UNIVERSITY OF PITESTI
FACULTY OF MECHANICS AND TECHNOLOGY
SCIENTIFIC BULLETIN
AUTOMOTIVE series, year XVI, no.20 (B)

NUMERICAL SIMULATION OF FLOWS AROUND TWO DIFFERENT SHAPED CARS USING CFD[1]

Ion TABACU [1], Cătălin NEACŞU [2], Victor IORGA[1], Mariana IVANESCU[1]

[1] University of Piteşti, [2] SC Automobile Dacia SA
E-mail: ion.tabacu@upit.ro, catalin.neacsu@renault.com, vicpit23@yahoo.com, mariana.ivanescu@upit.ro

Abstract:

In the automotive industry is in continuous change. One aspect that is very important is the aerodynamic development of new vehicles, and this can be done experimentally or by numerical simulation.

For a moving car, the drag refers to forces that oppose the relative motion of the car through air and is dependent of the vehicle velocity.

Often, in car aerodynamics is used the drag coefficient, c_d. This represents a dimensionless quantity and is used to quantify the drag of an object in a fluid environment, in our case, air. This coefficient is used in the drag equation and it is always associated with a particular area. Between the drag coefficient and the resistance encountered by the car is a direct proportionality.

In our paper we will evaluate, using the computational fluid dynamics (CFD) software FLUENT, the flow around two different shaped cars. The geometries will be simplified shapes of a sedan and a wagon with the same length, and height and width, the only difference being found on the rear part of the car.

Keywords: numerical simulation; aerodynamic; drag coefficient; CFD; Fluent

INTRODUCTION

In today automotive industry rapid changes in the customers demands represent one of the reasons to reduce the time and cost needed for the development of a vehicle. An important tool that helps the car makers is the numerical simulation.

Drag is proportional to the drag coefficient, frontal area and the square of vehicle speed. You can see a car traveling at 120km/h has to fight with 4 times the drag of a car traveling at 60 km/h. You can also see the influence of drag to top speed. If we need to raise the top speed of Ferrari Testarossa from 288km/h mph to 320 km/h like Lamborghini Diablo, without altering its shape, we need to raise its power from 390 hp to 535 hp. The same effect will have the reduction of its c_d from 0.36 to 0.29 can do the same thing [6]. This thing can be done using the CFD analysis.

The numerical simulations are very beneficial to the designers because they offer visual aids and data of the interactions that are occurring as well as flow trends that were not thought about before.

In our paper, we will evaluate the drag coefficient for two shaped cars, a sedan and a wagon. We will also visualize the velocities of air around those two cars, and the pressure on the surface of the cars.

AIR RESISTENCE THEORY

[1] This article was produced under the project "Supporting young Ph.D students with frequency by providing doctoral fellowships", co-financed from the EUROPEAN SOCIAL FUND through the Sectoral Operational Program Development of Human Resources

The main aspects that are studied by automotive aerodynamic are the following:

- Air drag and ways to minimize it;
- Effects of interaction with the air on the stability vehicles and method of improving aerodynamic stability;
- Effects of interaction with air on adhesion between the wheel with road and methods of enhancing it;
- Air movement inside the vehicle and proper choice of different suction and discharge holes for air ventilation to cool the body and various parts.

Current flow of air near the vehicle body is modeled by the relationship between pressure and velocity expressed by Bernoulli's equation:

$$P_{static}+P_{dynamic}=P_{total} \tag{1}$$

or:

$$p_s + \frac{1}{2}\rho v^2 = p_t \tag{2}$$

where:

ρ	=	air density
v	=	air velocity (relative to the vehicle)

As a result of contact between air and car body will appear three main flows: some part of air, flow will pass over the car body, another between the car body and road and the third flow will hit the car body

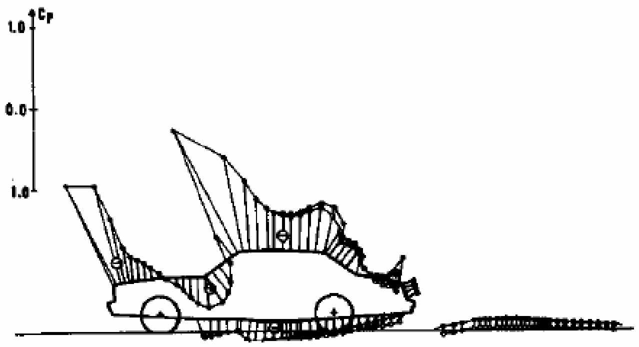

Figure.1. Pressure distribution along the longitudinal section [3]

The speed along the car body decrees at zero, that involving a variation of pressure. This phenomenon determines the force of drag cause by interaction with air. The force on a moving car due to air is:

$$R_a = \frac{1}{2}\rho \cdot c \cdot_x \cdot A \cdot v^2 \tag{3}$$

where:

ρ	=	air density(ρ=1.225 kg/m^3)
v	=	air velocity (relative to the vehicle)
c_x	=	drag coefficient for the reference area

We can calculate the basic reference car area with formula:

$$A = B * H \tag{4}$$

where:

B	-	width of the car
H	-	height of the car

MODEL CREATION

For the present paper, we will use two simplified car models with the same overall dimensions. The reference values for length, width and height and also the shape will be taken from blueprints chosen from the internet [5].
Starting from the blueprints, we will model the two models using Catia V5 software, keeping only the body of the car, without mirrors and detailing. After 3d modeling of the shape of the car, we will create the finite element model of that shape. In figures 2 and 3 are presented the wire-frames and meshes of the 3d models for both versions, and in table 1 can be seen the basic dimensions for the two models, and also the number of elements necessary to mesh the entire car model.

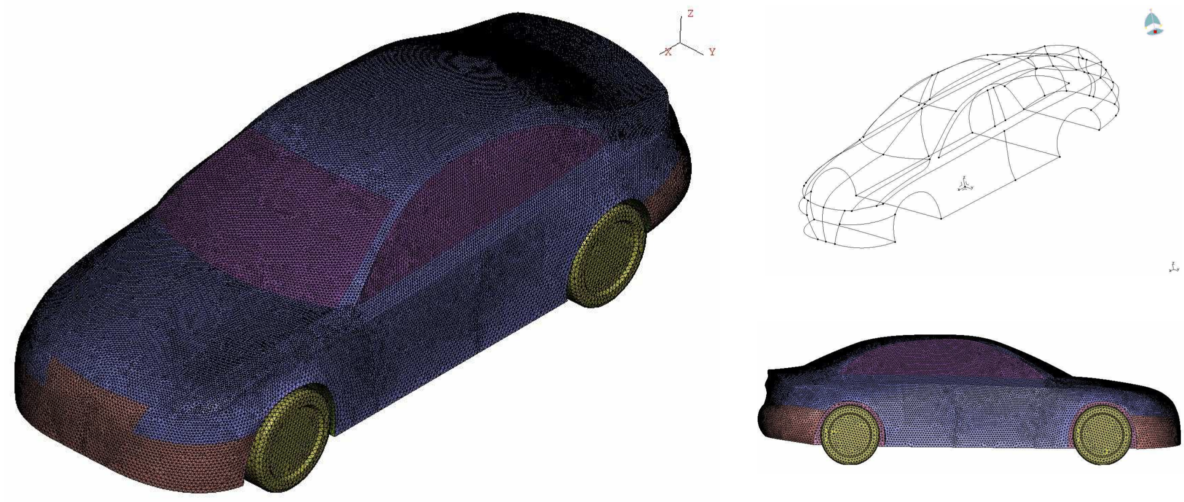

Figure 2. Wireframe and mesh for Sedan version

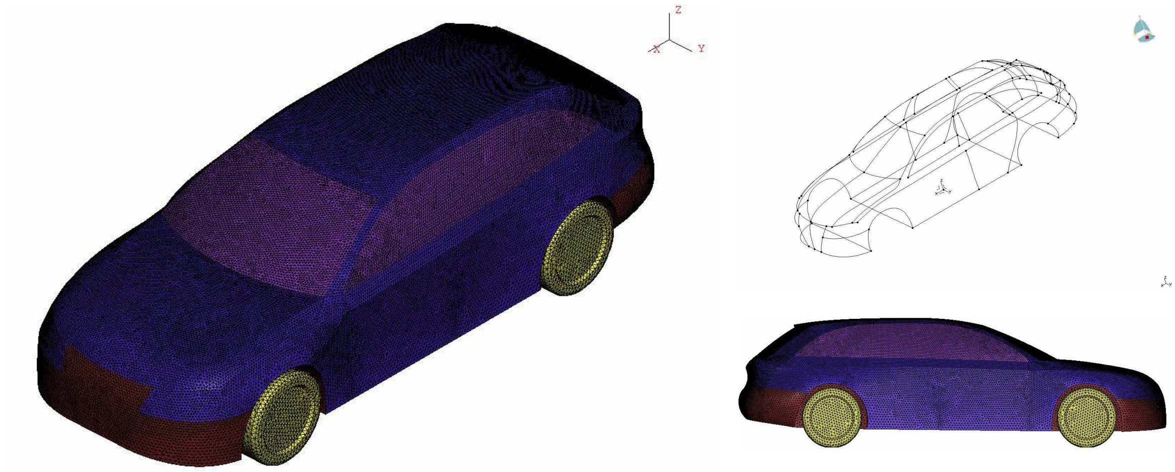

Figure 3. Wireframe and mesh for Wagon version

Version	***Sedan***	***Wagon(Avant)***
Length[m]	4,703	4,703
Width[m]	1,826	1,826
Height[m]	1,427	1,436
Number of elements	233670	232250

Table 1. Model characteristics

NUMERICAL SIMULATION

To analyze the aerodynamic features of the car are two possibilities: wind tunnel and computational fluid dynamics. Because it's proven efficiency and financial reasons, the later offers us a better solution. Other aspect that demonstrates the CFD advantages are the visualization and the accuracy provided.
The pre-processing was done using the Ansa software, as we mentioned earlier, and for the solving and post-processing we have used the Fluent software.

Theory

The governing equations for computational fluid dynamics are based on conservation of mass, momentum, and energy. *FLUENT* uses finite volume method (FVM) to solve the governing equations. The FVM involves discretization and integration of the governing equation over the control volume. The flow is said to be turbulent when all the transport quantities (mass, momentum and energy) exhibit periodic, irregular fluctuations in time and space. Such conditions enhance mixing of these transport variables. There is no single turbulent model that can resolve the physics at all flow conditions. *FLUENT* provides a wide variety of models to suit the demands of individual classes of problems. The choice of the turbulent model depends on the required level of accuracy, available computational resources, and the required turnaround time . For the problem analyzed in this paper, standard k - ε turbulent model is selected. The *k* - ε model is one of the most common turbulent models. It is a semi - empirical, two equation model, that means, it includes two extra transport equations to represent the turbulent properties of the flow. The first transported variable is turbulent kinetic energy *k*. The second transported variable is the turbulent dissipation ε. It is the variable that determines the scale of the turbulence, whereas the first variable *k* determines the energy in the turbulence.
The model transport equation for *k* is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart [4].
Transport equations for standard *k* - *ε* model

- for turbulent kinetic energy *k:*

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \qquad (5)$$

- for dissipation *ε*:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(C_k+C_{3\varepsilon}G_b)-\rho C_{2\varepsilon}\frac{\varepsilon^2}{k}+S_\varepsilon \qquad (6)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for *k* and *ε* , respectively. S_k and S_ε are user - defined source terms [4].

Virtual wind tunnel creation

After creating the basic shape of the car, we will create the virtual aerodynamic tunnel. It has a shape of a box, and is placed around the car. The entrance to the wind tunnel will be placed 10 meters in front of the car, and will be considered the place where air enters the domain. The exit of the wind tunnel will be placed 20.2 meters from the rear of the car, and will represents the place where the air exits from the domain. The other characteristics of the domain will be the width of the domain, 6 meters and the height of the domain, 5 meters. The characteristics are found in table 2, and in figure 4 we can see the representation of this, for the Sedan version. For the Wagon version, the domain will have the same characteristics.

	Length[m]
Length in front of the car	10.0
Length from the back of the car	20.2
Total length	35.0
Width	6.0
Height	5.0

Table 2. Virtual tunnel dimensions

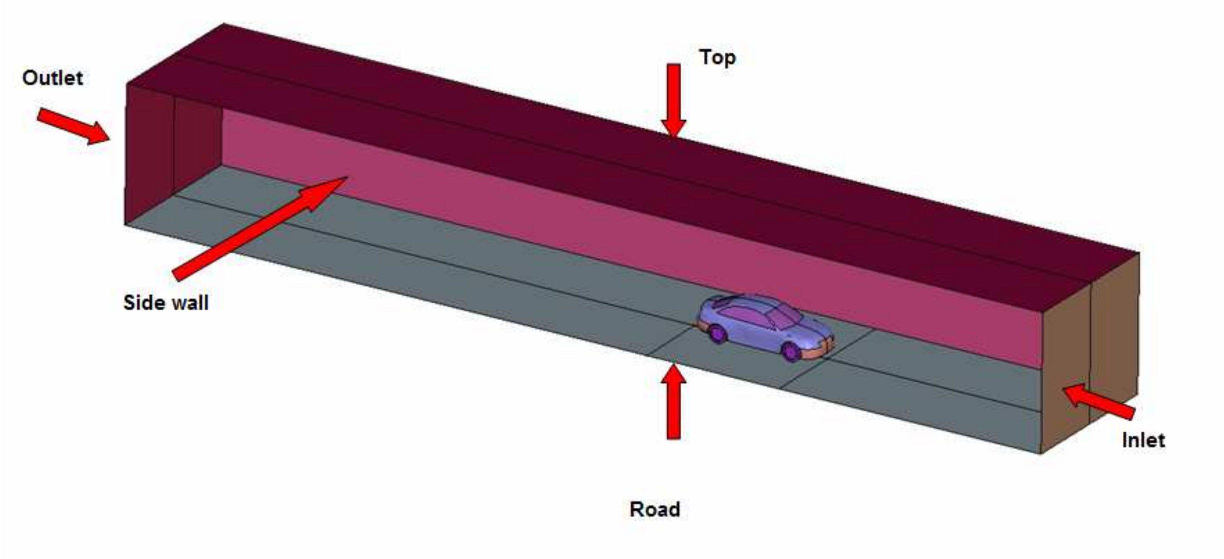

Figure 4. Computational domain

Boundary conditions

For our simulations will consider the air velocity at the inlet equal to 30m/s. The other boundary conditions are presented in table 3, and are taken accordingly to[2].

Boundary	***Conditions***
Inlet	*Velocity inlet*
Outlet	*Pressure outlet*
Road	*Wall - moving*
Top	*Symmetry*
Sides(left-right)	*Symmetry*

Table 3. Boundary conditions

To establish if the flow around the car is either turbulent or laminar, we must compute the Reynolds number for the given situation. Considering the air velocity equal to 30 m/s, the Reynolds number based on the length of the vehicle will be equal to:

$$\mathrm{Re} = \frac{\rho \cdot v \cdot L}{\mu} = \frac{1{,}225 \cdot 30 \cdot 4{,}8}{1{,}7894 \cdot 10^{-5}} = 45324372 \tag{7}$$

This value obtained for the Reynolds number corresponds to a turbulent airflow. Since the airflow around a car is turbulent, a model needs to be selected for the simulation of the turbulent flow. There appears to be four major turbulence models used in automotive industry: k-epsilon, k-omega, Lattice-

Boltzman and Large Eddy Simulation(LES). Of these models, k-epsilon and k-omega are most widely used with the k-epsilon said to be most stable.
The model that we will use for our simulation will be *Standard k-epsilon.*

RESULTS

In figure 5 and 6 we will present the velocity contours on both car geometries in the middle plane, in figure 7 and 8 the air path lines colored by the velocity magnitude, and in figure 9 and 10 the velocity distribution over the car body (post processing realized using CEI Ensight).

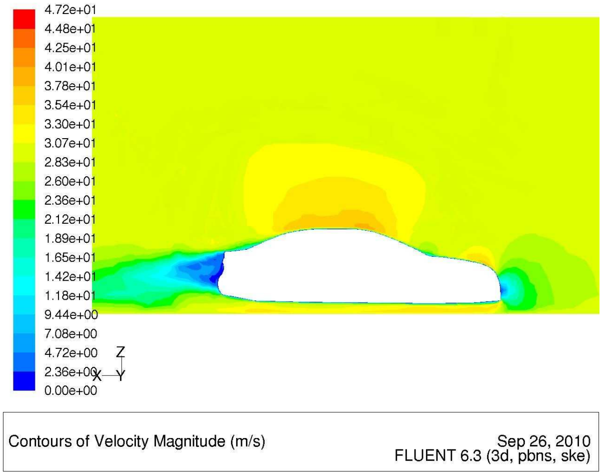

Figure 5. Velocity contours in the Sedan mid-plane

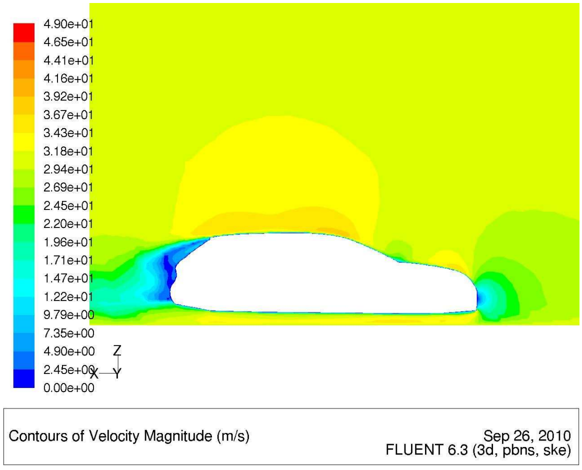

Figure 6. Velocity contours in the Wagon mid-plane

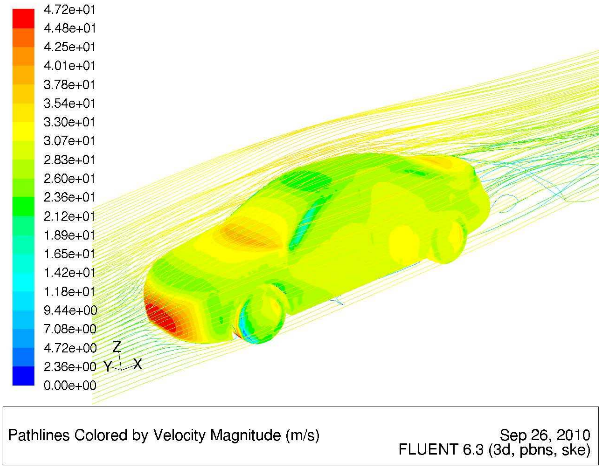

Figure 7. Velocity pathlines over the Sedan

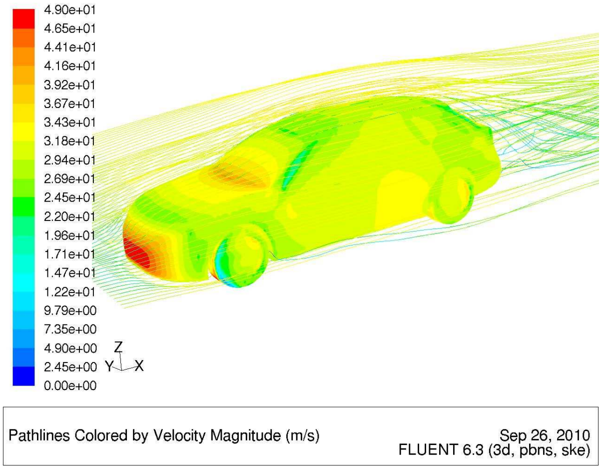

Figure 8. Velocity pathlines over the Wagon

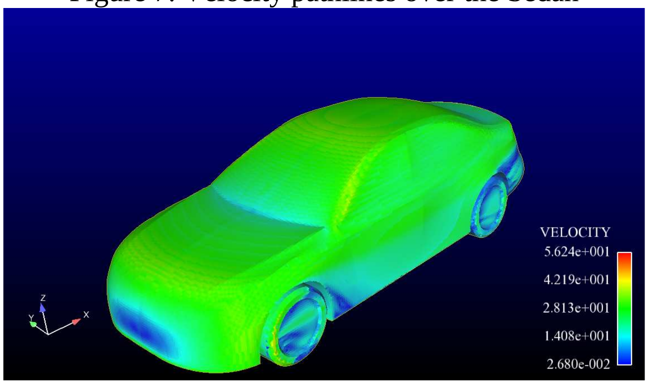

Figure 9. Velocity contours on the Sedan body - front view

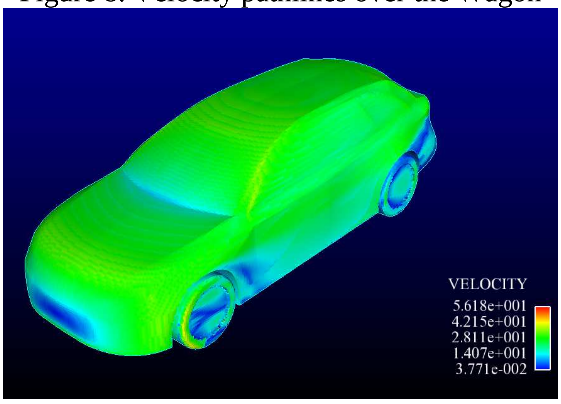

Figure 10. Velocity contours on the Wagon body - front view

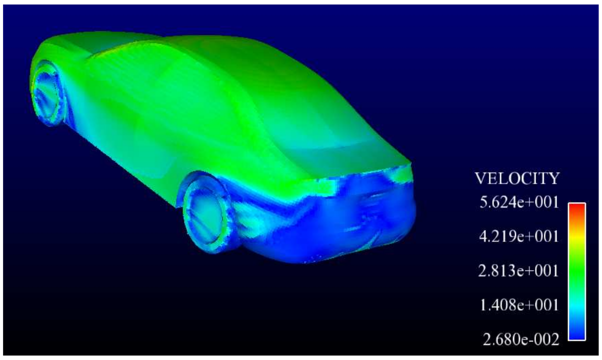

Figure 11. Velocity contours on the Sedan body - rear view

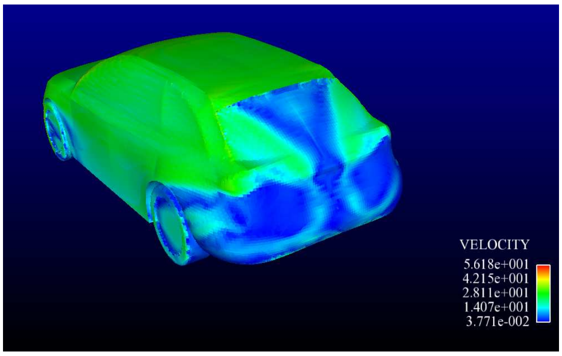

Figure 12. Velocity contours on the Wagon body - rear view

From figures 11 and 12 we can observe the zero velocity regions near the back of the cars. For the wagon version, the recirculation areas present on the rear windshield represents the zones that generate maximum of turbulence. Figures 13 and 14 show the static pressure contours for both bodies in the middle plane, while figures 15 and 16 shows the static pressure on the car body. Can be seen that air slows down and results 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 bottom of the car. As the air flows over the car hood, pressure is decreasing, but when it reaches the front windshield it briefly increasing. When the higher pressure air in front of the windshield travels over the windshield, it accelerates, causing the decreasing of pressure. This lowering in the pressure literally produces a lift force on the car roof as the air passes over it.

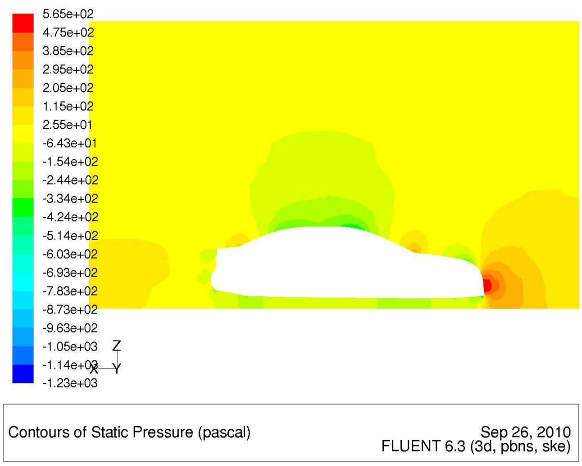

Figure 13. Static pressure contours in the Sedan mid-plane

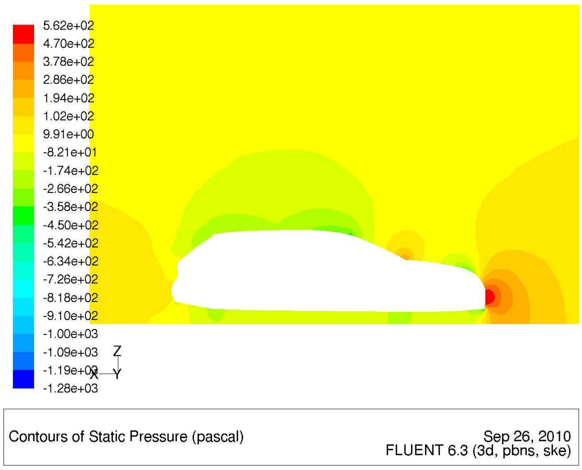

Figure 14. Static pressure contours in the Wagon mid-plane

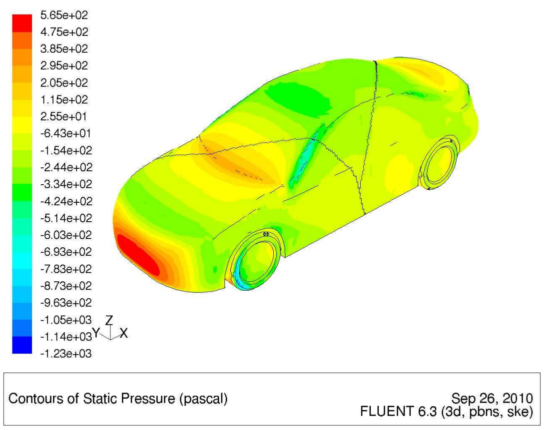

Figure 15. Contours of pressure on Sedan body

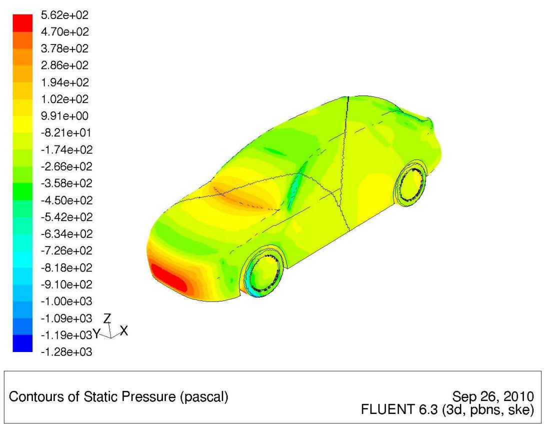

Figure 16. Contours of pressure on Wagon body

From figures 17 and 18 we can see that the flow around the sedan is less turbulent than the flow around the wagon.

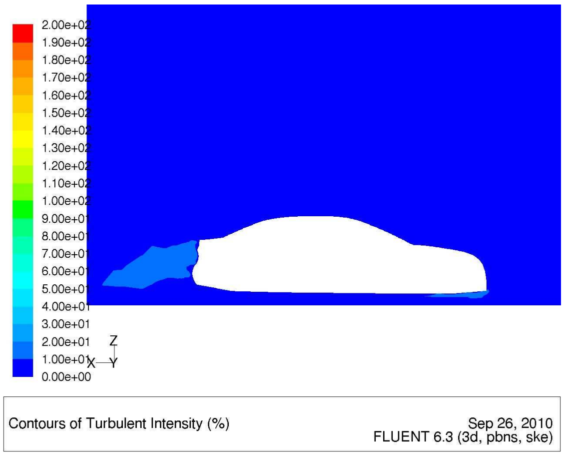

Figure 17. Turbulence intensity for the Sedan in the mid-plane

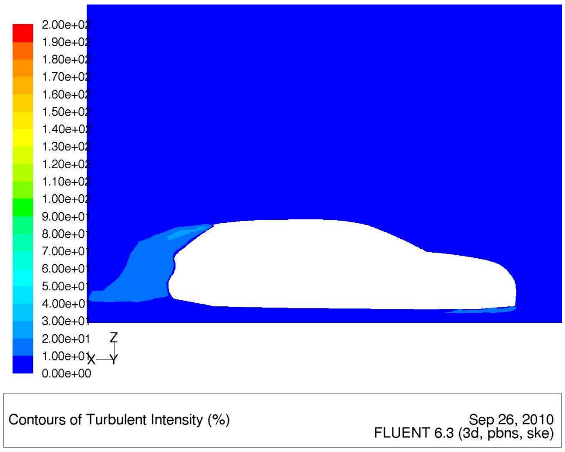

Figure 18. Turbulence intensity for the Wagon in the mid-plane

We can see that the shape of the rear have a big influence in the turbulence around the rear of the car. For the Sedan the maximum intensity of turbulence was about 73%, while for the wagon, the values is nearly around 157%.
As we stated in the first part of this paper, our aim was to evaluate the drag coefficients for the two car shapes. Using Fluent software, we have obtained the values presented in table 4, and we have compared those values to the values found on the internet [7] for cars that have the same external shape.

Body shape	***Simulation value***	***Real value[7]***
Sedan	0.31	0.27
Wagon	0.35	0.31

Table 4. Drag coefficient values for the car shapes

CONCLUSIONS

The pressure distribution on the mid plane of the domain along the middle plane of the vehicle is shown in figure 13, 14. The pressure at the back does not recover to the stagnation pressure level. As a result of the flow separation, the pressures at the back of the car are lower than at the front, which effect creates drag. We shall call this component of the drag force, which results from flow separations, the form drag.
Near the vehicle's body, a thin boundary layer exists, where the air speed is reduced to zero (on the surface). If this boundary layers stays attached to the vehicle (especially at the back), then very low drag coefficient can be obtained, but if the boundary layers separates, then the drag coefficient will usually be much larger (case of wagon)
We have also observed that the flow behind the wagon type car is more turbulent compared to the flow behind the sedan car. This gives us the explanation why a wagon/hatchback car need rear windshield wipers.
Looking at the obtained results, we can conclude that the obtained drag coefficients are closer to those found in literature for the given type of car.
Also, we can conclude that, given the number of elements used in the numerical simulation and the level of details kept on the geometry, the obtained results are satisfactory.
As a future work we will try to evaluate the drag coefficient using more detailed car geometries and also a finer discretization of the finite element model in the zones where we will have a turbulent flow, such as the back of both cars.

REFERENCES

[1] Katz, J. – “***Race cars aerodynamics***”, Bentley Publishers, 1995

[2] Marinari, A. - "***Analisi delle capacita di morphing nella generazione di griglie CFD in campo automobilistico***" - Tesi di Laurea, 2006-2007, Universita Degli Studi di Pisa, Italy

[3] Tabacu Ş., Tabacu I., Macarie T., Neagu E., „***Dinamica Autovehiculelor - Îndrumar de proiectare”***, Editura Universităţii din Piteşti, 2004

[4] *** ***Fluent 6.3.26 User's Guide***

[5] *** http://www.audi.ro

[6] *** http://www.autozine.org/technical_school/aero/tech_aero.htm

[7] *** http://www.cartype.com/pages/3181/audi_a4__2009

[8] *** http://www.up22.com/Aerodynamics.html