5th International Conference on Experiments/Process/System Modeling/Simulation/Optimization
5th IC-EpsMsO
Athens, 3-6 July, 2013

SHAPE OPTIMIZATION OF A HIGH PERFORMACE VEHICLE FOR DRAG COEFFICIENT REDUCTION

Konstantinos Tsikouris[1], Benjamin Leroy[1], Jorge Gines[1], Kasper Damkjaer[2]
[1]Icon Technology & Process Consulting Ltd, UK
Berkshire House, Thames Side, Windsor, Berkshire, SL4 1QN, United Kingdom
e-mail: k.tsikouris@iconCFD.com, web page: http://www.iconCFD.com

[2]Koenigsegg Automotive AB, Sweden

Keywords: Adjoint, shape optimization, CFD, open source, high performance vehicle, drag forces.

Abstract. *The study presented in this paper was carried out by Icon Technology & Process Consulting Ltd and its aim was to assist in the scope of optimizing the shape of a high performance vehicle in the final stages of the design. The target of the optimization process was to achieve a reduction of the drag forces acting on the surfaces of the car.*

A detailed aerodynamic simulation study of the car was performed with the use of CFD calculations, followed by the application of shape optimization methods. The surface sensitivity of the exterior of the car was calculated relative to a cost function of drag coefficient reduction. The shape of the external surfaces of the upper body was then morphed based on the aforementioned surface sensitivity values and a maximum moving coefficient.

The study was performed on the CAD mockup of the high performance vehicle model provided by Koenigsegg. The simulation study was conducted with the use of the Icon Adjoint Technology, which is part of the open source based CFD process iconCFD. The results in this paper are presented with the kind permission of Koenigsegg.

1 IMPLICATIONS

Increasing consideration of shape optimization of vehicles and the implications on their performance and fuel consumption has made computational methods and tools increasingly important. Numerical investigation of several geometries of vehicles and comparison with actual testing in wind tunnels has shown that shape optimization, even in the final stages of the design can be achieved and can contribute to the reduction of drag forces acting on the vehicle and, consequently, to its performance and environmental implications.

2 INTRODUCTION

The optimization of shape of vehicles has always been an important consideration in the automotive industry, with increasing demand for improvement of performance and efficiency. Especially nowadays, with awareness of the impact of the vehicle performance on the fuel consumption and, consequently, emissions and overall environmental implications being exponentially increased, the optimisation of vehicle performance has a great importance for the designers and manufacturers.

This paper presents an overview of the adjoint optimization methodology with the use of Computational Fluid Dynamics (CFD) and furthermore focuses on an example of the shape optimization of a high performance vehicle. The ultimate objective of this study is to investigate the feasibility of reducing the drag forces acting on the surfaces of the car by means of adjoint optimization of its external surfaces, as well as introducing the Adjoint iconCFD Optimize solver available in the open source based iconCFD service.

The first part of the paper describes the strategy followed in adjoint optimization in general, as well as more specifically with iconCFD Optimize. The stages of the optimization are presented in order to provide an understanding of the capabilities and demands of the adjoint optimization.

The second part of the paper illustrates the adjoint optimization of a high performance vehicle, the geometry of which was provided by Koenigsegg and was treated with the use of iconCFD Optimize. The stages of the optimization include the geometry and meshing setup and creation, the computational setup, the primal external aerodynamics simulation, the calculation of the surface sensitivities with the use of the adjoint solver, the morphing of the external surfaces based on their sensitivity values and, finally, the re-meshing of the domain based on the optimized geometry, the confirmation run and comparison of the results to the initial geometry ones.

The CFD methodology, solution domain, model set-up, solving and post-processing, as well as some comparison to experimental data of similar studies carried out with the iconCFD Optimize procedure is illustrated in this paper.

3 ADJOINT OPTIMIZATION OVERVIEW

3.1 Why Adjoint Optimization

Adjoint optimization is a fast, efficient gradient based optimization. Unlike traditional methods that require hundreds of evaluations, an optimized result can be achieved with the use of the adjoint method in a matter of a few CFD calculations iterations.

Adjoint optimization is also independent of the number of parameters giving flexibility in the use of deformation tools and can provide a choice between Shape and Topology optimization. In principle Shape Optimization calculates adjoint sensitivities according to one or more given cost functions, morphs the shape according to those sensitivities and is recommended for external aerodynamics. Topology Optimization calculates and provides the optimum flow path without performing any change on the computational domain but instead blocking the surrounding cells. This method is mainly recommended for internal flows. In both methods the modified - optimized geometry can be exported at the end of the optimisation loop for use as feedback in the design process.

3.2 Choice of software packages

In this study the iconCFD [1] service was used. The CFD solver and meshing tools, as well as the adjoint CFD Optimize deployed in iconCFD were utilised. iconCFD is based upon the OpenFOAM® toolbox [2]. Both are based on the finite volume method of discretisation of Navier-Stokes equations. The physical models, turbulence model, solver settings, meshing and solution strategy were selected to predict the physical phenomena encountered in this type of external aerodynamic applications.

Results of previous similar studies were shown to be comparable with reasonable accuracy to Wind Tunnel testing data.

The iconCFD Optimize Technology is currently being used in the automotive industry to perform final shape design of upper body styling surfaces and definition of under body panels and flow guidance structures to numerous vehicles ranging from Pickup trucks to SUVs and high performance cars. iconCFD Optimize is being used by Formula 1 to optimize wing designs and aero package. However it is currently being used in non-automotive applications e.g. optimization of filters, nozzles, ducting systems.

3.3 Methodology

3.3.1 Primal Run

- Frozen adjoint turbulence
- Efficient, high accuracy, robust simulation methodology for usage in production
- Robust and fast meshing of complex geometries
- Presented at SAE World Congress 2009 (SAE 2009-01-0333)
- DES validated with wind tunnel data, and compared to proprietary CFD codes
- Presented at SAE World Congress 2011(SAE 2011-01-0163)
- +85% of results are within 4% error (Absolute drag value prediction) (**Figure 6**)

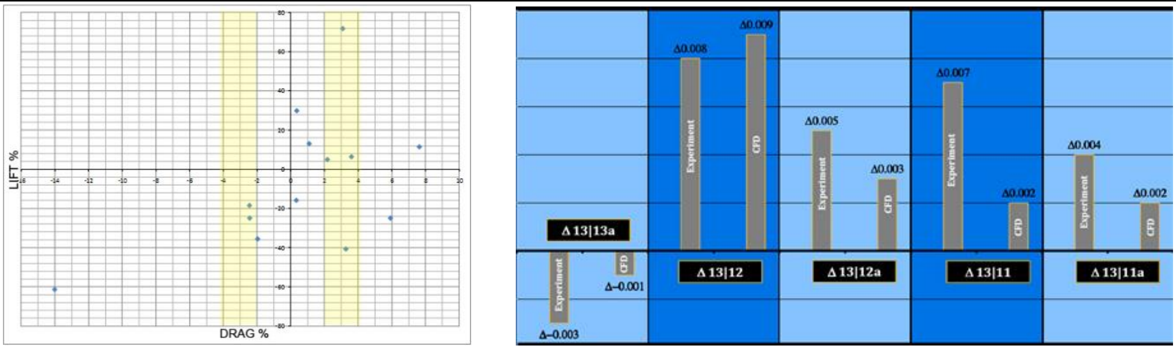


Figure 6 Drag & lift CFD value prediction compared to experiment

3.3.2 Adjoint Run - Surface Sensitivities calculation

- Frozen adjoint turbulence
- Output: sensitivity with respect to drag reduction
- Selection of faces to be optimized by mesh deformation
- Push & Pull indication is given by the surface sensitivities output

3.3.3 Morphing:

- Direct deformation tool - Fully parallelized
- Smoothing and limiting deformation picks
- User selection of moving patches
- Transition between moving and non-moving patches
- Surface and / or volume morphing

3.3.4 Exporting Morphed Geometry:

- Exporting the morphed geometry in STL format
- Various options

3.3.5 Confirmation Run:

- Successfully applied to 12 car projects
- Range of improvement [-0.015,-0.003]
- Maximum deformation [0.01,0.015]

4 CASE DESCRIPTION

4.1 Computational Geometry & Mesh

The detailed geometry of a high performance car (**Figure 7**) was provided by Koenigsegg. The underhood and all internal geometry had been removed. The geometry was watertight without any unnecessary details and features and was therefore used in order to produce the computational mesh. The geometrical characteristics of the wind tunnel were based on a best practice put together by previous similar studies.

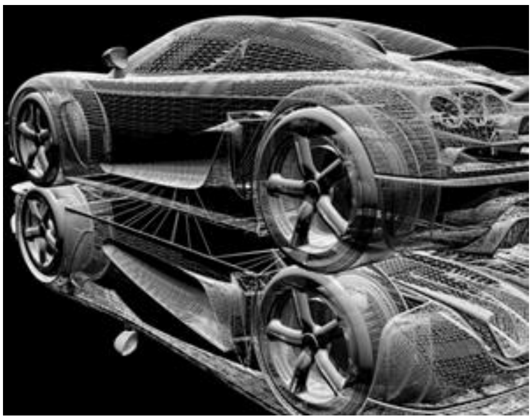


Figure 7 Geometry of the mockup model

The computational mesh was created by ICON with the iconCFD built-in meshing tool, foamProMesh. The mesh is a trimmed unstructured hex-dominant one with a minimum cell size of 5.8mm and 6 surface layers, returning a total mesh size of approximately 18.2 million cells. This resolution is acceptable for the nature of the problem under investigation but can also be increased, should greater detail and even more accurate results be requested. It is obviously very important to capture all details on the simulation but equally necessary to be able to deliver results on-time and have an acceptable turnaround.

In order to have enough accuracy near the coolers, as well as an increasing refinement in the vicinity of the vehicle compared to remote locations of the domain, refinement zones have been included. These are boxes that lie around the desired areas and define a local refinement region. (**Figure 8**)

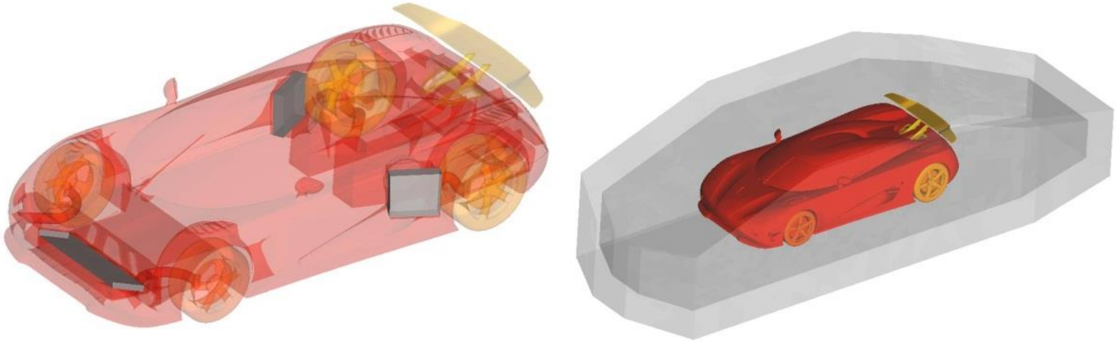


Figure 8 Refinement Zones

4.2 Computational Setup

The primal run was executed with the RANS solver adjointSimpleFoam. The Spalart-Allmaras turbulence model was used on a total number of 72 processors.

The surface sensitivity run was executed again with the RANS solver adjointSimpleFoam. However this time the turbulent field was frozen to the last state of the primal run and the laminar turbulence model was used on the same 72 processors.

The boundary conditions of the wind tunnel were also based on experience from previous similar studies.

The velocity at the tunnel inlet was set to 40m/s, whereas the outlet was set to a fixed pressure value of 0Pa. For this simulation rotating velocity boundary conditions have been applied to the four wheels, whereas the ground has been assigned moving ground boundary conditions.

Parts of the geometry have been treated separately in the calculation in order to be able to exclude some of them from the optimization procedure. The disc brakes and wing pillars, as well as the wheels have not been optimized.

5 THE PRIMAL RUN (EXTERNAL AERODYNAMICS RUN)

5.1 Primal Run Results

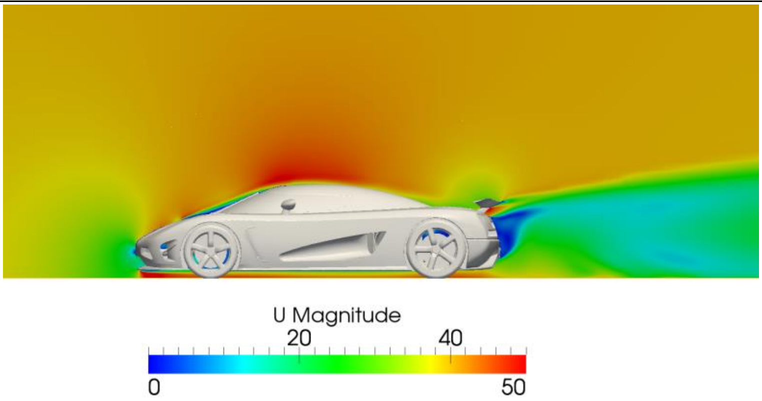
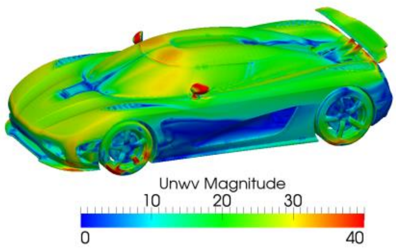


Figure 9 Velocity Magnitude, section on Y=0



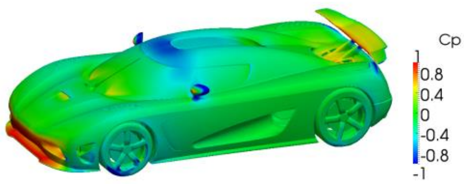


Figure 10 Near Wall Velocity Magnitude Unwv

Figure 11 Pressure coefficient Cp

6 OPTIMIZATION ADJOINT RUN

6.1 Surface Sensitivities Calculation

Figure 12 Surface Sensitivity Gw

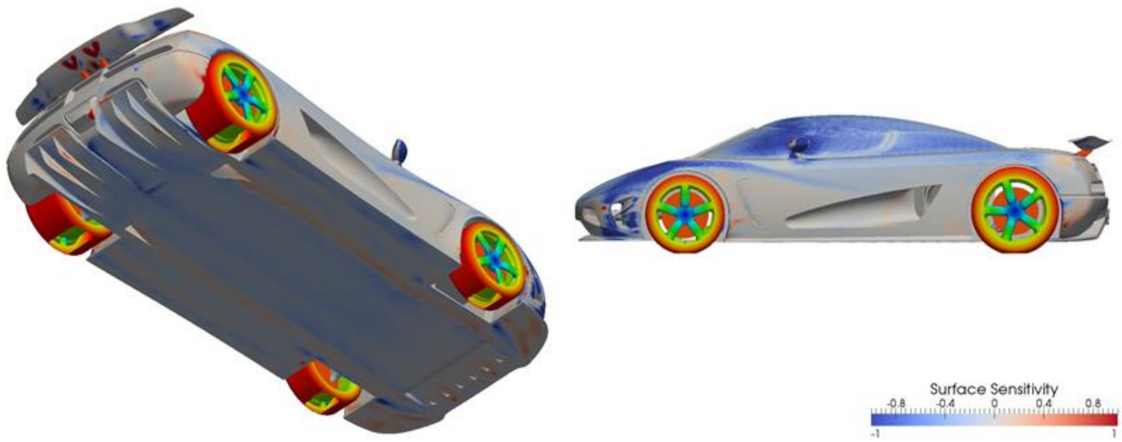


Figure 13 Surface Sensitivity Gw

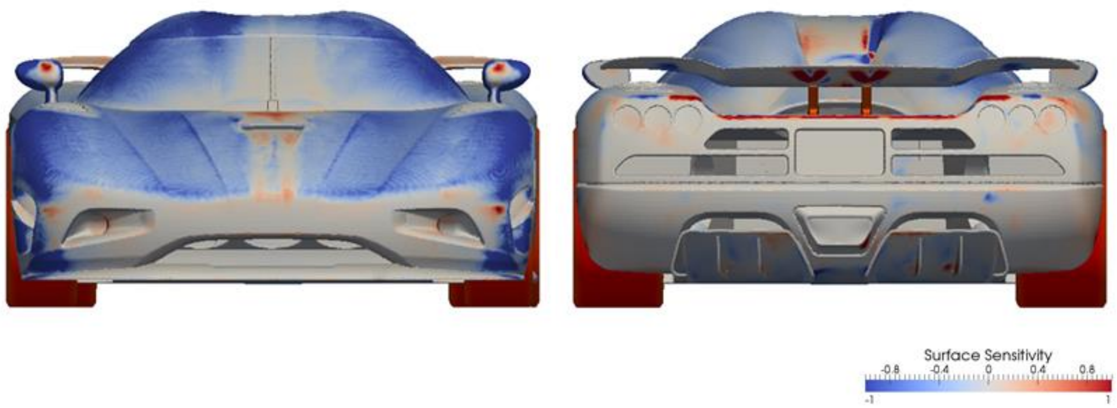


Figure 14 Surface Sensitivity Gw

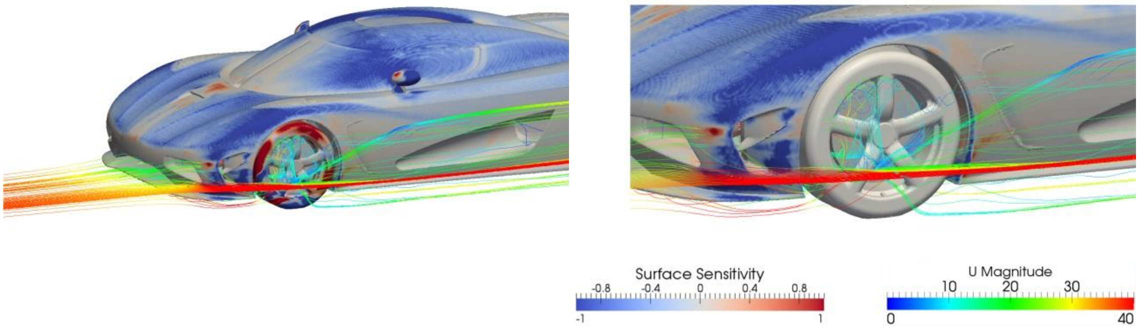


Figure 15 Velocity-coloured Streamlines superimposed on Surface Sensitivity Gw

6.2 Morphing

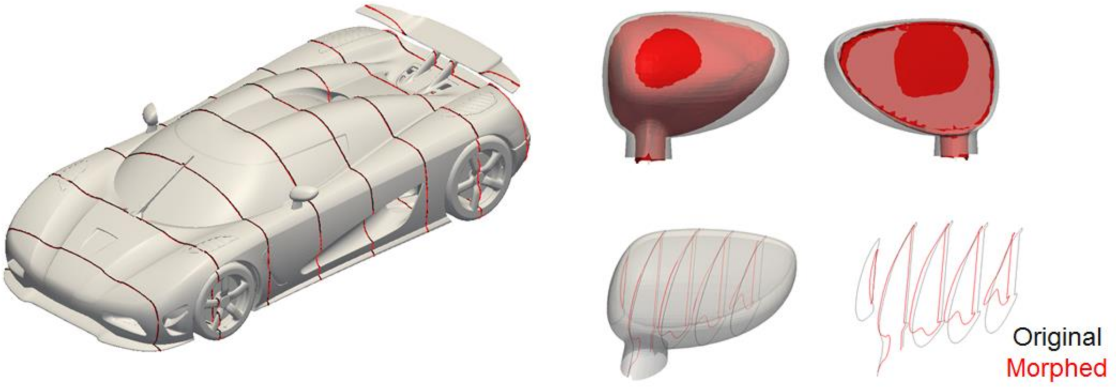


Figure 16 Comparison of original to morphed geometry

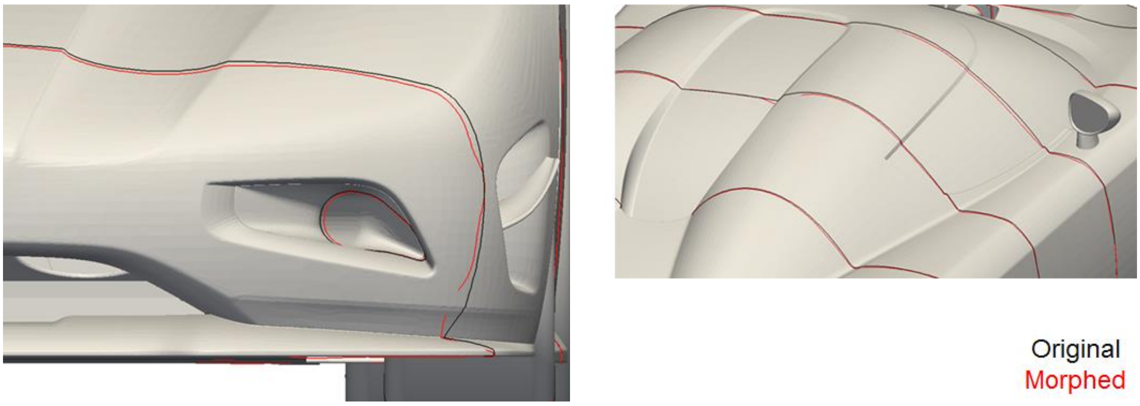


Figure 17 Comparison of original to morphed geometry

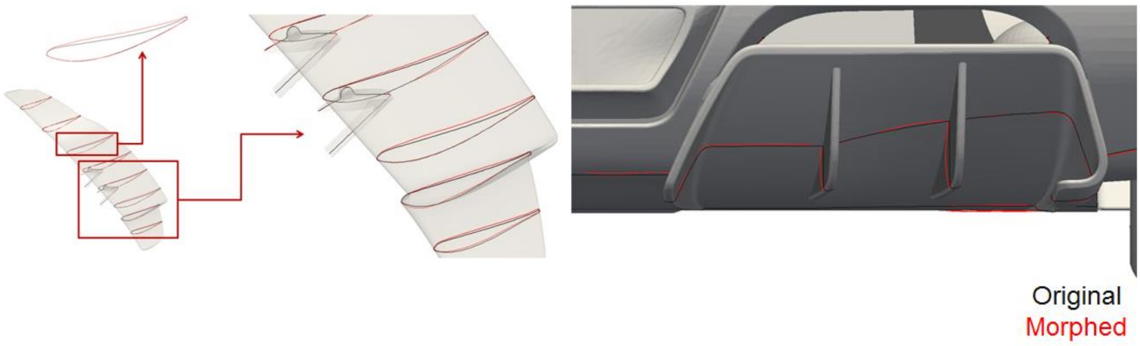


Figure 18 Comparison of original to morphed geometry

6.3 Re-meshing and Confirmation Run

6.4 Results and Discussion

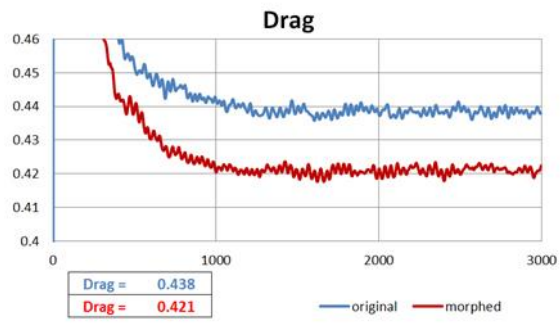


Figure 19 Drag (original & morphed geometry)

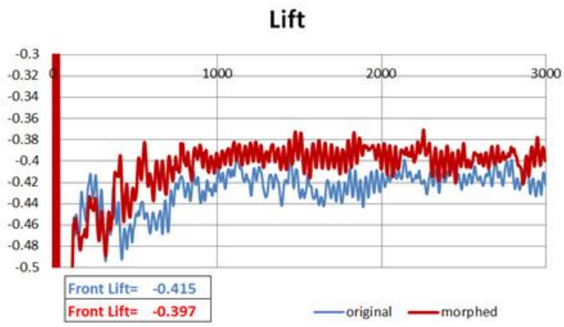
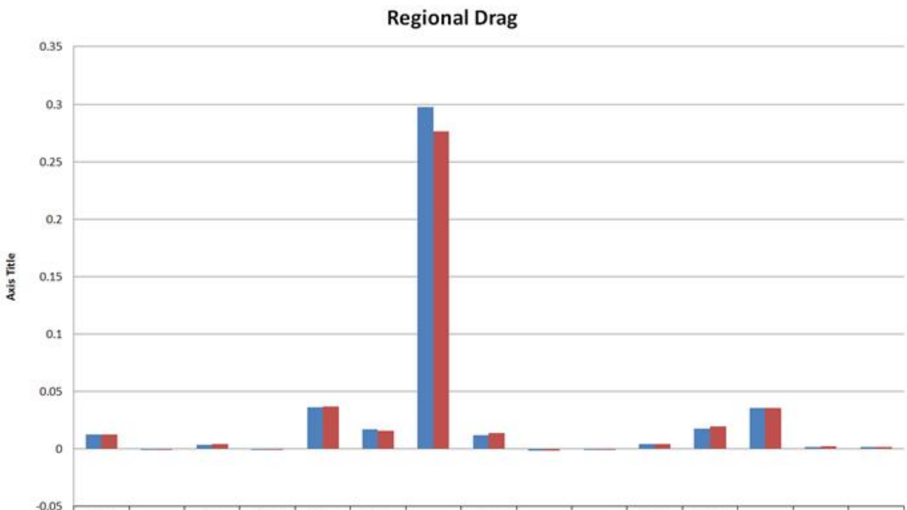


Figure 20 Lift (original & morphed geometry)



	Rear wheel Left	Rear brake left	Rear wing foot left	Front brake right	Rear wing blade	Front wheel right	Agera body	Rear wheel right	Front brake left	Rear brake left	Rear wing foot right	Front wheel left	Cooler front	Cooler rear left	Cooler rear right
Original	0.01283	-0.0003	0.003664	-0.00026	0.036339	0.016748	0.297564	0.012203	-0.00174	-0.00027	0.004194	0.017663	0.035813	0.002014	0.001917
Morphed	0.012664	-0.00045	0.003966	-5E-05	0.037158	0.015719	0.27613	0.013982	-0.00167	-0.00018	0.00409	0.019817	0.035873	0.002351	0.001662

Figure 21 Regional drag contribution (Original & Morphed geometry)

5 CONCLUSIONS

6 ACKNOWLEDGEMENTS

Authors wish to thank Koenigsegg Automotive AB for their kind permission on making public the results of this study.

7 REFERENCES

[1] Icon, 2013, http://www.iconcfd.com/es/services/iconCFD

[2] OpenFOAM® and OpenCFD® are registered trademarks of Silicon Graphics International Corp. The work described in this paper are not approved or endorsed by OpenCFD Limited."