World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:16, No:12, 2022

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

Scale Effects on the Wake Airflow of a Heavy Truck

A. Pérard Lecomte, G. Fokoua, A. Mehel, A. Tanière

***Abstract*—**Automotive experimental measurements in wind tunnel are often conducted on reduced scale. Depending on the study, different similitude parameters are used by researchers to best reproduce the flow at full scale. In this paper, two parameters are investigated, which are Reynolds number and upstream velocity when dealing with airflow of typical urban speed range, below 15 $m.s^{-1}$. Their impact on flow structures and aerodynamic drag in the wake of a heavy truck model are explored. To achieve this, Computational Fluid Dynamics (CFD) simulations have been conducted with the aim of modeling the wake airflow of full- and reduced-scaled heavy trucks (1/4 and 1/28). The Reynolds Average Navier-Stokes (RANS) approach combined to the Reynolds Stress Model (RSM) as the turbulence model closure was used. Both drag coefficients and upstream velocity profiles (flow topology) were found to be close one another for the three investigated scales, when the dynamical similitude Reynolds is achieved. Moreover, the difference is weak for the simulations based on the same inlet air velocity. Hence, for the relative low velocity range investigated here, the impact of the scale factor is limited.

***Keywords*—**Aerodynamics, CFD, heavy truck, recirculation area, scale effects, similitude parameters.

I. INTRODUCTION

AUTOMOTIVE aerodynamic researchers and engineers frequently rely on reduced scale model for their experimental and numerical studies. Full scale wind tunnels are rare and expensive, that is why reduced scales are commonly used.

Bulky vehicles, especially heavy trucks come under being modelled by reduced scales. Indeed, most experimental studies on heavy trucks have been conducted on reduced scale models [1]-[4], in many cases with the aim of reducing drag force.

In some studies, e.g. [1], the similitude parameter between real case and reduced scale is a non-dimensional number, that is the Reynolds number. In a study conducted on a 1/8 scale truck, a large range of Reynolds number has been investigated, highlighting that, for Reynolds numbers lower than 3 million, drag force values are quite close one another, with a relative gap of 1% [1]. In other studies, velocity has also been chosen to be identical between different scales. In [2], experimental measurements have been conducted on full and 1/10 scales, using the same velocities for the two scales. Drag coefficients were found to be close with a maximal gap around 11% [2]. Velocities between 100 and 180 kilometers per hour have been used to study drag reduction on a 1/4 scale heavy truck [3] and on a 1/20 scale of tractor-trailer [4]. It has been found that add on devices placed on the truck reduce drag coefficient of more than 30% [3]. Flow topology has also been investigated, showing the formation of a large vortex in the wake flow of the truck [4].

Numerical studies on heavy trucks, fewer in number, have also been conducted and have afforded a better knowledge of the flow topology around such a vehicle. The Large Eddy Simulation investigation on a simplified heavy truck showed two contra-rotating vortices in the wake of the vehicle [5].

In general, there is a real lack of studies on the scale effects in wind tunnel modelling, especially in automotive aerodynamics. Indeed, the scale influence on drag coefficient and flow topology is poorly known; especially on heavy utility vehicles travelling at typical urban speeds (below 15 $m.s^{-1}$), for future investigations on pollutants emission and dispersion in these areas. The purpose of this paper is to numerically study the influence of scale on some flow characteristics, that are drag coefficient and wake flow topology through velocity profiles. Two parameters are kept identical between three scale sizes: Reynolds number and inlet velocity.

After defining the numerical set up, validation approach is presented. Thereafter, results on flow topology and aerodynamics through drag coefficient are given for the two mentioned parameters.

II. METHODOLOGY AND VALIDATION

A. Studied Parameters

1) Recirculation Length

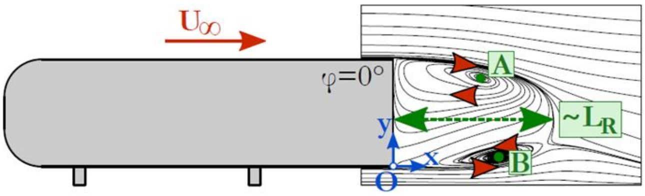

Fig. 1 Recirculation area and length on an Ahmed Body [6]

Recirculation length noted L_r is the first investigated parameter. This quantity represents the characteristic length of the recirculation region (Fig. 1), often used for wake flows studies such as those of the Ahmed Body [6]. To estimate this value, x-velocity profile, starting from the mid-height of the trailer in the wake of the heavy truck has been drawn (Fig. 2). This parameter is defined as the highest length for which the longitudinal velocity component is negative.

A. Pérard Lecomte is with the LEMTA (Laboratory Energy & Theoretical and Applied Mechanics), University of Lorraine, 2 avenue de la Forêt de Haye, 54500 Vandœuvre-les-Nancy, France (corresponding author, e-mail: aude.perard-lecomte@estaca.fr).

G. Fokoua and A. Mehel are with the ESTACA (Engineering School of Aeronautical Technics and Automotive Construction), 2 avenue Paul Delouvrier, 78180 Montigny-le-Bretonneux, France.

A. Tanière is with the LEMTA, University of Lorraine, 2 avenue de la Forêt de Haye, 54500 Vandœuvre-les-Nancy, France.

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

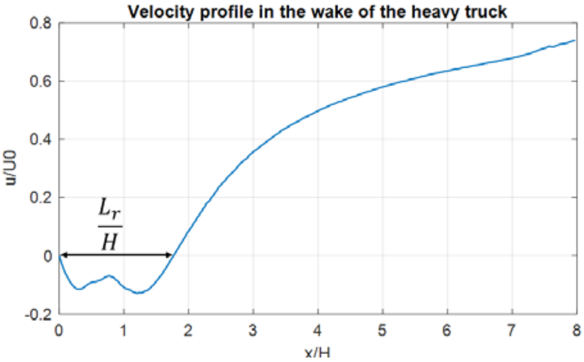

Fig. 2 x-velocity profile in the wake of the truck

2) Drag Coefficient

With the aim of characterizing scale effects on the heavy truck aerodynamics, drag coefficient has been characterized. Drag coefficient, noted C_d, is defined by (1):

$$C_d = \frac{F_d}{\frac{1}{2}\rho U_0^2 S_x} \tag{1}$$

F_d is the drag force, ρ is the fluid density, U_0 is the inlet velocity and S_x is the projected surface on x-axis.

B. Numerical Set Up

CFD-simulations have been conducted on a simplified heavy truck model, type class 8. The geometry used in the present study shown in Fig. 3 is the same as the one used in the experimental study [3]. However, some details like mirrors and under-trailer components have been removed to ease the meshing. For the same reason, wheels and trailer axles have been simplified. Three different scales of the heavy truck have been investigated: the full scale, the 1/4 scale and the 1/28 scale. The characteristic length of the vehicle is the height H of the truck's trailer (Fig. 3 and Table I).

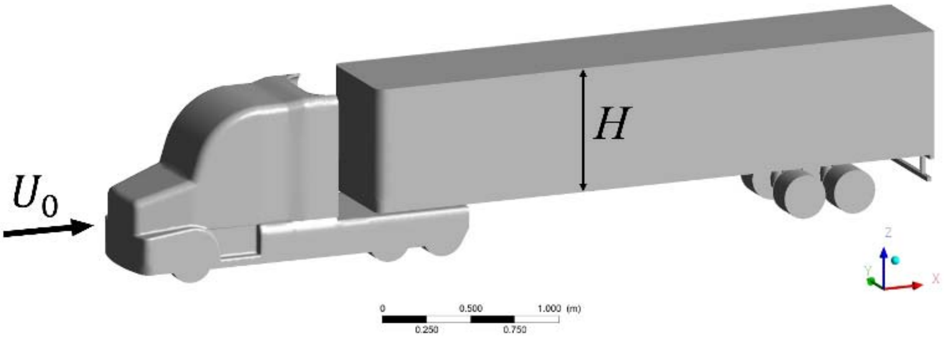

Fig. 3 Heavy truck model used in the present study

TABLE I
INVESTIGATED SCALES AND CORRESPONDING CHARACTERISTICS LENGTHS

Scale	1	1/4	1/28
H (m)	2.900	0.725	0.104

The heavy truck is positioned in a domain that has been dimensioned using two parameters: the height of the truck's trailer H and a form factor X, defined by (2). $L_{upstream}$ and $L_{upstream}$ are respectively lengths between the inlet of the domain and the front of the truck, and between the outlet of the domain and the back of the vehicle. L_{top}, L_{bottom} and L_{side} are respectively lengths on the top, on the bottom and on each side of the heavy truck (Fig. 4). Previous simulations not presented here have shown that boundary effects are negligible when the form factor X is higher than 9.5; the form factor X has also been set at 9.5.

$$\begin{cases} X = \frac{L_{upstream}}{H} \\ X = \frac{L_{downstream}}{3.H} \\ X = \frac{L_{top}}{H} \\ X = \frac{L_{bottom}}{3.6.10^{-3}.H} \\ X = \frac{L_{side}}{H} \end{cases} \tag{2}$$

The grid composed of tetrahedral elements has been dimensioned as follows. A coarse mesh size has been set in the full domain. A body of influence (Fig. 4) has been used around the truck, where the mesh size is finer. Due to a complex geometry, the grid has also been refined in other areas like staircases, deflector and trailer's axles. Inflation meshing has been applied on the truck's walls and on the domain's bottom wall, which are both stationary, to model the boundary layer. For the three scales, the first layer's height has been set so that the non-dimensional parameter y^+, defined by (3) does not exceed 12. u_τ is the friction velocity and ν the cinematic viscosity.

$$y^+ = y\frac{u_\tau}{\nu} \tag{3}$$

For each scale, three grids have been tested to investigate the mesh sensitivity on the wake flow. The grid size in two areas has been changed: in the full domain and in the body of influence region around the heavy truck (Fig. 4). Inflation meshing and local refinements remain unchanged from one grid to another. In the present paper, only the mesh sensitivity study of the 1/4 scale is presented. Characteristics and recirculation lengths of coarse medium and fine meshes are given in Table II.

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

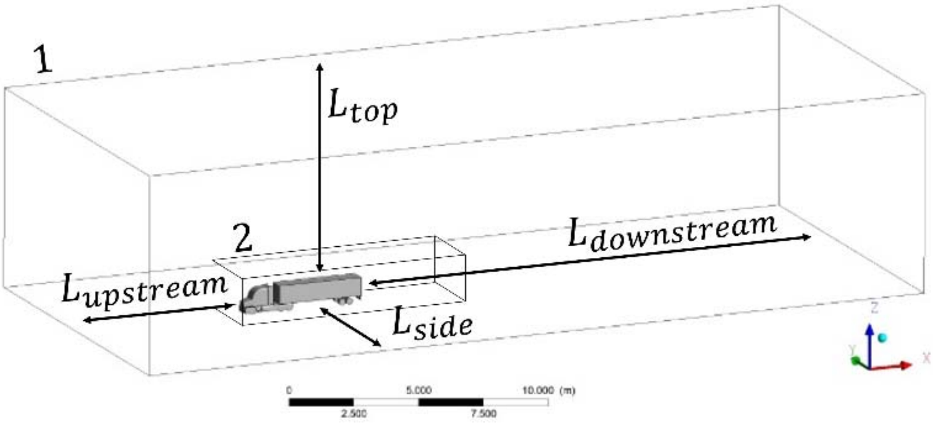

Fig. 4 Heavy truck in the calculation domain (1) and body of influence (2) for the 1/4 scale model

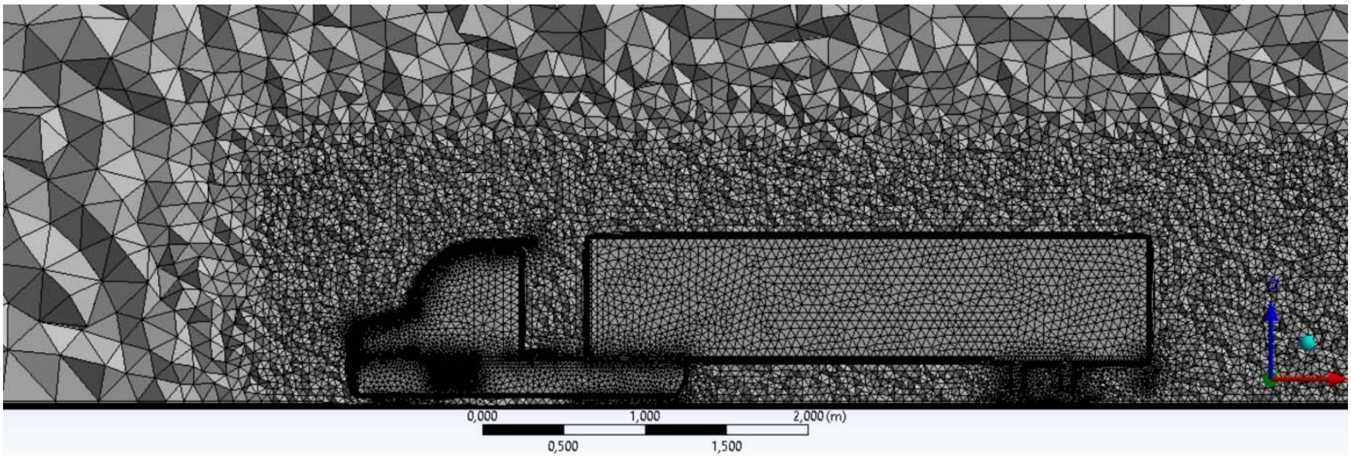

Fig. 5 Final grid for the 1/4 scale

TABLE II
GRIDS CHARACTERISTICS FOR THE GRID SENSITIVITY STUDY

Grid	Coarse	Medium	Fine
Domain mesh size (m)	0.30	0.25	0.20
Body of influence mesh size (m)	0.05	0.04	0.025
Number of elements (million)	5.3	8.1	21
L_r	1.62*H	1.77*H	1.78*H

Recirculation lengths of the medium and fine grid are very close to each other, with a deviation of 0.6%. In contrast, the coarse grid's recirculation length is around 9.3% shorter. In terms of recirculation length, the medium grid gives the same results as the fine grid, whereas it counts more than two times less elements. The medium grid (Fig. 5) has then been chosen for further simulations, offering the good compromise between accuracy and calculation duration.

A summary of the grid characteristics for the three scales is given in Table III.

TABLE III
GRIDS' CHARACTERISTICS FOR EACH STUDIED SCALE

Scale	1	1/4	1/28
Domain mesh size	0.41*H	0.34*H	0.34*H
Body of influence mesh size	0.07*H	0.06*H	0.05*H
Number of elements (million)	7.6	8.1	9.5

C. Validation

In order to validate the CFD-model, a first simulation has been completed and compared to the experimental study [3], based on drag coefficients. This simulation has therefore been set in the same conditions as in the experimental study of Landman et al. [3], that is on the 1/4 scale and with an inlet velocity of 29 m.s^{-1}. Results of drag coefficients are given in Table IV. Relative gap between the two drag coefficients is low, equal to 3.6% thus the present CFD-model is valid.

TABLE IV
EXPERIMENTAL AND NUMERICAL PARAMETERS

	Inlet velocity	Drag coefficient
Present study	29 m.s^{-1}	0.515
Experimental study [3]	29 m.s^{-1}	0.534

D. Simulation Model

Ansys-Fluent software has been used for CFD-simulations. Boundary conditions of velocity inlet and pressure outlet have been set. Heavy truck surfaces and bottom of the domain have been set as stationary wall. Symmetry condition has been applied to sides and top of the calculation domain. Transient simulations have been conducted with a time step of 10^{-2} s.

CFD-simulations have been conducted using the RANS approach. This method consists in applying the Reynolds

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

decomposition to Navier-Stokes equations, in which variables are the sum of the mean and fluctuating components. The equations of mass conservation (4) and momentum conservation (5) are then obtained.

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{4}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \rho\overline{u_i' u_j{}'}\right] \tag{5}$$

$\overline{u_i}$ and u_i' are respectively the mean and the fluctuating velocity components, $\bar{p}$ is the mean pressure. The dynamic viscosity and the density of the fluid are respectively given by μ and ρ. $\rho\overline{u_i' u_j{}'}$ is the Reynolds tensor. In the present study, the RSM has been chosen to close the system of equations. This turbulence model solves the transport equation for each Reynolds tensor component; thus, it considers the turbulence anisotropy. A near-wall treatment has been used through the two-layer models in order to model the near-wall flow.

III. Results and Discussion

Two cases have been examined: the first one is devoted to a constant value of the Reynolds number of 85000 for the three scales. In the other case the inlet velocity is kept constant and set at $U_0 = 12\, m.s^{-1}$ for each model size. This velocity corresponds for the full scale to a heavy truck travelling in an urban area, as the mean velocity in downtowns is around 42 kilometers per hour [7]. Corresponding velocities and Reynolds numbers are given in Table V for the three scales.

Reynolds number, given by (6) is based on the trailer's height *H*. ρ and μ are respectively the density and the dynamic viscosity of the air. Corresponding inlet velocities are given in Table V.

$$Re = \frac{\rho U_0 H}{\mu} \tag{6}$$

TABLE V
Reynolds Numbers and Corresponding Velocities

Scale	1	1/4	1/28
Velocity inlet for $Re = 85000$	0.4 m.s^{-1}	1.7 m.s^{-1}	12 m.s^{-1}
Reynolds number for $U_0 = 12\, m.s^{-1}$	2,400,000	600,000	85,000

For each case, recirculation length L_r and drag coefficient C_d have been assessed and are given in Tables VI and VII. Relative gaps between reduced and full-scales have been computed using (7) and (8). Relative gap between drag coefficients obtained numerically in the present study and experimentally by Landman et al. (2009) [3] has also been computed by (9):

$$\Delta C_d = \begin{cases} \Delta C_{d_{1,\frac{1}{4}}} = \left|\frac{C_{d\,1} - C_{d\,1/4}}{C_{d\,1}}\right| * 100 \\ \Delta C_{d_{1,\frac{1}{28}}} = \left|\frac{C_{d_1} - C_{d\,1/28}}{C_{d_1}}\right| * 100 \end{cases} \tag{7}$$

$$\Delta L_r = \begin{cases} \Delta L_{r_{1,\frac{1}{4}}} = \left|\frac{L_{r_1} - L_{r_{1/4}}}{L_{r_1}}\right| * 100 \\ \Delta L_{r_{s1,\frac{1}{28}}} = \left|\frac{L_{r_1} - L_{r_{1/28}}}{L_{r_1}}\right| 100 \end{cases} \tag{8}$$

$$\Delta C_{d_{exp}} = \left|\frac{C_{d\,exp} - C_d}{C_{d\,exp}}\right| * 100 \tag{9}$$

A. Reynolds Number Similitude Parameter

Velocity magnitude contours and streamlines in the central vertical longitudinal plane *y = 0* of the heavy truck are given in Fig. 6 for the three scales. Three major turbulent areas are observed: the gap area between the tractor and the trailer, in the under-trailer area and in the wake of the truck. In particular, the last one is further investigated.

In the near wake of the heavy truck, vortical structures are forming. For the three scales, this region is composed of a large vortex (A) coming from the under-trailer (Figs. 6 (a)-(c)). This vortex center is located between *x = 0.13H* and *x = 0.18H*; *z = -0.26H* and *z = -0.30H.* The vortex length and height respectively go from *0.59H* to *0.69H,* and from *0.58H* to *0.69H.*

A second vortex (B) is observable, at respectively *(x;z) = (0.89H;0.30H), (x;z) = (0.97H;0.30H)* and *(x;z) = (1H;0.28H)* for scales *1, 1/4* and *1/28*. The rotation direction of this vortex is opposed to the direction of the vortex A.

At *x = 1.52H* for 1/4 and 1/28 scales*; x = 1.23H* for the full scale, an additional structure (C) appears in the rear part of the recirculation area, where velocities are notably low. For the three models (Fig. 6), flow topology is comparable to the wake flow of the heavy truck studied experimentally in [4]. In this study, the recirculation area contains a large vortex and an additional structure comparable to vortex A and structure C. The first vortex is similar to vortices A (Fig. 6), as its center is located at *x = 0.18* [4].

Recirculation length and drag coefficient values obtained from airflow simulations at same Reynolds number corresponding to the three scales are given in Table VI. Velocity profiles in the wake of the heavy truck are also given in Fig. 7.

Recirculation length and drag coefficient values obtained from airflow simulations at same Reynolds number corresponding to the three scales are given in Table VI. Velocity profiles in the wake of the heavy truck are also given in Fig. 7.

TABLE VI
Results for Re = 85000

Scale	1	1/4	1/28
L_r	1.60**H*	1.66**H*	1.59**H*
ΔL_r (%)	-	3.7	0.6
C_d	0.575	0.539	0.552
ΔC_d (%)	-	6.3	4.0
$\Delta C_{d_{exp}}$(%)	7.7	0.9	3.4

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

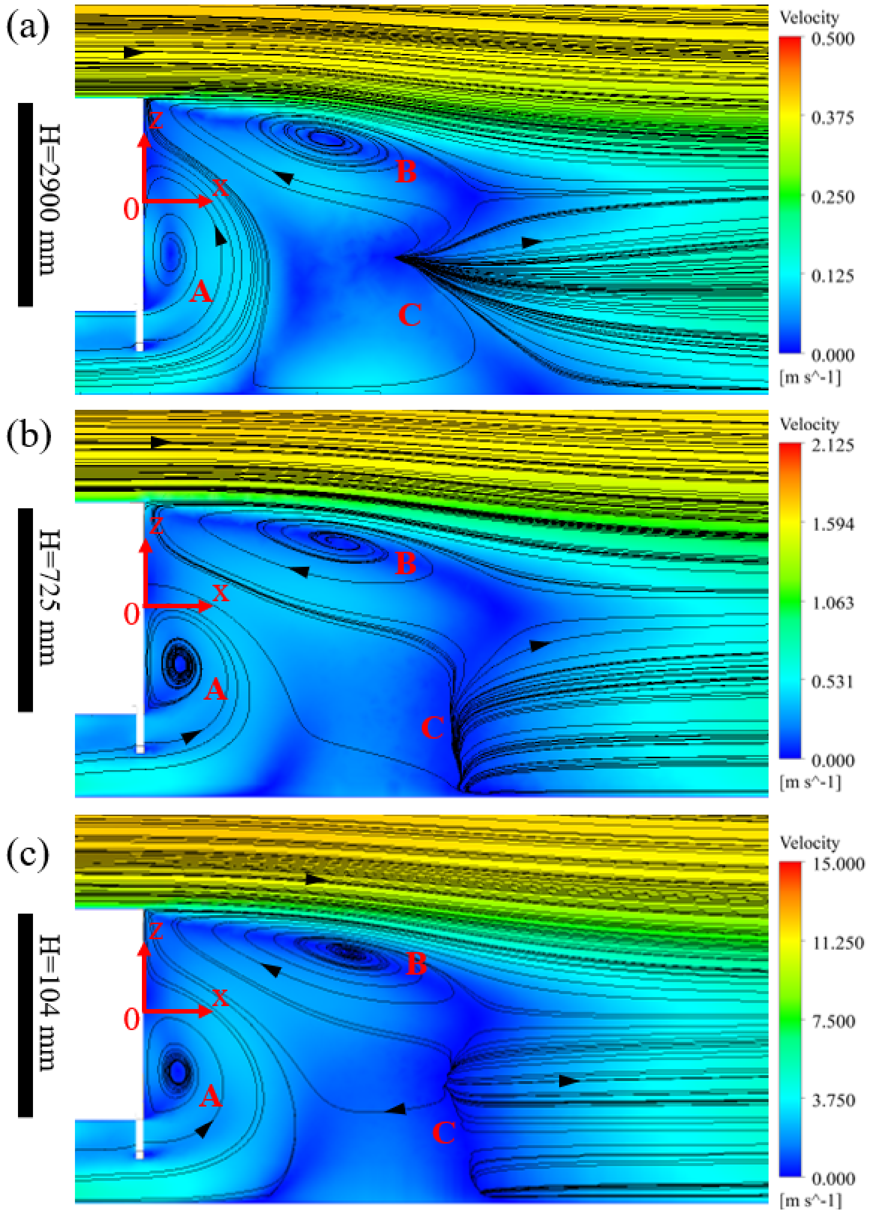

Fig. 6 Velocity magnitude contours and streamlines in the longitudinal plane $y = 0$ for Re = 85000 for scale (a) 1; (b) 1/4; (c) 1/28

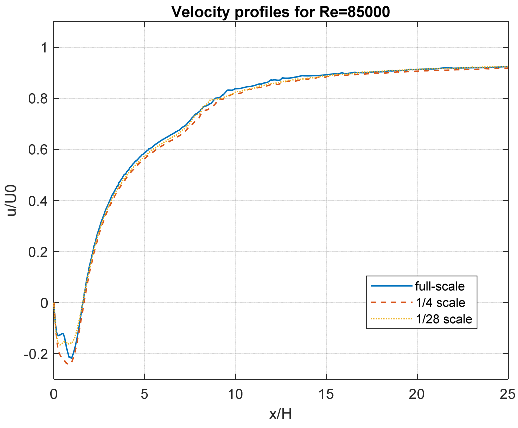

Fig. 7 x-velocity profiles in the wake of the heavy truck, for Re = 85000 for the three scales

B. Velocity Inlet as Similitude Parameter

Contours of velocity magnitude and streamlines around the three scales are given in Fig. 8, for the simulation conducted setting the same inlet velocity, $U_0 = 12\, m.s^{-1}$. As in the previous case, a large vortex with the flow coming from the under-trailer is formed (A). The center of this vortex is located around $x = 0.20H$ and $z = -0.25H$. The vortex size is similar for the three scales.

The second contrarotating vortex (B) centered at $x = H$, $z = 0.28H$ only appears on the 1/28 scale (Fig. 8 (c)). The additional structure (C) is visible at $x = 1.23H$ for the full scale (Fig. 8 (a)) against $x = 1.51H$ and $x = 1.45H$ for 1/4 and 1/28 scales respectively. It appears that as the scale-size and therefore Reynolds number is decreased, a second vortex (B) rotating clockwise emerges.

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf

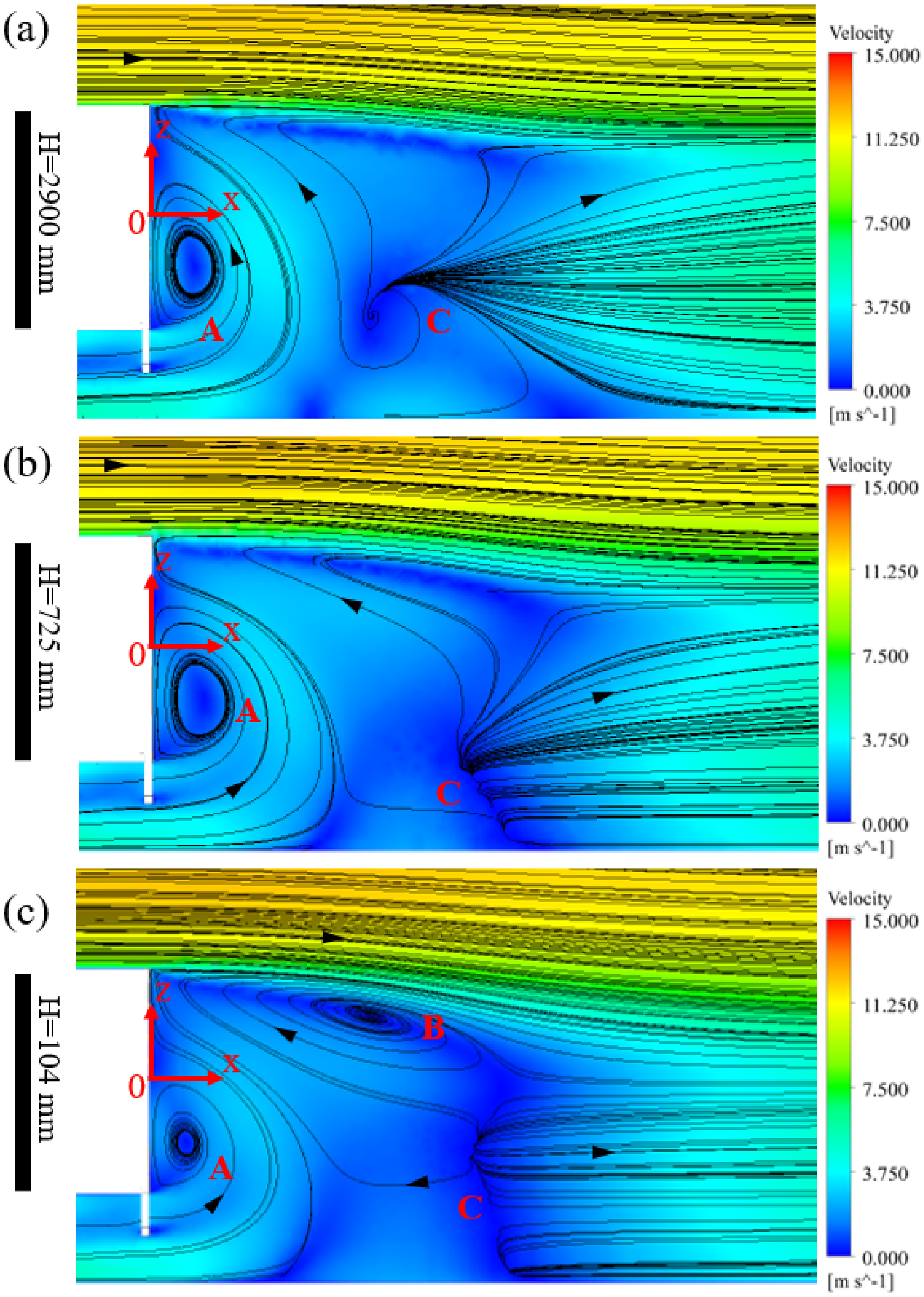

Fig. 8 Velocity magnitude contours and streamlines in the longitudinal plane $y = 0$ for $U_0 = 12\, m.s^{-1}$ for scale (a) 1; (b) 1/4; (c) 1/28

Results of the three scales simulated with the same inlet velocity $U_0 = 12\, m.s^{-1}$ are given in Table VII and velocity profiles in Fig. 9. Recirculation lengths for 1/4 and 1/28 scales are close with a gap of around 6.3%. However, recirculation length is much lower for the full scale, where the gap with other scales exceeds 30%. The trend of velocity profiles remains unchanged for the three scales: profiles are similar for $x > L_r H$. Regarding drag coefficients, the highest gap noticed equals 5.3%. Drag coefficients are close to experimental [3] value, with a maximal relative gap of 3.4%.

TABLE VII
RESULTS FOR $U_0 = 12\, m.s^{-1}$

Scale	1	1/4	1/28
L_r	1.29*H	1.69*H	1.59*H
ΔL_r (%)	-	31.0	23.3
C_d	0.524	0.517	0.552
ΔC_d (%)	-	1.3	5.3
$\Delta C_{d_{exp}}$(%)	1.9	3.2	3.4

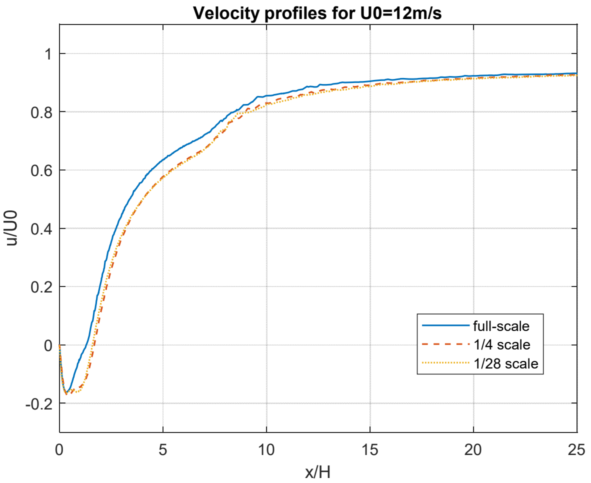

Fig. 9 x-velocity profiles in the wake of the heavy truck, for $U_0 = 12\ m.s^{-1}$ for the three scales

IV. CONCLUSION

In the present study, the issue of scale influence on flow characteristics is raised; when dealing with airflow of limited velocity range, below 15 m.s^{-1}. There is a question of whether similitude parameter, especially Reynolds number really influences aerodynamics and flow topology. To respond that, results of simulations obtained for three scales have been compared: full size, 1/4 and 1/28 models. On one hand, we ensured the dynamic similitude when the inlet velocity has been changed for the three scales to keep the Reynolds number constant. On the other hand, the inlet velocity has been chosen to be the constant parameter between different scales. Results and flow topology show the same trend regardless of the similitude parameter. In particular, drag coefficients are close to each other for the three scales using the two similitude parameters, with a maximal gap of 6.3%. Velocity profiles follow the same trend for the two similitude parameters, outside of the recirculation area, even if recirculation region seems to be bulkier at reduced scale for simulations keeping the same airflow velocity. Flow topology especially the main vortex looks similar for the three scales and for the two similitude parameters. However, some vortical structures do not appear on each studied case. Finally, when dealing with wind tunnel or numerical studies with reduced scale models, the dynamic similitude or the inlet velocity could be chosen to compare the results with a full-scale vehicle when the investigation concerns airflow with limited velocity range (below 15 m.s^{-1}).

REFERENCES

[1] B. L. Storms, D. R. Satran, J. T. Heineck and S. M. Walker, "A Study of Reynolds Number Effects and Drag Reduction Concepts on a Generic Tractor-Trailer," NASA Ames Research Center, 2004.
[2] K. R. Cooper and L. Jason, "Model and Full-Scale Wind Tunnel Tests of Second-Generation Aerodynamics Fuel Saving Devices for Tractor-Trailers," SAE Technical Paper, Vols. 2005-01-3512, 2005.
[3] D. Landman, R. Wood, W. Seay and J. Bledsoe, "Understanding Practical Limits of Heavy Truck Drag Reduction," SAE International Journal of Commercial Vehicles, Vols. 2009-01-2890, 2009.
[4] K. H. Lo and K. Kontis, "Flow Characteristics over a Tractor-Trailer Model with and without Vane-Type Vortex Generator Installed," Journal of Wind Engineering and Industrial Aerodynamics, vol. 159, pp. 110-122, 2016.
[5] A. H. Rao, G. Zhang, B. Minelli, Basara and S. Krajnovic, "An LES Investigation of the Near-Wake Flow Topology of a Simplified Heavy Vehicle," Turbulence and Combustion, vol. 102, no. 2, pp. 389-415, 2019.
[6] R. King, "Active Flow Control II," Springer Berlin Heidelberg, p. 417, 2010.
[7] ONISR, "Observatoire des Vitesses," 8 November 2021. (Online). Available: https://www.onisr.securite-routiere.gouv.fr/etudes-et-recherches/comportements-en-circulation/observations/observatoire-des-vitesses.

Open Science Index, Mechanical and Mechatronics Engineering Vol:16, No:12, 2022 publications.waset.org/10012820.pdf