CFD analysis for road vehicles - case study

Dan BARBUT*,[1], Eugen Mihai NEGRUS[1]

*Corresponding author
*,[1]"POLITEHNICA" University of Bucharest, Faculty of Transport,
Splaiul Independentei 313, 060042, Bucharest, Romania
danbarbut@gmail.com

DOI: 10.13111/2066-8201.2011.3.3.2

Abstract: *This is a case study on the influence of the lower part of road vehicles on the global drag characteristics. Reducing overall drag by redesigning the lower part of the road vehicles has a potential of almost 20% in the overall drag breakdown, mainly due to the viscous effects and the fluidic interaction of the flow under the car with the typical bluff body flow pattern behind the vehicle. A special parameterization is proposed for the global shape of the sedan car, with respect to the lower part of the body, taking into account most of the specificities of the system. For such a complex interaction, CFD analysis is probably the only efficient tool in order to assess specific design parameterization of a generic car shape. Building on the credibility of such instruments is one of the major goals of this paper. Also, with respect to a target sedan car configuration, examples of successful design strategies are presented. Based on the CFD results, possible strategies to be used in order to reduce viscous drag and global drag characteristics are proposed.*

Key Words*: CFD analysis, drag reduction, fluidic interaction, car aerodynamics*

1. INTRODUCTION

Car aerodynamics is a very complex area for new developments. This is mainly motivated by the importance of the drag reduction with respect to the reduction of the operating costs, and also with respect to the optimization of road vehicles configurations so that new standards of safety are met. Starting with some earlier studies for a more efficient shape for a faster race car, up to a fully optimized body with a very low fuel consumption, road vehicle design has been constantly developed, making usage of the most sophisticated mathematical models and computing hardware ([1]).

Most of the optimization work and concept car development has been initiated almost 100 years ago. Making usage of the basic theory in the fluid dynamics, this type of analysis has been constantly improved, both in the phenomenological aspects taken into account and the capability to include more complex geometries in the simulation process. There are various areas for interest with respect to their potential in the global drag structure for road vehicles. This type of drag breakdown analysis, as influenced by a more advanced approach in aeronautics, highlights the major contribution of 3 global components: shape drag, friction drag and vortex drag. In is important to understand the relation between this type of breakdown, since, for a road car of sedan type, each of the components has a significant importance, as presented in Figure 1.

Therefore, tools to be used in order to optimize a car configuration need to prove their capability to correctly estimate this structure, making usage of well advanced models mainly in turbulence and unsteady fluidic interactions ([2]).

With respect to the current state of the art in CFD analysis, there are several elements making a strong case in favor of this type of analysis a perfect tool for conceptual design.

Extensive usage of CFD for the optimization of a lower part of the car is to be presented in this paper, where specific tools in the pre-processing and post-processing phases of the development are emphasized.

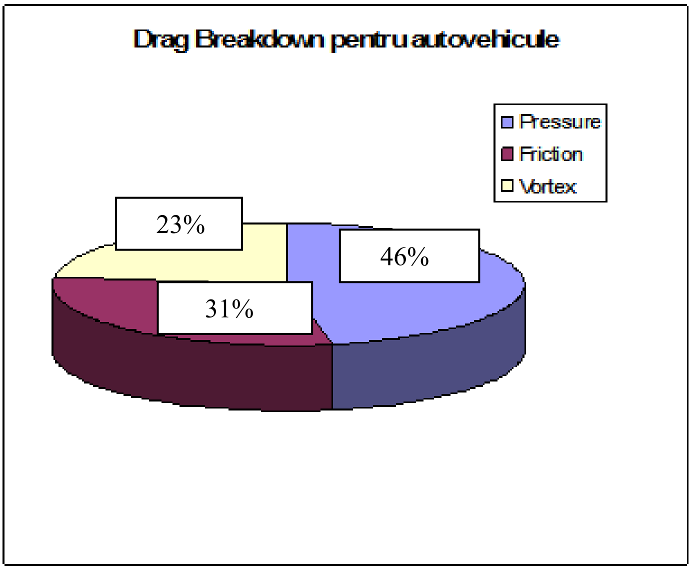

Fig. 1 – Typical drag breakdown for a sedan car

This paper also addresses a very efficient (parallel) implementations of multi-model DxUNSp CFD platform, under continuous development.

Latest IT technologies, mainly in Grid computing environment, have been introduced, and this capability to post-process the very complex output is one of the major features, as presented in Figure 2.

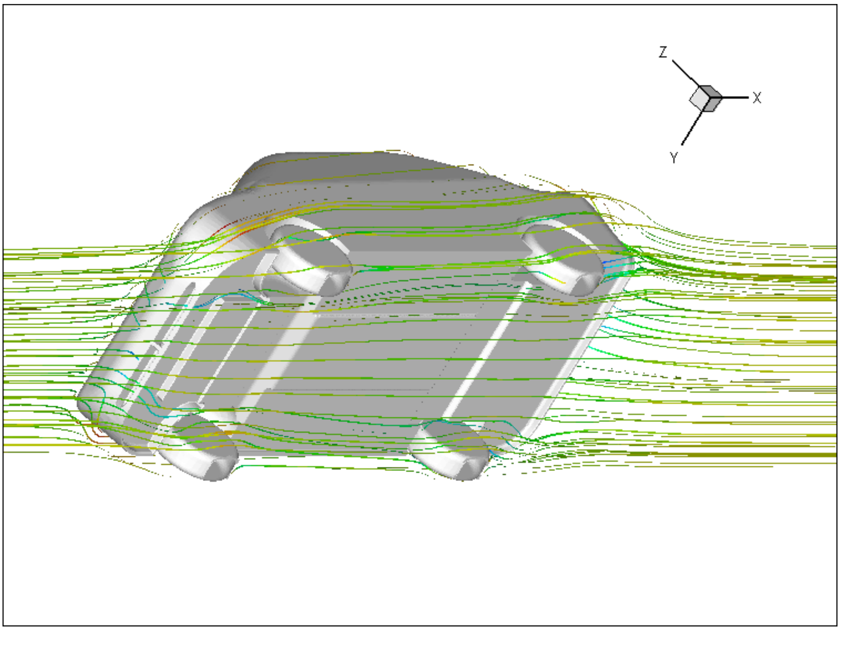

Fig. 2 – Typical CFD flow analysis for the lower part of a sedan car

At the same time, if we consider major components in the car structure in the popular description, from a very complex analysis of such components and cross-correlations with existing databases, a drag breakdown structure is presented in Figure 3 ([1], [2]).

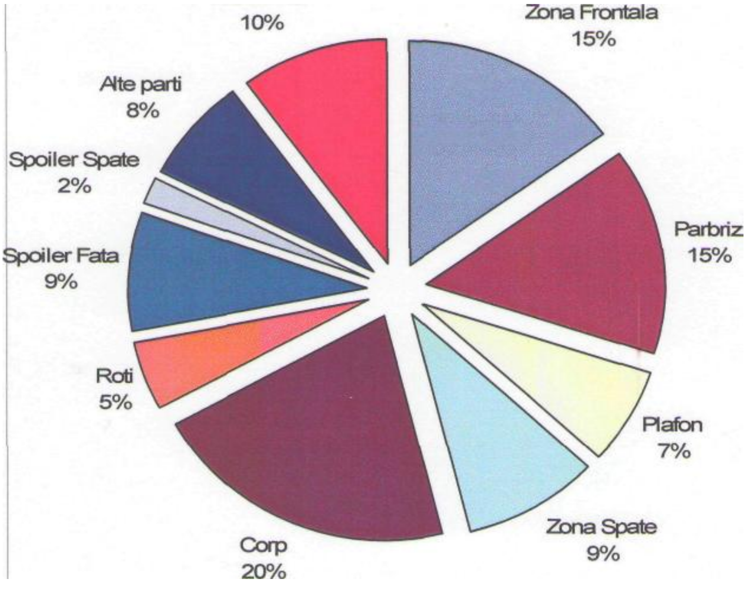

Fig. 3 – Sedan car - component drag breakdown

This is a first order split of the total drag on the main components, without taking into account induced effects, mainly to present the potential of various components in the total optimisation process.

2. CFD ANALYSIS FOR DRAG REDUCTION

There are two principles used to identify an optimum shape for the sedan car when we consider drag reduction as a main objective:

- the car needs to perform its basic functions. Therefore, shape optimization is performed with constraints, coming from this basic parameters of the basic functionality. This includes the total volume, a fixed geometry for the interior, frozen geometry for the wheels track.

- the optimization with respect to drag reduction is based on the need to improve cruise drag, without affecting other dynamic characteristics. This includes the need for a specific correlation between a down force to be obtained at a specific cruise speed, without introducing additional elements (e.g. wings) in order to achieve this.

Therefore, an interesting analysis for the sedan car would be to identify tools and a methodology for global drag reduction, by re-designing the lower part of the car, mainly the floor. CFD analysis is the appropriate tool for the work and this is presented in the next paragraph.

The reference model of the sedan car is presented in Figure 4. This corresponds to a normal geometry that is considered to meet all requirements for certification of a sedan car, both with respect to geometry and dynamic characteristics.

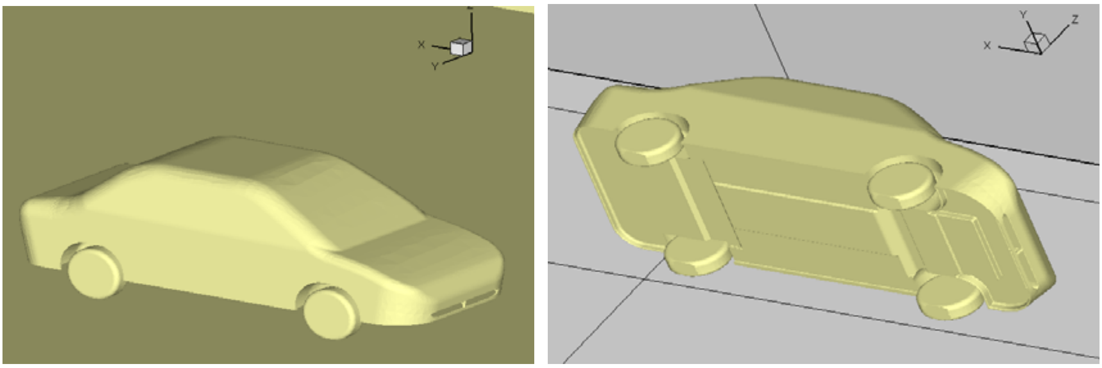

Fig. 4 – Reference configuration - sedan car

For this geometry, based on detailed CFD analysis, we consider a set of global data as basic reference:

Reference speed	V = 25 m/s (90 Km/h)
Reference Reynolds number	Re = 2.5 x 10^6
Length	b = 1.5m)
Reference total drag	CD = 0.2411
Reference lift	CL = 0.5662

The lift and drag calculated values are for a "clean" surface configuration, without all the additional elements existing on the real car. From experience, if we add all additional elements, the a total drag value for this configuration is in the range of CD=0.28 - 0.3.

In order to identify a new shape with lower drag, by only changing the lower part of the body, a parameterization of the shape is proposed in Figure 5.

This parameterization takes into account realistic elements possible to be changed without violating other global functional constraints.

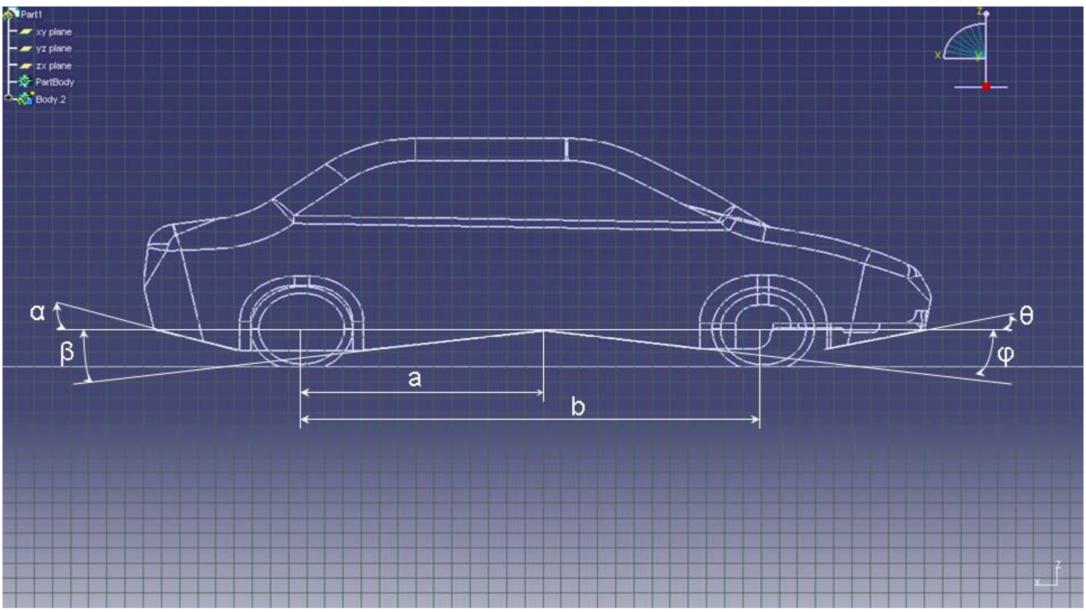

Fig. 5 – Reference configuration - lower side parameterization

Using this parameterization, we look for a new lower car shape so that the global drag is lower, using independent variation of this parameters. Main interest is for the variation of the angular parameters α, β, φ, θ (Figure 5).

The CFD analysis is performed based on a discretisation of the model in tetrahedras, as presented in Figure 6.

The CFD code used is DxUNSp ([3]), based on unstructured domains, with additional option for grid refinement close to the solid surfaces.

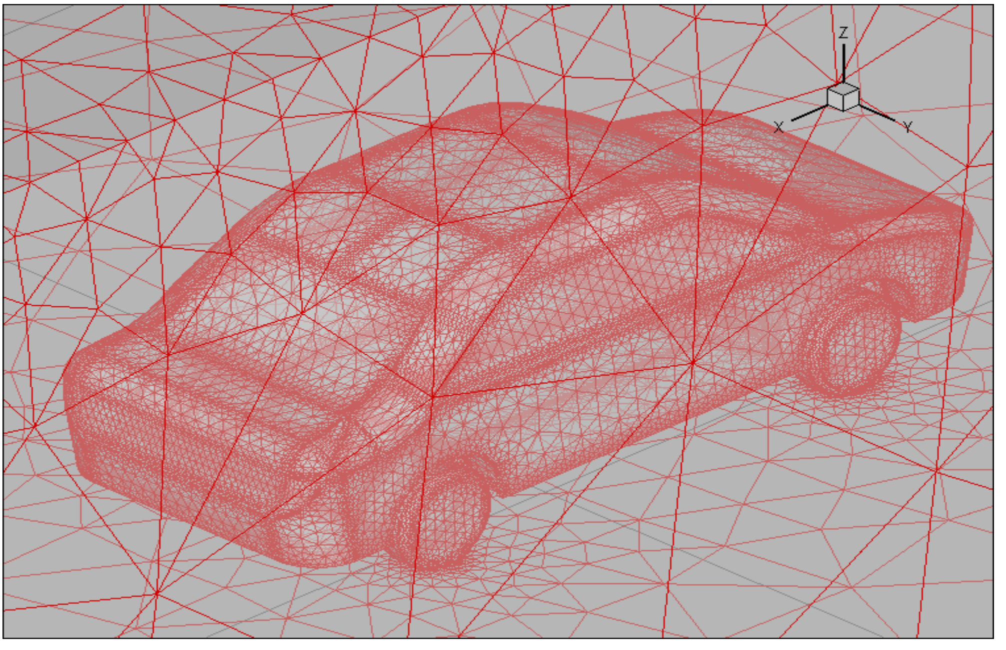

Fig. 5 – Reference configuration - basic discretisation

The basic discretization of the model and surrounding domain is presented in Figure 6. This is characterized by :

Exterior Domain - isotropic distribution with :
average triangle length : 0.15 m;
number of elements : 20.000.

Floor - anisotropic distribution (wheels taken into account) :
average triangle length : 0.025 - 0.15 m;
number of elements : 10.000.

Body of the car - anisotropic distribution on edges :
average triangle length : 0.005 - 0.05 m;
number of elements : 150.000.

Total number of surface elements : 180.208
Total number of volume elements : 1.536.278

In order to use the parallel version of DxUNSp code, domain was decomposed into sub-domains. The code implements Schwartz ([3]) domain decomposition algorithm, with a minimization of overlapping regions. Also, turbulence is considered using current state of the art k-eps model ([4]).

The computational domain was divided in 8 domains using MeTHIs software, with a global work distribution of 1:1.02. The code runs in URANS version in explicit time integration algorithm ([3]).

Basic analysis of various configurations were performed in order to evaluate the individual contribution of the lower side components to the global drag. CFD results have been post-processed using TecPlot software and complex representations are available for the flow pattern on the car and inside the domain. A typical result of this analysis has been presented in Figure 2.

A complex representation of the velocity distribution on the surface of the car, together with a complex streamline localization in presented in Figure 6.

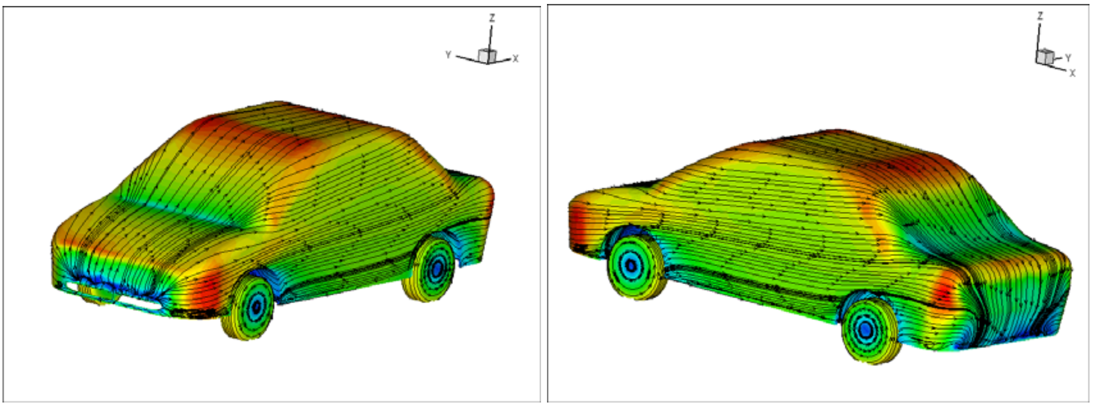

Fig. 6 – Reference configuration - Velocity and streamlines distribution

Based on this type of analysis, independent effect of the angular parameters has been recorded with respect to their potential for overall drag reduction.

A sinthesis of this preliminary analysis is presented in Figure 7.

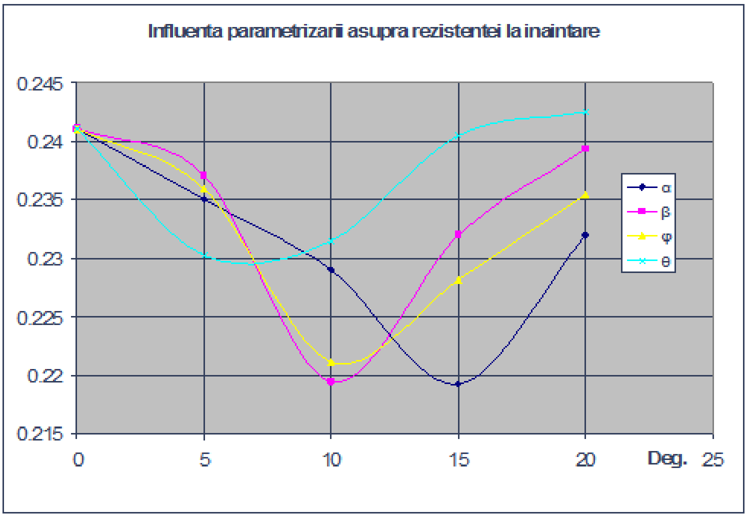

Fig. 6 – Reference configuration - Velocity and streamlines distribution

As an important remark, the individual variation of the angular parameters has a minimum value with respect to the total drag. This is an important factor to be considered in overall optimization process in future work.

3. CFD RESULTS ON OPTIMIZED CONFIGURATION

From the set of individual variations, it was important to identify a global optimum solution and to investigate the flow aspects coming from the CFD analysis. Two areas of major interest have been considered, front and rear spoilers.

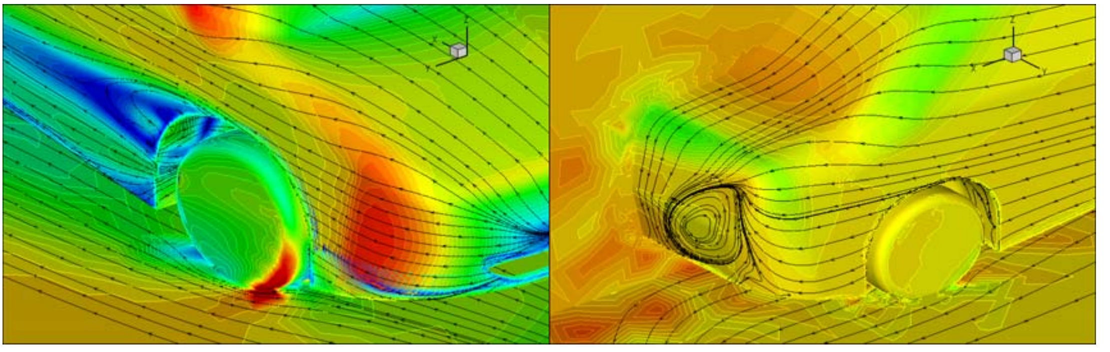

Fig. 7 – Front and rear spoiler flow analysis

The CFD analysis performed was also intended to investigate the flow pattern under the car. This is very difficult to evaluate in real conditions (also in wind tunnels).

The CFD analysis is the only possibility for a detailed analysis, as this is presented in Figure 8.

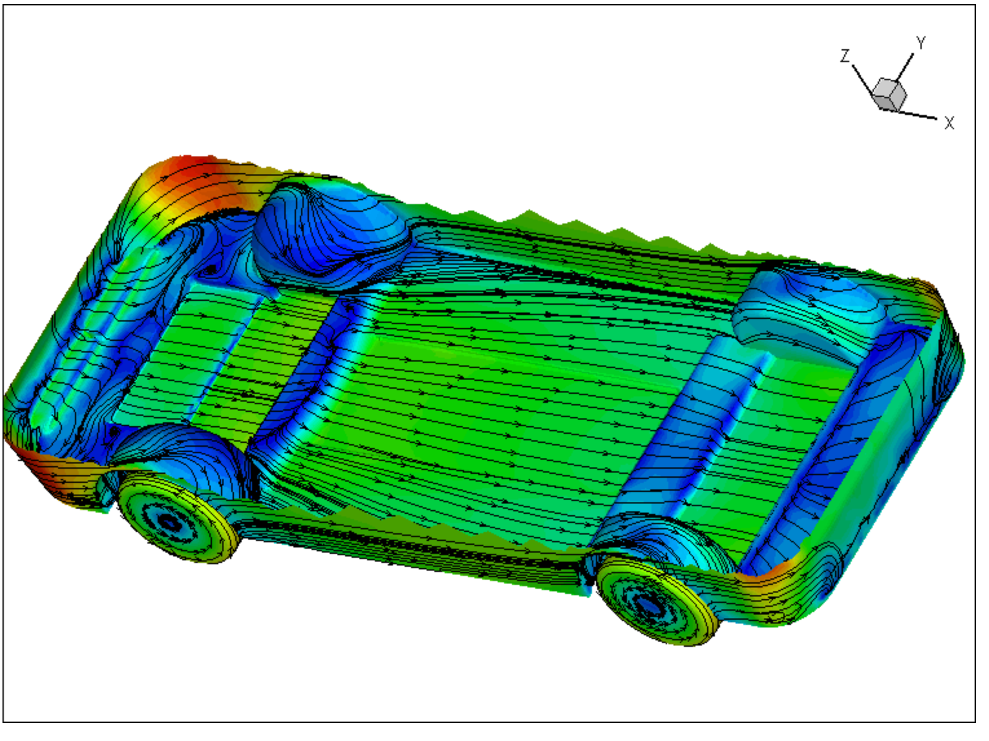

Fig. 8 – Front and rear spoiler flow analysis

This result obtained by post-processing the CFD results enables a strategy formulation with respect to the global optimization of the sedan car taking into account the flow pattern on the underside.

6. CONCLUSIONS

The objective of this analysis was to demonstrate the importance of the CFD analysis in car optimization, taking into account the flow under the car. This is the only available tool for this type of analysis with the potential to introduce important changes mainly to the optimization strategy in car industry.

Based on the work performed as part of the PhD thesis, several effects linked to the front and rear part of the lower side of the car have been identified. Their variation is presented in Figure 9.

For this particular sedan shape, drag has been reduced from the reference value of 0.2411 to 0.2105 (12.7%), a very promising value in the car industry.

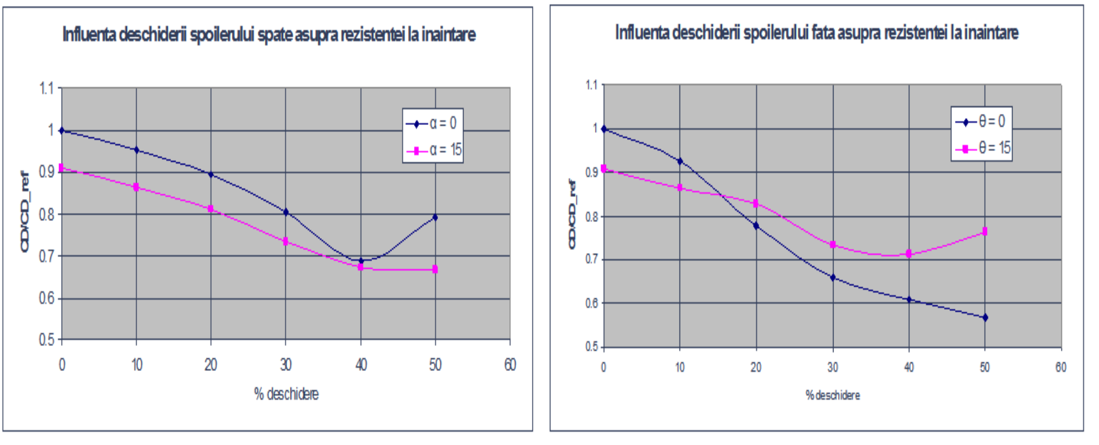

Fig. 9 – Spoiler redesign - CD global influence

Global optimization is possible using appropriate optimization tools. This was not the purpose of this paper.

However, such a complex approach is to be considered for further work.

Acknowledgement

This work has been performed for the partial fulfillment of the requirements of the PhD thesis of the main author.

REFERENCES

[1] W. H.Hucho, “Aerodynamik des Automobils”, VDI Verlag, 1994

[2] T. Kobayashi, K. Kitoh, “A Review of CFD Methods and Their Application to Automobile Aerodynamics”, SAE Paper 920338.

[3] C. Nae, “Flow Solver and Anisotropic Mesh Adaptation using a Change of Metric based on Flow Variables”, AIAA Paper 2000-2250.

[4] D. C. Wilcox, Turbulence Modeling for CFD, ISBN 1-928729-10-X, 2nd Ed., DCW Industries, Inc.., 2004.