Drag Analysis of Autonomous Vehicles in Different Arrays using CFD simulations

Filipe Fabian Buscariolo[1,3,5],Felipe Magazoni[2], Flavio Maruyama, Julio Cesar Lelis Alves and Leonardo D. Volpe[4],

[1]Imperial College London
[2]NETeF-USP
[3]NDF-USP
[4]GM North America
[5]McLaren Racing

E-mails: f.fabian-buscariolo16@ic.ac.uk, felipe.magazoni@gm.com, flavio.maruyama@gm.com, julio.alves@gm.com, leonardo.volpe@gm.com

ABSTRACT

Autonomous vehicles, which are defined as capable of sensing environment and navigating without any human input, are the top trend of the automotive industry. The computers responsible for the control are able to set the vehicle to optimum operation point. With the advent of Computational Fluid Dynamics -CFD software, it is possible to study drag reduction proposals when the vehicles drive at the velocity, which contributes to increase fuel economy.

In this context, based on a sedan vehicle virtual drag model, several simulations cases were developed considering different vehicle arrays and changing the distance between each one. The study aims to demonstrate, using virtual simulations, the potential drag coefficient reduction when vehicles are moving in a constant speed and which configuration leads to better performance increment. Taking the isolated vehicle, as the baseline value, all the vehicles in the different arrays were analysed.

Results show that the vehicles staying behind the first vehicle in the arrays have better drag coefficient performance. Considering the presented results, it is possible to apply this methodology to others types of vehicles and optimize the driving of autonomous vehicles.

INTRODUCTION

Autonomous cars can be defined as a vehicle that is capable of sensing the environment by using multiple types of electronic devices, such as radars, GPS, computer vision, and navigating through the streets and roads without any human input.

Experiments regarding autonomous cars have been conducted since 1920s and according to FORREST and KONCA (2007) [5], the first known worthy attempt to build an autonomous vehicle was in 1977. The research project was carried out by Tsukuba Mechanical Engineering Laboratory in Japan. The car functioned by following white street markers and was able to reach speeds of up to 20 mph on a dedicated test course.

The breakthrough in the development autonomous vehicles came in the 1980's with Carnegie Mellon University's Navlab and ALV projects in 1984 and the work of Mercedes-Benz, Ernst Dickmanns and his team at Bundeswehr Universität München in 1987. The project called Eureka Prometheus was able to achieve 96kph on the roads without traffic. Another important achieved was reached in 1995, with a car named CMU's NavLab 5, which has completed the first autonomous coast-to-coast drive of the United States. In July 2013, Vislab demonstrated BRAiVE, a vehicle that moved autonomously on a mixed traffic route open to public traffic.

Within the rising of the importance of autonomous vehicles in the mobility industry, SAE International published in 2014 an automotive standardization body named as J3016: Taxonomy and Definitions for Terms Related to On-Road Motor Vehicle Automated Driving Systems. This standard introduces a classification system based on six different levels, ranging from none to fully automated systems. In 2015, the USA states of Nevada, Florida, California, Virginia, and Michigan with Washington, D.C. allowed the testing of fully autonomous cars on public roads. In February 2017, automated cars, which are not fully autonomous yet, were permitted on public roads.

Considering the fact that autonomous vehicles are being permitted on public roads, according to LE VINE, ZOLFAGHARI and POLAK (2015) [7], there is a growing interest in the impacts on road network operations among transportation planners and road network managers.
In the future, considering all cars autonomous in a road, it will be possible to have them moving in constant velocity, which can be the top speed allowed of the road, in order to reduce the driving time. Another benefit is the capacity of maintain a fixed distance between the vehicles, which makes possible to increase the number of vehicles in a road, without compromising the safety of the drivers.

The movement of vehicles in both constant speed and distance between each other also implies in different aerodynamic behaviour from a single car. A possible approach to this behaviour is similar to an aircraft squad, bird's group or a cycling peloton.

According to TRENCHARD (2013) [12], a peloton may be defined as two or more cyclists riding in sufficiently close proximity to be located either in one of two basic positions: (1) behind cyclists in zones of reduced air pressure, referred to as 'drafting', or (2) in non-drafting positions where air pressure is highest. Cyclists in drafting zones expend less energy than in front positions. Similar to the cyclists, cars spend less energy while in the drafting zones and also increase their autonomy and reduce the emission levels.

Considering the autonomous vehicles as a peloton, this work focus on the effects of the drag coefficient values of the vehicles in different arrays and changing the gap between the vehicles, employing CFD as the main tool to quantify the drag coefficient for each vehicle.

Aerodynamics simulations are a reality in almost all automotive companies and it is used as one of the main tools to develop new high efficient vehicles and improve the aerodynamic behavior, combined with wind tunnel tests in order to reduce cost and development time.

According to MARUYAMA *et al.* (2015) [8], current car projects seek better performance regarding fuel consumption and aerodynamics plays an important role to achieve this goal. Every component of the car is studied in order to meet the required performance. Also, by improving the aerodynamic behavior of the vehicle, it is also possible to have higher top speeds, a desirable point for sports and performance cars.

TORRE and ÍÑIGUEZ (2009) [11], showed in their work that the power the cyclists need to travel requires overcoming aerodynamic or hydrodynamic drag forces, which can be substantially reduced when the group travels in an optimal arrangement. Cycle races, in which speeds of up to 54kph are frequent, offer great opportunities to appreciate the advantage of travelling in a group. It is presented a brief analysis of the aerodynamics of a cycling team in a time-trial challenge, showing how each rider is favoured according to his position in the group. The conclusion conclude that the artificial tail wind created by the team also benefits the cyclist at the front by about 5%.

PAHLE, et al., (2012) [9], pretended a work in which they experimentally evaluated several flight formations for two C-17 cargo airplanes in order to evaluate the effects of drag and fuel consumption reduction. Flight data recorded at test points within the vortex from the lead aircraft are compared to data recorded at tare flight-test points outside of the influence of the vortex. Since drag was not measured directly, reductions in fuel flow and thrust for level flight are used as a proxy for drag reduction. Estimated thrust and measured fuel flow reductions were documented at several trail test point locations within the area of influence of the lead's vortex. The maximum average fuel flow reduction was approximately 7-8%, compared to the tare points flown before and after the test points.

As the autonomous vehicles are becoming a trend, they can also be optimized in order to reduce the aerodynamic forces acting on the vehicles by moving as a cyclists' peloton.

1. OBJECTIVES

The main objective of this work is to evaluate the aerodynamic behavior in terms of the drag coefficient, considering an open geometry sedan vehicle available on GrabCad as the baseline main vehicle and two different arrays of 5 vehicles: in-line and V-shaped, similar to a cyclists' peloton, considering four different distances between each vehicle: half vehicle, one vehicle and two vehicles.

In the future, autonomous vehicles, combined with intelligent roads, will be able to set a fixed velocity and distance between the vehicles in order to optimize both the traffic and travel time. This paper aims to anticipate this by reproducing the same travel condition with current vehicles and evaluate which one is the most effective when related to drag performance.

The velocity in which the vehicles are simulated is 110kph, which is an average highway speed around most countries and is applied to both in-line and V-shaped arrays. Three fixed distances between the vehicles are also evaluated: 2.3m, which is inside the vehicle's rear wake region and represents half length of the vehicle; 4.6m, which represents one vehicle length; 6.9m, representing 1.5 times the length of the baseline vehicle; and 9.2m, which represents a length where the vehicle's wake has been dissipated and represents two vehicles lengths.

The drag coefficient for each vehicle, in each distance case, is compared to the baseline single vehicle value, in order to evaluate the effect of the cars moving together, evaluate the optimal distance between the cases studied and could be used to estimate the energy economy for the peloton moving.

2. TECHNICAL FORMULATION

In order to understand the flow behavior, it is necessary to calculate the variables associated to the fluid based on the conservation law, which is expressed by the expression below [2] [10],

$$\frac{\partial \rho\phi}{\partial t} + div(\rho\phi u_r) = div(\Gamma\, grad\, \phi) + S_\phi \quad (1)$$

where ρ is the fluid density, t is the time, u_r is the relative speed between the fluid and coordinate system, ϕ is the scalar variables (u, v, w, for 3 directions velocity; k,for the turbulent kinetic energy; ε, for the turbulence dissipation; and E for the Energy), Γ is the diffusion coefficient and S_ϕ is the source term.

By replacing the scalar variables in Equation 1 with the appropriate adaptations, arise the Navier-Stokes equations, which describe the motion of a fluid and its properties, and considering incompressible, isothermal Newtonian flow (density and viscosity are constants),

$$\nabla \cdot V = 0 \quad (2)$$

$$\rho \frac{DV}{Dt} = -\nabla P + \rho g + \mu \nabla^2 V \quad (3)$$

where P is pressure, μ is the viscosity, V is the velocity field and g is the acceleration vector.

As the averaged Navier-Stokes Equations, basis of most of the computational CFD codes, compose an open equation system due to the additional Reynold's stress terms. Hence, a turbulence model needs to be incorporated in order to close the equation system. Turbulence models considered in this study are the k-ε and k-ω SST models.

The k-ε turbulence model is composed by two equations that come directly from the differential transport equations, in which k represents the turbulent kinetic energy and ε represents the turbulence dissipation rate. The two equations are written below [4] [9],

$$\frac{\partial \rho k}{\partial t} + div(\rho k u_r) = = div\left(\frac{\mu_t}{\sigma_k}\, grad\, k\right) + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \quad (4)$$

$$\frac{\partial \rho\varepsilon}{\partial t} + div(\rho\varepsilon u_r) = div\left(\frac{\mu_t}{\sigma_\varepsilon}\, grad\, \varepsilon\right) + + C_{1z}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2z}\rho\frac{\varepsilon^2}{k} \quad (5)$$

where μ_t is the turbulent viscosity, C_{1z}, C_{2z}, σ_ε and σ_k are turbulence model constants.

Once the fluid behavior and properties are known, the next step is to calculate drag coefficient, which comes from the pressure acting over the vehicle due to the contact with air. Once the drag force (D) is an output from the Navier-Stokes solution, the drag coefficient of a vehicle, which is characterized by its frontal area A and vehicle speed V is expressed by the following equation below,

$$c_D = \frac{2D}{\rho V^2 A} \quad (6)$$

3. METHODOLOGY

In order to perform this study on CFD, a virtual wind tunnel will be considered, with test section similar dimensions to one of the largest wind tunnels, the General Motors (GM), with increased section length in order to accommodate the 5 vehicles array.

According to BUSCARIOLO, et al. (2016) [2], it is a closed circuit with temperature control to perform aerodynamic studies such as directional stability and thermal comfort in passenger cars, medium size "Pick Ups", sports cars and "SUVs (Sport Utility Vehicle). Its test section has a static floor with boundary layer suction system on the floor.

As stated in KELLY et al. (1982) [6], the GM's tunnel presents a cross section at the final nozzle of 56.16 m^2. It is built of coated metal with a thick 600 mm concrete layer to prevent sound waves propagation. The air flow is provided by a powerful fan with 13 m diameter blades and an approximate power of 3000 kW, able to reach a top speed of 250kph for 15 minutes. GM's wind tunnel scheme is shown in Figure 1.

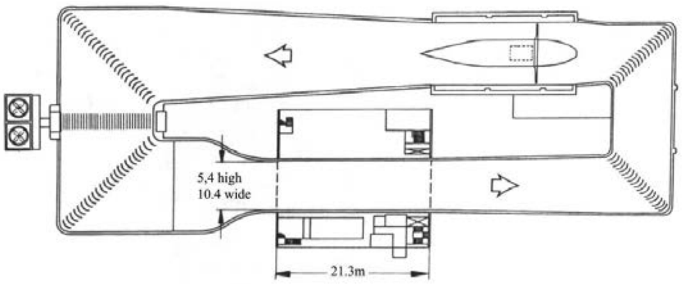

Figure 1. General Motors full-scale Wind Tunnel (Kelly, 1982) [6]

The GM's wind tunnel is equipped with a static floor combined with a boundary layer suction system to minimize its effect over the drag coefficient measured, according to KELLY et al. (1982) [6]. Many studies have been made to understand the behavior of the wind tunnel's floor boundary layer and how to avoid its interference in the aerodynamics measurements. Wheel rotation can currently not be simulated in the wind tunnel. This has a significant impact on the flow around the vehicle and was therefore also not considered in the CFD simulations.
For the CFD simulations the same parameters as shown in BUSCARIOLO and KARBON (2011) [1] are used, see the table below.

Table 1. Parameters for the simulations

Wind tunnel	General Motors, Warren-Michigan (similar parameters)
Air speed	110kph
Outlet pressure	Atmospheric pressure
Turbulence intensity	0.60 %
Boundary layer suction system	At the beginning of the test section
Test Section dimensions	(5.4 x 10.4 x 68.0) m (increased length to fit all vehicles)

Simulations are performed using the program Simerics®. and ANSA® are used to generate the surface mesh of vehicle and wind tunnel test section, including the volumetric mesh in all cases and the main solver for the flow equations;

Vehicle evaluated is an open source geometry from GrabCad (https://grabcad.com/library/tesla-model-3-1), with Af = 2.17 m^2 and which close corresponds to a hybrid vehicle with autonomous characteristics. It is represented by a virtual model, considering simplified underhood and closed fascia openings. The vehicle's model can be seen in Figure 2.

Figure 2. Representation of the car studied in this work.

For the load cases evaluated, Figures 3, 4, 5 and 6 show the in-line array for 5 vehicles with the distances between each other from half-vehicle length, one vehicle length, 1.5 vehicle length and 2 vehicles length.

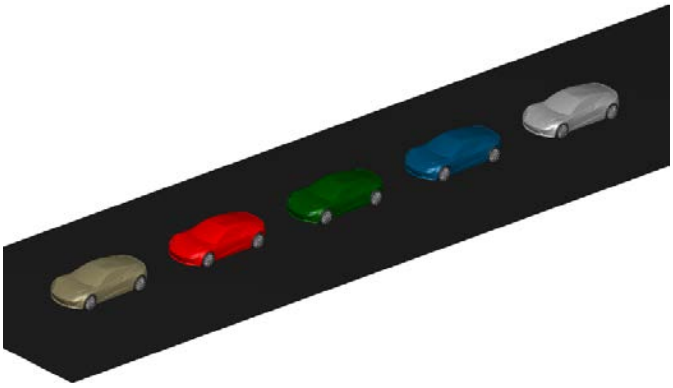

Figure 3. Representation of the in-line array with distance of half vehicle between each car.

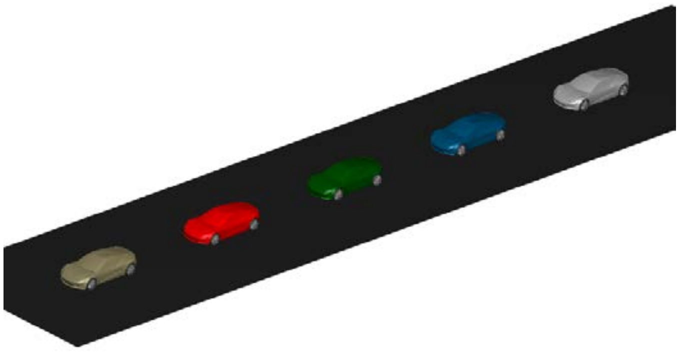

Figure 4. Representation of the in-line array with distance of one vehicle between each car.

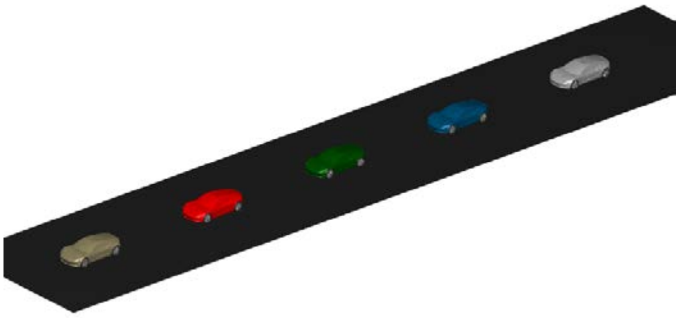

Figure 5. Representation of the in-line array with distance of one and a half vehicle between each car.

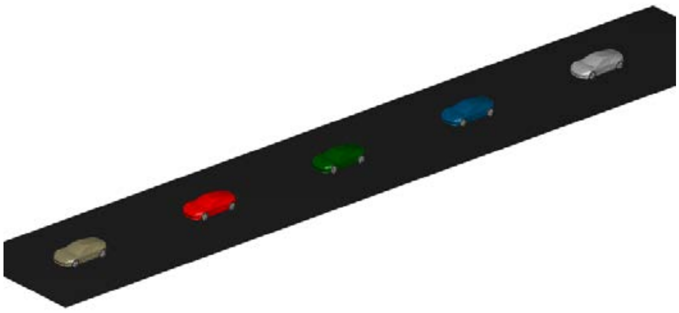

Figure 6. Representation of the in-line array with distance of two vehicles between each car.

Similar to previous images, Figures 7, 8, 9 and 10 show the V-Shaped array for 5 vehicles with the distances between each other from half-vehicle length, one vehicle length, 1.5 vehicle length and 2 vehicles length.

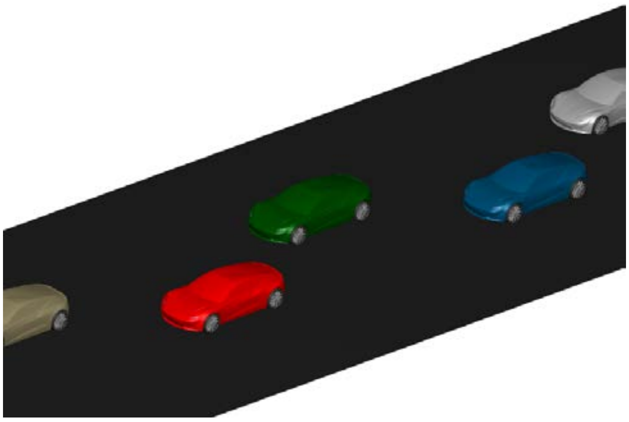

Figure 7. Representation of the V-Shaped array with distance of half vehicle between each car.

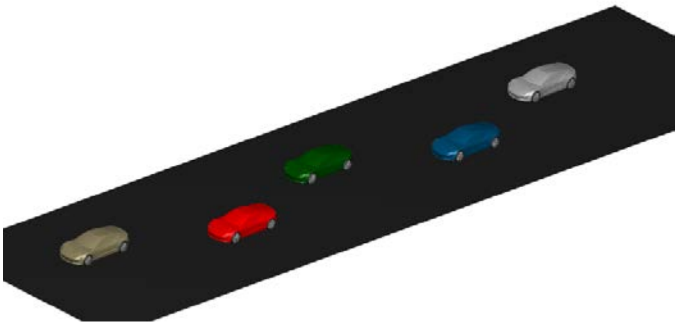

Figure 8. Representation of the V-Shaped array with distance of one vehicle between each car.

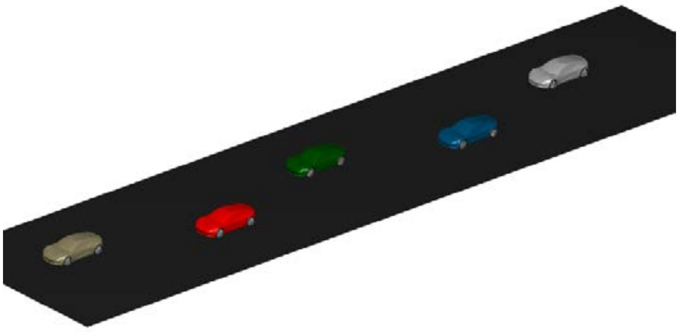

Figure 9. Representation of the V-Shaped array with distance of one and a half vehicle between each car.

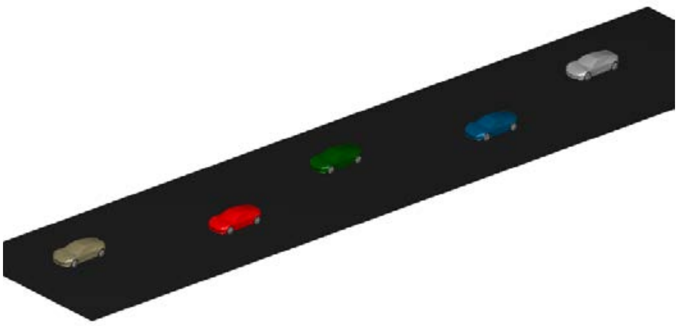

Figure 10. Representation of the V-Shaped array with distance of two vehicles between each car.

4. RESULTS

4.1. Drag Coefficient results

Considering the 4 load cases evaluated for the 2 cars arrays, in-line and V-Shaped, the drag coefficient for each vehicle was obtained and it is shown in table 2. For comparison reasons, the % of Cd increment is reported, based on the Cd obtained from the most isolated car position, which is car 5, in the V-Shape array, considering distance of two vehicle lengths, which closest correspond to a wind tunnel test of an isolated vehicle.

Table 2. Summary of Drag Coefficient increments for all load cases evaluated

Array	Vehicle Position	Delta Cd% - Distance Between Cars			
		0.5	1	1.5	2
In-line	Car 1	-23%	-9%	-6%	-1%
	Car 2	-45%	-34%	-22%	-22%
	Car 3	-41%	-23%	-10%	-7%
	Car 4	-36%	-16%	-9%	-3%
	Car 5	-24%	-7%	-9%	-9%
V-Shaped	Car 1	-11%	-5%	-4%	Baseline
	Car 2	-16%	-14%	-5%	-3%
	Car 3	-24%	-15%	-15%	-9%
	Car 4	-30%	-25%	-12%	-5%
	Car 5	-21%	-13%	-14%	-7%

Considering the results in Table 2, it’s possible to conclude that for the in-line cases, travelling the shortest distance, the half car length case, gives better enhancement on drag performance for all vehicles, with a maximum reduction of 45% for car 2, compared to baseline vehicle.

Considering the V-Shaped array, similar trend was observed and the best enhancement case matches with in-line array, being the half car length distance the best for drag reduction. The best reduction was observed for car number 4, which is in the secondary line, rather than behind a vehicle.

From mathematical results, it’s possible to conclude that there is an advantage in terms of drag performance once the vehicle is in the wake region of another vehicle. It was expected to have better performance for the vehicles, which are not explicit exposed to free stream, such as car 2, 3, 4 and 5 for in-line array and cars 3, 4 and 5 for V-Shaped array, once the air pressure is not directly acting over the vehicle.

The second point is the influence of the rear wake generated, which can both help the front and rear vehicles in terms of performance. For in-line arrays, as close the vehicles get, better performance is observed for all cases. This can be explained by the fact of the low-pressure zones are reduced and it might behave as a continuous flow, similar to a train. The low-pressure of the rear wake also reduce the pressure on the front end on the following vehicle, enhancing its performance in terms of drag and contributing to lower emissions and higher efficiency. For the V-Shaped array, best drag performance is on car 4 up to one vehicle length, due to both influences of the car in the front and on the upper line. For 1.5 vehicle length up to 2, car 3 heads the best performance.

Considering the in-line array cases, Figures 11, 12, 13 and 14 shows a middle-plane cut showing the wake profile contour colored by velocity for each case.

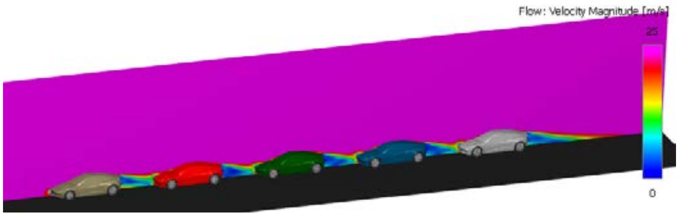

Figure 11. Velocity middle plane contour of the in-line array with distance of half vehicle between each car.

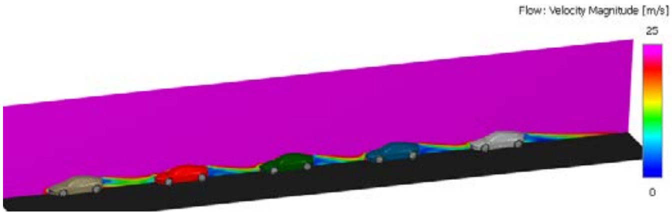

Figure 12. Velocity middle plane contour of the in-line array with distance of one vehicle between each car.

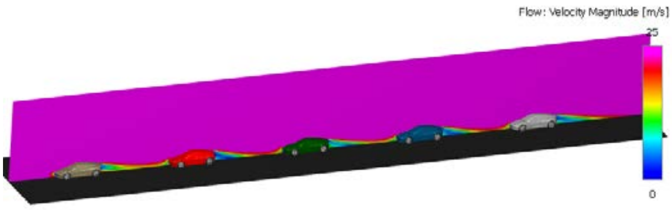

Figure 13. Velocity middle plane contour of the in-line array with distance of one and a half vehicle between each car.

Figure 14. Velocity middle plane contour of the in-line array with distance of two vehicles between each car.

In terms of performance for all vehicles, the best case to be considered for future applications would be the shortest possible distance between the vehicles and in-line array for cost saving, emissions reduction and efficiency in terms of having more vehicles moving. Once this test was based on autonomous car which can be fully controlled, it's and reliable option.

CONCLUSION

Most automakers are focusing their research on autonomous vehicles which are capable of sensing the environment by using multiple types of electronic devices, such as radars, GPS, computer vision, and navigating through the streets and roads without any human input.

They started to be studied in the early 1920's and since then, the automation tendency is rising. With new types of sensors and computers, it's also becoming feasible to have a real autonomous vehicle that can take people around the places.

For a fully integrated and intelligent operational system, the streets and roads should be also re-designed with new features and also to control and organize the way autonomous vehicles should move.

The future highways could be compared as a cyclists' peloton, in which cars will be grouped in lines, moving with fixed speed and distance between each other.

Focusing also on eco-friendly projects, in order to meet the emissions policies and also to increase the technical development, aerodynamics is one of the key contributors that make the emission targets achievable.

By moving as a peloton, the vehicles will be exposed to different levels of aerodynamic forces and behaviors, depending mainly on the distance between each one, compared to one single vehicle.

The main objective of this work is to evaluate the aerodynamic behavior in terms of the drag coefficient, considering an open geometry sedan vehicle available on GrabCad as the baseline main vehicle and two different arrays of 5 vehicles: in-line and V-shaped, similar to a cyclists' peloton, considering four different distances between each vehicle: half vehicle, one vehicle and two vehicles.

Results shows that there is a potential reduction for the drag coefficient as the vehicles get close to each other and stay in-line behavior, being the load case studies, in-line array considering distance of half vehicle between each other the best performance among the eight cases evaluated.

As the autonomous vehicles can be precisely controlled, it's possible to aim short distances between the cars without compromising safety, as the whole array can be set to stop at the same time, coordinated by a signal.

Further work might consider evaluate the same array with different body style vehicles in order to check if the same behavior is found and also reduce the distance between the vehicles.

REFERENCES

1. Buscariolo, F.F.; Karbon, K.J., “Comparative CFD Analysis Between Rotating and Static Cases of Different Wheels Opening Designs over a Performance Sedan”, SAE Paper, Nº 2011-36-0271, Society of Automotive Engineers, 2011.

2. Buscariolo, F.F.; Magazoni, F., Wolf, M., Maruyama F., Alves, J. C. L., Volpe, L. D., “Analysis of Turbulence Models Applied to CFD Drag Simulations of a Small Hatchback Vehicle”, SAE Paper, Nº 2016-36-0201, Society of Automotive Engineers, 2016.

3. Ferziger, P.; Peric, M., “Computational Methods for Fluid Dynamics”, 2nd Edition, Springer-Verland, Germany 1999.

4. Forrest, A., Konca, M., “Autonomous Cars and Soceity”, Worcester Polytechnic Institute, USA, 2007.

5. Kelly, K. B.; Provencher, L. G.; Schenkel, F. K., The General Motors Engineering Staff Aerodynamics Laboratory – A full Scale Automotive Wind Tunnel, SAE Paper, Nº 820371, Society of Automotive Engineers, 1982.

6. Le Vine, S., Zolfaghari, A., Polak, J., “Autonomous cars: The tension between occupant experience and intersection capacity”, Transportation Research Part C 52 (2015) 1–14, Elsevier, 2015

7. Maruyama, F., Alves, J. C. L., Volpe, L. D., Magazoni, F., Buscariolo, F. F., “Wheel Design Sensitive Analysis on Drag of Small Sedan Vehicle”, SAE Paper, Nº 2015-36-0168, Society of Automotive Engineers, 2015.

8. Pahle, J., Berger, D.,Venti, M., Duggan, C., Faber, J., Cardinal, K., “An Initial Flight Investigation of Formation Flight for Drag Reduction on the C-17 Aircraft”, AIAA Paper Nº AIAA 2012-4802, AIAA Atmospheric Flight Mechanics Conference, Minnesota, USA, 2012.

9. Tennekes, H.; Lumley, J. L.,”A First Course in Turbulence”, MIT Press, Cambridge, MA, 1972.

10. Torre, I., Íñiguez, J.,” Aerodynamics of a cycling team in a time trial: does the cyclist at the front benefit?”, EUROPEAN JOURNAL OF PHYSICS vol. 30, (1365-1369), IOP Publishing, 2009

11. Trenchard, H., “Peloton phase oscillations”, Chaos, Solitons & Fractals vol. 56, (194-201), Elsevier, 2013.

12.

NOMENCLATURE

$\boldsymbol{A}$	=	Area, m^2
$\boldsymbol{C_{1z}}$	=	Turbulence model constant
$\boldsymbol{C_{2z}}$	=	Turbulence model constant
$\boldsymbol{C_{des}}$	=	Calibration DES constant
$\boldsymbol{c_D}$	=	Drag coefficient
$\boldsymbol{D_\omega}$	=	Cross-diffusion term
$\boldsymbol{E_{ij}}$	=	Turbulence model constant
$\boldsymbol{D}$	=	Aerodynamic force, N
$\boldsymbol{G_k}$	=	Production of turbulence kinetic energy
$\boldsymbol{G_\omega}$	=	Generation of ω
$\boldsymbol{S_k}$	=	User-defined source terms
$\boldsymbol{S_\omega}$	=	User-defined source terms
$\boldsymbol{Y_k}$	=	Dissipation of k
$\boldsymbol{Y_\omega}$	=	Dissipation of ω
$\boldsymbol{g}$	=	Acceleration gravity, m/s^2
$\boldsymbol{k}$	=	Turbulent kinetic energy, m^2/s^2
$\boldsymbol{P}$	=	Pressure, Pa
$\boldsymbol{S_\phi}$	=	Source term, kg/m^2-s^2
$\boldsymbol{t}$	=	Time, s
$\boldsymbol{U}$	=	Flow variables
$\boldsymbol{u_r}$	=	Relative velocity between the fluid and coordinate system, m/s
$\boldsymbol{V}$	=	Velocity, m/s
$\boldsymbol{\varepsilon}$	=	Turbulence dissipation rate, m^2/s^3
$\boldsymbol{\beta}^*$	=	k-ω SST model constant
$\boldsymbol{\Gamma}$	=	Diffusion coefficient, kg/m-s
$\boldsymbol{\mu}$	=	Dynamic viscosity, kg/m-s
$\boldsymbol{\mu_t}$	=	Turbulent viscosity, kg/m-s
$\boldsymbol{\phi}$	=	General scalar variable
$\boldsymbol{\rho}$	=	Density, kg/m^3
$\boldsymbol{\sigma_\varepsilon}$	=	Turbulence model constant
$\boldsymbol{\sigma_k}$	=	Turbulence model constant
$\boldsymbol{\Delta_{max}}$	=	Grid spacing