Drag Coefficient Optimization for a Sports Car Using the Coupling Between LS-DYNA® ICFD Solver, LS-OPT® and DEP MeshWorks Software

M. Seulin[1], M. Le Garrec[1], A. Poncet[1], I. Çaldichoury[2], K. Gudlanarva[3]
[1]*DynaS+, Toulouse – France*
[2]*ANSYS LST, Livermore, CA – USA*
[3]*Detroit Engineered Products, Troy, MI - USA*

Abstract

Vehicle aerodynamics are one of the key points allowing to improve the vehicle dynamic behavior, to improve performance and to reduce fuel consumption. The vehicle aerodynamics have been studied in wind tunnels for several decades. Numerical simulations are increasingly used in addition of physical testing and permit to increase the number of design experimentations with cost and time savings.

When CFD engineers are looking into optimizing the global aerodynamics of a car, numerous factors are taking into considerations. A car is a very complex assembly that must fit with multi-physical requirements updated along the vehicle project (design aesthetic, crash safety, weight, vibrations, noise, performances, design manufacturing, etc.) to find the best compromise according to initial specifications.

DynaS+, ANSYS-LST and DEP are working closely with automakers across the world on various applications. Often, the automakers are sharing their work in conferences only several years after for obvious innovative competitive reasons. The aim of this work is to demonstrate what the current innovative technologies are, and methodologies used on aerodynamic applications using open source sports car data.

Like in the majority of aerodynamic studies, in the present work, the objective was to reduce the aerodynamic drag coefficient of our model. A design optimization was performed on the initial design with the help of the advanced morphing capabilities of the DEP MeshWorks© solution coupled with the optimization software LS-OPT and the Incompressible Computational Fluid Dynamics (ICFD) solver LS-DYNA.

Introduction

Today, two of the main engineering guidelines for automakers according to governmental regulations are the occupant safety and the environmental impact of the car. To reduce the environmental footprint of the vehicle, important work is done on new propulsion energies, internal combustion engines, body weight, and other innovative items. The engineering objective is to reduce the fuel consumption and the gas emissions, while maintaining the user safety and an attractive design.

Vehicle aerodynamics are one of the key points allowing to update the vehicle dynamic behavior, to improve performance and to reduce fuel consumption. The vehicle aerodynamics have been studied in wind tunnels for several decades. Numerical simulations are increasingly used in addition of physical testing and permit to increase the number of design experimentations with cost and time savings.

When CFD engineers are looking into optimizing the global aerodynamics of a car, numerous factors are taken into considerations. A car is a very complex assembly that must fit with multi-physical requirements updated along the vehicle project (design aesthetic, crash safety, weight, vibrations, noise, performances, design manufacturing, etc.) to find the best compromise according to initial specifications. In the automotive industry, aerodynamics is generally not the priority taken into consideration (depending on the automakers) whereas it is more important for plane manufacturers for obvious reasons. Nevertheless, even for a ground vehicle, they remain a crucial factor of performance and fuel consumption as shown in reference [1]. Designers and engineers usually work together to find a compromise between aesthetics and performances.

DynaS+, ANSYS-LST and DEP are working closely with automakers across the world on various applications. Often, the OEMs are sharing their work in conferences only several years after for obvious innovative competitive reasons. The aim of this work is to demonstrate what the current innovative technologies are, and methodologies used on aerodynamic applications using open source data.

A CAD model looking like a reference sports car was selected and downloaded from a free of rights database. It is not the actual car CAD model; however, its resemblance makes it the perfect demonstrator. This study is demonstrating current innovative technologies and methodologies on a global aspect, not a local and detailed one, leaving this work to the real OEMs.

Like in the majority of aerodynamic studies, in the present work, the objective was to reduce the aerodynamic drag coefficient of the model. A design optimization was performed on the initial design with the help of the advanced morphing capabilities of the DEP MeshWorks© solution coupled with the optimization software LS-OPT and the Incompressible Computational Fluid Dynamics (ICFD) solver LS-DYNA. The chosen sports car CAD model is shown below in Figure 1.

Figure 1 - Picture of the CAD model used for the study

In the following sections, the applied methodologies are presented starting with the design parameterization of the rear part of the vehicle using DEP MeshWorks©, then the coupling with LS-OPT and with the LS-DYNA ICFD solver. The detailed results include a comparison between the initial configuration and the optimized one.

Methodology

The ICFD solver of LS-DYNA has its own 3D fluid volume generator integrated within the solver. The required input of the solver is a triangle skin mesh of the studied model with the ICFD settings. The user remains free to input a 3D tetrahedral volume mesh, but it is not necessary, as LS-DYNA is able to generate it automatically. This automatic mesh generator also permits to use remeshing and mapping of the solution directly during calculations if the mesh deformations lead to poor CFD equations resolution.

The skin mesh for ICFD calculations was created using DEP MeshWorks©. It is a CAE platform having powerful tools to save drastic times in the modelling work part of the CAE engineer. Based on the CAD model, some cleaning and simplifications could be performed using dedicated tools for CFD meshes creation within DEP MeshWorks©: deletion of small features, and automatic wrapping of the external skin for CFD meshing.

Some features disrupting the global answer given by LS-DYNA with the wanted mesh size were simplified: the hood lines, the two tailpipes, the diffusor, the front face air intakes, the wheels and the careenage.

Basic meshing was performed segregating two principal areas: the front, where the flow is mostly laminar, and the back, where detachments and eddy shedding can generally be observed, as shown in the work of Mohd Nizam Sudin [1]. This led to a tentative mesh size of 80mm on the front side, and 10mm on the back. For the ICFD 3D mesh generator to work properly, this skin mesh needs to have the following characteristics:

- Unstructured: most triangle mesh generator, including DEP MeshWorks©, can chose between structured and unstructured;
- Homogenous: mesh size transitions need to be smooth; any small features must be properly integrated in the mesh;
- Good quality: the most relevant metric for triangle mesh quality is the aspect ratio; for this model, no aspect ratio was allowed to be above 6.

The basic skin mesh which was used can be seen here below:

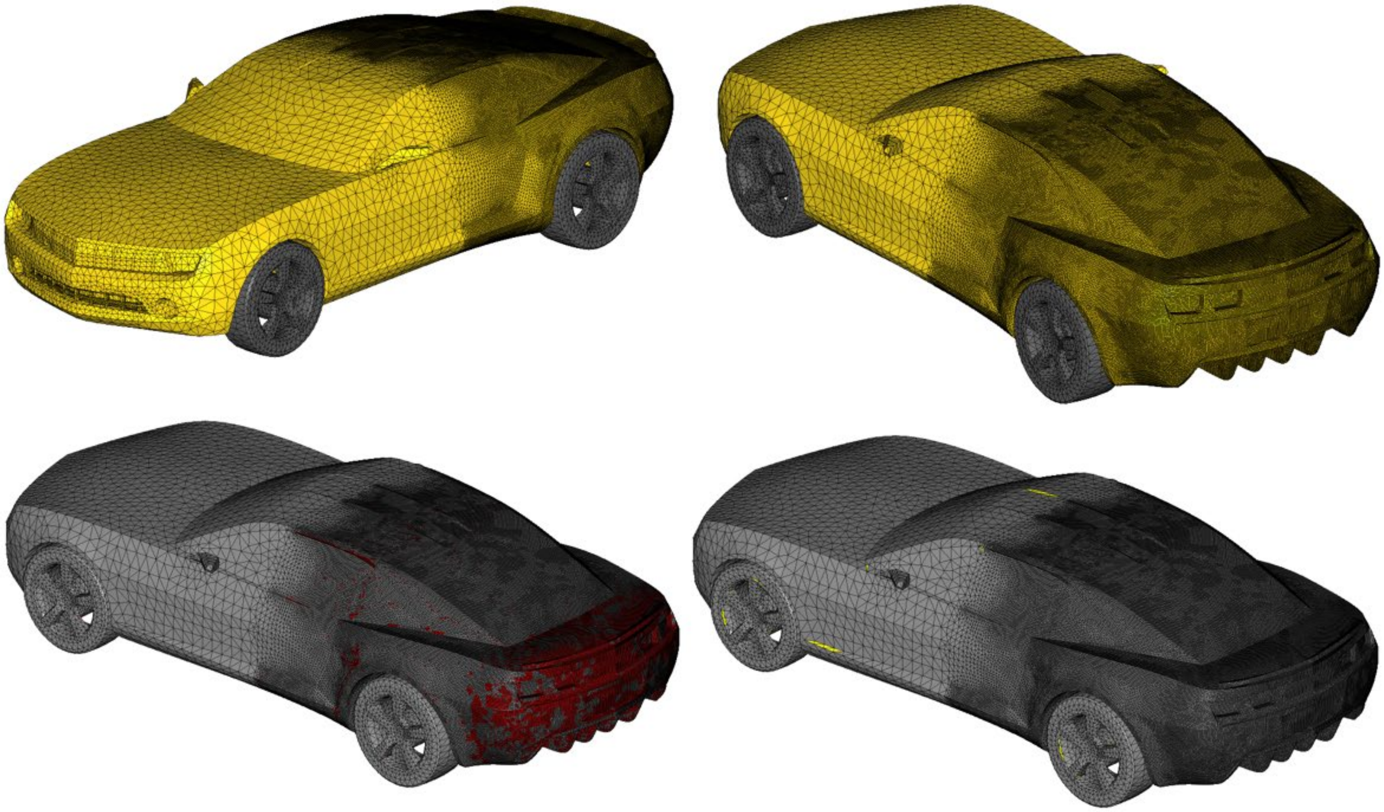
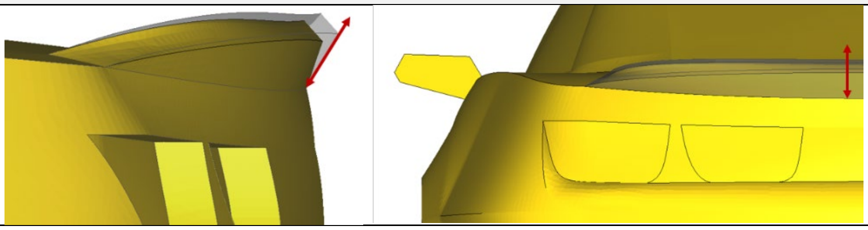
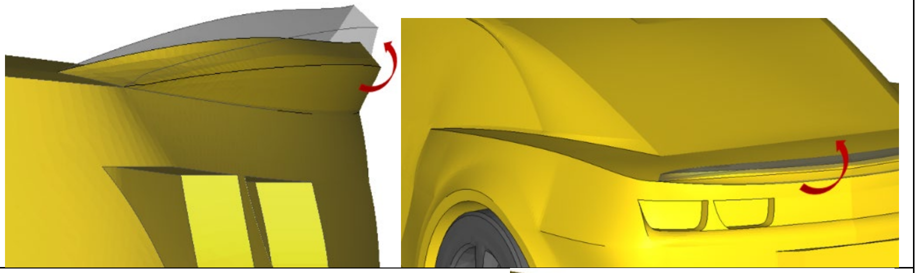
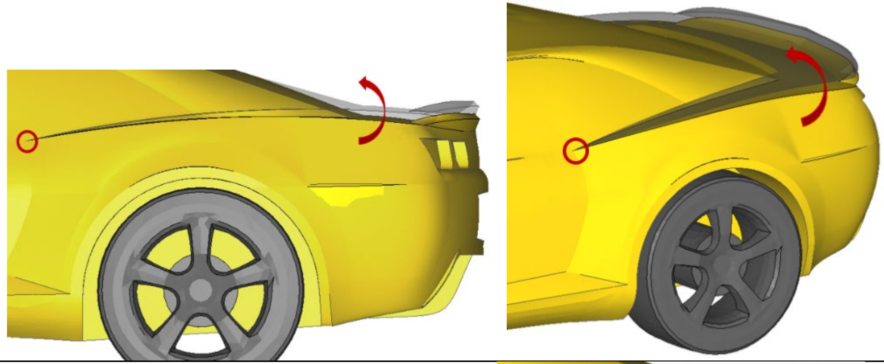
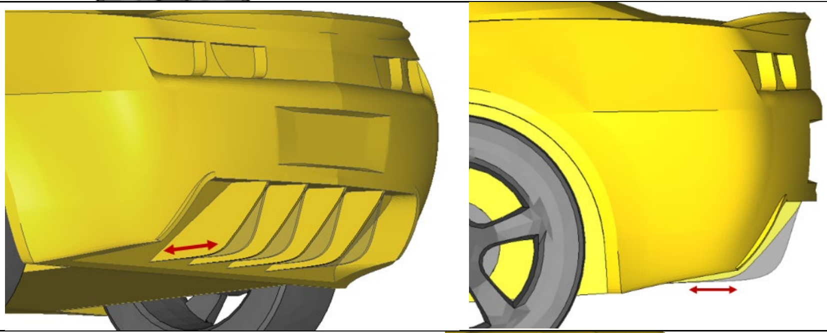
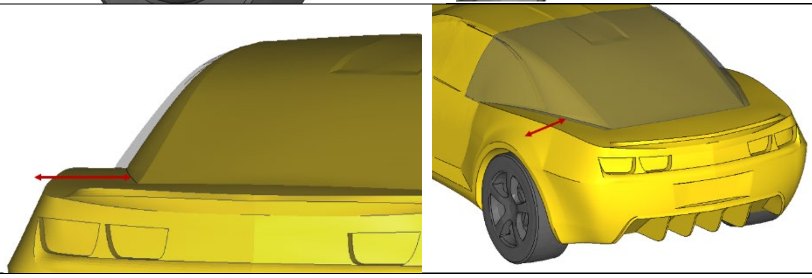


Figure 2 : Global mesh front and rear, element size under 10mm and elements with aspect ratio over 3

The next step was to identify the design regions that, aerodynamically speaking, could influence the drag force. It was decided to focus the study on the rear part of the vehicle, especially the spoiler, the rear glass configuration and the hood levelling as shown in the work of Krishnani, P. N. [2]. In order to explore the expected design space and being able to create a Design Of Experiments (DOE), advanced morphing capacities of DEP MeshWorks© coupled to LS-OPT were used, as in the work realized by C. Goubel [3]. DEP MeshWorks allows to easily parametrize FE and CFD meshes in order then to generate DOE. LS-OPT is used to do the sampling of the design space defined within DEP MeshWorks by the parameters' boundaries.

A large variety of morphing methods such as FreeForm, Field Based, Control Blocks (lower and higher order), Direct parabolic, Spherical, and Polycube morphing are available for various applications.

For this case, the Freeform morphing algorithm was retained for all parameters. The parameterization within DEP MeshWorks© is user-friendly. The parameters that were set for this study are described in the table below.

Parameter name	Value range	Description	Zone
Spoiler height	-0.002/0.015m	Translation of the spoiler top face in the length direction	
Spoiler angle	-2°/5°	Rotation of the spoiler top face around the spoiler bottom and the transversal car axis	
Chest height	-0.6°/1°	Rotation of the chest face around the nodes surrounded in red and the transversal car axis.	
Winglet angle	-0.06/0.02m	Translation of the winglet corner nodes in the longitudinal car axis.	
Top side Width	-0.02/0.02m	Translation of the side edge in the transversal car axis.	

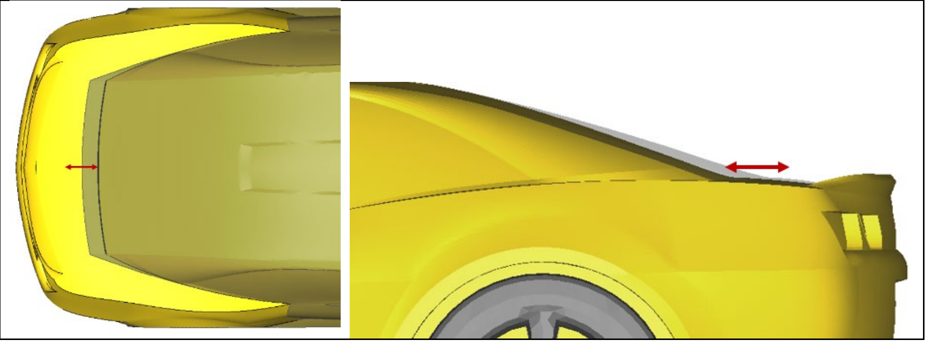
Bottom side width	-0.003 / 0.003m	Translation of the side edge in the transversal car axis.	
Rear glass angle	-0.05/0.05m	Translation of the bottom edge of the rear glass in the longitudinal car axis	

Table 1 - Description of the parameters involved

In this parameterization, the body architecture and dynamical vehicle behavior were not considered. Only the CFD engineer's point of view was taken into account. A comparison between the minimum and the maximum designs that can be obtained with the parameterization can be seen below Figure 3. It gives a good idea of the design space involved for the DOE.

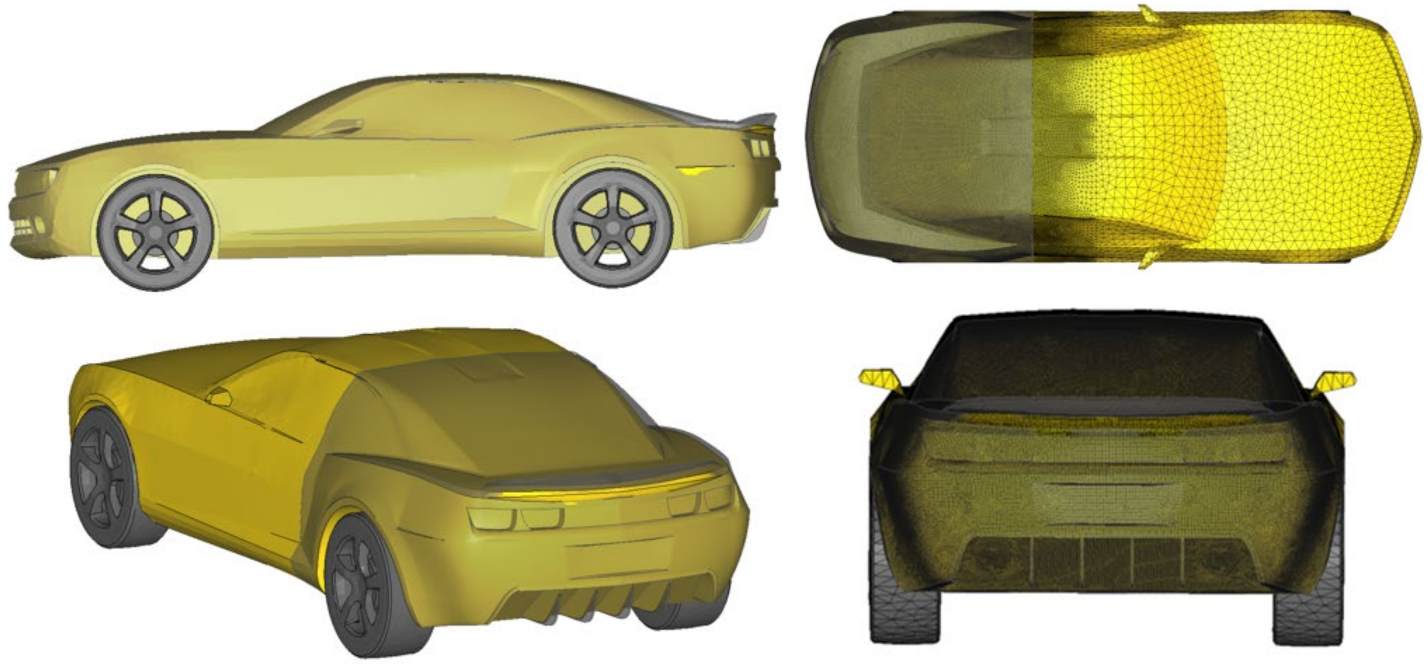


Figure 3 - Preview of the minimum and maximal designs according to the parameterization

Once the parametrization within DEP MeshWorks© was set, a LS-OPT optimization coupled with this parametrization was created. A Meta-Model based with a single stage optimization was defined using a Feedforward Neural Network. A DOE of 55 designs was automatically created. Each design applies a different value from the range defined for each parameter. There are 7 parameters in the LS-OPT study which correspond to the 7 parameters defined in the DEP MeshWorks© phase. A "MW_Mesh_stage" is calling DEP MeshWorks© to create required designs from the initial mesh, according to the sampling done by LS-OPT.

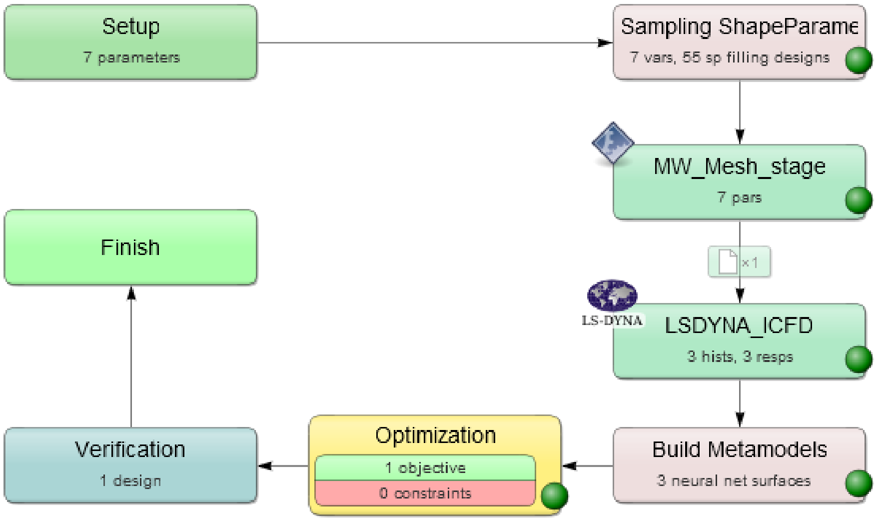


Figure 4 - Diagram of the optimization loop within LS-OPT canvas

CFD simulations were performed by the ICFD LS-DYNA solver. For every set of parameters, DEP MeshWorks© created a skin mesh of the vehicle, displayed Figure 6, directly readable by the main simulation model. This total model included a large domain (Figure 5), as used in the work of M. Le Garrec [4], with 3 vehicle lengths upstream, 9 downstream, 1 sidewards, and 2 upwards; a small gap was kept below the vehicle wheels. The LES turbulence model was chosen, with a boundary layer mesh control (Figure 7), as recommended in the LS-DYNA user manual [5]. A uniform inflow velocity of 26m/s was used, corresponding to roughly 90km/h or 55mph. The air properties are the usual values in normal conditions of pressure and temperature: $\rho=1.28 kg/m^3$, $\mu=1.7x10^{-5}$Pa.s. The initial mesh size was selected in order to reduce computation times, especially with the coarser front zone of the vehicle, displayed. For better mesh transitions in the rear zone, a binding mesh size was used in the form of a meshed surface wrapping the back of the vehicle.

Mesh quality is fundamental in CFD computations. The ICFD solver analyses the George quality criterion (LS/V), which must stay as close to 1 as possible. In addition, the use of an LES turbulence mode recommends y^+ (non-dimensional tangential velocity in the first element on the surface) values to remain close to 1. This last criterion is not managed here for time saving considerations, but the global drag values are unaffected.

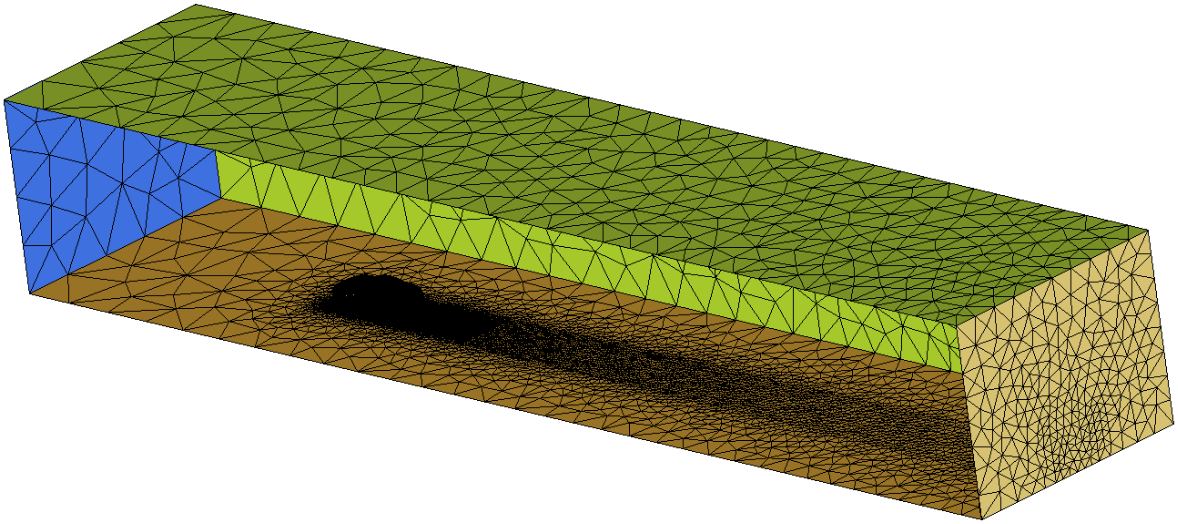


Figure 5 : CFD domain used with boundary conditions: inflow (blue), outflow (yellow), wall (orange), freeslip (green)

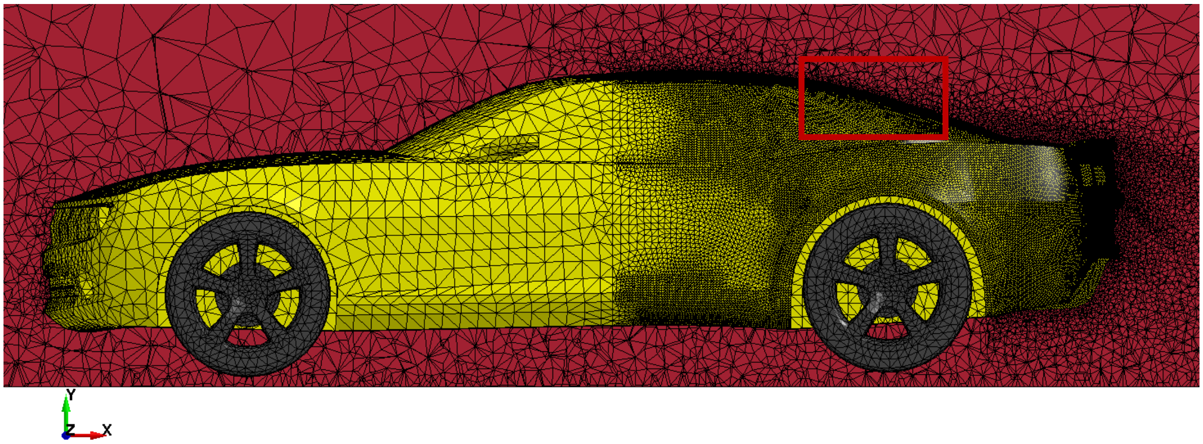


Figure 6 – Mid section of the automatic 3D fluid mesh based on the car model 2D skin mesh

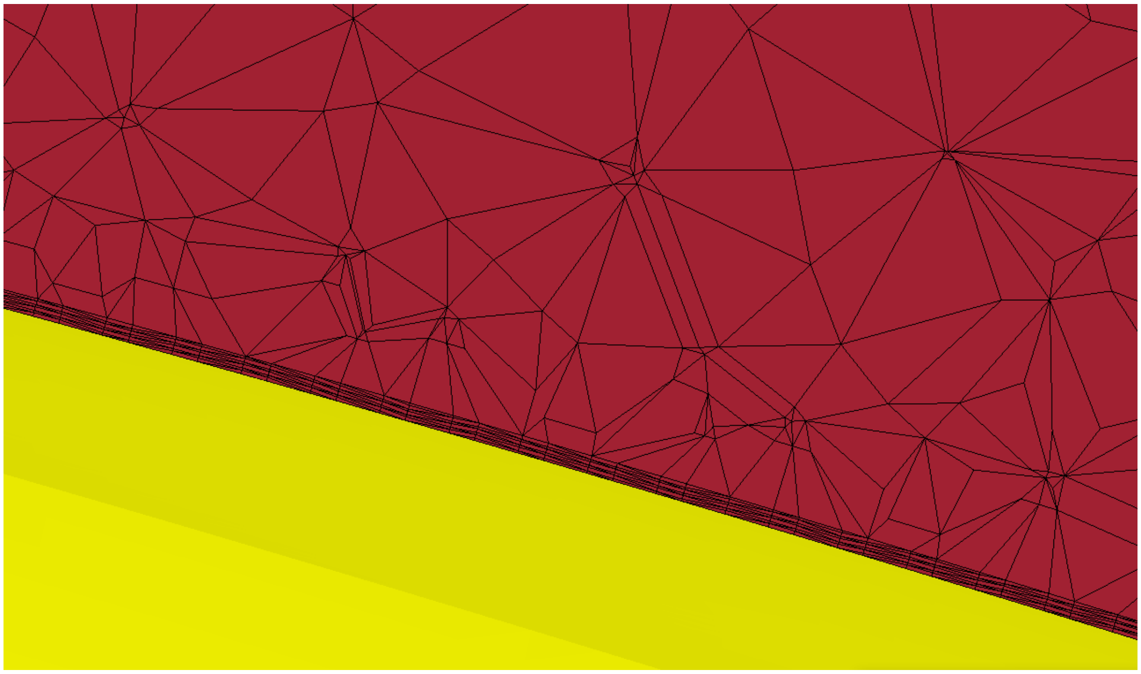


Figure 7 : Close up view on the boundary layer around the car

LS-OPT optimization objective was the Cd of the vehicle or drag coefficient. This coefficient is calculated as:

$$C_d = \frac{F_x}{\frac{1}{2}\rho A u^2}$$

With: F_d the measured drag force
$\rho = 1.28 kg/m^3$ the air density
$A = 2.6 m^2$ the vehicle cross-sectional area
$u = 26 m/s$ the inflow velocity

The simulation time for all computations was 0.04s. After an initial phase during which the flow develops around the vehicle, a steady phase could be observed. The instantaneous drag force was output during the last 0.01s of the simulation, and this curve was fed to LS-OPT, which computed an average force for the optimization. Figure 8, Figure 9 and Figure 10 show the results obtained for the initial design in the longitudinal plane of symmetry.

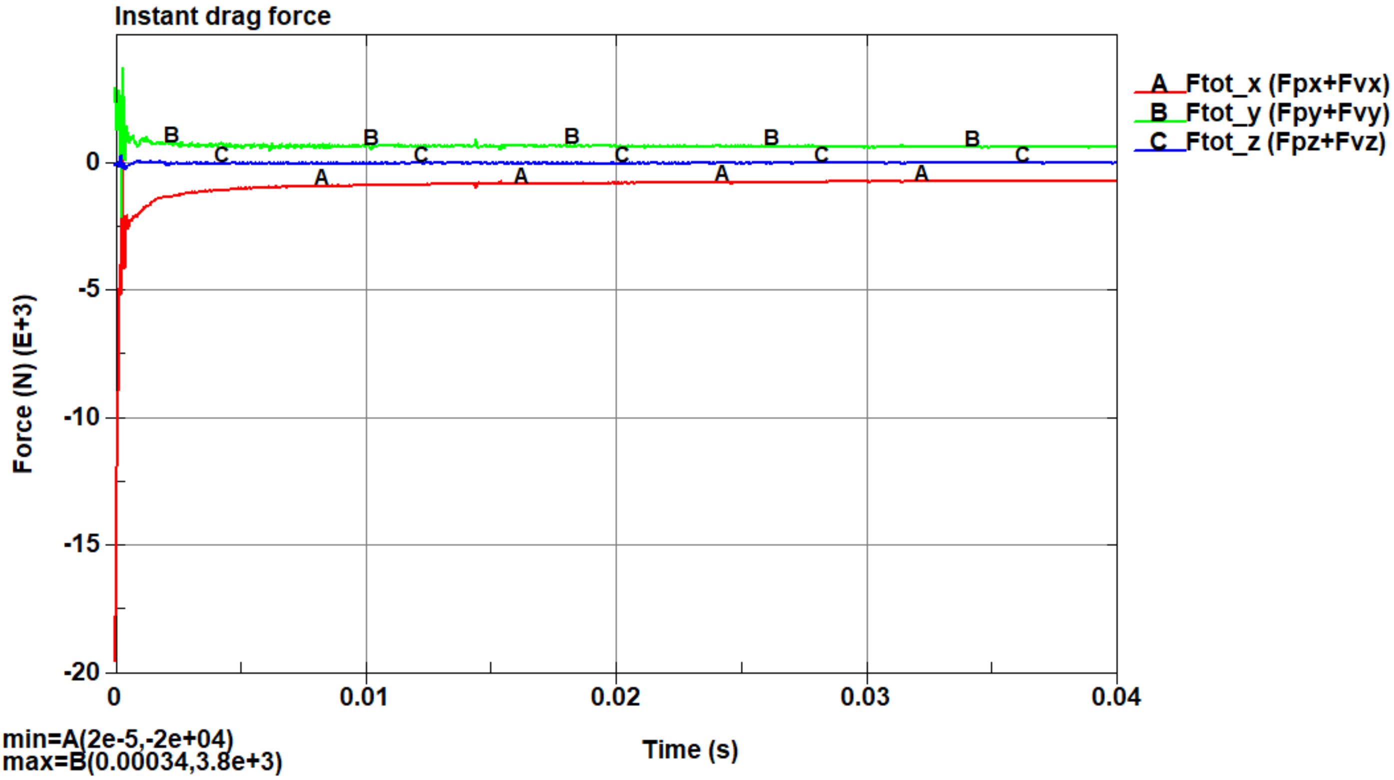


Figure 8 : Instant drag force on the initial design

Figure 9 : Fluid velocity on the initial design

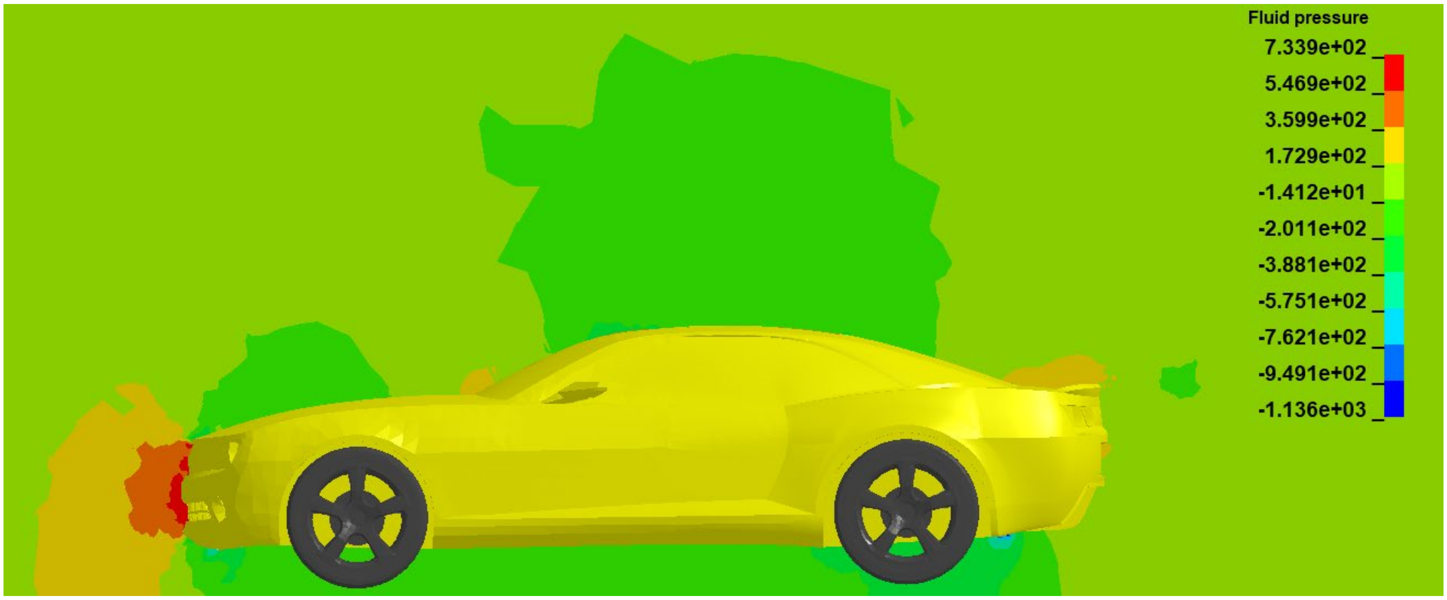


Figure 10 : Fluid pressure on the initial design

Results

The optimization study ran successfully, with a number of 55 points, 55 different designs. Each of them ran for two hours on 28 cores machines for a total optimization time of 2 days.

As shown on Figure 11, the meta model precision and adaptivity is good. Approximate responses on the metamodel surface is relatively equal to the one computed by LS-DYNA for the sampling points.

On Figure 12, the sensitivity shows that several parameters chosen for this optimization have an impact on the drag coefficient C_d. The sensitivity results are based on the metamodel. The confidence range is correct and significantly smaller than the influence of the most influent parameters. The spoiler angle and the chest height are the most influential parameters. If the value of those parameters is increased, then C_d is too.

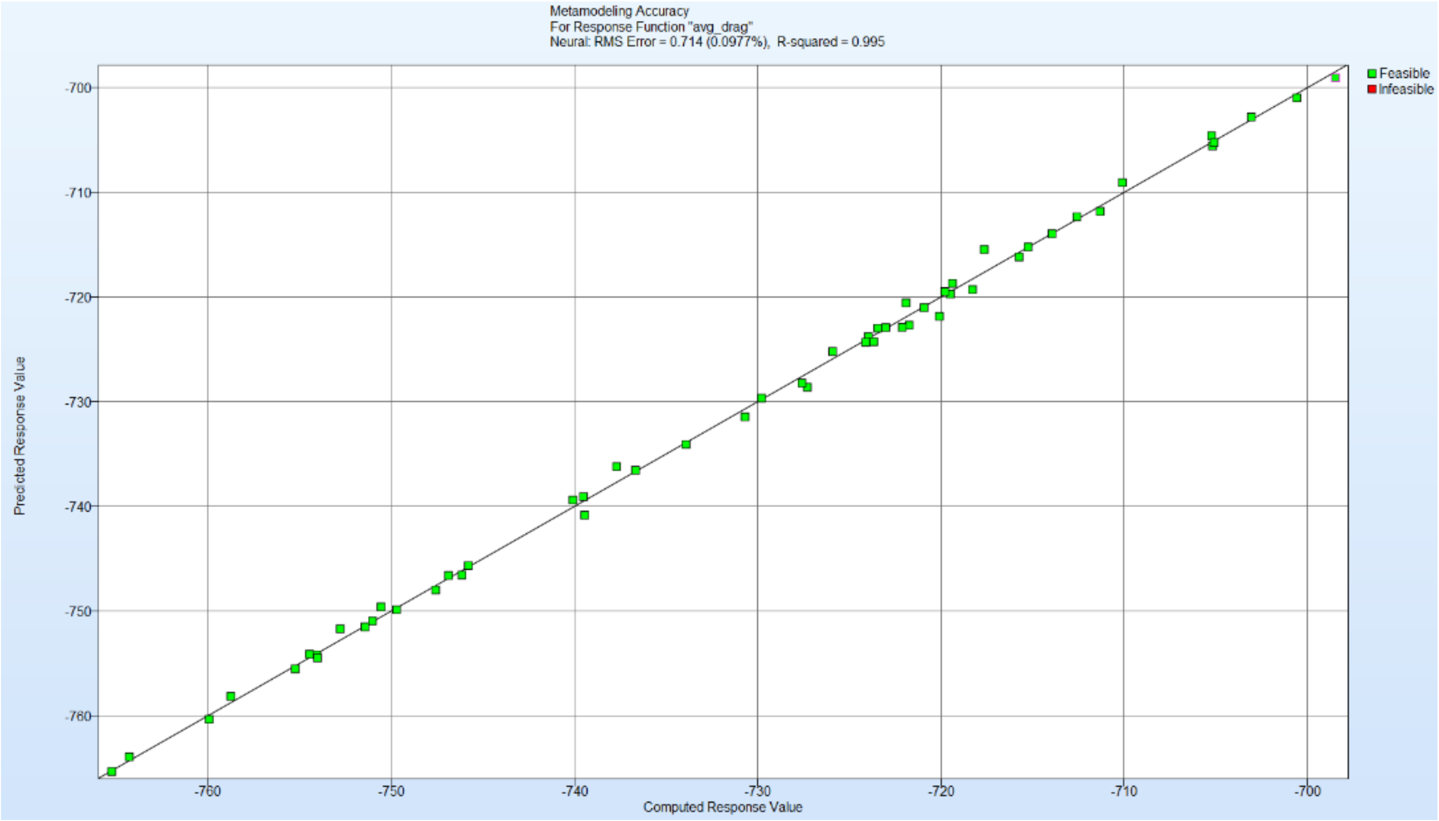


Figure 11 : Model answer precision

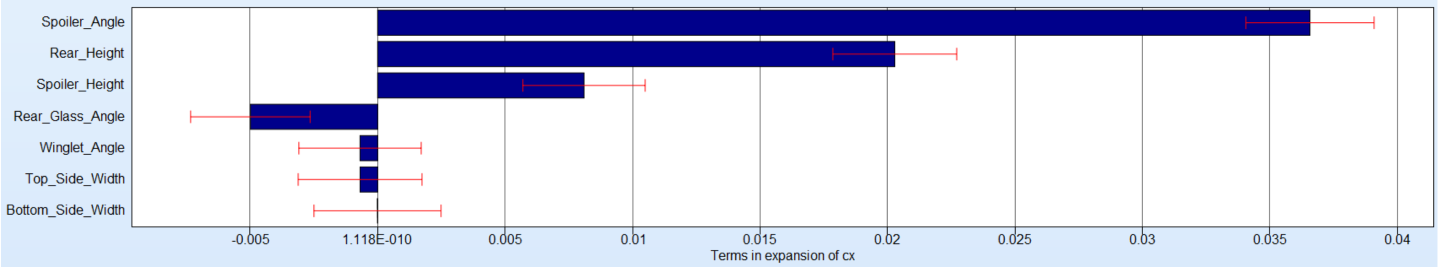


Figure 12 : Parameter influence

The optimal design found by LS-OPT with the objective to reduce the C_d, shows that the spoiler angle and the chest height are reduced at almost their maximum. The spoiler length is also increased. The best design shows an improvement of 0.012 on the drag coefficient corresponding to about a 2% improvement. The drag force is reduced from 711N to 698N.

On Figure 13, a visual comparison of the initial and the optimized design can be seen. Several zones are morphed, the chest is lower than the initial design, the spoiler angle has been changed, the winglet angle is more pronounced. The yellow model is the optimized design and the green one is the initial one.

The fluid velocity along the X axis in the plane of symmetry between the initial and optimized is displayed Figure 14 and Figure 15.

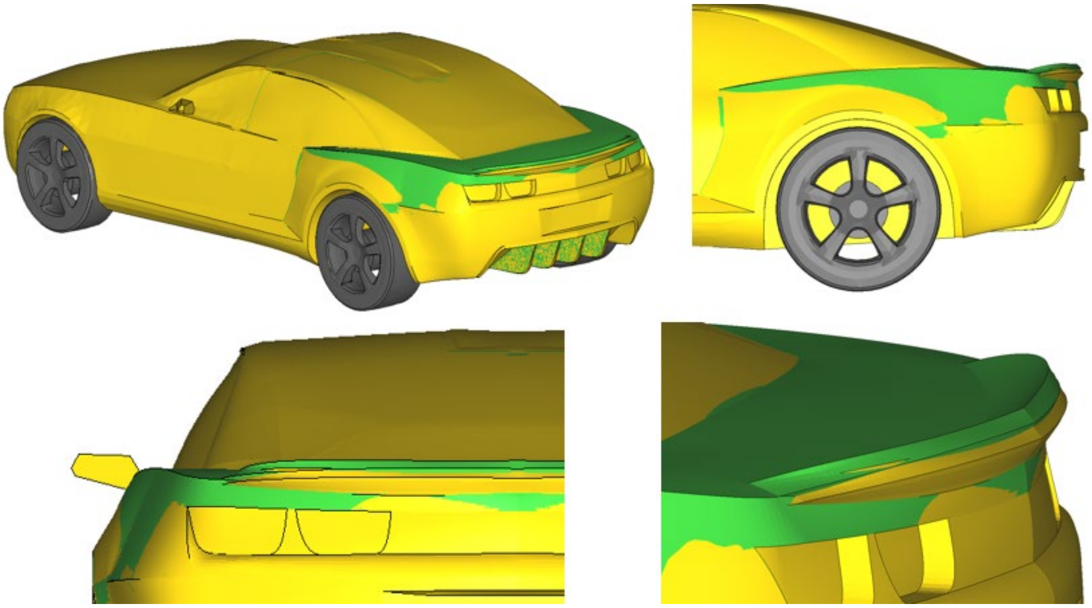


Figure 13 : Initial (green) and optimized (yellow) design comparison

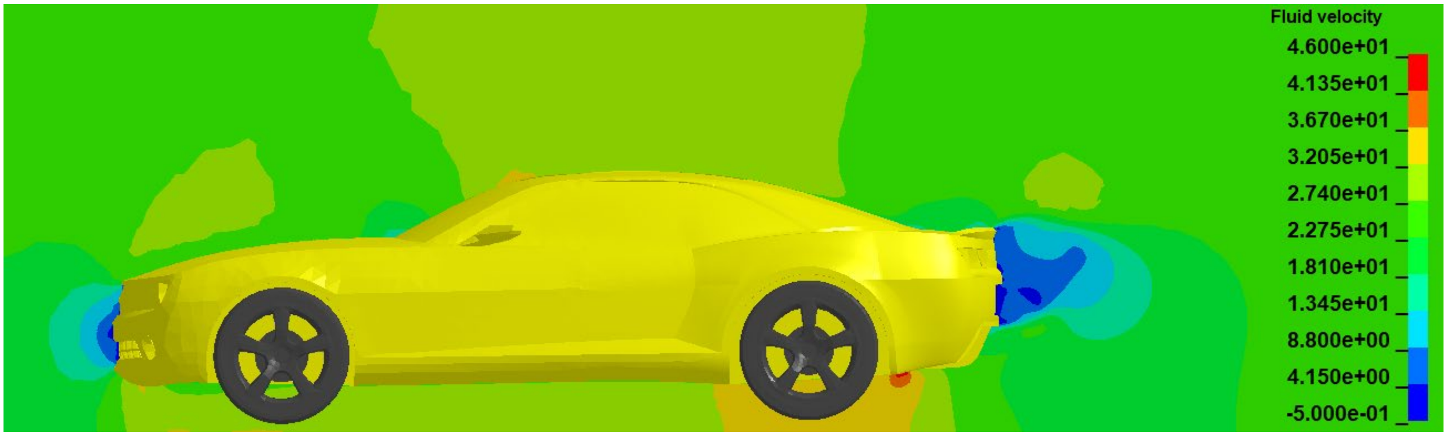


Figure 14 : Fluid velocity for the initial design

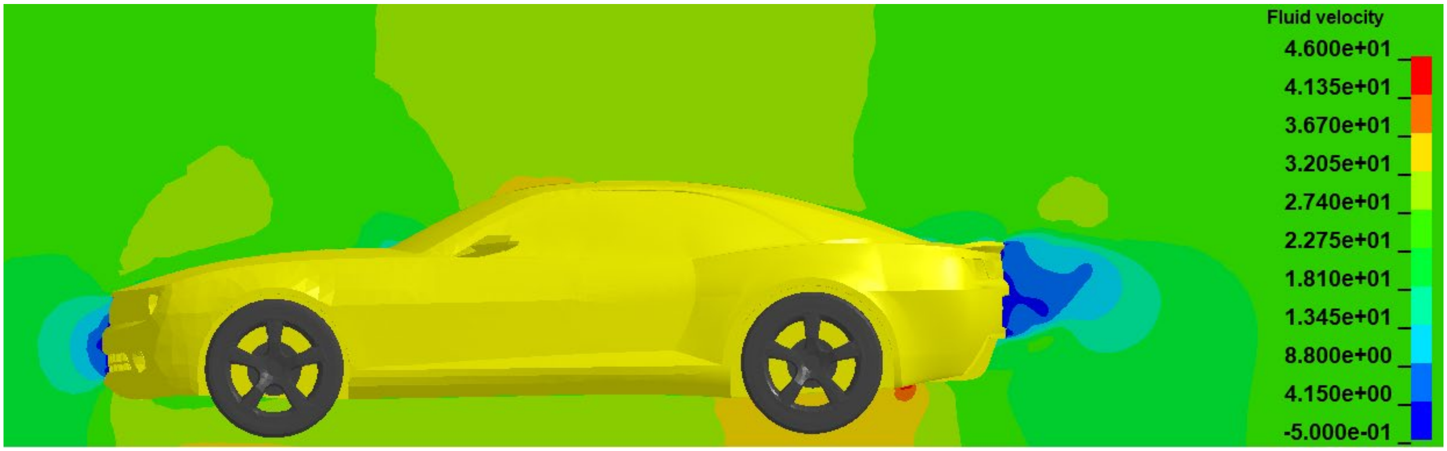


Figure 15 : Fluid velocity for the optimized design

Figure 16 shows a comparison between the initial and optimized streamlines. Trajectories are slightly different; the optimized ones present less turbulence than the initial ones. The initial configuration is displayed in grey and the optimized one in displayed in color from blue to red.

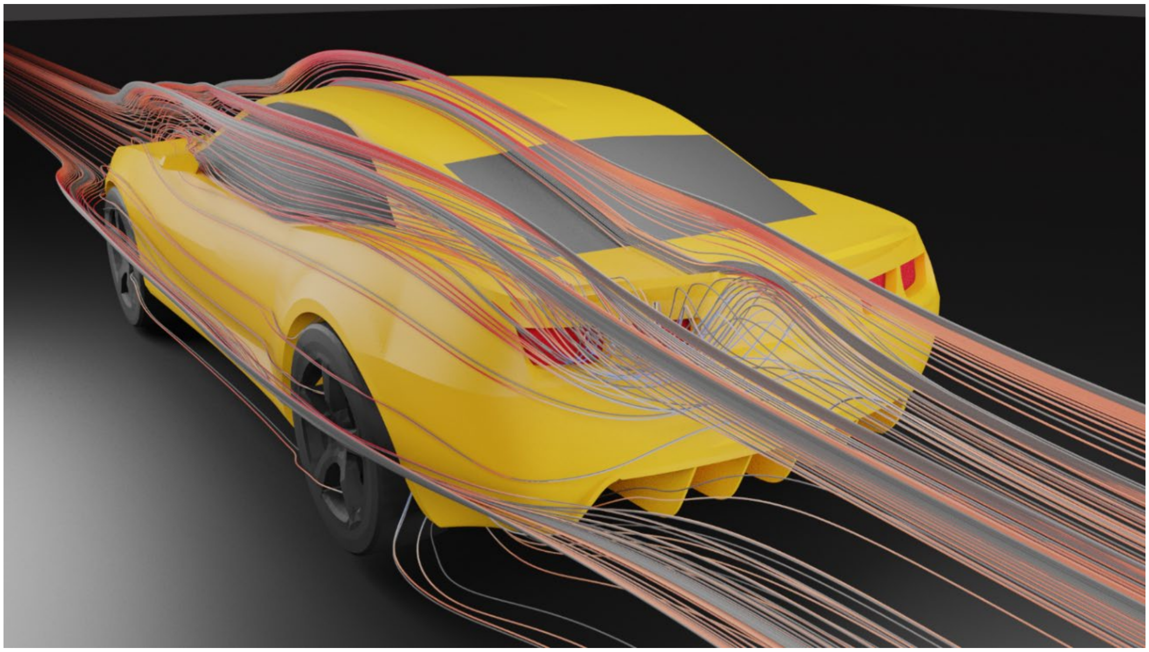


Figure 16 : Streamlines comparison between original and optimized designs

Conclusion

In this work, the C_d of a sports car was improved using morphing capabilities of DEP MeshWorks©, coupled with the optimization software LS-OPT and the ICFD solver of LS-DYNA. The drag coefficient C_d directly impacts fuel consumption, thus gas emissions and performance. The objective was to minimize its value for highway speeds. The improvement obtained with this study went from 0.660 to 0.648 corresponding to about 2% improvement which is significant in an industry having to adapt to ever tightening constraints. This study remains a demonstrator giving only an insight of the methodologies the OEMs are using nowadays. In real life, models may be more detailed, meshes more refined, and multi-physic complexity of a car to be taken into consideration. The high number of parameters taken into account, coupled to the relativity short calculation times allow engineers to better react to potentially late and urgent design changes and to offer crucial information as to which geometry change would have the highest impact on the aerodynamic profile of their model. As demonstrated by this study, the high versatility of the tools offered by the LS-DYNA suite, coupled to the powerful meshing capabilities of DEP-MeshWorks© make it a strong proposition for similar external aerodynamics applications, which DynaS+ will pursue alongside with their partners at DEP and ANSYS-LST.

References

[1] Mohd Nizam Sudin, “Review of Research on Vehicles Aerodynamic Drag Reduction Methods”, International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS Vol:14 No:02, 2014.

[2] Krishnani, P. N. (2009), “CFD study of drag reduction of a generic sport utility vehicle”, Master’s Thesis, Mechanical Engineering Department, California State University, Sacramento.

[3] C. Goubel, J.Y. Gouiffes, “Vehicule Restraint System optimization and robustness assessment using the coupling between LS-DYNA, LS-OPT and DEP MeshWorks software”, 12th European LS-DYNA Conference 2019, Koblenz, Germany

[4] M. Le Garrec, M. Seulin, “Airdrop Sequence Simulation using LS-DYNA ICFD Solver and FSI”, 15th International LS-DYNA Users Conference, 2018

[5] LS-DYNA Manual. http://www.lstc.com/download/manuals.