Strojnícky časopis – *Journal of* MECHANICAL ENGINEERING,
VOL 72 (2022), NO 2, 41 - 56

NUMERICAL ANALYSIS OF THE REAR WIND INFLUENCE ON THE TRUCK DURING TRAVEL

DOÇI Ilir[1], SHALA Ferat[1*]

[1] *Faculty of Mechanical Engineering, University of Prishtina, Bregu i Diellit, n.n. 10000 Prishtina, Kosovo*
** Corresponding author, e-mail: ferat.shala@uni-pr.edu*

Abstract: Rear wind is the air flow that blows in the rear side of the truck. Also known as a tailwind, this aerodynamic force can cause stability issues during forward travel. The methodology of the study is CFD numerical analysis of vehicle aerodynamics. The results will be acquired for the influential aerodynamic parameters, which includes velocity, pressure, vortex, stress, that will explain the impact of rear wind on the truck body. This is achieved with the aid of design, modelling, and simulations using software Comsol Multiphysics. The simulations were performed with freestream velocity of rear wind on the back and around the truck body. Virtual Wind tunnel is modelled around the truck to simulate the wind blow and distribution. There are two airflow regimes analysed: laminar flow and turbulent flow. Simulations for the laminar air flow were implemented for Compressed air flow and the velocity limit Ma<0.3. Simulations for the turbulent air flow regime were implemented based on Reynolds-Averaged Navier-Stokes Equations (RANS) and K-ω model. Anemometer device is used for the experimental measurements of the wind velocity around the truck. The wind is a function of air velocity and atmospheric pressure. The paper is in the fields of Vehicle aerodynamics, Fluid Mechanics, and Transport safety.

KEYWORDS: Rear Wind, Truck, Aerodynamic analysis, Laminar flow, Turbulent flow, Modelling, Simulations

1 Introduction

Rear winds are natural motion of the air that have various speed, direction and form, and very often unpredictable flow. It is rarely analysed as an air flow that impacts the truck's motion. Generally, it is considered as a wind that helps increase the speed of the vehicle, but not its motion stability, and can make braking more challenging [2]. It tends to influence driving movement, direction, and stability and depends on the velocity, turbulence, and direction of the wind.



Fig. 1 Direction of Rear wind on the Truck

In this paper we will analyse the influence of rear wind as an aerodynamic force on a heavy truck that is used for carrying packages during its forward travelling (Fig.1) [3].

DOI: 10.2478/scjme-2022-0015, Print ISSN 0039-2472, On-line ISSN 2450-5471

The numerical study is conducted on the three-dimensional truck model. The model of the truck is developed in the software Comsol Multiphysics. Fluid dynamics analysis using this software in general consists of designing the 3D model of the truck with main parts as a solid objects [4], [12], adding ambient around it in the form of Wind tunnel [7], [8], and implementing wind input parameters. Simulation will be accomplished for two cases of wind flow regime, laminar airflow, and turbulent airflow. Achieved results will be compared. Developed model is presented in Fig.2 and Fig.3 [4]. Model is constructed in the rear wind stream direction.

In the model, rear wind velocity is measured experimentally, and equals the wind gust at the speed v_{wg} = 16.31 m/s.

Main technical features of the truck are (Fig.1): *Load Capacity Q=7105 kg, Tare weight W= 11090 kg, Max Weight* W_{max} *= 13000 kg, Overall Length L = 9 m; Width W = 2.5 m; Height H = 3.4 m. Wheelbase is* l_t *= 5.5 m* [3]. We adopt the truck is traveling at the speed of *60 km/h* on a straight road.

In the development environment, besides the truck model created, around it will be added the virtual wind tunnel to simulate the rear wind blowing in the back and parallel of the side walls of the truck. The bottom of the tunnel is modelled as a flat ground; other sides are modelled as a freestream air motion (Fig.3).

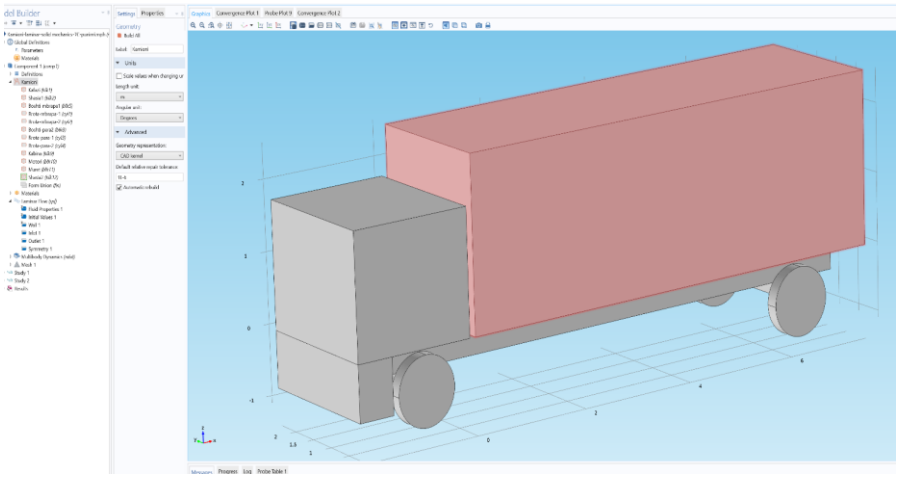


Fig. 2 Modelled truck and the environment in the Comsol Multiphysics software [4]

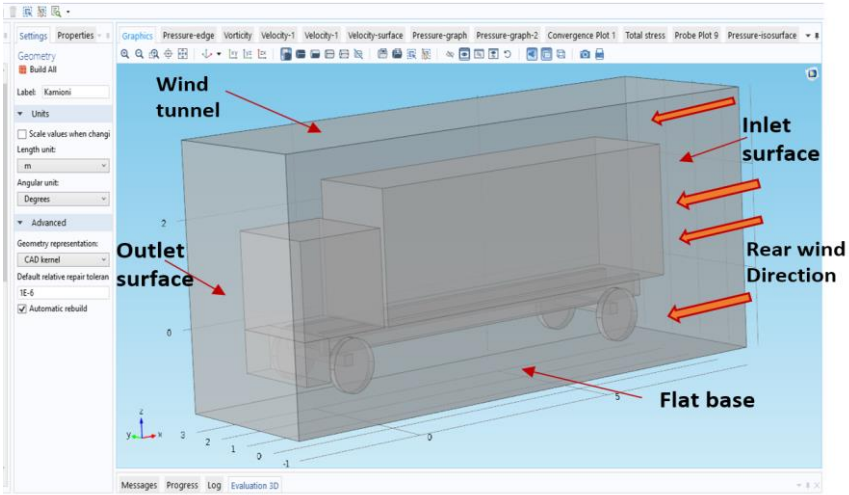


Fig. 3 Wind tunnel around the modelled truck [4]

Features of the modelled wind tunnel are *Length 12 m; Depth 5 m, Height 5 m* (Fig.3).

2 Theoretical Background

As we mentioned earlier, the air flow in a wind tunnel can have two motion regimes: *laminar flow* and *turbulent flow*. Analysis will be conducted for both regimes.

Laminar airflow is the case when air moves smoothly, without disruption between air layers, which are almost parallel in motion, and without swirls or multidirectional currents [1].

Turbulent airflow is the case when air layers show irregular movement with disturbances and swirls. It occurs when the air velocity is higher, and viscosity is low. [1], [5]

Most important parameter to define and distinguish these two regimes of fluid flow is Reynolds number (Re) [1]. It is the main parameter about the viscous behavior of all Newtonian fluids. It is a dimensionless number expressing the ratio between inertial and viscous forces, calculated by the formula [1], [6]:

$$R_e = \frac{\rho \cdot v \cdot L}{\mu} = \frac{\rho \cdot v \cdot L}{\upsilon} \tag{1}$$

where:

- ρ is the density of the fluid,
- v is the velocity of the fluid,
- L is the characteristic length of the body
- μ is the dynamic viscosity of the fluid
- ν is the kinematic viscosity of the fluid

In the Literature is also known the term *Transition regime* [1]. It is considered as a mixture of laminar and turbulent flow that occurs in the same flow, or the regime that separates the laminar and turbulent flows. The distinction between the flow regime depends on the range of Reynolds number, with these values [1], [6]:

$1 < Re < 100$ – laminar flow, strong Reynolds number dependence,
$100 < Re < 10^3$ - laminar, boundary layer theory useful,
$10^3 < Re < 10^4$ or $2300 < Re < 4000$ - transition to turbulence,
$10^4 < Re < 10^6$ - turbulent, moderate Reynolds number dependence,
$10^6 < Re < \infty$: turbulent, slight Reynolds number dependence.

For slow to moderate airflow where the velocities are small, the laminar regime is applied for numeric analysis. For higher stream velocities, turbulent flows are more appropriate.

The equations solved by the Laminar Flow interface are the Navier-Stokes equations for conservation of momentum and the continuity equation for conservation of mass [1], [4], [23]. These equations are partial differential equations that describes the movement of fluids in a medium.

The Reynolds - averaged Navier-Stokes (RANS) equations are used as the numerical method to model and analyse air flow for the turbulent flow in this paper [13]. These are motion time-averaged equations for the fluid flow. The fluid flow quantities are decomposed into their time-averaged and fluctuating components known as Reynolds decomposition [14]. These equations can be used to give approximate averaged solutions to the Navier-Stokes equations [1]:

$$\frac{\vartheta \overline{u_\iota}}{\vartheta x_i} = 0 \tag{2}$$

$$\frac{\vartheta \overline{u_\iota}}{\vartheta t} + \overline{u_J}\frac{\vartheta \overline{u_\iota}}{\vartheta x_j} = -\frac{\vartheta \bar{p}}{\vartheta x_i} + \nu \frac{\vartheta^2 \overline{u_\iota}}{\vartheta x_i \vartheta x_j} - \frac{\vartheta \tau_{ij}}{\vartheta x_j} \tag{3}$$

where:

$\overline{u_i}$– scalar component of mean velocity in the *i* direction, also filtered,
$\overline{u_j}$– scalar component of mean velocity in the *j* direction, also filtered,
$\bar{p}$ – mean pressure,
τ_{ij} – scalar components of Reynolds- stress tensor,
v - fluid kinematic viscosity,
t – time parameter.

3 Literature Review

There are lot of published papers concerning the wind influence on the truck, numerical analysis, CFD analysis and wind flow parameters. There are few concerning the rear wind influence, more as part of general studies of the wind flow around the truck. This was the reason we prepared this paper and conducted analysis.

We will name few of the authors that contributed to the topic.

In the papers on the topic of Wind influence on the trucks, authors Hamir Rahai et al. [9] have conducted numerical and experimental investigations of air flow on the Ahmed Body truck model, where they adopted the freestream mean velocity 24.5 m/sec for the numerical investigations and 16.67 m/sec for the experiments.

Author Shahrin H. Amirnordin [10] in his paper has studied the relative values of drag and lift forces acting on a passenger trailing a large vehicle under unsteady conditions. The simulation is conducted using *Fluent CFD* software for a two-dimensional flow domain at Reynolds number values Re $3.65 \cdot 10^6$ for a trailing distance of 0 to 30 meters.

Authors A. Alonso-Estébanez et al. [11] on their paper, using numerical approach known as Large Eddy Simulation (LES) present the effect of various testing truck infrastructure parameters on the coefficients of truck aerodynamics under cross wind conditions.

In their paper, authors Selvi Rajan S et al [7]. have tested the physical phenomena of the wind flow on a 1:50 scaled model of a launch vehicle. The simulation of the flow was implemented using Wind tunnel and Computational Fluid Dynamics (CFD).

Author T. Persson on his Mater thesis [8] has presented the design of wind tunnel in full scale and a study on a full-scale truck. He noted the importance of interpreting the truck behavior based on wind tunnel results.

A. Alshqirate et al. [15] have used CFD software *OpenFOAM* and implemented CFD analysis on full scale pickup truck for the speed range from 40 km/hr to 140 km/hr. They used k-ω model for turbulence modelling. On their work they investigate the influence of cargo area covering on the aerodynamics drag.

Concerning the effects of side wind or cross wind in vehicle models, and methods of analysis, papers are published by the authors Xiaoyu Zhang [19], E. Guilmineau et al. [20], H. Kozmar et al. [21].

Helena Martini [22] on her phd thesis have investigated the aerodynamic drag and cooling airflow for heavy trucks, under the circumstances of reconsidered total truck length legislation. Computational Fluid Dynamics (CFD) was the main tool for obtaining the results.

Concerning the wind effects on truck crashes, authors Robert Rescot et al. [24] have obtained the data for all heavy-vehicle crashes on the road I-70 in Kansas that involved strong winds over a three-year period. The objective was to design a model that can predict the possibility of such wind induced truck crashes.

The theoretical background about the Fluid dynamics is referenced in the books of White, Frank M. [1], Joseph H. Spurk et al. [5]. The references about the *Reynolds-Averaged Navier–Stokes Equations* are referenced in the paper of Giancarlo Alfonsi [13] and in the link [14]. The methodology of CFD analysis is referenced in the book of John D. Anderson Jr [6]. Magdolen Ľuboš et al. [18] have presented virtual simulations of vehicle manoeuvring which can be useful for the truck back wind flow.

The theoretical background about the Aerodynamic vehicle design is referenced on the book of Jason Moffat [12], Vehicle dynamics is referenced in the book of Thomas D. Gillespie [26].

4 Experimental measurements of the rear wind

For the implementation of the correct wind velocity in the study, measurements were done to implement them on the simulations for the accurate results [17].

The instrument called Anemometer was used to measure the wind velocity. It was mounted close to the tested truck [16] (Fig.4).

Measurements were acquired in the month of April, when the wind is strong in the city of Prishtina. Based on measurements, highest value of the wind speed is v_w *=28.6 mph= 46 km/h = 12.8 m/s*, and wind gust v_{wg} *= 36.5 mph = 58.74 km/h = 16.31 m/s* (Fig.5 & Fig.6). [17]



Fig. 4 Anemometer mounted close to the tested Truck for the measurements of the Wind speed [16]

Summary
April 1, 2022 - April 30, 2022

	High	Low	Average
Wind Speed	28.6 mph	0.0 mph	4.0 mph
Wind Gust	36.5 mph	--	5.2 mph
Wind Direction	--	--	SSW
Pressure	30.22 in	29.45 in	--

Fig. 5 Table of the measurements of the Wind speed in the period 04/01/2022 – 04/30/2022 [16], [17]

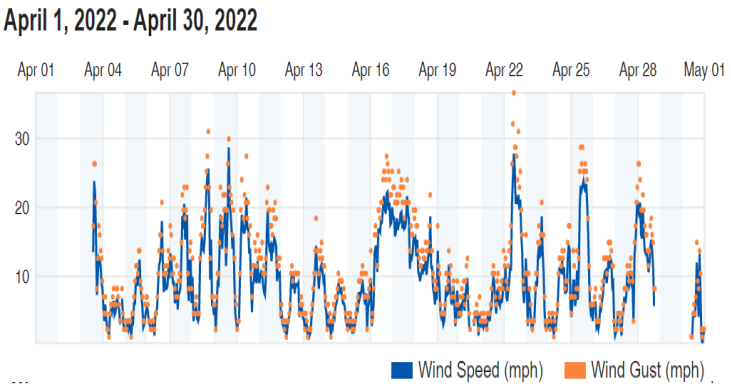


Fig. 6 Graph of the Wind speed measurements
in the period 04/01/2022 – 04/30/2022 [

For the calculation of results, and analysis, we will take the max speed of wind, the wind gust measured v_{wg} = *16.31 m/s.*

5 Modelling the grid and boundary conditions

For the modelling and analysis purposes, the CFD models are discretized in the mesh with 2D and 3D elements. Two kinds of grid were used in the CFD model: Free tetrahedral grid of the virtual tunnel for fluid dynamics analysis, which consists of 1 domain, and free tetrahedral grid for the solids and surfaces of the truck, which consists of 11 domains.

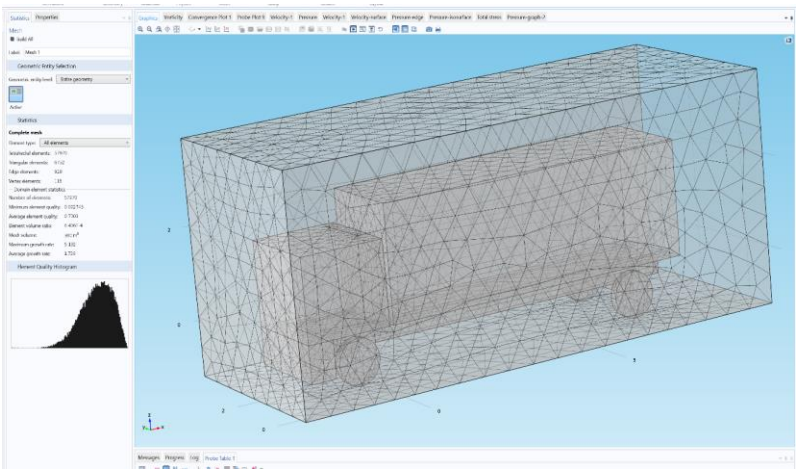


Fig. 7 Grid model of the truck with virtual tunnel

The grid size contains 57970 tetrahedral elements, 6752 triangular elements, 910 edge elements, vertex 118 elements. The mesh volume is 300 m^3 (Fig.7).

The grid has the property of expansion (growth rate) for better accuracy in the areas of the boundary layers. The grid has maximum growth rate 5.873 and average growth rate 1.735. The growth rate value is a local measure greater than or equal to 1 indicating the maximum element size growth rate between two neighboring elements [4].

The boundary conditions for solving the CFD model of the truck are [11]:

- **Inlet:** Is the surface of the virtual tunnel in the CFD model, which is parallel to the rear side of the truck trailer (Fig.8). In this surface the rear wind blows. The inlet surface is used for boundary conditions for which there is a net flow into the domain. The wind speed of the wind gust v_{wg} = *16.31 m/s* is defined as a normal inflow velocity rather than velocity field

with components x, y, z. When there is an Inlet surface defined, there should be also the outlet surface defined, with boundary conditions which can be different.

- **Outlet:** Is the surface of the virtual tunnel in the CFD model, which is parallel to the front side of the truck (Fig.9). The main physical parameter to define is Relative pressure, usually $p = 0$, with the condition of suppress backflow.

- **Tunnel Walls**: The bottom surface was defined as a solid surface, the road (Fig.9). It is positioned below the truck wheels, as a travelling road. Other three surfaces are adopted as a freestream airflow parallel to the truck's length.

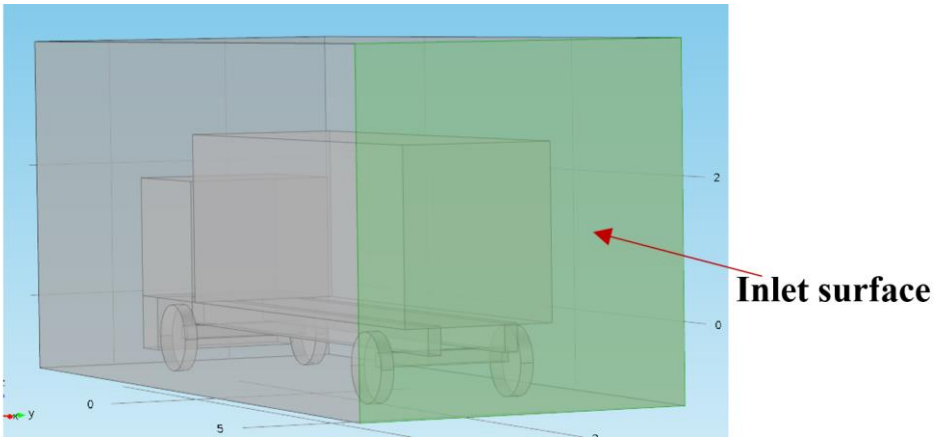


Fig. 8 Inlet surface-boundary behind the truck

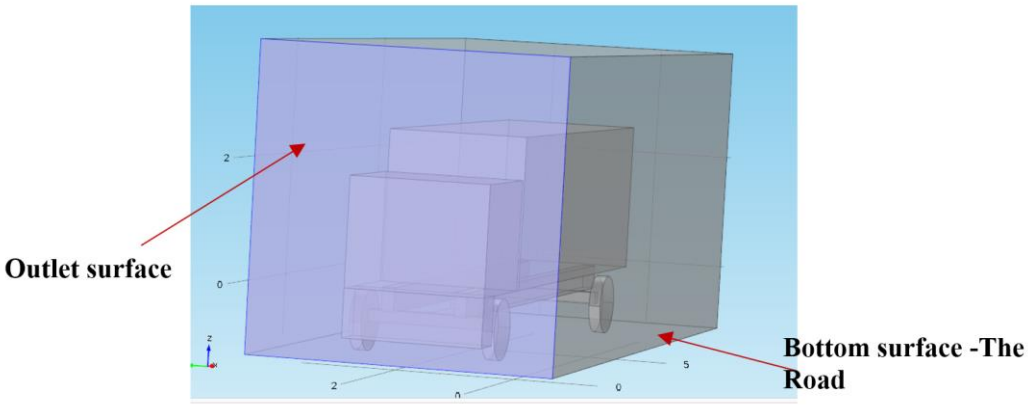


Fig. 9 Outlet surface-boundary in front of the truck

6 Numerical analysis for the Laminar flow regime and results

For the Laminar flow regime, wind flow will be implemented in the rear as inlet, with the speed of wind gust of $v_{wg} = 16.31\ m/s$. The atmospheric pressure level is $p_{ref} = 1\ atm$. Compressibility is for compressible flow, $Ma<0.3$ (Mach speed of air). Equation form is Study Controlled [4] of equations mentioned in Chapter 2, for Laminar flow. For this regime, there is no turbulence model type applied. The ambient temperature is adopted with value $T = 293.15[K] = 20^{o}C$.

The aim is to acquire the values of the wind velocity, pressure, and stress in the rear of the truck and around it, and the form of the wind vortex. The results further are given in the form of 3D and 2D graphs.

In fig.10 is the 3D graph with results of the velocity field and magnitude on and around the truck. Maximal values of the velocity are present in the front part and below the truck. They can reach values up to v_{max} = *33 m/s*, although the rear wind speed is v_{wg} = *16.31 m/s*.

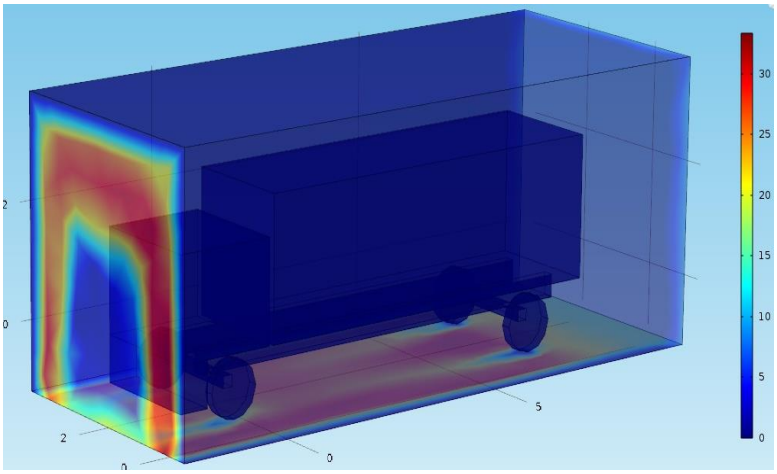


Fig. 10 Velocity field and magnitude around the truck, (m/s)

In fig.11 is the 3D graph of velocity vortex around the truck. Maximal values of the velocity vortex are present in the front part and below the truck (dark brown color). This is due to the cabin form of the truck, which extends low, close to the ground and creates extra resistance (drag) to the rear wind flow. Drag is the aerodynamic force and resistance against the vehicles motion through the *air* [12], [26].

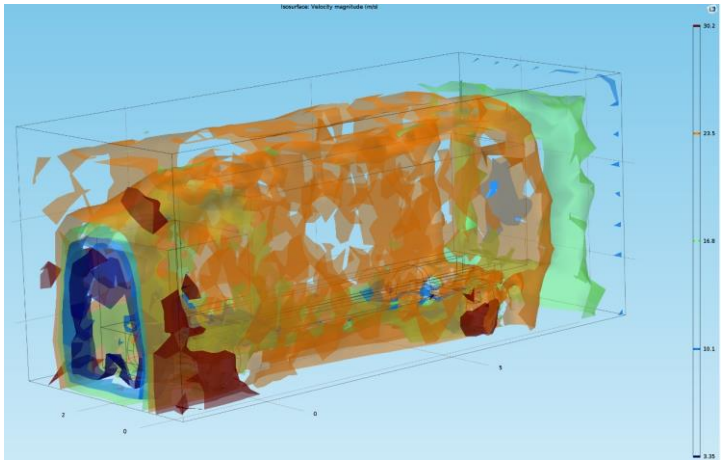


Fig. 11 Velocity vortex around the truck, (m/s)

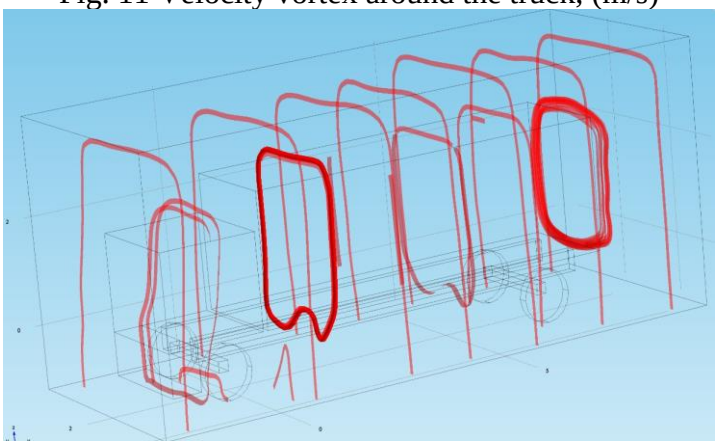


Fig.12 Vorticity field around the truck, (1/s)

In this case, drag is the resistance created from truck form and structure during the rear wind blow.

Maximal values of velocity reach up to $v_{max} = 30.2\ m/s$ at below the cabin, although the rear wind velocity is $v_{wg} = 16.31\ m/s$.

In fig.12 is shown 3D graph of the vorticity field around the truck. The curves of vorticity represent the local spinning motion of air around the truck. Maximal vorticity curves are present in the back side and front side of the truck. Highest vorticity is behind the cabin.

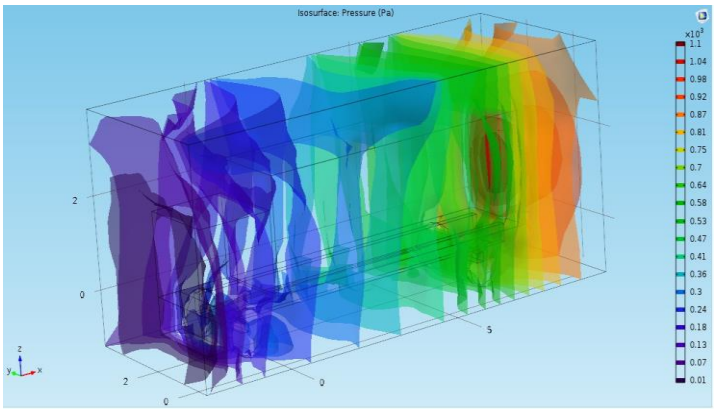


Fig. 13 Pressure on and around the truck (Side view), (Pa)

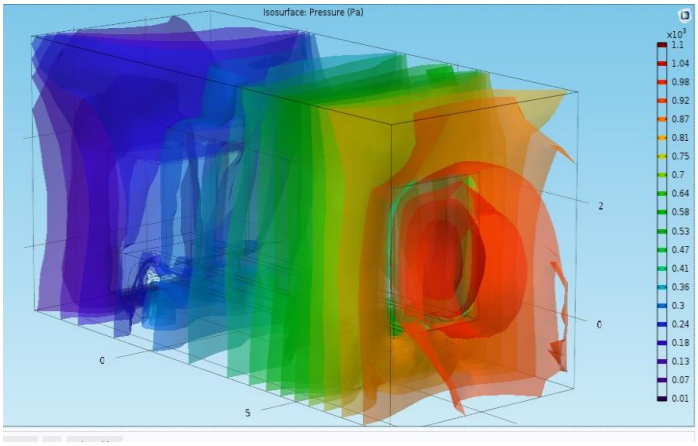


Fig. 14 Pressure on and around the truck (Rear view), (Pa)

In Fig.13 and Fig.14 are given 3D graphs in two views of the pressure distribution on the truck from the wind blow, in the form of isocurves. Maximal values of direct pressure (red and orange color) are concentrated on the rear of the truck, on the trailer. Pressure can reach maximal values $p_{max1} = 1100\ [Pa]$. The nature of pressure is turbulent in the rear of the trailer.

In the Fig.15 is the 2D graph of pressure acting on the back of the trailer on its edges. Its values are between *715 [Pa]* and *1000 [Pa]*.

In Fig. 16 is given the 3D graph of the Total stress acting on the truck from the rear wind. The highest values of stress is present in the rear part, on the trailer, rear axle and wheels. The max value of stress is $\sigma_{max} = 1.2 \cdot 10^3\ Pa$.

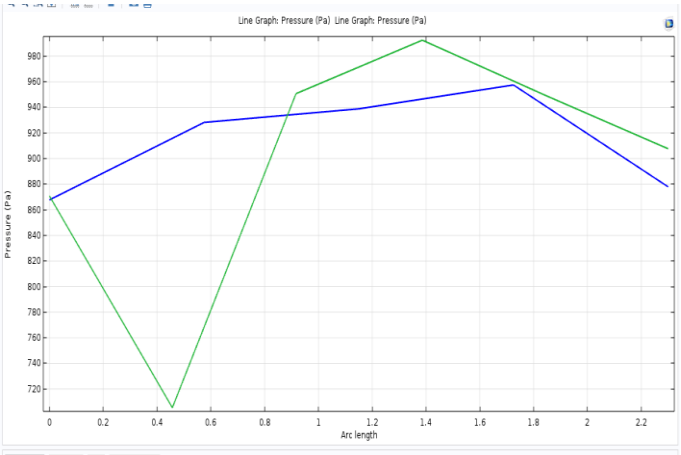


Fig.15 Pressure in the rear of the truck, on the trailer, resulting from rear wind

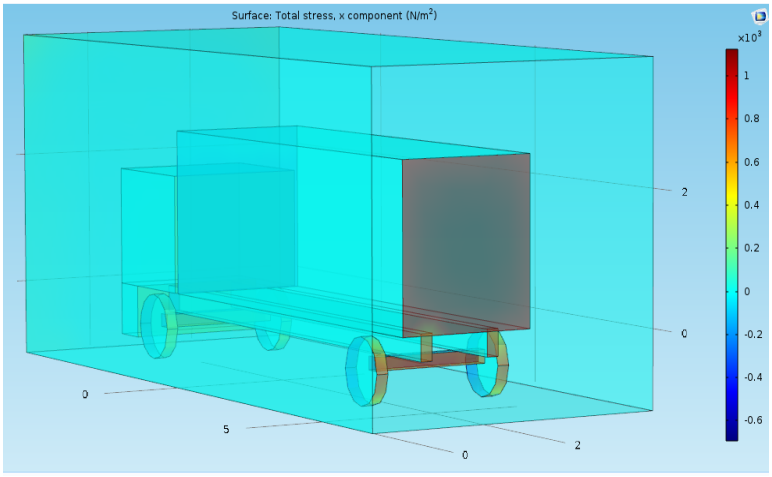


Fig.16 Total stress around the truck, rear view

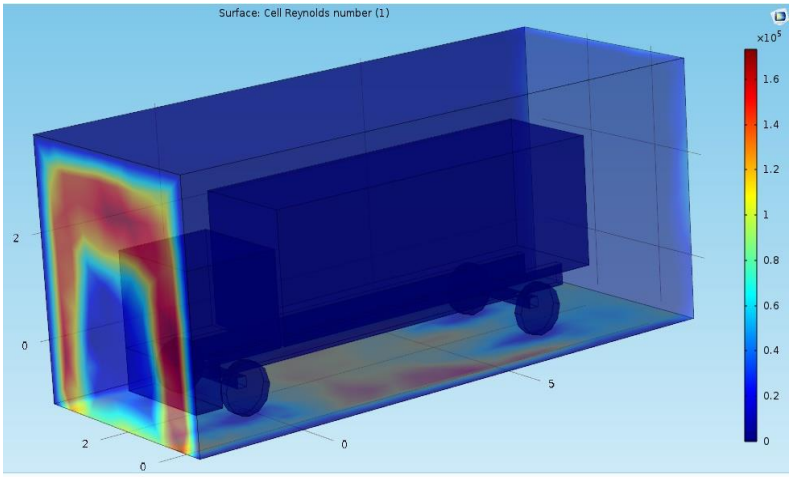


Fig. 17 Cell Reynolds number around the truck

In Fig. 17 is given the 3D graph of the Local Cell Reynolds number. It is calculated from the formula [4]:

$$Re^c = \rho \cdot u \cdot h/(2\nu) \quad (4)$$

ρ - local avg. mixture density,
u - local average cell velocity,
h - cell dimension,
2ν - local viscosity.

This Reynolds number is not identified with the character of the flow field, but with the stability of the numerical discretization.

Based on the results from Fig.16, the highest value of Cell Reynolds number is $Re^c = 1.75 \cdot 10^5$, in many parts around the truck (red color). Comparing with the numerical values from Chapter 2, this means that in many parts around the truck there is a moderate turbulent flow

7 Numerical analysis for the Turbulent flow regime and results

For the turbulent flow regime, initial aerodynamic parameters: wind velocity, atmospheric pressure level, compressibility, and ambient temperature are the same as in laminar flow. Difference is in the model type of wind flow. In this case turbulence model type is RANS and turbulence model is k-ω.

K-ω turbulence model approximates RANS equations (3), [13]. It is created on the solution of two modeled transport equations: one for the ω, which is the specific rate of dissipation of the turbulence kinetic energy *k* into internal thermal energy) and another for the turbulent kinetic energy *K* in the form of the one-equation models [14]. The typical form of the transport equation is:

$$\frac{\vartheta\omega}{\vartheta t} + \bar{u_\iota}\frac{\vartheta\omega}{\vartheta x_i} = -\gamma_1\frac{\omega}{\mathrm{K}}\tau_{ij}\frac{\vartheta\bar{u_\iota}}{\vartheta x_j} + \frac{\vartheta}{\vartheta x_i}\left(\frac{\nu_T}{\sigma_\omega}\frac{\vartheta\omega}{\vartheta x_i}\right) - \gamma_2\omega^2 + \nu\frac{\vartheta^2\omega}{\vartheta x_{i\vartheta x_j}} \tag{5}$$

where $\nu_T = \frac{\gamma^* K}{\omega}$, γ_1, γ_2, γ^*, σ_ω are constants.

The results are given in the form of 3D and 2D graphs. They present the influence of wind blowing in the turnulent regime on the back of the truck.

In the fig.18 is the 3D graph of velocity field and magnitude on and around the truck for the turbulent flow. Maximal values of the velocity are present in the front part and upper part of the truck. They can reach values up to $v_{max} = 40$ *m/s*, although the rear wind velocity is $v_{wg} = 16.31$ *m/s.*

Compared with the laminar flow, the velocity of the wind around the truck in turbulent flow regime is about 22% higher, and maximal influence is at the upper part of cabin and trailer.

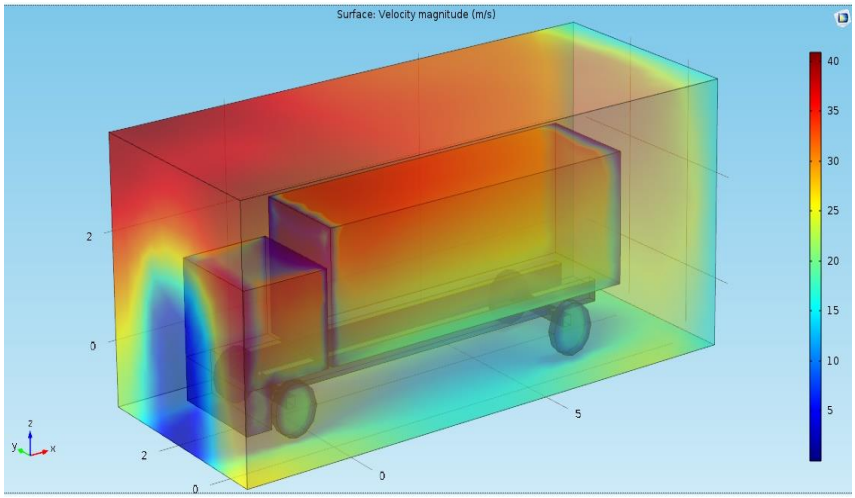


Fig. 18. Velocity field and magnitude around the truck, turbulent flow (m/s)

In fig.19 is the 3D graph of velocity vortex around the truck. Maximal values of the velocity vortex are present in the upper part and front of the cabin and trailer. This means that highest drag is around the cabin, which can lead to stability problems during driving [18],

[25]. Maximal values of velocity reach up to v_{max} = *36.8 m/s*, although the rear wind velocity is v_{wg} = *16.31 m/s*.

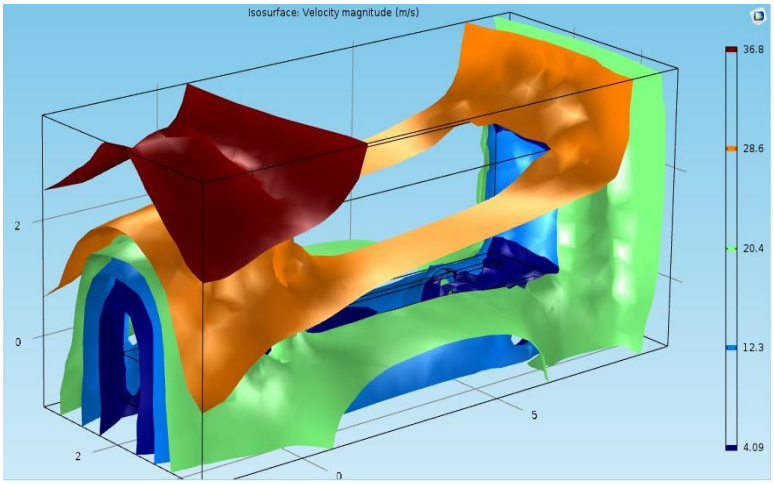


Fig. 19 Velocity vortex around the truck, (m/s)

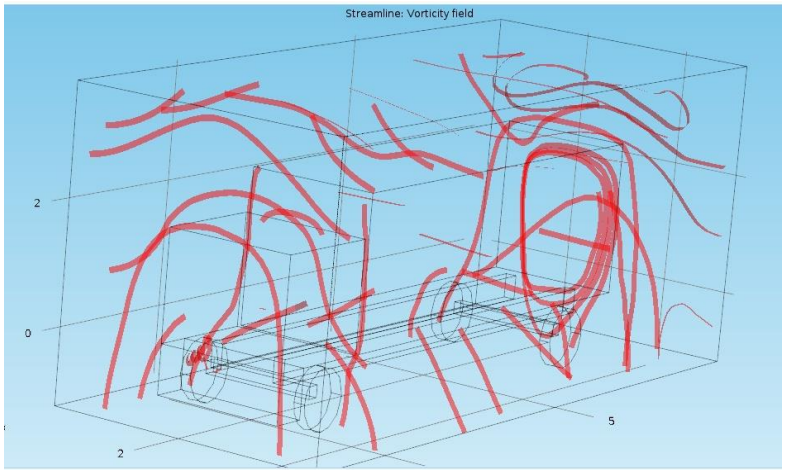


Fig. 20 Vorticity field around the truck, (1/s)

In Fig.20 is the 3D graph of vorticity field around the truck. The curves of vorticity represent the local spinning motion of air around the truck. Maximal vorticity curves (red and orange color) are present in the back and side parts of the truck.

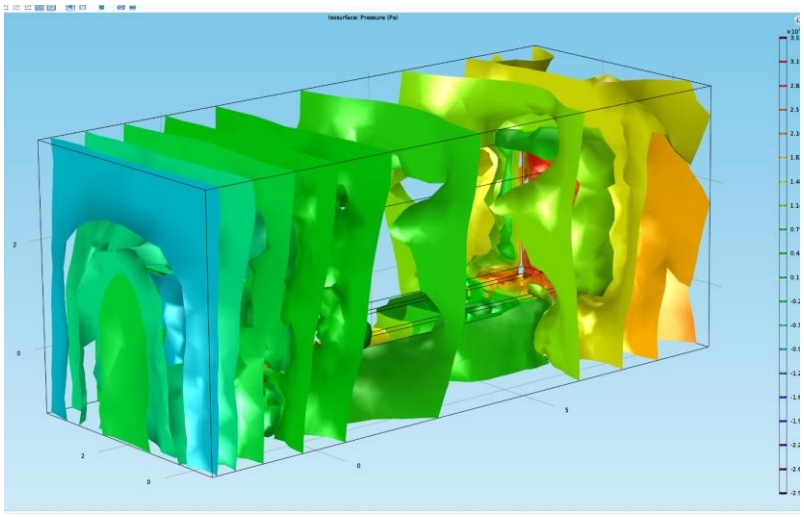


Fig. 21 Pressure on and around the truck (Side view), (Pa)

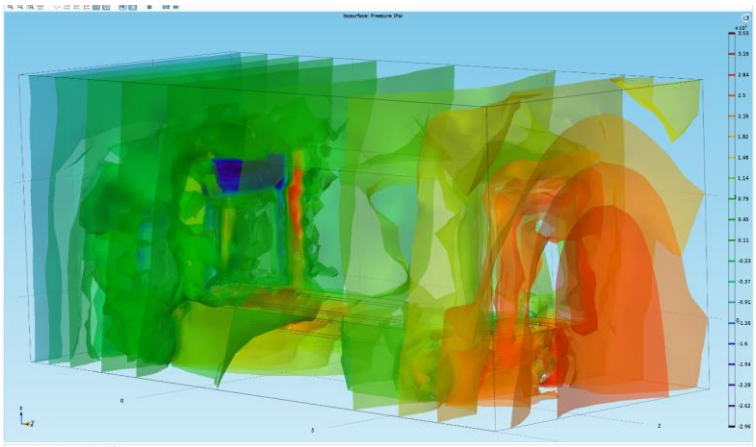


Fig. 22 Pressure on and around the truck (Rear view), (Pa)

In Fig.21 and Fig.22 are 3D graphs of pressure distribution from the wind blow on the truck, in the form of isocurves. Maximal values of direct pressure (positive values) are concentrated on the rear of the truck, on the trailer. Pressure can reach maximal values p_{maxt} = *35300 [Pa]*. On the other side, in the front side of the truck the pressure is low.

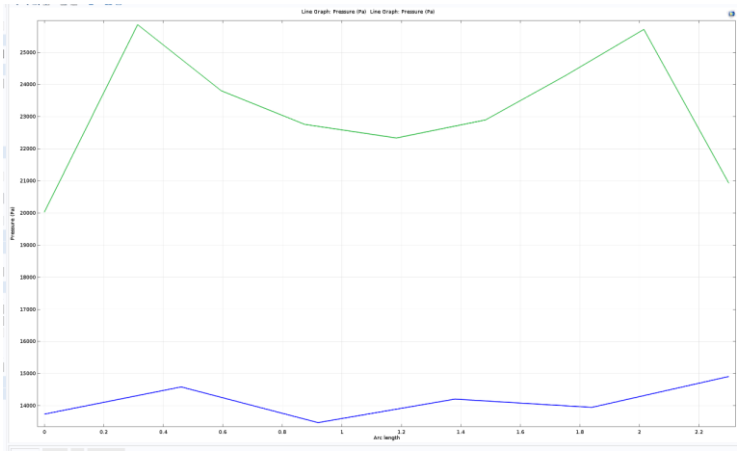


Fig.23. Pressure in the rear of the truck, on the trailer, resulting from rear wind

In the Fig.23 is the 2D graph of pressure acting on the back of the trailer on its edges during turbulent flow. Its values are between *13500 [Pa]* and *26800 [Pa]*. These values are about 24.3 times higher compared with the laminar flow, Fig.15.

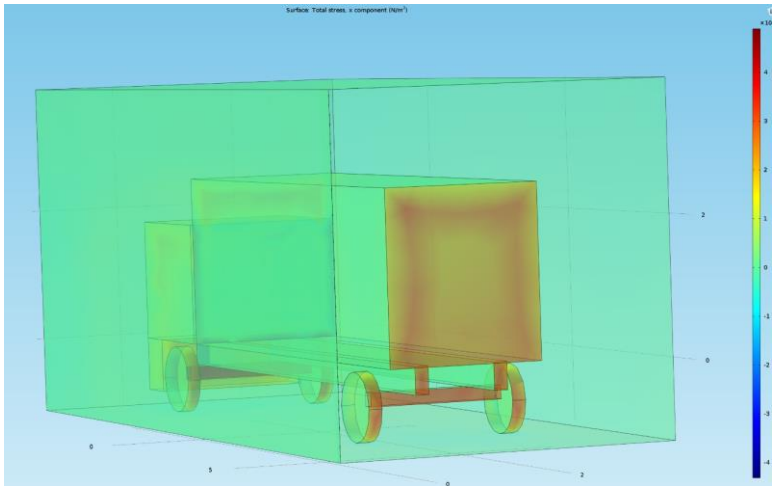


Fig. 24 Total stress around the truck, rear view

In Fig. 24 is given the 3D graph of the Total stress acting on the truck from the rear wind. It shows that the highest stress is present in the rear part, on the trailer, rear axle and wheels like in laminar flow. But here, highest values appear also at the front of the truck, in fron wheels and axle, and in some parts of cabin and front trailer. The max value of stress is $\sigma_{max} = 4.8 \cdot 10^4\ Pa$. These values are about 40 times higher compared with the laminar flow in Fig.16.

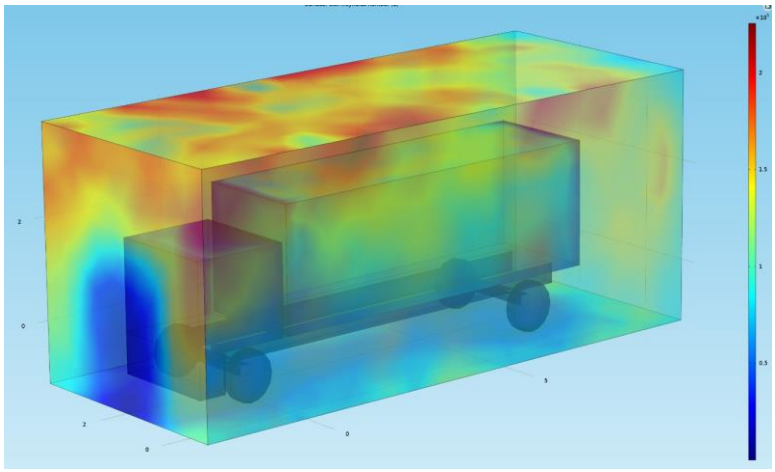


Fig. 25 Cell Reynolds number around the truck, turbulent flow

In Fig. 25 is given the 3D graph of the Local Cell Reynolds number parameter for turbulent flow. The highest value of Cell Reynolds number is $Re^c = 2.3 \cdot 10^5$, spread in many parts around and upper the truck (red color). Comparing with the laminar flow (fig.16), the maximal cell Reynolds number is 31% higher. Comparing with the numerical values from Chapter 3, the truck undergoes moderate turbulent flow.

CONCLUSIONS

Analysis of the wind flow around the truck is important for the safety during the travel. It is important to know how the rear wind influences the truck. This form of wind blow is not very well studied as a specific influence. It is believed that it helps in the speed of the vehicle, but that is not important for trucks, while they have speed limitations. Rear wind can produce negative effects, like instability during travel while trucks are big vehicles. The methodology of study in this paper is through numerical analysis of wind flow as a fluid flow using modelling and simulations with Comsol Multiphysics software. Previously, the velocity of rear wind is measured experimentally and implemented in simulations. There are two airflow regimes analysed: laminar flow and turbulent flow. Results are required for most influential aerodynamic parameters: wind velocity, pressure, vortex, and stress. Also, the Reynolds number was calculated to identify best the flow regime around the truck.

The laminar regime of rear wind flow studied in this case is most present during the year. The results given in 3D and 2D graphs tells us that in this flow regime there are some turbulent occurrences of airflow around the truck due to its form and big structure, which creates increased aerodynamic drag. For example, the highest values of velocity appear at the front and below the truck, at the cabin part, and they are almost twice higher compared to the initial rear wind velocity. Similarly, we conclude for vorticity and Cell Reynolds number. The results of pressure and stress shows highest values at the back of the truck. But, results of laminar flow are lower and less concerning, compared with those of turbulent regime.

The turbulent regime of wind flow studied in this case is less present during the year but must not be neglected due to the high magnitudes of aerodynamic parameters resulted. The results given in 3D and 2D graphs tells us that in this flow regime there are high turbulent

occurrences of airflow around the truck. The values are much higher than in the Laminar regime. For example, the highest values of velocity appear at the front and upper part the truck, and they are for 22% higher than those in Laminar regime. The highest values are for the pressure that appear at the rear of the truck. They are about 24.3 times higher compared with the laminar flow. This is a matter of concern! If the turbulent airflow is present, it can lead to traveling problems.

Conclusion is that rear wind affects the truck structure not only on the back of it, but also on sides and in the front. Rear wind influence is dynamic in nature, with oscillations in the form of irregular distribution. The magnitude of aerodynamic parameters is high. Besides the high pressure, the velocity of the airflow increases more than twice around the cabin compared to the initial wind velocity at the back of the truck. In this case, it will require from the driver to constantly correct the driving path on the road lane and control the driving speed.

REFERENCES

[1] White, Frank M. "Fluid mechanics, 7th ed.", McGraw-Hill series in mechanical engineering, **2011**.

[2] Driving tests, "Driving and riding in strong winds", https://www.drivingtests.co.nz/resources/driving-and-riding-in-strong-winds/

[3] "Mercedes Benz Antos 1840 L", https://roadstars.mercedes-benz-trucks.com/en_GB/events/2014/october/exhibits-distribution-transport.html

[4] "Comsol Multiphysics Help & User guide", COMSOL Multiphysics®,v.5.1, www.comsol.com.

[5] Joseph, H., Spurk, N. A. "Fluid Mechanics, Second Edition", Springer-Verlag Berlin, **2008**.

[6] John, D. Anderson Jr. "Computational Fluid Dynamics - The Basics With Applications", McGraw-Hill series in mechanical engineering, **1995**.

[7] Selvi Rajan, S., Santhoshkumar, M., Lakshmanan, N., Nadaraja Pillai, S. Paramasivam, M. "CFD Analysis and Wind Tunnel Experiment on a Typical Launch Vehicle Model", Tamkang Journal of Science and Engineering 12 (3), pp. 223, **2009**.

[8] Persson, T. "Wind Tunnel Effects on Truck Aerodynamics and Soiling", Master Thesis, Royal Institute of technology, Stockholm, **2017**.

[9] Hamir, R., Samuel, L., Jeremy, B. "Numerical Investigations of Air Flow Around a Model Freight Truck with a Rear Active Flow", Mineta Transportation Institute, **2019**.

[10] Shahrin, H. A. "Numerical analysis on the effects of air flow in the wake of a large vehicle on trailing a passenger car", Malaysian Technical Universities conference on engineering and technology, 28-29 June **2010**.

[11] Alonso-Estébanez, A., del Coz Díazb, J. J., Álvarez Rabanalb, F. P., Pascual Muñoza, P., García Nietoc, P. J. "Numerical investigation of truck aerodynamics on several classes of infrastructures", Wind and Structures 26 (1), **2018**.

[12] Moffat, J., "Aerodynamic Vehicle Design and Analysis", Blackpool and the Fylde College, **2016**.

[13] Giancarlo, A. "Reynolds-Averaged Navier–Stokes Equations for Turbulence Modeling", Applied Mechanics Reviews 62, **2009**.

[14] Chemeurope.com, “Reynolds-Averaged Navier–Stokes Equations”, https://www.chemeurope.com/en/encyclopedia/Reynolds-averaged_Navier-Stokes_equations.html#_note-0/

[15] Alshqirate, A., Ghafoor, D. Z., Borse, S. “CFD Analysis of Flow Over Pickup Truck with And Without Covering Cargo Area Using OpenFOAM”, International Journal of Fluid Mechanics & Thermal Sciences 90(1), pp. 40 – 51, **2022**.

[16] “Ambient Weather WS-2902A”, Smart Weather Station with WiFi Remote Monitoring and Alerts, https://www.ambientweather.com/amws2902.html

[17] “Weather Undergorund”, https://www.wunderground.com/dashboard/pws/IPRISHTI3/graph/2022-04-12/2022-04-12/monthly

[18] Magdolen, Ľ., Danko, J., Milesich, T., Kevický, I., Galinský, M., Bucha, J., Skyrčák, R. “Virtual Simulation of Overtaking Maneuver of Autonomous Vehicle“, Strojnícky časopis - Journal of Mechanical Engineering 71(2), pp. 179 – 188, **2021**. DOI: 10.2478/scjme-2021-0028

[19] Zhang, X., “Crosswind stability of vehicles under nonstationary wind excitation“, Dissertation, Karlsruher Institut für Technologie (KIT), April, **2015**.

[20] Guilmineau, E., Chometon, F. “Effect of Side Wind on a Simplified Car Model: Experimental and Numerical Analysis”, Journal of Fluids Engineering, **2009**.

[21] Kozmar, H., Butler, K., Kareem, A. “Transient cross-wind aerodynamic loads on a generic vehicle due to bora gusts”, Journal of Wind Engineering and Industrial Aerodynamics, **2012**.

[22] Martini, H. “Perspectives of Aerodynamic Drag and Cooling Airflow for Heavy-Duty Trucks - Reconsidering European Total-Length Legislation”, Phd Thesis, Chalmers University of Technology, Gothenburg, Sweden, **2016**.

[23] Bourhane, B., Kabar Y. “Numerical study of laminar forced convection of nanofluid in a new microchannel heat sink“, Strojnícky časopis – Journal of Mechanical Engineering 71(2), pp. 31 – 40, **2021**. DOI: 10.2478/scjme-2021-0015.

[24] Rescot, R., Jasrotia, R., Farlow, K., Lee, Y., Schrock, S. “Predicting and Mitigating Wind-Induced Truck Crashes on I-70 in Kansas”, Proceedings of the 2009 Mid-Continent Transportation Research Symposium, Ames, Iowa, 2009 by Iowa State University, **2009**.

[25] Hoxha, G., Shala, A., Likaj, R. “Vehicle Speed Determination in Case of Road Accident by Software Method and Comparing of Results with the Mathematical Model“, Strojnícky časopis – Journal of Mechanical Engineering 67(2), pp. 51 – 60, **2017**. DOI: 10.1515/scjme-2017-0017

[26] Gillespie, T. D. “Fundamentals of Vehicle Dynamics”, Society of Automotive Engineers, Inc., **1992**.