IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 41-47
www.iosrjournals.org

Seismic Analysis of Steel Fuel Storage Tanks

Parvathy Krishnakumar[1], Jini Jacob [1]
[1]*(Civil Department, SCMS College/ M G University, India)*

ABSTRACT : *. Seismic response of steel tank depends however on complex fluid structure interaction that results in global overturning moments and base shear induced by horizontal inertial forces, overturning moment causes an increase of vertical stresses in the tank wall and even uplift of the base plate, while base shear can lead to relative displacements between base plate and foundation. The responses of cylindrical steel tank under seismic effect is analyzed by means of finite element software package ANSYS which make use of modal analysis and response spectrum analysis . In this study, the seismic analysis of fuel storage tank with diameter of 10m and height 11m under four different liquid filling levels (25 %, 50%, 75%,100%) is considered. Tanks were designed according to American Petroleum institute (API) 650-2007. The seismic analysis of fuel storage tanks were calculated using IS 1893:2002 (Part 2)and dependence of various parameters on different aspect ratios were analyzed. Using ANSYS software package, modal analysis was carried out on all four tanks considered. Response spectrum analysis was also carried out on fuel storage tanks to study the effect of fluid level on tank behavior. Next the response spectrum analysis is done. The results of the normal stress in Y-direction are presented and found that normal stress in Y-direction of tank 1, tank 2, tank 4 have higher normal stress which exceeds the minimum yield strength(240MPa) of plate material A 516 M. Only the tank 3 (75% filling level) is safe under seismic effect.*

Keywords – *Base shear, Modal analysis , Normal stress ,Overturning moment , Response spectrum analysis*

1 .Introduction

Dynamic behavior of atmospheric storage tanks when subjected to earthquake is characterized by two pre-dominant modes of vibration: the first is related to the mass that rigidly moves together with the tank structure (impulsive mass), the other corresponds to the liquid sloshing (convective mass). Seismic response of steel tanks depends however on complex fluid-structure interaction that results in global overturning moments and base shear induced by horizontal inertial forces. Overturning moment causes an increase of the vertical stress in the tank wall and even uplift of the base plate, while the base shear can lead to relative displacements between the base plate and the foundation. When strong ground motions take place, large relative displacements are generated. Indeed, tank sliding reduces the maximum acceleration suffered by the equipment, however relatively small frictional factor may produce large relative displacements, hence large deformations and even failure of piping and connections can occur The major objectives of the study is to design of the fuel storage tank using American Petroleum Institute (API) 650-2007.Then a simplified seismic analysis of fuel storage tanks with the obligatory provisions of IS 1893:2002 (Part 2) were fully taken into account. After that modelling and analysis of the fuel storage tank using finite element software ANSYS 14 Workbench was done . Then evaluation of the behavior of liquid fuel tanks, when subject to an earthquake for different filling levels, 25, 50 and 75%, 100% using ANSYS 14 Workbench was done . Lastly, the modal and Response Spectrum analysis was done in order to find the responses of fuel storage tanks to earthquake.

Al Zeiny, (2004) [1] discussed the simplified closed form solution to be used to model the effect of liquid hydrodynamic pressure in the linear and non-linear analysis of cylindrical-vertical liquid storage tanks. It was concluded that the calculation according to EC 8, seems to be overly conservative for tanks with slenderness ratios H/R > ~1.5. ***Cooper T.W (1997)*** [5] studied ground supported cylindrical tanks having variable height, radius and thickness. In order to study the behavior of ground supported cylindrical tanks under dynamic loading a finite element method was undertaken**.** ***Carluccio and Fabbrocino, (2008)*** [3], analyzed a steel cylindrical tank for fuel storage subjected to different earthquake ground motions. ***DegaoZou and Xianjing Kong, (2000)*** [7] discussed the seismic analysis of cylindrical steel fuel storage tanks and damages und ergone by steel tanks during past earthquakes, such as the 1964 Anchorage (Alaska) and 1999 Izmit (Turkey) events showed that these structures are seismically vulnerable.

2. Structural Description and Tank Design

The tank considered for this analysis is steel cylindrical, ground supported tank containing fuel oil. The tank is anchored on a rigid concrete foundation. The size of the cylinder chosen was 10m in diameter, 11m in height. The studies are carried out on the completely full, partially filled(75%,50% 25%) with liquid. The general parameters considered for the design are as follows:

Table 1 General parameters of the tank considered for the design

Total fluid mass(m^3)	1000
Young's Modulus, E (GPa)	210
Poisson Ratio, v	0.3
Steel Density, ρ_s (kg/m^3)	7850
Liquid Density, ρ_l (kg/m^3)	890
Bulk Modulus, (GPa)	1.2

Storage tank design consists of 2 main sections – Shell Design and Roof Design. .Here tank is designed according to API 650-2007. The required shell thickness shall be the greater of the design shell thickness, including any corrosion allowance, or the hydrostatic test shell thickness. Plate material selected is A 516M whose minimum Yield Strength is 240 MPa.The thickness of shell plates shall be 25mm Minimum thickness of 12 mm is provided for tank bottom and 10mm is provided for tank roof. [2].

3.Seismic Design of Fuel Storage Tanks [As Per Is 1893:2002 (Part 2)]

According to the current design philosophy, two distinct modes of vibration for the contained liquid of rigid vertical tanks subjected to horizontal ground motion are defined: the rigid impulsive mode, in which the contained liquid follows the rigid motion of the tank and the convective mode, in which the contained liquid moves vertically due to the incompatibility with the shell of the tank. This vertical motion of the liquid surface is called sloshing. The dynamic analysis of a tank structure can be carried out using the concept of generalized single degree of freedom systems (SDOF), representing the impulsive and convective modes of vibration, as shown in Figure 1.

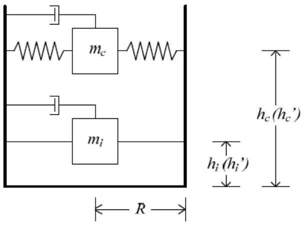


Fig. 1 Dynamic model of a steel circular tank

m_i and m_c denote impulsive and convective masses of liquid respectively, h_i is the height where the resultant of impulsive pressure on the wall is located, while h_i' is the height where the resultant of impulsive pressure on the wall and the base is located. Similarly, h_c is the height where the resultant of convective pressure on the wall is located, while $h_{c'}$ is the height where the resultant of convective pressure on the wall and the base is located. The inner radius of a tank is denoted as *R*.

3.1 Total Seismic Response

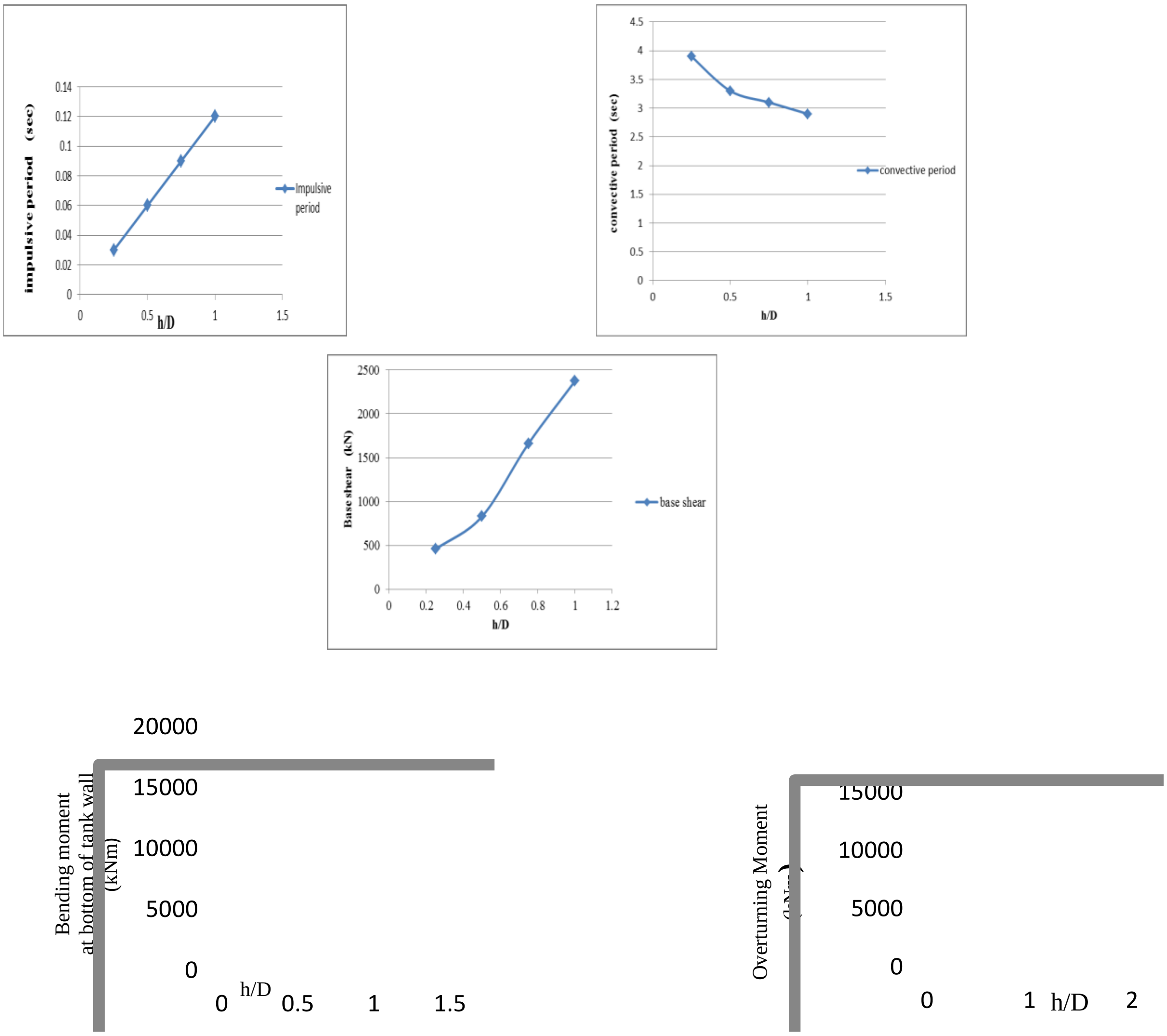


Fig.2 Total seismic response of analyzed tanks

4.Finite Element Modeling, Analysis and Results

The fuel storage tank was modeled in CATIA V5, the shell and roof of the tank, and fluid Stored in the tank were modeled as separate bodies. The CATIA Model Was Imported To ANSYS 14 Work Bench. The materials used for modeling fuel storage tank were structural steel and water element. The properties given for structural steel were density 7850 kg/m^3, Young's Modulus 210GPa, Poisson Ratio 0.3. For the water element the properties of fuel stored in the tank were given. The density of the fluid was given as 890 kg/m^3 and bulk modulus 1.2GPa. Half symmetry of the structure was considered and it was meshed with fine mesh. Four finite element models were created for tank T_1, T_2,T_3, T_4.

4.1Description and Generation of the 3D Finite Element Models

The problem of interaction between the shell and the liquid is modeled using Solid 186, Solid 187, Target 170 ,Contact 174, and Surface154 elements.

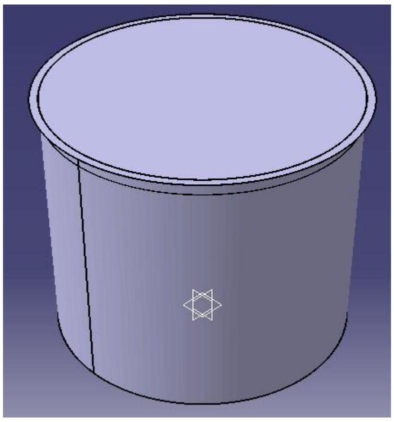


Fig. 3 Catia model of tank(T3)

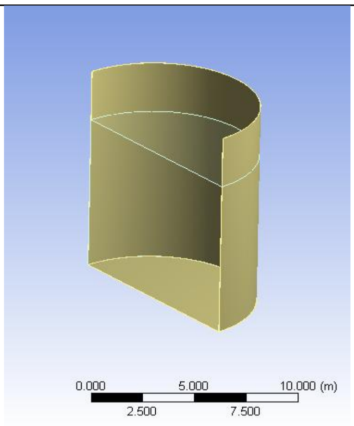


Fig. 4 Model of tank considering half symmetry(T3)

The meshing and the generation of the finite elements for the circumferential shell of the tanks were based on the range of the width of the plates that were used to construct them Damping for both tanks was assumed to be 2%.

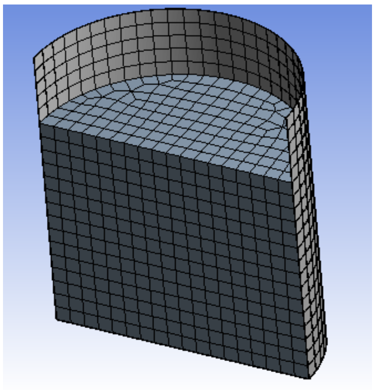
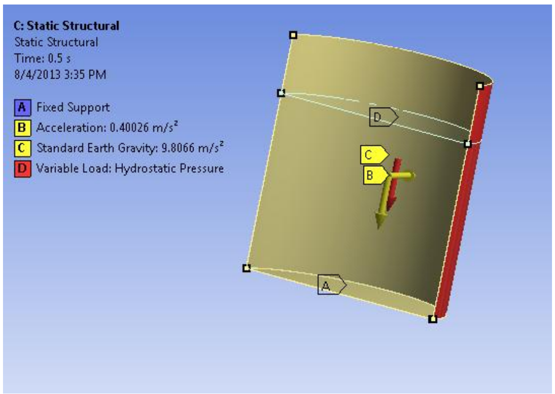


Fig. 5. Finite element mesh(T3)



C: Static Structural
Static Structural
Time: 0.5 s
8/4/2013 3:35 PM

A Fixed Support
B Acceleration: 0.40026 m/s²
C Standard Earth Gravity: 9.8066 m/s²
D Variable Load: Hydrostatic Pressure

Fig. 6. Loads and boundary conditions

4.2 Boundary Conditions and Loading

The reliability of the simulations and consequently the legitimacy of the results depend on the application of the correct boundary conditions between the two subsystems (liquid-steel tank)(figure 6). More specifically, boundary conditions were applied: i) on the top liquid surface incorporating the linear wave theory for the simulation of the maximum wave height due to seismic loading ii) between the circumferential shell of the tank and the contained liquid by proper kinematic coupling among the pertinent solid-shell nodes . The loads that were taken into account for the analyses are the dead load from the self-weight of the steel tanks and weight of the contained liquid, Uniform live load at the roof of tank, with a characteristic value of q_k = 1.20kPa (according to API 650) and Seismic loading (inputted as dynamic loading) according to Eurocode 8. The base of the tank was considered as fixed, hydrostatic pressure was applied to the shell sides and to the base of the tank. Standard gravitational force was applied to the entire structure for incorporating the dead load of the structure.

4.3 Modal Analysis

Modal analysis of the fuel storage tank was carried out in ANSYS 14 to study its behavior. The model created in CATIA was imported to ANSYS and was analyzed by considering half symmetry and by giving base of the tank fixed. Modal analysis was carried out for all four (T_1,T_2,T_3,T_4) tanks with different fluid fill levels.

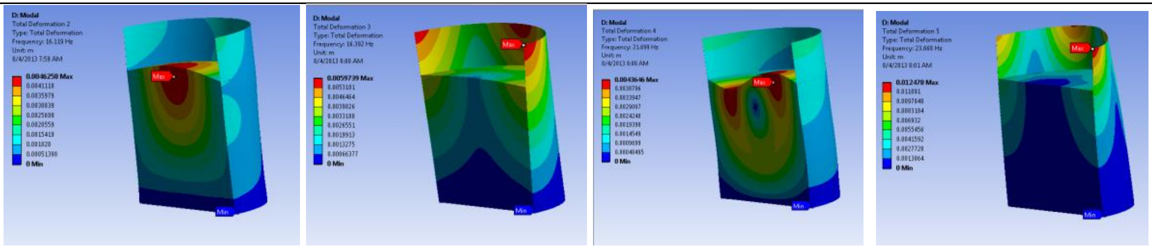


Fig. 7. First four mode shapes of Tank 3

Modal analysis was carried out, and it was found that for T_1 tank maximum deformation of 0.246 m occurs at the second mode for a frequency of 19.57 Hz. For T_2 tank max. deformation of 0.028 m occurs at the third mode for a frequency of 18.74 Hz. For T_3 tank max. deformation of 0.012 m occurs at the fourth mode for a frequency of 23.466 Hz. For T_4 tank max. deformation of 0.0040 m occurs at the second mode for a frequency of 14.40 Hz. The Modal analysis will give the eigen frequencies of the structure. Here run a response spectrum analysis where the ground acceleration is applied in the y-direction.

4.4 Response Spectrum Analysis

There are two steps in running a response spectrum analysis in ANSYS. First run a modal analysis which will give use the modes/eigenvalues of the structure. Secondly run the Response Spectrum analysis .

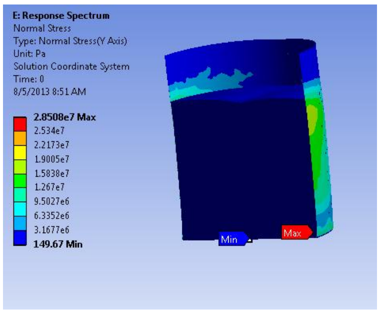


Fig. 8. Normal stress in Y-direction of tank 3

From the response spectrum analysis of all four tanks, tank 1 with 25% filling level possess higher value and tank 3 have lower value of normal stresses under seismic effects compared to other tanks. The normal stress is linearly varying with time. The tank 1 possess maximum deformation value of 9.169 e9 Pa, tank 2 have maximum deformation of 1.252e7 Pa, tank 3 have maximum deformation of 2.85e7Pa,and tank 4 have maximum deformation of 6.52e8Pa.

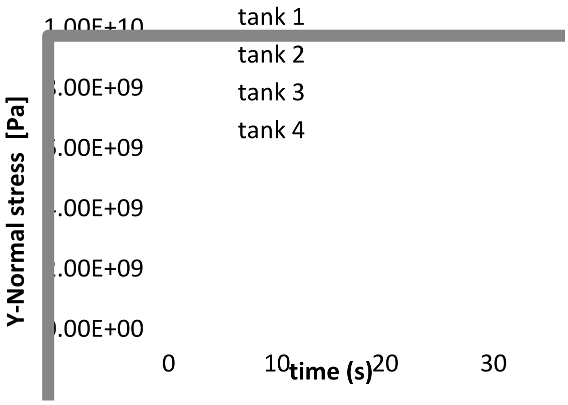


Fig. 9. Normal stress in Y -direction

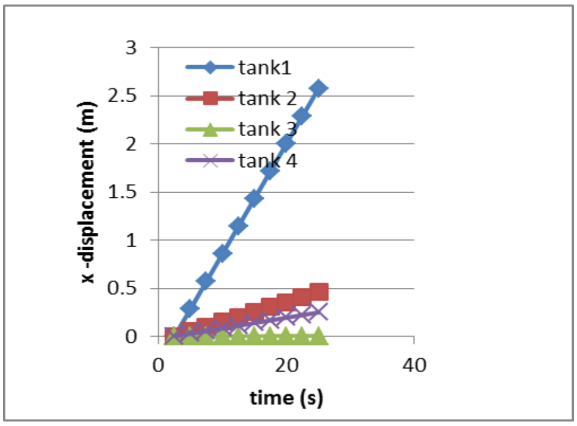


Fig.10. Y-displacement from the SRSS value

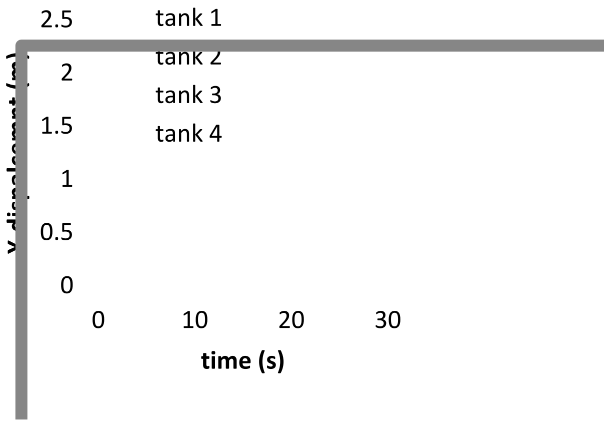


Fig. 11. X-displacement from the SRSS value

5.Conclusions

The response spectrum analysis of fuel storage tank was made on four tanks with different filling levels (25% ,50%, 75% ,100%) and the following conclusions are made.

- The study showed that the impulsive period of vibrations, base shear force and overturning moments increase almost linearly with the increase of the ratio *h/D*, whereas the convective period of vibrations is practically a constant value for the h */D*>1.5.
- But there should be better provisions for finding sloshing wave height, as of now, the code recommends to find it by means of varying parameter radius only. In addition other varying parameters like liquid height, frequency, stresses and strains should also taken into consideration for better varying results.

1. From the Finite Element modeling using ANSYS 14 Work bench, modal analysis and response spectrum analysis was done.

- From the modal analysis it was concluded that tank 1 with 25% water level have maximum deformation of 0.248m, and tank 4 with 100% water posses minimum deformation of 0.004m.Also the normal stress corresponding to four mode shapes in Y-direction is also find out from modal analysis. Maximum normal stress occur in third mode of tank 1 (25% filling level) and minimum normal stress occurs in third mode of tank 4 (100%filling level).
- From the response spectrum analysis , the results of the normal stress in Y-direction are presented .It was found that normal stress in Y-direction of tank 1, tank 2, tank 4 have higher normal stress which exceeds the minimum yield strength(240MPa) of plate material A 516 M.
- Only the tank 3 (75% filling level) is safe under seismic effects .So in order to safeguard steel tank under adverse circumstances, it is advisable to use high strength plate material or increase shell thickness.
- Also consider the freeboard requirement in design of tanks, so that a minimum of 1m clearance ,from the top of tank to liquid level have to be provided, so that impact pressures will not damage the roof of the structure during motion of liquid.

6. Acknowledgement

The authors sincerely thank the management authorities of SCMS engineering college, karukutty, for their consistent support and facilities provided

References

[1] Al Zeiny, “Simplified Modeling of Liquid-Structure Interaction in the Seismic Analysis of Cylindrical Liquid Storage Tanks*”***,** *13th World Conference on Earthquake Engineering Vancouver, B.C., Canada* August 1-6, 2004, Paper No. 1914.

[2] American Petroleum Institute (API) 650-2007, *Welded Steel Tanks for Oil Storage*, 11th Edition, August 2011.

[3] Carluccio and Fabbrocino, “Fem Seismic Analysis of Steel Tanks For Oil Storage In Industrial Facilities*”*, *The 14th World Conference on Earthquake Engineering* October, 2008,12-17 .

[4] Cho KH, Kim MK, Lim YM, Cho SY,“Seismic response of base isolated liquid storage tanks considering fluid–structure-soil interaction in time domain”. *Soil Dynamics and Earthquake Engineering, 2006,*839–52.

[5] Cooper, T. W. “A Study of the Performance of Petroleum Storage Tanks during Earthquakes, 1933– 1995”, *NIST No. GCR 97-720, U.S. Dept. of Commerce, National Institute of Standards and Technology, Gaithersburgh,1997*

[6] Criteria for Earthquake Resistant Design of Strucutres: Part 2 -Liquid Retaining Tanks (Revision of IS 1893:2002 (Part 2)).

[7] DegaoZou and Xianjing Kong, “A Simplified Seismic Response Analysis Method of Cylindrical Liquid Storage Tanks”, *12WCEE 2000*.7-13

[8] F.Saadi1 and A.AboudiAsl , “Fuel Reserve Tanks Dynamic Analysis Due to Earthquake Loading”, *World Academy of Science, Engineering and Technology 78 , 2011*

[9] G.C. Koli and Prof.V.V.Kulkarni,, “Simulation of Fluid Sloshing in a Tank”, *Proceedings of the World Congress on Engineering 2010 Vol II WCE 2010, June 30 - July 2, 2010, London, U.K.*

[10] Giovanni Fabbrocino and IunioIervolino, “Quantitative Risk Analysis of Oil Storage Facilities in Seismic Areas”, *Journal of Hazardous Materials A123 ,2005, 61–69.*