Journal homepage: http://www.journalijar.com
Journal DOI: 10.21474/IJAR01

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH

ISSN NO. 2320-5407

RESEARCH ARTICLE

MODELLING OF LIQUID PROPELLANT SLOSHIN A RECTANGULAR TANK BY VARYING BAFFLE HEIGHTS.

Ranjitha. E, G. Dinesh Kumar, Parthiban. S.
School of Aeronautical Sciences, Chennai, Tamilnadu, India.

Manuscript Info

Manuscript History:

Received: 14 February 2016
Final Accepted: 16 March 2016
Published Online: April 2016

Key words:
Fuel tank, baffle heights, slosh, CFD.

Corresponding Author

Ranjitha. E.

Abstract

Sloshing occurs in fractionally-filled containers that undergoes accelerated motion. It is a phenomenon of fluid-structure interaction. A study of a rectangular fuel tank filled with kerosene was simulated in this paper using Volume of Fluid (VOF) multiphase model. Simulations compared the velocity, turbulence and wall shear stress in the tank by varying the baffle heights. CFD analysis was carried out using a commercial finite volume package ANSYS FLUENT 16.0. The tank was set into motion by giving an acceleration of 9.81 m/s^2 in the Z direction. It was found from the CFD simulations that the sloshing in the fuel tank was significantly reduced with the introduction of baffles in the fuel tank. The damping effect of the baffles was clearly visible in the tank with baffles that is 50% of its total height from the obtained results. Further research can be undertaken by introducing holes and by varying the size of the holes in the baffles in order to further reduce the sloshing in the fuel tank.

Introduction:-

Sloshing phenomenon can be seen in all vehicles that undergoes accelerated motion. Propellant slosh has major significance in automotive as well as aerospace industry. Slosh refers to movement of liquid surface inside a container. Sloshing has greater impact on spinning spacecraft. This also generates unpleasant noise.The instability was found to be the result of interaction between liquid lateral sloshing modes and the attitude control. This tends the spacecraft to deviate from its trajectory.
Many control measures has made in order to decrease the turbulence and also the noise generated by them. The introduction of baffles is the known method for slosh reduction. Some automotive vehicles like water truck, full heights of baffles are used. Usage of full height baffles considers each compartment as separate sections. So the entire manufacturing process is constraint.

The main purpose of this work is to design baffle with different heights and analyze it to conclude the ideal solution. A rectangular fuel tank is considered and tests are performed with baffle of different heights. The computer simulations are used to validate the design. Multiphase models available in ANSYS CFD software will be investigated to find their usefulness for different flow cases as well as their limitations.

Literature review:-

Computational study of sloshing behaviour in 3-D rectangular tank with and without baffle under Seismic Excitation, Thesis Submitted to National Institute of Technology, Rourkela, by Puneet Kumar Nema*et al.*, [1]. This paper describes the modelling and meshing of the tank. This paper also gives the step-by-step procedure followed in the fluent for meshing. Following solution method is adapted from this paper: - Pressure-velocity coupling: Fractional step, Gradient: least square cell based, Pressure : Body force weighted, Momentum : Power law, Volume fraction : Geo-Reconstruct , Transient formulation :Non-iterative time advancement

CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing, 24th DAAAM International Symposium on Intelligent Manufacturing and Automation, 2013, VaibhavSingala*et al*., [2], JashBajajb*et al*., [3], NimishAwalgaonkara*et al*., [4], SarthakTibdewalc*et al*., [5], VIT University, Vellore - 632014, Tamil Nadu, India. This paper gives the boundary conditions for baffles, walls and interiors. In this project, Volume of Fluid (VOF) multiphase model in ANSYS FLUENT 12.0 was used to predict the motion of the Kerosene fuel inside the tank when the tank is under accelerated motion. The VOF model was designed to capture the position of interface between two or more immiscible fluids (air and Kerosene).

Design of fuel tank baffles to reduce kinetic energy produced by fuel sloshing and to enhance the product life cycle, VOL. 9, NO. 3, March 2014, ARPN Journal of Engineering and Applied Sciences,R. ThundilKaruppaRaj*et al*., [6], VIT University, India. This paper gives values of necessary initial conditions. Also, this paper investigated the effect of the vertical baffle heights on the liquid sloshing in a three-dimensional (3D) rectangular tank with 75% water fill level. They selected various ratios of baffle height to initial liquid height(h).For simulation of 3D incompressible, viscous, two phase flow in a tank partially filled with liquid and equipped with baffle, the volume of fluid (VOF) method based on the finite volume method has been used. Result shows that after a certain height (critical height) of baffle, the liquid does not reach at roof top and when baffle height is greater than liquid fill level, free liquid surface exhibit linear behaviour in each section.

Modelling:-

Fuel tank geometry:-

The fuel tank geometry considered is based on the work done by R. ThundilKaruppa Raj *et al*., [6]. The dimensions are given as follows.

- Length of the tank – 30cm
- Breadth of the tank – 10cm
- Width of the tank – 10cm

The rectangular tank has shown in Figure- 1.

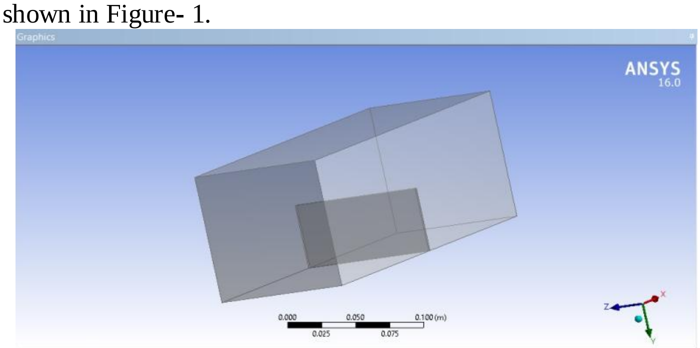

Fig. 1Rectangular tank with 50% baffle height

Meshing:-

The Catia design is imported for meshing and analyzing. Structured quadrilateral mesh was done on Ansys Fluent 16.0 commercial package. The mesh view of different baffle height designs were given as follows:

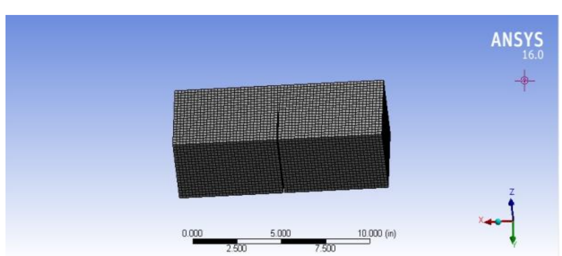

Fig.2 Meshed view of Rectangular tank with 50% baffle height

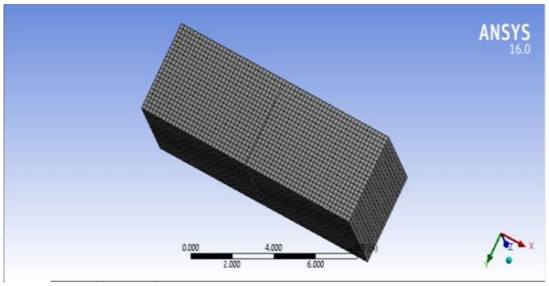

Fig.3 Rectangular tank with 75% baffle height

GRID INDEPENDENCE STUDY

The grid independent study is from 30000 nodes till 40000 nodes. The result does not vary after 36000 nodes. The result remains independent after 36000 nodes. So, 39667 nodes are used to capture the boundary layer.

GOVERNING EQUATIONS

Continuity Equation
$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (1.1)$$

Navier-Stokes Equation
$$\frac{\partial(\rho u)}{\partial t}+\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho x-\frac{\partial p}{\partial x}+\frac{1}{3}\mu\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+\mu\nabla^2 u \quad (1.2)$$

Conservation of momentum
$$\frac{\partial}{\partial t}\left(\rho\vec{V}\right)+\nabla.\left(\rho\vec{V}\vec{V}\right)=-\nabla p+\nabla.\left(\vec{\tau}\right)+\rho\vec{g}+\vec{F} \quad (1.3)$$

Turbulence viscosity equation
$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \quad (1.4)$$

Volume Fraction equation
$$\frac{1}{\rho_b}\left[\frac{\partial}{\partial t}(\alpha_b\rho_b)+\nabla.\left(\alpha_b\rho_b\vec{V_b}\right)=s_{\alpha_b}+\sum(\dot{m}_{ab}-\dot{m}_{ba})\right] \quad (1.5)$$

Boundary conditions

Number of fluids present	2 (Air and fuel)
Fuel considered	Kerosene
Acceleration	9.81 m^2
Time step size	0.0005s
Compressibility of fluid	Incompressible
Turbulence model	SST model
Density of fluid	780kg/m^3
Various heights of baffle	50%, 75%, 100% of total tankheight

Results and discussions

Simulation results are obtained from the design containing the baffles and by varying its height. The simulated result of 50% of total height baffle configuration is validated with the results of the experiments conducted by the R. ThundilKaruppa Raj *et al.*, [6]. The turbulent kinetic energy, velocity and wall shear stress of this model is lesser when compared to the results with the baffles of 75% and 100% of the total height baffle configuration. The figures show that the fluid experiences high pressure when they hit the wall of the tank. Also, 100% of total height baffle configuration considers each compartment as separate sections. So the height of the baffle is reduced to overcome this problem.

Parameters/ Height of the baffle	**Velocity (m/s)**	**Turbulence (m^2/s^2)**	**Wall Flux (Pa)**
50%	2.95038 $X10^{-6}$	0.9082082	4.90225 $X10^{-8}$
75%	1.4305775 $X10^{-19}$	0.9080748	0.007173 $X10^{-22}$
100%	0.517448 $X10^{-19}$	0.9084310	4.68040 $X10^{-22}$

Table 1. Parameters and values for various height configuration of the baffle

The average turbulent kinetic energy, velocity and wall shear stress are given in table 1. The simulation is simulated in a computer having an Intel Core i3 -4030U, 1.9GHz processor with 4GB RAM. The system took approximately 100 hours to solve the problem.

The concluded height of baffle is 5cm with the width of 2mm. Width of the baffle does not influence the sloshing. Bottom mounted baffles reduces the turbulent kinetic energy and influences sloshing to a great extent. A top mounted baffle reduces slosh but the noise created by them is high and hence it is not considered in this study.

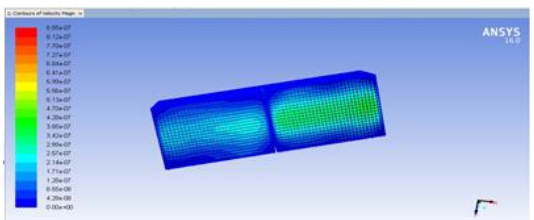

Fig. 4 Velocity distribution of rectangular tank with 50% baffle height

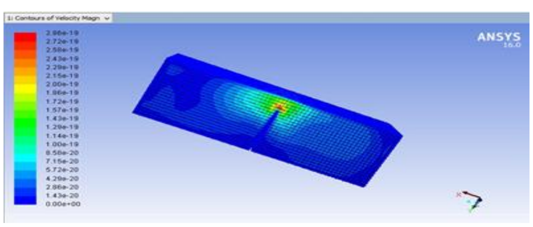

Fig. 5 Velocity distribution of rectangular tank with 75% baffle height

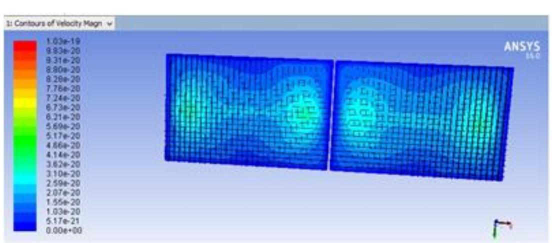

Fig. 6 Velocity distribution of rectangular tank with 100% baffle height

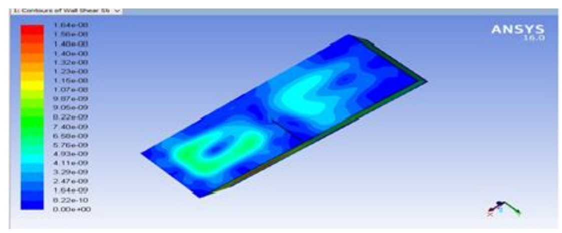

Fig. 7 Wall Shear stress distribution of rectangulartank with 50% baffle height

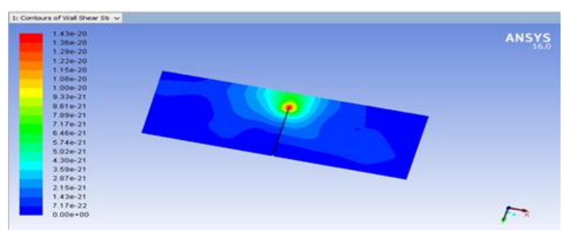

Fig. 8 Wall Shear stress distribute of rectangular tank with 75% baffle height

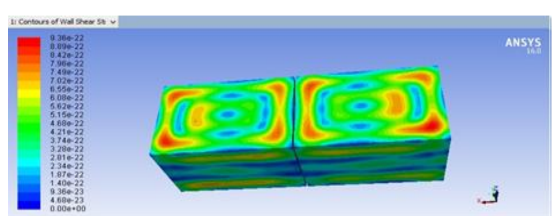

Fig. 9 Wall Shear stress distribution of rectangular tank with 100% baffle height

This study gives the following conclusions:
a) The Turbulent kinetic energy is high for the baffle with 100%of total height configuration.
b) Also, 100% of total height baffle configuration considers each compartment as separate sections.
c) The wall shear stress is comparatively high at the edges for baffle with 75% of total height configuration.
d) The baffles with 50% of total height give comparatively good results but it has to be placed alternatively with specified intervals.
Future work also includes varying of baffle angles which may have good influence in sloshing.

Acknowledgement:-

We wish to acknowledge the expertise of Mr. G. Dinesh Kumar, Assisstant Professor, Hindustan University, Chennai for his valuable suggestion in carrying out this work.

Nomenclature:-

ρ = Density
t = Time
u, v, w = velocity components in x, y, z direction
p = pressure
k = turbulent kinetic energy
m_{ab} = mass transfer from a to b phase
m_{ba} = mass transfer from b to a phase

References:-

[1] Ling Hou, Fangcheng Li, Chunliang Wu (2012), "A Numerical Study of Liquid Sloshing in a Two-dimensional Tank under External Excitations", J. Marine Sci. Appl. Vol 11: 305-310
[2] J.H. Jung, H.S.Yoon ,C.Y.Lee, S.C.Shin (2012), "Effect of the vertical baffle height on the liquid sloshing in a three-dimensional rectangular tank", Ocean Engineering. Vol 44: 79–89
[3] V. Singal ,Jash Bajaj, NimishAwalgaonkar, SarthakTibdewal (2014), "CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing", Procedia Engineering. Vol 69: 1365 – 1371
[4] KritThreepopnartkul, ChakritSuvanjumrat, "The Effect of Baffles on Fluid Sloshing inside the Moving Rectangular Tank", Journal of Research and Applications in Mechanical Engineering. Vol. 1 No.2
[5] T. Kandasamy, S. Rakheja, A.K.W. Ahmed, (2010), "An Analysis of Baffles Designs for Limiting Fluid Slosh in Partly Filled Tank Trucks", The Open Transportation Journal. Vol 4: 2332
[6] K. J. Craig · T. C. Kingsley (2007), "Design optimization of containers for sloshing and impact", DOI 10.1007/s00158-006-0038-6. Vol 33: 71–87
[7] Bernhard Godderidge , Stephen Turnock, Mingyi Tan, Chris Earl (2009), "An investigation of multiphase CFD modelling of a lateral sloshing tank", Computers & Fluids. Vol 38: 183–193
[8] A.VakilaadSarabi, M. Miyajima, K. Murata (2012), "Study of the Sloshing of Water Reservoirs and Tanks due to Long Period and Long Duration Seismic Motions", 15 WCEE LISBOA.
[9] A. Di Nardo, G. Langella, D. Mele, C. Noviello (2009), "Sloshing Phenomenon Analysis In Liquid Fuels Storage Tanks Subject To Seismic Event", International Journal Of Heat And Technology.Vol 27 NO 2
[10] M. Eswaran, U.K. Saha, D. Maity (2009), "Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation", Computers and Structures. Vol 87: 198–205
[11] MahmoodHosseini, HamidrezaVosoughifar, PegahFarshadmanesh (2013), "Simplified Dynamic Analysis of Sloshing in Rectangular Tanks with Multiple Vertical Baffles", Journal of Water Sciences Research, ISSN: 2251-7405 eISSN: 2251-7413 Vol.5, No.1: 19-30
[12] L. Khezzar, A. C. Seibi, A. Goharzadeh , (2009), "Water Sloshing in Rectangular Tanks – An Experimental Investigation & Numerical Simulation", International Journal of Engineering (IJE), Vol 3: Issue (2) 174-184
[13] Dongming Liu, Pengzhi Lin, (2009). "Three-dimensional liquid sloshing in a tank with baffles", Ocean Engineering. Vol 36: 202–212
[14] Mi-An Xue, JinhaiZheng, Pengzhi Lin, (2012), "Numerical Simulation of Sloshing Phenomena in Cubic Tank with Multiple Baffles", Journal of Applied Mathematics. Vol 2012, Article ID 245702, 21 pages
[15] Y.G. Chen, K. Djidjeli, W.G. Price, (2009), " Numerical simulation of liquid sloshing phenomena in partially filled containers", Computers & Fluids. Vol 38: 830–842