Numerical Study of the Liquid Oxygen Draining in a Spinning Tank with Baffles for Hybrid-Propulsion Rocket

Anant Singhal*, Deepak K. Agarwal, T. John Tharakan, S. Sunil Kumar

Propulsion Research and Studies
Liquid Propulsion Systems Centre
Thiruvananthapuram, Kerala, 695547
mail2anantsinghal@gmail.com

Abstract

Computational Fluid Dynamics (CFD) study of liquid oxygen draining in a spinning tank with and without baffles is carried out for hybrid propulsion rocket. The effect of tank spinning and the presence of baffles on the vortex formation and unusable propellant mass is brought out in this work. The interface of the draining propellant is captured using the Volume of Fluid (VOF) framework while the motion of baffles is simulated through mesh motion technique. Initially, analysis of liquid oxygen draining is carried out in a stationary tank without baffles. It is found that that a vortex is formed in the tank and gas entry in the feed line happens at 21s resulting in unusable propellant of 1.9kg. Subsequent analysis of liquid draining in a spinning tank without baffles shows that gas enters into the feed line earlier as compared to the stationary tank. Finally, the analysis is carried out for propellant draining in spinning tank with baffles. It is found that the momentum imparted by the baffles to the draining fluid increases the unusable propellant mass further compared to tank without baffle.

Keywords: *draining, tank, spinning, baffles*

1. Introduction

A hybrid propulsion rocket is being developed by ISRO that has solid propellant as fuel and liquid oxygen as the oxidizer. This hybrid propulsion rocket is envisaged to be aerodynamically spin stabilized. In one of the liquid oxygen storage tank configurations, baffles are placed on the tank inner walls along its length. The baffles are placed to reduce the propellant sloshing during flight. However, the spinning of the tank and the presence of baffles may cause vortex formation during propellant draining that can lead to a decrease in the usable propellant mass. In the present study, three-dimensional transient numerical simulations of liquid oxygen tank draining during flight is carried out to study the effect of tank spinning and baffles on the un-usable propellant mass towards the end of the flight.

2. Literature Review

Various experimental studies have been reported on the formation of vortex during draining of liquid in tanks [1-3]. It has been reported that a dip forms on the free surface of the draining liquid which intensifies into a gas-vortex. The gas enters the drain port at a critical height. In the context of rocket engines, this gas entry from the tank drain into the pump may lead to critical failure. Therefore, it is important to study and estimate the formation of gas vortex during draining of the liquid propellant. Various researchers have successfully proven the usability of VOF method to capture the formation of vortex during liquid draining [4-6].

Researchers have also reported the flow field in a stirred tank both with and without baffles on the tank wall. [7, 8]. A detailed experimental study of liquid motion in spinning, nutating tanks was reported in [9] to determine the effect of such liquid motion on the stability of the spacecraft. However, to the best knowledge of the authors, study on the flow

*Corresponding Author: Email: mail2anantsinghal@gmail.com

field during propellant draining in a spinning tank with and without baffles for launch vehicle application is not available in open literature. Therefore, in the present paper, a numerical investigation is carried out to bring out the effect of tank spin motion and baffles on the vortex formation and un-usable propellant in the tank. The following 3D transient CFD analyses were carried out for the draining of liquid oxygen from the tank.

Case1: Analysis of propellant draining in a stationary tank without baffles.

Case 2: Analysis of propellant draining in a spinning tank without baffles.

Case 3: Analysis of propellant draining in a spinning tank with baffles.

3. Computational Domain and Boundary Conditions

The 3D computational domain of the liquid oxygen tank is shown in Figure 1. The domain consists of the tank with baffles both in the cylindrical part and in the bottom dome of the tank. The cylindrical part of the tank is 1.76m in height and 0.3m in diameter. A set of four baffles is mounted longitudinally each on the tank cylindrical wall and in the bottom hemispherical portion of the tank. The domain for the cases 1 and 2 is similar as shown in Figure 2 except that the baffles are removed from the tank. The domain is meshed with polyhedral elements having a total mesh count of 27lakhs. The maximum skewness of the mesh is 0.85 while the minimum orthogonal quality is 0.16.

A pressure inlet boundary condition was applied at the vent in the top dome in order to maintain the ullage at a constant pressure of 35bar during liquid oxygen draining. A constant liquid oxygen mass flow rate of 6kg/s is specified at the drain port outlet. The tank was simulated to be spinning at a rate of 6rps. The vehicle acceleration of 50m/s^2 was considered as the body force acting on the propellant. The initial propellant loading of 128kg and resulting ullage volume of 6 litres was considered for the simulation. Each transient simulation was carried out for a duration of 22s.

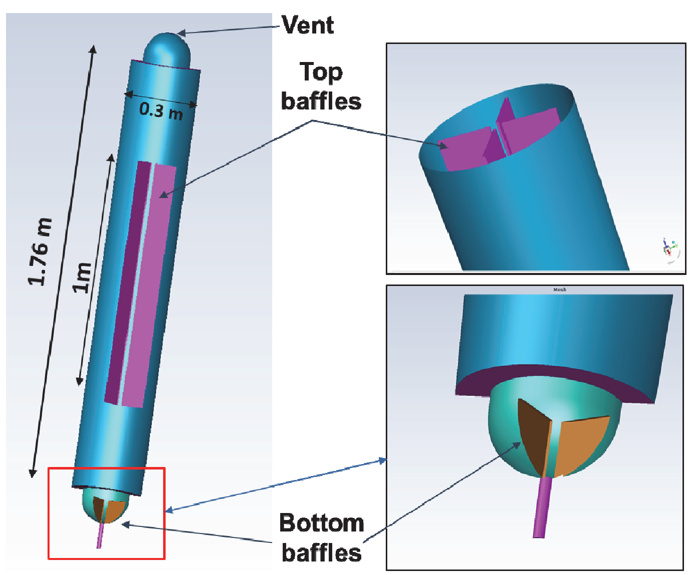

Figure 1: Computational domain of the LOX tank

ISBN - 978-93-95038-50-8

4. Numerical Methodology

The numerical simulations were carried out using ANSYS Fluent. The spinning of the tank having baffles is simulated using the mesh motion technique. The interface of the draining liquid oxygen is captured using Volume-of-Fluid (VOF) methodology with SST k-ω turbulence model. The pressure and velocity coupling is achieved using the Pressure-Implicit with Splitting of Operators (PISO) scheme. The discretization of the governing equations was carried out using second order upwind scheme in space. First order implicit scheme was used for time marching with a time step of $1\text{x}10^{-4}$s. Non-iterative time advancement was invoked for improved stability of the solution at each time step.

The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the qth phase in absence of mass transfer, this equation has the following form:

$$\frac{\partial}{\partial t}\left(\rho_q \alpha_q\right) + \nabla \cdot \left(\rho_q \alpha_q \vec{v}\right) = 0 \tag{1}$$

Volume fraction equation will not be solved for the primary phase; since it is computed based on the constraint,

$$\sum_{q=1}^{n} \alpha_q = 1 \tag{2}$$

Volume fraction equation is solved using the implicit formulation in this study. In the VOF model, a single momentum equation is solved throughout the domain.

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla \vec{v} + \nabla \vec{v}^T\right)\right] + \rho \vec{g} + \overrightarrow{F_t} \tag{3}$$

where ρ, μ are volume-weighted averaged properties. F_t represents the surface tension force at the interfaces.

Continuum surface force (CSF) model which accounts for surface tension between a phase pair as a continuous, volumetric body force term is applied in this study. It has the following form:

$$F_t = \sigma_{ij} \frac{\alpha_i \rho_i \kappa_j \nabla \alpha_j + \alpha_j \rho_j \kappa_i \nabla \alpha_i}{\frac{1}{2}\left(\rho_i + \rho_j\right)} \tag{4}$$

where curvature $\kappa = -\nabla \cdot \hat{n}$, interface unit normal $\hat{n} = \nabla\alpha / |\nabla\alpha|$

5. Results and Discussion

Validation Study

Initially, as a model validation, steady state axisymmetric analysis of liquid in a spinning tank was carried out. The obtained shape and rise of the liquid free surface was compared with the analytical result. A cylindrical tank having 200mm diameter and 600mm height was taken for the validation study. The tank was considered to be rotating at a speed of 12rad/s. The contour of liquid volume fraction obtained from CFD analysis is shown in Figure 2. It can be seen that CFD estimates the distance between the minimum and maximum height of the liquid free surface to be 75mm. The expression given by Eq. (5) was used to analytically estimate the height of the liquid free surface. The height of the free-surface is estimated to be 73.39mm using Eq. (5). Thus, error in the CFD model is within 2.2%.

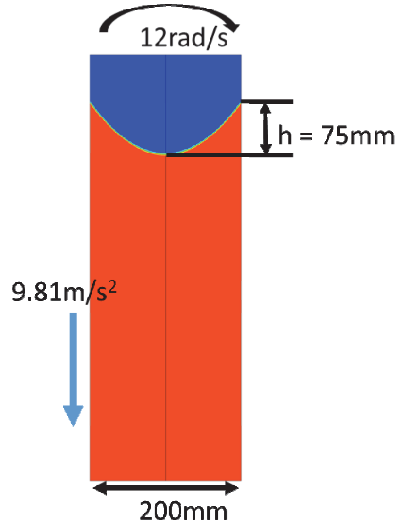

Figure 2: Shape of liquid free-surface in a spinning tank for model validation

$$h = \frac{R^2.\omega^2}{2g} \tag{5}$$

where R is the tank radius, ω is the angular velocity and g is the gravitational acceleration.

CASE 1: ANALYSIS OF PROPELLANT DRAINING IN A STATIONARY TANK WITHOUT BAFFLES

Initially, analysis of liquid oxygen draining in a stationary tank without baffles was carried out to estimate the propellant left-out in the tank at the instant when vapor enters the drain line. Figure 3 shows the liquid volume fraction in the tank at the start of draining (t=0s), 1s, 10s and the time instant at which the ullage gas enters the drain port (21s).

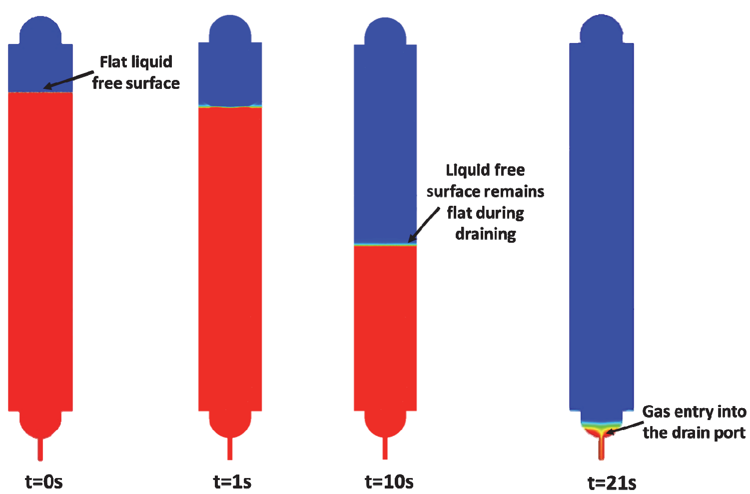

Figure 3: Contours of liquid (red color) volume fraction at various time instants for Case 1

ISBN - 978-93-95038-50-8

It can be seen that when the tank is not spinning, the free surface of the liquid that was flat initially remains to be flat throughout the draining. A dip is formed at the free surface of the propellant towards the end of the draining and the ullage gas enters the drain port at 21s. The mass of liquid oxygen remaining in the tank at this instant is 1.9kg and is termed as unusable propellant.

CASE 2: ANALYSIS OF PROPELLANT DRAINING IN A SPINNING TANK WITHOUT BAFFLES

Subsequent to case 1, analysis was carried out for a spinning tank without baffles to determine the effect of spin motion on the flow behavior and un-usable propellant. The initial condition for this case was the same as that for case 1. Figure 4 shows the contours of liquid volume fraction at t=5s, 10s and 20.28s (the time at which ullage gas enters the drain port).

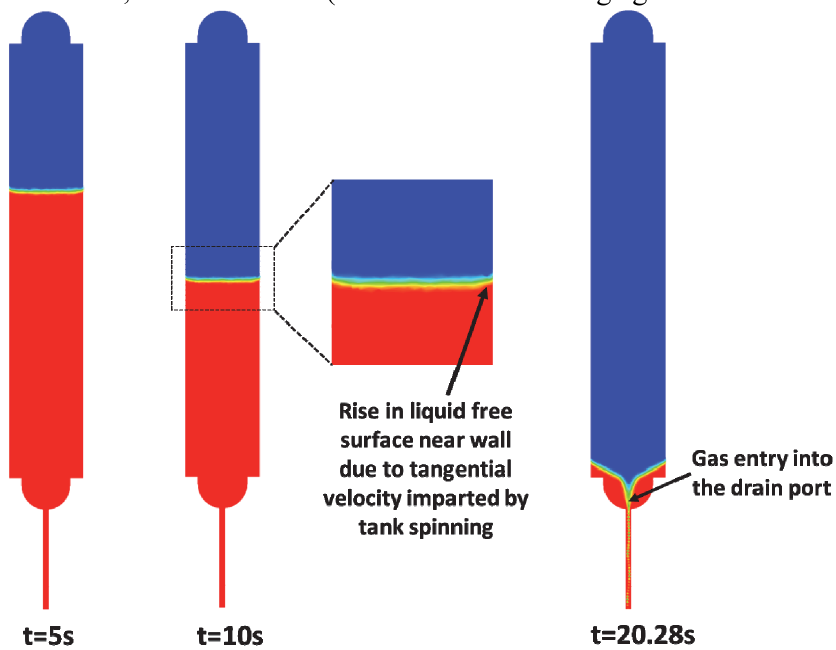

Figure 4: Contours of liquid (red color) volume fraction at various time instants for Case 2

It can be seen from Figure 4 that the free surface of the liquid assumes a curved shaped due to centrifugal force acting on the liquid resulting from spinning of the tank. The ullage gas enters the tank drain port at 20.28s and the un-usable mass of liquid oxygen in the tank is 6.3kg at this instant. The spinning motion of the tank imparts angular momentum in the draining liquid, thus increasing the vorticity of the liquid. The increased vorticity results in earlier dip formation in the liquid free surface which results in a higher left-out mass in case 2.

CASE 3: ANALYSIS OF PROPELLANT DRAINING IN A SPINNING TANK WITH BAFFLES

Figure 5 shows the contours of liquid volume fraction at various time instants for propellant draining in a spinning tank having baffles. The baffles that are fastened to the tank walls also spin along with the tank. The spinning baffles impart additional angular momentum to the liquid that results in a dip formation at the free-surface of the liquid during early stage of the draining at t=1s. The dip formation in case 3 results in early gas entry into the drain port compared to that in cases 1 and 2. The gas enters into the drain port at 20.02s in case 3 and the left-out propellant in the tank, at this time instant, is found to be 7.9kg.

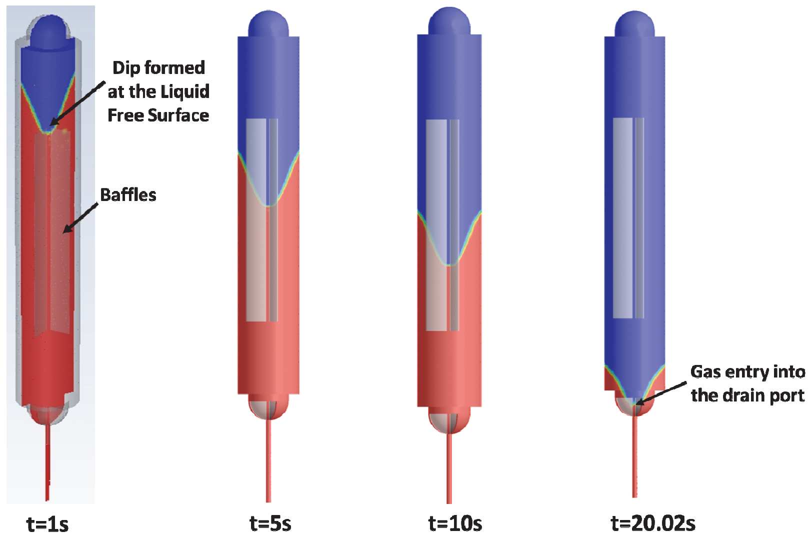

Figure 5: Contours of liquid (red color) volume fraction at various time instants for Case 3

6. CONCLUSIONS

Two-phase CFD simulations of liquid oxygen draining in the tank of a spin stabilized hybrid propulsion rocket is reported in the current paper. The simulations are carried out for spinning tank with and without baffles on its inner wall to determine the effect of spin motion and baffles on the un-usable propellant in the tank. It is found that the spinning of the tank increases the un-usable propellant mass from 1.9kg for a stationary tank to 6.3kg for a spinning tank without baffles. Further, adding anti-slosh baffles to the tank of a spin stabilized rocket increases the left out propellant to 7.9kg, thus increasing the mass penalty.

REFERENCES

1. Ramamurthi, K. & Tharakan, T.J, "Intensification of a vortex during draining", Can. J. Chem. Eng. 73 (1995) 292–299.
2. Ramamurthi, K. & Tharakan, T.J, "Flow visualisation experiments on free draining of a rotating column of liquid using nets and tufts", Exp. Fluids 21 (1996) 139–142.
3. Lubin, B.T. & Springer, G.S, "The formation of a dip on the surface of a liquid draining from a tank", J. Fluid Mech. 29 (1967) 385–390.
4. Robinson, A, Morvan, H, Eastwick, C, "Computational investigations into draining in an axisymmetric vessel", J. Fluids Eng. 132 (2010).
5. Zhou, Q.N. & Graebel, W.P, "Axisymmetric draining of a cylindrical tank with a free surface", J. Fluid Mech. 221 (1990) 511–532.
6. Agarwal, D, Basu, P, Tharakan, T.J, Salih, A, "Prediction of gas-core vortices during draining of liquid propellants from tanks", Aerospace Science and Technology 32 (2014) 60–65.
7. Rajavathsavai, D, Khapre, A, Munshi, B, "Numerical Study of Vortex Formation inside a Stirred Tank", International Journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering Vol:8, No:12, 2014.
8. Tamburini A, Gagliano G, Scargiali F, Micale G, Brucato A, Ciofalo M, "CFD Simulation of Radially Stirred Baffled and Unbaffled Stirred Tanks", Chemical Engineering Transactions, 2019, 74, 1033-1038 DOI:10.3303/CET1974173.
9. David J. Chato, D.J, Dalton, P.J, Dodge, F. T, Green, S. Liquid Motion Experiment Flight Test Results. NASA, NASA/TM—1998-208479. July 1998.