

Water Ingress Analysis and Splash Protection Evaluation for Vehicle Wading using Non-Classical CFD Simulation

Prashant Khapane and Vivek Chavan
 Jaguar Land Rover

Uday Ganeshwade
 Tata Technologies, Ltd.

ABSTRACT

Physical testing of a vehicle wading through water is performed to gauge its capability to traverse through shallow to deep levels of water, wherein various vehicle performance parameters are observed, recorded and analysed. Jaguar Land Rover (JLR) has instigated and established a comprehensive CAE test procedure for assessing the same, which makes use of overset mesh (in a CFD environment) for a non-traditional approach to vehicle motion. The paper presents investigations made into the established wading physics, in order to optimise the splashing and water jet modelling. Large Scale Interface model was implemented instead of the previously standardised VOF-VOF fluid phase interaction model, and a comparison is made between the two. The implemented wheel rotation approach was scrutinised as well and appropriate inferences are drawn. JLR has devised an extended simulation methodology, making use of the passive scalar model, to understand water impingement and splashing in a greater detail; which are the two critical performance parameters during vehicle wading. Passive scalar acts as a pseudo catalyst in the target fluid phase of the multiphase CFD simulation. Analysis carried out using this model gives the amount of water ingress from specific locations on the vehicle and also the amount of water subsequently splashing on the pre-set target locations or components in the engine bay. The assessment was performed on a simplified block model, scaled down with reference to vehicle dimensions, which permitted reiterative simulation setup with minimum computational cost. The methodology was successfully implemented on a full scale JLR vehicle later. Thorough physical testing of vehicles was carried out so as to understand and assess water ingress and in turn, drive further developments in CAE.

CITATION: Khapane, P., Chavan, V., and Ganeshwade, U., "Water Ingress Analysis and Splash Protection Evaluation for Vehicle Wading using Non-Classical CFD Simulation," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1327.

INTRODUCTION

Vehicle wading refers to driving the vehicle through a fluid of high density (much denser than air); which in most practical cases is water with or without suspended matter. As the vehicle drives, water impinges onto the underbody trims and onto other components such as the engine bay, transmission system and the subframe. This could potentially cause a number of problems for the vehicle [1], which have been listed in an order of increasing severity below:

- Damage to the underbody trims and wheel arches
- Water ingress into the transmission system
- Impingement onto the electric components in the engine bay
- Water ingress into the Air Intake System, causing hydraulic lock

Conventional approaches make use of prototype vehicle testing for studying damage to the vehicle underbody trims and engine bay due to wetting. This offers little insight into the water splashing, contamination sources and severity, since setting up a real-time

inspection of the same becomes extremely challenging and offers little help [1]. Issues are often detected very late in the development process and last minute changes end up bearing additional cost to the business.

A virtual analysis methodology was deemed necessary for understanding these problems, which would assist in this study during the early phases of vehicle development. Hence, JLR has established a comprehensive CAE process for understanding the effect of vehicle wading on vehicle performance and functionality [2]. Durability and Robustness (D&R), a vehicle engineering Attribute team within JLR uses the process to obtain the pressure loads on the underbody components and then predict the possible modes of failure for the same.

The complexity of physics involved and the unsteady nature of the simulation make it a peculiar application of CFD; while also adding to the computational cost. The patented process [2] involves the vehicle moving through a stationary water trough, which ensures that the dynamics of a physical vehicle moving through a flooded road is replicated as closely as possible and the analysis is accurate.

The virtual analysis methodology did not facilitate a thorough understanding of water ingress. D&R perceived the importance of thoroughly inspecting water ingress in the vehicle front end from different openings and understand how the vehicle can be designed for optimal splash protection. The main focus was to not only assess the water ingress, but also to effectively predict its most likely sources. This would then assist in a better overall understanding of vehicle wading and water ingress and, consequently aid vehicle design with regards to robustness to splash.

The approach outlined in this paper aims at enhancing the virtual wading analysis methodology that JLR has established and correlated [3]. The challenge was to make these modifications without unnecessary alterations in the simulation setup and increase in computational cost.

The key objectives of the study are as follows:

1. Improving the capability of the simulation to replicate real life behaviour of water splashing and contamination, as a vehicle moves through a flooded road, for different vehicle speeds and water depths.
2. Assessing water ingress and splash in an effective way so as to accurately predict the vulnerable components and openings.
3. Improve the overall virtual splash protection evaluation capabilities.
4. Achieve the above whilst limiting the increase in computational cost.

RELATED WORK

D&R has devised and standardised physical testing procedures and design rules at JLR, which specify the norms against which the vehicle should be assessed, depending on the vehicle category and use. The procedure simulates practical scenarios that the vehicles often encounter when driving through flooded areas. The chief requirement of the test is that the vehicles should remain fully functional afterwards. The CAE process [2] has been designed to cater to these requirements in the most efficient way.

Prior to the establishment of the virtual process, relevant work on the study of vehicle wading was extremely sparse. Zheng et al. [4] present one such study, where the focus of the CFD analysis was assessing firewall water scouring and its remedies.

Khapane et al. [3] have elaborated on the challenges faced in the development of a robust virtual analysis method of vehicle water wading and the devised solutions for the same. A non-classical approach was deemed necessary in order to accurately capture the effects of the bow wave formed in the water body [3]. This was achieved using chimera interpolation between the vehicle region and the background (domain) region. This allows relative motion between different continuum regions [6]. This enabled a more realistic and accurate flow pattern and bow wave in the water body as the vehicle moves through it. The trade-off in this case is the increased size of the computational domain, which directly results in an increased cell

count and computational cost. Simulation complexity also increases due to the Rigid Body Motion (RBM) between the vehicle region and the background region.

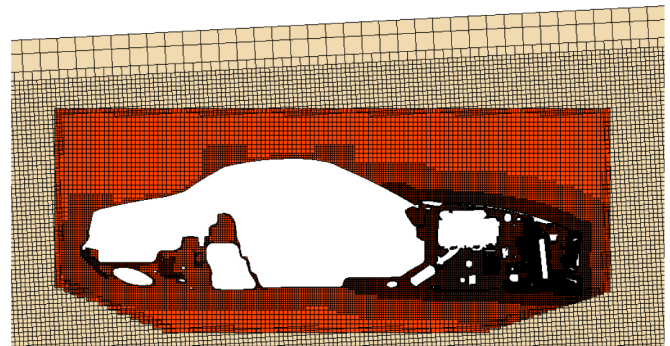


Figure 1. Cut section plane, showing the vehicle region (highlighted in dark orange) and the background region. The cells on the intersection of the two regions act as acceptor and donor cells.

The predominant focus of the methodology is to predict the failure modes of the underbody trims and wheel arches owing to high pressure forces from water. Khapane et al [5] have demonstrated this approach. The pressure load acting on the underbody components at predefined short intervals of time is recorded and mapped onto the solid meshed structural component. The FEA solver, Abaqus, is used to calculate the stresses acting on the component, along with the resultant deformation and the fixing loads (shear and axial forces). If the analysis indicates that underbody component is likely to undergo failure, specific design modifications made so as to ascertain that the component can sustain the loads.

DEEP WATER WADING AND SPLASH WADING

As a vehicle wades through water at low speeds, the inertial forces in the water body are low and hence, minimal splashing and water droplet breakage occurs. The Weber number and the Froude number are low and the magnitude of disturbances on the free surface of water is low as well [7]. A bow wave is formed in front of the vehicle, followed closely by a trough near the vehicle body. In case the vehicle halts or slows down, the water from the heightened bow wave impinges onto the vehicle and impinges onto the engine bay and underbody components. A vehicle can drive through higher depths of water at low speeds [1]. In this case, the components that get affected are often submerged in water, either fully or partially.

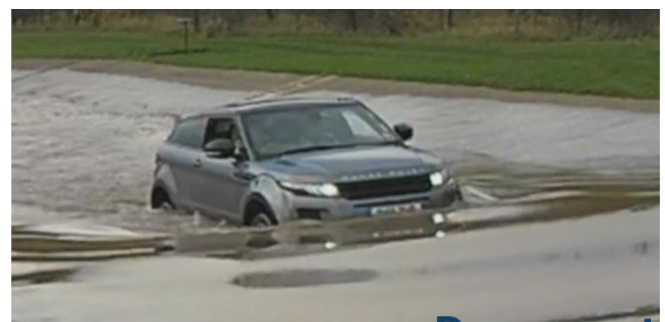


Figure 2. A Range Rover Evoque wading through water trough at a 7kph at Millbrook test facility. The bow wave formed at the front can be observed.

As the vehicle traverses at high speeds, the Weber and Froude numbers are high, owing to high inertial forces; resulting in a high degree of splashing and droplet formation. This results in impingement of water at high speeds into the engine bay, even if the vehicle drives through relatively low depths of water. The vehicle cannot drive through deep water at these speeds. For this case, the components that get affected are often not submerged in water, but instead encounter water impingement from small gaps and openings, at high speeds. This is more frequently encountered by vehicles in general [1], and the severity of impact depends on the actual vehicle speed and water depth.



Figure 3. A Range Rover Evoque wading through water trough at a 50kph at Millbrook test facility. High amount of splashing is observed.

The above two scenarios can be classified into cases of deep water wading and splash wading respectively. The severity of damage caused to the vehicle varies based on a number of factors, such as the vehicle design and structure, splash protection provided and the relative positioning of the vulnerable electrical components [1].

Even in cases of deep water wading, where the affected components are submerged, the relative movement of the components and accessories (such as the rotation of the FEAD pulleys and the movement of the belt) further propels and redirects water to other components in their vicinity, resulting in indirect impingement of water [1]. The D&R testing team has observed that the way in which the component is affected due to wetting varies depending on the component location and function. The underbody plastic trim components often undergo structural failure due to high pressure load of impinging water. Whereas, the electrical components located in the engine bay are likely to accumulate mud or salt that impinges along with water, over a period of time and eventually fail.

VEHICLE WADING: CAE ANALYSIS AND TESTING

Wading CAE serves a crucial role in vehicle development. The data obtained from virtual simulations is postprocessed and scrutinised to understand the modes of failure of underbody components and vulnerability of engine bay components to splash. This drives the improvements in underbody design and splash protection design well before a prototype vehicle is tested.

Physical testing, then, serves as a check for the vehicle from a functionality standpoint in conjunction with underbody robustness and effectiveness against splash. Physical testing provides a limited

understanding of water flow and ingress. The bow wave pattern of the water and/or the splurging around the vehicle is observed through visual inspection and recorded via an externally placed camera. The internal splashing of water in the engine bay is inspected by means of cameras mounted inside the engine bay [1]. The view of the camera gets eventually obscured due to the impinging water. However, physical testing is pivotal with regards to analysing vehicle behaviour, functionality and any unexpected problems, such as electrical component failure, cabin leakage and hydraulic lock.

D&R decided to investigate the ways in which the virtual analysis capabilities could be improved further. It is essential to understand the CFD simulation setup in order to identify the possible areas of development. Khapane et al. [3] have outlined the mesh and physics setup for wading simulations which is applied at JLR.

Table 1. Peculiar aspects of the CFD simulation setup, as described by Khapane et al. [3]

Parameter	Details
Eulerian Multiphase model	Volume Of Fluid (VOF)
Eulerian phases	Water & Air
Multiphase interaction model	VOF-VOF Phase interaction
Turbulence Model	SST (Menter) K-Omega
Wall Treatment	All y+ wall treatment

The physics setup described here is ideally suited for a low to medium speed vehicle wading assessment. The distance travelled by the vehicle per unit time is less; hence the use of a comparatively larger time step is justified. The VOF multiphase model is suited for simulations of flows where each phase constitutes a large structure, with a relatively small total contact area between phases. Hence, it is ideal for capturing a relatively smooth free surface of water. However, splash wading results in formation of water jets, large numbers of air bubbles in the water, and water droplets in the air. The method would then require a fine mesh (at least three cells across each droplet/bubble) to produce negligible modelling errors. [6]. CFD Modelling of the vehicle wading at very low depths is very complex to capture owing to the constraint of mesh size. On the other hand, deep water wading phenomena can be captured with the current meshing setup [3] since there isn't much splashing and droplet formation. As a result, the Courant–Friedrichs–Lewy (CFL) condition can be met for wading simulations at a low vehicle speed; and the simulation is stable [6].

In case of medium to high vehicle speeds (30kph and above), the default physics and mesh setup prove to be insufficient in order to correctly capture the splash and water impingement. This is attributed to the factors enlisted below:

- The vehicle travels a greater distance per unit time, and as a result, the default time step proves to be too big.
- The high amount of splashing cannot be captured with the default VOF multiphase model and coarser mesh.

- The wheel rotation modelling approach does not simulate an appropriate hydraulic jump and water jet motion, and as a result, the subsequent water ingress is under predicted.
- Satisfying the CFL criteria for high vehicle speed simulations is computationally expensive. Secondly, simulations that run with high CFL numbers are relatively unstable.

D&R tested a JLR sports car prototype (Test Vehicle 1) with an aim of understanding water impingement in the engine bay. Visual inspections highlighted the alternator as a critical component at risk of damage due to splashing over a period of time.

The alternator is highlighted in Figure 4. The gap between the vehicle subframe and the splash shield (red) was identified as the opening most likely to contribute to the ingress. The testing team wrapped the alternator with a porous jacket lining and measured the amount of water collected in the jacket after each test. A number of tests were conducted, with varying vehicle speeds and water depths (standardised as a wading matrix at JLR). Tests were conducted for low water depths (50mm and 100mm) and medium to high vehicle speeds (30kph, 50kph and 65kph). As expected, high vehicle speed runs showed a greater splash in the engine bay compared to medium speed.

The CAE team carried out a number of simulations to test how the output compares with the testing data. For high vehicle speed simulations, the fine droplet breakup was not resolved, and hence the amount of water impinging on the alternator didn't correlate.

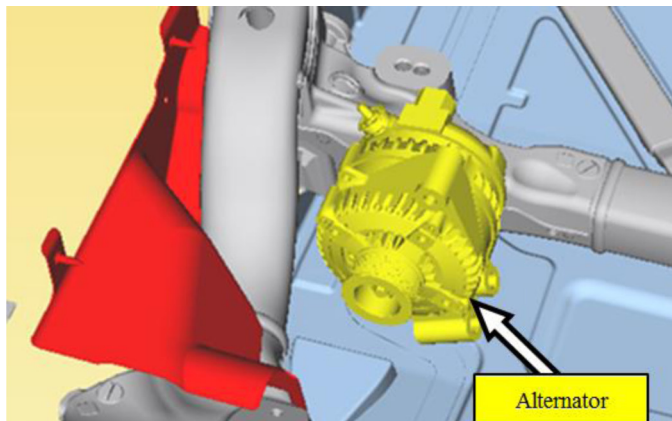


Figure 4. Above: CAD model shows the alternator and the vehicle structure for Test Vehicle 1 along with the splash shield.

There were additional issues with regards to quantifying the water retained by the porous jacket. The inertial and viscous porous resistances of the jacket were not known. However, the data collected from the physical tests was still useful in terms of obtaining ballpark values of fluid impingement. The CAE team ran a number of simulations by varying the porous resistance properties. The water absorption and retention data was recorded and scrutinised. For lower resistance values, the amount of water absorbed was higher; however, the amount of water retained by the sponge at the end of the simulation was low; and vice versa. After the initial CFD analysis, the most practical resistance values were chosen for the full vehicle assessment. Figure 5 shows an instance from the simulation with the alternator and the porous jacket modelled around it.

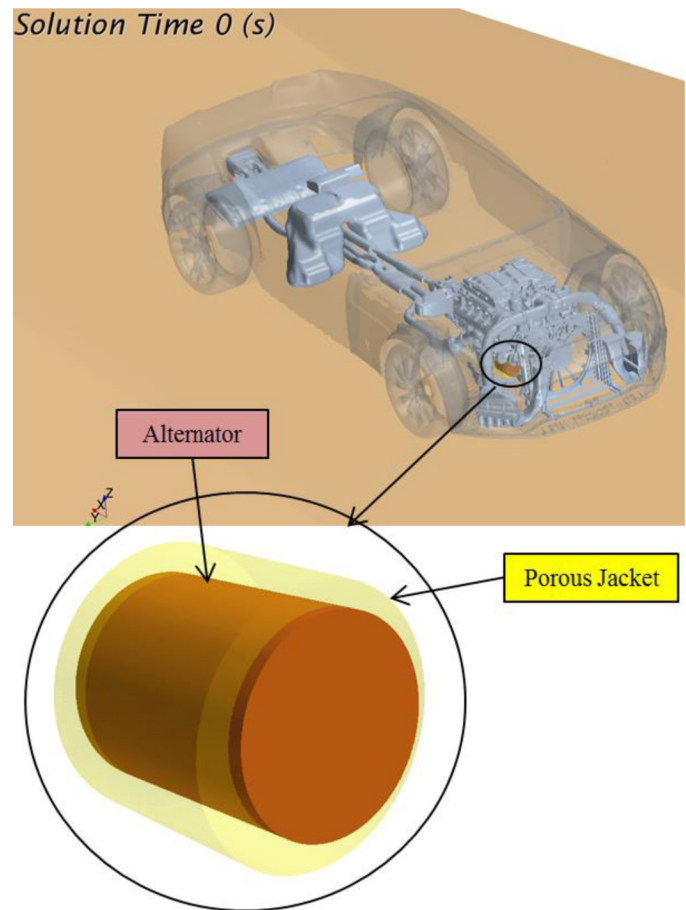


Figure 5. A scene from the CFD simulation with a view of the Engine Bay of Test Vehicle 1, showing the alternator surrounded by a porous jacket.

This test was useful for observing the water impingement and splurging phenomena in general. CFD assessment conducted for replicating the test showed potential areas for improvement with regards to splash protection evaluation and water ingress analysis, which have been discussed in the subsequent sections.

SPLASH PREDICTION USING CFD

The splash and water ingress into the engine bay occurs in the form of sharp jets and droplets. To capture these jets in the wading CFD simulation using VOF-VOF Eulerian multiphase interaction between water and air, would imply the use of a refined volume mesh of high cell quality [6]. Thus, the CPU solver time of the simulation would increase substantially. Simulations carried out for vehicle wading show that in order to accurately capture the impingement, volumetric refinement in the entire splash zone need to be approx. half (1.25-2.5mm) of the current cell size. Correspondingly, volumetric cell count for vehicle region increases 2-5 times and requires a smaller time marching for attaining simulation stability and solution accuracy. [8]. Thus, the CPU run time of the simulation increases to around 4-6 times that of a similar case for ingress wading simulation. This was deemed impractical for industrial virtual wading applications, owing to the strenuous focus on cost saving and quick turnaround time.