Advances in Engineering Software 83 (2015) 31–44

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Automotive fuel cell sloshing under temporally and spatially varying high acceleration using GPU-based Smoothed Particle Hydrodynamics (SPH)

S.M. Longshaw *, B.D. Rogers

School of Mechanical, Aerospace & Civil Engineering, The University of Manchester, Manchester, United Kingdom

ARTICLE INFO

Article history:
Received 13 August 2014
Received in revised form 23 December 2014
Accepted 7 January 2015
Available online 6 February 2015

Keywords:
Smoothed Particle Hydrodynamics (SPH)
Sloshing
Fuel cell
Motorsport
GPU
CUDA

ABSTRACT

Understanding how fuel sloshes in a fuel cell, as a vehicle races around a circuit, is an important but mostly unexplored factor when designing fuel containment systems. Cell designs are based on knowledge of how liquids slosh in other containers, with the design and placement of structures, such as weirs, based on engineering judgement.

This work aims to provide better understanding for this difficult problem with a view to improve future designs. A Graphics Processing Unit (GPU) based Smoothed Particle Hydrodynamics (SPH) model is presented to simulate the fuel sloshing problem, with results from a simplified and real fuel cell geometry shown and compared against real data recorded in a vehicle. The vehicle motion and accelerations are included in the SPH simulations using a body force within the momentum equation. Results show good agreement between the simulation and the real fuel movement, with bulk motion captured well for accelerations up to 5 times gravity.

Focus is placed on the practicality of the method for use as part of an industrial design process, therefore the amount of time needed to compute results is considered throughout. Computational performance is found to be within acceptable limits, while numerical accuracy is actively considered through the use of Kahan compensated summation. It is concluded that the model is successful in capturing the necessary fluid dynamics for it to be useful in fuel cell design. It is expected that the method will provide insight into current cell designs and highlight where improvements can be made.

1. Introduction

Understanding fuel tank sloshing in high-end competitive motorsport is currently limited. While vehicle fuel systems (from containment to transport mechanisms) receive careful design, knowledge of violent sloshing motion due to braking, cornering and acceleration on the efficacy of the system is still mostly based on engineering judgement. This is primarily due to laboratory analysis of the motion of fuel contained within a vehicle cornering at up to five times gravity being a difficult, and therefore expensive, problem. Conventional computational fluid dynamics (CFD) solvers used in industry are unable to capture the complexity of the physics involved such as violent free-surface flow, however the Lagrangian technique Smoothed Particle Hydrodynamics (SPH) has been shown to be able to capture sloshing problems well [1–7]. This work looks to form better understanding of fuel sloshing in a cell under highly varying accelerations, using SPH simulations accelerated on a Graphics Processing Unit (GPU), along with quantitative data obtained from video footage recorded within a fuel cell (henceforth cell).

Cells used in high performance automotive vehicles, such as those which take part in motorsport, follow designs which have evolved from simple enclosures to geometries constructed from materials such as Kevlar. They include complex fluid transport mechanisms, combined with baffles and scavenging techniques, to produce a containment system designed to force fuel towards the point where it is extracted for use by the engine. Despite this complexity, little work has been done to understand how these designs control the flow of fuel during vehicle motion.

Simulation of this problem using mesh-based Eulerian CFD techniques is difficult as it is expected the fuel will form complex surfaces, such as breaking waves, and that an initial volume will separate into multiple discrete volumes due to the interference of structures within the cell. The range of physical processes involved is also arguably beyond the capability of many commercial CFD solvers as, due to the complex geometry of the boundaries

* Corresponding author.
E-mail addresses: Stephen.Longshaw@stfc.ac.uk, SLongshaw@gmail.com (S.M. Longshaw), Benedict.Rogers@manchester.ac.uk (B.D. Rogers).

http://dx.doi.org/10.1016/j.advengsoft.2015.01.008
0965-9978/

and violent forces from motion, simulated fluid must be able to follow complex bounded paths due to high acceleration.

The Lagrangian SPH method has long been considered a potential option for solving this type of problem. However, in order to capture important fluid detail, such as whether it is trapped by a structure within the cell, the simulation needs to be in three dimensions and include enough particles to represent the fluid motion accurately. Simulations also need to take place over long time-scales, with durations of over a minute likely, whilst still capturing phenomena occurring in a fraction of a second. This leads to a computational demand that cannot realistically be tackled by normal high performance computing (HPC) resources within a time frame that is suitable for industrial application.

High accelerations also pose difficulties to current SPH methodologies, particularly the boundary conditions. Bounding particles with high velocities that rapidly change direction can only be achieved as long as the time step taken within the simulation is sufficiently small such that their interaction with the boundary is captured. This creates not only computational demand, but also requires that the software implementation and SPH algorithms are sufficiently robust to maintain numerical stability at small time steps. In cases where the time needed to compute results is a secondary consideration, a solution to this problem is to use algorithms which are naturally more stable but introduce greater computational costs. However, when the focus is on producing an accurate simulation from which engineering decisions can be made but can still be computed in a reasonable time, this option becomes less appropriate. Instead the challenge is to find ways to make less computationally intensive methods, such as simple boundary conditions or lower order integration techniques, fit for purpose.

The recent development of general purpose computing on GPUs (GPGPU) and its subsequent application to the SPH method [8–10] has led to simulations involving large numbers of particles (of the order of 10×10^6) able to be computed on GPU equipped workstations [11]. This technological advancement has rendered SPH a suitable solution to the fuel sloshing problem, as models with appropriate numbers of particles can be calculated in time scales suitable for the industrial design process. This also partially addresses the problem of maintaining simulation stability at small time steps, due to the fact that more simulation steps can be calculated in less time. However, careful selection of methods and determination of suitable governing parameters is needed, as the use of GPU based programming focused on computational performance raises new problems associated with numerical accuracy, i.e. using fast math libraries or the fact that using double precision floating point operations to overcome rounding errors is a less viable option due to the significant drop in performance when compared to single precision operations. With that aim, this work uses the open-source GPU accelerated *DualSPHysics* software package [8] by applying its existing formulations to the problem and developing its algorithms where necessary.

The work presented herein shows that the classical weakly-compressible SPH method, combined with a simple boundary condition and physical approximations, such as the artificial viscosity scheme [12] and Shepard density filter [13], is able to capture the bulk motion of fuel within a complex cell while it is subjected to temporally and spatially varying forces of up to five times gravity. Applied accelerations are captured using an inertial measurement unit (IMU) and comparison is drawn against real video footage of fuel sloshing in the cell, with a good correlation between simulation and reality found.

The use of GPGPU programming to implement the model is also shown to be an effective solution for SPH, as it allows the design engineer to iterate their own simulations on a private workstation, rather than relying on the more traditional industrial work-flow of the dedicated CFD team of engineers paired with an expensive many-processor HPC resource. In reducing the time needed to simulate an SPH model from a few days to a few hours and combined with the powerful pre- and post-processing tools of *DualSPHysics*, it is shown that SPH can be utilised within an engineering environment that relies on rapid prototyping.

This paper is structured as follows, the basic fuel cell arrangement is described, then the SPH methodology is introduced, followed by a description of the GPU implementation used. Simulation results using a simplified cell geometry, as well as fluid level results and computational run-times are then presented and discussed. Finally, conclusions are drawn and future developments outlined.

2. The fuel cell

Fuel cells used in motorsport are complex systems which transport fuel to ensure the engine has a constant supply, requiring containment and control of flow as the fuel sloshes due to violent movement. This is currently achieved using physical barriers such as baffles, combined with both passive and active fuel transport mechanisms such as pumps or one-way valves. To add to design complexity, some types of motorsport have requirements that the amount of fuel held in the vehicle at any one time is tightly controlled, with regulations such as a minimum quantity of fuel remaining in a specific portion of the cell at the end of a race.

With these considerations, a detailed understanding of the hydrodynamics of the fuel during typical racing conditions becomes a fundamental criterion for informed design of the cell. Knowledge of this detail is currently limited and careful redesign of a cell could result in notable performance improvements in the modern motorsport vehicle. There is then potential that these advancements can filter down to more typical road-going vehicles.

Due to confidentiality it is impossible to show the geometry of the actual fuel cell, which also includes visualisations of simulated flow compared with in-tank video footage. For this reason, a simplified cell geometry is used throughout the majority of the results for illustrative purposes and any important differences between the simplified cell and actual cell described. In Section 5.3 we present comparisons of the computed fuel levels with those measured in the real cell.

2.1. Cell design

While the exact details of cell designs are kept confidential within the motorsport industry, it is possible to describe the principles of design using a simplification of the current technology. Fig. 1 shows the main components of a design, using a simplified geometry that shows the relevant portion of a cell, with some intricate detail removed. While numerical results in Section 5.3 are obtained from simulations performed using a real cell design, the same key design points remain in the simplified variant, in the form of a central obstruction and appropriate dimensions. The origin of the coordinate system should be considered to be the geometric centre of the cell, found according to the box which bounds it in its entirety.

The cell is split into distinct areas in order to impede the momentum of the fuel; only one of these areas is shown here. This design introduces an interesting complexity in that it is necessary to ensure that fuel is constantly being forced between the separate areas of the cell in order to replenish fuel collected from the final drain point in the bottom-most cell. To ensure that the points of fuel transport are always supplied, even when it is violently sloshing, various forms of obstructions are utilised (which are not