REVISTA INTERDISCIPLINAR DE PESQUISA EM ENGENHARIA

CILAMCE 2016

XXXVII IBERIAN LATIN AMERICAN CONGRESS
ON COMPUTATIONAL METHODS IN ENGINEERING
BRASÍLIA - DF - BRAZIL

DEVELOPMENT OF A NUMERICAL METHODOLOGY FOR THE ANALYSIS OF AERODYNAMICS SURFACES

Rangel Silva Maia

Dr. Manuel Nascimento Dias Barcelos Junior

rangel.maia@gmail.com

manuelbarcelos@aerospace.unb.br

University of Brasilia

Special Industry Area, 72.444-240, Brasilia, DF, Brazil

Dr. Henrique Gomes de Moura

hgmoura@yahoo.com

University of Brasilia

Special Industry Area, 72.444-240, Brasilia, DF, Brazil

Abstract. *The cars aerodynamics study aims to evaluate the distribution of pressure and shear stress through the passage of the fluid on its surface. With a laminar flow in this passage, the loss of performance by the drag is minimized and hence there is a lower fuel consumption. Good performance in high-speed corners is due to the aerodynamic development, since it aims to keep the vehicle in contact with the ground all the track. The main factor that contributes to a better tyre grip is the force that the vehicle imposes on the ground. One way for maximizing this force without increasing its mass, is the use of aerodynamic forces seeking maximum lift along with the lower drag force. The use of airfoils is therefore a challenge to find a balance between them. Due to high project costs for manufacturing and testing, carrying out the development in computer programs is an economical way to improve the car's aerodynamics. This paper aims to develop a methodology for computer simulation of aerodynamic surfaces. It is important to mention this is a complementary work to the use of wind tunnels, not replacing, but providing an economical alternative with satisfied numerical precision in the design of aerodynamic surfaces.*

Keywords: *CFD, Airfoils, StarCCM+, Formula Student (SAE), Aerodynamics*

1 INTRODUCTION

The cars aerodynamic have two primary concerns: the increase of negative lift (known as downforce) and decrease of drag. These aerodynamic forces are generated by the pressure and shear stress distributions on the surface of the body.

The downforce is the responsible for the increment of vertical force that will be added to the weight. This addition helps to push the tyre against the track, and thus increasing the contact patch and the cornering ability. On the other hand, the drag acts in the opposite direction of the body by imposing a resistance to its movement.

The downforce is in the opposite direction at the wing (airfoil) of a car compared with aircrafts, whose force acts in the opposite direction to the weight force. The difference in air flow velocity passing over the leading edge and trailing edge of the wing creates a pressure difference. It ”pushes” the wing toward the low pressure zone, causing a lift force.

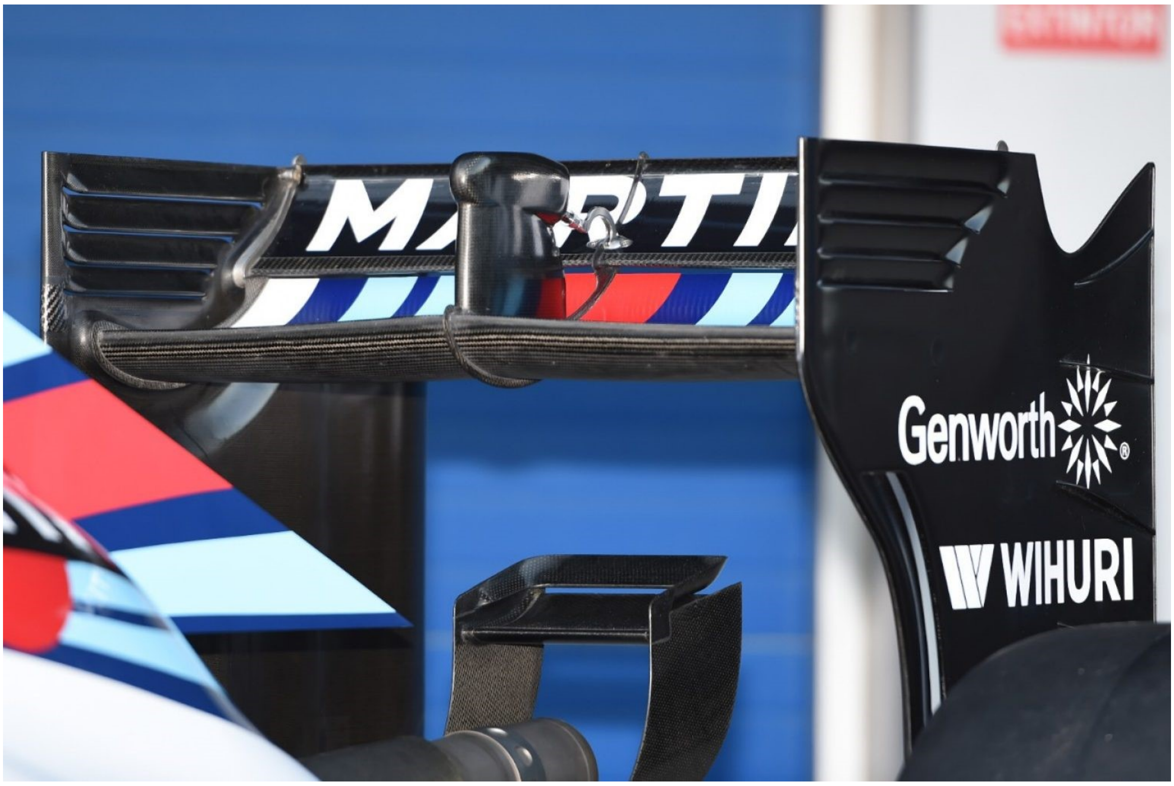


Figure 1: rear wing of an F1 car, (FORMULA 1, 2015)

The Formula 1 teams bet on aerodynamics changes to make the cars faster and faster, and hence summing crucial gains in time of each lap. There are two ways of aerodynamic research: wind tunnel experiments and fluid dynamics programs. The first one has a high cost involved in both the expenditure of materials for construction of scale models and the energy used to generate the experiments; or at room construction with large turbines, high-precision scales, etc.

The Computational Fluid Dynamic (CFD) is a numerical methodology that uses the fluid dynamics equations to allow engineers predict the performance of a project when exposed to both internal or external flow. CFD programs can simulate problems involving gases and liquids, or solids whether CSD (Computational structures dynamics) is available for the fluid-structure interaction.

The CFD programs emulate virtual environments of wind tunnels with the aid of computer simulation, and thus allowing engineers to analyze the aerodynamic efficiency of models before they are manufactured in order to reduce expenses with prototypes and time in project development.

The most obvious and important aerodynamic devices in a car are the front and rear airfoils. They have different profiles depending on the downforce required in a particular track. Slow tracks, with many curves, require wing profiles that maximize lift while high speed tracks, with long straights, require a drag decrease with reduced size of airfoils.

A range of researches has been performed on the aerodynamic characteristics of race cars. As a result of the competitive nature of motorsport, these projects are usually not published before they become outdated (KIEFFER; MOUJAES; ARMBYA,).

Despite the computer simulation provides similar results to those obtained in real tests, it is still needed to carry out experiments in wind tunnel. The evaluation of the cars aerodynamics in wind tunnel is applied as a validation of the results obtained in CFD platforms. This allows comparison between the forces and pressures acting on the vehicle on both tests to ensure that the CFD analysis are similar to the ”real world.”

In motorsport it is important to use a wing profile that meets the needs of downforce profits and drag reduction. With the increase of drag, it is apparent the decrease in vehicle performance and fuel consumption enhancement.

The main component of the lift comes from the pressure that acts on the body surface. It is originated from the difference of velocities in the upper and lower camber.

On the other hand, the drag force comes from the frictional drag due to the fluid viscosity (on the boundary layer next to the body) and also from the normal pressure distribution. The fluid viscosity causes a friction in the boundary layer. So, this friction generates a shear stress which contributes to the drag. With the turbulent boundary layer, the flow is retarded and thus drag is further increased.

Unlike laminar flows, fluid viscosity is not only a function of temperature in turbulent flows, and if it does not make a direct simulation of the equations that describe the movement of the fluid (do not solve numerically the equations that describe the fluid movement in an extremely refined mesh) the viscosity needs to be adjusted. This correction is made by an equation that describes the behavior of properties associated with turbulent flow. It complements the modeling and solves numerically the problem in more feasible meshes from the computational cost point of view.

The Reynolds Number, a dimensionless number that can determine whether a flow is more laminar or turbulent, is applied to the similarity condition (different fluids under the same boundary conditions and initial conditions have forces with constant ratio , with geometrically similar bodies (KATZ, 1995). This condition is the ratio of the inertial forces over the viscous forces:

$$Re = \frac{Inertial forces}{Viscous forces} = \frac{\rho u \frac{\partial u}{\partial x}}{\mu \frac{\partial_2 u}{\partial x_2}} = \frac{\rho u d}{\mu} = constant \quad (1)$$

Where d is the characteristic dimension of the body, u is the velocity and ρ is the fluid density. According to (KATZ, 1995), for airfoil chords with higher Reynolds than 10^5 the flow is turbulent. In formula SAE cars, the Reynolds number varies between 200,000 and 600,000 according to (PAKKAM, 2011).

The interest in racecar projects is that the airfoils work in the linear region of the lift curve, near the stall condition, and thus on the maximum lift. In projects that aims to reduce drag, it is important there is no detachment of the boundary layer by gradient of adverse pressure. The study of the air flow over vehicles mind that this detachment is as laminar as possible in order to reduce turbulence.

One way to study the properties of the turbulent flow in CFD is using turbulence models for the closure of the equations that describe the fluid movement. Without these models, the resolution of the governing equations for turbulent flows would only be solved with extremely refined mesh.

The development of numerical methods for aerodynamic surfaces analysis seeks to fill the lack of a wind tunnel, however, instead of rule out the experiment in this environment, it provides a reliable study to select, evaluate and improve airfoils in a more affordable way. On the same hand, in the 2010 season, Virgin Racing team tried to develop a car completely through a CFD program without the use of wind tunnel. They continued not being competitive as other teams were.

CFD simulations may show fluid behavior, aerodynamics improvement, however, they cannot provide exact values of the forces, but approximations. Wind tunnels provide the actual results of the forces with a vehicle half model (as in Formula 1) and assess the actual need for changes in the model.

As racecars require a greater concern with the downforce, the wing design is made to speeds where negative lift is most required than drag decrease. One type of chord profile widely used in researches with low Reynolds number and high lift is the ”s1223” developed by (SELIG; GUGLIELMO., 1997), which will be used in this work.

In the second part of this work, profiles of Wortmann type (FX72-150a; FX63-137 and FX74 cl5-140), Selig (s1223), Eppler (E423) and Liebeck (LA203a and LNV109a) will be compared using the methodology established in the first part.

2 OBJECTIVES

The aim of this work is to develop a numerical methodology of wing profiles for motorsport, especially for the Formula SAE electric car of University of Brasilia. The methodology will be assessed from the computational cost point of view and quality of results. The specific objectives include:

1. Creation of the mesh and analysis of its refinement and boundary layer in 2D.

2. Simulation and analysis of turbulence models at different angles of attack in order to observe its behavior within the linear region (the region in the lift graphics where the lift increases with the increasing of the angle of attack and before the stall on the surface).

3. Study of the boundary layer.

4. Simulation of a reduced scale model (quasi-2D) in order to validate the three-dimensional one.

5. Profile analysis in real scale according to the parameters of (SELIG; GUGLIELMO., 1997; WILLIAMSON et al., 2012).

3 METHODOLOGY

In this paper is developed a numerical methodology for the airfoil design of a Formula SAE team. The simulations are done through StarCCM+, a commercial package of CFD program which uses the finite volume method for simulations with turbulent flows. This method divides the domain in a finite number of small volumes (CD-ADAPCO, 2015).

Three steps are considered for this methodology:

The first one is the creation of the mesh (convergence study) in which the degree of refinement and the boundary layer are studied by evaluating the computational cost and quality of results. The numerical methodology is validated by comparison between experimental results available in the literature.

In the second stage, studies compare the turbulence models available in the CFD program for the approximation of Reynolds Average Navier Stokes (RANS) model (a turbulence model to provide the convergence for the governing equations). This turbulence model choice is based on the computational cost aligned to the quality of results.

Finally, the airfoil is discussed in full scale under different angles of attack for the stall study on the surface. The studies are done in 2D, quasi-2D and 3D simulations.

Two computational equipments are used for this work:

Table 1: Computational settings of the equipments.

	Equipment 1	**Equipment 2**
Processor	Intel ® Core ^{TM} i7-4500U CPU @1.8GHz 2.4GHz	Intel ® Xeon ® CPU E5620 @2.4 GHz 2.4 GHz (2 processor
Memory (RAM)	12 GB DDR3	26 GB
System	Windows 8 64 Bits	Windows 7 64 Bits.
Graphics cards	NVIDIA ® GeForce ® GT 720M com 2GB VRAM dedicada	NVIDIA GeForce GTX 295

The meshes 1 and 2 were generated by the Equipment 1 due to lower computational cost, while for the mesh 3, the Equipment 2 was used. As an estimate of the computational cost, Equipment 2 generated mesh 1 for three-dimensional full-scale model (with 850 mm thick) within 2000 seconds, getting 9,400,000 elements. The Table 3 shows the number of elements for each mesh.

The points of the wing geometry used were obtained from the (AIRFOILTOOLS, 2015). This tool imports the values of the profile geometry defined by (SELIG; GUGLIELMO., 1997) which is available in the virtual address of the University of Illinois, USA. The profile Selig s1223 is also tested in a wind tunnel by (WILLIAMSON et al., 2012), and used in this work for numerical validations. The upload of the curve points is made to the software via a CSV file (*Comma Separated Values*).

In the work of (WILLIAMSON et al., 2012), were used two Reynolds numbers: 200,000 and 250,000. For the first part of this study, the analysis were made to these values of Reynolds according to the parameters used in the wind tunnel test developed by (WILLIAMSON et al., 2012) and (SELIG; GUGLIELMO., 1997). With 300 mm of chord and 850 mm of span, as well as air density of $1.18415kg/m^3$, it is possible to find the velocity by the Reynolds Equation. It is important that these values keep close to the values of the problem studied (the average speed of formula SAE vehicles in the endurance race, skid pad or autocross, which is 12.5 m / s according to (EQUIPE ICARUS, 2015)).

For the boundary conditions, the inlet is set to the velocity in the regions of the front and above the airfoil; the outlet as the pressure distribution in the regions below and rear the airfoil; and a symmetrical condition for the symmetry plane in the regions of the sides of the control region for the 2D case and quasi-2D. For the three-dimensional case, a wall condition is adopted to the sides of the volume control.

The lift coefficient (C_L) is calculated by Eq.2:

$$C_L = \frac{F_y \cdot cos\,(\alpha) + F_x \cdot sen\,(\alpha)}{0,5 \cdot \rho \cdot V_{infty}^2 \cdot A} \tag{2}$$

In which α is the angle of attack , V_∞ is the velocity in the middle, ρ is the fluid density, A is the wetted area (the chord multiplied by the span), F_x is the horizontal force and F_y is the vertical force.

The Errors are given by:

$$Erro\,(\%) = \sqrt{\left(\frac{C_l - C_{lexp}}{C_{lexp}} x100\right)^2} \tag{3}$$

Where C_{lexp} is the lift value obtained by the wind tunnel by (WILLIAMSON et al., 2012).

3.1 Generation of the mesh

One of the most important areas in CFD is the creation of the mesh. It must always has a compromise between the number of cells and the quality of the mesh. More cells provide a better quality, but the simulation will take longer. Fewer cells, however, lead to less accurate results of the real model, but will be quicker, which is one of the crucial things when dealing with high performance racing.

The mesh is generated from an automatic option of trimmed type combined with prism layers in regions close to the wall. (CD-ADAPCO, 2015) states that it is a robust and efficient method for capturing high quality mesh in simple problems, such as the flow of air over an airfoil. Moreover, according to the work of (AIGUABELLA MACAU, 2011), the *polyhedral* mesh has the ability to deal with complex problems and not precise geometries, nevertheless *trimmed* one converges faster.

In the first stage in the creation of the methodology, three types of meshes are considered for the angles of attack of 0, 5 and 10 degrees: a less refined mesh, a medium and a more refined one. The lift coefficient obtained with these meshes for different angles of attack are compared with the experimental values.

The convergence of the simulations is made by numerical iterations aligned with an analysis of the graphics resolution of the governing equations. In this step, the grid is analyzed for three angles of attack and the object is focused in the linear region of the lift curve. Finally, the thickness of the boundary layer is refined to assess the quality improvement.

The control volume is defined by amounts of airfoil chord as indicated in the work of (MAIA, 2014). With the chord given by L, the control region necessary for a good analysis of the flow over a wing has a 5L away distance from the leading edge to the inlet. From the top of the control volume to the top of the wing, from the bottom of the wing to the bottom of the control volume and from the trailing edge to the outlet, the distance is 10L. It is important to care this region does not be too small so that the amount of aerodynamic force due to the viscosity of the wall does not influence the flow and does not be large enough to increase the computational cost. This region can be visualized by the figure below.

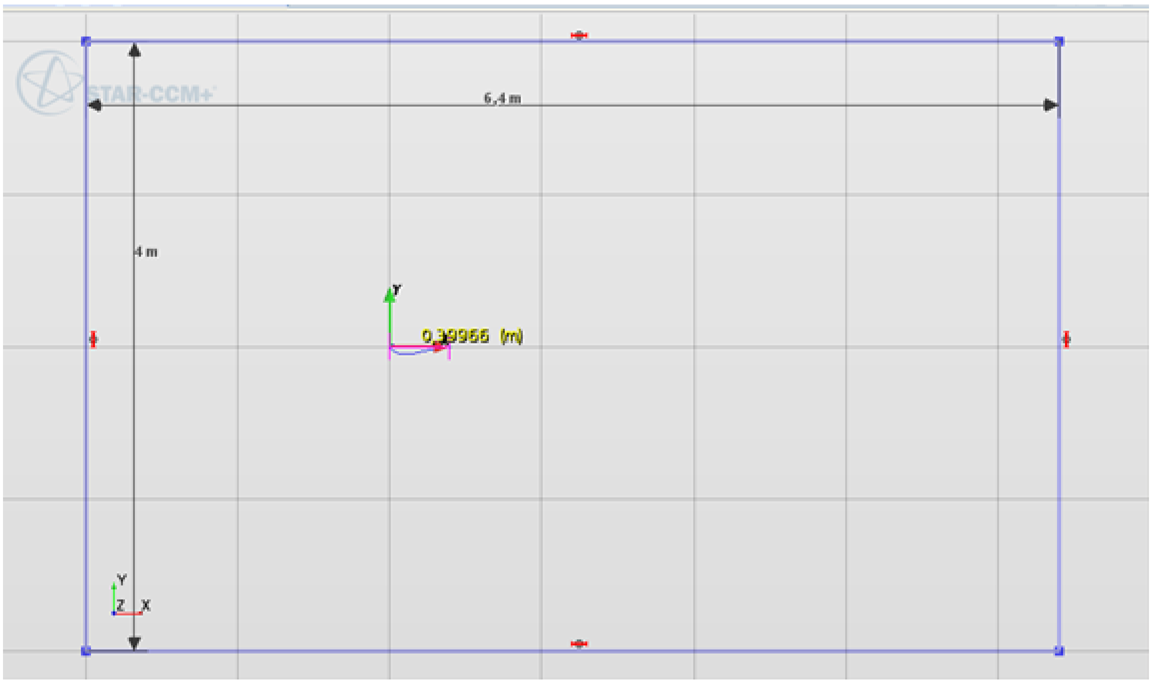


Figure 2: Definition of the control volume

The creating process of the mesh is made with three types of refinement: low (Mesh 1) , medium (Mesh 2) and high (Mesh 3). The lift coefficient of the coarse mesh were compared with the experimental values. Gradually the mesh was being refined until a point where the quality of the results were satisfactory in the point of view of the computational cost (it is called less refined mesh). From this point it was reached more two levels of meshes quality: a medium refined and a more refined one. These three meshes were used and compared in the study of the methodology. The profile for analysis of these meshes was tested at angles of 0, 5 and 10 degrees. The Table 2 shows the parameters definitions used in each mesh.

Table 2: Parameters used for the mesh refinement.

Parameters	Mesh 1	Mesh 2	Mesh 3
Base size	100 mm	100 mm	100 mm
Target surface size	5%	2.5%	2.5%
Minimum surface size	1%	0.1%	0.01%
Number of prism layers	4	4	4
Prism layer stretching	1.3	1.3	1.3
Prism layer total thickness	5%	5%	5%
Volume growth rate	Fast	Medium	Slow
Maximum cell size	50mm	50mm	50mm

The base size parameter is a control variable as other parameters are defined as a percentage of it. In this experiment, it was defined as 100 mm, once the chord is 300 mm.

It can be noted by the Table 2 the mesh 3 is denser than mesh 2, and these, in turn, is denser than mesh 1. This is justified by the quantity of elements from one to another.

The program calculates the elements size until the target surface size, however does not discard zones that need more refinement. In these zones, the elements which size is lower than the minimum surface size value are discarded.

The number of prism layers parameter indicates the quantity of elements layers present in the boundary layer. This parameter was set to 4 elements. A total thickness of these layers is defined by the parameter *prism layer total thickness*.

The thickness of each layer is determined by the previous layer with the prism layer stretching parameter. In this problem, the thickness is set to 1.3 times the previous layer.

The volume growth rate parameter indicates the generation rate of the mesh size from one layer to another. A fast growth rate, increase cell size quickly. On the other hand, a lower rate, indicates that the cell uses multiple mesh layers providing a gradual transition. This parameter can be better understood by the following figure.

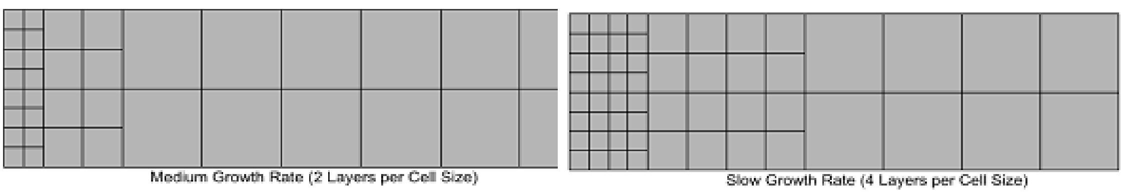


Figure 3: Growth rate.

3.2 Turbulence model

In this section, the profile is analyzed to angles of attack of 0, 5, 10, 12, 15 and 18 degrees, with the k-epsilon, k-omega and Spalart-Allmaras turbulence models. The lift coefficients are compared with the experimental values. It is important to remember that the experimental values of work done by (WILLIAMSON et al., 2012) and (SELIG; GUGLIELMO., 1997) were conducted in a controlled environment with the aid of a wind tunnel and hence the reliability of these results.

The mesh used for each angle of attack is the same, changing only the x, y and z components to obtain the flow angle recquired.

The turbulence model is defined according to the problem. For simulations involving flow over an airfoil, the RANS models meet satisfactorily the problem requirements and provide accurate values for the resolution of the governing equations (CD-ADAPCO, 2015).

For the flow problem on the formula SAE vehicle wing, a steaty state was adopted. Thus, as the fluid is air and the compressibility effects are negligible, the equations are solved for ideal gas due to this have good solution when it comes to determining the lift coefficient.

The solution of segregated flow solves the flow equations in a separated way and then the moviment and moment equations are connected by a corrective approximation. There are three types of segregated fluid energy used in StarCCM+. The temperature one is used in this paper.

The turbulence models adopted was the realizable k-epsilon with two layers; k-omega SST (Menter); and Spalart-Allmaras standard model.

In the first model, the transport equations are solved to the turbulent kinetic energy k and its dissipation psilon. It is able to deal with general flow and not only external ones (AIGUABELLA MACAU, 2011). The two layer model is a type of K-epsilon which uses low reynolds number to apply K-epsion in the sublayers.

The K-omega SST (Shear Stress Transport) uses the especific dissipation rate instead of the dissipation used in K-epsilon. It solve the problem of applying the standard model to practical problems and it is sensible to the refinement of the mesh.

The last model solves a single transport equation to determine the turbulent viscosity. This method is indicated for problems with associated boundary layer and flow with smooth detachment as a flow on airfoils.

4 RESULTS ANALYSIS

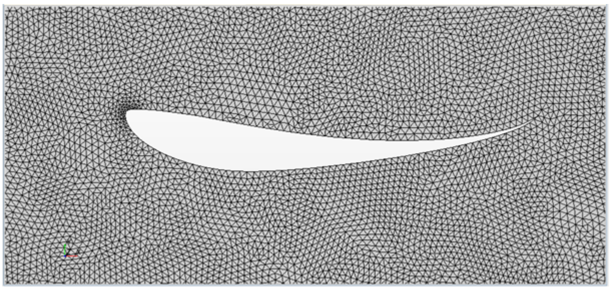
4.1 Mesh Convergence

The following figures show the level of convergence of each mesh for zero degrees of angle of attack. In order to assure the mesh of 2D is the same as 3D, the meshes were generated firstly in 3D and with then converted to 2D by the command ”convert to 2D”.

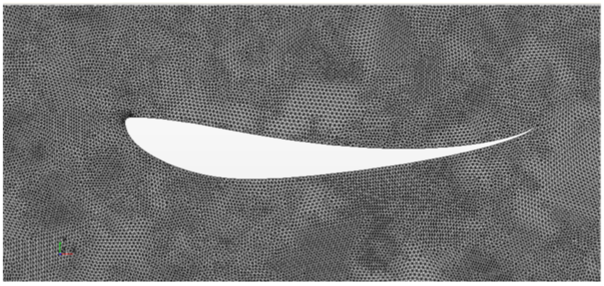
Table 3: Number of elements of each mesh.

	Mesh 1	Mesh 2	Mesh 3
Elements	2.183.143	9.849.521	9.850.255

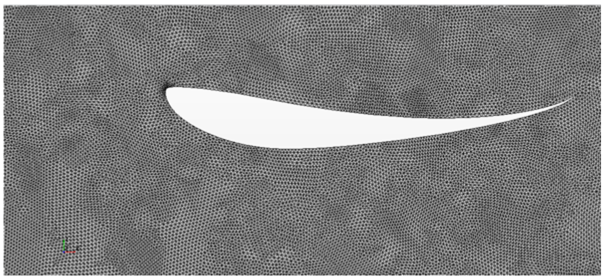
This table indicates the number of elements generated for each mesh. The larger it is, the more refined will be the mesh.



(a) Mesh 1



(b) Mesh 2

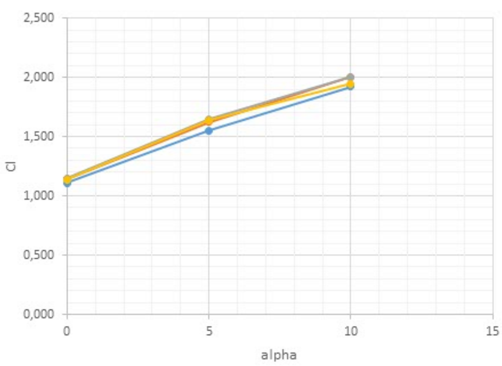


(c) Mesh 3

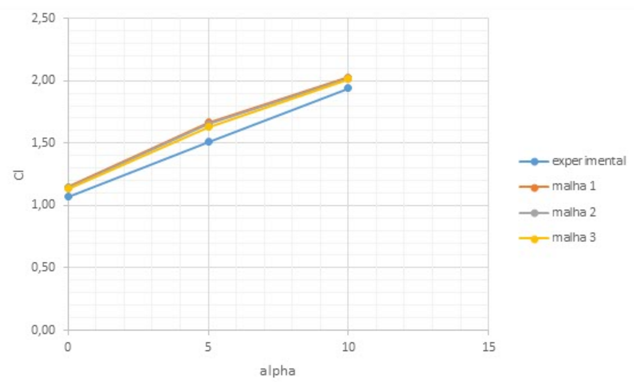
Figure 4: Refinement of each mesh.

The Figure 4b and 4c appear to have the same refinement, as they were set to have a target surface size of 2.5 mm. However, the probable diference between them is the first one does not allow elements lower than 0.1 mm, and in the second one this number is 0.01 mm. In other words, they are not necessarily differents as they are only different if the simulation require more refinement at some point.

The simulations are done with 200,000 and 250,000 for Reynolds for each mesh and results are shown in the Fig. 5a and 5b. The blue curve is the experimental one obtained by (WILLIAMSON et al., 2012) with the s1223 profile in 0, 5 and 10 angle of attack; The red one is for the Mesh 1, the gray is for the Mesh 2 and the yellow one correspond to the Mesh 3. The graphics correspond to the angle of attack (alpha) in function of lift.



(a) 200,000 Reynolds



(b) 250,000 Reynolds

Figure 5: Lift coefficient curve.

The Figures 5a and 5b show the lift curve in the linear region for the three meshes aforementioned. It is importante that the airfoil works inside this zone in order to avoid lift loss caused by stall.

The convergence study shows that the results are very close for the different meshes and for the experimental curve. Analysing both graphics for 200,000 and 250,000 Reynolds, it is possible to see that the use of more refined meshes does not improve the results.

4.2 Convergence of turbulence model

In the study of the convergence models, it is also used the two values of Reynolds, however, more angles of attack is simulated in order to know the behavior of each model throughout all the linear curve.

The Spalart-Allmaras model uses one equation while the k-epsilon and k-omega use two equations for the description of the fluid flow.

The Figure 6 represents the results for the simulation with 200,000 Reynolds of the different turbulence models.

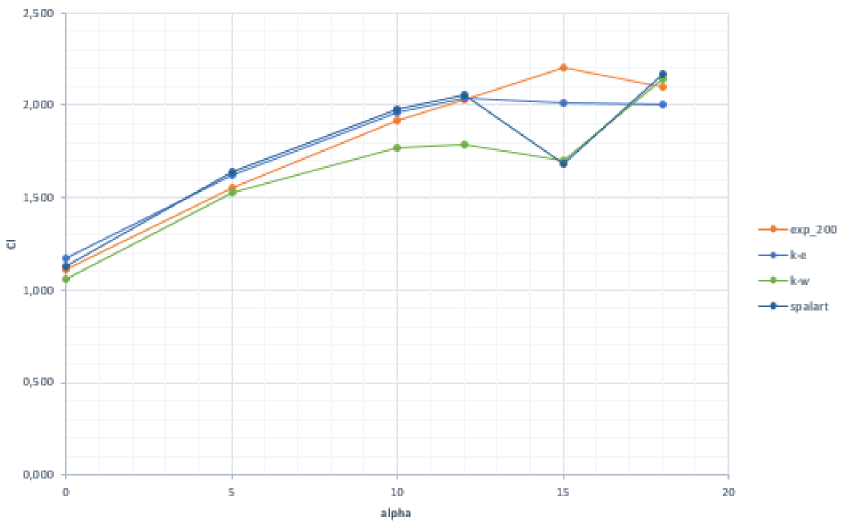


Figure 6: Lift curve of 200,000 Reynolds for different turbulence models.

For the Figure 6, it is possible to see that as the angle of attack increase, the error also increases in a significant way. In this region, the k-epsilon and k-omega models need more refinement. This can be explained by the fact that the equations of fluid flow and the governing equations of fluid dynamics do not converge totally.

The mean errors for each model is calculate by the Eq. 4

$$\overline{Error}\,(\%) = \frac{\sum_i^n Error_i}{n} \tag{4}$$

In the above equation, the angles above 15 degrees are discarded as these values are out of the linear curve and so the fluid are stalled. Thus:

k-epsilon =4,23$\%$

k-omega =9,63%

Spalart-Allmaras=7,17%

According to Figure 6 it is possible to conclude that k-epsilon and Spalart-Allmaras models have better approximations when compared with k-omega. However, evaluating for the computational cost, the Spalart is better because it uses only one equation as aforementioned.

For the 250,000 Reynolds, the results of the errors are:

k-epsilon=5,25%

k-omega=8,57%

Spalart-Allmaras=10,53%

Despite the errors of k-epsilon show better results, these values are not too significant between the models and so the Spalart model is chosen for the following sections.

4.3 Boundary Layer

The small region next to the surface is important to the description of the turbulence flows. This is duo to the fact that the turbulence begins when the fluid is dettached from the surface. The Equation 5 refers to the thickness of the boundary layer for flat plates (ANDERSON, 2001). The calculations are done for Reynolds of 200,000 with Spalart model model.

$$\delta = \frac{0.3747L}{Re^{0.2}} = \frac{0.3747 * 0.3}{200.000^{0.2}} = 9.786mm \tag{5}$$

The boundary layer is tested with 10, 20 and 30 layers. The Figure 7 shows the improvement of the results with the use of boundary layer. The simulation with 10 layers in yellow has the less error and proved to be useful.

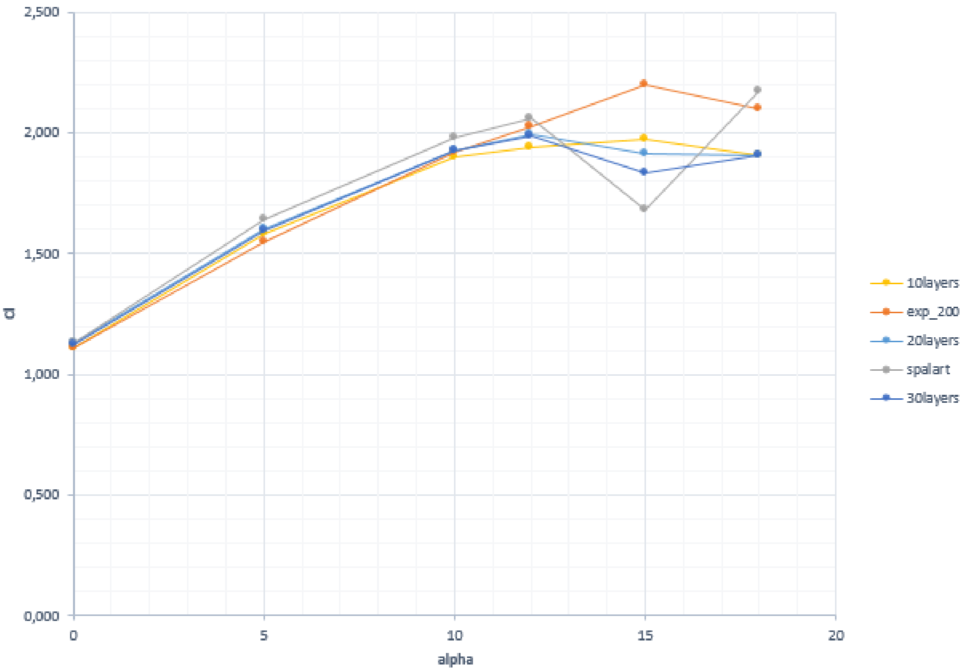


Figure 7: Curve of Lift coefficient for Spalart model with 200,000 of Reynolds

The boundary layer was studied and provided more accurate values in comparison with the computational results according to the Fig.7. The difference for further refinement of the boundary layer can be justified by the fact that the increase of layers results in numerical rounding errors.

4.4 Quasi-2d

The quasi-2D is a model 3D with so small thickness that can be compared with 2D. This profile is studied in order to validate the 3D case from the 2D. This model is also known as sandwich and can be saw by Fig. 8. The thickness is set to be 20 mm.

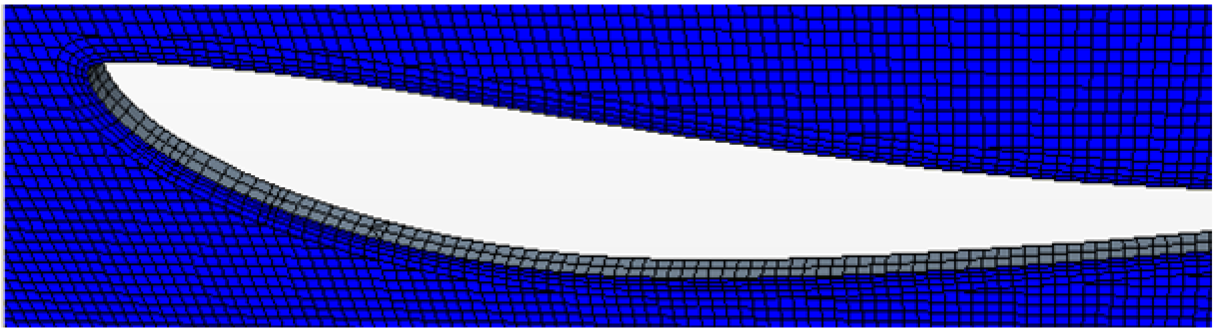


Figure 8: Thickness of the simulated airfoil

The results for the quasi-2D profile is shown by the Fig.9. The curves are simulated for 200,000 of Reynolds with Mesh 1.

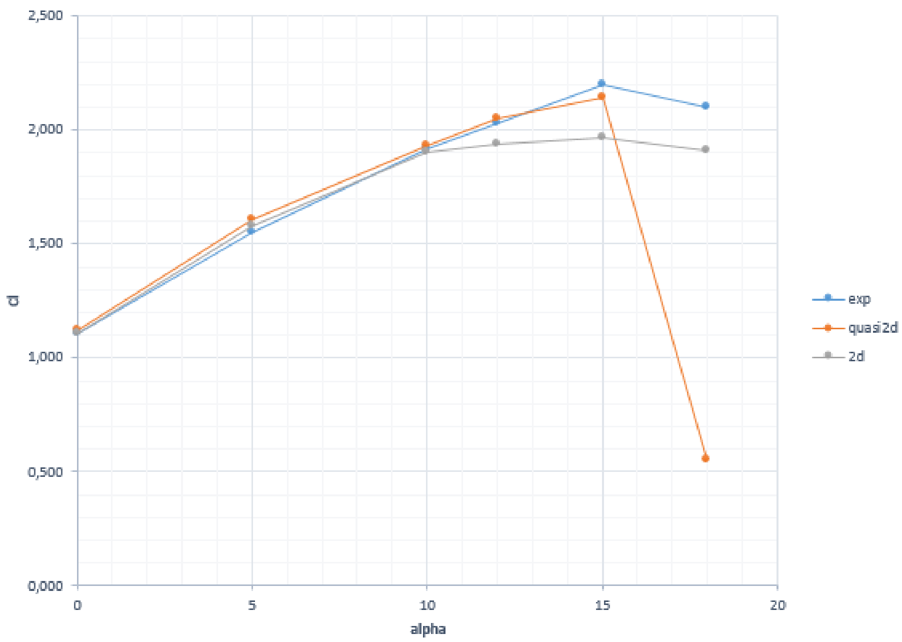


Figure 9: Comparison between quasi-2D, 2D and experimental curves.

It can be seen by the Figure 9 that the quasi-2D model has a large error for 18 angle of attack. This can be explained by the presence of gradient of adverse pressure such vortex structures which are commum when it occurs the detachment of the boundary layer.

4.5 3D

The 3D simulation is represented by the Fig. 10.

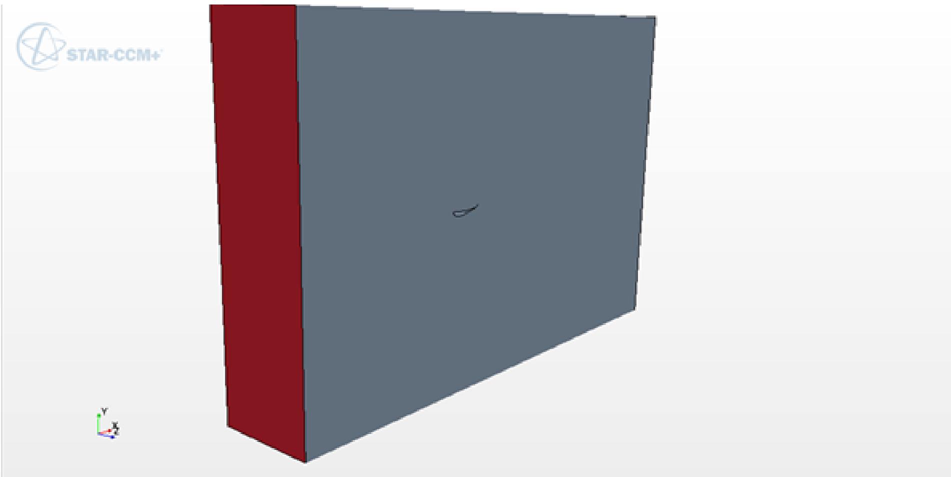


Figure 10: 3D model of the profile in the control volume.

The results of the simulation for 3D are shown in Fig. 11. For this case, it is simulated only three angle of attacks due to the high computational cost.

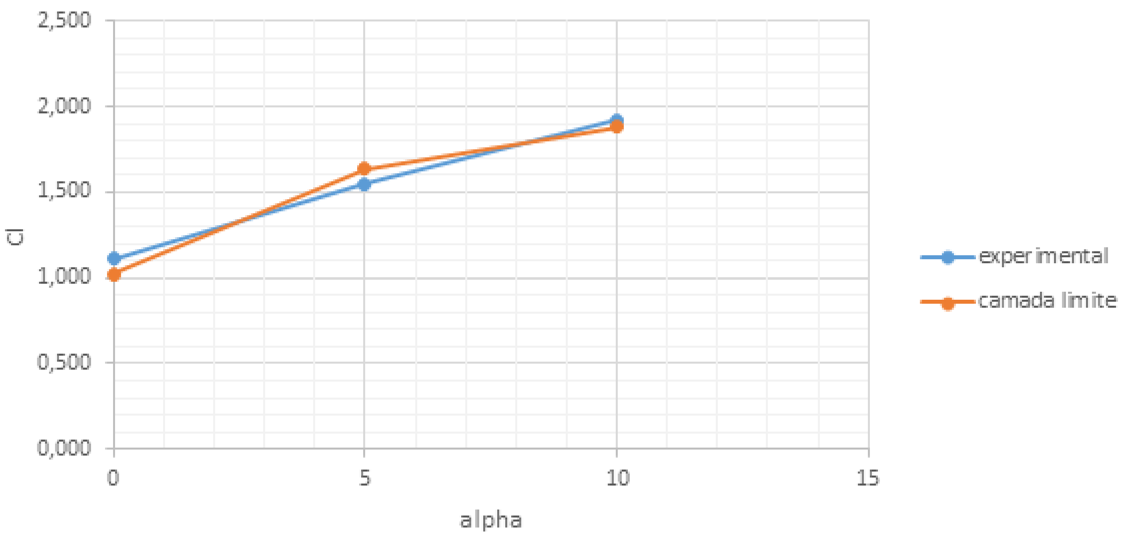


Figure 11: 3D curve compared with the experimental one.

In the Table 4 it is possible to see the errors of the linear region for full scale model.

Table 4: Results for the 3D simulation.

AOA	3D Calculated CL	Experimental CL	3D Error (%)
0	1,02	1,11	7,73
5	1,64	1,55	5,67
10	1,88	1,92	2,07

4.6 Numerical Methodology

From the simulations done, it can be proposed the following numerical methodology for the analysis of aerodynamics surfaces:

1. Study and development of a mesh able to capture the fluid behavior from known airfoils profiles in the literature;

2. Evaluating the use of the boundary layer;

3. Study of the turbulence model in order to see which one better match with the problem;

4. Simulation of the sandwich element in order to validate the 3D case;

5. Simulation of the profile for real conditions.

5 CONCLUSIONS

The aim of this study was to develop a numerical methodology for the design of an airfoil for race cars, especially the formula SAE electric vehicle of the University of Brasilia team.

The studies of mesh convergence were satisfactory on the quality of results within the linear range of the airfoil lift curve and the use of boundary layer was satisfactory to justify its refinement.

The K-Epsilon and Spalart-Allmaras proved to have a good approximation of the governing equations of the fluid flow in the linear region and the Spalart-Allmaras was indicated in order to have less computational cost.

The results for Quasi-2D were obtained with low errors when compared with experimental values in the linear range. However, an analysis for a greater number of Reynolds could provide a confirmation of the results.

In the latter case, the analysis of the full scale profile were satisfactory with small errors. However, further analysis would need to be made to check the behavior of all linear region before the stall and after the stall.

The methodology developed was adequate for the analysis of aerodynamic surfaces and therefore it could be used for different profiles of airfoils. Although require a greater Reynolds range for greater reliability in the results, observing the limit used in vehicles of the competition Formula SAE (less than 1e6 according to (MSC SOFTWARE, 2015)).

One possible source of errors is due to the analysis of the convergence, which it is up to the engineer to decide whether it occured or not from the residual and force graphics.

REFERENCES

AIGUABELLA MACAU, R. Formula one rear wing optimization. master thesis. Universitat Politecnica de Catalunya, 2011.

AIRFOILTOOLS. Airfoil plotter. s1223-il.acesso em 21 de maio de 2015. 2015. Disponível em: ⟨Airfoiltools.com⟩.

ANDERSON, J. *Fundamentals of aerodynamics. Traducao.* [S.l.: s.n.], 2001.

CD-ADAPCO. Steve portal. documentation. 2015. Acesso em: 5 maio 2015. Disponível em: ⟨https://steve.cd-adapco.com/Home⟩.

EQUIPE ICARUS. Poli ufrj. 2015. Disponível em: ⟨http://www.equipeicarus.poli.ufrj.br/?page=competicao⟩.

FORMULA 1. Inside f1. understanding f1 racing. aerodynamics. acessado em 3 de maio de 2015. 2015. Disponível em: ⟨http://www.formula1.com/content/fom-website/en/championship/inside-f1/understanding-f1-racing/aerodynamics.html⟩.

KATZ, J. *Racecar aerodynamics.* [S.l.: s.n.], 1995.

KIEFFER, W.; MOUJAES, S.; ARMBYA, N. Cfd study of section characteristics of formula mazda race car wings. mathematical and computer modelling. p. 1275–1287.

MAIA, R. S. Cfd analysis concept, school of engineering and technology. University of derby, 2014.

MSC SOFTWARE. Academic case studies, cal poly pomona formula sae team. n.p., 2015. web. 2015.

PAKKAM, S. High downforce aerodynamics for motorsports. [raleigh, north carolina]. North Carolina State University., 2011.

SELIG, M. S.; GUGLIELMO., J. J. 'high-lift low reynolds number airfoil design'. journal of aircraft 34.1. p. 72–79. Web., 1997.

WILLIAMSON, G. A. et al. Summary of low speed airfoil data, vol 5. University of Illinois, Department of Aerospace Engineering, Urbana-Champaign, 2012.