"EFFECT OF CHANGE IN LOCATION OF ESSENTIAL ELEMENTS OF SADDLE SUPPORTS"

Akash Gawde[1], Mrugank Divekar [2], Tanveersingh Dhanjal[3], Umesh Tendulkar[4], Preethakumari[5]

[1,2,3,4]*Student, Department of Mechanical Engineering, Lokmanya Tilak College of Engineering, Maharashtra, India.*
[5]*Professor, Department of Mechanical Engineering, Lokmanya Tilak College of Engineering, Maharashtra, India.*

Abstract – *Pressure vessels are subjected to various types of loadings viz. internal / external pressure, operating conditions thermal loadings, nozzle loadings, hydro test pressure, wind and seismic loadings, etc. These loads are transferred to the foundation through saddle supports. Thus, in addition to pressure boundary, one of the important elements in the construction is saddle supports. The structural analysis of supports is being carried out for two configurations and results compared.*

Key Words: **Pressure vessel, saddle supports, ANSYS, FEA, stress analysis, design of saddle**

1. INTRODUCTION

Pressure vessel is an enclosed container in which a fluid, in the form of liquid or gas, is stored under the desired pressure. In addition to the internal pressure, external forces due to wind, seismic, piping, dead loads, etc. are to be considered for design. Furthermore, hydrostatic tests are to be performed at higher pressure to ensure that the pressure vessel could withstand the desired loads. Pressure vessels are supported on support systems, which must withstand the above-mentioned loads. Therefore, stress analysis must be performed on the supports to check the safety of the design. Finite Element Analysis (FEM) enables simulation of the theoretical loads acting on the pressure vessel and thus helps to optimize and validate the design. In the present work, supports were modelled on SOLIDWORKS and the analysis were performed on ANSYS 18.1 simulation software.

2. LITERATURE REVIEW

L. P. Zick (1951)[1] presented a study in which he discussed various stresses acting in cylindrical vessels. Using these stresses, it is possible to determine which pressure vessels must be designed only based on internal pressure. It also helps to develop stiffening rings for those requiring it.

Shen Naijie (1995)[2] found out the stresses in the saddle supports of pressure vessels by experimental and theoretical analysis, using electric strain gauge and double Fourier series expansion method. Also, a trial and error method has been proposed to determine the contact pressure distribution pattern.

N.EL-Abbasi(2001)[3] In this research, a three-dimensional finite-element analysis of the pressure vessel resting on a flexible saddle framework was created. It evaluates and addresses the effects of saddle length, saddle width, plate extension, and support overhang on the resulting stress fields in the vessel and the support.

Shafique M. A. Khan (2010)[4] The stress distribution was discovered in various parts of the saddle, such as wear, network, flange and base plate, using 3D finite element analysis. Based on the optimum values of the support distance ratio from the end of the vessel, the effects of load shift and various geometric parameters were studied, and recommendations were made.

3. DESIGN OF PRESSURE VESSEL AND SADDLE SUPPORTS

The design of pressure vessel were carried out on Solidworks and analyzed on ANSYS.

3.1 Design of Pressure Vessel

The pressure vessel considered in our analysis is meant to carry LPG as its working fluid, with the following general dimensions-

Table -1

Dimensions of pressure vessel	
Shell outside diameter, D	2133.6 mm
Shell length L	5000 mm
Spherical head outside diameter	2133.6 mm
Corrosion allowance	1.28 mm
Thickness	91.8 mm

The material used for pressure vessel and saddle is SA-516 GR. 70 with the following properties –

Table -2

Properties of Saddle Material	
Material	SA-516 GR.70
Density	7750 kg/m^3
Modulus of Elasticity	1.92E+11 N/m^2
Poisson ratio	0.3
Yield Strength	260 MPa
Operating pressure	1.69 MPa
Design Pressure	6.8 MPa
Operating temperature	297 K

The design, cross section and model of the pressure vessel are as shown below -

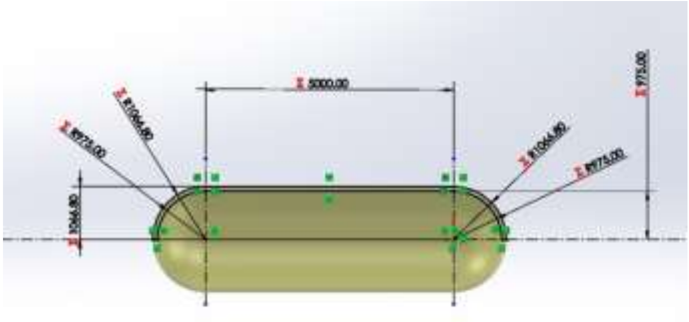

Fig -1: Dimensions of CAD model of pressure vessel

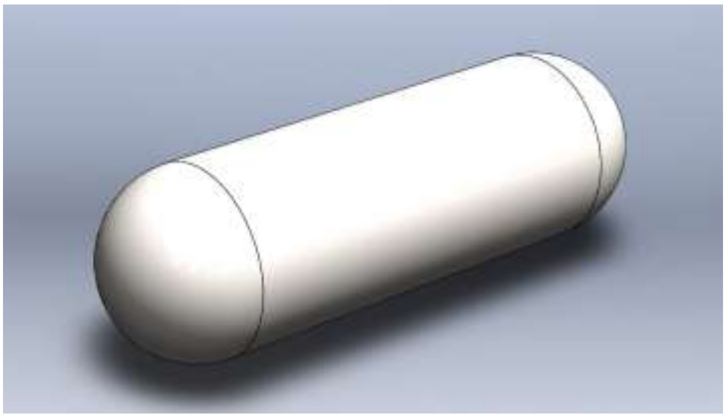

Fig -2: CAD model of pressure vessel

3.2 Design of Saddle

The dimensions for saddle were taken from the Dennis Moss book (2013)[5].

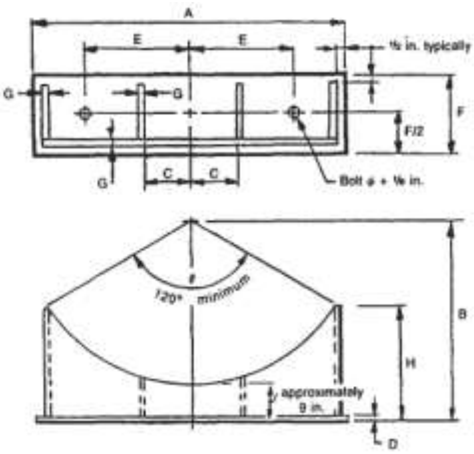

Fig -3: Saddle Dimensions

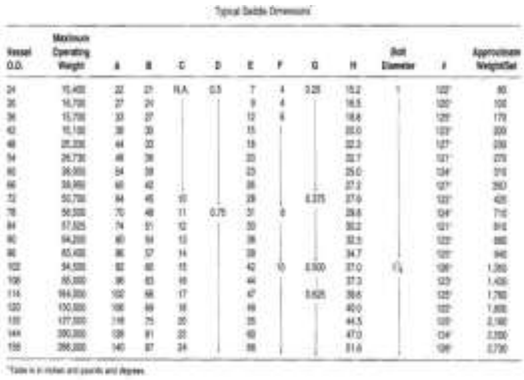

Fig -4: Typical Saddle Dimensions

Based on the above table, we have selected 84-inch diameter pressure vessel and designed the saddle according to the dimensions given. In order to have further clarity, the table given below has been prepared-

Table -3

Typical Saddle Dimensions	
Vessel O.D.	84 inches
Maximum Operating Weight	57525 pounds
A	74 inches
B	51 inches
C	12 inches
D	0.75 inches
E	33 inches
F	8 inches
G	0.375 inches
H	30.2 inches
Bolt Diameter	1.25 inches
Saddle Angle (Θ)	121°
Approximate Weight / Set	810 pounds

The saddle modeled in Solidworks is as shown below.

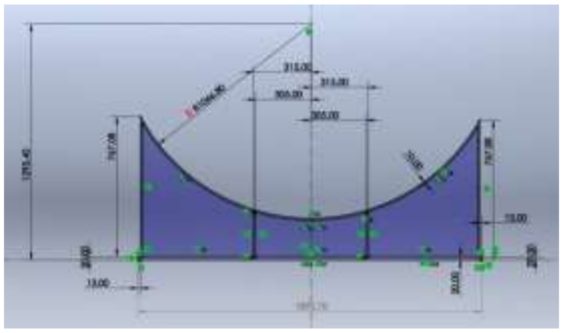

Fig -5: Dimensions of saddle in CAD

3.3 Forces and Boundary Conditions

The forces considered for analyzing the saddle are as follows-

i) Gravitational Force

The primary force that a saddle has to bear is the force due to its own weight. The mass of the saddle came out to be 237 kg.

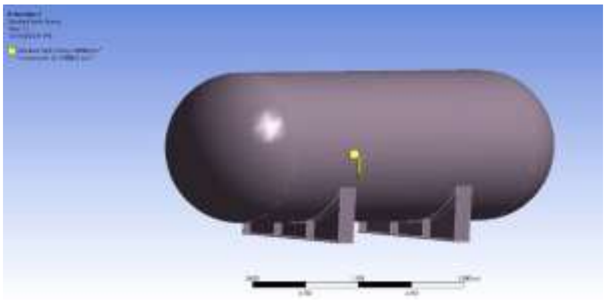

Fig -6: Gravitational force

ii) Pressure Force

The pressure vessel was designed for operating at 1.69 MPa pressure. The force due to the expansion of the vessel due to the internal pressure acts on the saddle.

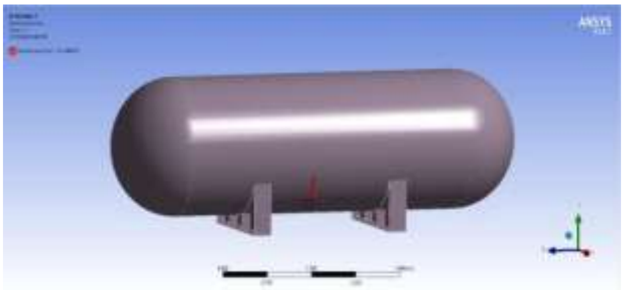

Fig -7: Pressure Force

iii) Wind Loads

Wind load was calculated for the Zone-4 structure in India, which came out to be 20 kN in longitudinal direction and 5 kN in lateral direction.

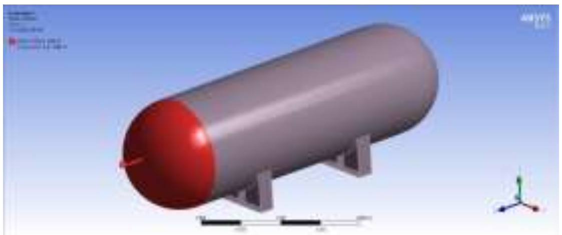

Fig -8: Wind load in lateral direction

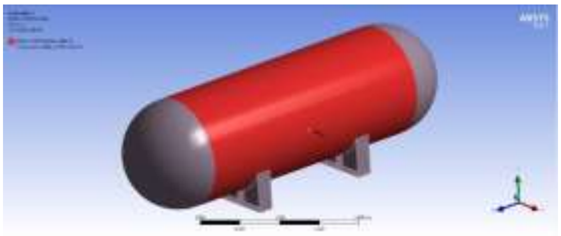

Fig -9: Wind load in longitudinal direction

iv) Constraints

Both of the saddles were fixed at their base to simulate no movement in the saddles.

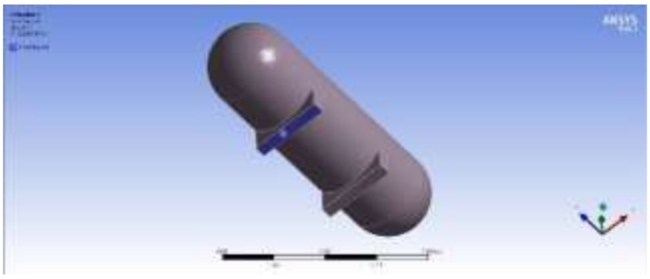

Fig -10: Fixed Support (Saddle 1)

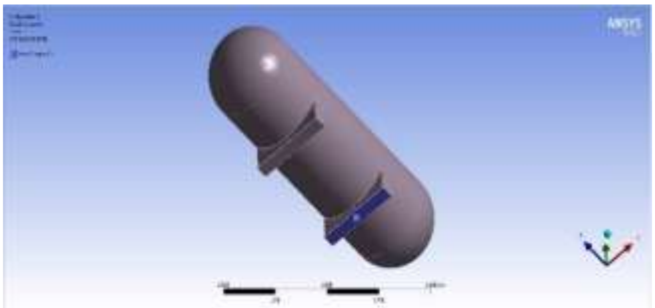

Fig -11: Fixed Support (Saddle 2)

4. ANALYSIS AND ITERATIONS

I) Iteration 1

As the first iteration, the saddle was designed with two vertical ribs on the sides and one web at the centre, which is considered as the baseline design. The saddle and its cross section are shown below.

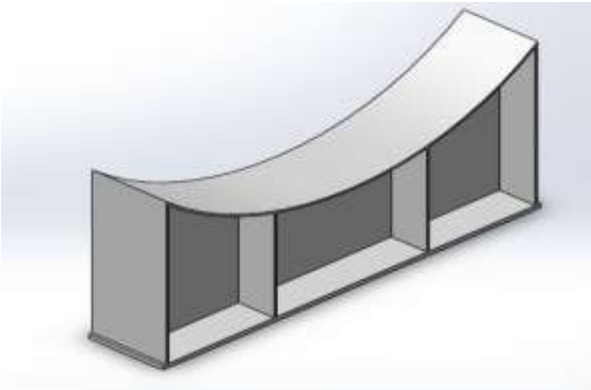

Fig -12: CAD model of saddle (Iteration 1)

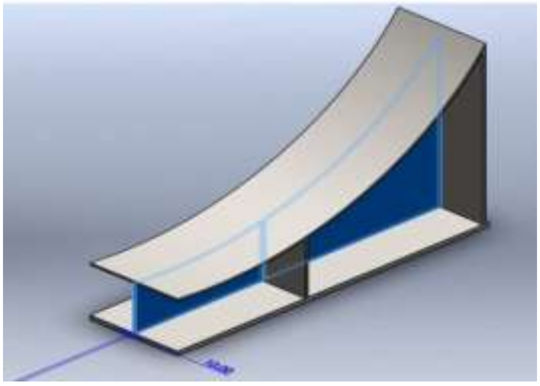

Fig -13: Section CAD model of saddle (Iteration 1)

Analysis of the structure using ANSYS indicated maximum Von Mises stress of 31 MPa in the saddle.

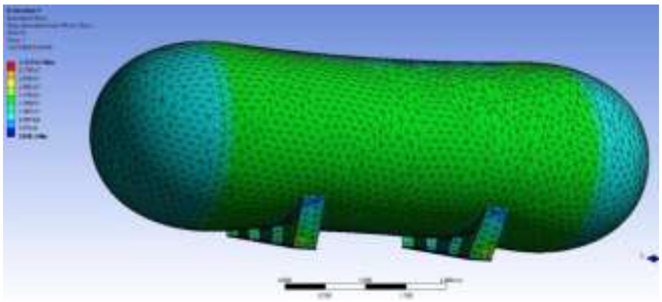

Fig -14: Von Mises Stresses of Iteration 1

II) Iteration 2

In the second iteration, the saddle was designed with two side webs without the inner centre web. The two centre ribs were retained the saddle and its cut section are shown below.

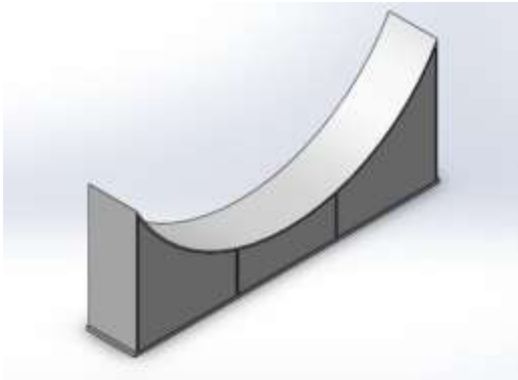

Fig -15: CAD model of saddle (Iteration 2)

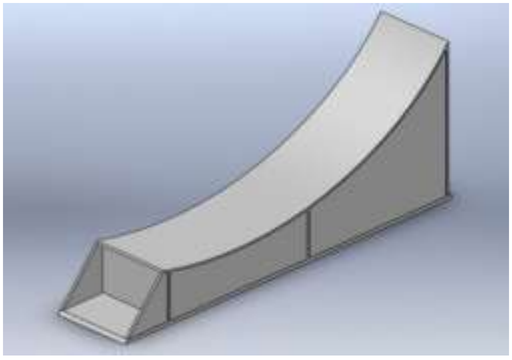

Fig -16: Section CAD model of saddle (Iteration 2)

Analysis of this structure indicated maximum Von Mises stress of 36 MPa in the saddle.

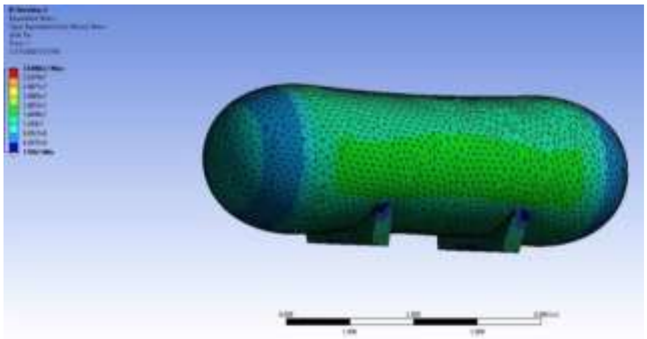

Fig -17: Von Mises Stresses of Iteration 2

III) Iteration 3

In the third iteration, the saddle was designed with two side webs only, without any ribs. This was like a box type structure. The saddle and its cut section are shown below.

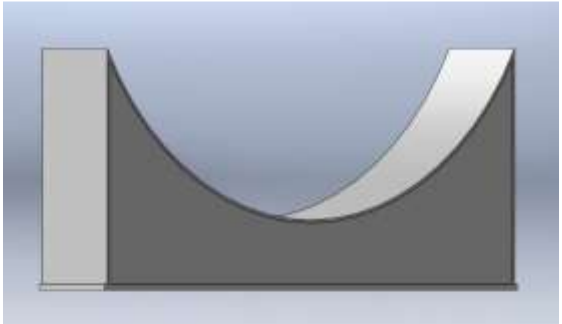

Fig -18: CAD model of saddle (Iteration 3)

Fig -19: Section CAD model of saddle (Iteration 3)

The maximum Von Mises stress was found to be 40 MPa in the saddle.

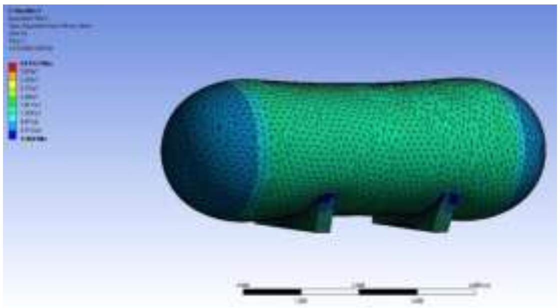

Fig -20: Von Mises Stresses of Iteration 3

5. CONCLUSION

The following are the observations from the ANSYS analysis of different saddles.

Table -4

Saddle Optimization Results		
Iterations	Weight (kg)	Factor of Safety
1 (Baseline)	237	8.39
2	289	7.20
3	279	6.38

As the initial design was over-designed with a factor of safety greater than 8, the design was modified. Although we could reduce the factor of safety to considerable levels, the weight of the saddle increased in subsequent iterations. Therefore, further modifications are necessary in the design for reducing the weight of the saddle. The modifications can be in the form of providing patterned gaps or holes in the low stressed areas.

ACKNOWLEDGEMENT

We are extremely grateful to Mr. Punit Ravani (Manager-Design & Engg., L&T) for his continued support and for providing invaluable guidance to us.

REFERENCES

[1] Zick, L. P. "Stresses in large horizontal cylindrical pressure vessels on two saddle supports." Welding Journal Research Supplement 30, no. 9 (1951): 435-445

[2]Shen Naijie Zhu Jitao, Lu Wenge "Stress state in the saddle zone of pressure vessels and piping", International Journal of Pressure Vessels and Piping Volume 63, Issue 2, 1995, Pages 155-164

[3]N El-Abbasi SA Meguid, A Czekanski "Three-dimensional finite element analysis of saddle supported pressure vessels", International Journal Of Mechanical Sciences. - 5 : Vol. 43.

[4] Khan, Shafique. "Stress distributions in a horizontal pressure vessel and the saddle supports." International Journal of Pressure Vessels and Piping 87, no. 5 (2010): 239-244

[5] Dennis R. Moss, Michael Basi, Pressure Vessel Design Manual fourth edition, 2013.

[6] ASME Boiler and Pressure Vessel Code. Section VIII. Pressure Vessels Division, 2. New York: ASME; (1989)

BIOGRAPHIES

Akash Gawde, pursuing B.E. in Mechanical Engineering from Lokmanya Tilak College of Engineering affiliated to Mumbai University.

Mrugank Divekar, pursuing B.E. in Mechanical Engineering from Lokmanya Tilak College of Engineering affiliated to Mumbai University.

Tanveersingh Dhanjal, pursuing B.E. in Mechanical Engineering from Lokmanya Tilak College of Engineering affiliated to Mumbai University.

Umesh Tendulkar, pursuing B.E. in Mechanical Engineering from Lokmanya Tilak College Of Engineering affiliated to Mumbai University.

Preethakumari,
Professor, Department of Mechanical Engineering, Lokmanya Tilak College of Engineering.