MULTIPHASE WATER FLOW SIMULATION OF A VEHICLE'S ROOF

Filipe Fabian Buscariolo[1], Attila Budavari[2], Daniela Mantovani[2], Eduardo Almeida[2], Gilvan Rossi[2], Roberto Bigarella[2], Roberto Pereira Ramos[3]

[1] GM Brazil / NDF-USP

[2] GM Brazil

[3] GM North America

E-mail: filipe.buscariolo@gm.com

ABSTRACT

Considering the increment of computational power and the accuracy on the results, virtual engineering tools are becoming more popular among the industries, especially inside the automotive, seeking development time and cost reduction. Taking advantage of the modern resources, it was developed a simulation methodology in order to verify the water flow behavior from the roof to the side ditch of the vehicle's roof. Within this methodology it is possible to virtually test a vehicle without a real prototype, analyze the roof performance and suggest design changes without any prototype part being made, which implies in cost and development time reduction.

Keywords: CFD (computational simulation), Multiphase Simulation, Water Management..

INTRODUCTION

Considering the growth of computational power, CFD simulations became a good solution to predict performance in a fluid system. This fact also made possible to perform more complex simulations in a production response time, such as multiphase fluid simulations, involving two or more fluids.

Considering the BUSCARIOLO and VOLPE (2014) [1] work, basically, multiphase CFD simulations can be divided into two groups:

- Reactive: fluids interact chemically or physically to generate new products and properties changes. Example: combustion.

- Non-Reactive: two or more fluids are present in the computational domain, however they do not interact chemically or physically. Example: water ingestion or management simulations.

This paper aims to contribute with the growing trend of using multiphase simulations in the industry. Considering a rainy condition over a vehicle, the roof's ditch tends to conduct water off from the side windows and avoid the occupants to get wet when opening the door or the side windows.

The main purpose is, by applying computational simulation correlated with a physical test, propose a methodology to identify water path on the vehicle's roof ditch and identify any leakage points due to rain before any prototype is built, aiming to improve the vehicle's development cost and time.

OBJECTIVES

As mentioned before, the present work was based on real condition of a car exposed to certain rain conditions in which the vehicle's roof ditch should be able to conduct water off to the plenum without any leakage to the side windows, which may cause the occupants to get wet when opening doors or side windows.

In order to reproduce the rain condition over the vehicle's roof, a rain simulation device which conducts water and can uniformly distribute over the length of one door was adopted and is shown on Figure 1.

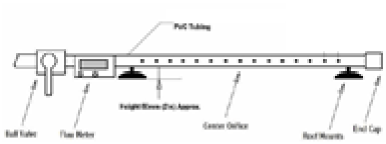

Figure 1. Rain simulation device.

The rain simulation device is assembled over the vehicle's roof, pointed to one of the doors, if it is a four doors car or to one of the doors, if it is a coupe car, then it drops water over and its path can be tracked to analyze the ditch performance due to this rain flow. If there's no leakage and the ditch can conduct all water, the vehicle's roof and ditch have both a good design. If water overpasses the ditch and reach the side window, roof and ditch designs should be improved to meet the criteria. A picture of the rain simulation device assembled over the vehicle is shown on Figure 2.

Figure 2. Rain simulation device assembled on a vehicle

The test mentioned before can only be performed if a physical model is available or for validation and by that time, the project is in an advanced stage in which design changes will cost more than changes executed at early stages of the projects.

Considering the points mentioned above, this paper’s main objective is to propose a multiphase simulation methodology, correlated with a physical test, in which the water path over the vehicle’s roof to the ditch can be identified and propose some design changes in order to avoid leakage points at early stages of the project.

Two studies will be shown: first one will be a correlation study and second one will be a comparison between simulation and physical test, in which the virtual CFD case was performed before, in order to confirm simulation confidence level.

METHODOLOGY

The proposed methodology in this work is based on BUSCARIOLO and VOLPE (2014) [1] work, using correlation of the physical test with proposed multiphase CFD simulation for water path over the vehicle’s roof until the roof ditch.

In order to reproduce the rain configuration, a virtual rain simulation device was assembled over a virtual model, considering the same water flow and position of the real rain simulation device. In order to save computational time and have a faster response, just one upper side of the car was considered, once the focus of the simulation is the vehicle’s roof ditch water flow. In this work, only the device pointed to the front door will be tested for both physical and CFD tests. A schematic virtual assembly of the virtual device is shown on Figure 3.

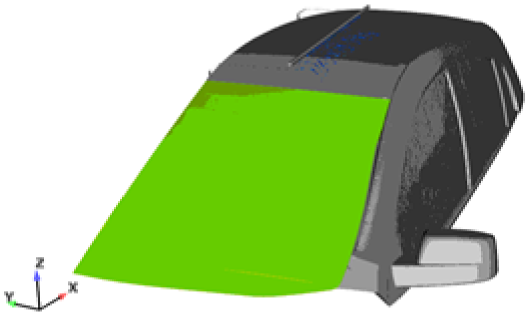

Figure 3.Virtual rain simulation device assembled on a virtual vehicle

A virtual wind tunnel was assembled over the upper side portion of the car in order to make possible the water flow calculation. Tunnel dimensions should be enough to fit the model inside and have good mesh quality. A representation of the model is shown on Figure 4.

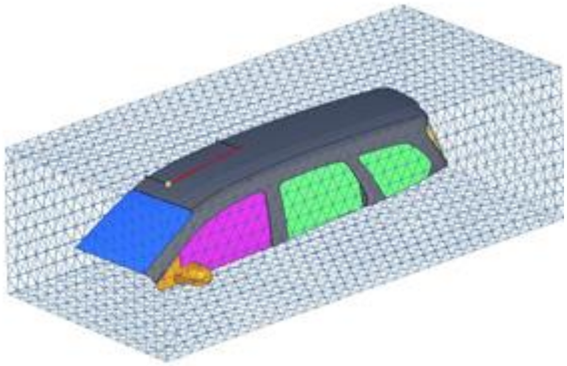

Figure 4.Virtual model assembly

CFD simulations were performed on a 68cpus cluster with total simulation time of 12 seconds in order to have constant flow to identify possible leakage on a 14 Million volumetric cells model.

Model Additional Considerations

Based on BUSCARIOLO and VOLPE (2014) [1] work, for the model configuration of the multiphase, it was considered the Volume of Fluid (VOF) method which is a numerical technique for tracking and locating the free surface. It belongs to the class of Eulerian methods which are characterized by a mesh that is either stationary or is moving in a certain prescribed manner to accommodate the evolving shape of the interface.

VOF is an advection scheme—a numerical recipe that allows the programmer to track the shape and position of the interface, but it is not a standalone flow solving algorithm. The Navier–Stokes equations describing the motion of the flow have to be solved separately. [3]

The turbulence model considered was the K-epsilon (k-ε) which is the most common model used in Computational Fluid Dynamics (CFD) to simulate turbulent conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k). The second transported variable is the turbulent dissipation (epsilon) which determines the rate of dissipation of the turbulent kinetic energy. In this case, swirl was considered in order to represent better highly perturbed flow.

The simulation was set as a transient with a fixed time step lower enough to the simulation reach convergence and represent the flow path.

RESULTS

First Comparison: physical test vs CFD proposed simulation methodology

In this case a comparison between the physical test in which the simulation methodology was based and the virtual results are shown and presented good correlation level. Figure 5 shows physical test result and Figure 6 shows CFD simulation results.

Figure 5.Physical test model result

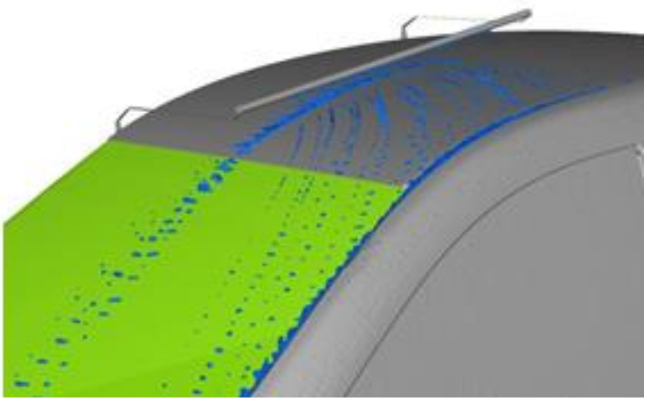

Figure 6.Virtual test model result

In this first comparison, physical test predicted no leakage on the test as shown on Figure 5 above. CFD multiphase simulation methodology also didn't find any leakage points during the simulation time, indicating correlation with physical test.

Second Comparison: CFD proposed simulation methodology vs physical test

In this second case a virtual model applying the proposed simulation methodology presented on the first comparison will be compared with a physical test. The key point here is that for the second comparison, virtual test was performed before the physical test, in order to evaluate the confidence level of the methodology. Figure 7 shows CFD simulation result and Figure 8 shows physical test result.

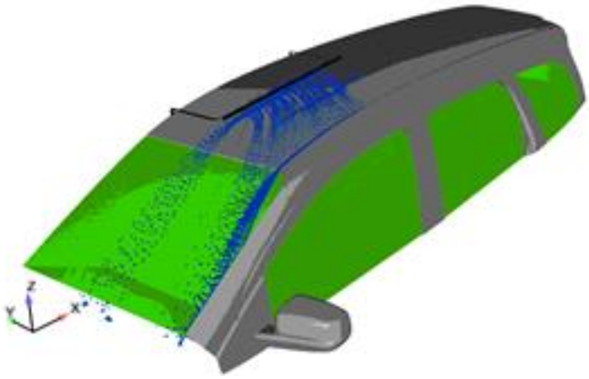

Figure 7.Virtual test model result

Figure 8. Physical test model result

In the second comparison, CFD multiphase simulation methodology predicted no leakage on the test as shown on Figure 7 above. Physical test also didn't find any leakage points during the test time, indicating correlation and indicating high confidence level of the simulation methodology.

CONCLUSION

The Industry and Automakers are always seeking for solutions to improve vehicles development cost and time. The use of computational simulations is becoming a common practice inside the industries nowadays. Considering the hardware and software improvements among the years, the complexity level of the simulations is also growing.

The use of CFD simulations are also increasing among automakers in order to improve vehicle's development time and also save cost with physical tests.

Nowadays, CFD codes have more powerful algorithms and, together with the increment of the number of processors, multiphase analysis, which are fluid simulations that consider two or more fluids in the same domain, become a nice solution and in a reliable production time.

In the present work, the main objective proposed a multiphase simulation methodology, correlated with a physical test, in which the water path over the vehicle's roof to the ditch can be identified and propose some design changes in order to avoid leakage points at early stages of the project.

Model was set considering VOF approach and K-epsilon turbulence model. Total simulated time was 12 seconds in a transient condition on a 68 cpus cluster.

The methodology was capable to reproduce the water path from the vehicle's roof to the ditch, as shown before, with a good correlation with physical test. This fact is really important once this approach can be used at early stages of the project to evaluate part performance and also save money by improving the design and performance of certain parts before any physical test.

First Comparison Conclusions

The proposed methodology was based on the test shown on the first comparison. The rain simulation device was assembled in the same position for both test and simulation and virtual parameters were set in order to reproduce the same effect.

Results shown correlation in which no ditch leakage point was found in both physical test or in the simulation. This step indicated a good start for the simulation methodology and it was tested again on second comparison.

Second Comparison Conclusions

For the second comparison, methodology was tested before the physical test was performed, in order to evaluate the confidence level of it.

Considering the simulation results, no leakage point was found, during total virtual test time. Afterwards, physical test was performed at the same condition and also did not find any leakage points.

This fact confirms the high confidence level of the proposed methodology which can be applies to projects at early stages of the projects.

For further works, a comparison, considering the same basis can be performed to the rear door as shown on Figure 9.

Figure 9. Rain simulation device assembled on the rear door

Another complementary work would be the rain simulation device assembled on the deck lid of station wagons, in order to prevent the water flow entering the trunk due to rain or any other wet condition and a correlation with the virtual methodology.

REFERENCES

1. BUSCARIOLO, F.F.; VOLPE, L.J.D., Water Ingestion and Pressure analysis of Automotive Vehicles using Multiphase CFD, SAE Paper, Nº 2014-36-0251, Society of Automotive Engineers, 2014.

2. FLUENT 14.0., User´s Guide, Fluent Inc., 2011.

3. HUCHO, W. H., Aerodynamics of Road Vehicles, From Fluid Mechanics of Vehicle Engineering, 4th Edition, SAE International, 1998.

4. Hypermesh 11.0 User´s manual Altair Computing, Inc., 2011.

5. KELLY, K. B.; PROVENCHER, L. G.; SCHENKEL, F. K., The General Motors Engineering Staff Aerodynamics Laboratory – A full Scale Automotive Wind Tunnel, SAE Paper, Nº 820371, Society of Automotive Engineers, 1982.

6. WHITE, F., Fluid Mechanics – fouth edition, McGraw Hill, 1999