



DESIGN OF FUEL TANK BAFFLES TO REDUCE KINETIC ENERGY PRODUCED BY FUEL SLOSHING AND TO ENHANCE THE PRODUCT LIFE CYCLE

R. Thundil Karuppa Raj, T. Bageerathan and G. Edison

School of Mechanical and Building Sciences, VIT University, Vellore, Tamil Nadu, India

E-Mail: thundil.rajagopal@vit.ac.in

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 moulding. So the present work mainly focuses on the simulation of the slosh experiments to analyse 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. For this work, the fuel tank is modelled with Solidworks 2011. The fuel tank is meshed in ICEM CFD. It is solved in Ansys CFX 12.0. 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, CFD, sloshing.

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 gasoline. So the noise generated by the fluid bothers the occupants.

Some control measures has 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 moulding 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 is 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.

LITERATURE REVIEW

Kouji Kamiya and Yoshihisa Yamaguchi [1] made fuel sloshing simulation experiments similar to the experimental results and reduced the development period. Hoi Sum IU *et al.*, [2] worked on simulation and practical experiments of a fuel tank to relate the results of the both to conclude an optimum result. Won-Joo Roh *et al.*, [3] analyzed the characteristics of impact pressure on the tank wall to find noise source on the wall which is influenced by both the inertia of fuel mass and the dynamic motion of sloshing flow. Also, a parametric study has been done to decrease impact pressure by changing its dimensions and shape. Manuel J. Fabela-Gallegos *et al.*, [4] conducted an experimental study on the effects of fill level and number of baffles on the sloshing using an instrumented scaled experimental tank of elliptical transversal section with water as liquid cargo.

Korang Modaressi-Tehrani *et al.*, [5] studied the role of transverse baffles on Transient Three- Dimensional Liquid Slosh in a Partly-Filled Circular Tank which in reducing not only the magnitudes of longitudinal slosh force but also the lateral load shift and slosh force, roll, pitch and yaw moments. Masashi Kamei *et al.*, [6] confirm the correlation of the sloshing noise performance and the main factors related to the sloshing noise. This was done to shorten the development time for a fuel tank. G. R. Yan *et al.*, [7] validated the fuel tank model by demonstrating the experimental data acquired with a scale model tank.



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. They concluded that highly effective anti-slosh effect is also observed for the partial baffle designs under higher fill levels.

Fan Li *et al.*, [8] 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. Stephen Sibal *et al.*, [9] presented both LS-Dyna's Arbitrary Lagrange-Euler (ALE) and Abaqus' Coupled Eulerian- Lagrange (CEL) methods for predicting the structural performance of a fuel tank system and demonstrated that a fuel tank systems and their components can be numerically evaluated before the products release. Jong-Suh Park *et al.*, [10] verified the reliability of the FSI method and suggested a new CAE analysis processes to predict fuel sloshing noise. Mahesh Balthi *et al.*, [11] evaluated the slosh performance of plastic tanks, carried out a study to predict the dynamic behaviour of the fuel inside a fuel tank during transient driving conditions. The developed a new model without hampering the slosh noise phenomenon.

Pierre De Man and Jules-Joseph Van Schaftingen., [12] presented an experimental validation of the source path- receiver approach to slosh noise from a fuel tank on a commercially available passenger vehicle. They correlated between predicted and real noise, confirming the validity of the approach. Eric Frank *et al.*, [13] 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.

Veera Venkata Sunil Vytla and Yuya Ando [14] compared the force acting on the fuel tank and found that they have same trend as the mechanism that triggers the sloshing. They 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.

CFD SIMULATIONS

FUEL TANK GEOMETRY

The fuel tank considered is based on the work done by Hoi Sum IU *et al.*, [2]. The dimensions are as shown in Figure-1.

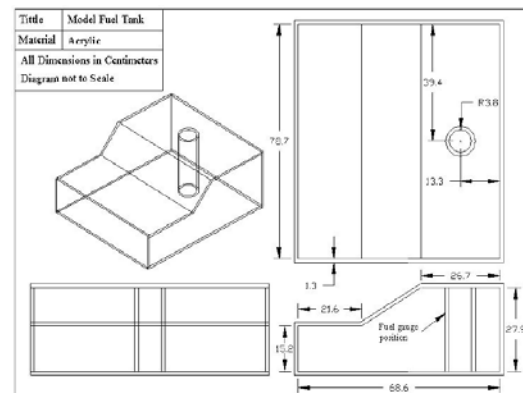


Figure-1. Fuel tank dimensions [2].

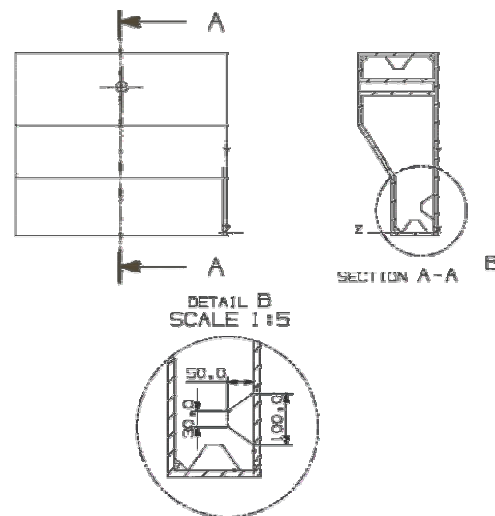


Figure-2. Dimension of the baffle (Dimensions in mm).

MODELLING AND MESHING

The meshed fuel tank is shown in Figure-2. 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 3 and 4. Ansys CFX is used for the simulation purposes. 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 [15].

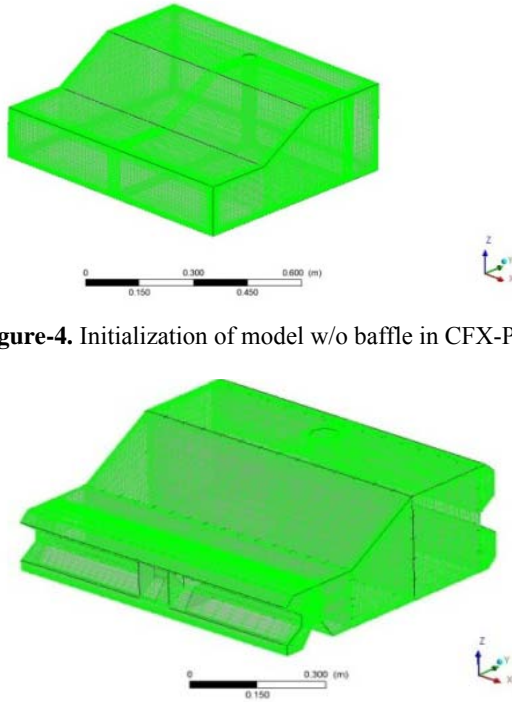


Figure-4. Initialization of model w/o baffle in CFX-Pre.

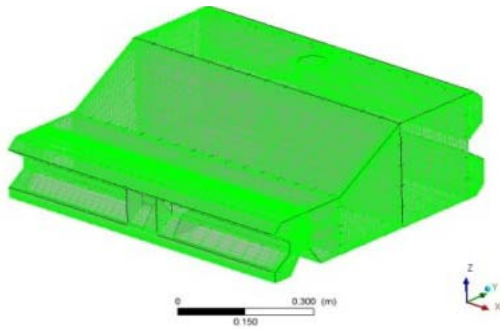


Figure-5. Initialization of model with baffle in CFX-Pre.

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.

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.

$$\text{Conservation of mass } \nabla(\rho V) = 0 \text{ ----- (1)}$$

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

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

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

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-1. 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 incompressible fluid
Void region pressure	1013 kg/cm-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

CORRELATION ANALYSIS

The Hoi Sum IU *et al.*, [2] has not mentioned the number of grids and turbulence model they used. So the correlation analysis is carried out between the simulation

results of Hoi Sum IU *et al.*, [2] and this work. The results are taken at the time steps of 2 sec, 2.5 sec and 3 seconds. The correlation factor is 0.97. This makes it visible that there is a high correlation between these simulation values.

**Table-2.** Correlation analysis result of Turbulent Kinetic energies with 6.5 cm water level.

Time (Sec)	Present work (cm^2/s^2)	Hoi Sum IU <i>et al.</i> , (cm^2/s^2)	Correlation factor
2.0	895.261	39.6	
2.5	762.382	13.1	
3.0	646.666	3.4	0.975469

RESULTS AND DISCUSSIONS

Simulation results are obtained from the No-baffle design and the design containing the baffles. The simulated result of No-baffle configuration is validated with the results of the experiments conducted by the Hoi Sum IU *et al.*, [2] 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 5, 6, 7 and 8 shows that the fluid experiences high pressure when they hit the wall of the tank. The readings are taken from 1.25 to 1.55 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.

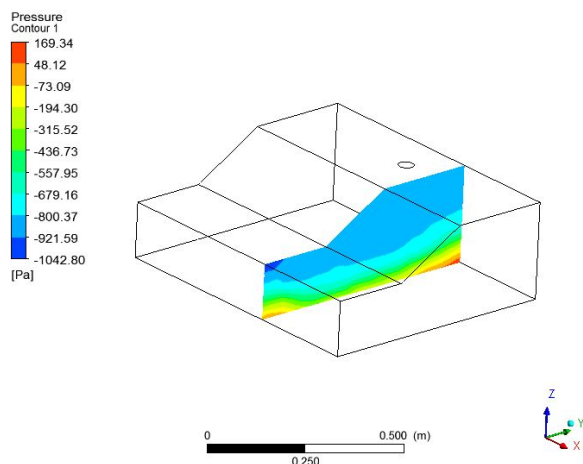
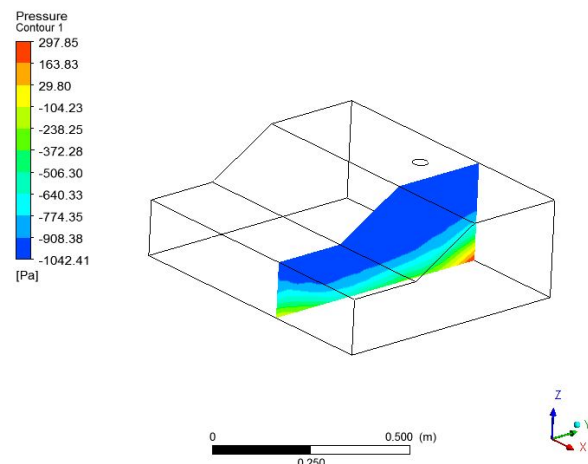
The baffles are placed at the walls near the deep and shallow ends of the tank. It will decrease the velocity

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. The simulation is simulated in a computer having a CORE 2Duo 2.2 GHz processor with 4GB RAM. The system took approximately 120 hours to solve the problem.

The baffle is of height 5cm. The width of the baffle does not influence the sloshing [2]. The baffle has effect over sloshing when the fluid is in less height than the height of the baffle. The baffle does not influence sloshing when the fluid height is high than the baffle's height. It is because the baffle will be submerged in the water and the baffle has no use. Bottom mounted baffles influences the sloshing and reduces the turbulent kinetic energy to a great extent. When the water travels from the deep end to shallow end, the velocity of the water is also reduced to great extent.

Table-3. Parameters and values at 0.1 second.

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

**Figure-6.** Pressure at plane at 1.25 seconds.**Figure-7.** Pressure at plane at 1.35 seconds.

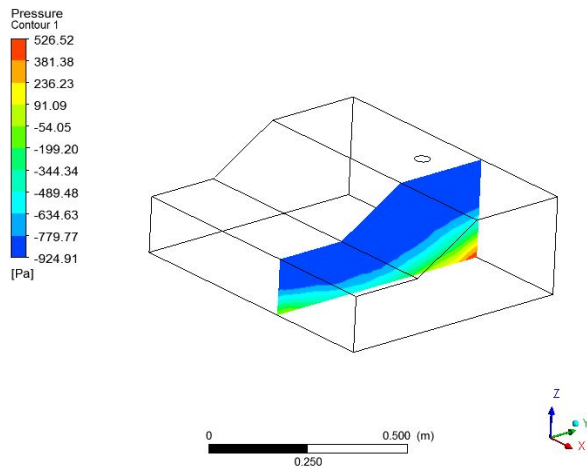


Figure-8. Pressure at plane at 1.45 seconds.

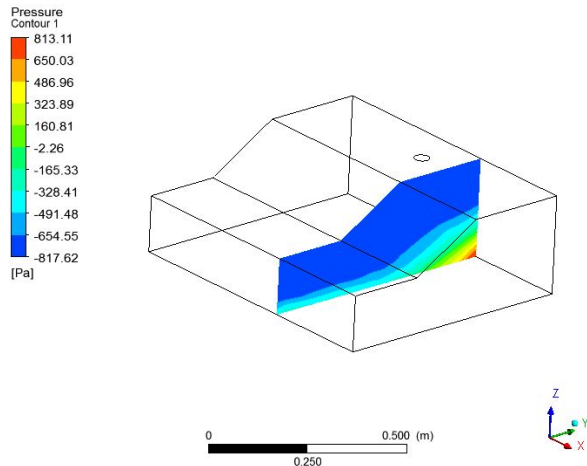


Figure-9. Pressure at plane at 1.55 seconds.

Top mounted baffles are not considered in this study. They reduce sloshing but the noise created by them is high [2]. The air bubble formed by them during sloshing tries to escape from one end to other end, which creates unpleasant noise [2]. Side mounted baffles are considered here. These side mounted baffles reduces the velocity of the fluid when the fluid travels in a high velocity from one end to other end. They also reduce the velocity of the fluid which is about to hit the top of the tank. So the velocity is reduced by the side mounted baffles. The bottom and side mounted baffles are with openings which helps in maintaining the correct fuel level at the fuel gauge reader. It may lead to misread the fuel level when the baffles do not have openings. The velocity of the water is reduced when the water moves towards the shallow end. Then the velocity of water is not reduced much when the water moves toward the shallow end. It is because the bottom mounted baffles is only placed near the shallow end and not at the deep end. So when the water moves to the deep end the velocity of water is more. The baffles have to be

placed at both the ends so that the velocity of the water is reduced.

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%.

ACKNOWLEDGEMENT

We wish to acknowledge the expertise of Mr. Balaji Ranganathan from FL Smidth, CFD Division, Chennai for his valuable suggestion in carrying out this work.

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] Kouji Kamiya and Yoshihisa Yamaguchi, "Simulation Studies of Sloshing in a Fuel Tank", SAE TECHNICAL PAPER SERIES, 2002-01-0574, 2002.
- [2] Hoi Sum IU, W. L. Cleghorn and J. K. Mills, "Design and Analysis of Fuel Tank Baffles to reduce the Noise Generated From Fuel Sloshing", SAE TECHNICAL PAPER SERIES, 2004-01-0403, 2004.
- [3] Won-Joo Roh, Sok-Hyun Cho and Jae In Park, "Noise Reduction by Decreasing Impact Pressure", SAE TECHNICAL PAPER SERIES, 2005-01-1913, 2005.
- [4] Manuel J. Fabela-Gallegos, Andres Ramirez-Valencia, Carlos A. Favela-Gallegos, David Vazquez-Vega and Miguel Martinez-Madrid, "Experimental Assessment of Baffles and Their Effect on the Longitudinal Sloshing Force in a Scaled Elliptical Tank", SAE TECHNICAL PAPER SERIES, 2005-01-3577, 2005.
- [5] 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.



- [6] Masashi Kamei, Junichi Hanai, Wataru Fukasawa and Takaomi Makino, “Establishment of a Method for Predicting and Confirming Fuel Tank Sloshing Noise”, SAE TECHNICAL PAPER SERIES, 2007-01-1538, 2007.
- [7] G. R. Yan, S. Rakheja and K. Siddiqui, “Baffle Design Analysis for a Road Tanker: Transient Fluid Slosh Approach”, SAE TECHNICAL PAPER SERIES, 2008-01-2670, 2008.
- [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.
- [9] Peiran Ding, Fan Li and Stephen Sibal, “An Integrated Fuel Tank System Simulation”, SAE TECHNICAL PAPER SERIES, 2011-01-0792, 2011.
- [10] 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.
- [11] Mahesh Balthi, Chetan Prakash Jain, Mohan Makana and Raghavendra Katti, “Development of a Quantification Methodology for Slosh Noise Associated with Dynamic Fuel Flow Behavior in Passenger Car Tank”, SAE TECHNICAL PAPER SERIES, 2011-28-0089, 2011.
- [12] Pierre De Man and Jules-Joseph Van Schaftingen, “Prediction of Vehicle Fuel Tank Slosh Noise from Component-Level Test Data”, SAE TECHNICAL PAPER SERIES, 2012-01-0215, 2012.
- [13] Eric Frank, Chris Moon Jason Rae and Michael Popovich, “Optimization of Test Parameters and Analysis Methods for Fuel Tank Slosh Noise”, SAE TECHNICAL PAPER SERIES, 2013-01-1961, 2013.
- [14] Veera Venkata Sunil Vytla and Yuya Ando, “Fluid Structure Interaction Simulation of Fuel Tank Sloshing”, SAE TECHNICAL PAPER SERIES, 2013-01-0641, 2013.
- [15] 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.