ASTFE
American Society
of Thermal and Fluids Engineers

5th Thermal and Fluids Engineering Conference (TFEC)
April 5–8, 2020
New Orleans, LA, USA

TFEC-2020-31916

STUDY OF THE INTERACTIONS BETWEEN TAILPIPE EXHAUST FLOW AND AHMED BODY WAKE

C. Blanchard[1,2], R. Balachandar[1*], F. Murzyn[2]

[1]University of Windsor, Ontario, N9G 3P4, Canada
[2]ESTACA West Campus, Laval, 53000, France

ABSTRACT

Air quality is a key issue. Cars contribute significantly to the emission of gaseous pollutants. It is crucial to analyse how these pollutants disperse in the wake of a vehicle and infiltrate the car cabin exposing passengers to high concentrations. Here, a 3D numerical study of the flow developing downstream of a simplified car model (squared-back Ahmed body) is presented when a gas (Nitrogen) is emitted from the exhaust pipe. A Reynolds-Average-Navier-Stokes (RANS) model (k-Ω SST) is used coupled with a multi-component gas method. Parameters for the simulation correspond to experimental investigations led in a wind tunnel to allow comparisons and the validation of the results. Based on the height of the car and the incoming velocity, the Reynolds number is 5×10^4. Altogether, these conditions correspond to an urban environment. This study focuses on the mixing between air and nitrogen and the flow dynamics in the close wake. We show that the tailpipe flow has no significant influence on the wake flow meaning that it is fully offset by the momentum of the incoming flow. Comparisons with experimental data obtained in wind tunnel at the same reduced scale are provided. Strong agreements are found for mean and turbulent velocities and for Reynolds stresses validating our model. The results of the volume fraction of nitrogen are also discussed indicating that the gas tracer is captured by the recirculation region due to turbulent structures. These results could provide interesting indications regarding the positioning of air intake in order to minimize pollutant infiltration.

KEY WORDS: Ahmed body, RANS, Turbulent wake flow, Mixing, Tailpipe

1. INTRODUCTION

Recent relevant data bring out the new challenges due to air pollution. Road transportation is responsible for up to 90% of Ultra Fine particles (UFP) emissions in number [1]. UFP and gaseous pollutants such as NO_x and CO_2 are among the most harmful. Indeed, they are associated with strong adverse health effects as well as with environmental issues related to the global warming. We are especially exposed in urban areas where traffic load is huge. Gas and particles can infiltrate the car cabin exposing drivers and passengers to high levels of pollutants. Indeed, the exhaust gases and UFP released from the tailpipe of a leading vehicle can penetrate the passenger compartment through the air intake system and leakage of the following vehicle [2]. To prevent this, it has become crucial to understand how pollutants disperse in the wake of a car. To date, some experimental studies were conducted to understand this complex phenomenon where turbulence plays a key role. In wind tunnel experiments, an Ahmed body is usually used as a simplified car model to make the comparison easier between the different investigations [3]. For such a car model, the governing parameter is the rear slant angle [4]. The flow topology is modified with this angle. A recent experimental study was conducted to discuss the influence of this rear slant angle on the dispersion of ultrafine particles in the wake

---

*Corresponding Author: rambala@uwindsor.ca

of a car [5]. However, experiments are complex and time consuming. Then, numerical simulations are required. They provide a solution to depict the 3D flow variations in a whole domain around the body for different conditions. Combining experimental studies with numerical simulations is then relevant and can bridge the gap for a better understanding of pollutant dispersion. In the present paper, we present a three-dimensional and unsteady simulation of the flow downstream of a squared back Ahmed body. It is based on Reynolds-Average-Navier-Stokes (RANS) equations along with k-Ω Shear Stress Transport (SST) model. The equations were solved using Star-CCM+v11.06 a finite-volume solver with a step time of 1 ms. We aim at discussing the influence of the tailpipe exhaust flow (Nitrogen) on the wake flow. New simulations were performed at the University of Windsor following the previous work on the topic [6].

2. COMPUTATIONAL MODEL

Parameters for the simulations were chosen based on the experimental investigations of Rodriguez [5]. The dimensions of the car model are presented on Fig. 1a (lengths are in mm). Based on its height (h), the Reynolds number is 4.95×10^4, the inflow velocity being U_∞=14.3 m/s. x, y and z are the horizontal, vertical and transversal directions, respectively. The origin O is located in the centreline of the test section (z/h=0) and on the rear face of the vehicle. In the experiments, the exhaust pipe has an internal diameter d=4 mm. It was located at x/h=0, y/h=0 and z/h=-0.31. Its eccentric positioning is due to the current geometry used in the automotive industry. The gaseous pollutant (nitrogen) is ejected from the tailpipe with a constant velocity U_p=10.6 m/s. This condition was selected according to a kinetic similitude corresponding to a vehicle in an urban environment. For more details about the experimental conditions, one can refer to Rodriguez [5]. The dimensionless ground clearance is h_s/h=0.28. Basically, in the close wake, a recirculating region develops (L_r, being its length). It can be divided into two regions R1 and R2 (Fig 1b) associated with a negative streamwise velocity. Two counter rotating vortices are formed. On Fig 1b, the bottom is also defined as the wall-jet region (WJ) and FS denotes the freestream region. For more details about the flow topology (3D), one can refer to [5]. Figure 2 is a sketch summarizing all relevant information.

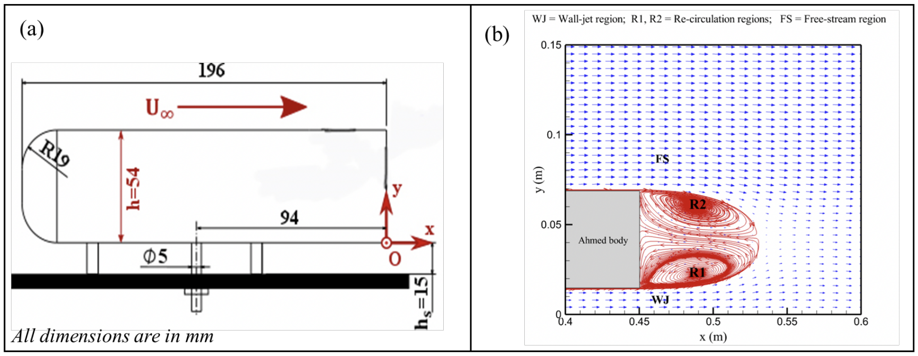
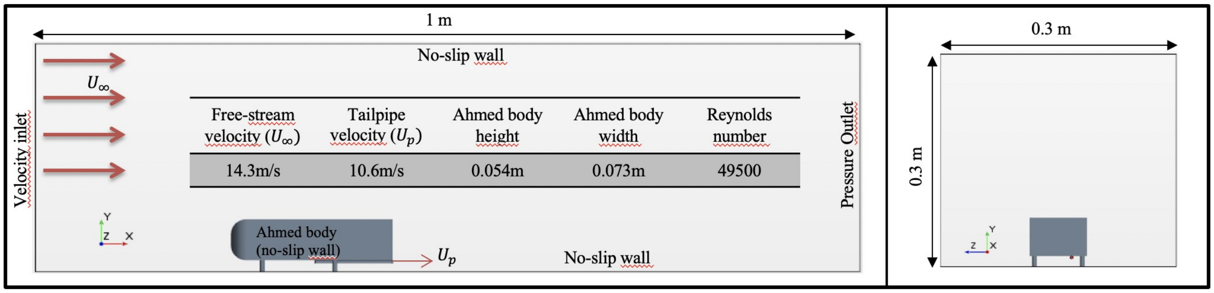


Fig. 1: (a) Side view of the Ahmed body, (b) Flow field behind the Ahmed body from [6]



Free-stream velocity (U_∞)	Tailpipe velocity (U_p)	Ahmed body height	Ahmed body width	Reynolds number
14.3m/s	10.6m/s	0.054m	0.073m	49500

Fig. 2: Computational domain

The mesh used during these simulations is composed of 4.9 million hexahedral cells and 20 prism layers to capture the flow interactions on the wall boundaries. There are also four refinement blocks from around the Ahmed body to the close region near the tailpipe. RANS model along with k-Ω SST was used to run the simulations. It was chosen for computational cost convenience rather than IDDES. Comparisons with Lienhart et al. [6] showed that the difference between the two techniques was not relevant enough to be considered as an obstacle as a first step. The physical model chosen to represent the mixing between air and exhaust gas (Nitrogen) was a multi-component non-reacting gas model consisting. The two gas were considered as the unique phase with a miscible mixture of two substances. The simulations were running for a physical time of 5s to reach a steady state. The residuals were considered converging below 10^{-6}. The simulations were also launched steady and switched to an unsteady state after 1s for a faster convergence.

3. RESULTS AND DISCUSSION

A calibration study was conducted based on the experiments of Rodriguez [5] and the numerical simulations of Lienhart et al. [6] to validate the RANS model. Overall, the flow topology was well captured even if the recirculation length was a little bit overestimated. In our simulations, we found $L_r/h=1.85$ while Rodriguez [5] and the literature gave $L_r/h=1.39$ and 1.55 ± 0.18, respectively. This tends to indicate that the RANS technique can lead to a satisfying accuracy compared to the IDDES method.

Based on this, we compared the mean horizontal velocity (U) and the Reynolds shear stresses for different vertical profiles located at x/h=0.83 and x/h=1.57 obtained from the experiments [6] with our numerical simulations when the exhaust gas (Nitrogen) is ejected from the tailpipe. In these cases, we were aligned with the tailpipe (z/h=-0.23) and closed to the rear face of the car (close wake). Fig. 3 presents the corresponding results. Left part is for the dimensionless horizontal velocity and right part for the dimensionless Reynolds shear stresses.

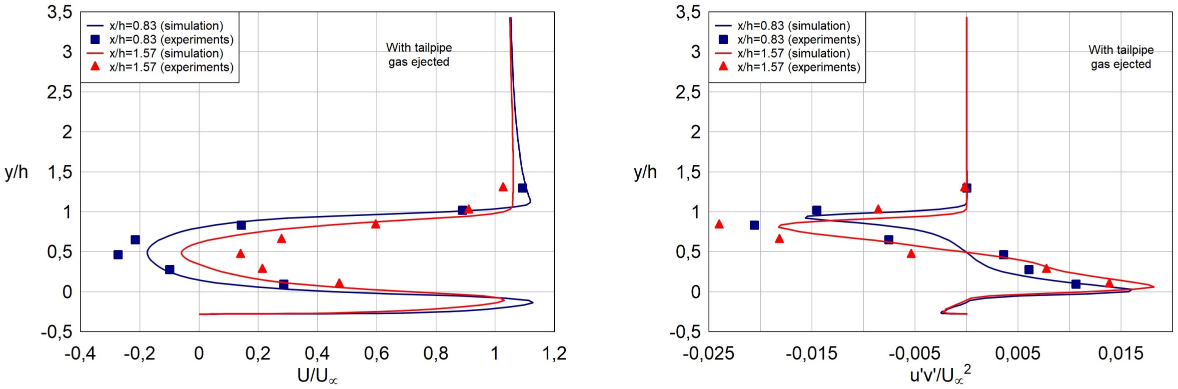


Fig 3: Velocity and Reynolds shear stresses vertical profiles in the wake of the car. Experimental and numerical results

Figure 3 shows that the velocity is increasing in the WJ region and decreasing in the recirculation area. Above the vehicle (y/h>1), the velocity is uniform. Overall, this is in agreement with the experimental results. The positive and negative peaks in the Reynolds stresses profiles match with the shear layers between WJ and R1 and between R2 and FS, respectively. For each dimensionless position x/h, experimental data and numerical results are in good agreement for both mean horizontal velocity and Reynolds shear stresses showing that our simulations are able to reproduce the flow generated in the wind tunnel.

Then, we assessed the influence of the exhaust system by comparing our numerical results with and without the tailpipe with those obtained by Rodriguez with the pipe (Fig. 4). From the numerical simulations, we show that there is no significant change in the vertical profiles of U with and without the pipe. A very small impact of the gas velocity is denoted near the tailpipe inlet. Nevertheless, it is fully offset by the contribution of the momentum brought by the incoming flow. For x/h=1.57, a slight difference is observed with the experimental results which is mainly due the RANS model, which overestimates the recirculation length.

Lastly, Fig. 5 gives a top view of the mixing of the exhaust gas (Nitrogen) in the wake flow behind the Ahmed body while Fig.6 shows a side view. An analysis of the flow streamlines confirms the results of Rodriguez [5]. There is a significant concentration of nitrogen on the opposite side of the pipe. This is due to the horizontal and vertical turbulence structures, which develop in the wake of the car enhancing the mixing. Finally, the flow carries the gas downstream with a concentration peak far downstream, which is close to the axis of symmetry of the Ahmed body (z/h=0), which is coherent with a squared-back geometry according to Rodriguez [5].

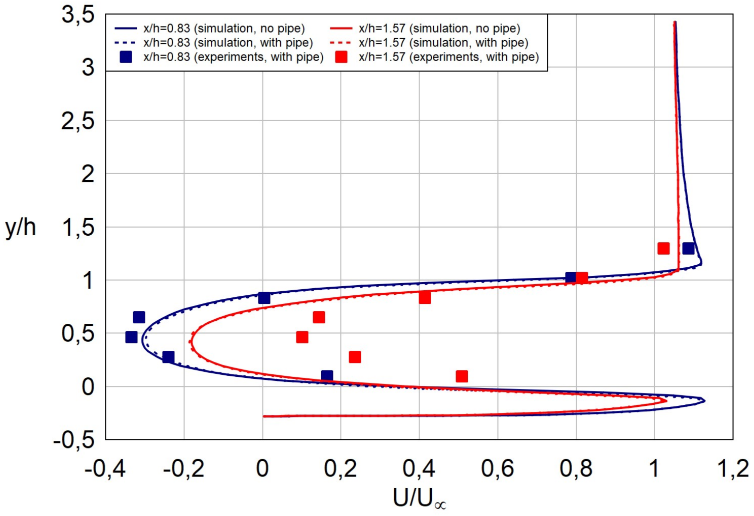


Fig 4: Comparison between experiments and numerical simulations (with and without exhaust gas)

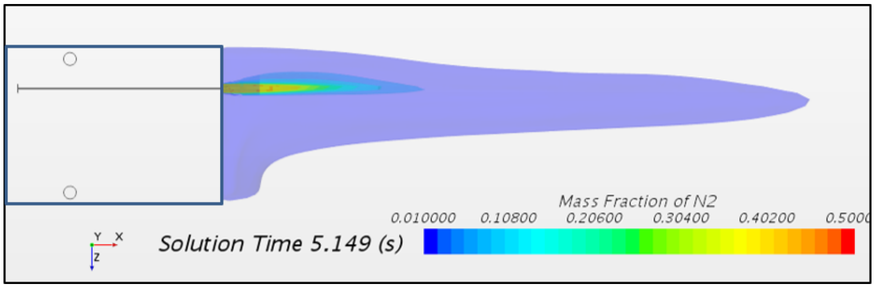


Fig. 5: Volume fraction of Nitrogen (top view)

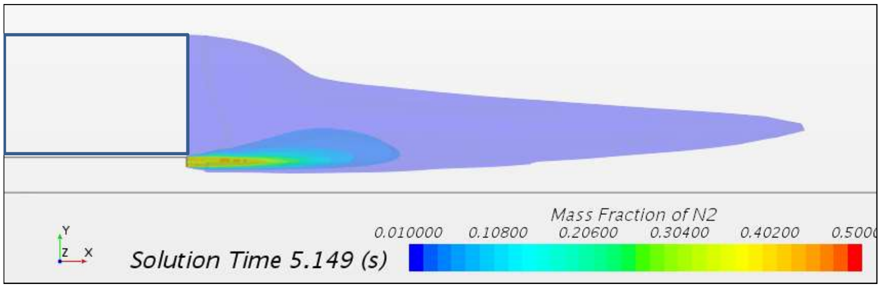


Fig. 6: Volume fraction of Nitrogen (side view)

4. CONCLUSIONS

In this paper, the mixing of exhaust gas in the wake flow of an Ahmed body was studied. Numerical results were compared with experimental data for validation. Overall, we confirmed that the exhaust gas has no significant impact on the wake flow structures. However, for this specific geometry (square back model), it can be noticed that the exhaust gas is carried in the symmetry plan of the vehicle. This result is important regarding the infiltration of pollutants into the car cabin of a following vehicle as well as for the positioning of the air intake system. The results provided by the RANS technique could be improved by using IDDES models. Lastly, other geometries and rear slant angles can be studied in further studies.

REFERENCES

[1] Mejia, J.F., Morawska, L. and Mengersen, K, "Spatial variation in particle number size distributions in a large metropolitan area", Atm. Chem. Phys, 8, 1127-1138, (2008). **Journal paper**

[2] Knibbs, L.D., de Dear, R.J., Morawska, L. "Effect of cabin ventilation rate on ultrafine particle exposure inside automobiles », *Env. Science and technology"*44, no. 9 (2010), **Journal Paper**

[3] Ahmed, S.R., Ramm, G. and Faltin, G. "Some salient features of the time-averaged ground vehicle wake", SAE Technical paper 840300, Detroit (USA), (1984). **Conference Proceedings**

[4] Ahmed, S.R., Influence of base slant on the wake structure and drag of road vehicles, Vol. 105, Transactions of the ASME (1983), **Conference Proceedings**

[5] Rodriguez, R. « *Etude expérimentale de la dispersion de particules ultrafines dans le sillage de modèles simplifiés de véhicules automobiles* », *PhD Thesis*, Ecole Centrale de Nantes, 270 pages, (2018), in french, D**issertation**

[6] Lienhart, R., Jesudhas, V. Balachandar, R., Barron, R., Rodriguez, R. and Murzyn, F. "CFD study of flow around Ahmed body", *Proc. of 4th Thermal and Fluids Eng. Conf.,* TFEC-2019-27816 (2019), **Conference Proceedings**