iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: I Month of publication: January 2017

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Design of Fuel Tank Baffles to Reduce Kinetic Energy Produced By Fuel Sloshing

Mohammed Iqbal Khatib[1], Mohammed Asif Kattimani[2], Mohd Nayeemuddin[3], Mohd Mujeebuddin[4], Mohammed Ahmed[5]
[1,2]*Assistant Professor of Mechanical Department, Lords Institute of Engineering and Technology, Hyderabad*
[3,4,5]*Students of Lords Institute of Engineering and Technology, Hyderabad*

Abstract: Fuel sloshing occurs in vehicle when it accelerates or decelerates. It generates high kinetic energy with unpleasant noise. This fuel sloshing leads to vehicle imbalance. This vehicle instability may occur when the fuel to weight ratio is high. In automobiles, the fuel sloshing generates unpleasant noise which is not expected from the present ones. So, this work presents the use of baffles at different positions in the fuel tank to suppress the fuel sloshing. Generally, this phenomenon is seen in High Density Polyethylene (HDPE) tanks which are strong and light in weight. Introduction of baffles in the HDPE tanks is an onerous process through blow molding. So, the present work mainly focuses on the baffle design. The result depends upon the number of baffles, location of the baffle and its shape. The highest noise is generated only when the fuel hits the top of the tank. The baffle is designed in such a way that these noises are reduced. The height of the baffle should be sufficient enough to reduce the flow of fuel. The work mainly focuses over the selection of appropriate height of the baffle which gives optimum result with less effect on the fuel capacity of the tank. The Turbulent kinetic energy, force and velocity produced by the fuel during sloshing are calculated. The kinetic energy produced by the fuel produces the stress at the ends when reaching the ends of the tank. So, the use of baffle reduced the noise and as well as the stress created at the ends. The product life cycle of the tank is also improved.
Keywords: fuel tank, baffle, product life cycle, sloshing.

I. INTRODUCTION

In the recent years, the perseverance for quality in vehicle has become important. Braking and acceleration is a basic phenomenon in automobiles. Due to this, the vehicle tends to have inertia effects. This phenomenon generates sloshing in the fuel tank which generates unpleasant noise in the vehicle. This has increased the perseverance over the search for quiet automobile. Generally, the fluid in the fuel tank is diesel or petrol. So, the noise generated by the fluid bothers the occupants.

Some control measures have to be made in order to decrease the turbulent kinetic energy and the noise generated by them. The flow of the fuel has to be changed in the fuel tank in order to have these reduced. So, baffles are used to have a control over the flow of the fuel. In steel fuel tanks, full height baffles are used. But now-a-days High Density Polyethylene (HDPE) have attracted the automotive manufacturers due to its light in weight and durable property. So, the entire manufacturing process is constraint. The tanks are manufactured by blow molding process. The extent of building baffles in HDPE is difficult as compared to the steel tanks. So, the design of baffles plays a key role in reducing the slosh and as well as the ease for manufacturing. The full height baffles are difficult to build in the HDPE fuel tanks. So, the design must be fulfilling the needs of the manufacturer and the customer. The main purpose of this work is to design different baffle design and analyze to conclude the ideal solution. We have considered a fuel tank model and performed tests with different baffle design. So, the computer simulations will be used to validate the design.

II. LITERATURE REVIEW

The validated CFD model was subsequently formulated for a full-scale tank and simulations are performed under excitations idealizing the straight-line braking maneuvers to investigate the anti-slosh role of four different transverse baffles concepts. We concluded that highly effective anti-slosh effect is also observed for the partial baffle designs under higher fill levels.

Mohammed Iqbal[1] used the combination of CFD (Computational Fluid Dynamic), FE (Finite Element) and Acoustic simulation methods, to evaluate the radiated fuel tank slosh noise performance using CAE methods. Mohammed Asif [2] verified the reliability of the FSI method and suggested a new CAE analysis processes to predict fuel sloshing noise. Mohd Mujeeb [4] evaluated the slosh performance of plastic tanks, carried out a study to predict the dynamic behavior of the fuel inside a fuel tank during transient driving conditions. We developed a new model without hampering the slosh noise phenomenon.

Mohd Nayeem[3] presented an experimental validation of the source path- receiver approach to slosh noise from a fuel tank on a commercially available vehicle. They correlated between predicted and real noise, confirming the validity of the approach, Mohammed Ahmed[5] identified the current capabilities and discussed optimal parameters of testing component level fuel slosh noise, and explored the merits of various NVH analysis methods that can be used to quantify slosh noise.

We compared the force acting on the fuel tank and found that they have same trend as the mechanism that triggers the sloshing. We also compared the deformation of the node located on one of the two tank chambers and found that the deformation of a node on the tank did not change much for the given braking scenario with the changing water levels.

III. FUEL TANK GEOMETRY

The fuel tank considered is based on the work done. The dimensions are as shown in Figure-1

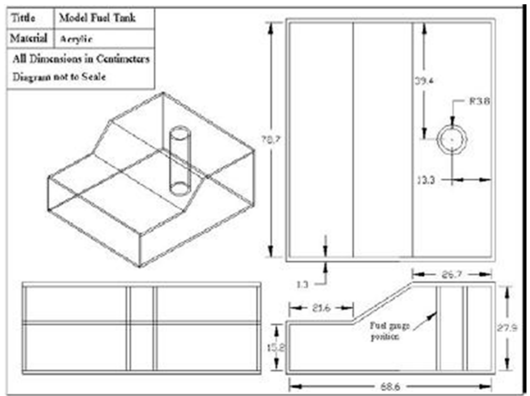


Figure.1 Fuel tank dimensions

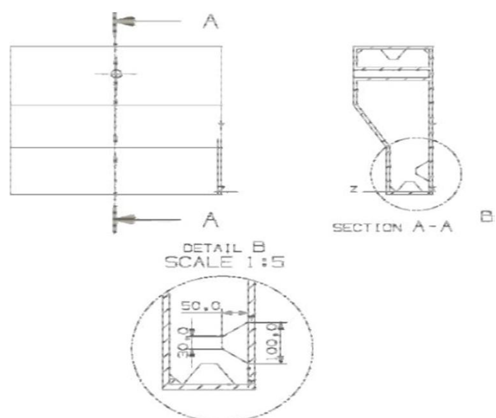


Figure.2. Dimension of the baffle (Dimensions in mm).

IV. MODELLING AND MESHING

The meshed fuel tank is shown in Figure-3. The fuel tank is modelled in Unigraphics NX and meshed using Ansys ICEM - CFD.

Figure.3. Meshed fuel tank used in simulation

Initialization of the model is made in Ansys CFX Pre-as shown in Figures 4 and 5Ansys CFX is used for the simulation purposes.

Volume 5 Issue I, January 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

The working fluid in the tank is water. The initial conditions which are considered are follows. The entire model is discretized using hexahedral mesh elements which are accurate and involve less computation effort. Fine control on the hexahedral mesh near the wall surface allows capturing the boundary layer gradient accurately.

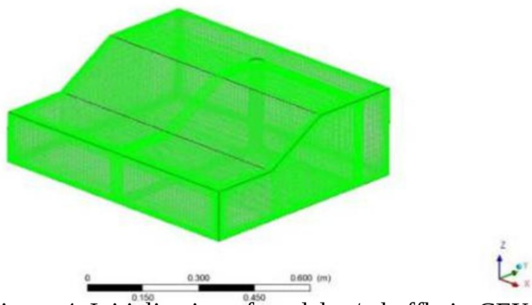


Figure.4. Initialization of model w/o baffle in CFX-Pre

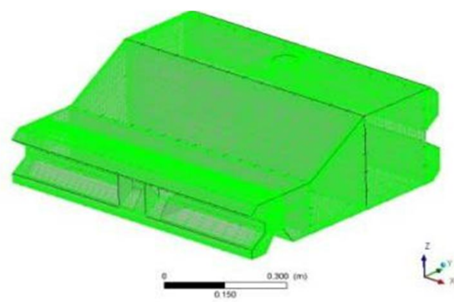


Figure. 5. Initialization of model with baffles in CFX-Pre

V. GRID INDEPENDENCE STUDY

The grid independent study is carried out starting with 100000 nodes till 500000 nodes. The variation in the results doesn't occur after 400000 nodes. The result is independent to the number of grids after 400000 nodes. So, 496000 nodes are used to capture the boundary layer.

VI. GOVERNING EQUATIONS

The 3-D flow inside the fuel tank has been simulated by solving the appropriate governing equations (1), (2), (3) and (4). Turbulence is taken care by shear stress transport (SST) model.

$$C\ conservation\ of\ mass\ \nabla(\rho V) = 0 \quad -------- $$

$$x-momentum = \frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \quad ----- (2)$$

$$y-momentum = \frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g - -(3)$$

$$z-momentum = \frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \quad ----- (4)$$

VII. BOUNDARY CONDITIONS

In ANSYS CFX Pre-processor, the fluid domains are defined. The flow in this study is turbulent, Hence Shear Stress Transport model is chosen.

Table-I.
Initial conditions and values.

Initial conditions	Corresponding values
Density of the fluid	0.001 kg/cm^3, density of water
Number of fluid present	1, air is considered to be the void region
Gravity	981 cm/s^2 downward along the z-axis
Time step size	0.0005 sec
Duration of the simulation	4 seconds
Fluid temperature	293.0 K
Compressibility of the fluid	Incompressible, water is considered to be an in compressible fluid
Void region pressure	1013 $kg/cm\text{-}s^2$, 1 atmosphere pressure
Initial pressure field	Hydrostatic pressure in the z-direction
Displacement	-15.5[cm]*sin((pi*t)/(1[s])) cm
Turbulence model	Shear stress turbulence model (SST)
Height of water	6.5 cm

VIII. RESULTS AND DISCUSSIONS

Simulation results are obtained from the design containing the baffles. The simulated result of No-baffle configuration is validated with the results of the experiments conducted by us by correlation analysis. The turbulent kinetic energy, velocity of water and average force of the baffled model is lesser when compared to the results without the baffles. The turbulent kinetic energy is high when the baffles are not used. The figures 6, 7 shows that the fluid experiences high pressure when they hit the wall of the tank. The readings are taken from 1.25 to 1.35 seconds because the water moves from the shallow end to deep end. So, the baffles are created to reduce the flow of the fluid when it is about to hit the wall.

The baffles are placed at the walls near the deep and shallow ends of the tank. It will decrease the velocity. of the water flow, which tries to hit the wall with high velocity. The comparison of average velocity of water of no-baffle and baffled model makes it visible.

Table.II.
Parameters and values at 0.1 second

Parameter	Time (sec)	No-baffle configuration	With-baffle configuration
Avg. turbulent	0.1	1736 cm^2/s^2	0.13996 cm^2/s^2
Avg. velocity of water	0.1	89.933 cm/s	22.804 cm/s
Avg. force	0.1	0.002460 N	0.002086 N

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

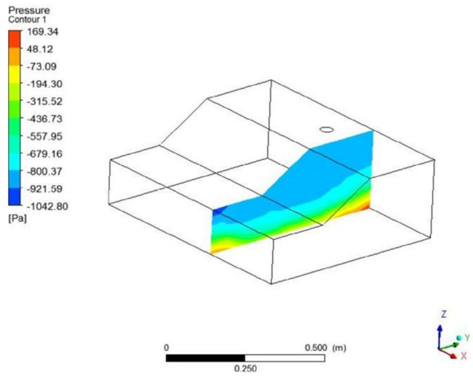


Figure.6. Pressure at plane at 1.25 seconds

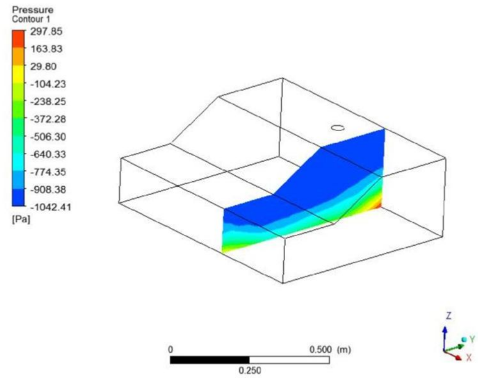


Figure.7.Pressure at plane at 1.35 seconds

The work arrives to following conclusions from the computer simulations.
The Turbulent kinetic energy is high when there is no baffle.
The height of the baffle influences the sloshing of the fuel not the width of the baffle.
The side mounted baffles has reduced the velocity of the fluid which is about to hit the top of the tank.
The baffles have reduced the Average velocity of the fluid up to 74.64%.

IX. ACKKNOWLEDGEMENT

We are very much thankful to Head of the Department, Mechanical Engineering and project guide for their inspiring guidance with helpful suggestions for the project. We are greatly obliged to the entire staff of the Department of Mechanical Engineering, Lords Institute of Engineering and Technology, Hyderabad

Table .III

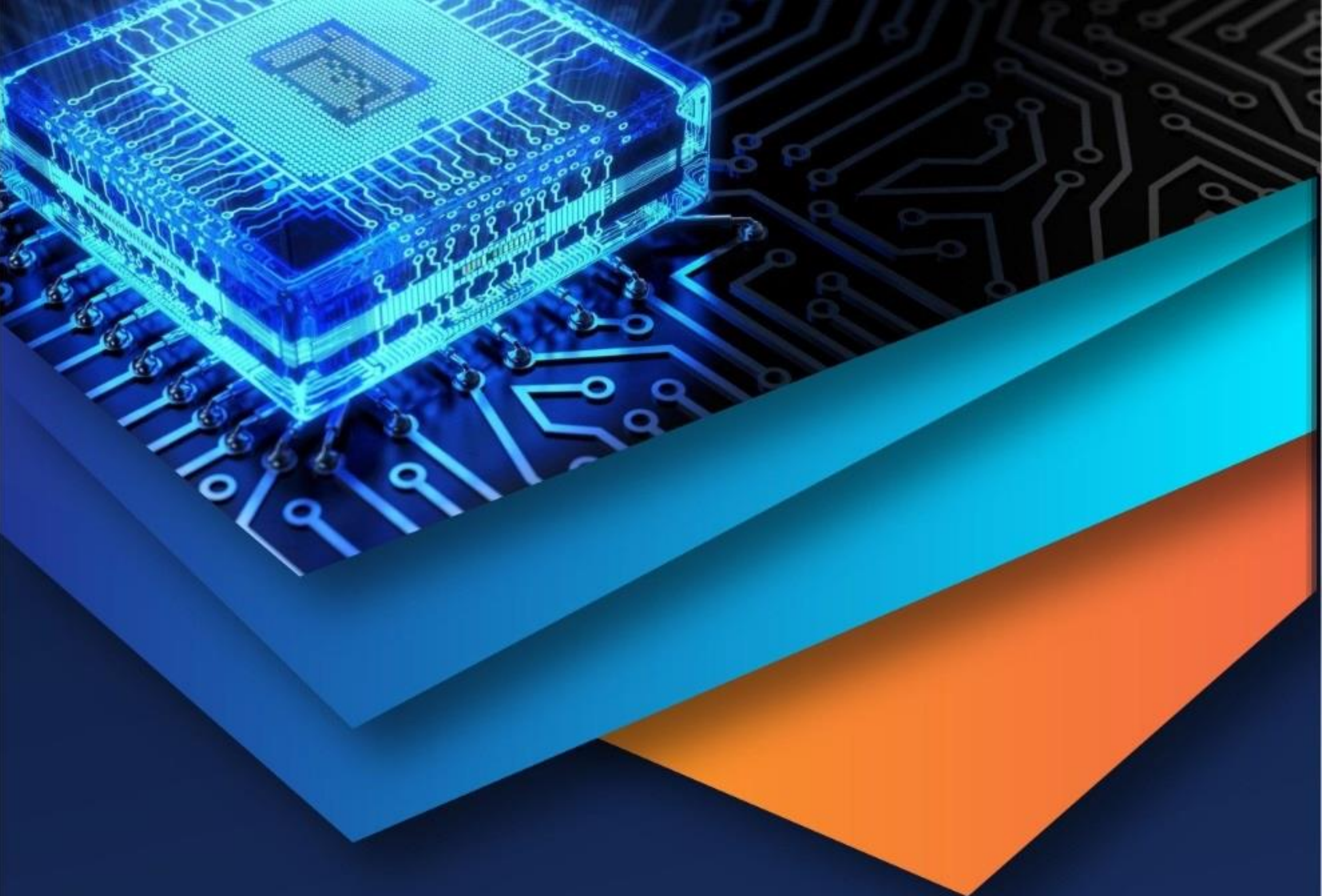
Nomenclature
V= Velocity vector [-]
x, y, z = position co-ordinates, [-]
g = gravitational force [ms^{-2}]
p = pressure [pa]
Greek symbols
ρ= *density*, [kgm^{-3}]
T= shear stress, [Nm^{-2}]

REFERENCES

[1] Hoi Sum IU, W. L. Cleghorn and J. K. Mills, "Design and Analysis of Fuel Tank Baffles to reduce the Noise Gener From Fuel Sloshing", SAE TECHNICAL PAPER SERIES, 2004-01-0403, 2004.
[2] Korang Modaressi-Tehrani, S. Rakheja and I. Stiharu , "Role of Transverse Baffle Designs on Transient Three-Dimensional Liquid Slosh in a Partly-Filled Circular Tank", SAE TECHNICAL PAPER SERIES, 2005-01-3594, 2005.

Volume 5 Issue I, January 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

[3] Kouji Kamiya and Yoshihisa Yamaguchi, "Simulation Studies of Sloshing in a Fuel Tank", SAE TECHNICAL PAPER SERIES, 2002-01-0574, 2002.
[4] Peiran Ding, Fan Li and Stephen Sibal, "An Integrated Fuel Tank System Simulation", SAE TECHNICAL PAPER SAE SAE TECHNICAL PAPER SERIES, 2011-01-0792, 2011
[5] Thundil karuppa raj and srikanth ganne, "Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow", THERMAL SCIENCE, Vol. 16, No. 4, pp. 1165-1174,2012
[6] Jong-Suh Park, Seung-Chan Choi and Seok-Gil Hong, "The Prediction of Fuel Sloshing Noise Based on Fluid-Structure Interaction Analysis", SAE TECHNICAL PAPER SERIES, 2011-01-1695, 2011.
[7] Won-Joo Roh, Sok-Hyun Cho and Jae In Park, "Noise Reduction by Decreasing Impact Pressure", SAE TECHNICAL PAPER SERIES, 2005-01-1913, 2005.
[8] Fan Li,Stephen D. Sibal, Ian Francis McGann and Raphael Hallez, "Radiated Fuel Tank Slosh Noise Simulation", SAE TECHNICAL PAPER SERIES, 2011-01-0495, 2011.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY