NMTFD-2 Deliverable Task 2 Report: Fuel Tank Sloshing

Onkar Marathe	(23146448)	onkar.marathe@fau.de
Gokhul Shriram Ravi	(23212226)	gokhul.shriram.ravi@fau.de
Adithya Umanath Rai	(22811294)	adithya.rai@fau.de

July 24, 2023

Introduction:

The following study requires us to simulate the sloshing of a fuel tank. The task required us to design a case study that emulates the results provided in the research paper titled " Validation of Slosh Modeling Approach Using STAR-CCM" by David J Benson and Wanyi Ng. The methodologies used in this deliverable follow that of the paper given. The results of our task as well as how it compares to the research paper will be discussed in the sections below.

(I) Computational Domain

The computational domain consists of a right circular cylinder inside which a liquid propellant is made to slosh. The tank is half full and the interface angle between the 2 fluids is set at 3 degrees. The propellant is Nitrogen Tetraoxide with a density of $1450 Kg/m^3$ and the gas used is Nitrogen with a density of 1.19 Kg/m^3. The model is simulated under a gravitational force of $9.81 m/s^2$ in the downward direction. The surface tension between the two fluids is 0.07. The tank has a diameter of 1.2m and a height of 1.4m. The axis of the tank is oriented along the Z-axis as shown in the figure below. The propellant is given by the area shaded in red, while the blue area represents Nitrogen gas in the propellant tank. The white area signifies the interface between propellant and gas.

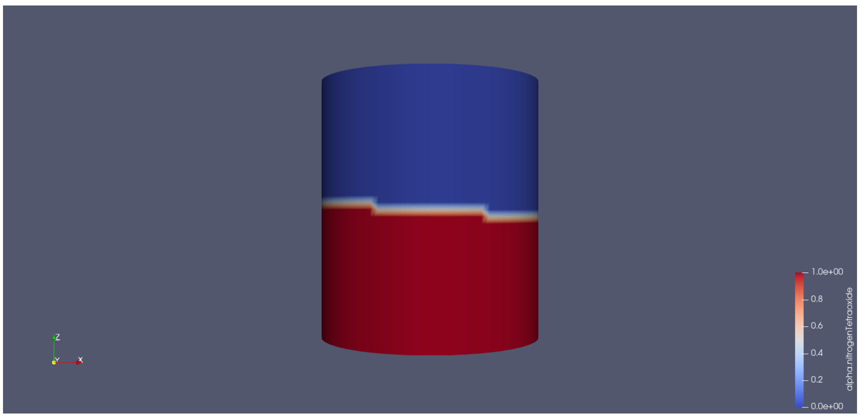

Figure 1: Right circular cylinder bare tank

(II) Governing Equations and CFD Model

The Volume of Fluid (VOF) approach is used in this study. The VOF approach is an interface-capturing method where an additional equation for volume fraction α is solved for each control volume apart from the conservation equations for mass and momentum. α represents the volume fraction, i.e., the ratio of the volume of fluid to that of the volume of the cell. The tracking of interfaces between the phases is accomplished by the solution of the volume fraction continuity equation, which is defined as:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) = s_\alpha$$

where

$\frac{\partial \alpha}{\partial t}$ represents time rate of change of the volume fraction
$\nabla \cdot (\alpha \mathbf{u})$ represents divergence of the convective flux of the volume fraction

Mass transfer between the phases can also be included, but in this study, the mass transfer between the phases is taken to be 0 i.e. $s_\alpha = 0$.

For the solver, we use interFoam solver. interFoam is a multiphase, transient, incompressible, isothermal solver used for multiphase simulations involving 2 immiscible liquids. The interFoam solver can be used under both laminar and turbulent conditions; this report however examines the behaviour of propellant under laminar conditions.

(III) Meshing and Grid

The VOF approach cannot accurately determine the details of the interface smaller than the mesh size, therefore it is important to make sure that the grid is sufficiently fine for the results to be considerably accurate. In this report, the comparison is made between blockMesh, snappyHexMesh and AMR at the interface for a predefined grid count for the forces.

Table 1: Effect of Mesh Refinement on Forces

Mesh size	Coarse	Medium	Refined
Cell Count	**40000**	**135000**	**320000**
Force at 1 sec	8.95211	7.554737	5.66819
Force at 2 sec	0.1504618	2.645789	0.807149
Force at 3 sec	0.2517157	2.564096	3.421348
Force at 4 sec	1.207224	5.894231	4.449798
Force at 5 sec	0.3730329	2.831901	1.238271
Force at 7 sec	0.6313422	3.884947	3.358638
Force at 10 sec	0.5541361	2.8055	2.956913

Plotting these values we get

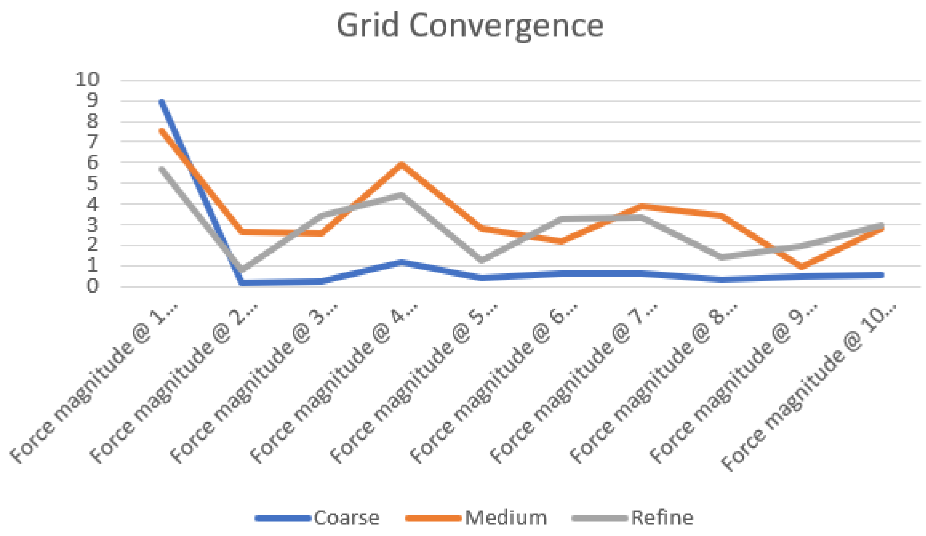

Figure 2: Effect of Mesh Refinement

Table 2: Study of Effect of Type of Mesh on

Type of Mesh	**blockMesh**	**snappyHexMesh**	**Adaptive**
Cell Count	**320000**	**320000**	**320000**
Force at 10 sec	0.7548714	0.7548714	2.956913
Force at 20 sec	1.038544	1.038544	0.9085521
Force at 30 sec	0.07443103	0.07443103	0.2464818
Force at 40 sec	0.0636293	0.0636293	0.04512248
Force at 50 sec	0.1363862	0.1363862	0.06993932
Force at 60 sec	0.06854738	0.06854738	0.05009001
Force at 75 sec	0.002217166	0.002217166	0.01307762

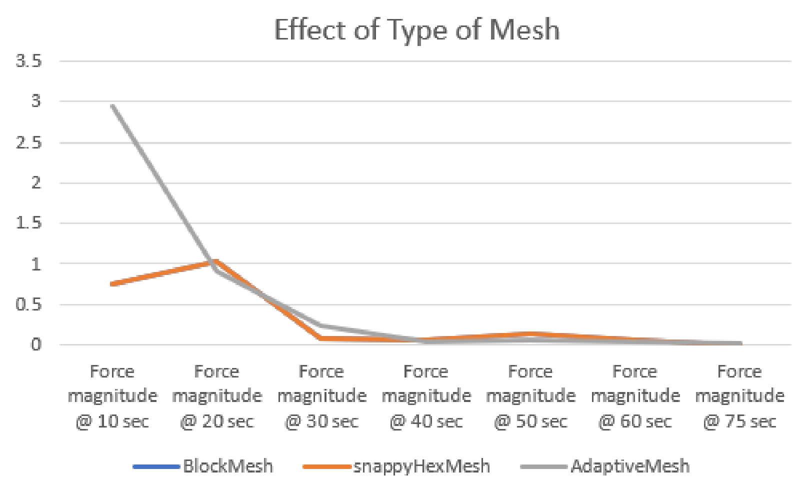

Figure 3: Effect of Mesh Refinement

(IV) InterFoam- nOrthogonal and nCorrectors

The interFoam solver solves the Navier Stokes equations for two incompressible, isothermal immiscible fluids. That means that the material properties are constant in the region filled by one of the two fluids except at the interface. The solver uses a PISO loop to solve the equations. PISO is a pressure-velocity calculation procedure for the Navier-Stokes equations, which is an extension of the SIMPLE algorithm. It involves first solving the momentum equation to estimate intermediate velocity and then solving Poisson's equation to arrive at the pressure correction factor. Next, the pressure field and velocity fields are corrected and subsequently, the second poisson equation is solved to obtain the second pressure correction factor. The above steps are repeated with the updated values for pressure and velocity until the solution is obtained. The solution is said to be obtained if the values of correction factors of both the velocity and the pressure tend to 0.

Here, we examine the results obtained by varying the number of times the corrector loops are run.

Table 3: NCorrector and NOrthogonal correctors

NCorrector and NOrthogonal	**Force Magnitude**	**Cell Count**
1 and 1 - Adaptive	0.7498991	40000
1 and 2 - Adaptive	0.6853428	40000
2 and 1 - Adaptive	0.3730329	40000
2 and 2 - Adaptive	0.2299449	40000
3 and 1 - Adaptive	0.3751212	40000
3 and 2 - Adaptive	0.3020203	40000

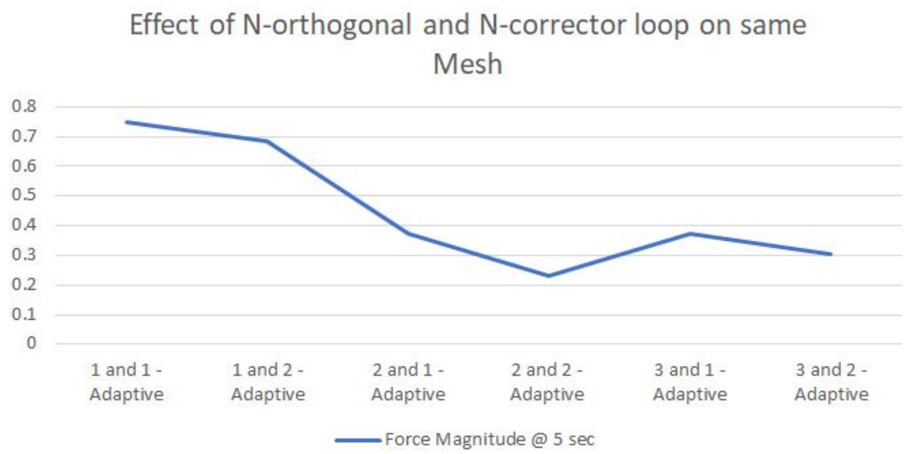

Figure 4: Plot of nOrthogonal and nCorrector values

It is essential to add only the required amount of nOrthogonal corrector loops. For this case and meshing strategy, nOrthogonal loops should be set at 1, since the value of max. orthogonality is around 73-83.

(V) Results

The results of the analysed case are discussed here. The two important parameters of interest here are the centre of mass and the forces on the tank wall of the bare cylinder. The values of these 2 parameters are plotted over 75 seconds as highlighted below.

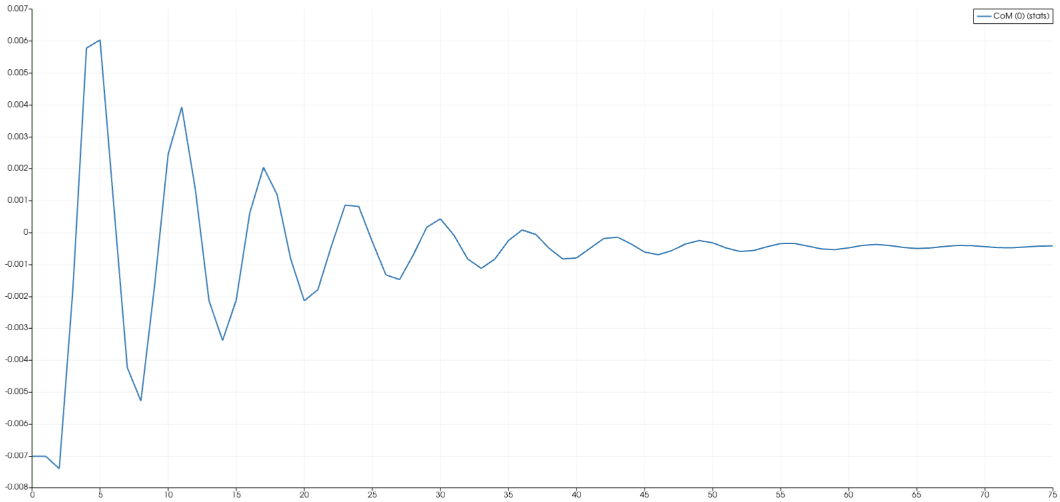

Figure 5: Plot of Center of mass over time

The above plot shows the evolution of the location of the centre of mass over time. The movement of the centre of mass is initially at 6mm in the beginning at around 0.5 seconds and then proceeds to dampen and then eventually almost flat lines around the 75-second mark.

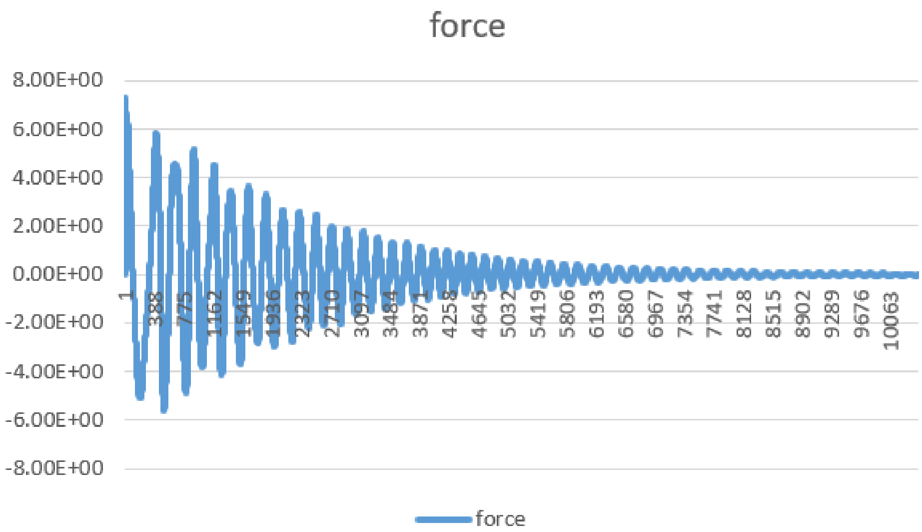

Figure 6: Plot of forces on tank wall over time

The plot above depicts the evolution of forces on the tank wall over the course of 75 seconds. It is the maximum of about 7N in the first crest and dampens progressively until it almost flatlines around the 75-second mark. It is interesting to note that both the movement of the centre of mass and the forces are sine waves that experience damping over time. Also, BlockMesh and snappyHexMesh behave almost identically to each other, even though the meshing strategy is different. It is also worthwhile mentioning that these plots are plotted for a finer grid size.

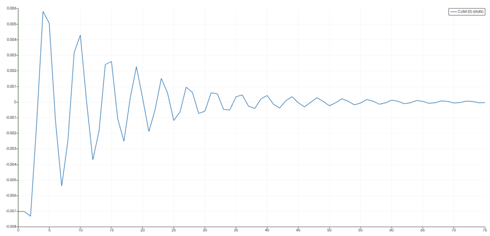

Figure 7: CoM - blockMesh

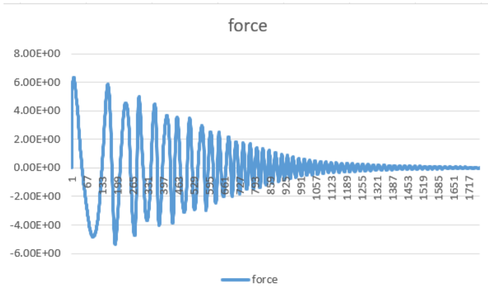

Figure 8: Forces - blockMesh

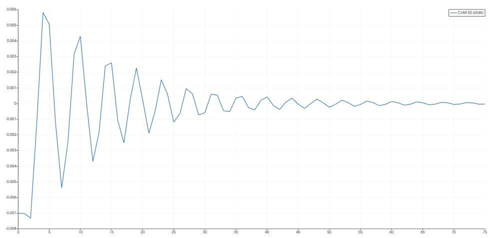

Figure 9: CoM - snappyHexMesh

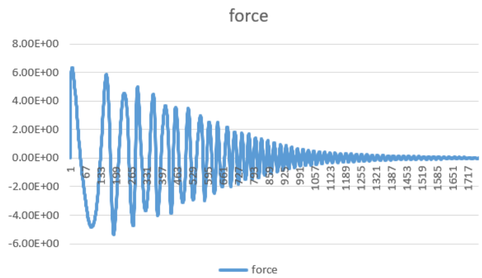

Figure 10: Forces - snappyHexMesh

The frequency of sloshing can be calculated by considering an equivalent mechanical pendulum, the formulae for which are furnished below:

$$L = \frac{R}{1.841tanh(\frac{1.841h}{R})}$$

$$f = \frac{1}{2\pi}\sqrt{\frac{g}{L}}$$

where,
L is the length of the equivalent pendulum,
R is the radius of the cylinder,
h is the fill height,
g is the settling acceleration,

In this case, the values of R=0.6, h = 0.7. Therefore, L is evaluated as

$$L = \frac{0.6}{1.841tanh(\frac{1.841*0.7}{0.6})} = 0.33491$$

and the frequency is calculated as

$$f = \frac{1}{2\pi}\sqrt{\frac{9.81}{0.33491}} = 0.861$$