Fuel tank sloshing: digital modelling with meshless CFD

by Giuseppe Usai[1], Michele Merelli[2]
1. Dallara Automobili - 2. EnginSoft

The highly competitive nature of motorsport drives designers to seek even the slightest advantage over their rivals. One potential area of improvement lies in reducing fuel sloshing and optimizing fuel extraction from the tank.

In this context, a new approach to fuel tank optimization is proposed. It aims to simplify the workflow by significantly reducing computational time compared to traditional finite volume method (FVM) CFD (computational fluid dynamics) tools. The proposed approach also explores the use of free solid bodies within the tank to mitigate fuel sloshing.

For applications such as these, conventional FVM CFD tools are computationally expensive and time-consuming. Using moving particle simulation (MPS), the proposed method provides a more efficient solution by using a Lagrangian formulation of the Navier-Stokes equations as opposed to the Eulerian approach used in FVM CFD. The analyses conducted demonstrated that meshless CFD software such as Particleworks can perform these types of simulations in less time and with fewer hardware resources than traditional FVM CFD methods.

Introduction and boundary conditions

One way a racing car manufacturer can improve the performance of a car is to lower the centre of gravity (COG) as much as possible. In fact, the performance of a racing car is affected by the position of its centre of gravity (COG): a higher COG results in greater lateral load transfer during cornering.

When the vehicle undergoes lateral acceleration, a “rolling” moment is induced by the fact that the COG is at a certain height above the ground. This moment must be balanced by the forces exerted on the tyres by the ground. The result is that the outer tyre is loaded than the inner tyre, resulting in less lateral “grip” and therefore slower lap times. Moreover, a higher roll moment causes more body roll, usually resulting in a loss of downforce, which is essential for maximizing cornering speed.

Another critical aspect to consider is the engine. At the bottom of each fuel tank are the so-called “feed pumps”, which draw fuel from the tank to the high-pressure pumps that feed the engine’s injection lines. Due to the high accelerations a racing car is subjected to, often exceeding 2g, it can happen that the fuel pumps do not receive

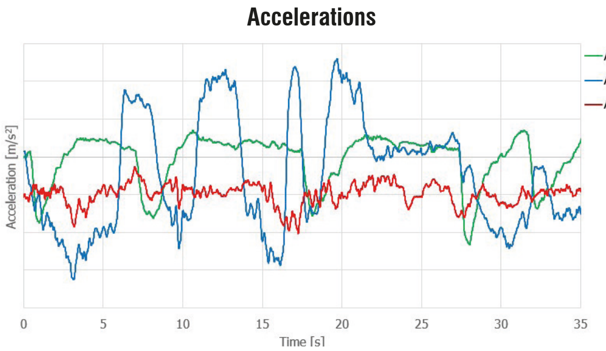

Fig. 1. Variation in the cost of a product depending on the stage of development.

enough flow to keep the engine running properly, causing a sudden power loss. The mass of the fuel cannot be neglected, and a proper analysis could be performed to limit the sloshing (movement) of the fuel when accelerating. The tools that can be used to develop this analysis are traditional CFD methods and meshless CFD.

This paper examines the MPS approach [1,2] with the objective of evaluating its cost-effectiveness compared to FVM CFD and studying the use of free solid bodies inside the tank to mitigate fuel sloshing. Specifically, the first part of the analysis aims to find the best compromise between computation time and accuracy by changing the particle size (PS). Then, a new solution involving the use of PTFE beads (commonly known as "Teflon®") to limit the sloshing was investigated.

A modern racing car was used as a case study. Its tank contains six feed pumps at the bottom and the accelerations shown in Fig.1 are imposed on the tank. Only the lap summary was considered, avoiding the study of the parts of the track where the accelerations are relatively low, since these sections do not produce results relevant to the analysis. The parameters observed are runtime, x, y, and z COG positions, and fuel flow rate in each pump.

Results of the particle size sensitivity study

Before analysing the effect of particle size (PS) on the results, it is important to note that the Particleworks simulation at PS 2.5mm and CFD are aligned. By changing the PS, a negligible difference is observed in the x and y coordinates of the COG. However, in the z direction, a maximum error of 0.9% is observed when the PS is increased from 2.5mm to 3mm, which increases to 1.0% and 2.1% when the PS is changed to 3.5mm and 4mm, respectively.

Examining the volume of fuel extracted from the tank, Fig.2 shows the instantaneous normalized volumetric flow rate of pump 3. The simulations performed with the 3mm and 3.5mm particle sizes show volumetric flow rates that closely match the 2.5mm simulation. However, the 4mm PS simulation deviates significantly from the predicted flow rate of the 2.5mm simulation.

Compared to traditional CFD tools, Particleworks allows for significantly reduced pre-processing and simulation times. Considering first the fluid-only simulation, the runtimes are shown in Fig.3: all Particleworks simulations were run on a single Nvidia A40 processor for a physical time of 36s, while the CFD simulations were run on a dedicated 8-node cluster with a total of 512 cores.

The results show that a PS of 3.5mm is the optimal compromise between accuracy and computational time, so this discretization was used for the solid sphere model.

Solid sphere model

A peculiar approach to the sloshing problem is to partially fill the tank with Teflon spheres, as revealed by some racing teams. To understand why it's important to study this scenario, consider the following examples:

- In distance-based races (e.g. Formula 1, Formula 2) where refuelling is not allowed, the tank will most likely be designed for the worst-case scenario of very high fuel consumption. On circuits where the expected fuel consumption is less than the tank capacity, teams can gain an advantage by limiting fuel sloshing by introducing other bodies into the tank, since the tank will not be completely full.
- In endurance racing (e.g. WEC, IMSA) cars undergo a balance of performance (BOP) process, where the series organizer intervenes in certain parameters to ensure a level playing field that allows virtually anyone to win a race. Some of the parameters that can be affected by BOP are: 1) engine power, 2) car weight, 3) stint energy (combination of fuel load and battery level).

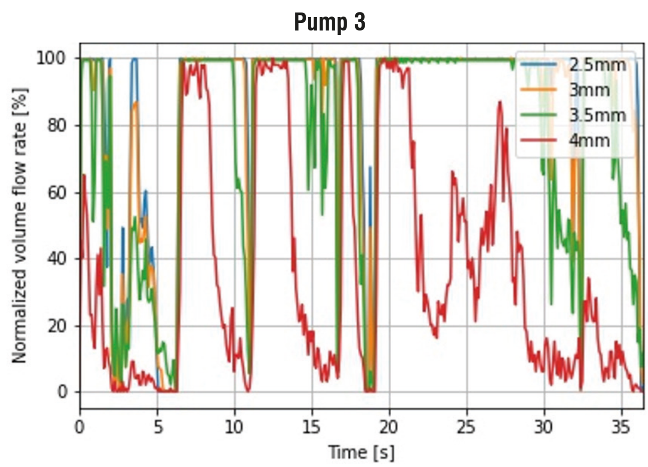

Fig.2. PS effect on the flowrate of pump 3.

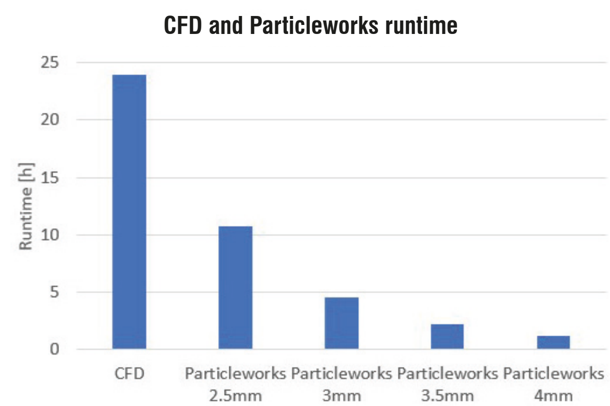

Fig.3. Comparison between FVM CFD and MPS in terms of runtime.

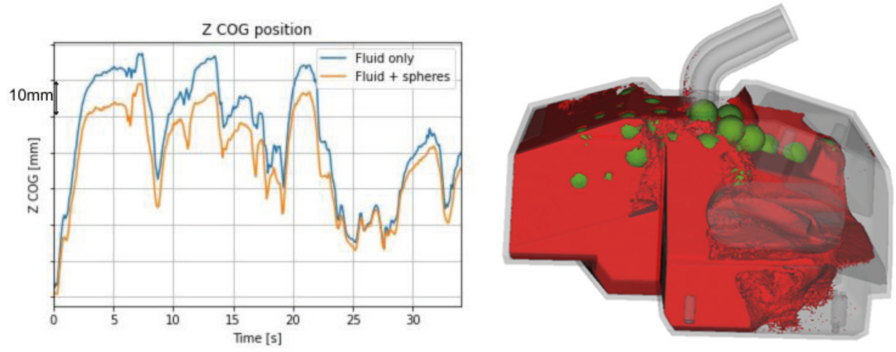

Fig.4. Left - Comparison of the Z COG position between the simulation with fuel only and fuel plus spheres. Right - The Teflon spheres (green) inside the tank.

Obviously if a given car is forced to refuel less than its maximum, it will have a partially empty tank immediately after the pits, and thus an advantage can be gained by limiting fuel sloshing.

This type of simulation (spheres are solid bodies and subject to buoyancy and gravity forces) cannot be performed with FVM CFD tools used by Dallara. We studied this solution with Particleworks.

The simulation runtime with Teflon spheres increased to 31 hours, mainly because of the reduced time step and higher particle count. Fig.4 shows the results of the analysis and compares them to a simulation where only fluid is present in the tank. It is evident that the inclusion of the spheres plays a significant role in limiting the movement of the COG.

Comparing the COG positions in the x, y, and z directions with the condition in which the tank is filled with fuel only, the addition of the spheres results in a reduction of the COG motion by approximately 14% in the x direction, 8% in the y direction, and 17% in the z direction.

Interestingly, no significant changes in fuel flow rate were observed during the transition from one configuration to another. This suggests that the presence of the spheres does not significantly affect the fuel extraction process.

Conclusions

The meshless CFD approach has proven to be an effective tool for the optimization of fuel tanks in racing applications. Through a particle size sensitivity study, it was determined that a particle size of 3.5mm provides highly accurate predictions of the COG position and flow rate. In particular, the use of Particleworks resulted in a significant reduction in both pre-processing and simulation times compared to traditional CFD tools.

The reduced run time allows Dallara to run sloshing simulations on more tracks. Whereas FVM CFD took 24 hours to simulate 36 seconds of physical time, the mesh-less approach allowed Dallara to run 400 seconds of physical time in the same timeframe.

In the final stage of the analysis, a comparison was made to assess the effect of introducing solid Teflon spheres into the fuel tank. The presence of these spheres was found to reduce fuel sloshing by approximately 14% in the x direction, 8% in the y, and 17% in the z direction, while no discernible effect on flow rate was observed.

Ultimately, the analysis allowed Dallara to better understand the sloshing problem and be prepared to extract maximum performance from fuel tank design in future projects.

Article from: www.nafems.org/publications/resource_center/bm_jul_24_3/

About Dallara Automobili

Dallara Automobili was founded in 1972 by engineer Giampaolo Dallara, who is also its current president, assisted by engineer Andrea Pontremoli as partner and Managing Director. The company headquarters are in Varano de' Melegari (province of Parma in Italy), Mr. Dallara's hometown. The company also has offices in Stradella di Collecchio (Parma in Italy) and Indianapolis (USA).

Today, the Dallara Group today is one of the world's leading companies specializing in the design, development and production of high-performance racing cars and is currently the sole supplier of cars for the IndyCar, Indy NXT, Formula 2, Formula 3, Euroformula, Formula E and Super Formula championships. Dallara builds cars for the WEC, ELMS, IMSA and NASCAR championships and provides expert advice and professional support to manufacturers and racing teams in the development of both competition and road cars.

In recent years, the company has also developed solutions for the aerospace and defence sectors. The Dallara Group currently employs over 750 people.

For more information:
Michele Merelli – EnginSoft
m.merelli@enginsoft.com

References

[1] S. Koshizuka, "Moving-particle semi-implicit method for fragmentation of incompressible fluid", *Nuclear Science and Engineering,* pp. 421–434, 1996.

[2] H. Isshiki, "Extension of Smoothed Particle Hydrodynamics (SPH), Mathematical Background of Vortex Blob Method (VBM) and Moving Particle Semi-Implicit (MPS)", *Scientific Research Publishing*, pp. 414–445, 2014.