THE IIER - 767TH INTERNATIONAL CONFERENCE ON RECENT INNOVATIONS IN ENGINEERING AND TECHNOLOGY ICRIET, DOHA, QATAR, DECEMBER 10-11, 2019.

FINITE VOLUME AND SMOOTHED PARTICLE HYDRODYNAMIC SIMULATION OF ROCKET FUEL TANK SLOSHING WITH BAFFLES

O. Anwar Bég[1], Matei Valter[2], Tasveer A. Bég[3], M.E. El Gendy[4], Ali Kadir[1] and W.S.Jouri[1]

[1]Department of Mechanical/Aeronautical Engineering, Salford University, Manchester, M54WT, UK.

[2]Numerical Simulation Engineer, Renault, Şos, de centură nr. 17 Chiajna, Bucharest, Romania.

[3] Renewable Energy and Computational Multi-Physics, Israfil House, Dickenson Rd., Manchester, M13, UK.

4. Computational Mechanics Consultant, Al Rehab, Cairo, Egypt.

Emails: O.A.Beg@salford.ac.uk (Presenter); m.valter@renault.org; tasveerabeg@gmail.com; mgendy93@gmail.com; A.Kadir@salford.ac.uk; w.s.jouri@salford.ac.uk

1.INTRODUCTION

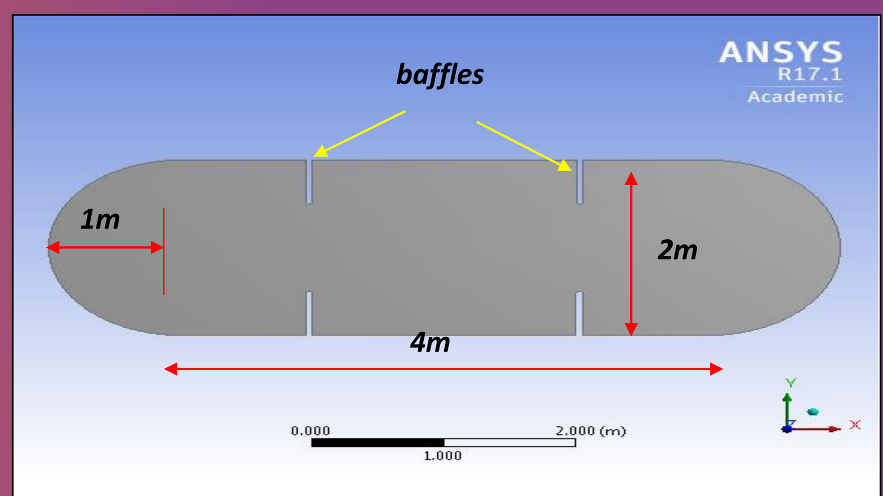
- The continuing efforts to reach deeper destinations in the cosmos has sustained interest in spacecraft propulsion. Presently chemical propulsion remains the only feasible mechanism for space travel. Systems therefore require fuel tanks to escape earth's gravity and to achieve fast entry into space. Fuel tanks may contain oxygen, hydrogen or other fuels. These are generally in a liquefied state and therefore may experience sloshing [1]. Energy dissipation within the fuel tank can have a drastic increase on the spacecraft nutation, also known as wobble, which can result in catastrophe. The gyroscope-like nature of the stabilisation spin means that the spacecraft changes the orientation of the rotational axis by a nutation angle 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 nutation angle and if not controlled it may lead to incomplete depletion of the propellant. Sloshing, in fluid dynamics [2], can be defined as the displacement of liquid inside another object, which is also typically experiencing motion. For this phenomenon to take place the liquid must have a free surface to alter the systems dynamic characteristics. Baffles have the characteristic of being able to minimise the kinetic energy of the fluid and thus reduce the uneven increase in pressure on the tank walls. Several baffle designs are considered 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 [3,4] are conducted and compared with water. Validation with an SPH (smoothed particle hydrodynamics) solver [5,6 is included demonstrating 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.

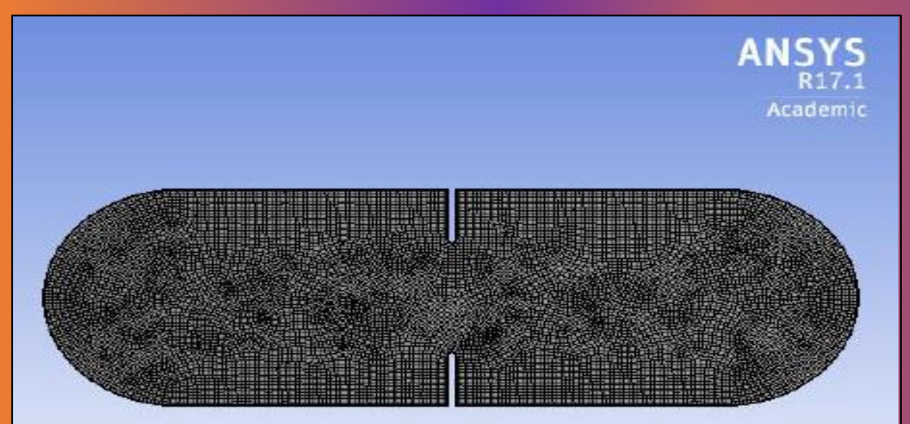
2. MATHEMATICAL MODEL AND GEOMETRIC MESH

A typical LOX fuel tank is shown below. The simulated geometry is shown below right and is a two dimensional geometric model (2 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 (1), the conservation of momentum (2) where all terms have the usual notation [3,4]:

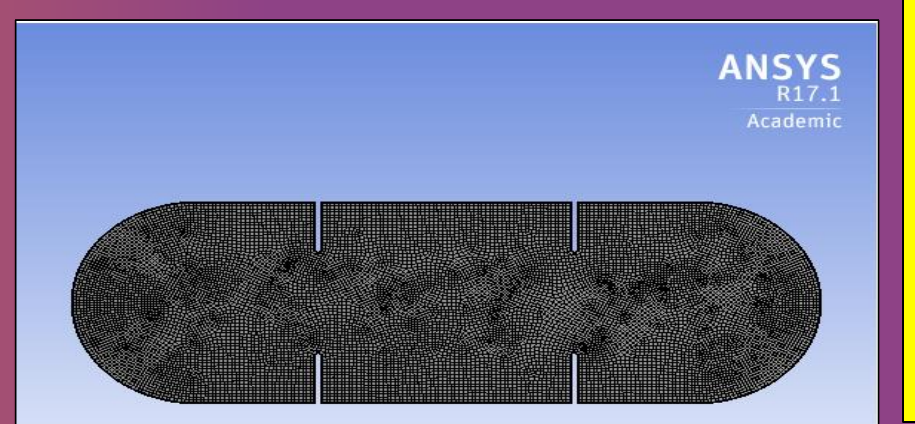
$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \qquad (1)$$

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \rho \vec{g} + \nabla \cdot \tau_{ij} \qquad (2)$$





Fuel Tank with 2 Baffles; 15874 elements



Fuel Tank with 4 Baffles; 16075 elements

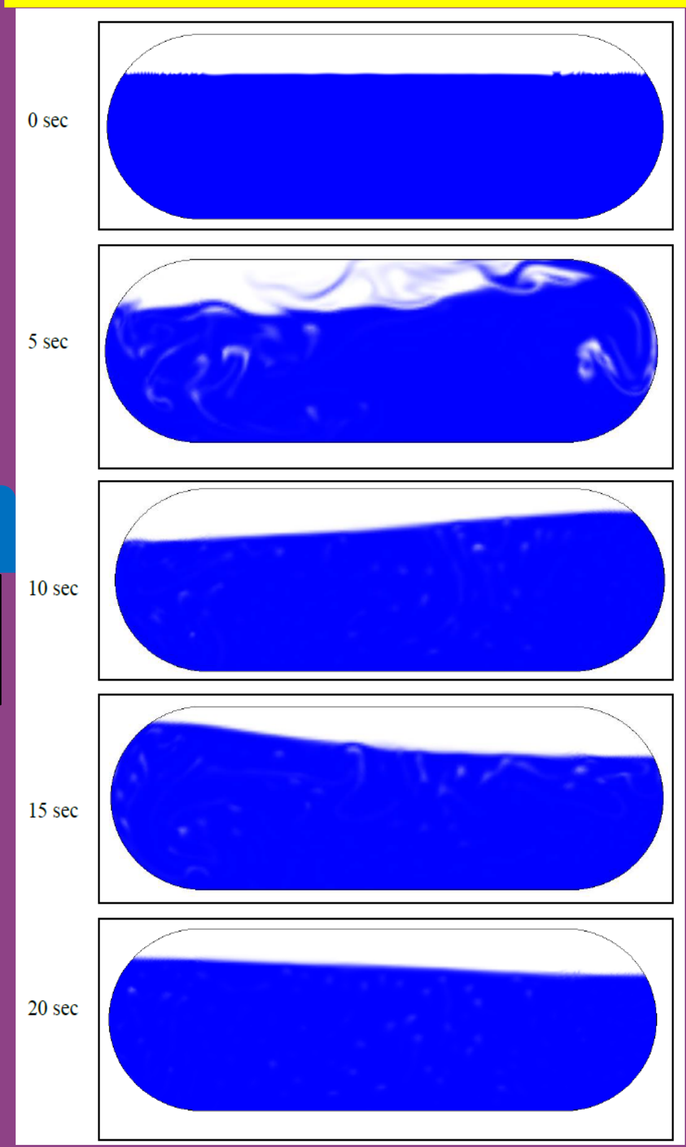
Quadrilateral elements are used. All material data and slosh specifications are shown below:

Liquid Oxygen		
Density	1141	kg/m³
surface tension	0.013	N/m
T - fluids in the tank	-182.778	deg C
T - fluids in the tank	90.372	K
Viscosity at 90 deg K	0.0002	kg/ms

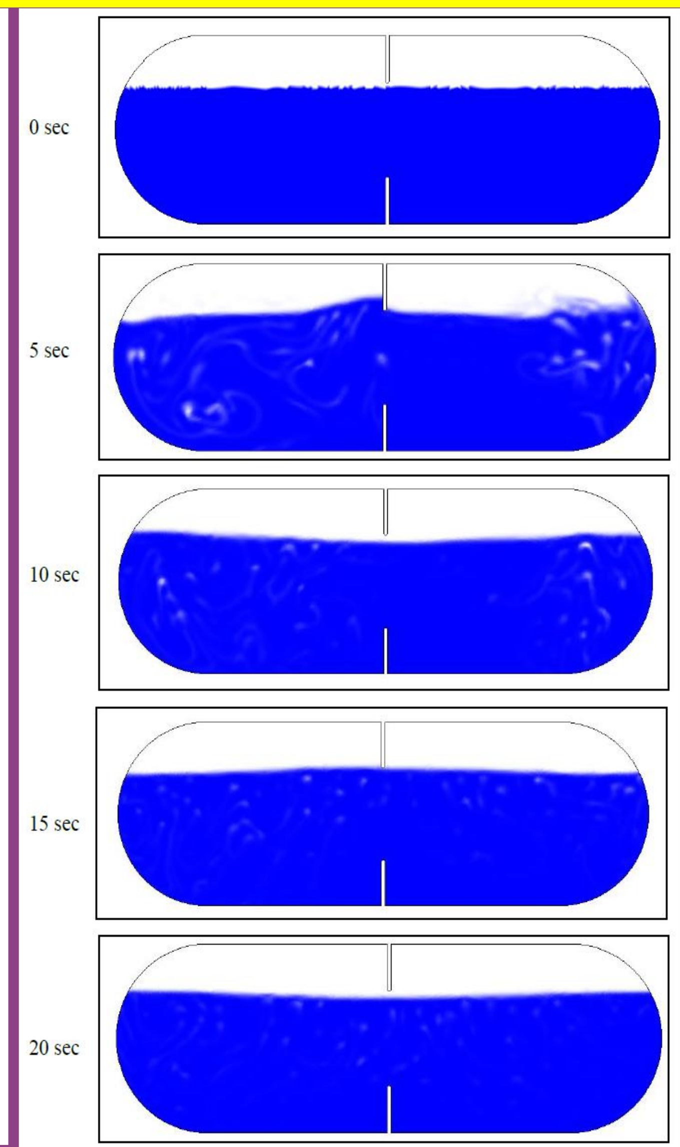
Parameter	Formula	Units
Fluid Height (*fluidHeight*)	0.6	m
Fluid Density (*fluidDensity*)	1141	kg/m³
Fluid Volume Fraction (*fluidVolF*)	if(y<fluidHt,1,0)*if(y>-1.1 [m],1,0)	
Hydro Pressure (*HydroPress*)	fluidDensity*g*(fluidHeight-y)*fluidVolF	Pa

3. ANSYS CFD SIMULATIONS

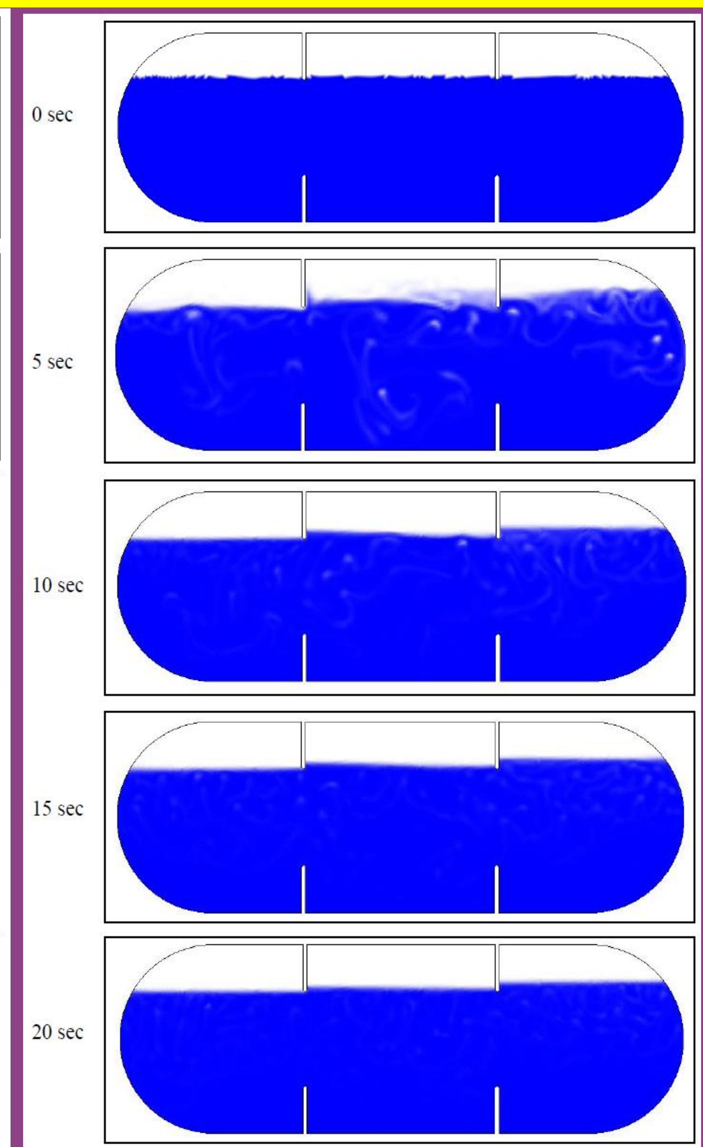
CFD Simulations consist of snapshots of the different mass distributions in the tank (propellant volume fraction) at five different timesteps (0 - 0 seconds, 100- 5 seconds, 200 - 10 seconds, 300 - 15 seconds and finally 400- 20 sec), and they will be presented first. The main reason for which the fluid 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: I = kg*m², 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.



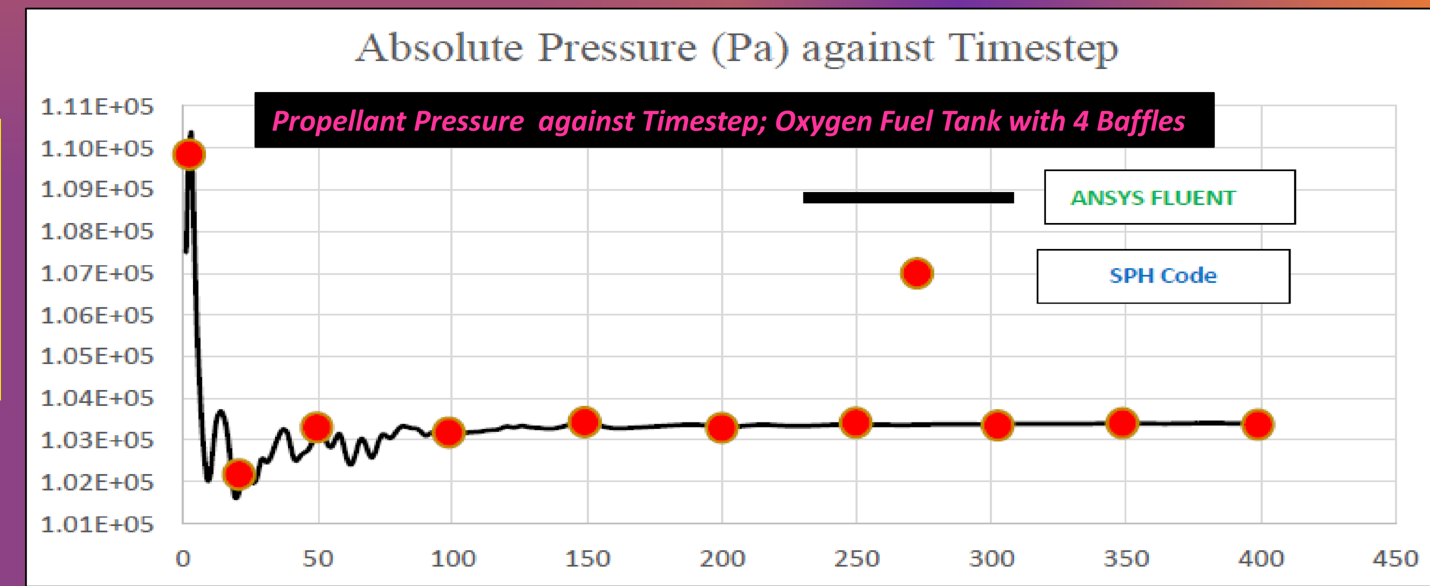
Volume fuel fraction with NO Baffles

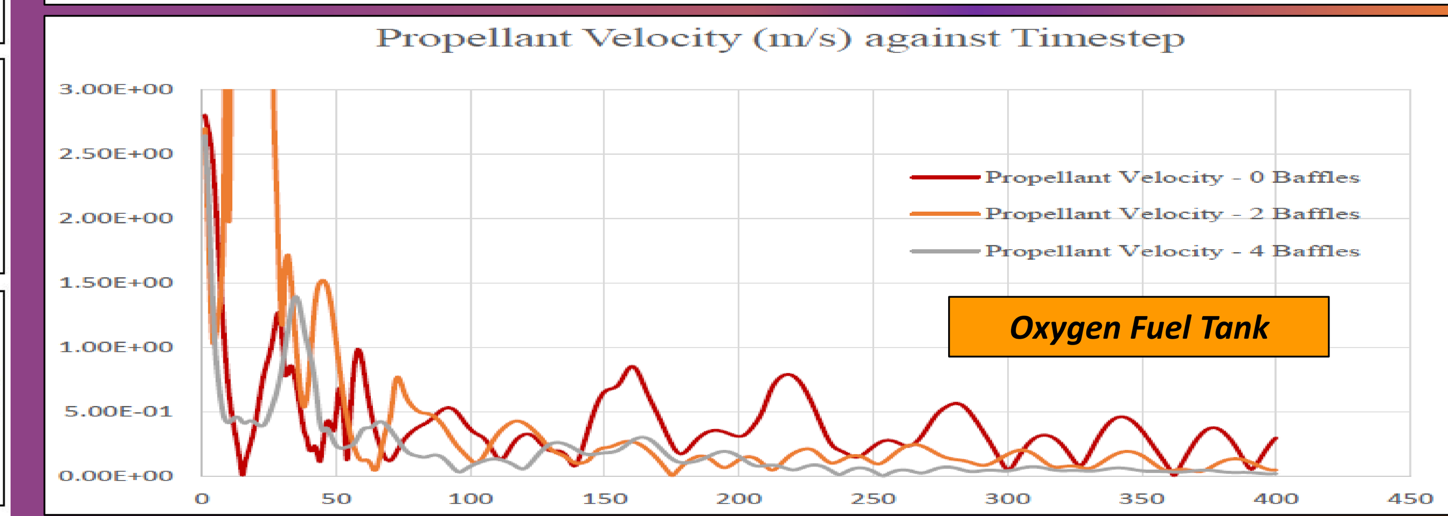


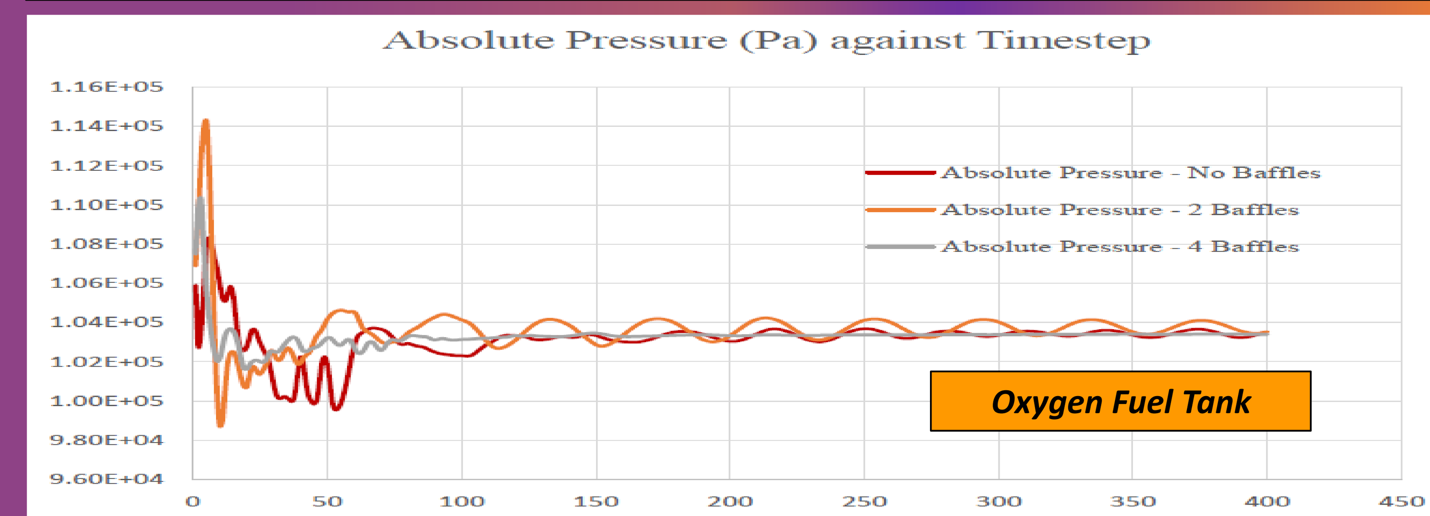
Volume fuel fraction with 2 Baffles



Volume fuel fraction with 4 Baffles







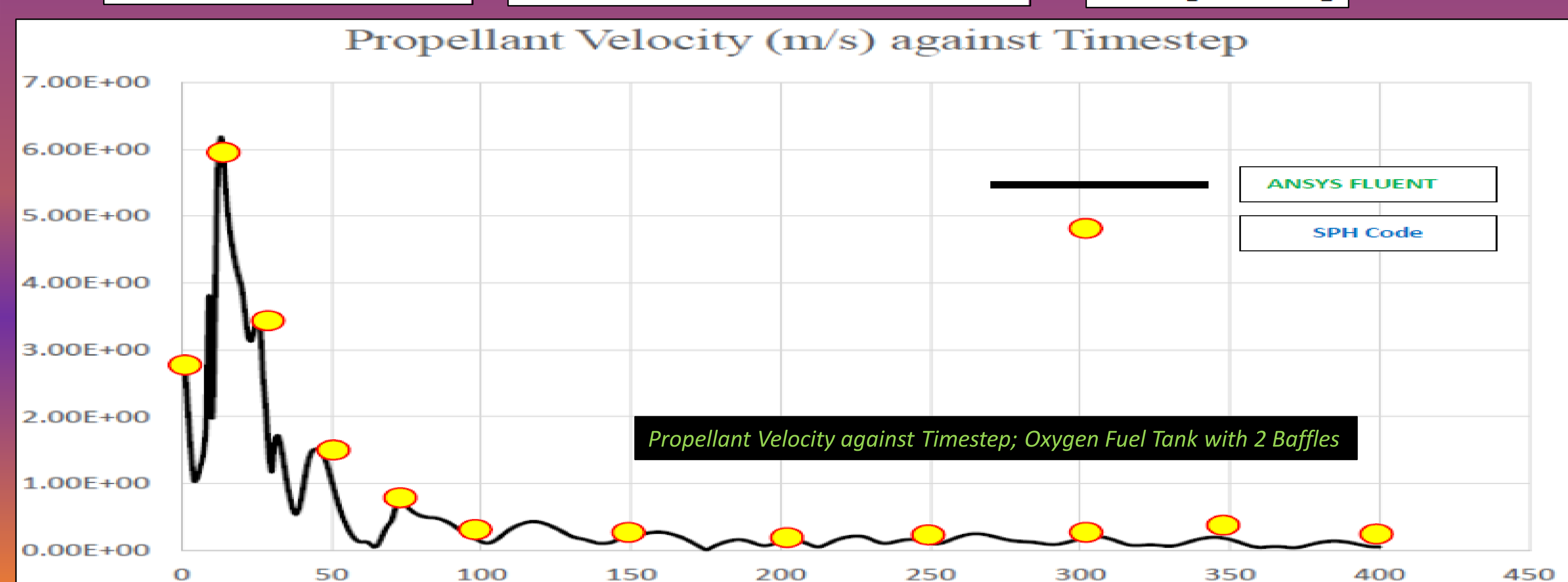
4. SPH VALIDATION

SPH is a "meshless method" originally introduced for astrophysical fluid dynamics in the 1970s BY American researchers, which avoids the re-meshing, numerical dissipation and free surface smearing issues inherent to finite volume methods such as ANSYS FLUENT. It uses a Lagrangian formulation and discretizes the flow domain by a collocation technique using an interpolant or smoothing kernel W(r,h), where r is 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 are naturally accommodated in the code. It has been used extensively in complex fluid dynamics problems [5, 6]. The interpolating points may be thought of as particles each carrying a mass m, and a velocity v. The SPH kernel employed is spline-based and vanishes for separations > 2h. This implies that summations involve only proximate "neighbours". By implementing an efficient grid based search algorithm that scales linearly with the number of particles, rapid convergence is achieved. To represent liquid oxygen fuel, the SPH particles were given a density of 1141 kg/m³ and viscosity of 2 × 10-4 kg/ms. Particles were created for all fuel tank designs. However here we only show the velocity and pressure validation for oxygen fuel. 58,346 particles were deployed. The SPH discrete equations of motion are such that there is no smoothing of density at the free surface in the continuity equation whilst the momentum equation is orchestrated in symmetrized from to conserve linear and angular momentum and *Πab* is the viscous term. The equation of state used in this simulation is also given where γ = 7 for the liquid oxygen fuel, Po is defined to restrain the maximum fluid compression to less than 1% . Excellent agreement is obtained between FLUENT and SPH.

$$\frac{d\rho_a}{dt} = \sum_b m_b (\mathbf{v}_a - \mathbf{v}_b) \cdot \nabla_a W_{ab}$$

$$\frac{d\mathbf{v}_a}{dt} = -\sum_b m_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_{ab} + \mathbf{F}_a$$

$$P = P_o \left[\left(\frac{\rho}{\rho_o} \right)^{\gamma} - 1 \right]$$



5. DISCUSSION/CONCLUSIONS

- The ANSYS FLUENT simulations demonstrate that there is a clear 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 studies of NASA etc. The larger the changes in the velocity the higher the fluctuations in absolute pressure. However, when analysing the tank with 2 baffles it can be seen that even if the oscillations in velocity are significantly decreased, when compared to the tank with no baffles there are still significant variations in the *absolute pressure* values. Nevertheless, those fluctuations are more stable, unlike the previous case, as the wavelength remains relatively constant, during the last 17.5 seconds of the simulation, but the amplitude is slowly decreasing over time. It emerges that when *analysing the sloshing phenomenon, by minimising the fluctuations in the propellant velocity, the changes in absolute pressure within the tank will not be dramatic but rather the pressure exerted on the tank wall will increase gradually, in stead of abruptly, ensuring a more stable and safe system for spacecraft.* The baffles aid in the loss of kinetic energy of the fluid. The fuel tank with 4 baffles is clearly the most effective design, out the fuel tanks tested, while the tank with no baffles was the least efficient. For the final 100 timesteps when analysing the results, it is found that the average velocity of the liquid oxygen within the fuel tank is only 20% of the velocity of the propellant in the tank with no baffles. For the last 200 timesteps the percentage is reduced even more, reaching a value of only 16.5% which means that for the last 10 seconds, on average, the propellant velocity in the fuel tank with four baffles in approximately 6 times less when compared to the other fuel tank. This is a significant finding. Finally, when comparing the average velocities for the entire simulation the differences are still significant. The average velocity of the fluid in the fuel tank with no baffles is 1.628 m/s while for the tank with baffles it is only 0.776 m/s which indicates that by introducing the 4 baffles, the overall sloshing velocity of the liquid oxygen is reduced by a factor of 1.5, confirming the excellent damping features of baffles in rocket fuel tanks. Excellent verification of the ANSYS FLUENT simulations has also been achieved with the SPH algorithm. Future studies will consider 3-dimensional visualizations with both techniques.

REFERENCES

- [1] H.A. Cole Jr, Baffle thickness effects in fuel sloshing experiments, *Technical Report, NASA Ames Research Centre, Mountainview, California, USA* (1966).
- [2] R. A. Ibrahim, *Liquid Sloshing Dynamics*: Cambridge University Press, New York, USA (2005).
- [3] ANSYS FLUENT ver 17.2, Theory Manual, Swanson Analysis Systems, Pennsylvania, USA (2017).
- [4] M. Valter, Rocket fuel tank sloshing (simulation with ANSYS FLUENT CFD), *BEng (Hons) Aircraft Engineering and Pilot Studies Project Dissertation, Aeronautical and Mechanical Engineering, Salford University, Manchester, UK*, May (2017).
- [5] O. Anwar Bég, B. Vasu, T. Sochi and V.R. Prasad, Keller box and smoothed particle hydrodynamic numerical simulation of two-phase transport in blood purification auto-transfusion dialysis hybrid device with Stokes and Darcy number effects, *Advanced Biotechnology and Bioengineering, 1,* 80-100 (2013).
- [6] O. Anwar Bég, S.S. Motsa, Eemaan T. A. Bég, A. Kadir, A.J. Abbas and Ayesha Sohail, Numerical study of nonlinear heat transfer from a wavy surface to a high permeability medium with pseudo-spectral and smoothed particle methods, *Int. J. Applied Computational Mathematics,* 3, 3593–3613 (2017).