

Development of Prediction Method for Engine Compartment Water Level by Using Coupled Multibody and Fluid Dynamics

Yoshiteru Tanaka, Jun Yamamura, Atsushi Murakawa, Hiroshi Tanaka, and Tsuyoshi Yasuki
Toyota Motor Corporation

ABSTRACT

When vehicles run on the flooded road, water enters to the engine compartment and sometimes reaches the position of the air intake duct and electrical parts and causes the reliability problems. Numerical simulation is an effective tool for this phenomenon because it can not only evaluate the water level before experiment but also identify the intrusion route. Recently, the gap around the engine cooling modules tends to become smaller and the undercover tends to become bigger than before in order to enhance the vehicle performance (e.g., aerodynamics, exterior noise). Leakage tightness around the engine compartment becomes higher and causes an increase of the buoyancy force from the water. Therefore the vehicle attitude change is causing a greater impact on the water level. This paper describes the development of a water level prediction method in engine compartment while running on the flooded road by using the coupled multibody and fluid dynamics. MPS (Moving Particle Semi-implicit) method was used to analyze the free surface flow and 3-dimensional multibody dynamics analysis was applied to calculate the suspension displacement due to the hydrodynamic force. As a result, the accuracy of the prediction method was within $\pm 5\%$ and the time history of the water level and suspension displacement showed a good agreement with the experiment. And the results showed that the phenomenon of the water rising was categorized into 2 patterns depending on the vehicle speed and water depth. Through the discussion, the importance of considering the vehicle attitude is indicated and it is concluded that the running conditions could be categorized by regarding the force acting on the vehicle when running on the flooded road.

CITATION: Tanaka, Y., Yamamura, J., Murakawa, A., Tanaka, H. et al., "Development of Prediction Method for Engine Compartment Water Level by Using Coupled Multibody and Fluid Dynamics," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1328.

INTRODUCTION

The automobile is expected to be reliable in the various environments and one of these environments is the flooded road. When vehicles run on the flooded road as shown in [Figure 1](#), water enters to the engine compartment and sometimes reaches to the position of air intake duct and electrical parts and causes reliability problems. The performance when running on the flooded road affects the engine compartment design and it is important to reflect in the vehicle design in the early stage of the new car development. Numerical simulation is an effective tool for this field, because it can not only evaluate the water level before experiment but also identify the intrusion route and understand the phenomenon of water entry and derive the countermeasures.

Yamamura et al. developed the simulation method to predict the water level around the air intake area using CFD and identified the factors determining the water level using multiple regression analysis [1]. This research assumed that the vehicle was fixed in a stationary position and the water flowed at a rate equivalent to the vehicle's normal traveling speed.

Recently, the gap around the engine cooling modules tended to be smaller and the undercover tended to be bigger than before in order to enhance the aerodynamics and exterior noise performance. Leakage tightness around the engine compartment becomes higher and causes an increase of the buoyancy force from the water to the vehicle when running on the flooded road. Therefore the vehicle attitude change was causing a greater impact on the water level. Furthermore, it was more difficult to determine the condition where the water level became the highest within the expected performance.

This paper describes the establishment of the simulation technique to predict the water level in the engine compartment considering the vehicle motion due to the force from the water. The cases where the vehicle speed, water depth and the suspension modeling varied are conducted and the water rising mechanism is discussed.



Figure 1. Situation when the vehicle runs on the flooded road.

SIMULATION METHOD

Numerical Methodology

In this study, coupled multibody and fluid dynamics method was developed to predict the water level around the vehicle.

The water flow around the vehicle was simulated and the buoyancy force from the water to the moving vehicle was calculated by the fluid dynamics analysis. Its force was sent to the multibody dynamics analysis and the vehicle attitude was calculated. Furthermore the vehicle attitude was fed back to the fluid dynamics analysis. In other words, the fluid dynamics analysis and the multibody dynamics analysis were strongly coupled.

Fluid Dynamics Analysis Method

The MPS (Moving Particle Semi-implicit) method was used to simulate the free surface flow. The MPS method is the incompressible flow methodology with free surfaces [5] and mesh-free Lagrangian method. In this method fluids are represented by particles and fluid behaviors are calculated as the movement of particles. Therefore grids are not necessary for this method. This point was very beneficial because it was not necessary to recalculate the simulation meshes in accordance with the vehicle attitude and contributed to reducing the simulation time.

The accuracy of the MPS method for the free surface flow has already been confirmed through application to breaking waves problem [6].

Governing equations are expressed by using mass continuity [equation \(1\)](#) and Navier-Stokes [equations \(2\)](#).

$$\frac{\partial \rho}{\partial t} = 0 \quad (1)$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \mathbf{u} + \mathbf{g} + \nabla \phi \quad (2)$$

where ρ denotes the density, t the time, $\mathbf{u}$ the velocity P the pressure, ν the kinematic viscosity, $\mathbf{g}$ the gravity force and ϕ surface tension potential. Governing equations are discretized by particle interaction models developed by Koshizuka et al. In particle interaction models, a weight function ([equation \(3\)](#)) is used to calculate the interactions with its neighbors.

$$w(r) = \begin{cases} \frac{r_e}{r} - 1 & (r < r_e) \\ 0 & (r \geq r_e) \end{cases} \quad (3)$$

Interactions act just only to the particles where the distance is within a finite distance r_e . Potential model is used to calculate the surface tension [7].

Numerical setup and property setup for the above governing equations are shown in [Table 1](#) and [Table 2](#). Particle size was determined based on the convergence check of the simulation results.

Table 1. Numerical setup for the MPS method

Particle Size (Initial Distance of particles)	8mm
r_e (Influence radius)	16.8mm

Table 2. Property setup (Characteristics of water)

Density	1000 kg/m ³
Kinematic viscosity coefficient	1.0×10^{-6} m ² /s
Surface tension coefficient	0.072 N/m
Contact Angle	90 deg

In this research, fluid dynamics analysis was performed using Particleworks Ver. 5.1 from Prometech Software, Inc..

As a preliminary calculation of this research, this MPS method was applied to the water traveling problem of the simplified vehicle shown in [Figure 2](#) (vehicle model) and [Figure 3](#) (test region). Four cases in which vehicle initial velocity and water depth were changed were conducted. Numerical setup for this calculation was different from [Table 1](#) because the size of simplified vehicle is much smaller than that of full vehicle. Particle size was set to 3mm and influence radius was set to 6.3mm. [Figure 4](#) shows the comparison of the maximum water level in front of the simplified vehicle between experiment and simulation. Water level was defined as the height from the slope road. From [Figure 4](#), simulation results showed a good agreement with experimental results. Accuracy was within $\pm 5\%$. Through these calculations, the validity for vehicle's waterway operation was confirmed in this research.

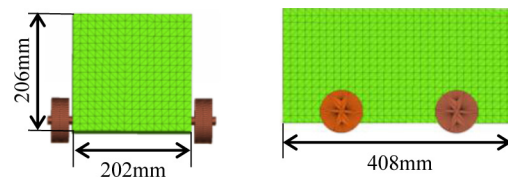


Figure 2. Simplified vehicle

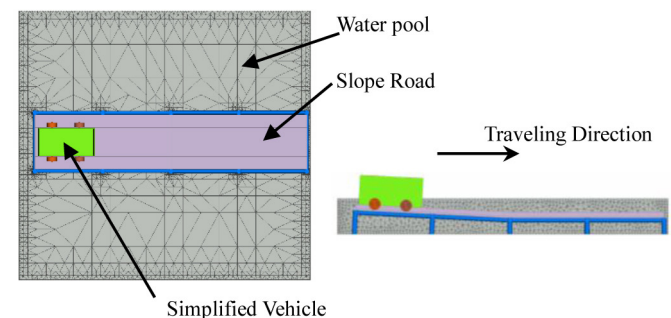


Figure 3. Test region

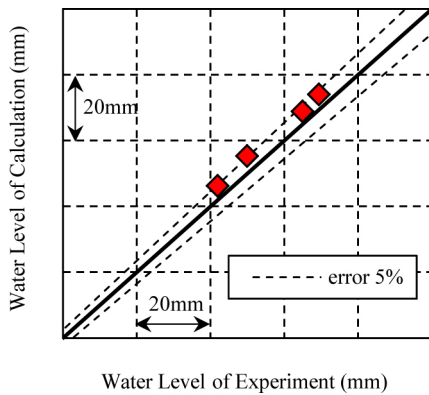


Figure 4. Comparison between experiment and calculation

Multibody Dynamics Analysis Method

Mechanical components are modeled as the rigid bodies and mass properties are assigned to the bodies. Each rigid body is constrained by fixed, revolute and translational joint. External forces are acted on this mechanical system and the movements of the bodies are calculated under the differential equations of motion and algebraic constraint equations [8].

In this research, multibody dynamics analysis was performed using RecurDyn V8R4 from FunctionBay, Inc..

Analysis Geometry

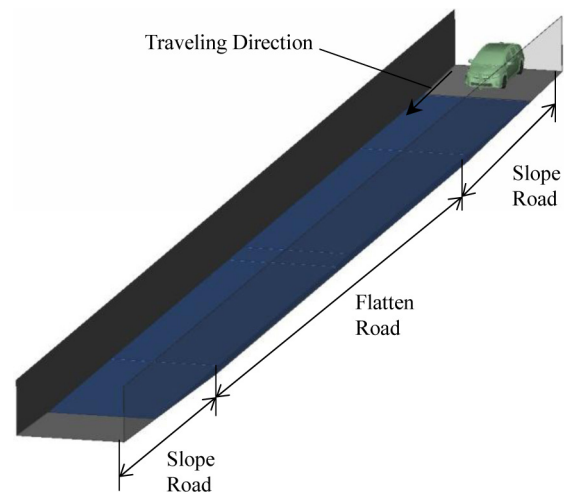
Fluid Dynamics Model

The model of fluid dynamics analysis was composed of analysis region (Figure 5) and vehicle model (Figure 6).

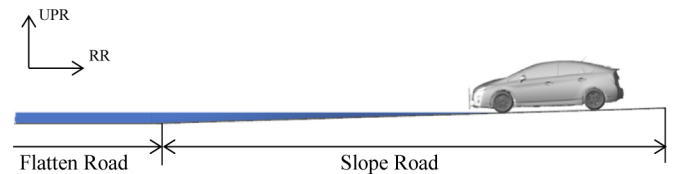
Analysis region was based on the submerged proving ground road at Toyota. Slope angle, width of water path and height of side wall are the same as the experimental booth. The vehicle model was composed of 10 parts, vehicle body, four under-springs, tires and wheels and rear beam. The Modeling of these parts was reproduced only at the exterior surface, similar modeling with aerodynamics analysis. The facial area and shielding status around the heat exchanger, which greatly affected to the water level in the engine compartment, were modeled down to details (Figure 7). Parts were independent of each other and connected through the constraints of multibody dynamics analysis.

The heat exchangers in the compartment were modeled as the volumetric region and passing water resistance was defined as the function of the water velocity and pressure difference (Figure 8).

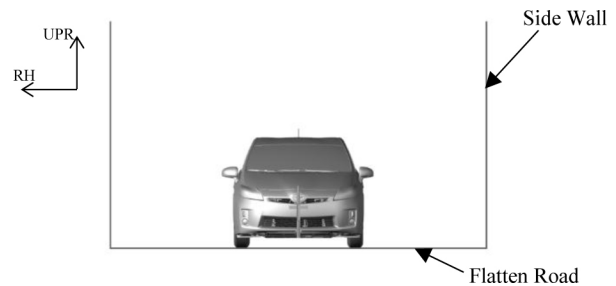
The polygon wall boundary condition were imposed to the wall of all parts. This condition was very useful in terms of reproducing the complex geometry accurately and different from the conventional condition which use wall particles.



(a). Overall view

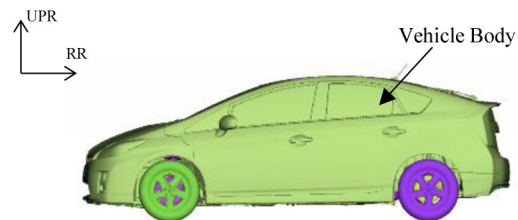


(b). Side View



(c). Front View at Flatten Road

Figure 5. Analysis region



(a). Overall view of vehicle model

Figure 6.

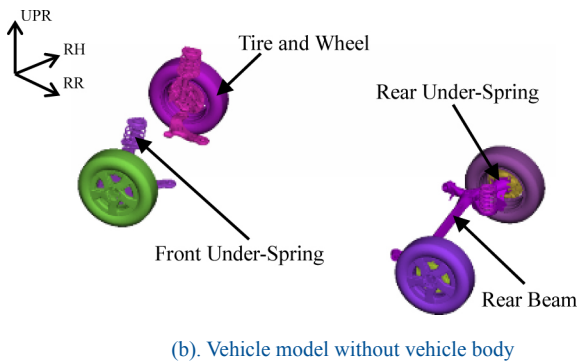


Figure 6 (cont.). Vehicle model

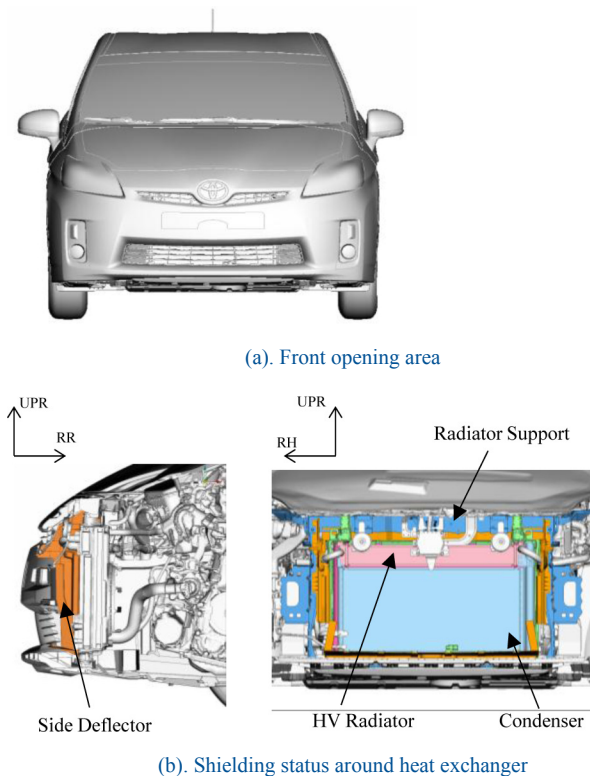


Figure 7. Details of Vehicle

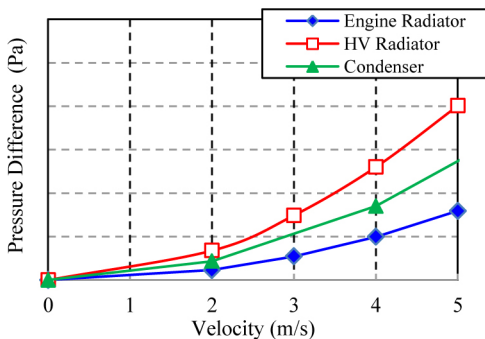


Figure 8. Heat exchanger characteristics for water

Multibody Dynamics Model

Multibody dynamics model shown in Figure 9 was used. This model was composed of 10 parts as well as vehicle model for fluid dynamics analysis. All parts were assumed as the rigid bodies and mass and inertia moment are defined for each rigid body. All parts were constrained with each other.

For front suspension, two under-springs were connected with each tire and wheel by revolute joint and these two under-springs were connected with vehicle body by translational joints. For rear suspension, two under-springs were connected with each tire and wheel by revolute joint and these two under-springs were fixed with rear beam and rear beam was connected with vehicle body by translational joint. Spring and damper coefficients are defined to the four translational joints.

Cylinder to surface contact conditions were imposed between tires and ground in order to reduce the numerical fluctuations. Rolling resistance coefficient was imposed to the surface of tires to calculate the rolling resistance.

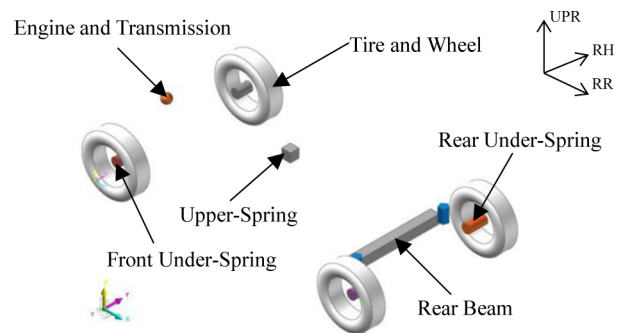


Figure 9. Multibody dynamics model

SIMULATION CONDITIONS

Simulation Cases

Table 3 shows simulation cases. Eleven cases were simulated. In these cases, vehicle speed, water depth and suspension model were changed in order to validate the developed method, compare the water level depending on vehicle conditions and clarify the necessity to use coupled multibody and fluid dynamics analysis.

Two suspension models, flexible model and rigid model, were compared. Flexible model was the developed method in this research in order to reproduce the vehicle attitude and rigid model had the vehicle attitude fixed in the stationary state. In other words, the vehicle could not move in the vertical direction even if the buoyancy force from the water acted on the vehicle.