

Effects of Installation Environment on Flow around Rear View Mirror

Haidong Yuan, Zhigang Yang, and Qiliang Li
 Tongji University

ABSTRACT

External rear view mirror is attached at the side of the vehicle which is to permit clear vision for the driver to the rear of the vehicle. When the vehicle is running, the flow field around external rear view mirror is highly three-dimensional, unsteady, separated and turbulent which is known to be a significant source of aerodynamic noise and a contributor to the total drag force on the vehicle. While among all the researches on the flow field around external rear view mirror, different installation environment were employed. The external rear view mirror is mounted on a production car in most researches which presents the real condition and it can also be mounted on the ground of a wind tunnel, a specially designed table, or a generic vehicle model based on the SAE model. While, the relationship between the flow field around external rear view mirror and the installation environment is not very clear.

A new open-source realistic car model which is called the DrivAer model is used as the baseline case. The external rear view mirror of the DrivAer model is also mounted on SAE model, a designed table and mirror-only as contrastive cases. The hybrid RANS-LES method (DES) is used to investigate both the time-averaged and unsteady flow field around the external rear view mirror. When the external rear view mirror is mounted on the realistic car model, the approach flow is accelerated and deflected. Vortex shedding exists in the near wake of the mirror and an anticlockwise swirling structure is formed in the far wake. Both the time-averaged and unsteady flow field are different in other three contrastive cases. Further the influence of the installation environment on the flow field around the external rear view mirror is analyzed.

CITATION: Yuan, H., Yang, Z., and Li, Q., "Effects of Installation Environment on Flow around Rear View Mirror," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1517.

INTRODUCTION

External rear view mirror is one of the most important automobile adds-on. Its primary function is to permit clear vision for the driver to the rear of the vehicle. Conventional reflective mirrors will be the main method used for rear vision for several years to come, and none of the world's automobile laws permits using electro-optical systems instead of conventional reflective mirrors.

The flow field around external rear view mirror is highly three-dimensional, unsteady, separated and turbulent. The unsteady flow field in the wake of the external rear view mirror of a typical production car is known to be a significant source of aerodynamic noise [1], [2] and a contributor to the total drag force on the vehicle [3]. Studies show that unsteady flow field in the wake of the external rear view mirror also contributes to the vibration of the mirror glass which affects effective rear view vision from external mirrors [4].

Wake flow structure around external rear view mirror has been studied extensively. In early wake, an alternating vortex was found along the upper and the lower edge of the mirror housing, and in downstream, a vortex wrap like a tip vortex was also found [5]. Studies showed that within the half distance of the mirror span the

recirculation zone appeared and the intrinsic vortex was shed with the frequency of 19.6 Hz, also showed that a conical envelop of the vortex sheet was developed [6].

While among all the researches on the flow field around external rear view mirror, different installation environment were employed. Researchers experimentally investigated the wake flow structure around an external rear view mirror which is mounted on a production car [5]. The experiments were performed in an open circuit wind tunnel, and the wake flow structure was captured by hot wire anemometry and two-dimensional laser Doppler velocimetry. This kind of installation environment is the most accurate one and also the most expensive one, and it needs a full-scale wind tunnel. In smaller wind tunnels, experiments were performed with a real external rear view mirror mounted on the ground, and researchers measured the pressure field around a full-scale side mirror mounted on the ground of a wind tunnel [6]. GM performed experiments about characterization of the unsteady flow field behind two outside rear view mirrors mounted on the ground of a small wind tunnel and both PIV and dynamic pressure system were used [7]. When side mirror is mounted on the ground, the boundary layer on the ground affects the flow field around the mirror. To avoid the significant flow boundary layer on the wind tunnel floor and to facilitate the installation of the flow and noise measuring instruments, the mirror was mounted on a specially designed table in an acoustic wind tunnel [8], [9]. A real

scale mirror and mounting jig which were mounted in the middle of a wind tunnel test section were used to demonstrate the body frame interaction which resulted in noise increase [10]. A generic vehicle model based on the SAE Type 4 (fullback) was used for an investigation of wind noise caused by the A-pillar and the side mirror flow [11]. The side mirror was mounted on a full-scale SAE model. The generic model is simple enough to exclude excess deviation keeping the interaction with the vehicle body.

Flow field around external rear view mirror has relationship with the installation environment. The purpose of the present paper is to investigate the effects of installation environment on flow field around the external rear view mirror.

In this paper, the external rear view mirror is mounted on production car model, SAE model, specially designed table and mirror-only model respectively. The hybrid RANS-LES method (DES) is employed to investigate both the time-averaged and unsteady flow field around the external rear view mirror.

MODEL SETUP

Baseline Model

A new generic DrivAer model is introduced to close the gap between the strongly simplified models and the highly complex production cars [12]. It consists of three car configurations i.e. estate, fastback and notchback shown in Figure 1. Each configuration has several variants such as with and without mirrors and wheels, with smooth and detailed underbody. The CAD geometry is available for researchers and has been subject to experimental investigations. The present study focuses on the flow structure around the front side window. The notchback DrivAer model with side mirrors and wheels is chosen as the baseline model. A smooth underbody without detailed features and a stationary floor are used.

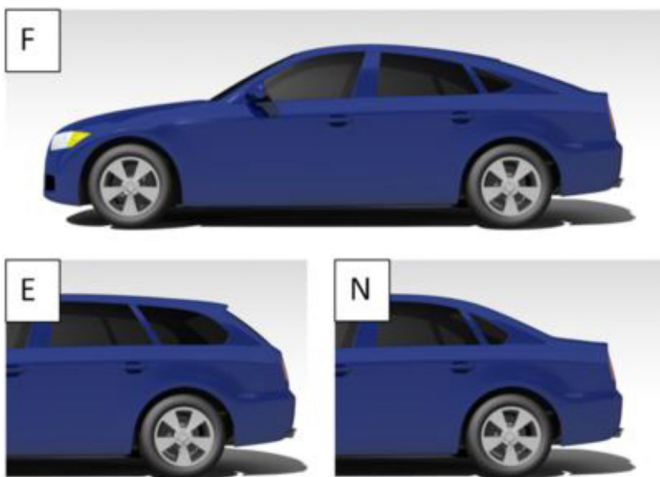


Figure 1. DrivAer model with different tops

Contrastive Models

Three contrastive models are chosen in this study i.e. SAE model, a specially designed table and mirror-only shown in Figure 2. The mirror of DrivAer model is used in contrastive models. For SAE model the mirror is mounted on the SAE Type 4 (fullback) model and its position refers to previous study [11]. For table model the mirror is vertically mounted on a specially designed table where the edges of the table need to be aerodynamically shaped to minimize or eliminate non-mirror related flow structure. The last contrastive model is a mirror-only model which is hanging in the air without any approach body. To simplify the model and save computational resources, all support components were removed.

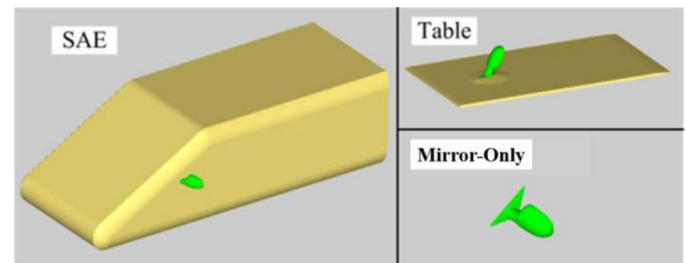


Figure 2. Contrastive Models: SAE, Table and Mirror-Only.

COMPUTATIONAL SET-UP

Turbulence Modeling Approach

Computational Fluid Dynamics (CFD) is now one of the most important design tools for the automotive industry. Both computational resources and commercial software are now sophisticated enough to handle the often complex geometries that are common in automotive design. Generally, two types of flow separation can be distinguished in the flow field around a moving car [13]. First, flow can separate on edges such as the front edge of the hood, the sides of the fenders, the front of the windshield and the rear of the vehicle. The second type of separation is three-dimensional by nature around the A- and C- pillars. Studies have shown that DES offers a clear advantage over RANS models in terms of the force coefficients, pressure coefficients distribution and capture of the three-dimensional flow feature [14], [15]. The DES model has been chosen in the present study.

DES is a hybrid RANS-LES method which attempts to provide a compromise between accuracy and computational expense by only using LES in regions of flow which are challenging to RANS models (such as separated flow) but then use RANS models elsewhere. The time-dependent IDDES (based on SST model) [16] used in the present paper adds additional functionality to shield the attached RANS boundary layer from the incorrect activation of LES modes, as well as wall-modelled LES capability to allow some of the boundary layer to be resolved by LES together with a quicker transition to resolved turbulence [17], [18]. All simulations in this paper used the same turbulence modeling approach.

Computational Grid and Boundary Conditions

The size of computational domain is $11L$ (L is the length of the car) $\times$ $11W$ (W is the width of the car) $\times$ $8H$ (H is the height of the car) and a schematic of the domain and a car are shown in Figure 3. A velocity inlet condition is imposed at the start of the domain ($4L$ upstream of the car) with a freestream velocity $U = 40\text{ms}^{-1}$ ($\text{Re} = 4.71 \times 10^6$ based upon $L = 1.85$). The turbulence at the inlet boundary is specified by intensity and viscosity ratio which are 1% and 20 respectively. A pressure outlet condition with 0 gauge pressure is imposed $6L$ downstream of the rear of the car. Slip wall condition is imposed on each side wall and top wall of the domain. The floor is set to a no-slip condition with non-rotating wheels to match the experimental non-ground simulation case. No-slip wall boundary condition is also used on the surface of the vehicle and mirrors.

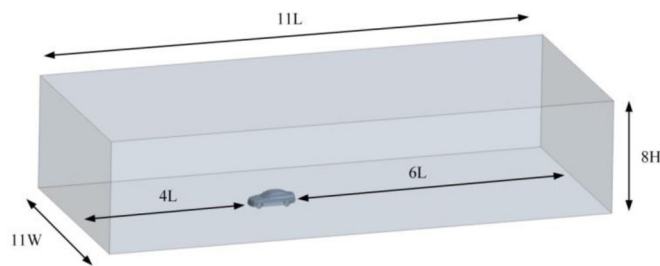


Figure 3. DrivAer computational domain size

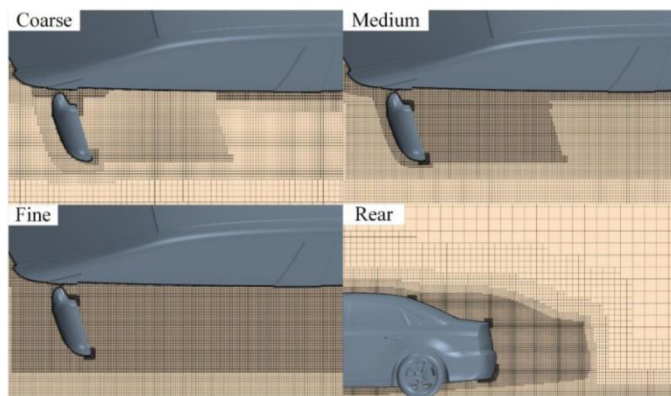


Figure 4. Slices through front window region of three meshes and rear of DrivAer used for the DES calculations

Table 1. Mesh type and resolution

Mesh Name	Mesh type	y^+	Cell Count
Coarse	Hexahedral + prism	<1	28×10^6
Middle	Hexahedral + prism	<1	35×10^6
Fine	Hexahedral + prism	<1	51×10^6

The topology of the computational grid is trimmer, which contains prism layers at the wall boundaries and a perfect hexahedral grid in the rest of the domain, and the prism layers are connected to the hexahedral grids by trimming them. The resulting mesh is composed predominantly of hexahedral cells with trimmed cells next to the surface. Trimmed cells are polyhedral cells but can usually be

recognized as hexahedral cells with one or more corners and/or edges that are cut off. The initial surface mesh is trilateral and the final is almost quadrilateral which is the projection of the volume cells. The prism layer is also hexahedral and its transition to core cells is controlled by the volume growth rate. Three grids are generated for DES calculations, which are summarized in Table 1 and illustrated in Figure 4. Each mesh is refined in the boundary layer to allow the integration of the transport equations to the wall and thus a non-dimensional wall-distance, $y^+ < 1$ was ensured over the car. Near-wall mesh has 20 prism layer cells to ensure that the boundary layer was well captured. The refinement regions are around front side window and rear portion of the car. Contrastive calculations used the same domain and boundary conditions. Trimmed mesh strategy was employed for all contractive cases with refinement in mirror region.

Numerical Approach

A segregated incompressible unstructured finite-volume solver was employed for DES simulation. To discretize the convective terms of the momentum equations, a hybrid numerical scheme [19] was used, which switches between a bounded Central Differencing Scheme (BCDS) in regions where LES is active to a second order upwind scheme where RANS modes are active. A second order upwind scheme was used for the turbulent quantities. The unsteady SIMPLE algorithm was used for the pressure-velocity coupling.

An Algebraic Multigrid (AMG) method was employed using a V cycle for the momentum equations and a flex cycle for the turbulence quantities. The solution was initialized using the steady RANS results.

The temporal discretization was done using second-order implicit scheme. The time step was 5×10^{-4} s. Time-averaged statistics were gathered for 20 convective flow units (convective flow unit = L/U) after 5 convective flow units. All contractive models were simulated by the same numerical approach.

Numerical Validation

The numerical method is validated by reference to experiment results in published papers [12]. The results for the drag coefficient are shown in Table 2 as well as the pressure coefficients in Figure 5, 6& 7. The mesh sensitivity study shows a constant trend of increasing C_d from the coarse mesh to the fine mesh and all average percentage differences from the experimental data are less than 3%. Pressure coefficient lines in Figure 5, 6& 7 for all three meshes are almost coincident and are close to the experiment data. The turbulent viscosity ratio, ν_t/ν can be thought of as a measure of the contribution of the SGS (modeled turbulence) to the overall dissipation level. This value decreases with mesh refinement shown in Figure 8. The turbulent viscosity ratio in the middle mesh case is less than 20 in the front window region which is considered enough for this investigation [14]. In this paper, the middle mesh scheme is employed for all simulations.

Table 2. Drag Coefficient for each mesh together with average percentage differences from the experimental data.

Model	C_d	Percentage difference
Exp.	0.258	
Coarse	0.261	1.2%
Middle	0.263	1.9%
Fine	0.264	2.3%

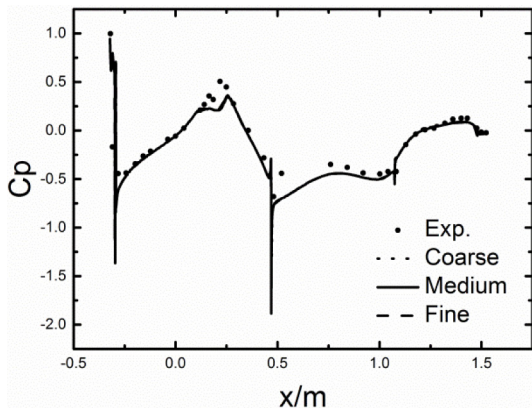


Figure 5. Mean Pressure Coefficient over the top of the Notchback configuration for each mesh.

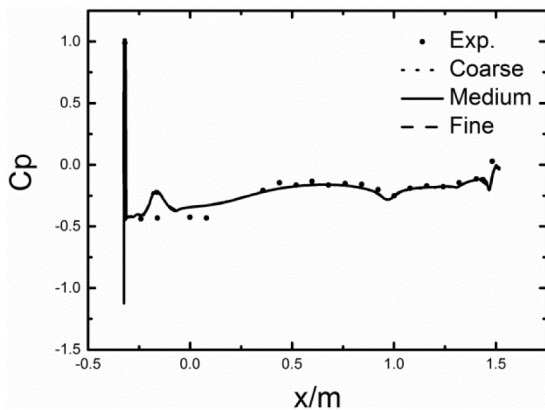


Figure 6. Mean Pressure Coefficient over the bottom of the Notchback configuration for each mesh.

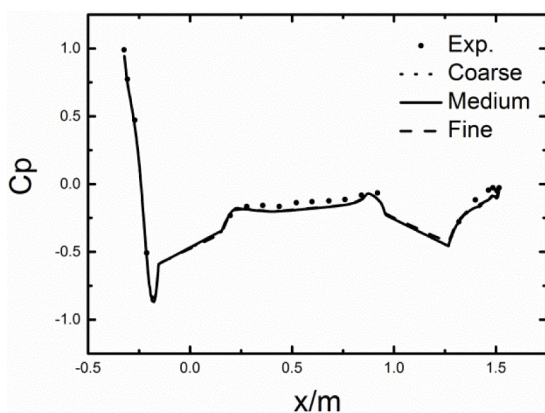


Figure 7. Mean Pressure Coefficient over the side of the Notchback configuration for each mesh.

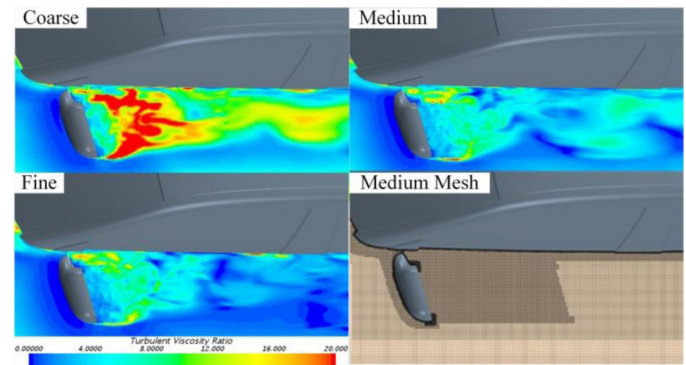


Figure 8. The turbulent viscosity ratio in front window region for three meshes

RESULTS

Coordinate System

To describe the flow field around external rear view mirror and compare different cases, the coordinate system was established shown in Figure 9. The original point was set in the central of and behind the mirror. The post-processing was made by points, lines and planes in X-, Y- and Z-planes.

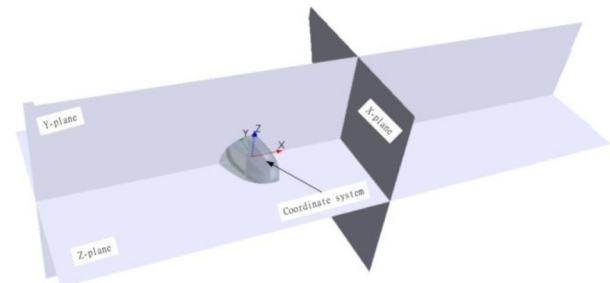


Figure 9. Coordinate system for analysis

DrivAer Model

Time-Averaged Flow Field

Approach Flow

As shown in Figure 10, the approach flow is deflected both in Z-plane and Y-plane. The u-velocity of the far away approach flow is larger than the freestream u-velocity, it decreases close to the mirror and is accelerated around the edge of the mirror shown in Figure 11&13. The approach flow far from the mirror yaws away from the vehicle body in the Z-plane and pitches down in the Y-plane shown in Figure 12&14. Both deflection and acceleration imply the existence of A-pillar vortex [20], [21]. The flow field around front side window is mostly determined by A-pillar vortex and also by the external rear view mirror. The blockage effect of the mirror causes flow deflection and acceleration around the mirror body and a wake which can interact with the vehicle body [10].