Analysis of unsteady aerodynamic on road vehicles drivability.

Javier SANCHEZ RIOS
Candidate for the Degree of Master of Engineering
Supervisor: Nobuyuki OSHIMA
Division of Mechanical and Space Engineering.

Abstract

Nowadays the Computational Fluid dynamics is being used widely on the automotive industry at different fields and applications on the vehicle development. This close relationship between design and simulation seems to be more notorious due to continuing advances in computer hardware and software as well as advances in the numerical techniques to solve the equations of fluid flow, resulting in more realistic simulation. The vehicle aerodynamics is one of the fields were the CFD is commonly used when designing a vehicle.

The main advantages of the CFD is the ability to simulate the flow going through the vehicle and be able to view and measure on detail the flow structures action on the vehicle at any desire coordinate without any physical model. Also by using CFD, running conditions or specific traffic situations can be simulated.

Another factor that companies started to pay more attention on recent years is the driving comfort or drivability on their vehicles. On way that companies evaluate the drivability is through the lift fluctuations and the pitching oscillations. There are multiple factors that may affect the drivability of the vehicles; road conditions, suspension set up, maneuvering and fluctuation acting from the surface.

Through experimentation it has been found that the strong low frequency, between the range of 0.1 Hz and 4 Hz, aerodynamic fluctuations contribute negatively to the drivability on the automobile. Higher frequency oscillations are contributed to the mechanical oscillations, such as the tires and vehicle motion.

But because the RANS simulations cannot capture the unsteady aerodynamics, this performance is studied by experimentation. From here the importance of simulate unsteady aerodynamics on the vehicles.

The unsteady simulation approach has been done primordially by Universities and Research centers that can develop, validate and improve specialized numerical code that should produce satisfactory results.

As was established before, the automotive companies are always optimizing and improving their designs to get overall better performance. One example of this constant evolution is the prototype design proposed by Honda Motor Company. This new design changes the body geometry around the rear tire area. The physical vehicle was tested by Honda Motor Company at the Sakura wind tunnel.

The results exhibited by the wind tunnel experimentation displayed a better performance than the original parts. Now, it is important to identify what is exactly changing on the flow field that may be causing this different performance. Specifically, for the driving comfort, because the sedan car is widely used by the general public so it is expected to have a smooth drive.

Despite that the Sakura Wind tunnel is a Full-Scale Wind Tunnel with a moving belt incorporated to the test. The advantage of a Full-Scale Wind tunnel and the moving belt are that this features allow to measure more accurate data. However, is virtually impossible or really complicated to identify the flow structures in detail using only the wind tunnel results. In addition to that, as mentioned before the RANS simulation are not able to capture the fluctuations on the vehicle. Hence, an unsteady simulation may be able to identify the flow mechanism causing this difference on both vehicles.

The present study, in collaboration with Honda Motor Company, is a comparison between the Baseline model and a new geometry on the vehicle side sill and lateral rear bumper of a sedan-type vehicle. The experimental results showed a better performance from the new design, but by the wind tunnel results was not possible to observe how the flow differs between both options.

Therefore, the present study aims to achieve the following objectives:
1. To investigate how the fluctuation is different between the vehicles and if this fluctuation may be affecting the drivability.
2. To identify the physical mechanism of how the unsteady aerodynamics, influence the drivability.

Therefore, the experimental set up was simulated in three yaw angles; 0, 3, and 5 degrees, for both vehicle configurations Baseline and the Aerodynamic proposal. To capture the unsteady aerodynamics the numerical method for time-resolved flow simulation is the Large Eddy Simulation (LES) method, applied through the in house developed software Front Flow Red (FFR).

Figure 1- Rear bumper and side sill views of both vehicles.
Beginning with the time average results we observed how the flow develops on both designs and the differences on the flow field. From the time average results the main flow differences were identified. It was clear that the Front pack on Aero option plays a critical role by changing the flow distribution on the vehicle. This part presented a different pressure distribution and new vorticity structures. Also the new shape on the rear bumper also contributed to smooth the flow on the rear part of the vehicle. However, the difference seemed slightly and it is hard to identify a specific mechanism responsible of the fluctuations affecting the drivability. Furthermore, is not possible to analyze fluctuations by using only time average results. Therefore, the unsteady aerodynamic results were analyzed.

The first approach on the analysis of the lift fluctuations was the visualization of the pressure RMS on the vehicle surface and the flow field. By observing the RMS distribution on the vehicle it was found that the Baseline option had its fluctuations concentrated on the edge of the rear bumper and the Aero option not only showed lower values of RMS but also a wider distribution on the surface of the car. On the flow field the Aero option showed a flow field with less fluctuation but similar distribution as the Baseline option.

To know if this fluctuations may affect the drivability, the PSD was performed on the lift forces. The frequency spectral analysis showed us some strong fluctuations close to the ~0.4 Hz frequency.

The DMD was applied to the pressure and velocity magnitude. The DMD results on the 0.4 Hz mode revealed fluctuation distribution very similar to the one found on the POD modes. This being the Baseline strongly concentrating the fluctuation on the rear bumper area and the Aero option distributing the fluctuation on all the rear vehicle surface. Finally, the mechanism causing the differences was proposed. It is believed that the Front pack on the Aero option by separating the flow from the tire it contributes to a drag and lift reduction which allows to a better distribution, as seem on the modal analysis, the energy does not concentrate on specific parts of the vehicle. The new shape on the Aero option rear bumper also dissipates the vorticity better than the straight shape used on the Baseline option. This new concave shape allows the flow to be better distributed on the rear vehicle surface.

References.

