Numerical Analysis of Pressure Vessel Saddle Support Subjected to Various Loading Conditions

Harshana S[1], Anjali Sudhakar[2]

[1]M. TECH. (Structural Engineering) Student, Department of Civil Engineering, Younus College of Engineering and Technology, Kollam, Kerala, India
[2] Assistant Professor Department of Civil Engineering Younus College of Engineering and Technology, Kollam, Kerala, India

---***---

Abstract - *Pressure vessels are closed containers used to hold fluids at a pressure which is different from surrounding pressure. A cylindrical pressure vessel which is oriented horizontally is known as horizontal pressure vessel. This type of pressure vessel is supported by using saddle supports at two ends of the vessel. Therefore, the most severely stress affected area of a horizontal pressure vessel is the contact area between saddle support and the pressure vessel, which is named as saddle horn area and the major weight of this pressure vessel rests on its saddle support. This paper mainly focuses on numerical analysis of saddle support subjected to various loading conditions. The loading conditions considered here are the wind load, seismic load, internal pressure and its combinations and Finite Element Analysis (FEA) using ANSYS software is conducted. The saddle support is modelled and analyzed in ANSYS software. The results obtained from the study are utilized for further study.*

Key Words: Saddle support, Saddle horn area, Finite Element Analysis, ANSYS, ASME

1. INTRODUCTION

Pressure vessels are closed containers used to hold fluids at a pressure which is different from surrounding pressure. A cylindrical pressure vessel which is oriented horizontally is known as horizontal pressure vessel. This type of pressure vessel is supported by using saddle supports at two ends of the vessel. Therefore, the most severely stress affected area of a horizontal pressure vessel is the contact area between saddle support and the pressure vessel, which is named as saddle horn area and the major weight of this pressure vessel rests on its saddle support. The stress affected zones in saddle support due to wind load, seismic load, internal pressure and its combinations on saddle support is analyzed.

[1] The stress distributions at various parts of pressure vessel saddle support are calculated using ANSYS software. SA 516 Gr 70 material was used. Maximum localized stress arises at the saddle to vessel interface near the saddle horn area. The maximum value of stress at saddle horn is 206.84 MPa, which is below the safe value of 221 MPa given by Von Misses stress criteria for the present horizontal cylindrical vessel and its saddle support model.

[2] Details of saddle design were taken from Megyesy and analysis was done in ANSYS. The effect of changing the load & various geometric parameters were investigated. The highly stressed area beside the pressure vessel at the saddle horn is the flange plate. Value of 0.25 for ratio of distance from tangent line to saddle center to the length from tangent-to-tangent line of pressure vessel generates minimum stress. Slenderness ratio less than 16 found to generate minimum stresses.

[3] Existing design of saddle support is gathered from Design hub. Redesign 3D model of saddle support is constructed in SOLIDWORKS 2017. 3D model was redesigned by using different materials such as alloy steel, 1023 carbon steel sheet and AISI 347 annealed stainless steel and also by adding different number of ribs such as 2, 4, and 6. 10-ton force was applied to each saddle (20-ton force in total) and static analysis was conducted. Optimized model was simulated by Von Misses stress. The most suitable material for 6 ribs saddle support applied for future usage is AISI 347 stainless steel.

1.1 Objectives

- To carryout FEA of saddle support using ANSYS based on various loading conditions.
- To determine the stress affected zones in saddle support.

1.2 Scope of Work

- Welded connection for parts of saddle support is considered as bonded contact.
- Bolted connection for base plate is considered as fixed.
- Pressure vessel is not designed.

2. INPUT DATA

2.1 Model

A three ribbed model of saddle support is developed in ANSYS on the basis of Process Industry Practices using following details:

Vessel diameter (D) – 2210 mm
Vessel mean radius (R_m) – 1111 mm
No. of saddles (n) – 2
Wear angle - 120°
Web angle - 120°
Bolt hole diameter – 25.4 mm
Spacing between saddle – 971.5 mm

Table – 1: Model details

Parts	Base plate	Web plate	Wear plate	Rib plate
Length (mm)	2008	1982	-	-
Width (mm)	230	204	254	204
Thickness (mm)	13	13	10	13

2.2 Loading

The loads assigned to pressure vessel are as follows:

Wind load (P1):
Wind design code – ASCE 7-98/2002
Wind pressure – 1.4545 kPa

Internal pressure (P2):
Based on ASME Section VIII division 1 code
Internal pressure – 0.345 MPa

Seismic load (P3):
Seismic design code – ASCE /SEI 7-05
Seismic load – 31290.62495 N

Load combination:
P – P1 + P2 + P3

2.3 Material

Table-2 Material properties

Elements	Saddle, shell & head	Bolts
Material	SA 516 Gr 70	SA 193 Gr B7M
Modulus of Elasticity (GPa)	198.373	200.373
Poisson's Ratio	0.3	0.3
Density (kg/m³)	7750	7750
Yield strength (MPa)	260	550
Allowable stress limit (MPa)	138	138

3. ANALYSIS

Stress due to wind pressure, seismic load, internal pressure and load combinations in saddle support is analyzed and stress affected zones in saddle support were identified for further study.

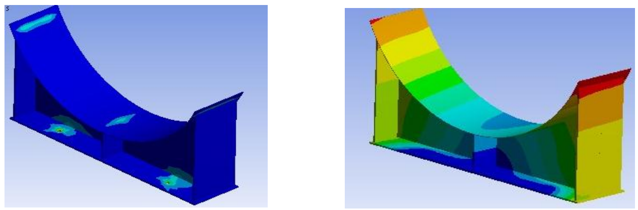

Fig -1: Stress & deformation corresponding to wind load

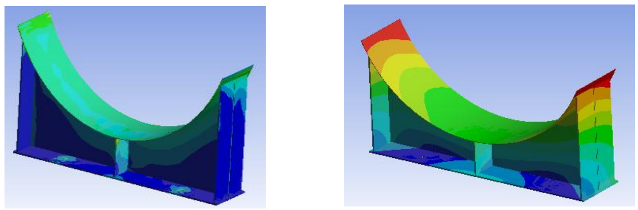

Fig -2: Stress & deformation corresponding to internal pressure

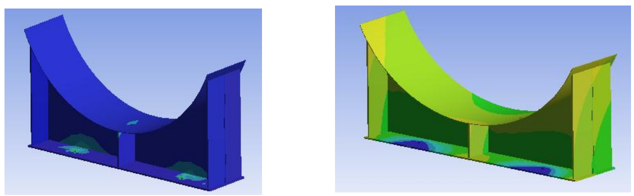

Fig -3: Stress & deformation corresponding to seismic pressure

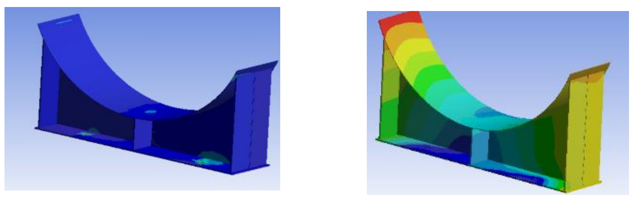

Fig -4: Stress & deformation corresponding to load combinations

Table – 3: Result of Analysis

Load	Stress (MPa)	Deformation (mm)
Wind load	137.44	0.68014
Internal pressure	53.738	0.25915
Seismic pressure	47.11	0.15435
Load combinations	168.07	0.92524

4. CONCLUSION

- FEA of saddle support using ANSYS based on various loading conditions was done.
- The stress affected zones in saddle support were found to be at the regions of bolt hole, edges of wear plate and at the center of wear plate.
- These results are utilized for further study.

5. SCOPE FOR FUTURE WORK

- The stress affected zones can be checked for linearized stress conditions.
- The linearized stress values can be checked for safety conditions as per ASME Section VIII code.
- The saddle support can be optimized using different parameters to reduce the weight of saddle support.
- The optimized model can be checked for transportation loads.

REFERENCES

[1] Kumar N, Angra S, Mittal VK. Support analysis of horizontal pressure vessel using FEA. In Applied Mechanics and Materials 2014 (Vol. 592, pp. 1220-1224). Trans Tech Publications Ltd.

[2] Khan SM. Stress distributions in a horizontal pressure vessel and the saddle supports. International Journal of Pressure Vessels and Piping. 2010 May 1;87(5):239-44.

[3] Rosli AH, Abdullah MA. Design Optimization of Saddle Support for The High-Capacity Pressure Vessel Using Finite Element Analysis. Progress in Engineering Application and Technology. 2021 Dec 5;2(2):1079-89.

[4] Adithya M, Patnaik MM. Finite Element Analysis of Horizontal Reactor Pressure Vessel Supported on Saddles. International Journal of Innovative Research in Science, Engineering and Technology. 2013 Jul;2(7):3213-20.

[5] Agale MC, Awachat MP. Finite Element Analysis of Horizontal Pressure Vessels Saddle. International Journal of Innovations in Engineering and Science. 2017;2(2).

[6] Mackenzie D. Stress linearization concepts and restrictions in elastic design by analysis. InPressure Vessels and Piping Conference 2017 Jul 16 (Vol. 57908, p. V01AT01A039). American Society of Mechanical Engineers.

[7] Arunkumar S, Moorthy PE, Karthik N. Design optimization of horizontal pressure vessel. Materials Today: Proceedings. 2020 Jan 1;26:1526-31.

[8] Wadkar VV, Malgave SS, Patil DD, Bhore HS, Gavade PP. Design and analysis of pressure vessel using ansys. J. Mech. Eng. Technol. 2015;3(2):1-3.

[9] Salins SS, Mohan M, Stephen C. Finite Element Investigation on the Performance of Pressure Vessel Subjected to Structural Load. InAnnales de Chimie-Science des Matériaux 2021 Jun (Vol. 45, No. 3, pp. 201-205).

[10] Nayak A, Singru P. Study of Effect of Angle of Contact and Angle of Extension of Wear Plate on Maximum Stress Induced in Horizontal Pressure Vessel. InAdvances in Industrial Machines and Mechanisms 2021 (pp. 511-519). Springer, Singapore.