MOBILITY & VEHICLE MECHANICS

https://doi.org/10.24874/mvm.2023.49.02.04
UDC: 629.3.015.3;62-184;629.1-42

EXPERIMENTAL AND NUMERICAL STUDIES OF REDUCING DRAG FORCE A SEMI-TRAILER TRUCK MODEL USING THE CABIN SPOILER

Stjepan Galamboš [1], Dalibor Feher[2], Nenad Poznanović[3], Nebojša Nikolić[4], Dragan Ružić[5], Jovan Dorić[6]*

Received in August 2022 *Revised in September 2022* *Accepted in October 2022*
RESEARCH ARTICLE

ABSTRACT: The paper obtains the procedure of design and analysis of the geometric parameters of the aerodynamic spoiler on the cabin of a semi-trailer truck. The procedure was included in order to obtain the optimal geometric shape and position of the spoiler, which would reduce the drag force of a semi-trailer truck. The procedure of optimizing the geometric shape and position is based on Design of Experiments and Response Surface Methodology. The research includes physical experiments in a wind tunnel and Computational Fluid Dynamics simulation. A scaled model of a semi-trailer truck is used as object of the optimization process.

KEY WORDS: *CFD, optimization, HVAC, efficiency*

*[1]Stjepan Galamboš, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, galambos@uns.ac.rs, https://orcid.org/0000-0003-3491-2860 (*Corresponding author)*
[2] Dalibor Feher, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, daliborfeher@uns.ac.rs, https://orcid.org/0000-0001-7548-9697
[3] Nenad Poznanović, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, npoz@uns.ac.rs, https://orcid.org/ 0000-0003-1426-6953
[4] Nebojša Nikolić, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, nebnik@uns.ac.rs, https://orcid.org/ 0000-0003-4657-3640
[5] Dragan Ružić, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, ruzics@uns.ac.rs, https://orcid.org/ 0000-0002-3320-4750
[6] Jovan Dorić, University of Novi Sad, Faculty of Technical Sciences, Trg Dositeja Obradovića 6, 21000 Novi Sad, Serbia, jovan_d@uns.ac.rs, https://orcid.org/0000-0003-3491-2860

EKSPERIMENTALNA I NUMERIČKA ISTRAŽIVANJA SMANJENJA SILE UZGONA POMOĆU SPOJLERA KABINE NA MODELU TEGLJAČ-POLUPRIKOLICA

REZIME: U radu je prikazan postupak projektovanja i analize geometrijskih parametara aerodinamičkog spojlera na kabini poluprikolice. Procedura je imala za cilj dobijanje optimalnog geometrijskog oblika i položaja spojlera, čime bi se smanjila sila otpora poluprikolice. Procedura optimizacije geometrijskog oblika i položaja zasnovana je na dizajnu eksperimenata i metodologiji površine odgovora. Istraživanje uključuje fizičke eksperimente u aerotunelu i simulaciju računarskom dinamikom fluida. Objekat optimizacije je bio skalirani model tegljača-poluprikolice.

KLJUČNE REČI: *CFD, optimizacija, HVAC, efikasnost*

EXPERIMENTAL AND NUMERICAL STUDIES OF REDUCING DRAG FORCE A SEMI-TRAILER TRUCK MODEL USING THE CABIN SPOILER

Stjepan Galamboš, Dalibor Feher, Nenad Poznanović, Nebojša Nikolić, Dragan Ružić, Jovan Dorić

INTRODUCTION

The aerodynamics of a semi-trailer truck is an important scientific discipline which includes many aspects regarding the proper behaviour of vehicles during operation. Most often unfavourable aerodynamic shapes and large dimensions make a semi-trailer truck a reasonable test object. Also, a large number of kilometres travelled during exploitation, increases the need for improvement in the field of external aerodynamics. By reducing the aerodynamic drag on specific parts of the semi-trailer truck, as well as by changing the air flow, the overall aerodynamic drag of the semi-trailer truck will be reduced, which directly reflects on a lower fuel consumption. Filippone and Mohamed in their paper [1] indicate the importance of aerodynamics in long-distance commercial vehicles. In paper [1], the authors present a model for calculating the fuel consumption of commercial vehicles. Unfavourable geometric shapes of a semi-trailer truck and large air gaps between the elements of the body contribute to the creation of local aerodynamic drag in the form of vortex air flow. This phenomenon, as well as the definition of the areas in which the greatest local drag occurs, are presented by Tyrrell in [4] and Wood in [3]. Some simple solutions of aerodynamic spoilers of commercial vehicles and their analysis authors Wood and Bauer are presented in their work [2]. They achieved fuel savings of 10% at three different velocities. Computational Fluid Dynamics (CFD) is a useful tool for describing phenomena in the field of vehicle aerodynamics through simulations. The air flow analysis by CFD simulations in the area between the road and the vehicle floor is presented in [5] by Huminic A. and Huminic G. A 3D air flow simulation of was performed using a RANS (Reynolds Average Navier-Stokes) model for a larger spectrum of velocities and the values of Reynolds number between 2.4x106 and 14.1x106. In scientific field exists a large number of mathematical procedures and methods using to find an optimum in the problem to be solved. A presently optimization procedure, which leads to a good enough setting of the experiment, is the Design of Experiment (DoE). The procedure was used by McCallen et al. in [6], where they presented the analysis of existing as well as the creation of improved aerodynamic spoilers. The paper combines the use of CFD simulations and experimental measurements in a wind tunnel. Norouzi et al. in [7] were done a numerical research of medium-heavy trucks, for the purpose of reducing the force of aerodynamic drag for the values of Reynolds number between 7x105 and 1.6x106. The Finite Volume Method were used for simulate the flow field and pressure distribution around the truck. For the turbulent model, the standard k-epsilon was used to simulate the turbulent flow characteristics. Resourcing the relevant literature which include the optimization of aerodynamic spoilers of a semi-trailer truck, the authors of the paper found some interesting details as well as airfoil shape and position for the purpose of its application in an aerodynamic spoiler on a semi-trailer truck cabin. This motivated the authors to do a research in that direction. To that end, an optimization procedure has been devised, combining a few known methods. Firstly, a scaled CFD model of a semi-trailer truck without a spoiler has been created and verified in the wind tunnel by experiment. After that, the verified model has been used in the optimization procedure of airfoil parameters. The first step of the optimization procedure was defining airfoil parameters. The analysis was performed for all level combinations of parameter by using

Full Factorial DoE with impact to identify the zone of the optimal solution. The analysis implied adapting the parameter boundaries, as well as increasing the number of parameter levels. This used the Response Surface Methodology (RSM) based on Central Composite Design. By applying regression analysis on the RSM results, the response surface equation was formed. Finally, by mathematical minimizing the equation of response surface the optimal shape and position of the airfoil were found.

1. TESTING OBJECT

The testing object of the research is a semi-trailer truck. A 3D CAD model of truck and semi-trailer, scaled 1:10, was created, with some simplifications that do not have a direct issue on the aerodynamics. Figure 1 shows the CAD model of a startup configuration of a semi-trailer truck with dimensions.

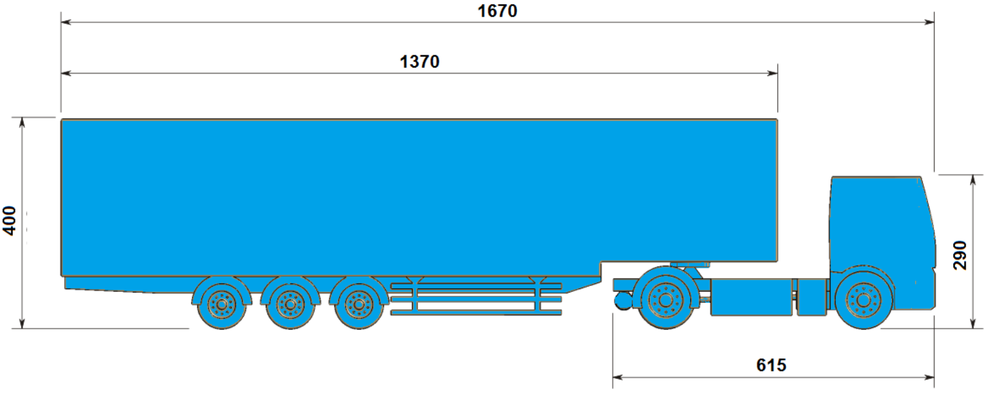


Figure 1. CAD model of the testing object with dimensions

In the reason of aerodynamics analysis, the CAD model was transformed to CFD (Computational Fluid Dynamics) model with adding virtual testing around which imitate a real wind tunnel. The model of the wind tunnel around the CAD model is a half cylinder 6000 mm long and radius of 700 mm. The CAD model has a frontal area of 0.0979 m^2. For better air flow spreading around the model and to minimise the impact of stationary floor, the CAD model was lifted up from the bottom of the wind tunnel for the value of 5 mm.

Some important properties within CFD model is shown in table 1.

Table 1. CFD important set-up

Parameter (or function)	Value (or explanation)
Software	AC Adapco - Star CCM+
Boundary conditions	Tunnel entrance - Velocity Inlet, Tunnel exit - Pressure Outlet, Tunnel and model - Wall
Inlet air flow velocity	Range between 60 and 90 km/h, with increment of 5 km/h
Reynolds number	Range between 1.8x106 and 2.7x107
Turbulent model	RANS k-Epsilon
Mesh type	Prismatic Polyhedral volumetric
Base cell size	100 mm for tunnel and 5 - 10 mm for all CAD model parts
Number of mesh cells	around 300,000

The values of base cell was adopted by mesh independency analysis, which included two mesh types (k-Epsilon and k-Omega) and a few values of base size (between 20 and 200 mm). Figure 2 shows CFD mesh model of the semi-trailer truck in the wind tunnel.

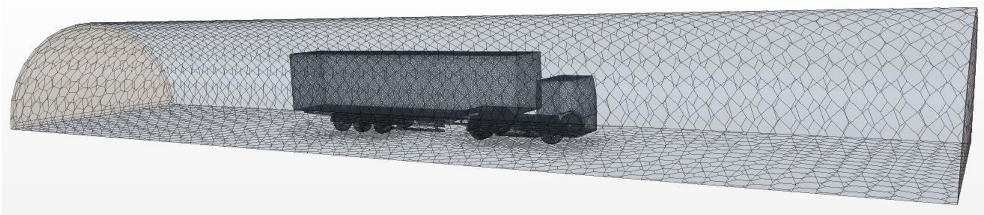


Figure 2. CFD mesh view of the testing model

After CFD set-up, the simulation was done for seven different velocity modes. Main outlook parameter was the value of model drag force. Table 2 shows values of drag force for seven air flow velocities.

Table 2. CFD initial results

Air flow velocity [km/h]	*Drag force [N]*
90	41.92
85	37.47
80	33.22
75	29.22
70	25.48
65	21.99
60	18.81

2. OPTIMIZATION PROCEDURE

The main purpose of this part of the research is to find optimal shape and position of a spoiler on the top of the truck cab. The optimization procedure includes a few steps. The initial spoiler had a shape of airfoil defined over four variable parameters and some constant parameters. Figure 3 shows the initial shape and position of the spoiler on the top of the truck cab.

2. 1 Factorial Design of Experiments

This part of the optimization procedure has aim to find best position and shape of the spoiler over CFD virtual simulations where was changed variable parameters H, L, R1 and R2. All simulations done for the highest air flow velocity of 90 km/h. The real variable parameter values were changed on the three levels, shown in the table 3. Four parameters with three levels gave 81 combinations of a different virtual CFD experiments.

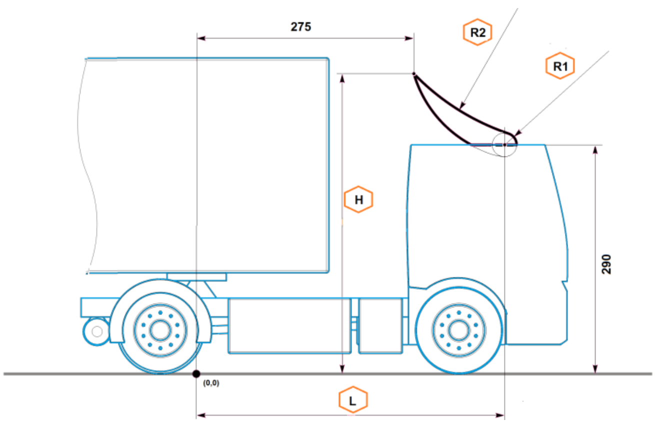


Figure 3. Initial position of the spoiler with parameters

Table 3. Full Factorial DoE values

	Values of parameters			
Level code	H [m]	L [m]	R1 [m]	R2 [m]
-1	0.35	0.35	0.010	0.1
0	0.39	0.38	0.015	0.3
1	0.42	0.40	0.020	0.5

Table 4 shows results of Full Factorial DoE. Combination number 16 gave the lowest drag force of 32.98 N. Also, Full Factorial DoE shows which level code of variable parameters has tendency for lower drag force.

Table 4. Full Factorial DoE results

Combination №	Factor level code				CFD simulation results	
	L	H	R1	R2	Drag coefficient, c_W [-]	Drag force, F_W [N]
1	-1	-1	-1	-1	0.942	34.02
2	-1	-1	-1	0	0.942	34.01
⋮	⋮				⋮	⋮
16	**-1**	**0**	**1**	**-1**	**0.913**	**32.98**
⋮	⋮				⋮	⋮
81	1	1	1	1	1.064	38.41

2. 2 Response Surface Method

In this part of the optimization procedure Full Factorial DoE is expended by using five-level Central Composite Design (CCD). Four parameters over five level code is 625 combinations of Full Factorial DoE, but thanks to CCD that number is reduced only on the 31 experiments. Table 5 shows the values of five level codes. Table 6 gives results of CCM within Response surface method. With this part of the optimization procedure the drag force was reduced to value of 31.93 N. The CCD yields 31 possible experimental settings, the last 7 of which are the central points of the design. The coefficients in the response surface function of aerodynamic drag force, Equation 1, are obtained by regression analysis of the dates from Table 6. The response surface coefficients are derived for uncoded parameter units.

Table 5. Level codes of parameters of CCM

Level code	L [m]	H [m]	R1 [m]	R2 [m]
-2	0.34	0.38	0.010	0.3
-1	0.35	0.39	0.015	0.4
0	0.36	0.40	0.020	0.5
1	0.37	0.41	0.025	0.6
2	0.38	0.42	0.030	0.7

Table 6. Results of Central Composite Design

Combination №	Factor level code				CFD simulation results		Combination №	Factor level code				CFD simulation results	
	L	H	R1	R2	Drag coefficient cw [-]	Drag force Fw [N]		L	H	R1	R2	Drag coefficient cw [-]	Drag force Fw [N]
1	-1	-1	-1	-1	0.945	34.11	17	-2	0	0	0	1.033	37.32
2	1	-1	-1	-1	0.943	34.06	18	2	0	0	0	0.991	35.78
3	-1	1	-1	-1	0.948	34.23	**19**	**0**	**-2**	**0**	**0**	**0.884**	**31.93**
4	1	1	-1	-1	0.936	33.79	20	0	2	0	0	0.896	32.36
5	-1	-1	1	-1	0.966	34.88	21	0	0	-2	0	0.897	32.41
6	1	-1	1	-1	0.917	33.10	22	0	0	2	0	0.918	33.15
7	-1	1	1	-1	0.970	35.03	23	0	0	0	-2	0.886	31.99
8	1	1	1	-1	0.901	32.55	24	0	0	0	2	0.905	32.69
9	-1	-1	-1	1	0.946	34.15	25	0	0	0	0	0.887	32.03
10	1	-1	-1	1	0.940	33.95	26	0	0	0	0	0.887	32.03
11	-1	1	-1	1	0.949	34.26	27	0	0	0	0	0.887	32.03
12	1	1	-1	1	0.926	33.45	28	0	0	0	0	0.887	32.03
13	-1	-1	1	1	0.962	34.74	29	0	0	0	0	0.887	32.03
14	1	-1	1	1	0.904	32.64	30	0	0	0	0	0.887	32.03
15	-1	1	1	1	0.977	35.26	31	0	0	0	0	0.887	32.03
16	1	1	1	1	0.908	32.79							

$$
\begin{aligned}
F_{Wth} &= 44.53 - 5.910 \cdot H - 0.561 \cdot L - 0.422 \cdot R1 - 0.968 \cdot R2 + 1.2114 \cdot H^2 + \\
&+ 0.1106 \cdot L^2 + 0.2688 \cdot R1^2 + 0.1599 \cdot R2^2 - 0.1291 \cdot H \cdot L - 0.4578 \cdot H \cdot R1 - \\
&- 0.0523 \cdot H \cdot R2 + 0.0513 \cdot L \cdot R1 + 0.0519 \cdot L \cdot R2 + 0.0159 \cdot R1 \cdot R2
\end{aligned} \tag{1}
$$

Presented equation describes the model behaviour with an accuracy of 95.8%. The parameter values that provide the minimum of the aerodynamic drag force response function are determined by numerical quasi-Newton minimization, Table 7.

By applying parameter values from the Table 7 to the CFD model, an aerodynamic drag force of 31.75 N was obtained. Figure 4 shows the optimized spoiler design.

Table 7.
Parameter values that provide the minimum aerodynamic drag

Parameter	Value [m]
H	0.3627
L	0.4002
R1	0.0210
R2	0.4912

Figure 4. Optimized spoiler on the top of the truck cab

3. EXPERIMENTAL PROCEDURE

The aim of the experimental testing is to verify the CFD simulations. The experimental testing was done in the "Miroslav Nenadović" wind tunnel within the Faculty of Mechanical Engineering in Belgrade, Serbia. That is a close type of underground wind tunnel with a circular air flow. The main dimensions of the testing area are 6000x2900x2100 mm. Maximum air velocity of around 400 km/h is reached by a four-blade fan with the motor electrical power of 210 kW.

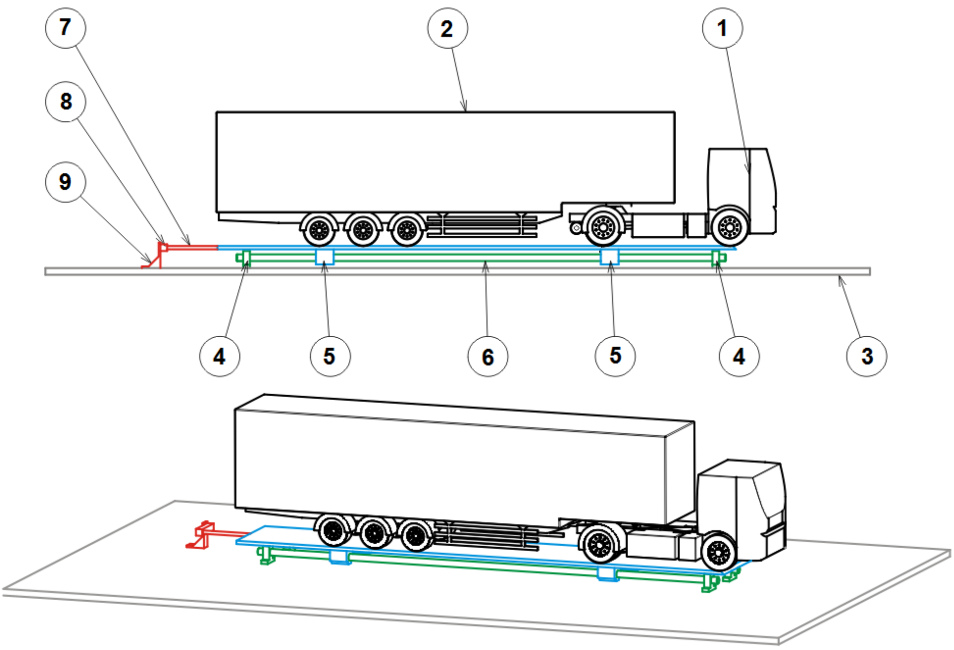


Figure 5. Experimental testing facility

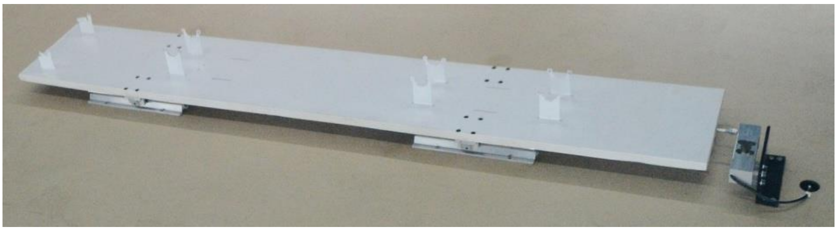
For the aim of measuring aerodynamic drag force, which is a horizontal component of vehicle resistance, an aerodynamic drag force measuring facility was built. Figure 5 shows a schematic representation of the facility for measuring the drag force of the semi-trailer truck model in a wind tunnel. This model, with dimensions of 1670x255x400 mm, is made of wood in a scale of 1:10 compared to the actual semi-trailer truck. The models of the truck 1 and the semi-trailer 2 are joined to the board 9 via a support. The models 1 and 2 are lifted up above the board for a vertical distance of 5 mm, same in CFD model. The board 9 is fastened via sliders 5 to sliding guides 6 through which the longitudinal translational movement of the board 9 is provided, together with models 1 and 2. The board 9 rests at the force measuring cell 8. The measuring cell CZL623B was used with the comprehensive full scale error 0.03% and rated output 2 ± 0.02 mV/V. The signal from the measuring cell is transferred to the universal measuring amplifier HBM QuantumX MX840A. Data acquisition was performed by the software HBM catman Easy -AP ver. 3.5.1.

Experimental measurements were performed on the following configurations of the measuring facility:

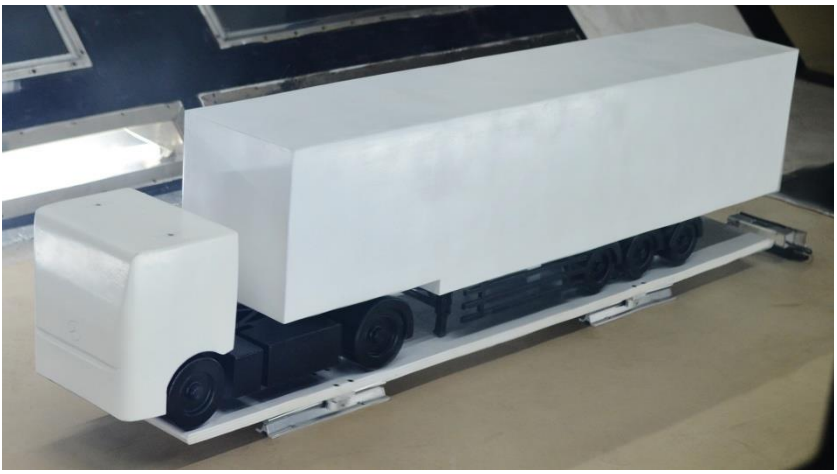
- drag force measurement of board 9 only, without the semi-trailer truck model.
- drag force measurement of the semi-trailer truck model without an aerodynamic spoiler;
- drag force measurement of the semi-trailer truck model with aerodynamic spoiler on the cabin of the truck.

Figure 6 shows the actual model configurations during measurements in the wind tunnel.

a)



b)



c)

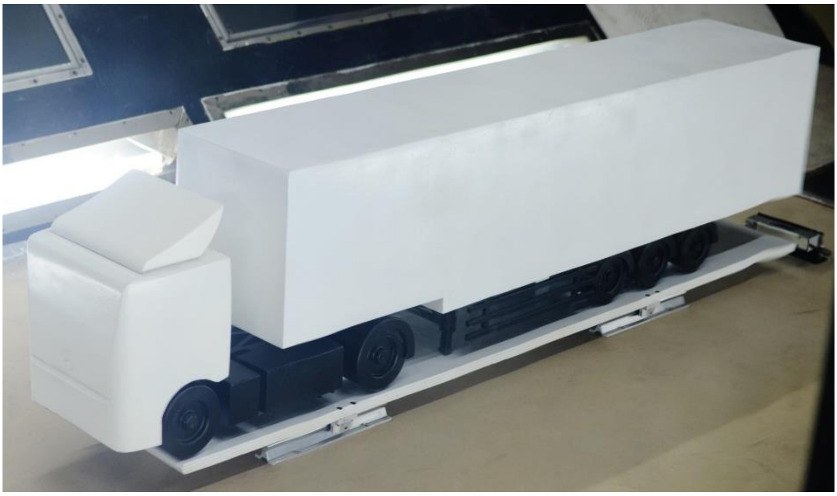


Figure 6. Testing models by three configurations in the wind tunnel

Within each measuring facility configuration, one measurement involved reading the aerodynamic drag force value at a single air flow velocity mode. Measurements were performed at seven velocity values in the range of 60 to 90 km/h, with an increment of 5 km/h. At each velocity mode, two measurements were performed for about 20 seconds.

4. RESULTS

Table 7 gives a comparative view of the results obtained by CFD simulations and experimentally for all three considered configurations and all velocity values. The obtained results in Table 7 show an acceptable agreement between the results achieved by CFD simulations and experimental tests in the wind tunnel. The adopted type of mesh and turbulent model within the CFD simulation provides good repeatability of the results for all considered airflow velocity values and all configurations of the measuring facility. The tested model during the experimental measurement showed stability and balance at the highest velocity values of air flow. The results of CFD simulations show that the value of aerodynamic drag force is higher than the corresponding one obtained experimentally by 1.5 to 3 N, at all the velocities. This deviation occurred because the CFD model did not take into account the friction force between sliders and their guide rails in the experimental facility. Due to the correct rounding design of the board leading edge and its distance in relation to the tested model, a very small aerodynamic drag force of the board without the model was observed, with a value of order of 2 N at the highest considered air flow velocity.

Table 7. Comparison results between simulations and experiments

Configuration of the measuring facility	v [km/h]	$F_{W\,CFD}$ [N]	$F_{W\,EXP\,1}$ [N]	$F_{W\,EXP\,2}$ [N]
a)	60	1.01	0.88	0.90
	65	1.18	1.04	1.04
	70	1.35	1.19	1.18
	75	1.55	1.36	1.37
	80	1.75	1.55	1.53
	85	1.97	1.53	1.77
	90	2.20	1.92	1.99
b)	60	18.81	16.44	16.78
	65	21.99	20.01	20.23

	70	25.48	23.13	23.59
	75	29.22	26.87	27.09
	80	33.22	30.79	30.88
	85	37.47	34.95	35.17
	90	41.92	38.23	38.68
c)	60	14.06	12.57	12.91
	65	16.48	15.10	15.42
	70	19.09	16.90	17.07
	75	21.88	19.70	19.89
	80	24.96	22.40	23.06
	85	28.13	25.64	26.13
	90	31.52	29.89	30.35

5. CONCLUSION

The goal of the research was to develop an optimizing procedure of the geometric shape and position, used as an aerodynamic spoiler on a semi-trailer truck cabin. The research included virtual experiments and physical experiments. The procedure was conceived as a combination of several known methods: Full Factorial Design of Experiments, Central Composite design, regression analysis and minimization of Response Surface equation.

Four parameters (top edge height from the ground - H, horizontal position of spoiler front radius centre - L, spoiler front edge radius - R1 and spoiler leading edge radius - R2) defining the spoiler shape and position. It turned out that parameter H has a dominant influence on the aerodynamic drag force. The influence of the parameter R2 is significantly smaller, while the parameters L and R1 have the least impact on the drag force.

As a result of Central Composite design of experiments, a quadratic response surface function of drag force was generated. Spoiler parameters are optimized by minimizing the response surface function. CFD simulation results were validated by testing in the wind tunnel.

The procedure presented in the research can also be performed by adding more parameters, including parameters that describe variations in the third dimension of the spoiler. Although such a research requires much more virtual experiments to be performed, it could lead to further improvement of truck aerodynamics.

ACKNOWLEGMENTS

This research has been supported by the Ministry of Education, Science and Technological Development through project no. 451-03-9/2021-14/ 200156: “Innovative scientific and artistic research from the FTS activity domain”.

REFERENCES

[1] Mohamed-Kassim, Z., & Filippone, A. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transportation Research Part D: Transport and Environment, 15(5), 275–284. https://doi.org/10.1016/j.trd.2010.02.010

[2] Wood, R. M., & Bauer, S. X. S. (2003). Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor -Trailer Trucks. SAE Transactions, 112, 143–160. JSTOR. https://doi.org/10.4271/2003 -01 -3377

[3] Wood, R. M. (2006). A discussion of a heavy truck advanced aerodynamic trailer system. Int. Symp. Heavy Veh. Weights Dimens., 9th, University Park, PA, 1–14.

[4] Tyrrell, C. L. (1987). Aerodynamics and Fuel Economy -On -Highway Experience (SAE Technical Paper No. 872278). SAE International. https://doi.org/10.4271/872278

[5] Huminic, A., & Huminic, G. (2009, April 20). CFD Study Concerning the Influence of the Underbody Components on Total Drag for a SUV. https://doi.org/10.4271/2009 -01 -1157

[6] McCallen, R., Salari, K., Ortega, J., Castellucci, P., Browand, F., Hammache, M., Hsu, T. -Y., Ross, J., Satran, D., Heineck, J. T., Walker, S., Yaste, D., DeChant, L., Hassan, B., Roy, C., Leonard, A., Rubel, M., Chatelain, P., Englar, R., & Pointer, D. (2004). DOE's Effort to Reduce Truck Aerodynamic Drag—Joint Experiments and Computations Lead to Smart Design. In 34th AIAA Fluid Dynamics Conference and Exhibit (Vol. 1–0). American Institute of Aeronautics and Astronautics. https://doi.org/10.2514/6.2004 -2249

[7] Norouzi, M., Pooladi, M. A., & Mahmoudi, M. (2016). Numerical investigation of drag reduction in a Class 5 medium duty truck. Journal of Mechanical Engineering and Sciences, 10, 2387–2400. https://doi.org/10.15282/jmes.10.3.2016.15.0221