OPTIMIZATION OF FUEL TANK USING BAFFLES TO REDUCE SLOSHING EFFECT

[1]Ujjal Kalita, [2]Twisampati Roy Chowdhury, [3]Shahid Khan,
[4]Alok Kumar Mishra, [5]Santosh Kumar Behera, [6]Manminder Singh Malhi
[1]ujjal.19355@lpu.co.in, [2]mishraalok721@gmail.com
[1]Assistant Professor, [2, 3,4,5,6] UG Student, School of Mechanical Engineering, Lovely Professional University, Jalandhar, India.

*Abstract :*Sloshing is a phenomenon which is seen widely occurring in liquid filled tanks which have large volume. Sloshing is the phenomenon which is the interaction of fluid and the structure of the tank. In this paper we are undertaking the task to reduce the sloshing effect in aircraft fuel tanks by introducing two different baffle designs. A baffle is a device which is used to restrict the flow of a fluid. In this research paper we are using the Boeing 747-200 which is one of the largest passengers carrying aircraft currently in operation. The designs were constructed on the Solidworks, and for simulating the effects of sloshing we are using Ansys simulation software. The computational Fluid Dynamics module is best to simulate conditions for the sloshing as various parameters can be controlled and dictated accordingly. The paper consists of results and comparisons between the current model in operation and the two designs of baffles used to reduce the effect of sloshing. The results are promising and saw that the designs are successful in achieving the objective of the paper. In oval shaped design of fuel tank and in partial horizontal and vertical combination design of fuel tank, we have managed to reduce the turbulence kinetic energy by 78.704% and 99.935% respectively. Further, attempts can be undertaken to improve the designs of the baffles used and achieve further reductions in sloshing.

***IndexTerms* - Sloshing effect, Boeing, Fuel tank, design.**

I. INTRODUCTION

Sloshing can be observed in all vehicles undergoing accelerated motion. The study of fluid- structure interaction or sloshing is of very importance in industries like aircraft, aerospace transportation chemical, petroleum, offshore, shipping. The ships and submarines are also affected by sloshing as the large ballast tanks are filled with fluid at times. These tanks may cause the vessel to experience rolling motions which can contribute to capsize the ship during high waves. Larger volumes are more prone to sloshing the passenger aircrafts have wings which are very flexible structurally, which enables it to deform significantly when encountered with a strong atmospheric turbulence. This flexibility allows the fuel in the wings to move turbulently and cause sloshing. Sloshing of fluid inside the tanks can create very huge problems for the aircraft, which occurs due to the sudden change in loads acting on the vessel. To design the equipment detailed understanding of the liquid under sloshing is required. Violent and long-term sloshing can make the aircraft very unstable. The fluttering of the wings can make the structure week and prone to accidents. The main objective of the paper is to find suitable solution in the reduction of turbulent kinetic energy of the fluid.[1]Ahmed Adel Mohamed Abdel-Raheem and his research partners in paper 'Aerodynamic study of Boeing 747-400 aircraft' shows that Boeing 747 uses "BACXXX" airfoil. Hence, in reference with the paper we are also conducting our study in the same airfoil. We have designed and carried out simulation for sloshing in that particular airfoil section in Ansys 19.2 CFX simulator using Volume of Fluid (VOF) multiphase model for liquid Jet A fuel [2] used in the aircraft.In the article 'Aviation Fuels Technical Review by Chevron', we have found that Boeing 747 uses Jet-An oil as fuel so we are also taking the same reference. From the paper 'Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation' by M. Eswarana, U.K. Saha, D. Maity. We can observe sloshing effect in most of the vehicles that contains a fluid tank and undergoes sudden acceleration [3]. Using these modules, we can compare graphical data in a variety of ways and get a closer look on how the fluid is behaving in different parameters.

II. MODELLING AND DESIGNING

Below an original 3D model of wing section fuel tank of B747 is shown.

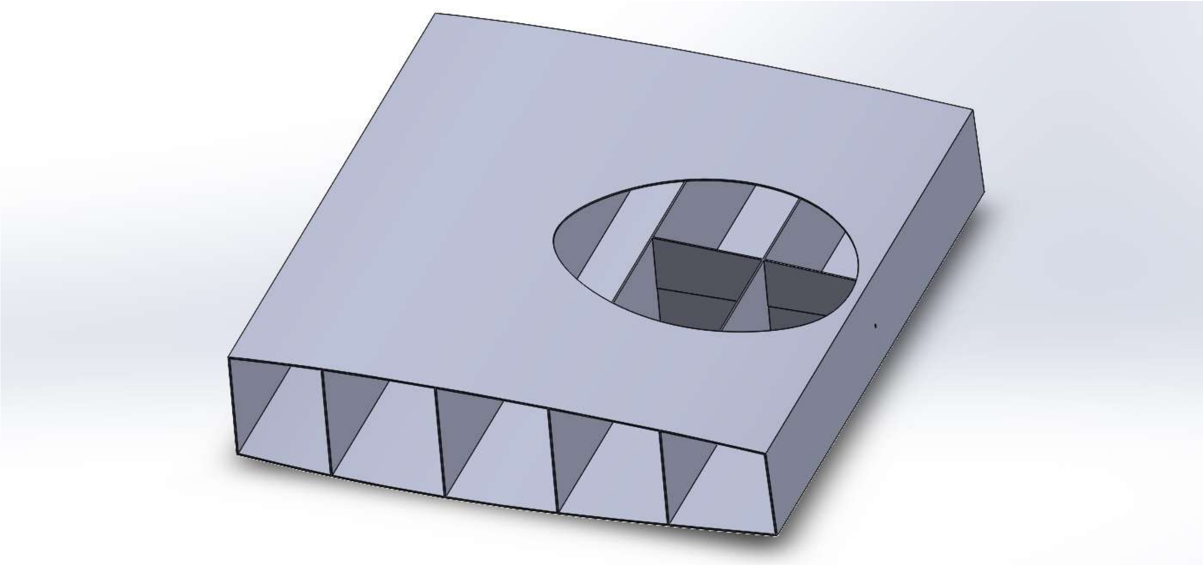

Figure 1: 3D model of the fuel tank of wing section of B747

BACXXX airfoil is the airfoil of wing of Boeing 747. From the research done by J.R. Cho, H.W. Lee, S.Y. Ha, 2005, "Finite element analysis of resonant sloshing response in 2d baffled tank" we have found that it has a maximum thickness of 11.3% at 35% of chord and maximum chamber of 1.4% at 15% of chord [5].

Designing – From the paper written by RoshanAmbade, Rajesh Kale 2013 "CFD analysis of sloshing within tank",we have used the dimensions of [6]. A fairly middle section / portion of the airfoil is taken to model this structure with accurate dimensions. We have used Ansys 19.2[7] as our analysis software to analyze the flow in the fuel tank. 1m water level is taken in the tank during simulation. We have stimulated by assuming the aircraft height is 9,000m [1]. The chord length of the airfoil is 14.6304m [1]. By the reference of the paper 'Fuel Tank Modeling and Fuel Temperature Simulation of an Aircraft in Steady-State and Transient-State Methods' by KANG Zhenye, LIU Zhenxia, REN Guozhe, LV Yaguo, temperature inside the tank was kept at 276K [8]

III. COMPUTATIONAL METHODOLOGY

Figure 2 represents the original 2d view of structure of fuel tank designed in solidworks. It is then imported in Ansys and simulation was carried out. Then we did the same thing for other two models and from the visuals or data generated from the simulation of CFX, we have compared them. We have set both the fuel tanks identical boundary conditions and gave a velocity in longitudinal direction. Due to the initial velocity, the fluid inside the tankexperienced a change in motion and to retaliate, it generated a sloshing phenomenon.

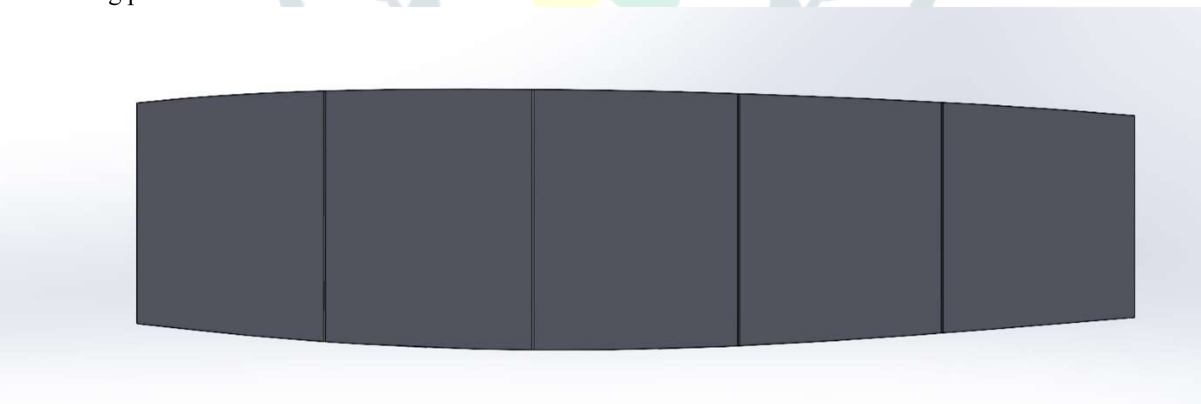

Figure 2: 2D cross section of fuel tank

EQUATION USED

Continuity Equation

$$\oint \frac{\partial \rho}{\partial t} dV + \int \Delta(\rho V c) dA = 0$$

Momentum Equation

$$\frac{d}{dt} \iiint \rho V dv + \oiint \rho(V.n) V dA = \oiint -pndA + \iiint \rho g d$$

Process

All the process which we have went through for the simulation, is stated below.

1) Geometry – Already discussed above, we have designed three designs, one is the fuel tank of Boeing747 and another two are our modified design to reduce sloshing effect.

2) Mesh –Firstly, we renamed the two symmetric faces of the airfoil as ‘Symm1’ and ‘Symm2’ and the remaining boundaries as ‘Wall’ for future reference.

Secondly, we chose ‘Sweep’ method for meshing and went for manual source. We opted free face mesh type as the front face i.e ‘Symm1’ face. We then selected the front face ‘Symm1’ for face sizing, and put the face size mesh as 50mm. a decent mesh was formed.

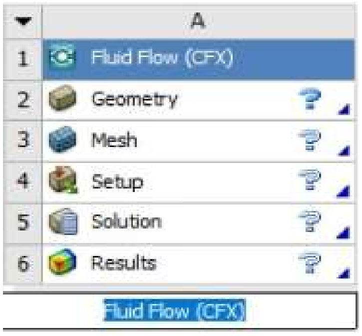

Figure 3: All the steps used for the simulation

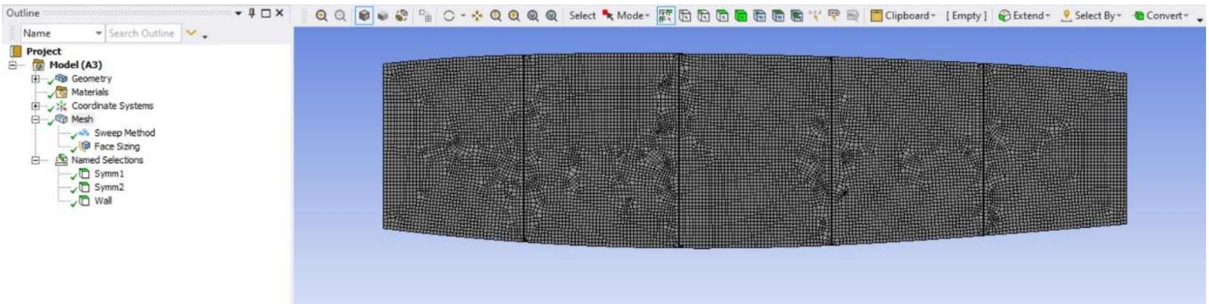

Figure 4: Meshed body of B747 fuel tank cross section

3) Setup – In setup, we chose analysis type as transient and set the total time as 20s. Then gave the timestep 0.1 and started the simulation from0s. We specified 3 boundaries for the section as ‘Symm1’ and ’Symm2’ as symmetric and ‘Wall’ as wall. We set time step interval as 2.

Step 4: We defined 4 equations for the sloshing simulation which holds as the backbone for the sloshing simulation, those are as follows:

a) HydroP – fluidDen*g*(fluidHt-y)*fluidVF
b) fluidDen – 780(kg/m^3) [2]
c) fluidHt – 0.2(m)
d) fluidVF – if(y<fluidHt,1,0)*if(y>-1.1[m],1,0)

Where Hydrop – Hydrostatic pressure; fluidDen – Density of the fluid;fluidHt – Level of the fluid inside the tank; fluidVF – This defines fluid volume fraction.
In default domain section, we defined two fluids, air and Jet A fuel (B747 fuel) and set the buoyancy reference density, 1.185 (Kg/m^3). The importance of k-epsilon model is very relevant towards this simulation, so here we have used the same for the simulation.

From the Research done by Anna Maiorova, AleksandrVasil'ev and OganesChelebyan on ‘Biofuels in Aircraft Engines’, it is found that surface tension coefficient between the fluid layer is 0.028 (N/m) [9]. An initial velocity of 5m/s was imparted along the chord line axis, which defines that when the aircraft is stationary and suddenly increasing its velocity, it will face a sudden change in motion which will initiate the sloshing motion.

We did the whole simulation in 276K temperature and HydroP function as hydrostatic pressure.

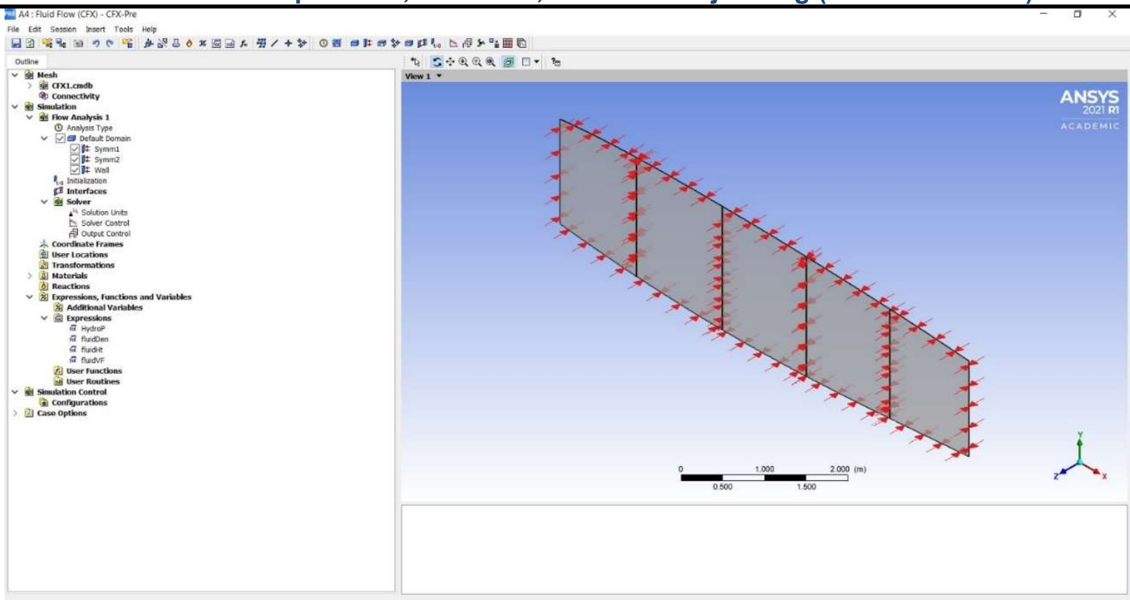

Figure 5: Setup methods in Ansys CFX

5) We did the same with our version of design got simulation that is generated from CFX solver. Below are the screenshots of the simulation that we got from our simulation.

Below are the screenshots of sloshing simulation of actual model of B747 wind section of fuel tank at different timesteps.

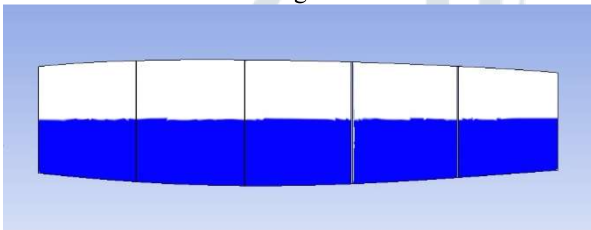

Figure 6 : timestep=0

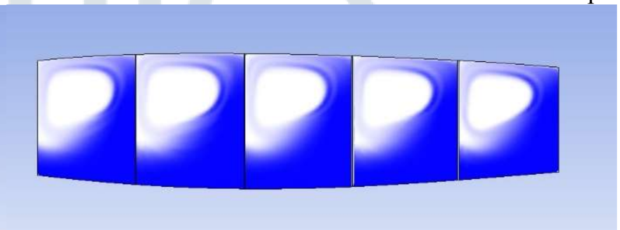

Figure 7 : timestep=2

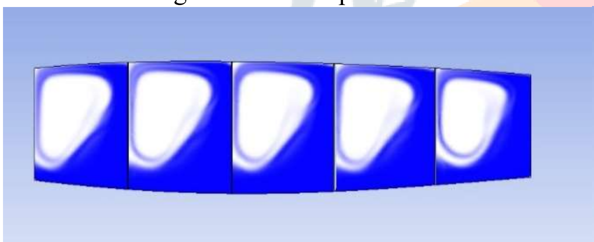

Figure 8 : timestep=4

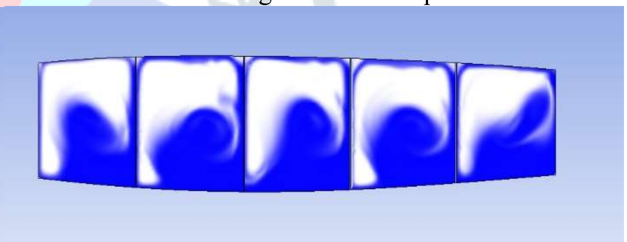

Figure 9: timestep=8

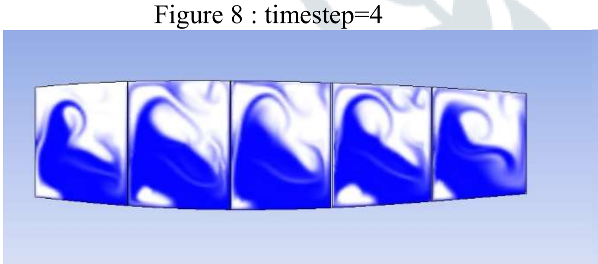

Figure 10 : timestep=12

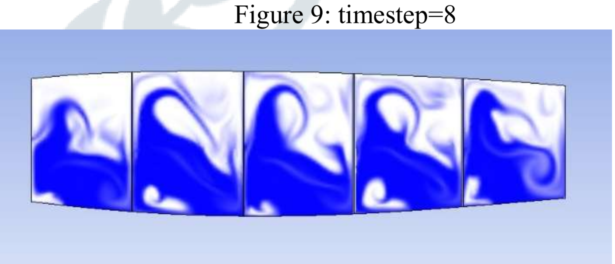

Figure 11: timestep=14

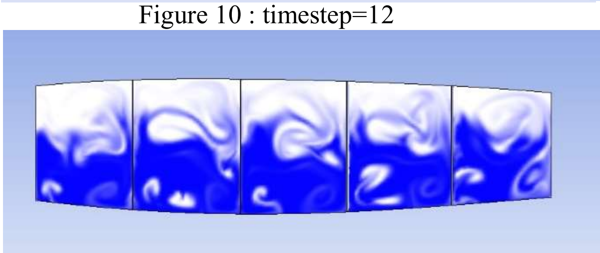

Figure 12: timestep=18

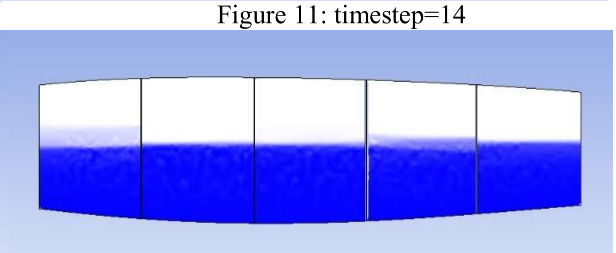

Figure 13: timestep=200

All the screenshots given here from figure 17 to figure 24 are from the actual B747 modelfrom fuel tank of wing section. At figure 6 i.e., in timestep 0, no velocity is being introduced so no net motion of the liquid particles inside the tank are observed. In figure 7which represents timestep 2, it is observed that the fuel is forming a vortex like structure and hitting the walls of the box shaped sections of fuel tank. This hitting of fuel creates much disturbances and turbulence inside the fuel tank. In figure 8which is representing timestep 4, a strong prominent vortex shape is forming and that is imparting lots of pressure on the wall of the fuel tank, thus creating much unwanted turbulence. In figure 9, representing timestep 8we can still see the movement of fuel inside the tank. Figure 10 and figure 11 representing timestep 12 and 14, after getting striked at the right sided wall of the fuel tank box section, now it gets hit by left side also. This leads to a continuation of a vigorous turbulence. Till timestep 18 also in figure 12, the fuel is not settling down and not getting neutralized. Finally, the sloshing motion comes to an end in 200th timestep in figure 13.

Below are the screenshots of sloshing simulation with elliptical baffles at different timesteps.

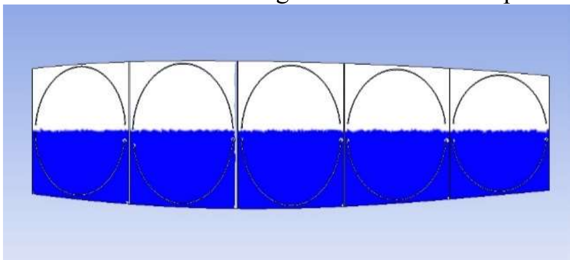

Figure 14 : timestep=0

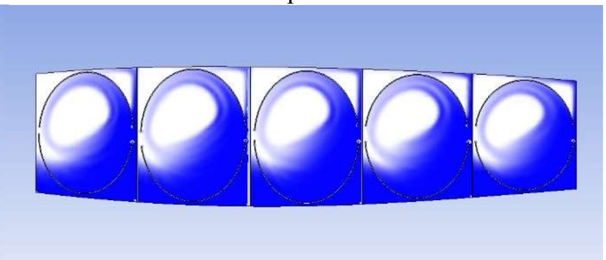

Figure15 : timestep=2

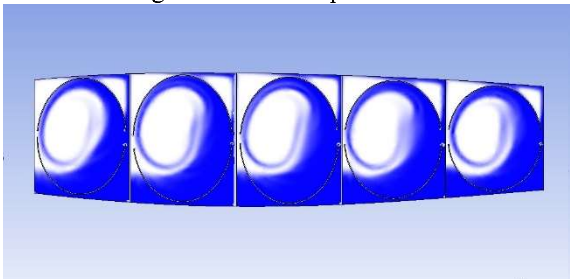

Figure 16 : timestep=4

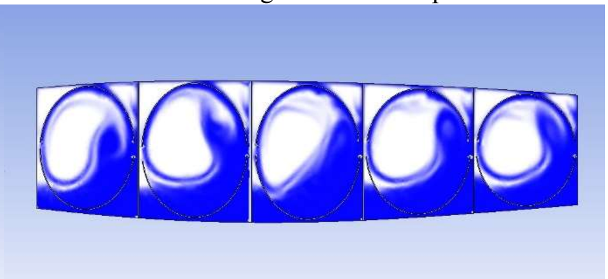

Figure17 : timestep=8

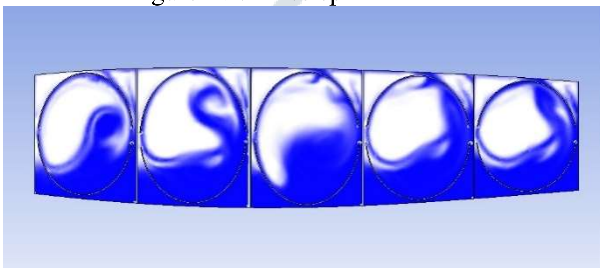

Figure 18 : timestep = 12

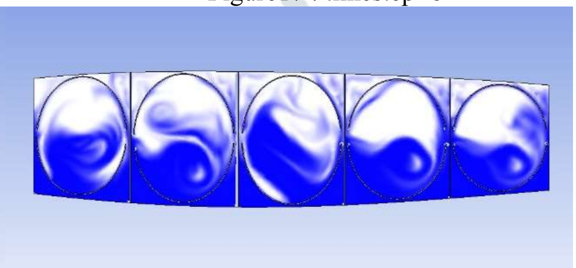

Figure 19 : timestep=14

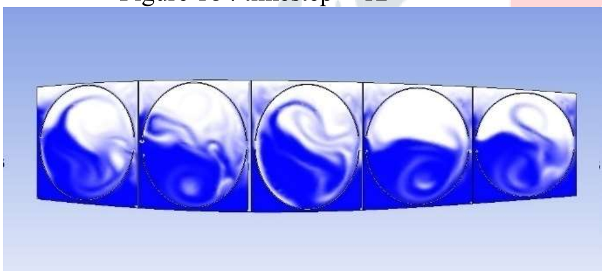

Figure 20 : timestep=18

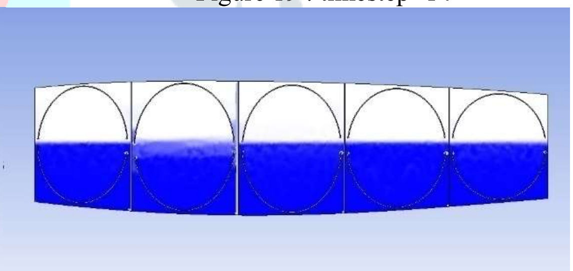

Figure 21 : timestep=200

This simulation is a collective of timesteps of simulation of elliptically curved baffles. In fig. 14 we can see the fluid is motionless at timestep 0. At timestep 2 (fig. 15) the fluids climb up from the right side of the baffle and moves forward towards the left side of the tank making almost a full loop around it. In the next fig. 16 the front of the fluid wave crashes on the bottom of the tank and rolls inwards. In fig. 17 the fluid rolls back and disturbs the bottom of the tank and makes it more turbulent. At timestep 12 in fig. 18 the liquid seems to make another half wave emerging from the right and climbing up to the top of the curve. The first and the second wave seems to have combined and appear to have high liquid volume. In the next figure (Fig. 19) the second wave has collapsed half way which suggests the momentum and energy of the fluid is lost to much extent but still it is very turbulent in the central part. In fig. 20 the fluid is splashing around in different directions but seems to be collecting back into the central region. At timestep 200 which is figure 21 the liquid is calm and is back in its initial state as it was at timestep 0. In this simulation we can see that the liquid mostly is contained within the walls of the elliptical baffle and the force of the liquid is not directed to the walls of the outside tank. This means that the disturbances caused are negligible and the design seems effective in reducing sloshing.

Below are the screenshots of sloshing simulation with a combination of partial horizontal and partial vertical baffles at different timesteps.

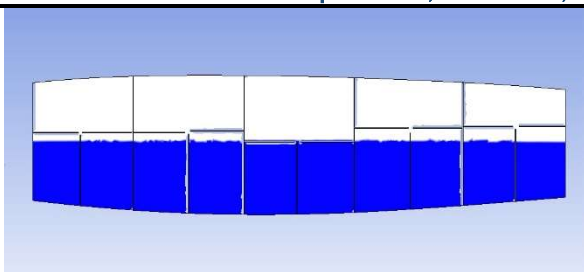

Figure 22 : timestep=0

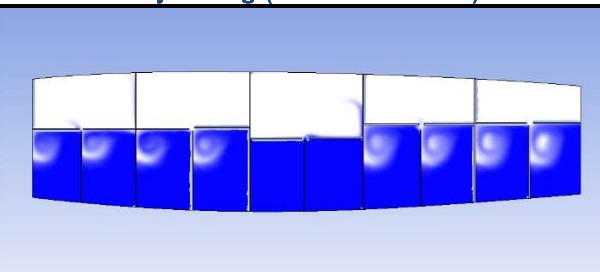

Figure 23 : timestep=2

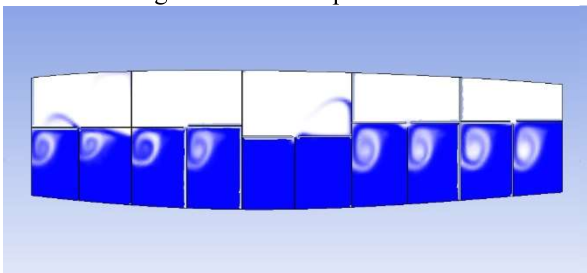

Figure 24 : timestep=4

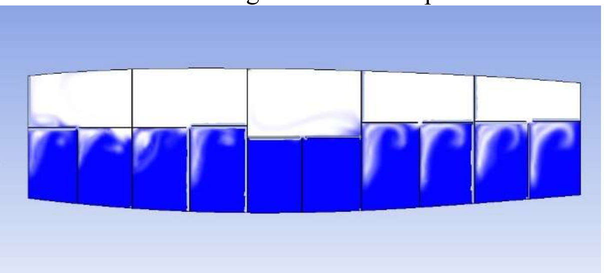

Figure 25: timestep=8

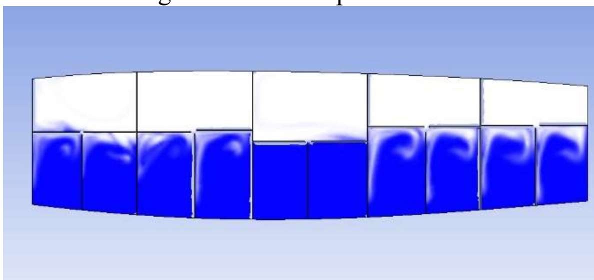

Figure 26 : timestep=12

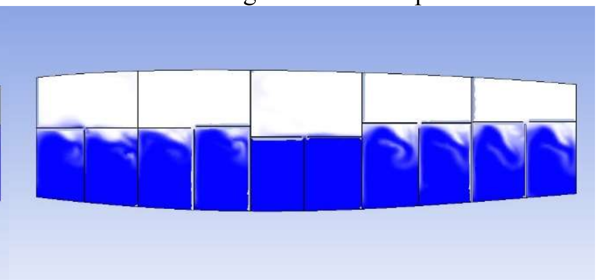

Figure 27 : timestep=14

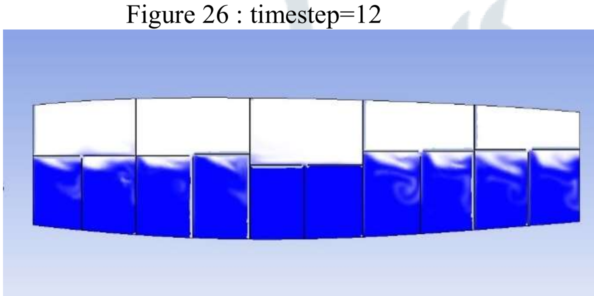

Figure 28 : timestep=18

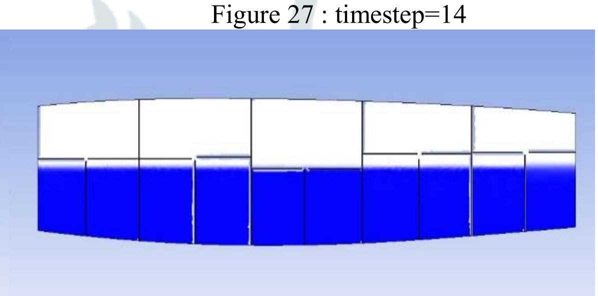

Figure 29 : timestep=200

In figure 22 through 29 the sloshing simulation with a horizontal baffle working in tandem with a vertical baffle. The simulation is run for twenty seconds and is of two hundred timesteps. At timestep 0 (Fig. 22) the liquid is completely calm and is motionless as no force is acting on it. in the next figure (fig. 23) at timestep 2, the tank is in motion and the liquid is seen to be reacting along with-it showing motion and forming a forward moving wave. At timestep 4 in figure 24 the liquid is rolling back in on itself. It is not very turbulent as the liquid is restricted and its motion is limited. In Fig. 25, the liquid wave is collapsing and mixing with the fluid in the center. The fluid in the upper section is moving backwards from the wave collapse. Figure 26 is more or less same as the last one since the fluid is still moving and rolling inwards. In figure 27 the liquid is not very turbulent and is seen to be coming back down to a stop, a little splashing can be seen around the horizontal baffle. In the next figure (Fig. 28) the liquid in the middle section of the tank is back to its initial condition while the rest of the section is not very turbulent and the splashing in the liquid is also very less compared to the earlier figures. In the last figure 29 the liquid is calm and is back in its initial position as it was in the figure 22, completing the two hundred timesteps

IV. Conclusion

- The minimum turbulence kinetic energy in original model is 3.55e-4, in oval shaped baffle tank it is 7.56e-5and in partial horizontal-vertical designed tank it is 2.29e-7. This simulated data proves that our two designs are so effective in reducing sloshing effect.
- From our version of design i.e., by adding an oval kind of section inside the baffle it is thoroughly observed that in the fuel tank we have minimized the sloshing effect, and most of the sloshing movement is contained within the oval shaped structure only so almost no force is imparted on the walls of the fuel tank thus creating lesser disturbances.
- From the screenshots of our second model simulation, it is thoroughly observed that as the fluid is not getting enough space to move around freely because a combination of horizontal and vertical baffles which are placed in such a way that it obstructs the fluid flow inside the tank, it fails toaccumulate and hit the wall at once, disturbance cause due the force impact on the walls of tank is much less now.

References

[1] Ahmed Adel Mohamed Abdel-Raheem, PoulaAwad Suleiman Hanna, Islam Atef Mohamed Abdel-Hameed, Aerodynamic Study of Boeing 747-400 Airfoil.
[2] Aviation Fuels Technical Review by Chevron

[3] M. Eswaran, U.K. Saha and D. Maity, Effect of baffles on a partially filled cubic tank: Numerical simulation and experimental validation Computers and Structures 87, 198–205, 2009.
[4] J.R. Cho, H.W. Lee, S.Y. Ha, 2005, "Finite element analysis of resonant sloshing response in 2d baffled tank" J. Sci. &Vib., Vol 288, Issues 4–5 Pages 829-845
[5] RoshanAmbade, Rajesh Kale 2013 "CFD analysis of sloshing within tank", IJIRTS, Vol 2, Number 3
[6] Fuel Tank Modeling and Fuel Temperature Simulation of an Aircraft in Steady-State and Transient-State Methods by KANG Zhenye, LIU Zhenxia, REN Guozhe, LV Yaguo
[7] Biofuels in Aircraft Engines by Anna Maiorova, AleksandrVasil'ev and OganesChelebyan; DOI :http://dx.doi.org/10.5772/59871
[8] Bernhard Godderidge , Stephen Turnock , Mingyi Tan , Chris Earl, 2009 "An investigation of multiphase cfd modelling of a lateral sloshing tank" *Comp. & Fluids,*Vol**38**, Issue 2, Pages 183-193
[9] J.R. Cho, H.W. Lee, S.Y. Ha, 2005, "Finite element analysis of resonant sloshing response in 2d baffled tank" *J. Sci. &Vib.,* Vol**288**, Issues 4–5 Pages 829-845
[10] RoshanAmbade, Rajesh Kale 2013 "CFD analysis of sloshing within tank",*IJIRTS,*Vol**2**, Number 3
[11] Fuel Tank Modeling and Fuel Temperature Simulation of an Aircraft in Steady-State and Transient-State Methods by KANG Zhenye, LIU Zhenxia, REN Guozhe, LV Yaguo
[12] Pizzoli, M.Aerotec.Missili Spaz, 2020, "Investigation of sloshing effects on flexible aircraft stability and response", *Springer*, Vol**99**, Pages297–308
[13] An Introduction to Computational Fluid Dynamics: The Finite Volume Method by HenkKaarleVersteeg, WeeratungeMalalasekera
[14] F. Gambioli , R. AlegreUsach , J. Kirby , T. Wilson , P. Behruzi,2019, "Experimental evaluation of fuel sloshing effects on wing dynamics" *IFASD,* Vol-**142,** Savannah, Georgia, USA
[15] Anna Maiorova, AleksandrVasil'ev and OganesChelebyan, Biofuels in Aircraft Engines, Intechopen publishing
[16] Oddm. Faltinsen and Alexander N.Timokha, 2010, "A multimodal method for liquid sloshing in a two- dimensional circular tank" *J. Fluid Mech.*, Vol. **665**, Pp. 457–479
[17] Oddm. Faltinsen and Alexander N.Timokha, 2001, "An adaptive multimodal approach to nonlinear sloshing in a rectangular tank" *J. Fluid Mech.*, Vol**432**, Pp. 167–200
[18] J. F. Kuo, R. B. Campbell, Z. Ding, S. M. Hoie, A. J. Rinehart, R. E. Sandström And T. W.Yung, 2009, "LNG tank sloshing assessment methodology—the new generation",*ISOPE,* Vol**-09-062**
[19] Gurinder Singh Brar, Simranjit Singh, 2014, "An experimental and cfd analysis of sloshing in a tanker",*ICIAME,* Vol**14**, Pages 490-496
[20] Franco Mastroddi, Francesco Saltari, Michael Wright, Arnaud G. Malan, Simone Simeone, Francesco Gambioli, 2019, "Aircraft fuel sloshing roms for aeroelastic analyses" *IFASD,* Vol**92**, Savannah, Georgia, USA