

# COMPUTER SIMULATION OF CAR AERODYNAMIC PROPERTIES

Ivan Janoško<sup>1</sup>, Tomáš Polonec<sup>1</sup>, Peter Kuchar<sup>1</sup>, Pavel Máchal<sup>2</sup>, Martin Zach<sup>3</sup>

<sup>1</sup>Department of Transport and Handling, Faculty of Engineering, Slovak University of Agriculture in Nitra, Tr. A. Hlinku 2, 94976 Nitra, Slovak Republic

<sup>2</sup>Department of Project Management, Faculty of Regional Development and International Studies, Mendel University in Brno, Zemědělská 1/1665, 613 00 Brno, Czech Republic

<sup>3</sup>Expert Engineering Department, Institute of Lifelong Learning, Mendel University in Brno, Zemědělská 1/1665, 613 00 Brno, Czech Republic

## Abstract

JANOŠKO IVAN, POLONEC TOMÁŠ, KUCHAR PETER, MÁCHAL PAVEL, ZACH MARTIN. 2017. Computer Simulation of Car Aerodynamic Properties. *Acta Universitatis Agriculturae et Silviculturae Mendelianae Brunensis*, 65(5): 1505–1514.

The paper presents an optimization of car aerodynamic properties using the simulation software STAR-CCM+. For real simulation was used tested car Fiat 127 which was modified on the performance car. The basic objective of this work is using computer simulations to obtain knowledge about the impact of individual body parts on the results of aerodynamic drag, downforce or lift. Based on the results, bodywork modifications will be designed to improve the aerodynamic characteristics of the body, but would not disrupt the basic shape and appearance of the vehicle. The modifications will be again subjected to tests in simulation software. On the modified body was significantly reduced torque of the front axle, while increased of rear axle (cca 1250 N.m). This caused a significant stabilizing effect on the rear axle. The results of simulation tests before and after use bodywork modifications are processed in graphical and numerical form.

Keywords: aerodynamic resistance, lifting force, car stability, car modification.

## INTRODUCTION

Vehicle aerodynamics, whether conventional or sport cars, is very important. The main reasons why vehicle manufacturers around the world deal with aerodynamics are mainly reduced consumption, increased performance, improved vehicle stability and ultimately reduce the noise level when the vehicle is traveling. It also has a big impact on his handling, maneuverability and the ability to transmit traction and braking forces to the road at high speeds (Sloboda *et al.*, 2008). Since the detection of aerodynamic characteristics using wind tunnels is very expensive, based on theories and knowledge gained in measurements have been developed simulation software with mathematical models which allow to calculate, with some accuracy, the behavior of the flowing medium in a given area, as well as its effects on any object placed in this space. Using this software could apply calculations also to complex and large objects (e.g. vehicle). Calculations require very powerful computers with

multi-core processor. The calculation time can range from a few hours to several days. Even though these processes are time and computationally consuming, virtual simulations are much cheaper than the real model tests in a wind tunnel (Vančura *et al.*, 2010).

Currently there are several specialized computer programs oriented in fluid dynamics (inter alia) know as CFD. Computational Fluid Dynamics (CFD) presents an analysis of the flow, heat transfer and associative phenomena as chemical reactions by using computer simulation. This technique is very powerful and represents a wide range of applications in industrial and non-industrial applications. There is only few good enough CFD software that can satisfy demanding simulations of aerodynamic of the vehicle. Currently, most of the organizations use Fluent, Autodesk CFD, STAR-CCM + and PowerFlow.

Program ANSYS Fluent, thanks to its wide application in the analysis of fluid flow, heat exchange and mass transfer and chemical reactions

is one of the most common and most widely used commercial CFD programs. The basis of the ANSYS Workbench environment menu from which can be selected systems and by correct selection obtain the desired results (ANSYS, 2016).

Autodesk Simulation CFD software provides fast, accurate and flexible fluid flow which helps predict product performance, optimize design and validate product behavior before manufacturing. Minimizing reliance on costly physical prototypes and getting innovative products to market faster. The program is not directly designed to simulate the aerodynamics of cars. It is almost an adjunct to modeling programs to verify product characteristics (Autodesk, 2016).

PowerFlow is CFD simulating program from Exo company. It imports fully complex model geometry and accurately and efficiently performs aerodynamic, aeroacoustic and thermal management simulations. The biggest advantage of the program consists in calculating the aeroacoustic. This include for example underbody wind noise, gap noise, mirror, whistle and tonal noise, sunroof

and window buffeting, pass-by, noise and cooling fan noise (Exa Corporation, 2016).

Program STAR-CCM+ from CD-adapco company is one of the most used software to aerodynamics simulations. STAR-CCM+ is designed to entirely automate the simulation workflow and perform iterative design studies with minimal user interaction. STAR-CCM+ is developed to be accurate, efficient, easy to use and user friendly (CD-Adapco, 2009).

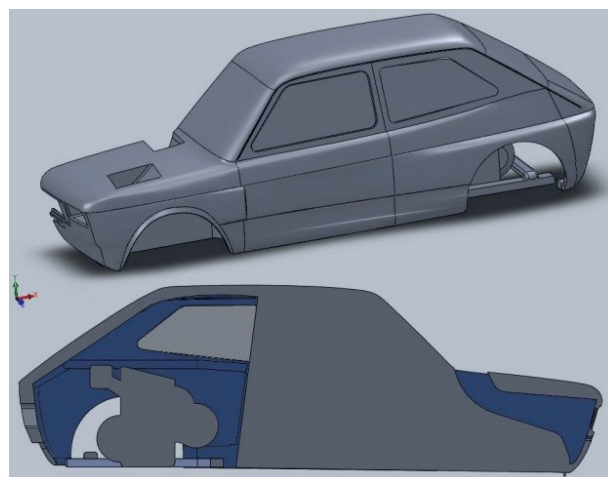
The paper aimed to assess the design and aerodynamics modification to the bodywork for the race car FIAT 127 apcially modified (motor power over 310 kW) for accelerating dragster race car at a distance 200 and 400 m. Competing car showed a significant directional instability at higher speeds (about 180–200km.h<sup>-1</sup>) tending to swim along the way and rotate about Z axis and becomes uncontrolled skid.

## MATERIALS AND METHODS

Researched object was a model of experiment vehicle Fiat 127 (Fig. 1). Body of vehicle was



1: Modified race car FIAT 127 equipped with motor Lancia 2.0 16V, 310 kW  
Source: own



2: Computer model of left side of car body  
Source: own

hatchback type with a negatively sloping front face. 3D computer model of the vehicle was created by technique of projection views on plains and the consequent creation of spatial curves. Combining the spatial curves were created outside surfaces of the bodywork. Adding a volume to surfaces create a solid model, in which was created inner mudguards, engine space and various vents, which could be also found on the real vehicle. In order to monitor the flow into the engine compartment and in various inner body points, a model of vehicle's left side was created (Fig. 2). Since the vehicle is completely symmetrical, there was no need to do simulation testing also for the other half of the body.

Computer model of testing vehicle was created in 1:1 ratio. The test environment consisted of a tunnel with dimensions  $5 \times 5.5 \times 55$  m. Before meshing, the body and the environment were inverted, so that the volume of the body was cavity and the air around and inside the car became a volume (Fig. 3). Inverted model was then imported into the simulation program STAR-CCM+.

To compile the simulation was necessary to enter environmental characteristics, test conditions and define the properties of objects into the program (Tab. I). Body was divided to individual parts (e.g. hood, roof, etc.), whose properties were meant to be examined. The next step was meshing model, thus creating a network of polygons. Computer model of the vehicle and wind tunnel was created in CAD computer software SolidWorks 2010. To simulate the action of aerodynamic effects on the body was used computer program STAR-CCM+.

The simulation was carried out on a computer with 12 core processors.

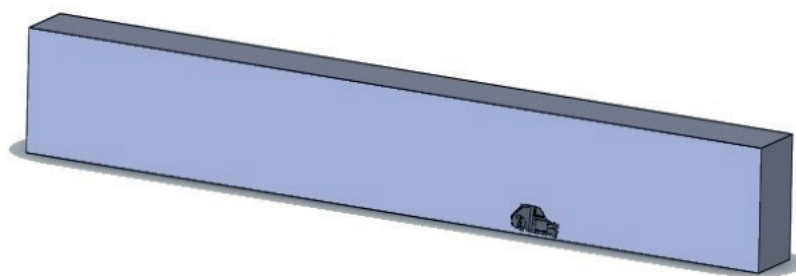
For a calculation of local air flow speed in any point, could be applied modified Bernoulli's equation (1). For these calculations is necessary to know value of ambient air density, speed of ambient air (vehicle speed), pressure of ambient air (atmospheric pressure) and static pressure in measured point. From this equation (1) is possible to see that, as the air flows faster around the vehicle, pressure will be reduced.

$$\begin{aligned} \frac{p}{\rho} + \frac{v^2}{2} = \text{const.} &\Rightarrow \frac{p_x}{\rho} + \frac{v_x^2}{2} = \\ &= \frac{p_{amb}}{\rho} + \frac{v_{amb}^2}{2} \Rightarrow v_x = \\ &= \sqrt{\left( \frac{p_{amb}}{\rho} + \frac{v_{amb}^2}{2} - \frac{p_x}{\rho} \right) \cdot 2} \end{aligned} \quad (1)$$

where:

$p_x$  ..... static pressure in measured point [Pa]  
 $p_{amb}$  ..... pressure of ambient (atmospheric pressure) [Pa]  
 $v_x$  ..... local speed in measured point [ $\text{m.s}^{-1}$ ]  
 $v_{amb}$  ..... speed of ambient air (vehicle speed) [ $\text{m.s}^{-1}$ ]  
 $\rho$  ..... air density [ $\text{kg.m}^{-3}$ ]

To present a data of drag and lift in non-dimensional form, the measured forces must be divided by the square of the velocity, air density and the cross-sectional area of the vehicle. This will define the drag and lift coefficients. For a calculation of aerodynamic drag coefficient can be used



3: Virtual wind tunnel with car model  
Source: own

I: Properties of environment (air) sets in simulation software

	Value	Unit
Air speed (vehicle speed)	52.778	$\text{m.s}^{-1}$
Density of environment (air density)	1.22	$\text{kg.m}^{-3}$
Temperature of environment	20	$^{\circ}\text{C}$
Viscosity of environment	$1.8 \cdot 10^{-5}$	$\text{N.s.m}^{-2}$
Atmospheric pressure	101.325	kPa

equation (2). Aerodynamic lift coefficient could be calculated by equation (3).

$$C_X = \frac{F_x}{\frac{1}{2} \cdot \rho \cdot v^2 \cdot S} \quad (2)$$

where:

$C_X$ .....drag coefficient [-]

$F_x$ .....aerodynamic drag force [N]

$\rho$ .....air density [ $\text{kg} \cdot \text{m}^{-3}$ ]

$v$ .....vehicle speed [ $\text{m} \cdot \text{s}^{-1}$ ]

$S$ .....frontal area of vehicle [ $\text{m}^2$ ]

$$C_Y = \frac{F_y}{\frac{1}{2} \cdot \rho \cdot v^2 \cdot S} \quad (3)$$

where:

$C_Y$ .....lift coefficient [-]

$F_y$ .....aerodynamic lift force [N]

$\rho$ .....air density [ $\text{kg} \cdot \text{m}^{-3}$ ]

$v$ .....vehicle speed [ $\text{m} \cdot \text{s}^{-1}$ ]

$S$ .....frontal area of vehicle [ $\text{m}^2$ ]

STAR-CCM+ simulation software calculates the amount of the forces and torques in all three axis X, Y, Z. STAR-CCM+ also allows graphical display of streamlines, flow velocities, and the pressure on various parts of the model. Calculated amounts of forces which acting on individual body parts are shown in Tab. II.

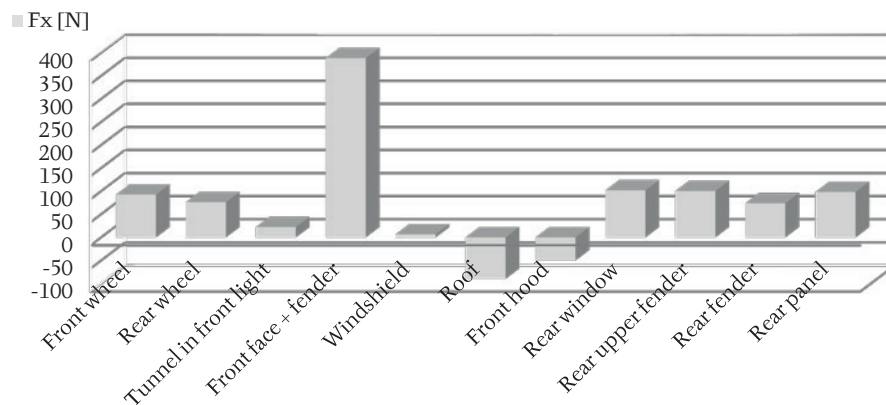
The amount of force “ $F_x$ ” acting on various parts of the body is graphically illustrated in Fig. 4. Another simulation output was the amount of a numerical representation of lift or downforce ( $F_y$ ) acting on various parts of car. Graphical illustration of force “ $F_y$ ” is shown on Fig. 5. Negative values mean the opposite direction of the force action. Lift force with a negative value, is referred as the downforce.

Directional vectors of acting forces and torques:

- Direction of force  $F_x$  – horizontally to the road plane, positive value means direction from the front to the rear of the vehicle,
- Direction of force  $F_y$  – perpendicular to the plane of the road, positive value means direction from bottom to top. Negative values mean the opposite

II: Calculated X, Y, Z aerodynamic forces for selected parts of car; Source: own

	Half car model			Full car model		
	$F_x/2$ [N]	$F_y/2$ [N]	$F_z/2$ [N]	$F_x$ [N]	$F_y$ [N]	$F_z$ [N]
<b>Floor</b>		-586.69			-1173.38	
<b>Front wheel</b>	46.43	37.74		92.85	75.49	
<b>Rear wheel</b>	38.05	18.20		76.10	36.39	
<b>Tunnel in front light</b>	11.21	-8.99		22.42	-17.97	
<b>Front face + fender</b>	194.29	79.44	90.92	388.57	158.88	181.84
<b>Windshield</b>	3.38	5.24		6.76	10.48	
<b>Roof</b>	-44.75	777.30		-89.49	1554.60	
<b>Front hood</b>	-24.88	70.13		-49.76	140.27	
<b>Rear window</b>	50.93	76.10		101.85	152.20	
<b>Rear upper fender</b>	50.18	109.48		100.37	218.97	
<b>Rear fender</b>	36.90	13.89		73.79	27.78	
<b>Rear panel</b>	49.22	-13.22		98.44	-26.44	
<b>Complete car</b>	<b>610.10</b>	<b>93.49</b>		<b>1220.20</b>	<b>186.97</b>	



4: The amount of force  $F_x$  acting on individual parts of stock car  
Source: own

direction of the force action. Lift force with a negative value, is referred as the downforce,

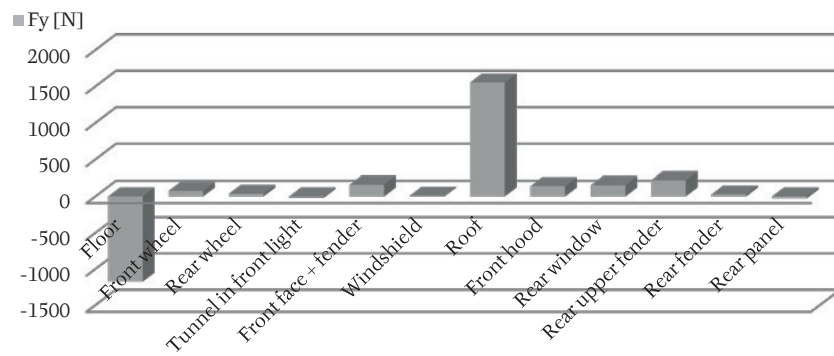
- Direction of force  $F_z$  – horizontally to the road plane, positive value means direction from centre to the left side of vehicle,
- Torque  $M_x$  – axis of rotation parallel to the axis “z”, positive value means counterclockwise direction (Katz, 2006).

A very important output of performed simulations was also express of value and direction of torques, generated by aerodynamic forces, acting on the individual axles. The center of rotation of these torques are in the middle of the axis of rotation of the front and rear axles. Calculated effects of forces  $F_y$  acting on individual vehicles parts and consequently of the torque applied to the vehicles axles, implicit that torque has a positive sense and loads the front axle more, causing the rear axle to be lifted at high speed.

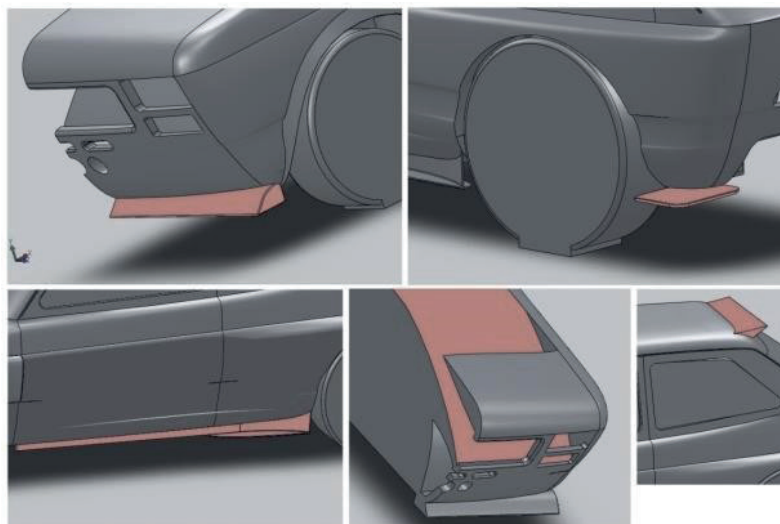
According to the results of the simulations were selected body parts which are most involved in the formation of undesirable aerodynamic properties of the vehicle.

In terms of aerodynamic drag (force “ $F_x$ ”) it can be seen that on the stock body was higher value of drag at the front face and the front fender, followed by rear window, rear panel and the top of the rear fender. On the contrary, on the roof and front hood has force “ $F_x$ ” opposite direction. On individual parts of the body also acts downforce or lift force “ $F_y$ ”. If the force “ $F_y$ ” is positive, it is referred as a lift force and consider it undesirable because it lifting vehicle, which impairs traction between tire and road. If the force “ $F_y$ ” has negative direction, it is referred as the downforce and in our case it is considered as desirable because it improves tire contact with the road. From the simulation results it can be seen that the stock car body has a largest lift force on the roof of the vehicle. Much smaller lift force is applied to almost all parts of the body except the floor. Floor of the vehicle is only one monitored place where the force “ $F_y$ ” has negative direction and helps to reduce the overall lift of the vehicle.

Results of simulations were evaluated and modifications of the body were proposed to improve the aerodynamic characteristics of the vehicle. To improve the aerodynamic characteristics of vehicle,



5: The amount of force  $F_y$  acting on individual parts of stock car  
Source: own



6: Modified 3D model of car body (light color show modifications)  
Source: own

reduction of the lifting force ( $F_y$ ) in problem areas of the body, while maintaining or only a slight increase of drag force ( $F_x$ ) has been tried. At the same time, it was aimed to increase opposite torque  $M_{z2}$  on rear axle to prevent the unloading of the rear axle at high speeds.

To improve the aerodynamic characteristics of the vehicle had been proposed these modifications:

- Increasing the downforce on the floor of the vehicle by means of reducing the volume of air flowing between the body and the road,
- Reducing the aerodynamic drag of the front face and the front part of the body (decrease  $c_x$ ),
- Reduction of lift force acting on the roof of the vehicle by improving airflow around the body,
- Increasing the down force acting on the rear axle,
- Restrict ingress of air from the sides of the vehicle to the space between the vehicle floor and the road,
- Reducing the impact of rotation of the wheels on the air flow around the body.

Any used modifications subject to the condition that they must not radically change the outer contour and the basic shape of the body. Based on the proposed modifications was created a modified three-dimensional computer model of the body. On Fig. 6 are (by light color) shown modifications that were made on the car body.

## RESULTS AND DISCUSSION

Modified three-dimensional computer model of the car with applied the proposed solutions was subjected to simulations in the program STAR-CCM+ under the same test conditions as the stock model of car. The calculated values of the forces acting on the individual parts of the vehicle model, before and after modification are shown in Tab. III. At Fig. 7 is shown a graphical comparison of the size of the force “ $F_x$ ” on the individual vehicle parts for stock and modified version of the body. Another output was the comparison forces “ $F_y$ ” acting on individual parts of the stock and modified body (Fig. 8). The modifications of body brought with it a change in the value of the frontal area of the vehicle and also change the aerodynamic coefficients (Tab. IV). Tab. V shows a comparison of the calculated values of the moments acting around the axis Z. The center of rotation of moment is placed in axe of the rear axle.

Graphical comparison of trajectories of streamlines and color resolution of airflow speeds around the stock and modified body are shown in Fig. 9.

The simulation program has also provided the possibility of graphical representation of air pressure acting on the individual points of the car model. This graphical representation is determining in which areas of bodywork occurs negative pressure or positive pressure. In Fig. 10 can be seen pressure

III: Aerodynamic forces acting on selected parts of car body before and after the modification (values are valid for half of a car body)  
Source: own

	Stock car			Modified car		
	$F_x$ [N]	$F_y$ [N]	$F_z$ [N]	$F_x$ [N]	$F_y$ [N]	$F_z$ [N]
<b>Floor</b>		-586.69			-619.86	
<b>Front wheel</b>	46.43	37.74		33.32	42.52	
<b>Rear wheel</b>	38.05	18.20		27.85	30.58	
<b>Front part of hood</b>	x	x		-112.75	284.15	
<b>Front spoiler</b>	x	x		58.33	-131.95	
<b>Rear roof spoiler</b>	x	x		8.28	-12.48	
<b>Tunnel in front light</b>	11.21	-8.99		x	x	
<b>Front face + fender</b>	194.29	79.44	90.92	194.34	39.40	100.41
<b>Windshield</b>	3.38	5.24		21.73	-12.17	
<b>Roof</b>	-44.75	777.30		-77.15	565.77	
<b>Front hood</b>	-24.88	70.13		x	x	
<b>Front hood (modified)</b>	x	x		62.08	42.59	
<b>Rear window</b>	50.93	76.10		51.58	77.41	
<b>Rear upper fender</b>	50.18	109.48		41.13	63.17	
<b>Rear fender</b>	36.90	13.89		36.79	13.79	
<b>Rear panel</b>	49.22	-13.22		105.92	47.75	
<b>Rear wheel spoiler</b>	x	x		1.21	-4.82	
<b>Side skirt</b>	x	x		1.79	-2.45	
<b>Rear spoiler on side skirt</b>	x	x		0.40	-8.32	
<b>Total (1/2 of the car)</b>	<b>610.10</b>	<b>93.49</b>		<b>614.41</b>	<b>-251.59</b>	

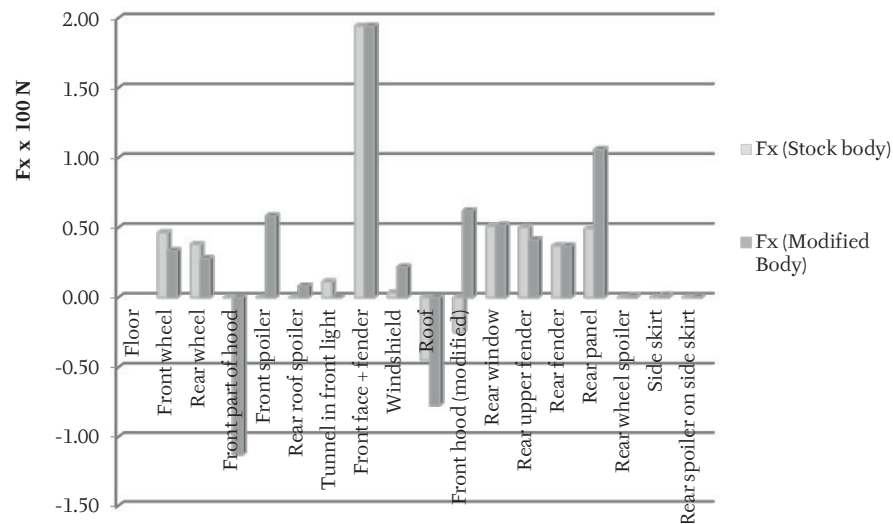
## IV: Comparison of aerodynamic coefficients and frontal area

	Stock car	Modified car
Aerodynamic drag coefficient, $C_x$	0.4251	0.4197
Aerodynamic lift coefficient, $C_y$	0.0651	-0.1719
Frontal area of car [ $m^2$ ]	1.7467	1.7815

Source: own

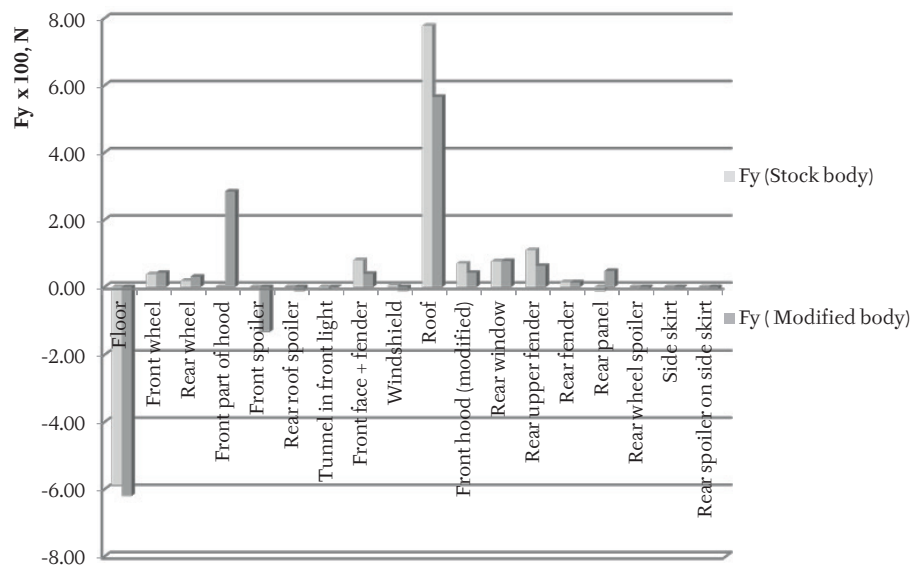
## V: Comparison of torques generated by aerodynamic forces

Moment	Stock car	Modified car
$M_{z2}$ (rear axle) [Nm]	286.61	1251.15



7: Comparison of "Fx" forces acting on selected parts of stock and modified car

Source: own



8: Comparison of "Fy" forces acting on selected parts of stock and modified car

Source: own

distribution on stock car (top) and modified car (below). With deep red color are marked areas with zero relative pressure, i.e. the pressure same as atmospheric. In areas where the color is colder, pressure is lower than atmospheric. A gray and white color shows the places where the pressure is greater than atmospheric.

During the solving of the aerodynamic problems of racing car Fiat 127 has been designed and tested several solutions to improve stability. Selected modifications, which have based on simulations positive-going impact on this parameter projected are listed in the paper. Knowledge and assessments resulting from the analysis of the aerodynamic properties of the vehicle body of the program STAR-CCM+, have been used in real conditions. It was mainly performance by increasing downforce  $F_y$ . The proposed modifications were implemented in practice, on the vehicle and thoroughly tested on the race track. Car really showed improved longitudinal and lateral stability, which provide better maneuverability and greater safety of the vehicle.

The results of computer simulations are confirmed and discussed by basic theory and practice tests. Generally, air resistance is the result of normal air pressures on body surface and friction forces acting in the tangential direction of air flow around the bodywork. The undesired pressure above and below the vehicle (high vacuum under the vehicle, slight overpressure under the vehicle) causes the air to swirl in the transverse plane of the vehicle (so-called induced air resistance), which is another component of the overall air resistance (Vlk, 2001).

Airborne resistors are also included in the overall air resistance of the vehicle as a result of the passage of air through the cooler and the ventilation system, and the resistances produced by swirling and frictioning air on the rotating wheels (Vlk, 2001, Sloboda *et al.*, 2008).

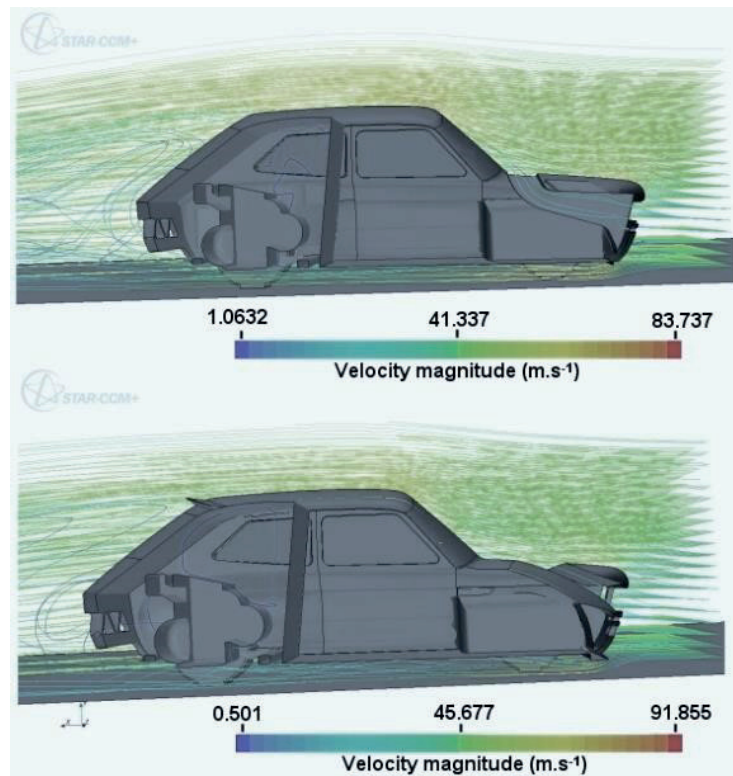
Buljac *et al.*, 2016 and Damjanovic *et al.*, 2011 in their CFD aerodynamics analysis of car came to the end, that during moving vehicle, the air slows down when it approaches the front of the car and results in that more air molecules are accumulated into a smaller space. Once the air stagnates in front of the car, it seeks a lower pressure area, such as the sides, top and the bottom of the car. As the air flows over the car hood, pressure is decreasing, but when reaches the front windshield, it increases briefly. When the higher-pressure air in front of the windshield travels over the windshield, it accelerates, causing the decrease of the pressure. This lower pressure literally produces a lift – force on the car roof as the air passes over it. Therefore there must be use some modification to improvement downforce of vehicle. The wing is a very efficient aerodynamic add-in, because it creates lots of downforce and thereby with small effect to increasing drag. Rear wing generally considered to improve the race car aerodynamics as a favourable downforce is obtained while the drag force is increased as well. The rear wing has big significance to the turbulences. It can be seen that in case of the redesigned car geometry there is less turbulences behind the car and the turbulent zone is cleaner.

## CONCLUSION

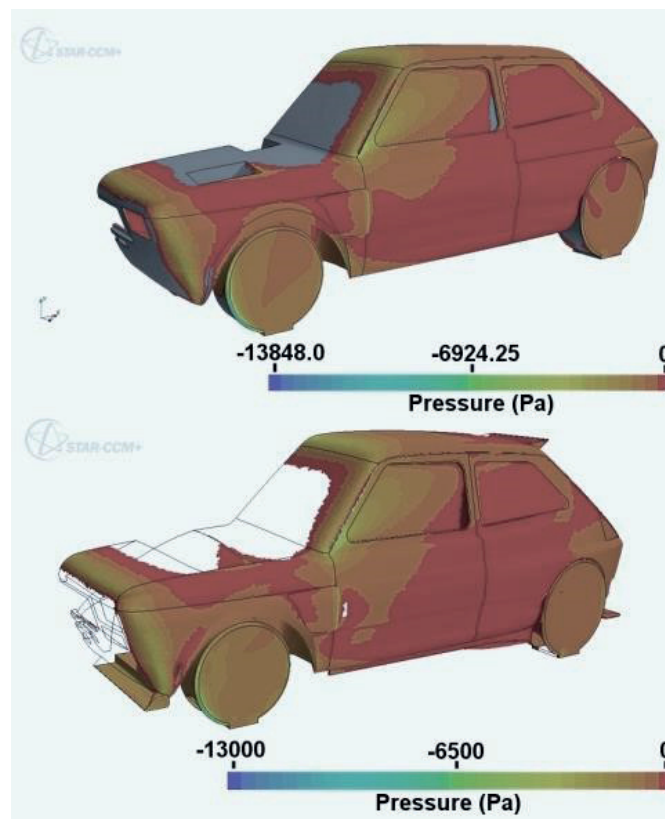
From the performed simulations, it is clear that the vehicle body was not designed by the manufacturer for using at high speeds. At a speed of  $190 \text{ km} \cdot \text{h}^{-1}$ , the resulting lift force ( $F_y$ ) exceeds almost 187 N. Aerodynamic simulation by a computer program STAR-CCM+ has helped to build up a picture of the aerodynamic properties of individual body parts of experimental vehicle and identify those parts of bodywork, which modification could lead to an improvement of these properties. After analyzing the test results of the original bodywork, modifications of car computer model were implemented, so that airflow was optimised around the problematic parts in order to increase overall downforce and transfer the point of action closer to the rear axle of the vehicle. At the same time, the goal was set to not increase too much aerodynamic drag of the vehicle, and also to not change the basic outer contour and shape of bodywork radically. Simulations performed on the modified model partially confirmed the correctness of our solutions. The resulting lift force of bodywork ( $F_y$ ) changed its size and direction. Instead of the original value of lifting force 187 N modified body generate downforce 503 N. On the modified body was also reduced torque in the axis of the front axle MZ1, while increased torque MZ2 in the axis of the rear axle (to  $-1251 \text{ Nm}$ ). This change caused a loading the rear axle, without adverse lifting effect on the front axle. The proposed modification was really implemented on race car FIAT 127 and verified in the race, noting increasing driving stability even at high speeds.

## Acknowledgements

Supported by the Grant Agency VEGA of the Ministry of Education of the Slovak Republic and Slovak Academy of Sciences, Grant VEGA No. 1/0337/15 „Research aimed at influence of agricultural, forest and transport machinery on environment and its elimination on the basis of ecological measures application”.



9: Graphical comparison of streamlines and velocity magnitude on stock (up) and modified (down) car  
Source: own



10: Graphical comparison of pressures on stock (up) and modified car (down)  
Source: own

## REFERENCES

- ANSYS. 2016. *Inc. ANSYS Fluent*. [Online]. Available at: <http://www.ansys.com/Products/Fluids/ANSYS-Fluent> [Accessed: 2016, May 24].
- AUTODESK. 2016. *Inc. Autodesk CFD Showcase Exporter 3.0*. [Online]. Available at: [https://apps.autodesk.com/productline/cs/Detail/HelpDoc?appId=7295736932499277815&appLang=en&os=Win32\\_64&mode=live#generalinf](https://apps.autodesk.com/productline/cs/Detail/HelpDoc?appId=7295736932499277815&appLang=en&os=Win32_64&mode=live#generalinf) [Accessed: 2016, May 12].
- BULJAC, A., et al. 2016. Aerodynamic Performance of the Underbody and Wings of an Open-Wheel Race Car. *Transactions of FAMENA*, 40(2): 19–34.
- DAMJANOVIĆ, D., et al. 2011. CFD analysis of concept car in order to improve aerodynamics. *A Jovo Jarmuve, A Magyar Jármuipar Tudományos Lapja*, 1(2): 63–70.
- CD-ADAPCO. 2009. *User Guide Star-CCM+ Version 4.04.011*. Melville, USA: CD-Adapco.
- EXA CORPORATION. 2016. *PowerFlow*. [Online]. Available at: <http://exa.com/product/simulation-tools/powerflow-cfd-simulation> [Accessed: 2016, May 26].
- KATZ, J. 2006. *Race car aerodynamics: Designing for speed*. Cambridge, USA: Bentley publishers.
- SLOBODA, A., FERENCÉY, V., HLAVŇA, V., TKÁČ, Z., et al. 2008. *Konštrukcia kolesových a pásových vozidiel: teória, konštrukcia, riziká*. Košice: Viena.
- VANČURA, J., NOVOTNÝ, P., PORTEŠ, P. and BLAŤÁK, O. 2010. Aerodynamic characteristics computation of open wheel racing car. *MECCA - Journal of Middle European Construction and Design of Cars*, 13(2): 22–27.
- VLK, F. 2001. *Dynamika motorových vozidel*. Brno: Nakladatelství a zaslátelství VLK.

## Contact information

Ivan Janoško: [ivan.janosko@uniag.sk](mailto:ivan.janosko@uniag.sk)  
Tomáš Polonec: [xkuchar@is.uniag.sk](mailto:xkuchar@is.uniag.sk)  
Peter Kuchar: [30045@is.uniag.sk](mailto:30045@is.uniag.sk)  
Pavel Máchal: [pavel.machal@mendelu.cz](mailto:pavel.machal@mendelu.cz)  
Martin Zach: [martin.zach@mendelu.cz](mailto:martin.zach@mendelu.cz)