AERODYNAMIC OPTIMIZATION OF A FORMULA STUDENT CAR

[1]Argyrios Apostolidis*, [2]Athanasios Mattas, [3]Aggelos Gaitanis, [4]Nikolaos Christodoulou
[1]Aristotle Racing Team, Greece, [4]BETA CAE Systems S.A., Greece

KEYWORDS – CFD pre-processing, meshing, morphing, optimization, Formula Student

ABSTRACT – Aristotle Racing Team has a ten-year history in Formula Student competitions, while in the last two years the team decided to evolve a complete aerodynamic package to vastly improve on-track performance.

The aerodynamic development of a Formula Student car has been proved to be a time-consuming process, which involves the design of numerous aerodynamic components using CAD software, geometry clean-up for mesh generation, model set-up, running of CFD simulations and finally result assessment. The efficiency of this loop defines the number of optimization increments, and hence the performance of the aerodynamic package. The use of ANSA meshing capabilities played a significant role in the minimization of the human time and the automation of the process, while preserving the quality of the results by the extensive use of the morphing tool.

In this paper three main studies were conducted using ANSA's morphing tool. The determination of the basic characteristics of the diffuser (length, angle and height), and of the relative position of the elements of both front and rear wings (slot gap and angle of attack), in conjunction with generation of an aerodynamic map for different front and rear ride heights, a vital datum for a detailed vehicle dynamics analysis.

In addition, the batch mesh, task optimization and scripting capabilities of ANSA allowed for a fully automated mesh generation and simulation of multiple design variants without any human interference, decreasing the time to a minimum.

1. INTRODUCTION

Formula SAE competitions challenges international students to design, build and race a formula style car. Aristotle Racing Team throughout, its ten-year presence in the European races, has been successful in both static and dynamic events. In the last two years, in order to remain on the forefront of technical development, the team opted to introduce a complete aerodynamic package for the first time in its history.

The success of a Formula Student race car heavily relies on the testing period. Maximizing on track testing has great impact on improving its reliability, performing experiments and finding the optimum set-up parameters for the car. This implies that the design and optimization of the car should be as efficient as possible to meet the tight timeframes without compromising the performance.

The aerodynamic development of last year's race car is based on Computational Fluid Dynamics, due to its lower cost and easier accessibility than wind tunnel or track experiments. CFD tools provide a complete picture of the flow around an object, however the accuracy of the simulation depends on various parameters such as, the detail of the model, hardware and software. Minimizing the CAE modelling time can be achieved by the implementation of ANSA tools like "Batch Mesh", "Morphing Tool" and "Task Optimization", while at the same time maintaining the high quality of the model. The present work presents the use of ANSA tools on optimization of diffuser's shape, wing's position and finally on the production of an aerodynamic map for different ride height set ups.

2. MODEL SET-UP

The CFD pre-processing method that has been used for the purpose of the paper, starts with the translation of the geometry from a CAD software to ANSA, which ensures the optimum quality input. The model then must be prepared for meshing by correcting errors which occur during the translation such as intersections or open holes.

2.1. Batch Mesh

The batch mesh tool is then used for the generation of surface mesh, layers and volume mesh. With the use of batch mesh, it is easy to automate the process of meshing by specifying the meshing characteristics for each part of the car (minimum and maximum size of elements, number of layers etc.) and then reuse it for different design variants of a part, saving up to 50% of human time. Moreover, the batch tool along with the scripting capabilities of ANSA can produce high quality meshes in shorter times, without opening ANSA's graphical interface but instead running it in parallel on University's Linux cluster.

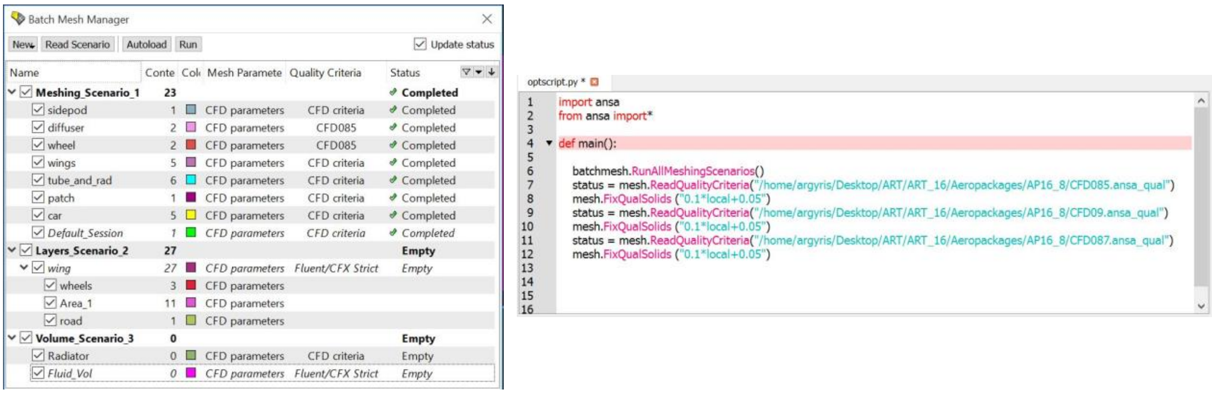

Figure 1: Batch Mesh Tool and scripting for parallel simulation on a cluster

2.2. Cutting Plane

For the analysis of the rear wing and the diffuser an approach for extensive mesh reduction has been developed. After a full car CFD simulation a transversal plane before the helmet of the drive is being generated, where the three components of the velocity and the turbulent viscosity are obtained. Then, the same plane is being introduced into the mesh, as shown in Figure 2, that cuts the geometry reducing the size of the domain up to 50%.

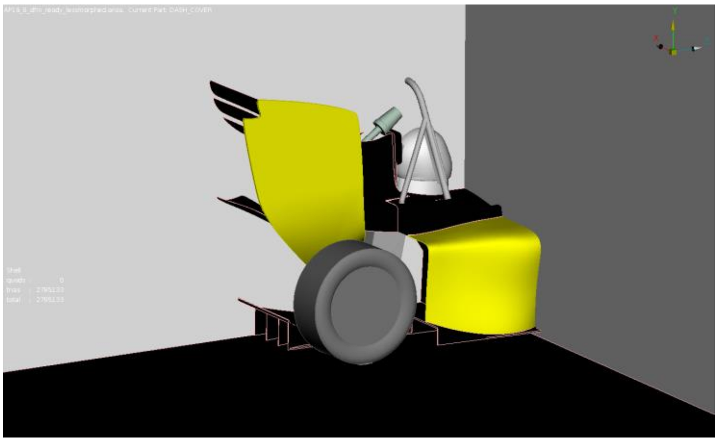

Figure 2: Reduced model for Rear Wing with cutting plane

After the generation of the reduced mesh the previously obtained data are imported as inlet boundary conditions on the cutting plane.
The selection of the position of the cutting plane is of great importance. The rear wing influences the front of the car and by changing its design the flow characteristics in front of it will change also. Therefore, if the cutting plane is too close to the rear wing, the inlet boundary conditions will impose a wrong flow direction that will deface the CFD simulation.

3. MORPHING TOOL AND OPTIMIZATION

3.1. Diffuser

The diffuser is the most efficient aerodynamic device on the car, that can generate significant amount of downforce with minimum drag penalty, while in the same time the produced upwash, affects the performance of the rear wing and vice versa. However, its operation is heavily compromised from the presence of both front and rear wheels, thus a robust design is a key target.

The most efficient way to define the basic geometry of the diffuser is with the assistance of Morphing Tool, that allows user to automatically reshape the diffuser by changing key dimensions within certain limits. The whole CAE modelling time is greatly decreased by removing the use of CAD software, geometry clean up and surface mesh creation.

To produce an efficient morphing model attention must be paid to ensure the morphing boxes correctly represent the diffuser’s geometry. The design variables used, were the diffuser’s height and length. Finally, the optimization task manager produced 30 different designs that cover the whole sampling space using the statistical method Latin Hypercube.

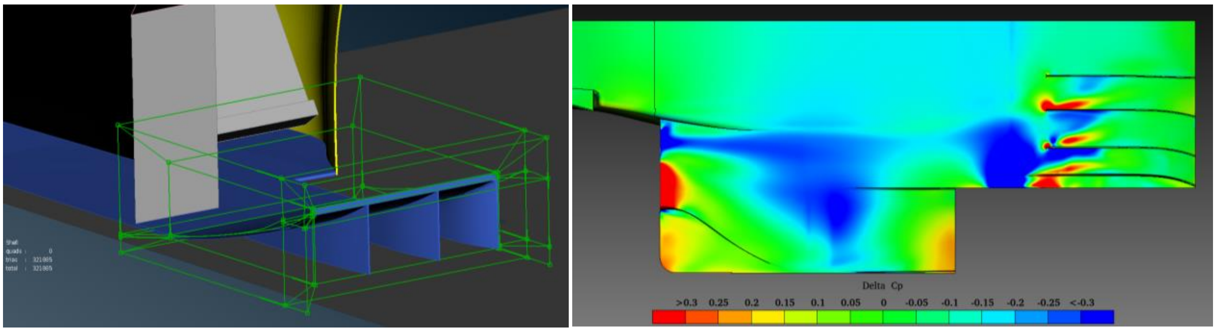

Figure 3: Diffuser’s morphing boxes and Pressure Coefficient differences between initial and final diffuser design

As seen in the above picture the pressure coefficient across the diffuser is greatly decreased after the optimization of the baseline model. This is due to the improved pressure recovery at the end of the diffuser, which affects the suction of the air at the entry. Moreover, it should be noted that in the final design the negative effect of the rear tyre wake had been minimized, resulting in greater amounts of downforce.

3.2. Wing Location

Another major area of study on the car is the relative location of the wing elements. The main plane on both front and rear wings is the main downforce producing element, however the two flaps play a significant role on the overall performance of the wing. The high velocity flow on the lower surface of the flap allows the flow to leave the main plane at higher speed. This damping effect reduces the pressure recovery of the main plane and favors more aggressive angles on it.

The gap between each element determines how well the two elements will interact. Decreasing the gap results into higher downforce levels, until the gap becomes small enough that the two boundary layers are merged into a thicker one, which increases the likelihood of flow separation on the forward element. By increasing the gap the positive effect that the trailing element has on the forward one can be lost.

Finally, the different track characteristics of each event necessitate different downforce and drag levels on the car. The easiest way to find the optimum configuration for each track is by having multiple slots for different angles of the elements.

The above described studies were carried out in the shortest amount of time with the use of DFM tool, without the need of constant CAD changes and surface mesh creation. Figure 4 depicts the effect of the gap between front wing’s elements. For small gaps the confluent boundary layer limits the downforce of the wing, and for gaps over 25 mm the flap effect is minimized.

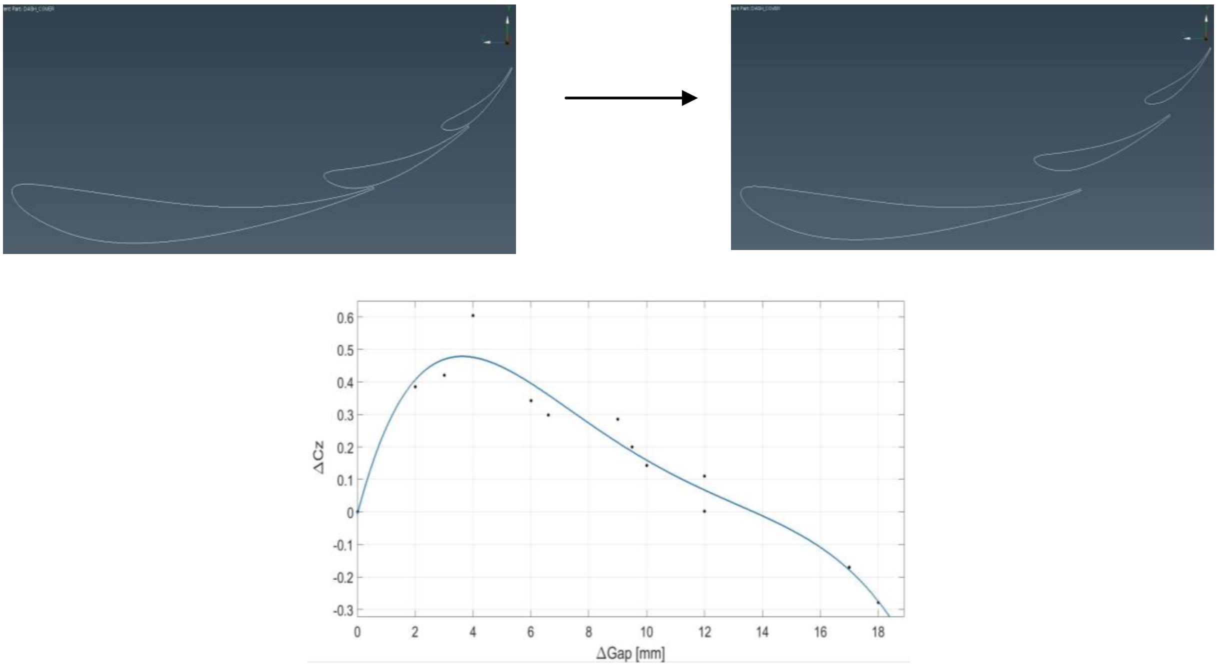

Figure 4: Downforce variations with increasing slot gap

3.3. Aerodynamic map

A vital tool for the analysis of the behaviour of the car on track is the vehicle dynamic simulation. To accurately represent the impact of aerodynamic forces, an aerodynamic map for different ride heights should be provided by the aerodynamic department. Due to various limiting factors the only way to generate such a map is by performing CFD analysis on multiple ride height configurations. Aerodynamic forces greatly impact the dynamic behaviour of the car, especially during braking or cornering. During these stages the ride height of the car, both front and rear, varies, resulting in a similar change to the aerodynamic loads, thus heavily influencing its dynamic response. A robust aerodynamic package is of great importance for improving the overall performance of the car.

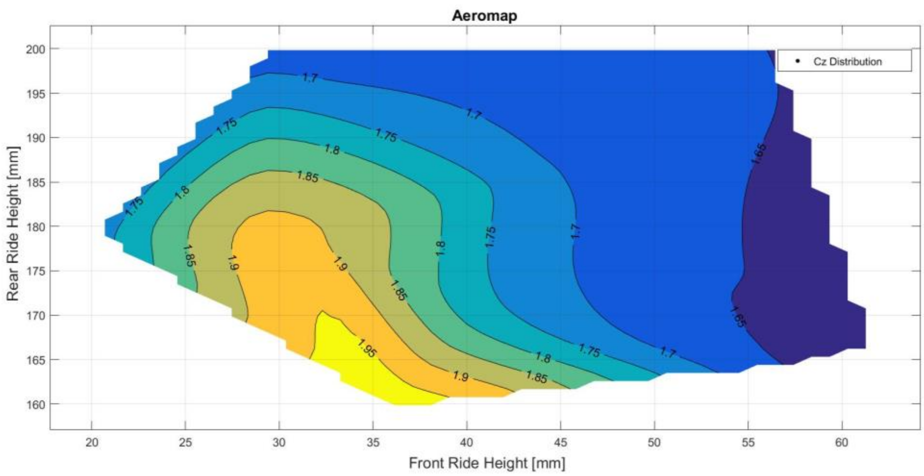

Figure 5: Downforce coefficients for different front and rear ride heights

It is obvious in Figure 5 that a lower front ride height leads to greater downforce, as a result of the strengthened ground effect on both front wing and on the diffuser. However, if the rear ride height overpasses a certain value the diffuser and the main plane of the front wing stall due to the high rake angle, resulting to an abrupt decrease in downforce. Using again the DFM tool of ANSA the simulation time reduced up to 60% for one design iteration as shown in Table 1.

	Standard Method	DFM & Batch Mesh
CAD	2h	-
Geometry Clean-up	1h	-
Morphing	-	0.5h
Surface Mesh	1h	-
Volume Mesh	2h	2h
TOTAL	6h	2.5h

Table 1: Total time for meshing per iteration

4. CONCLUSIONS

ANSA's powerful pre-processing tools allow for the aerodynamic optimization of every part on a Formula Student car with minimum human effort. The above described methodology increased the downforce of the car up to 40 % and at the same time provided a clear picture of the car's aerodynamic behaviour in different conditions, vital for the vehicle's dynamic simulation. Utilizing the Batch Mesh and Morphing Tool the total saved time goes up to 200 hours, while maintaining the quality and the accuracy of the simulations.

REFERENCES

(1) ANSA version 17.0.0 User's Guide, BETA CAE Systems, July 2015

(2) Xin Zhang, Jonathan Zerihan, "Aerodynamics of a Double-Element Wing in Ground Effect", AIAA Journal Vol. 41, No. 6, June 2003

(3) Xin Zhang, Jacques Heyder-Bruckner, "Detached-Eddy Simulation of a Double-Element Wing in Ground Effect", 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition 4 - 7 January 2010, Orlando, Florida

(4) John Axerio, Gianluca Iaccarino, Emin Issakhanian, Kin Lo, Chris Elkins, John Eaton "Computational and Experimental Investigation of the Flow Structure and Vortex Dynamics in the Wake of a Formula 1 Tire", Paper Number 09B-0082