VI International Conference on Particle-based Methods – Fundamentals and Applications
PARTICLES 2019
E. Oñate, M. Bischoff, D.R.J. Owen, P. Wriggers & T. Zohdi (Eds)

EVALUATION OF PUDDLE SPLASH IN AUTOMOTIVE APPLICATIONS USING SMOOTHED PARTICLE HYDRODYNAMICS

MURALEEKRISHNAN MENON[1], G. V. DURGA PRASAD[1], KEVIN VERMA[1], and CHONG PENG[1]

[1] ESS Engineering Software Steyr GmbH
Berggasse 35, 4400 Steyr, Austria
muraleekrishnan.menon@essteyr.com; www.essteyr.com

Key words: Fluid dynamics, Divergence Free SPH, Graphics Processing Unit

Abstract. The impact of splashing water as a car moves through water-soil puddle or flooded areas is significant to the automotive industry. Due to the multifold advantages of simulating this problem, several numerical approaches exists to understand the fluid dynamics and the fluid-structure interaction in such scenarios. The current research focuses on obtaining fluid dynamics of the splashing phenomena as a first step towards simulating such problems through a Computational Fluid Dynamic (CFD) approach by adopting the Smoothed Particle Hydrodynamics (SPH). As a mesh-free Lagrangian-based method, the SPH framework tracks particle behavior in the computational domain at each instant of dynamic simulations. In contrast to traditional grid-based methods, SPH is well suited for simulating fluid dynamic problems involving free-surfaces, multi-phase flows, and involving objects with high degree of deformations.

The current study presents results and discusses the observations from simulating a car Body-In-White (BIW) geometry with four tires that moves through a water puddle as a normal car would. The SENSE solver developed at ESS Engineering Software Steyr adopts the SPH framework and presents several formulations. The solvers are implemented on Graphics Processing Units (GPU) to enable its usage for industrial applications. The framework is developed to be easily parallelizable allowing multiple GPU simulations, that are useful for industrial problems involving huge number of particles. In addition to the ease of scaling, it also permits computations at higher particle resolutions when needed to handle specific physical constraints of a problem. The simulations discussed in the current study were performed using the Divergence-Free (DF-SPH) formulation in SPH, and implemented on multiple-GPU devices, with a particle discretization that allows to see the properties to the order of 5 *mm*.

1 INTRODUCTION

On a rainy day, it is inevitable in most parts of the world, that one needs to drive through stagnant water on roads, negotiate through ditches filled with rain water. Rain water, splashes and hits the lower side of the chassis and spreads onto the engine bottom, and other areas under engine hood. This can damage a vehicle's engine, cause the brake rotors to warp from rapid cooling when immersed in water, cause loss of power steering, and short circuiting of electrical components in a vehicle. Further the presence of water puddles at specified locations for long periods leads to the potential risk of rusting of metal objects. The acidic nature of polluted water (rain water mixed with mud particles) will damage the paint and eventually reduce the life of chassis. Automotive companies are continuously trying to reduce the cost of production and improve the design aspects to ensure longevity of products as well as customer satisfaction. The impact of splashing water on a car body as it moves through a water-soil puddle region or flooded areas is significant due to several reasons. These aspects have been of interest to the automotive manufacturers and component suppliers for several years. They have primarily relied on experimental investigations to improve design aspects of chassis, engine casing, and other relevant components. Recent years have seen increased usage of numerical methods and simulations to understand these phenomena. A preliminary investigation of fluid dynamics and the fluid-structure interaction in such scenarios can be helpful in estimating the impact at a reduced cost to the manufacturer. The current research focuses on establishing fluid dynamics of the splashing phenomena as a first step towards simulating such complex problems for automotive industry.

In the automotive industry, the adverse effects of puddle splashing are wide ranging and it is important to estimate the impact at an early stage of the design process [1, 2]. While negotiating a puddle with depth larger than the ground clearance of automobile, water is sucked into the engine through air intake passages which are present in the lower part of most modern cars (especially due to low ground clearance), which causes the engine to seize. This is known as 'hydrostatic lock' – water enters an engine cylinder and during the compression stroke will lock the engine piston in place. This in turn overloads the connection rod, causing it to deform and cause significant engine damage. An example of such a situation is shown in figure 1. In extreme case, this could even cause permanent and irreversible damage to engine or engine parts. Manufacturers provide protective and safety mechanisms to deal with such scenarios: the air intake passage is provided with water trap, which drains the sucked water back out; electrical wiring connectors in the engine bay are water-tight sealed with silicone gaskets, and so on. However, a detailed assessment of these protective mechanisms help the designers to preempt potentia dangers and device more advanced mechanisms to mitigate such adverse scenarios. A prior investigation of water splash scenario using advanced numerical simulation tools help to identify probable areas of a car body that are vulnerable for damage, if and where water enters, and to estimate potential risks of object deformation from suspended solid particles splashing with the water. There are several advantages of computational analysis of such scenarios, as opposed to setting up realistic simulations in an experimental lab.

Figure 1: Example of a flood situation that could cause static engine hydrolock [1].

With advancements in the field of Computational Fluid Dynamics (CFD), there are numerous approaches to simulating the splash phenomena for automotive application. The most traditional and established methods are Finite Volume based methods which rely on a spatial numerical grid to descritize the fluid and solid domains.However, such Eulerian-based methods are dependent on the quality of volumetric meshing, and are disadvantageous from the perspective of computational resources as well as time efficiency of simulation. Adding to this is the complexity of modeling scenarios such as splashing, which involves moving boundaries and rotating objects that force very low time steps. An alternative and promising approach is adopted in this study, which involves particle-based meshless method known as Smoothed Particle Hydrodynamics (SPH). Lack of a numerical grid makes SPH method more suited to handle complex geometries and moving objects. Nowadays, the SPH approach is more and more commonly used for hydro-engineering applications involving free-surface flows where the natural treatment of evolving interfaces makes it an attractive method.

Some relevant outcomes of simulating this problem is understanding the impact on underbody of the car, estimating water penetration in the engine compartment area, and visualizing the water splash distribution.

2 COMPUTATIONAL APPROACH: SENSE

The SPH solver used in this study was developed at ESS Engineering Software Steyr GmbH and will be referred to as SENSE solver. The pre-processor and post-processor parts of the SENSE were also developed in-house to be compatible with the SPH solver, and were used to set-up the problem as well as to analyze the results. The SPH is primarily a particle-based method that solves equations of motion for each particle in the domain based on their interaction with neighbouring particles. The SENSE approach for solver design is to simulate fluids and solids for specific physics-based problems such that simulation of industrial problems may be perfoemd with minimal to zero expertise in CFD. The computations are performed on Graphics Processing Units (GPU), which are much faster than Central Processing Units (CPU). Additionally, the algorithm for SPH is highly parallelizable on GPU devices, providing high speed-up possibilities for scaled industrial solutions [3].

The main idea behind SPH is the use of integral interpolants to evaluate field properties of discretized particles at each time step (see [4, 5], [6] , and [7]). Based on the treatment of incompressibility, there are several options/approaches available in SPH technique. The solver used for this study adopts a Divergence Free formaulation to treat the incompressibility and hence is known as the DF-SPH solver **Peng2019**. The SPH approach primarily involves two stages - an integral interpolation of physical properties based on a suitable kernel function $W(r, h)$, and a field discretization of fluids (or solids) into particles that carry physical properties as a function of position $f(r)$. The integral interpolant for a field property $f(r)$ is written as,

$$\widehat{f}(\mathbf{r}) = \int_{\Omega} f(\mathbf{r}')W(\mathbf{r} - \mathbf{r}', h)d\mathbf{r}', \tag{1}$$

where r' is the neighbouring particles in a compact domain defined by the smoothing length h, W is the kernel function, and the integration is performed on the entire domain Ω. As the domain is discretized as particles, the integral interpolant maybe written in a summation form as,

$$\langle f \rangle (\mathbf{r}) = \sum_{b} f(\mathbf{r}_b)W(\mathbf{r} - \mathbf{r}_b, h)\Omega_b, \tag{2}$$

where r_b and Ω_b denote the position and volume of all particles b in the compact domain. An additional advantage of SPH method is the possibility to represent the gradient of the field property using the gradient of the kernel function due to the imposed symmetry condition and mathematical identities. The gradient of a preprty $f(r)$ can hence be written as,

$$\langle \nabla f \rangle (\mathbf{r}) = \sum_{b} f(\mathbf{r}_b)\nabla W(\mathbf{r} - \mathbf{r}_b, h)\Omega_b, \tag{3}$$

where ∇W is the gradient of the kernel, and hence the gradient of the field variable $f(r)$ dependents only on the field value and not on its gradient. This makes the implementation and computations much easier.

For fluid dynamic application such as the one covered in this study, SPH technique solves the Navier-Stoke's equations for incompressible fluid flow written in the Lagrangian form. The primary equations of interest are that of continuity and momentum conservation,

$$\frac{d\rho}{dt} = -\rho\nabla \cdot \mathbf{u}. \tag{4}$$

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\left(\nabla \cdot \mu\nabla\right)\mathbf{u} + \mathbf{f}, \tag{5}$$

where $\rho = const$ is the density, $\mathbf{u}$ the velocity, t the time, p the pressure, ν the kinematic viscosity and $\mathbf{f}$ are accelerations due to external forces. These equations are then written in SPH discretized format following equations 2 and 3 methodology and using certain mathematical identities. The SPH discretized mass and momentum equations are solved using explicit time integration scheme to march forward in time. In this work the second-order Symplectic integrator is employed. With the solution of the two conservation equations, we can obtain the motion of the particles and the evolution of the carried variables. More details of the fluid dynamic formulations maybe obtained by review of [8]. Conventiomally, the imcompressibility in SPH is treated by assuming a Weakly Compressible formulation, which involves the use of an additional equation of state. For stability purposes, this imposes a very small time steps making the simulation slower especially when it involves huge number of particles. To allow fast and efficient computations of industrial cases, a Divergence-Free formulation is used for incompressibility treatment, more information on which may be obtained from Chitneedi et. al. [9].

Despite all the benefits of particle-based modeling, one of the main disadvantages of SPH is the huge numerical complexity. In order to model real life phenomena in appropriate resolution, typically up to several hundred millions of particles need to be considered, which frequently results in large execution times. However, the discrete particle formulation renders the SPH method suitable for parallelization, which allows for a massive speedup – e.g. using the *General Purpose Computations on Graphics Processing Units* (GPGPU) technology. SPH solutions utilizing the computational power of GPUs have initially been introduced by [10] as well as [11], where the *Open Graphics Library* (OpenGL) was employed. Later, SPH implementations based on the *Compute Unified Device Architecture* (CUDA) have been developed by [12].

However, in order to simulate huge domains involving millions of particles, a single GPU device is usually not sufficient anymore. In these cases, the particle domain needs to be distributed over several devices – yielding a *multi-GPU architecture* as presented in [13]. These SPH multi-GPU solutions employ a spatial subdivision of the domain to partition the whole domain into individual subdomains. These subdomains are distributed to the corresponding GPUs and executed in parallel. Receiving optimal performance for SPH on multi-GPUs is a highly non-trivial task due to the massive amount of synchronization needed between the distinct subdomains. In order to overcome these shortcomings a range of optimization techniques are introduced as discussed in [14, 15]. Employing these techniques eventually allows for efficiently applying SPH to engineering applications that involve millions of particles.

3 COMPUTATIONAL CASE STUDY

In automotive industry, the comprehension of fluid dynamics in splashing scenario is relevant to improve component designs. The SENSE solver provides a convenient platform to perform fluid dynamic simulations without significant expertise. Preliminary simulations were conducted for a typical puddle splashing scenario, where a car body is moving through stagnant puddle of water at a nominal speed. The SENSE pre-processor capabilities were used to setup the initial and boundary conditions enabling SPH-based computations on multiple GPU devices, and the results were analyzed for visualizing the fluid dynamics and identifying impact regions. The current section covers the approach

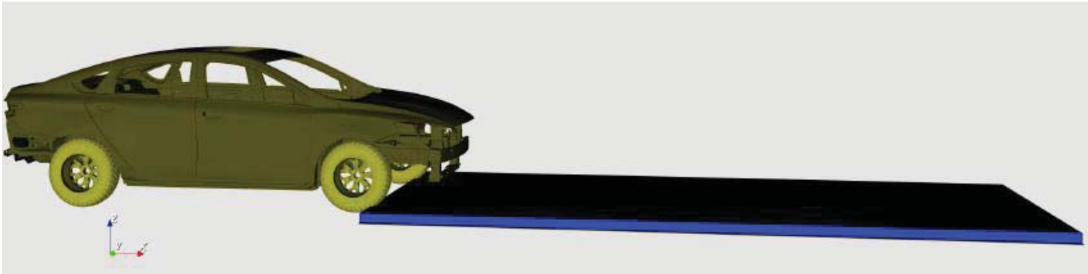


Figure 2: Sample of problem setup showing car geometry at the edge of water puddle.

adopted to setup the problem and presents some preliminary results and observations.

The puddle splashing case is setup with a Body-in-White (BIW) geometry, which is fitted with four tyres such as to emulate real car movement through puddle. The car geometry is approximately 4 m long, 2 m wide, and 1.2 m high, which is defined to moved along a straight line in the X-direction at a speed of 30 km/h. For the current scenario, the puddle consist of only water with density $\rho = 1000\ kg/m^3$ and kinematic viscosity, $\nu = 8.94x10^{-4}\ m^2/s$. At the beginning of time, the car is placed outside of the puddle such that the front tyres are slightly in contact with the edge of the puddle. The puddle is a rectangular region 6 m long and 80 mm deep. To ensure that the splashing water stays within the computational domain, the puddle is designed to be wider than the total width of the car geometry. The computational domain is further extended in the axial and lateral directions to avoid bouncing of splashing fluid particles. Figure 2 shows a sample of the problem setup at the beginning of time. It is also relevant to note that the Lagrangian-based solver performs a single-phase SPH simulation that considers only water without the effects of air. Additionally, surface tension effects and fluid drag are not currently modelled.

The computational domain is discretized into fluid and solid particles whose physical properties are tracked by the SPH fluid dynamic equations. For the SPH solver, a smoothing length of $h = 0.0055\ m$ is chosen along with an $\frac{h}{dr}$ value of 1.1, which ensures that results are captured to the resolution of 5 mm particle size. The initial setup consist of 13.6 million particles including both fluid and solid domains. Dynamic simulations are run for a time period such that the car geometry traverses till the end of the axial domain. Following sub-section presents some results from one such simulation, and discusses the observations. The numerical computations were deployed on multiple GPU devices, each having the configuration of an NVIDIA GeForce GTX 1080Ti with 11 GB memory.

3.1 Simulation Results

The results presented in this sub-section are from simulation of the BIW geometry moving at 30 km/h through a water puddle 80 mm deep and longer than the length of the car body itself. Simulation were deployed on 8 GPU devices with the configuration mentioned above. It takes approximately 0.72 s for the car geometry to move from the left part of the puddle till the other end. The total time required to simulate this physical sit-

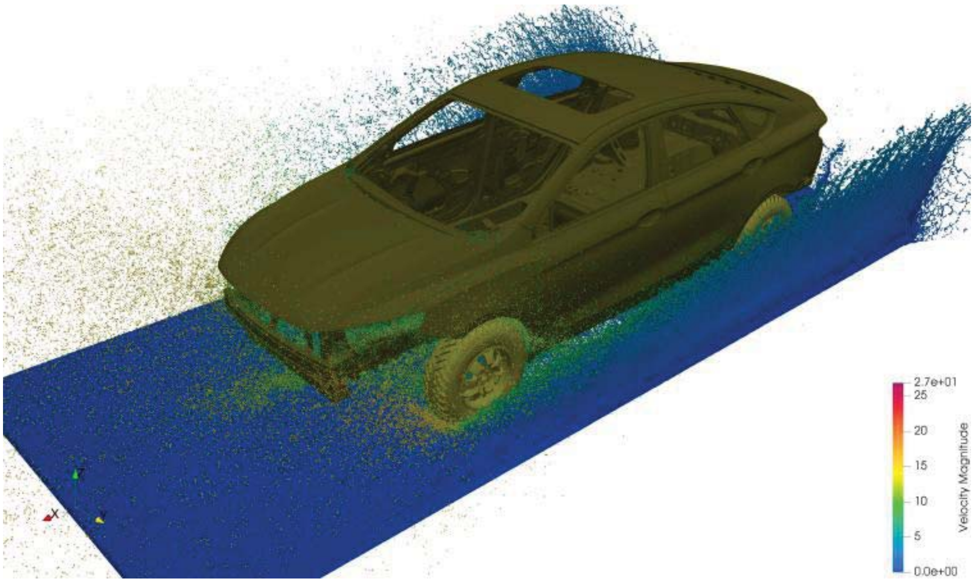
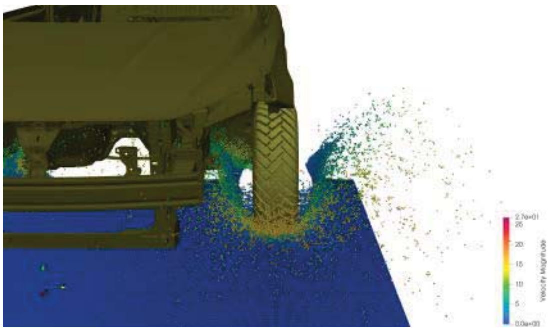


Figure 3: Puddle water distribution presented at $t = 0.5\ s$, coloured by velocity magnitude of water.



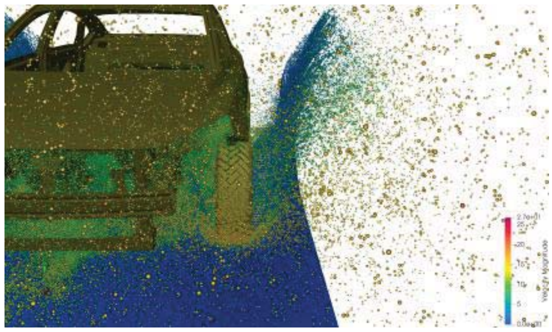


Figure 4: Velocity contour represetnation of water particles splashing near the front-right tyre shown at two stages. Left: $t = 0.1\ s$ and right: $t = 0.5\ s$.

uation was 2.5 $hours$. As an example of fluid dynamics during car movement through the puddle, some velocity contours of water movement and some other observations relevant to impact analysis are shown below.

Understanding the dynamic motion of puddle water and potentially suspended solid particles is significant to the impact caused by car moving through flooded areas. As this study covers only the fluid dynamics, we discuss velocity profile of the water puddle at different stages of simulation. At time $t = 0.5\ s$, the splash phenomena looks as shown in figure 3, where the car has moved completely into the puddle region. Initially, as the front tyres start moving into the puddle, water is pushed outside. Figure 4 shows this fluid distrbution at two time stages - $t = 0.1\ s$ and $t = 0.5\ s$.

A different perspective of the fluid distribution can be seen in figure 5 showing the top view of the domain at the same stages of simulation. Additionally, figure 6 shows only

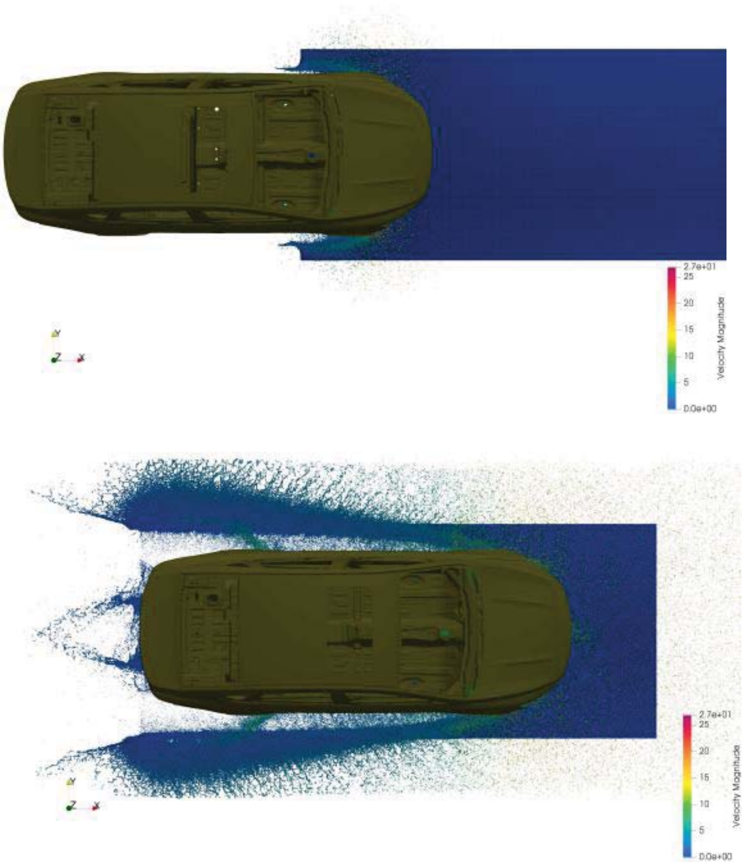


Figure 5: View of splash phenomena in the domain seen from above, fluid particles coloured by velocity magnitude. Top: $t = 0.1$ s and bottom: $t = 0.5$ s.

the fluid domain at $t = 0.5$ s. It may also be noted that the tread marks left by the tyres are noticeable here. This representation of free-surface fluids can be improved by increasing particle resolution, i.e. updating solver parameters to discretize the domain as smaller particles.

Another aspect of these simulations are the ability to understand impact on the car body due to splashing, which is of high importance to manufacturers and suppliers. From preliminary analysis, we are able to show the contact time of fluid particles hitting the solid car body. An example is shown in figure 7 that shows a contour plot if time (in s) that the car body comes in contact with splashing water, seen from underneath. This gives an initial estimation of car body regions impacted by splashing. Additionally, figure 8 also shows the contact impact on the front-underneath part at different time stages. The comparison of contact-time legend in figure 8 shows how regions with higher impact can be isolated.

The SENSE approach makes it easier to study several scenarios in a short time span. For example, the effect of different puddle depths or flooding situation can be simulated by merely changing a few parameters in the problem setup. Studying the effects on another

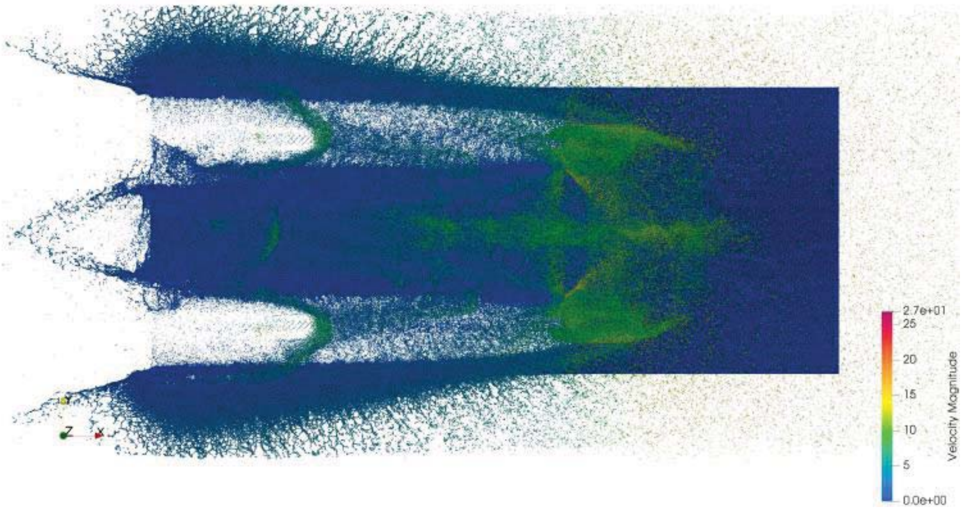


Figure 6: Representaion of fluid distribution viewed from top at $t = 0.5$ s, fluid particles coloured by velocity magnitude.

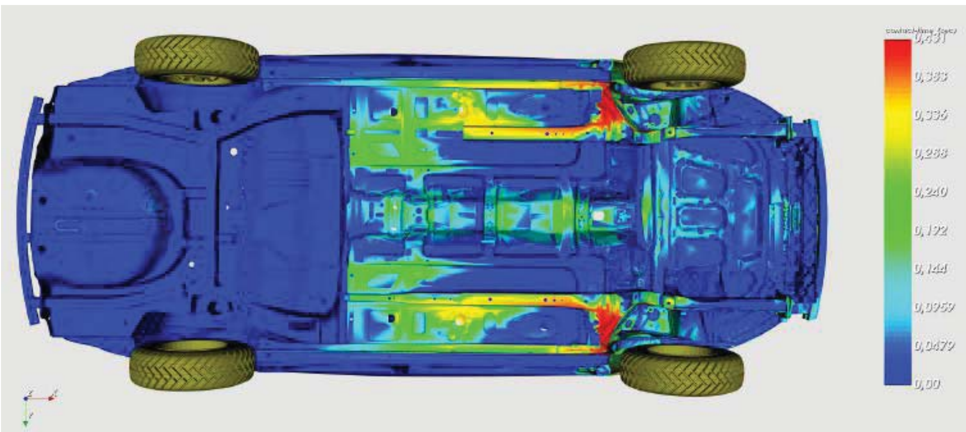


Figure 7: Depiction of splash impact underneath the car surface, shown as the fluid-solid contact time on the car body at $t = 0.5$ s.

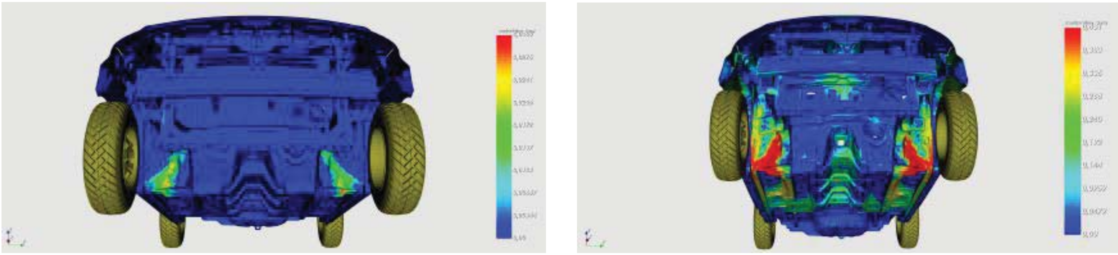


Figure 8: Comparison of impact at front-bottom of car body. Left: $t = 0.1\ s$ and right: $t = 0.5\ s$.

car geometry is also fairly easy by replacing the geometry in the pre-processor. It was also observed in the analysis that the particle-based study of such splashing scenarios makes it possible to evaluate where water seeps into the car body.

4 CONCLUSIONS

The current research presents a Lagrangian particle-based approach to study water splashing phenomena applicable to the automotive industry. Results are presented for an example fluid dynamic study that covered a Body-in-White with tyres moving through a water puddle region. Simulating splash phenomena using an advanced Lagrangian-based solver like this, allows to fairly understand the dynamic free-surface flow as well as to get an estimation of fluid impact on the car body. Implementing on GPUs, with an easy-to-scale framework for parallalezation on multiple devices, industrial cases can be evaluated in a relatively short period of time. This expands the scope of scenarios that can be studied in a given time frame, providing a good balance of expenditure and amount of data available for design updates.

REFERENCES

[1] Tutu, A. Engine hydrolock - how water can damage or destroy your engine. (Accessed Jun 2019).

[2] DrivingFast.net. Driving in deep water. (Accessed Jun 2019).

[3] Szewc, K., Mangold, J., Bauinger, C., Schifko, M., and Peng, C., 2018. “Gpu-accelerated meshless cfd methods for solving engineering problems in the automotive industry”. In SAE Technical Paper, no. 2018-01-0492, SAE International.

[4] Monaghan, J., 1992. “Smoothed particle hydrodynamics”. *Annual Review of Astronomy and Astrophysics*, ***30***, pp. 543–574.

[5] Monaghan, J., 2012. “Smoothed particle hydrodynamics and its diverse applications”. *Annual Review of Fluid Mechanics*, ***44***, pp. 323–346.

[6] Liu, M., and Liu, G., 2003. *Smoothed Particle Hydrodynamics: A Meshfree Particle Method*. World Scientific, Singapore.

[7] Violeau, D., 2012. *Fluid Mechanics and the SPH method: theory and applications*. Oxford University Press.

[8] Liu, M. B., and Liu, G. R., 2010. "Smoothed particle hydrodynamics (sph): an overview and recent developments". *Archives of computational methods in engineering,* ***17***(1), pp. 25–76.

[9] Chitneedi, B. K., Peng, C., and Verma, K., 2019. "Modeling flood waxing for automotive cavity protection using divergence-free smoothed particle hydrodynamics". In Proceedings of the14th SPHERIC Workshop.

[10] Kolb, A., and Cuntz, N., 2005. "Dynamic particle coupling for gpu-based fluid simulation". In Int. Proc. of the 18th Symposium on Simulation Technique, pp. 722–727.

[11] Harada, T., Koshizuka, S., and Kawaguchi, Y., 2007. "Smoothed Particle Hydrodynamics on GPUs". In Proceedings of 5th International Conference Computer Graphics, pp. 63–70.

[12] Herault, A., Bilotta, G., and Dalrymple, R., 2010. "SPH on GPU with CUDA". *Journal of Hydraulic Research,* ***48***, pp. 74–79.

[13] Domínguez, J. M., Crespo, A. J. C., Valdez-Balderas, D., Rogers, B. D., and Gómez-Gesteira, M., 2013. "New multi-gpu implementation for smoothed particle hydrodynamics on heterogeneous clusters". *Computer Physics Communications,* ***184***(8), pp. 1848–1860.

[14] Verma, K., Szewc, K., and Wille, R., 2017. "Advanced load balancing for SPH simulations on multi-GPU architectures". In IEEE High Performance Extreme Computing Conference (HPEC), pp. 1–7.

[15] Verma, K., Peng, C., Szewc, K., and Wille, R., 2018. "A multi-gpu pcisph implementation with efficient memory transfers". In IEEE High Performance extreme Computing Conference (HPEC), pp. 1–7.