analytical & FEA Analysis methods using 3D solid work modeling software and post- processing methods for analysis of nozzle which is fitted on pressure vessels according to ASME codes. From this analysis, the mechanical design of a Nozzle & pressure vessel can be easily verified by a third party organization to ensure the quality of a pressure vessel system that it can be easily fulfills the requirement of the ASME code.

REFERANCES

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

[2] **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, 62-64Published Online November 2013.

[3] **Dharmit Thakore,"**Finite Element Analysis of Pressure Vessel and Nozzle Junction" Revision:Date:

29 March 2013.

[4] **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.

[5] **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.

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

[7] **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

[8] **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.

[9] **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.

[10] **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.