International Journal of
Engineering Science and Futuristic Technology
A Peer-reviewed journal

ISSN : 2454-1338(O), ISSN : 2454-1125(P)
Impact Factor : 3.885 (Calculated by I2OR)
IJESFT 17 (2016) 001-010

Static and Dynamic Analysis of Pressure Vessel with Vertical orientation using PVElite

Vipul J. Solanki [1*], Prof.D.A.Patel [2]

[1]PG student, Mechanical Engineering Department, SPCE Visnagar, GTU, Gujarat, India

[2]Associate Professor, Mechanical Engineering Department, SPCE Visnagar, GTU, Gujarat, India

A B S T R A C T : Pressure vessels are widely used in private sectors and industries all over the world. Its safety and quality is major issues to avoid the failure of the system. In this paper static and dynamic analysis of pressure vessel with vertical orientation is carried out. Vertical orientation pressure vessel is 6m in height and support height is above 1m. Commercial software PVElite is used to analyses the static behavior of pressure vessel under internal pressure and dynamic behavior is also studied. To understand the dynamic behavior of the pressure vessel in actual conditions response spectrum analysis is performed in mastering software ANSYS 15.0. Manual calculation is carried out for seismic evaluation based on Indian Standards Codes IS: 1893 to validate the results.

Keywords: Vertical Pressure Vessel, IS: 1893 RSM, PVElite 2008, Response Spectrum Analysis

1. Introduction

Pressure vessels are the container for fluid and gases under high pressure. Due to high pressure, stresses are induced in pressure vessel, if this stresses are more than the permissible stresses then the failure of pressure vessel occurs. The wind and seismic design is a major requirement in oil and gas industries equipment design. Normally oil and gas plants are away from the residential area. However if heavy earthquakes or wind emerge, that will cause disaster due to failure of equipment as these plants are handling many harmful toxic fluids and gases. Many disasters have happened all over the world in these industries due to excess wind and seismic effects. So it is necessary to manufacture pressure vessels under standard codes.

2. Literature Survey

2.1. Static Analysis

Authors have done the mechanical design of pressure vessel using graphical based software named PVElite [1]. Analyses were carried out on head, shell, nozzle and saddle. The input parameters are type of material,

Corresponding author e-mail: vipulsolanki.mech@gmail.com
Tel.: +91 8154893978
Journal access: www.adpublication.org

Nomenclature

I	Moment of inertia
F	Force or weight of vessel
δ	Deflection of pressure vessel system
f	Natural frequency of vessel system
A_h	Design horizontal acceleration spectrum value

pressure, temperature, and diameter and corrosion allowance. Analysis performed the calculations of internal and external pressure, weight of element, allowable stresses, vessel longitudinal stress check, nozzle check and saddle check. The same design method is applied in paper [2] for local stress analysis using PVElite according to welding research council WRC 107. The high stresses at intersections are caused by discontinuity shear stresses and moments which exist to maintain compatibility at the junctions. Research can be explored to into account other parameters. Selection material referring to ASME, standard can also been developed. There are other parameters that are not considered such as thermal loads, wind loads, seismic loads, transportation load, erection load and fabrication methods etc. this insufficiency can be overcome by mastering software like ANSYS. Another probabilistic design of pressure vessels using ANSYS PDS feature is carried out in paper [3]. Probabilistic design uses Gaussian distribution for various input parameters and simulation uses Monte Carlo simulation technique for sampling. Monte Carlo simulation approach and analysis was looped through 100 samples points considering the variations defined in the input variables and the corresponding statistical analysis of the output parameters.

2.2. Dynamic Analysis

The wind and seismic design is a major requirement in oil and gas industries equipment design. Normally oil and gas plants are away from the residential area. However if heavy earthquakes or wind emerge, that will cause disaster due to failure of equipment as these plants are handling many harmful toxic fluids and gases. The seismic design based only on analytical approach does not give exact solution all the time and it is not suitable for all types of vessel geometry. [7]The FEM is required with analytical approach to ensure the design is safe. They found that the passed analytical saddle design is failing during FEM simulation and it requires more thickness to with stand the external forces based on FEM. Accordingly thickness has been improved as 16mm and the FEM was redone and found that the saddle is safe with 16mm shell thickness. Here further fabricator can use external or internal stiffening rings wherever required for saving the vessel from excess external and internal loading conditions. The behavior of a structure system in an earthquake or seismic is one of vibration under variable conditions of acceleration [8]. The usual normal simplified approach to the problem is based on the assumption that the structural system is a rigid body which undergoes the accelerations of the supporting ground on which it's standing. The horizontal force which acts on the structure are equal to its mass times the ground acceleration, and has the same ratio to the weight as the ground acceleration has to that of gravity [9]. These structural codes give values of this ratio that are based on engineering experience and judgment.

3. Objective of paper

The salient points of the proposed problem are as follows:

- Static and dynamic analysis is carried out in commercial software named PVElite. Stress values are observed for vertical air receiver due to applied input pressure in structure. Different load combinations are performed in PVElite software and governing analysis is presented for existing problem.

- To identify the design capacity level of pressure vessel structure and maximum allowable lateral force based on shear coefficients of seismic loadings by perform the dynamic analysis.

- To specify the damage level of pressure vessel structure by performs the response spectrum analysis in ANSYS 15.0. IS: 1893 codes are performed to evaluate the seismic response of structure.
- To obtained the natural frequency response of structure with mathematical and software approach.

4. Project Approach

In this paper, commercial software PVElite is used to generate the stresses due to the internal pressure and earthquake load on the pressure vessel structure. PVElite is mostly used customized software for vessels like pressure vessel, heat exchanger, tanks and boiler, to evaluate stresses. This stresses than validate with the result obtained from the mathematical calculation and mastering software ANSYS 15.0. In ANSYS 15.0 the static analysis is carried out and results are presented. To understand the dynamic behavior of pressure vessel response spectrum modal analysis is carried out and results are generated.

The work is done on the case study given by Air and Gas Process Technologies Pvt. Ltd. Vatva GIDC, Ahmedabad. This work is based on the vertical orientation pressure vessel of air receiver. Company gave the details of current vertical pressure vessel of air receiver. The model of air receiver is made in PVElite software. PVElite is customized software for vessels like heat exchangers, pressure vessel, boilers, tanks etc.

5. Mathematical Approach

5.1. Frequency calculation

Loading condition of vertical pressure vessel under seismic forces is similar to that of a cantilever beam when the load increases uniformly toward the end.

Step 1: Moment of inertia

$$I = \frac{\Pi}{4}\left(r_o^4 - r_i^4\right) \qquad I = 0.01833 m^4$$

Step 2: Force or weight of vessel

$$P = \frac{F}{A} \qquad F = 141130.44 N$$

Step 3: Deflection of pressure vessel system

$$\delta = \frac{\omega \cdot L^3}{8 \cdot E \cdot I} \qquad \delta = 1.19 \times 10^{-3} m$$

Step 4: Natural frequency of vessel system

$$f = \frac{1}{2\Pi}\sqrt{\frac{g}{\delta}} \qquad f = 14.44 Hz$$

Hence, the natural frequency of vessel system is found to be 14.44 Hz by mathematical approach. Vertical distribution of lateral force at each level is presented here with mathematical calculation as well as stress calculation in vessel structure is presented.

5.2. Mathematical approach for seismic calculations is carried out based on Indian Standard criteria for earthquake resistant design of structures PART 1 GENERAL PROVISIONS AND BUILDINGS (Fifth Revision)

Lateral Load Distribution with Height by the Static Method:

Vessel location: Vatva, Ahmedabad, Gujarat, India

The approximate fundamental natural period of vibration (T), in seconds, of a moment-resisting frame building without brick panels may be estimated by the empirical expression: for steel frame building.

$$T_a = 0.085 \times h^{0.75}$$ (Ref. clause 7.6.1 IS: 1893)

$$T_a = 0.32\,seconds$$

Design seismic base shear,

$$V_b = A_h \times W \qquad V_b = 5820.53Kg$$

Where,

$W = \sum W_i$,

A_h = Design horizontal acceleration spectrum value as per 6.4.2 (IS: 1893), using the fundamental natural period T, as per 7.6 in the considered direction of vibration.

Table 1: Input parameters for seismic data

Sr. No.	Parameters	Value
1	Zone number	III
2	Zone factor	0.16
3	Soil type	Medium soil
4	Period of vibration	0.32 seconds
5	Value of S_a/g	2.50
6	Damping factor	2.00
7	Force factor	4.00
8	Importance factor	1.00

Table 2: Vertical distribution of design force at each level

Level	w_i (N)	h_i (m)	$w_i h_i^2$	$\frac{w_i h_i^2}{\sum w_i h_i^2}$	Lateral force at each level (N) $V_b \frac{w_i h_i^2}{\sum w_i h_i^2}$
6	13894	6	68.18	0.052	303
5	22.903	5	572.57	0.4312	2510
4	22.903	4	366.45	0.2759	1606
3	22.903	3	206.13	0.1552	903
2	22.903	2	91.61	0.0689	401
1	22.903	1	22.903	0.0172	100
		Total	1327.8		

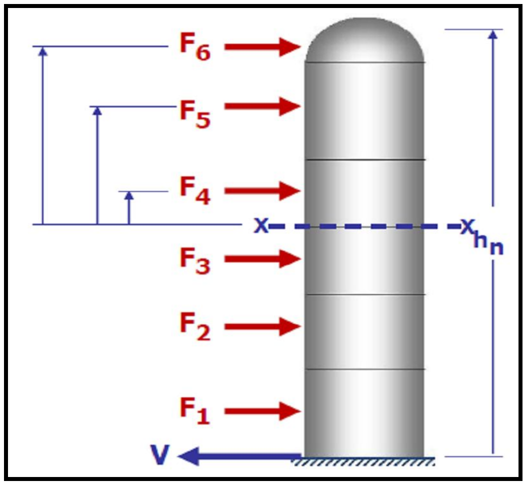

Fig.1 Diagram for Lateral force in vessel at different level

6. Software Approach

6.1. PVElite Software

PVElite is commercial software for shells like heat exchanger, tanks and boiler. The modal of pressure vessel with vertical orientation is made in PVElite 2008. The results of stresses due to internal pressure are calculated from program. Moreover, combined load cases are carried out with different load combinations and results are generated.

Table 3: Inputs parameter for vertical pressure vessel

Components	Parameters	Units
Vessel	Material	SA 516 Gr.60
	Outside diameter	2500mm
	Thickness	8mm
	Height	6000mm
	C.A.	1.5mm
Heads	Material	SA 516 Gr.60
	Type	Ellipsoidal heads
Inlet nozzle 1	Size	400mm
	Schedule	40mm
Outlet nozzle 2	Size	400mm
	Schedule	40mm
Outlet nozzle 3	Size	350mm
	Schedule	40mm
Outlet nozzle 4	Size	250mm
	Schedule	40mm
Manhole	Size	400mm
	Schedule	40mm
Support	Material	SA 516 Gr.60
	Type	Skirt

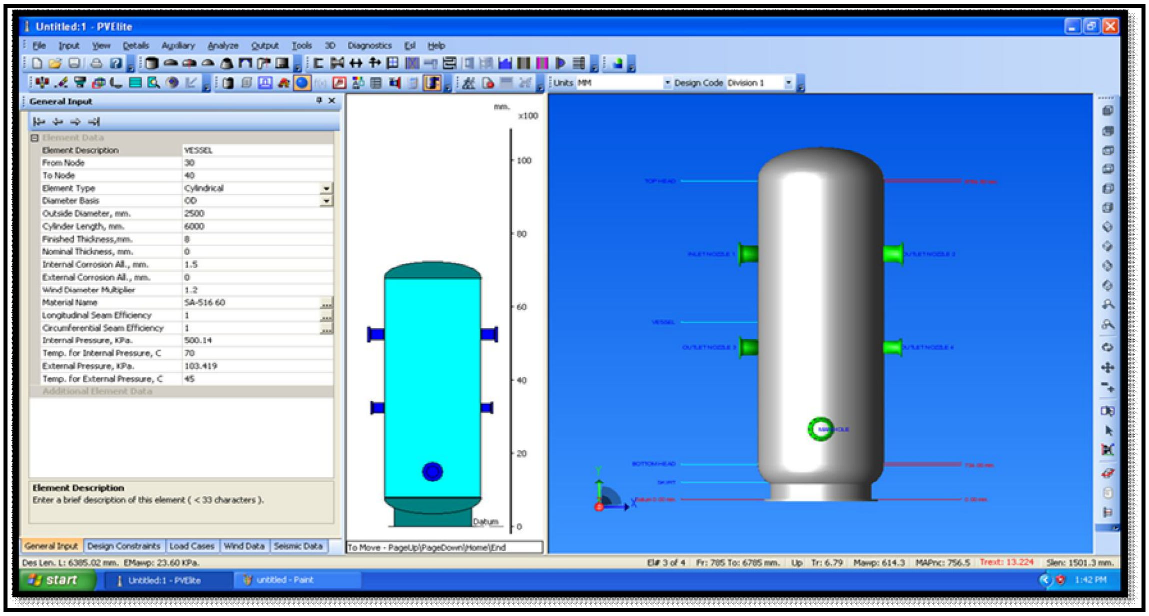

Fig.2 3D model of Vertical pressure vessel in PVElite 2008

6.1.1. Different Load Combinations result

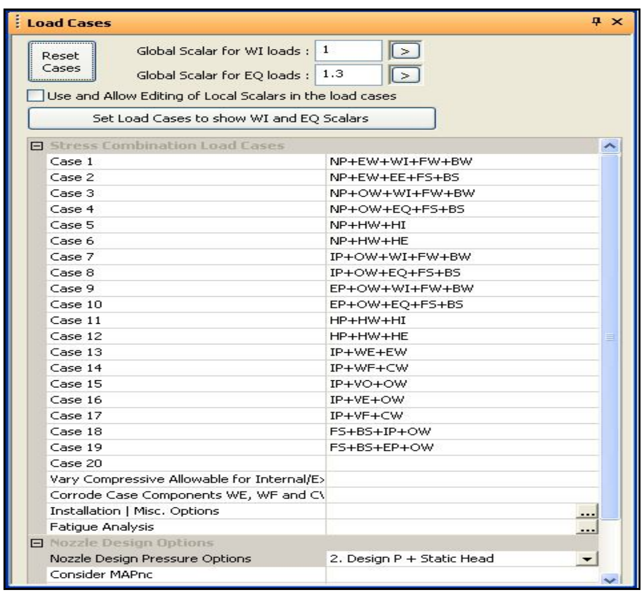

Fig.3 Different load combinations in PVElite

Governing Element: TOP HEAD
Governing Load Case 18: FS+BS+IP+OW
Absolute Maximum of the all of the Stress Ratio's = 0.8282

Table 4: *Governing Load case 18: FS+BS+IP+OW*

From Node	Tensile stress	Allowable Tensile Stress	Compressive stress	Allowable compressive stress	Tensile ratio
10	-	117.90	-1.35	-78.66	-
20	97.06	117.90	-	-64.99	0.8232
30	97.15	117.90	-	-64.99	0.8240
40	97.65	117.90	-	-64.99	0.8282

6.2. ANSYS 15.0 Software

6.2.1. Modal response spectrum analysis

We apply modal analysis to conclude the vibration characteristics like mode shapes and natural frequencies of a construction component while it is being planned. It also can be starting points for another analysis like dynamic analysis, such as a spectrum analysis, transient dynamic analysis, or a harmonic response analysis. The mode shapes and natural frequencies are vital parameters in the design of a structure for dynamic load environment. It is also required if we want to do a spectrum analysis or a mode superposition transient or harmonic analysis. Modal analysis is done on a pre-stressed construction. Modal analysis in the ANSYS relations of parts is a linear analysis.

A response spectrum analysis in which the outcome of a modal analysis are used to recognized spectrum for estimate stresses and displacements in the model. It is mostly used in position of a time-history analysis to establish the response of systems to time-dependent or random loading situation like seismic, ocean wave loads, wind loads, rocket motor vibrations, and jet engine thrust and so on.

The method for a Single-point response spectrum analysis done by following steps:

1. Set up the model. 2. Apply loads and find the solution. 3. Find the Model result. 4. Find the spectrum result.

5. Develop the Modes. 6. Merge the Modes. 7. Review the results.

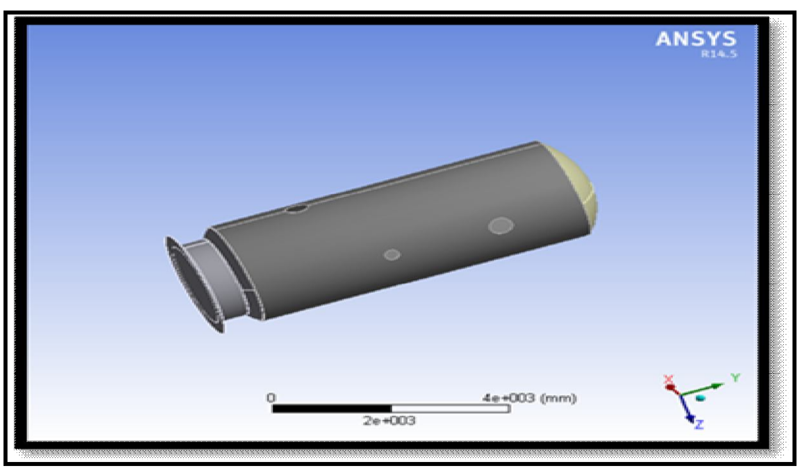

Fig.4 3D model in ANSYS Workbench 15.0

Meshing Details:

Name of Element = 1 x 1 Square

No. of nodes = 47504

No. of elements = 23627

6.2.2. Result of frequency response

Frequency value = 10.568 Hz

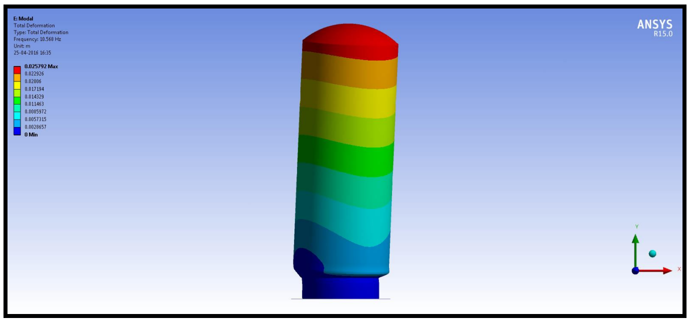

Fig.5 Natural frequency result

Table 5: Displacement value with respect to Frequency

Sr. No.	Frequency (Hz)	Displacement (mm)
1	1	10
2	3	20
3	25	30

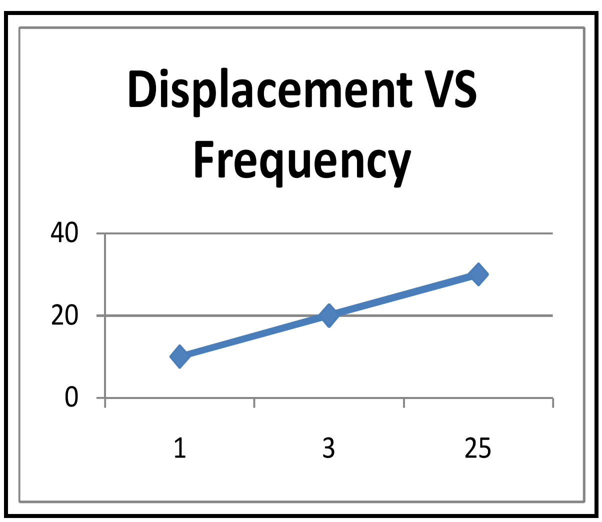

Fig.6 Displacement VS Frequency

Table 6: Validation for natural frequency

Sr. No.	Parameter	Mathematical approach IS: 1893	Software approach PVElite 2008	Software approach ANSYS 15.0
1	Natural frequency	14.44 HZ	11.34 Hz	10.56 Hz

Conclusion

- Static and dynamic analysis of pressure vessel with vertical orientation in PV-Elite software has been carried out with different load combination and stress result is obtained.
- From the analysis, Load case 18 for internal design pressure, operating weight, axial stress in the vertical direction due to seismic force and lateral force for seismic load is govern the whole analysis as maximum of all the stress ratio as 0.8282. Tensile stress value is less than the allowable tensile stress value and design is safe for the structure. i.e. 97.65 N/mm^2 < 117.90 N/mm^2
- Natural frequency value for vertical pressure vessel is estimated through PVElite 2008 which is found to 11.34 Hz for existing structure as well as mathematical value found to be 14.44Hz, and ANSYS 15.0 modal analysis observed the result of natural frequency as 10.56Hz. This natural frequency value specifies the damage level of pressure vessel structure.
- Structure displacement found to be 10mm, 20mm and 30mm for natural frequency value of 1Hz, 3Hz and 25Hz respectively.
- Mathematical approach to evaluate dynamic behavior of structure is performed by IS: 1893 codes & standards which shown the seismic response of structure. Maximum allowable lateral force at each level in vessel system is calculated.
- To consider the wind load as a alternate to seismic load condition, future work can be possible with Dynamic Wind Analysis. In mastering software ANSYS computational fluid dynamics (CFD) can be performed to evaluate wind load results.

Reference

Journal Papers:

[1] Prashant Adalinge and Gaur A V, ‘Deterministic and Probabilistic Design of Pressure Vessels’, *International Journal of Engineering Research & Technology (IJERT)*, 3.8 (2014), 1540–50.

[2] Kumar Vijay and Kumar Pardeep, ‘Mechanical Design of Pressure Vessel by Using PV-ELITE Software’, *International Journal of Scientific and Research Publications*, 4.4 (2014), 1–4.

[3] Vyas Binesh P, Tayade R M and Kumbhani Ankit D, ‘Design Of Vertical Pressure Vessel Using PVElite Software’, *International Journal of Engineering Research & Technology (IJERT)*, 2.3 (2013), 1–8.

[4] Magucki K., Szyc W., Lewinski J., “Minimization of stress concentration factor in cylindrical pressure vessels with ellipsoidal heads” *International Journal of Pressure Vessels and Piping 79 (2002) 841–846*

[5] Chavan Sandip S, Kalaskar Sanjay H and Patil Pradip B, ‘Effect of Wind Load on Pressure Vessel Design by Using Non- Linear Finite Element Analysis (FEA)’, *International Journal of Engineering Research & Technology (IJERT)*, 2.12 (2013), 2566–71.

[6] Diamanti Kalliopi, ‘Seismic Analysis and Design of Industrial Pressure Vessel’, in *Computational Methods in Structural Dynamics and Earthquake Engineering*, 2011.

[7] Senthil A.M., Dev Anand M., Anis Milton G., “Development of finite element based wind and seismic design procedure for horizontal pressure vessel”, *International journal on modeling optimization and computing (ICMOC-2012), Procedia Engineering* 38 (2012) 3998 – 4004

[8] Modi Jiger, Prof. Joshi S J, Prof. Shah D B, “Pressure Vessel Design against Wind and Seismic Load”, *International Conference on Engineering (2013), Procedia Engineering 00 (2013) 000–000*

[9] Blocki J., Combescure D., Mazzone G., “Assessment of the dynamic behaviors of the ITER Vacuum Vessel”, *Fusion Engineering and Design 88 (2013) 1938– 1941*

Books:

[10] Indian Standard, Criteria for earthquake resistant Design of structures Part 1 general provisions and buildings, fifth revision
[11] Dennis r. moss, pressure vessel design manual, Third edition
[13] Warren C. Young, Richard C. Budynas, Roark's formula for stress and strain, Seventh Edition