

# CFD Analysis of Sloshing Within Tank

Roshan Ambade<sup>1</sup>, Rajesh Kale<sup>2</sup>

P.G. Student, Department of Mechanical Engineering, RGIT, Mumbai, India<sup>1</sup>

Associate Professor, Department of Mechanical Engineering, RGIT, Mumbai, India<sup>2</sup>

## Abstract

Tanker trucks are the most preferred means of transporting liquid cargo and a large percentage of them carry flammable liquids. Along with the usual dangers that are associated with tanker truck accidents, there are some added dangers such as fires, explosions, leaks and spills. Liquid loads behave very differently to dry loads. When the liquid starts to slosh in the tank, it causes huge weight shifts. It builds momentum and does not settle down quickly. This paper presents the steps involved in designing a 3D model for the simulation of water sloshing phenomenon in a 2.85 x 9.81 x 9.6 m elliptical tank subjected to a sudden (impulsive) impact. The design encompasses the construction of a 3D geometry of the tank in CATIA v5 followed by the discretization of the tank using ANSYS ICEM 14.5 using hexahedral meshing technique. This mesh is imported to ANSYS FLUENT 14.5 and appropriate boundary conditions are applied to it. The results are then validated with the experimental results. Further, in this research paper the effect of using baffles at the base of the tank is simulated by using ANSYS FLUENT 14.5 and found out that the sloshing effect decreases with the use of baffles at the base of the tank. The kerosene level of 40% full is used for the analysis. Examination of CFD capability to predict the behavior of the free surface of the fluid during the container initial motion and after impact with and without baffle is the focus of this paper.

**Keywords:** water sloshing, computational fluid dynamics, baffles, 3D model

## Introduction

The liquid sloshing in a tank has been studied intensely for a long time because of its fundamental significance in flow physics and its practical importance in a wide range of applications such as in ships, satellites, rockets, trucks and even stationary petroleum containers. It is known well that the hydrodynamic load exerted by liquid sloshing can cause severe structural damage. One of the passive devices to control liquid flow and suppress liquid sloshing is the baffle

installed inside a tank. Sloshing of liquid in a container is a phenomenon encountered within various industries like chemical, petroleum, offshore, shipping etc. A free surface, i.e., a boundary between two homogenous fluids such as liquid and air, is required for sloshing to occur. The problem of water sloshing in closed containers is a big problem and has been a topic of studies since a very long time. This phenomenon is occurred due to the sudden change in loads. In order to design the equipment detailed understanding of the liquid under sloshing is required. Sloshing is the result of the acceleration/deceleration of the container body. The main concentration is on the pressure concentration of the containment walls and its temporal peaks that can reach as the tanker reaches to twice its rated load capacity. In road tankers, the liquid surfaces experiences large disturbances even on small disturbances which leads to large stability problems. Effect of sloshing can cause capsizing in ships i.e., it can turn the ship on its side or upside down. This can also affect trucks .

Traditionally the non linear potential theory and experimentation on scaled models were used to assess the sloshing loads. But the recent approach for analysis of sloshing loads is Computational Fluid Dynamics (CFD) investigated by Goddridge [1] et al. The results showed that the sloshing natural frequency and the inertia of the system is affected by the fluid level. Potential flow theory has some limitations that is why CFD is now considered as the most viable tool for analysing such fluid dynamics problems [2-5]. Several studies were conducted on sloshing of fluids and the extensive review by Ibrahim et al. [6] provide a thorough review of the subject liquid sloshing dynamics. Initial work started in the early 1960's with the study of the influence of liquid propellant sloshing on the flight performance of jet propelled vehicles. Chwang & Wang [7] applied nonlinear theory to calculate the pressure force in accelerating rectangular and circular container. Moreover, Popov et al. [8] studied the effect of acceleration and curvature on the fluid motion in rectangular containers and observed that the dynamic coefficient is influenced by the aspect ratio of fluid height to length. The study revealed that maximum sloshing occurs in square containers with 30 – 60% fluid level and that maximum forces occur at a fluid level ranging between 75 – 93%.

A similar study conducted by Ye and Birk [9] investigated the pressure variation at the walls of a horizontal cylindrical vessel during and after impact where fluid sloshing takes place at fluid levels less than 95% full. Faltinsen et al.[10] studied the transient loads on sloshing tanks and observed five distinct transient phases with different amplitudes. Chen and Chiang [11] conducted a simulation study on a simple two-dimensional rectangular tank with rigid walls subjected to horizontal and vertical accelerations using an in-viscid and incompressible fluid to examine the nonlinear behaviour of fluid motion.

This research paper focuses on the liquid flow dynamics using 3D CFD model of an accelerating fluid inside a rectangular tanker subjected to sudden impact load. This paper specifies that a road tanker is accelerating on the road and experiences a sudden impact due to collision with another body on road resulting in the sloshing of the liquid inside the container. The study involves the CFD modeling of the tanker and 3D simulation of the of the liquid inside the tanker for the pressure variation on the wall of the tanker by using Ansys ICEM and FLUENT software. The process involves geometry creation, grid generation, time step iterations followed by results, post processing and discussions on the results.

## Project Objective

A cargo truck carrying kerosene container/tank accelerates constantly at  $10 \text{ m/s}^2$  for 2 seconds and then stops suddenly. CFD analysis is carried out to study the free surface movement of the liquid kerosene inside the tank and pressure distribution on the walls of tank. The volume of kerosene inside the tank is 40% of the total container volume. The analysis is carried for two cases at various instant of time:

- Without baffles
- With baffles (thickness of the baffle is assumed to be zero)

## Validation

Before getting in to the actual problem, to examine the CFD capability to predict the behavior of the free surface of the fluid in moving container during the container's initial motion and after impact, a validation case has been chosen in which the experimental results [12] are available and CFD approach is used to match the results properly. Thus initially a validation case is solved and the learning from the validation case is then applied actual problem.

## Geometry and mesh

The 2D geometry and meshing of sloshing tank is shown as below.

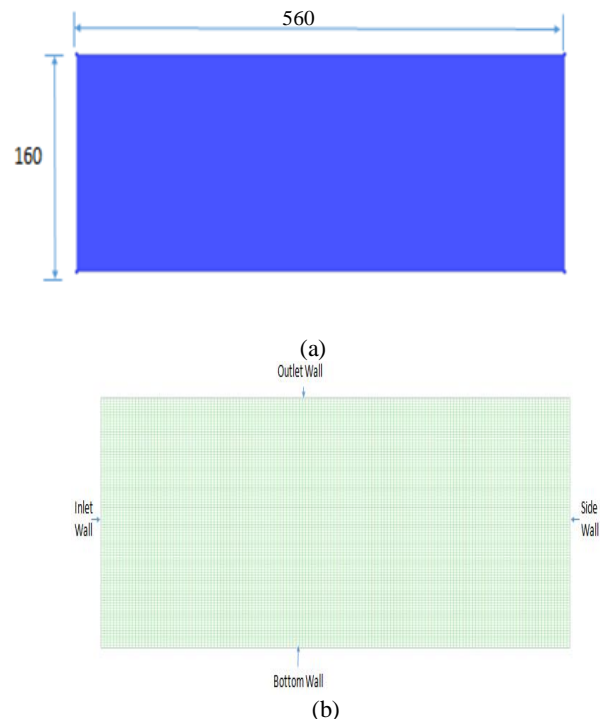


Fig 1(a) geometry (b) meshing of 2D sloshing tank

The geometry and mesh generation process is carried out by using Ansys ICEM 14.5. After grid independency test a mesh with 13261 quad mesh count is finalized. The free surface fluid motion inside the tank have modeled in two parts

1. Rectilinear motion with zero initial speed (Before Impact for 1.98 sec)
2. Post-impact fluid sloshing (After Impact from 1.98sec to 2.98 sec)

The simulation process is carried out in ANSYS FLUENT. The transient simulation is done using VOF (volume of fluid) multiphase model. Material involved is water liquid. The volume of water filled is 50% of total volume. The two phases used here are air and water liquid. The primary phase is air and secondary phase is water liquid. The X and Y gravity values are given as  $X = 0.63 \text{ m/s}^2$ ,  $Y = -9.81 \text{ m/s}^2$ . The time step used is 0.01 sec and total number of time steps are 198. The horizontal body-force was set to zero for the periods at and after impact.

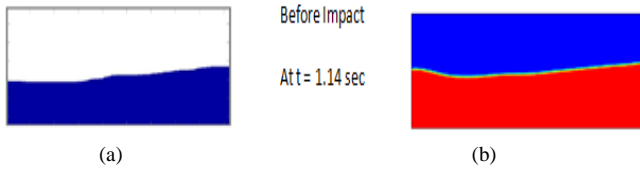


Fig 2 Comparison between (a) experimental results (b) numerical results at t=1.14 sec before impact

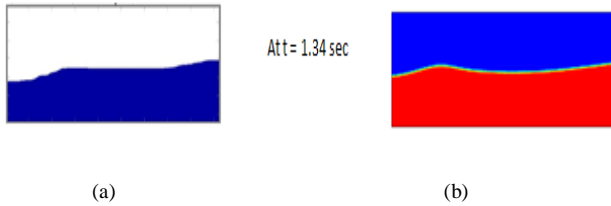


Fig 2 Comparison between (a) experimental results (b) numerical results at t=1.14 sec before impact

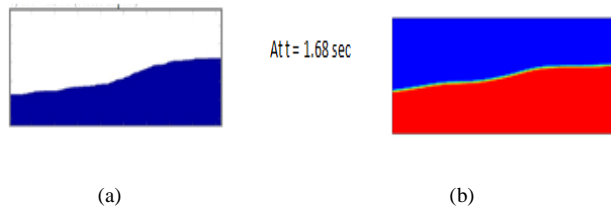


Fig 4 Comparison between (a) experimental results (b) numerical results at t=1.68 sec before impact

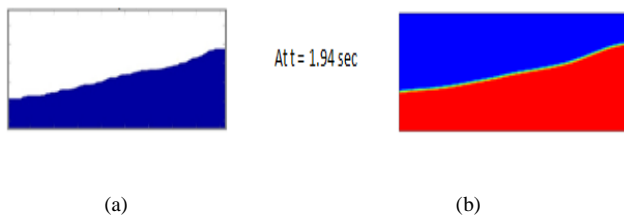


Fig 5 Comparison between (a) experimental results (b) numerical results at t=1.68 sec before impact

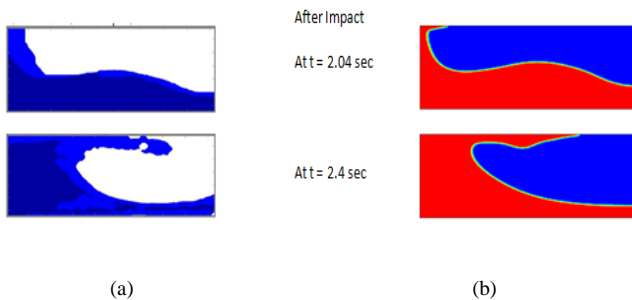


Fig 6 comparison between (a) experimental results (b) numerical results at t=2.04 sec and t=2.4sec after impact

The above comparison of flow visualization between experimental results and the CFD results shows a good agreement. This has validated CFD capability to predict the behavior of the free surface of the fluid in moving container. Thus the same solution methodology can be used to study the free surface movement of the liquid kerosene inside the tank with and without baffles.

## CFD Analysis of Fluid Sloshing within Tank

### Geometry

The geometry of the tank is generated by using CATIA v5. The geometry has the following specifications.

- Length of the cylinder = 9.6 m
- Length of major axis of ellipse = 2.85 m; length of minor axis of ellipse = 1.81 m.
- Total volume of the tank = 43.35 m<sup>3</sup>
- Volume of kerosene or 40% tank volume = 17.34 m<sup>3</sup>.

Mesh generation process is carried out using ANSYS ICEM CFD. Structured hex meshing information is given below:

- Element type – Hexahedral
- Total elements – 0.8 million cells
- Mesh quality > 0.48 (0-worst quality, 1-best quality)

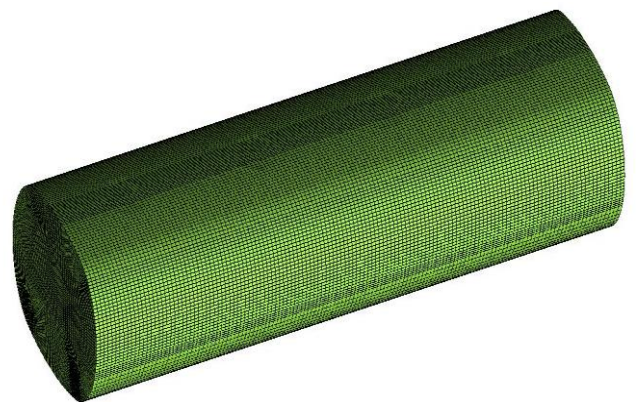


Fig 7 Structured mesh of tank

Simulation Set-up

The simulation process is carried out in ANSYS FLUENT. The simulation is done using VOF (volume of fluid) multi-phase model. The simulation was done for 4 seconds of sloshing with a velocity profile such that the velocity is increasing from 0 to 2 seconds with an acceleration of  $10 \text{ m/s}^2$  and for the next two seconds i.e., 2 to 4 seconds the velocity is zero. The energy equation is turned off because there is no exchange of heat energy or the process is assumed isothermal.

The two phases used here are air and kerosene. The primary phase is air and secondary phase is kerosene. The properties of the two phases are given below.

**Air properties:**

Density =  $1.225 \text{ kg/m}^3$   
 Viscosity =  $1.7894 \times 10^{-5} \text{ kg/m-s}$ .

**Kerosene properties:**

Density =  $780 \text{ kg/m}^3$   
 Viscosity =  $0.0024 \text{ kg/m-s}$ .

**CFD Results:-**

The contours of the sloshing at  $t = 0.5 \text{ s}$ ,  $1.0 \text{ s}$ ,  $1.5 \text{ s}$ ,  $2.0 \text{ s}$ ,  $2.5 \text{ s}$ ,  $3.0 \text{ s}$ ,  $3.5 \text{ s}$ ,  $4.0 \text{ s}$  are shown below for both the cases i.e., without baffle and with baffle.

Comparison of contours before break and after break is done as shown below:

Before applying break:



Fig 8 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=0.5 \text{ sec}$  before applying break

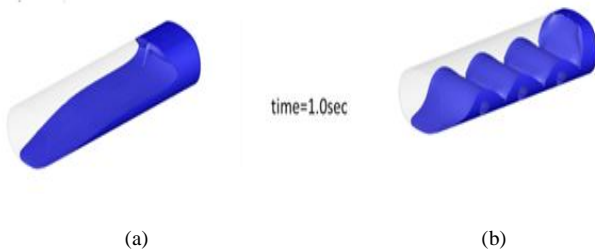


Fig 9 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=1.0 \text{ sec}$  before applying break



Fig 10 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=1.5 \text{ sec}$  before applying break

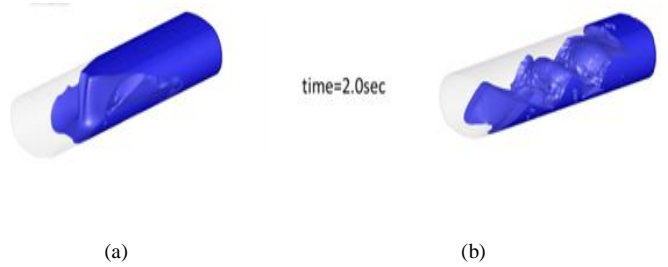


Fig 11 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=2.0 \text{ sec}$  before applying break

After applying break:



Fig 12 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=2.5 \text{ sec}$  after applying break



Fig 13 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=3.0 \text{ sec}$  after applying break



Fig 14 Comparison between CFD results of (a) without baffle (b) with baffle at  $t=3.5 \text{ sec}$  after applying break



Fig 15 Comparison between CFD results of (a) without baffle (b) with baffle at t=4 sec after applying break

Pressure variation on tank surfaces:

To evaluate the “strength” of sloshing, pressure variation on the tank surfaces with respect to time is plotted. To evaluate effect of baffles, the maximum pressure on the tank walls at given time is plotted. When there is more sloshing, the tank fluid moves with more velocity and hence has more dynamics pressure. This fluid then exerts this pressure on the tank wall. Due to presence of baffles, the sloshing reduces and hence the dynamic pressure exerted on the tank walls.

Fig 16 shows pressure variation on tank walls with respect to time when there were no baffles in the tank. At the start, fluid is at rest and hence the maximum pressure is near to zero. The maximum pressure on the tank surface is seen between 2 to 2.5 seconds. It is seen that the maximum pressure is about 2.25 bars is exerted on the walls. It is also seen that there are local picks in the maximum pressure exerted on tank walls.

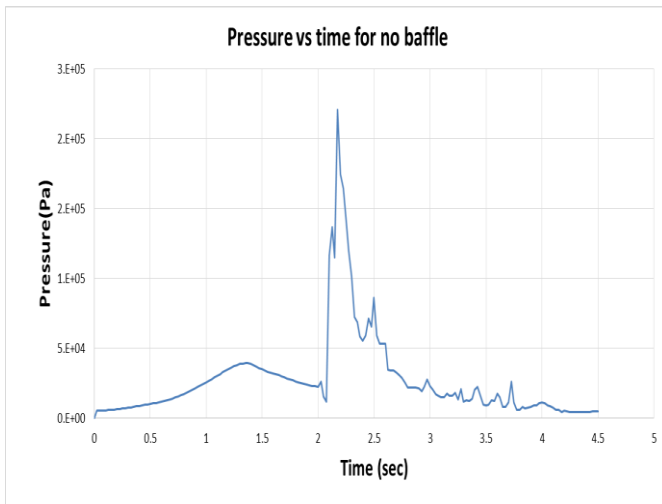


Fig 16: Pressure variation on tank walls with no baffles

Fig 17 shows pressure variation on tank walls with respect to time when baffles were inserted in the tank. The overall pressure variation pattern is same as that of without baffles.

With about zero pressure during the start, the maximum pressure exerted is 1.4 bars. The maximum pressure on the tank surface is seen between 2 to 2.5 seconds.

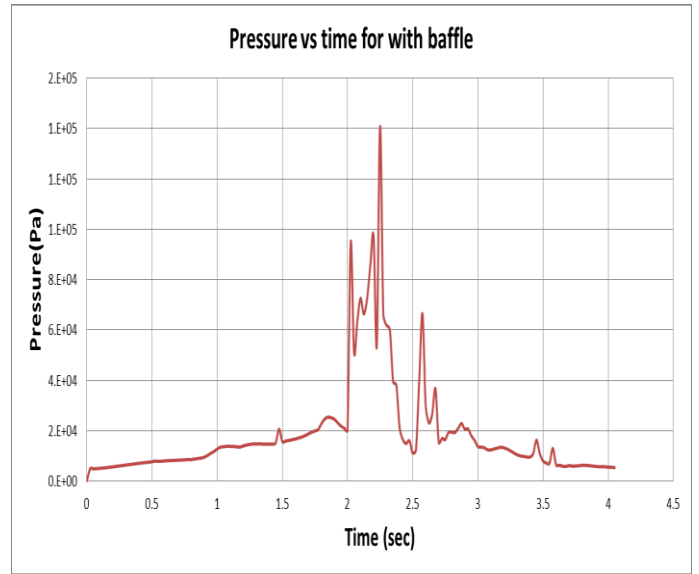


Fig 17: Pressure variation on tank walls with baffles

Fig 18 shows the comparison between tank with and without baffles. It is seen that the maximum pressure exerted on the tank walls is reduced when baffles are used. Without any baffles, the maximum pressure on the wall is 2.25 bars, which is reduced to 1.4 bars with baffles. This shows that the intensity of sloshing has reduced with use of baffles. It’s not only the maximum sloshing, but overall sloshing is dampened with usage of baffles.

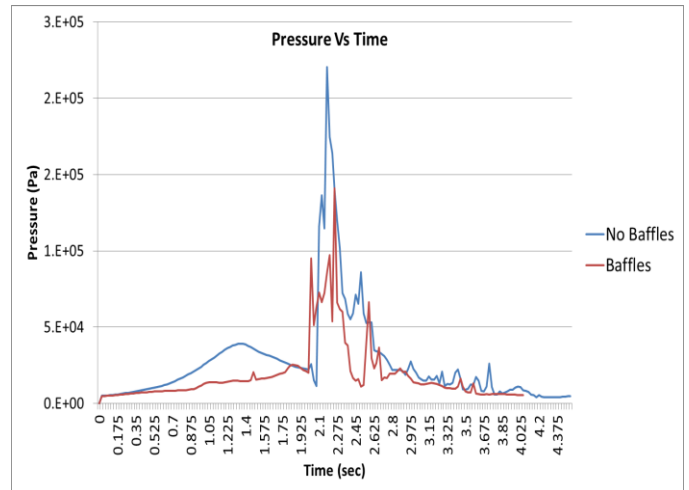


Fig 18: Pressure variation comparison

**CONCLUSION**

The comparison of pressure contours clearly shows that the pressure distribution on the tank surface is much better when baffles are present. Thus, tank with baffle is much better design for reduced sloshing. Volume of fluids (VOF) is suitable to predict the maximum tank sloshing, the wall stress and liquid overflow. It is also concluded that baffles are effective means of preventing or minimising sloshing intensity. However, baffles can change the model characteristics of a structure and increase its total weight. And there exist many design variables such as size, interval, installation position, the number of baffles having an effect on sloshing, which is left for the future work.

**References**

- [1] B. Godderidge, M. Tan, S. Turnock, and C. Earl. "A Verification and Validation Study of the Application of Computational Fluid Dynamics to the Modelling of lateral Sloshing" Ship Science Report No 140, Fluid Structure Interaction Research Group, University of Southampton, August 2006.
- [2] Simulation of Fluid Sloshing in a Tank by Mr. G.C. Koli, Prof.V.V.Kulkarni, Non-Member, IAENG Proceedings of the World Congress on Engineering 2010 Vol II WCE 2010, June 30 - July 2, 2010, London, U.K.
- [3] Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation M. Eswarana, U.K. Saha, D. Maity Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati 781 039, Assam, India Department of Civil Engineering, Indian Institute of Technology Kharagpur, Kharagpur 721 302, West Bengal, India.
- [4] A. Cariou, and G. Casella. "Liquid Sloshing in Ship Tanks: a Comparative Study of Numerical simulation", Marine Structures, Vol. 12, pp. 183-198, 1999.
- [5] A.C. Kassinos, and J. Prusa. "A Numerical Model for 3-D Viscous Sloshing in Moving Containers" ASME-Publications FED, Vol. 103, pp. 15-86, 1990.
- [6] R.A. Ibrahim, V.N. Pilipchuck, and T. Ikeda. "Recent Advances In Liquid Sloshing Dynamics" Applied Mechanics Reviews, Vol. 54, No. 2, pp. 133-199, 2001.
- [7] A.T. Chwang, and K.H. Wang. "Nonlinear Impulsive Force on an Accelerating Container" J. of Fluids Eng., Vol. 106, pp. 233-240, 1984.
- [8] G. Popov, S. Sankar, T.S. Sankar, and G.H. Vattas. "Liquid Sloshing In Rectangular Road Contain-

- ers", Computers Fluids, Vol. 21, No. 4, pp. 551-569, 1992.
- [9] Z. Ye, and A.M. Birk. "Fluid Pressures in Partially Liquid-Filled Horizontal Cylindrical Vessels Undergoing Impact Acceleration," Journal of Pressure Vessel Technology, Vol. 116 No. 4, pp. 449-459, November 1994.
- [10] O. Faltinsen, O. Rognebakke, N. Alexander, and A.N. Timokha. "Resonant three dimensional nonlinear sloshing in a square-base basin, part 2. effect of higher modes", Journal of Fluid Mechanics, Vol. 523, pp. 199-218, 2005.
- [11] B.F. Chen, and H.W. Chiang. "Complete 2D and Fully Nonlinear Analysis of ideal fluid in tanks", Journal of Engineering Mechanics, pp. 70-78, 1999.
- [12] L. Khezzar, A. C. Seibi & A. Goharzadeh "Water Sloshing in Rectangular Tanks – An Experimental Investigation & Numerical Simulation" International Journal of Engineering (IJE), Volume (3) : Issue (2).

**Biographies**

**ROSHAN AMBADE** received the B.E. degree in Mechanical Engineering from the Shivaji University, Kolhapur, Maharashtra state, in 1993. He is Post graduate student at Rajiv Gandhi Institute of Technology, Mumbai. His research area is analysis of thermo fluid systems by CFD. Mr. Roshan Ambade may be reached at [roshanambade@yahoo.co.in](mailto:roshanambade@yahoo.co.in)

**RAJESH KALE** received the B.E. degree in Mechanical Engineering from the University of Amravati, Maharashtra, the M.E. degree in Mechanical Engineering from the University of Gulbarga, Karnataka. He is an associate Professor of Mechanical Engineering at Rajiv Gandhi Institute of Technology, Mumbai. His teaching and research areas include thermal Engineering, Heat and Mass transfer, Thermodynamics, Electricity Generation sector, Design of Thermal systems. Prof. Rajesh Kale may be reached at [rajesh.kale@mctrigit.ac.in](mailto:rajesh.kale@mctrigit.ac.in)