

Re-Design for Automotive Window Seal Considering High Speed Fluid-Structure Interaction

Wenfeng Zhu, Chunyu LI, and Yao Zhong
Tongji University

Peijian Lin
China Ship Development and Design Center

ABSTRACT

Automotive window seal has great influence on NVH (Noise-Vibration-Harshness) performance. The aerodynamic effect on ride comfort has attracted increasing research interest recently. A new method for quantifying and transferring aerodynamics-induced load on window seal re-design is proposed. Firstly, by SST (Shear Stress Transport) turbulence model, external turbulent flow field of full scale automotive is established by solving three-dimensional, steady and incompressible Navier-Stokes equation. With re-exploited mapping algorithm, the aerodynamics pressure on overall auto-body is retrieved and transferred to local glass area to be external loads for seals, thus taking into account the aerodynamics effect of high speed fluid-structure interaction. This method is successfully applied on automotive front window seal design. The re-design header seal decreases the maximum displacements of leeward and windward glass with 9.3% and 34.21%, respectively. The improvement of fitting stability shows the effectiveness this seal re-design considering high-speed fluid-structure interaction.

CITATION: Zhu, W., LI, C., Zhong, Y., and Lin, P., "Re-Design for Automotive Window Seal Considering High Speed Fluid-Structure Interaction," *SAE Int. J. Mater. Manf.* 10(2):2017, doi:10.4271/2017-01-9452.

1. INTRODUCTION

Automotive window seals are embedded between metallic DIW (Door in White) panels and nonmetallic door glass, including header, front run-channel, rear run-channel and belt-line, shown in Fig.1. Header seal provides insulation for the top edge of glass when the glass is in fully up position. The window glass moves up and down through the space between the inner seal and outer seal of belt-line, while its front and rear edges embed within the run-channels [1]. It needs not only allow door glass to move smoothly and steadily, but also insulate water and noise.

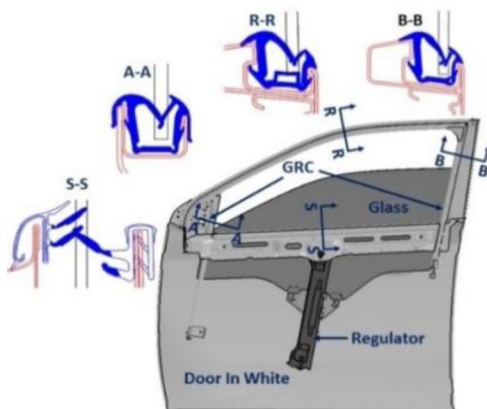


Fig. 1. Illustration of automotive window seals

Being intimately associated with human sensibility, windows seals have great influence on NVH performance. To satisfy human factors and ergonomics, its general design trend is mainly oriented on the isolation of noise and vibration [2,3,4]. A. Stenti et al studied seals low-frequency dynamic analysis by non-linear FE model, pointing out important performance factor of seal design for NVH applications [5,6, 7]. Julia A. Cordioli build full non-linear hybrid FE-SEA model to analysis seal geometrical deformation/contact on sound transmission [8, 9, 10]. E. Dikmen et al developed hyper-elastic and viscoelastic models of the weather-strip seal to predict dynamic performance [11, 12].

After reducing main noise sources and transmission paths to the interior, those once thought as secondary began showing significant contributions. Seal noise generation is estimated by models and found that the sound transmission from the exterior to the interior through side-glass windows and seals as the primary aerodynamic noise sources [8, 13]. In cases where seals do not perfectly block the flow through the gaps, as shown in Fig.2, seal wall vibrations are induced by unsteady pressures produced from the turbulent flow over the automotive surface [9, 10]. In recent years, with weight-lighting and highway network increasing, automotive is frequently driven under higher speed condition. By unsteady pressures produced by turbulent flows over the auto-body, external aerodynamic load leads to vibration, greatly increasing the interior noise level [14].

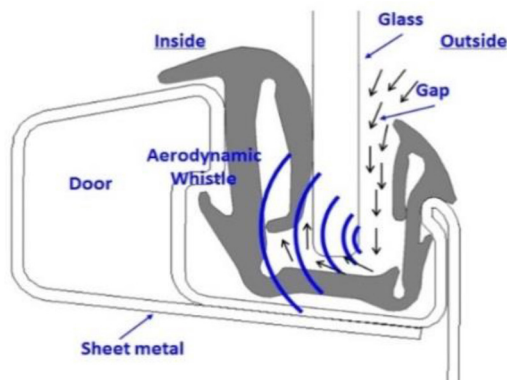


Fig. 2. Aerodynamic noise generated by seal-glass gap

When evaluating the aerodynamic performance, wind-tunnel measurements are often used. F. Cheli et al investigated a high-sided lorry in flat ground scenario and measured mean aerodynamic forces and moments by means of a six-component dynamometric balance [15]. Andreas Kremheller et al used three different kinds of wind tunnel testing on the aerodynamics development process of the new Nissan Qashqai [16]. Proved by many publications [17, 18, 19], CFD is feasible to quantify aerodynamic effect by capturing actual fluid flow with respect to time and cost efficiency.

It is valuable and interesting to re-design window seal aiming to reduce aerodynamics sound transmission. Recently, numerical investigation is proposed for automotive window sealing design with respect to optimize sealing performance [16, 18, 20, 21, 22]. However, major difficulties still exist in quantification of overall aerodynamic effect, especially for its transformation to local window seal. In this paper, a method for automotive window seal re-design is presented considering high speed fluid-structure interaction.

The structure of this paper is organized as the following. [Section 1](#) presents introduction on this research's background. [Section 2](#) is about the CFD modeling to quantify additional external dynamics load. SST (Shear Stress Transport) turbulence model is established and three-dimensional, steady and incompressible N-S equations are solved to obtain external flow field of full scale automotive. [Section 3](#) presents re-exploited mapping algorithm, addressing the problem of load transfer. The lateral aerodynamics pressure on overall auto-body is retrieved and transferred to local window area as external loads for seals. Finally, an automotive front window seal re-design is used as case study in [Section 4](#). The fitting stability improvement shows the effectiveness of seal re-design considering the new high-speed fluid-structure interaction.

2. CFD FOR EXTERNAL AERODYNAMICS FLOW FIELD

Door glass is flexibly constrained inside window seals along perimeter. Because of viscoelasticity and low strength, the door-seal fitting status could be easily impaired by unexpected exterior load. High speed fluid-structure interaction induces vibrations, causing varied gaps between holding seals and glass. However, it is unfortunately neglected in previous window sealing design. To further improve NVH performance, the full-scale automotive CFD

model is constructed. Through solving three-dimensional, steady and incompressible Navier-Stokes equations, the external aerodynamics flow field is achieved.

2.1. Computational Model and Boundary Condition

[Fig.3](#) presents the computational domain used in the CFD of a realistic mass-production automotive type. Since only auto-body external flow field and exterior force is interested, automotive under-body areas, such as the fuel tank and rear suspension, et al, are simplified and replaced by a bottom plane.

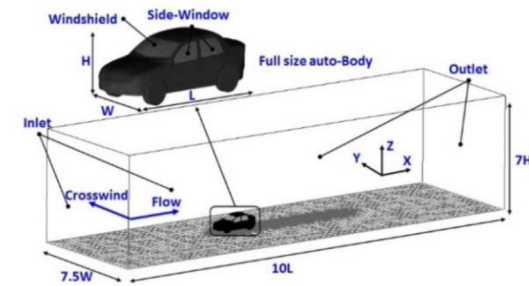


Fig. 3. CFD computational domain set up

Tab. 1. Geometrical dimensions of the car model in CFD

Characteristic	Size
Length. L	4m
Width. W	1.6m
Height. H	1.45m
Ground clearance	0.26m

The automotive body CAD model is built by using CATIA software. Its geometrical dimension is shown in [Tab.1](#). It is then imported to mesh generation module to generate simulation mesh. To achieve both higher calculation accuracy and efficiency, the domain should be meshed and simulated closing to the three-dimensional streaming flow inside the infinite space using the least time. The length, width and height of the computational domain are 10L, 7.5W and 7H, where L, W and H are auto-body overall length, width and height, respectively. The distances to the upstream and downstream boundaries are 3.5L and 5.5L.

2.2. Mesh Generation and Boundary Conditions

2.2.1. Mesh Generation

To achieve higher calculation precision, the Delaunay triangle method is adopted to generate unstructured mesh grid in whole calculation flow field. ICEM software is adopted as mesh generator, which can meet the strict request for mesh partition with auto mesh dense in boundary layer and local mesh dense in rapid change region of flow fluid.

The polyhedral meshes are created with the meshing tool ICEM. For the surfaces where mainly attached flow is expected, a number of prisms layers are created closest to the vehicle surface in order to account for the viscous effects against the vehicle surface [17]. On automotive surface, six prism layers with a height of 5mm are created to resolve the boundary layer. Tetrahedral mesh is mapped and generated on the basis of the triangular surface mesh generation, which finally formed multilayer diamond mesh on the surface and near-surface of the vehicle. Such thick mesh can build unstructured grids in the entire computing space basin to better adapt to the irregular computational area, just as shown in Fig.4. The total mesh number is 1,719,606.

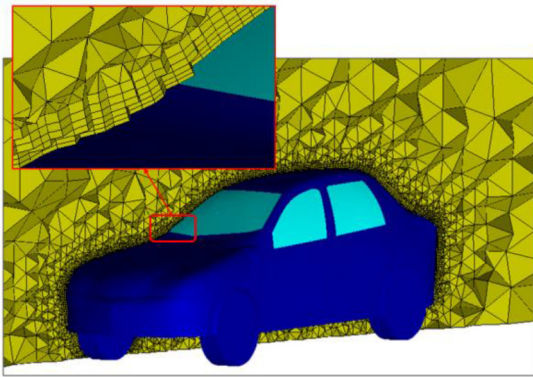


Fig. 4. Full-scale mesh generation and refinery area

Tab. 2. Boundary conditions

Boundaries	types
Inlet & left wall surface	$U = -12.3\text{m/s}$, $V = 27.78\text{m/s}$
Outlet & right wall surface	Non-pressure outlet boundary
Car body surface	Non-slipping wall surface
Ground	Non-slipping wall surface
Automotive roof surface	Free-slipping wall surface

2.2.2. Boundary Condition

Resultant-wind method is adopted for computing external aerodynamics flow field [15, 23]. Inlet velocity condition is a constant velocity, which is the result of square root of automotive speed and a crosswind speed. A speed of 100km/h or about 27.78m/s and a crosswind of 12.3m/s are finally selected and assumed. The outlet condition of the computational domain is a constant pressure with the coefficient of pressure equal to zero (assumed atmospheric pressure) [22]. The boundary conditions are shown in Table.2. All numerical analysis is achieved through ANSYS Workbench.

2.3. Basic Equation and SST Turbulence Model

When running the external aerodynamics simulations, different turbulence models for the RANS-based method are evaluated. The SST model is developed to effectively blend the robust and accurate formulation of the $k - \omega$ model in the near-wall region with the

free-stream independence of the $k - \epsilon$ model in the far field. To achieve this, the $k - \epsilon$ model is converted into a $k - \omega$ formulation. The SST model is one of the most widely used models for aerodynamic flows. It is typically somewhat accurate in predicting the details of the wall boundary layer characteristics.

Since automotive external flow field is normally regarded as a steady, isothermal and incompressible 3D flow field, its computational fluid dynamic is adopted with one-way fluid-solid coupling steady state calculation. According to automotive speed, feature dimension size, air density and viscosity, the Reynolds number is $3.01\text{e}6$, which is great than $4\text{e}3$. Because the complex geometries of auto-body would cause flow separation easily, turbulent flow instead of laminar is adopted to retrieve the flow streamline. Consequently, SST model is proposed and employed to predict the model state under the operating conditions.

2.4. Model Validation and Results Discussion

Auto-body surface pressure distribution, aerodynamic forces of whole passing course, can be really simulated and re-exhibited [21]. The center-line pressure distribution is shown in Fig.5. From two pressure distribution images, airflow through the upper grille and lower bumper openings created maximum positive pressure acting on the radiator and intercooler systems. Another region with positive pressure is the windscreen. On engine hood, ceiling surface and rear end, there are negative pressures of relatively small scale, as vacuum is induced when the eddy current passed through the front. Reference [16] presented similar distribution obtained by wind tunnel test and CFD analysis. Besides it, there exists trailing vortex of a narrow pressure gradient range. In addition, the velocity vector distribution is shown in Fig.6, from which the detached-eddy at rear end is quite clear. By depicting fluid streamlines, the effectiveness of aerodynamic features like velocity can be easily explained and showed.

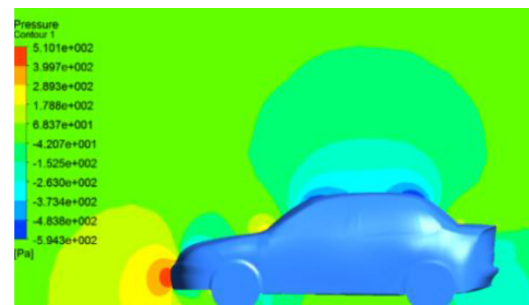


Fig. 5. Center-line pressure distribution

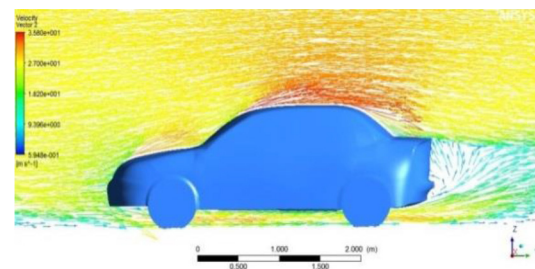


Fig. 6. Center-line velocity vector distribution