Finite volume and smoothed particle hydrodynamic simulation of rocket fuel tank sloshing with baffles

Beg, OA, Valter, M, Beg, TA, EL Gendy, M, Kadir, A and Jouri, W

<table>
<thead>
<tr>
<th>Title</th>
<th>Finite volume and smoothed particle hydrodynamic simulation of rocket fuel tank sloshing with baffles</th>
</tr>
</thead>
<tbody>
<tr>
<td>Authors</td>
<td>Beg, OA, Valter, M, Beg, TA, EL Gendy, M, Kadir, A and Jouri, W</td>
</tr>
<tr>
<td>Type</td>
<td>Conference or Workshop Item</td>
</tr>
<tr>
<td>URL</td>
<td>This version is available at: <a href="http://usir.salford.ac.uk/id/eprint/51359/">http://usir.salford.ac.uk/id/eprint/51359/</a></td>
</tr>
<tr>
<td>Published Date</td>
<td>2019</td>
</tr>
</tbody>
</table>

USIR is a digital collection of the research output of the University of Salford. Where copyright permits, full text material held in the repository is made freely available online and can be read, downloaded and copied for non-commercial private study or research purposes. Please check the manuscript for any further copyright restrictions.

For more information, including our policy and submission procedure, please contact the Repository Team at: usir@salford.ac.uk.
1. INTRODUCTION

The continuing efforts to reach deeper destinations in the cosmos has sustained interest in spacecraft propulsion. Presently chemical propulsion remains the only feasible mechanism for space travel. Systems therefore require fuel tanks to store earth’s gravity and to achieve fast entry into space. Fuel tanks may contain oxygen, hydrogen or other fuels. These are generally in a liquid state and therefore may experience sloshing [1]. Energy dissipation within the fuel tank can have a drastic increase on the spacecraft operation, also known as wobble, which can result in catastrophes. The gyroscopic-like nature of the oscillation spins means that the spacecraft changes the orientation of the rotational axis by a rotation, and for an ideal flight it should have a relatively small and constant value. However, as sloshing takes place within the tank, it will increase the rotation angle and if not controlled may lead to incomplete depletion of the propellant. Sloshing, in fluid dynamics [2], can be defined as the deformation of liquid inside another object, which is also typically experiencing motion. For this phenomena to take place the liquid must have a free surface to alter the system’s dynamic characteristics. Baffles have the characteristics of being able to minimize the kinetic energy of the fluid and thus reduce the increase in pressure on the tank walls. There are designs differ with different fuels. The main focus is on liquid oxygen however due to its popularity in rocket propulsion and ANSYS Fluent finite volume CFD simulations [3] are conducted and compared with water. Validation with an experimental particle hydrodynamics solver [4] is used to demonstrate excellent agreement. The analysis shows that sloshing can be mitigated with judicious selection of baffles which damp the oscillations and are relatively easy to install.

2. MATHEMATICAL MODEL AND GEOMETRIC MESH

A typical LOX fuel tank is shown below. The simulated geometry is shown below right and is a two dimensional geometric model (1 baffles). ANSYS Fluent uses the finite-volume method to solve the governing Navier-Stokes equations for a fluid which are derived from the conservation mass equation, the conservation of momentum (2) where h are the usual notation (1)-(4).

3. ANSYS CFD SIMULATIONS

CFD Simulations consist of snapshots of the different mass distributions in the tank [propellant volume fraction] at five different timesteps (0 - 10 seconds, 100 - 15 seconds, 200 - 30 seconds, 300 - 45 seconds and finally 600 - 90 seconds) and will be presented below. The main reason for which the fluid volume fraction of the propellant is presented, besides the visual aid, is that the mass redistribution, which is associated with sloshing, has a direct effect on the moment of inertia of the fuel tank. The moment of inertia is defined using the following units: [17 kg m²/ s²], so the direct correlation between sloshing and moment of inertia is clear. Ultimately, significant fluctuations of the moment of inertia in the fuel tank can severely affect the stability of the spacecraft.

4. SPH VALIDATION

SPH is a “smoothed particle hydrodynamics” method originally introduced for astrophysical fluid dynamics in the 1970s by American researchers, which avoids the re-analyzing, numerical discretization and free surface modeling issues inherent to finite volume methods such as ANSYS Fluent. It uses a Laplacian formulation and discretizes the flow domain by a collocation technique using an interpolant or smoothing kernel φ(r), which is a position and h the smoothing length. SPH is ideal for free surface or sloshing dynamics there is no mesh to generate or move and free surfaces and material break up as they are naturally accommodated in the code. It has been extensively used in complex fluid dynamics problems [5, 6]. The interpolating points may be thought of as particles each carrying a mass, and a velocity, u. The SPH kernel employed is spherically based and consists for separations > h, φ(r) has a change of smoothing of density at the free surface in the continuity equation whilst the momentum equation is orchestrated to conserve linear and angular momentum and is the viscous term. The equation of state used in this simulation is also given where p = ρ for the liquid oxygen fuel. Po is defined to restrain the maximum fluid compression to less than 5%. Excellent agreement is obtained between FLUENT and SPH.

5. DISCUSSION/CONCLUSIONS

The ANSYS Fluent simulations demonstrated that there is a close correlation between the velocity of the propellant and the absolute pressure within the tank, which is the expected result when comparing to the actual fuel tank sloshing tests. This high degree of match has been achieved with the SPH algorithm. Validation with an experimental particle hydrodynamics solver [4] has been extensively used for water and fuel, and thus the tank with no baffles is as shown in Figure 1. The results are compared with literature and the code is seen to produce a good match. However, when compared to the tank with no baffles there are still significant variations in the absolute velocity. Nevertheless, these fluctuations are rather small, within the percent ranges, in the presented cases. For a practical example, during the last 1547 seconds of the simulation, the fluid is slowly decreasing over time. This suggests that, on the present day, the fluid velocity can also be considered in the relative movement problem of the tank with no baffles. The table below shows the results from the SPH simulations for the fuel tank with no baffles and with baffles. The table is used to compare the tank with and without baffles. It shows that the fuel tank with baffles has a higher pressure than the tank with no baffles. The pressure is the major concern in this application. The results are compared with literature and the code is able to produce a good match. In summary, the tank with baffles is more efficient than the tank with no baffles. The baffles absorb the energy and reduce the velocity of the fluid. The results are presented in the following sections. The final section contains a discussion of the results and future work.

REFERENCES


**REFERENCES**


