1
Introduction

Nowadays in the world of the motorsport industry, among the several research fields involved in the design of race cars, aerodynamics is the most rewarding in terms of performance improvements per resource and time invested. Using aerodynamic devices to generate negative lift enhances the overall car performance, decreasing lap times. They generates an extra force that helps to vary notably the dynamic behavior of the car enabling faster cornering speeds as well as shorter breaking lengths.

The first devices used in motor-sport were inverted wings mounted directly on the rear suspensions, supported by stilts. Nowadays, formula cars use two inverted wings located in the front and in the back. The purpose of them is to maintain the equilibrium in the car while producing maximum aerodynamic downforce.

In the first years of development, the design of these devices was accomplished using classical wing theories, wing tunnels measurements or just previous experience. Almost all changed when some teams started to use numerical simulations to develop them. Since then, the role of Computational Fluid Dynamics CFD in the study and development of car aerodynamics changed. All the progress CFD attained during the past two decades contributed to make it a main tool in the designing process and the optimization of aerodynamic devices. Currently, CFD data is considered as important as experimental measurements in the prediction of forces and flow characteristics, since it allows the engineers to analyze the complete flow field around the car, not to mention the possibility of studying other situations that would not be practical nor economical with experimental methods such as transient aerodynamic forces on overtaking.

Some additional advantages that CFD simulations offer are the possibility of dealing with complicated configurations, be it, large systems or isolated elements, the reduction of time and cost of designing iterations and specially visualize the flow field in order to gain an understanding of how it works. There are also some disadvantages that have impeded CFD to overcome experimental measurements as the main tool in aerodynamic studies, for instance, low reliability of the results, difficulties to generated the grid, and the cost of software licenses.

Even so, the development of robust discretization schemes, unstructured grid generators and the incredibly fast growth in computational power, enabled engineers to solve the Navier-Stokes equations over complicated geometries contributing to the generally acceptance of CFD as a tool for aerodynamic design.

Today, CFD offers several techniques to deal with the most diverse engineering and scientific problems. For example, among the various methods to solve for turbulent regime, the Reynolds Average Navier-Stokes (RANS) method stands out, offering a good trade between required computational resources and accuracy. Although its results are not completely reliable, the amount of qualitative data one can extract from a simulation represents a valuable starting point to the enhancement of any system or device. So one can say that the maturity and the advantages, the RANS method offers, overcomes the disadvantages the method still has.

On the other hand, more complete simulations such as Direct Numerical Simulations or Large Eddy Simulations represent more reliable numerical tools but still are limited to the exploration of simple flows at moderate Reynolds numbers. Therefore, in the next years, until enough computational resources are available and affordable, both techniques will continue being part of simplified analysis and sources of validation for turbulence models.

1.1 Objectives

Understanding the characteristics and the effects of the flow structures, present in the flow over low aspect ratio wings (race-car wings), is the starting point in the process of designing new wings or optimizing existing geometries. The success in these processes lies in the effective control or elimination of the negative effects caused by some flow structures as well as taking advance of the positive ones to improve the overall wing performance. As it was mentioned the most cost-effective way to accomplish this task is to use the complete flow description provided by CFD simulations.

Since the design and optimization of aerodynamics devices in the motorsport industry are constrained to produce accurate results in very short periods of time at relative low cost, numeric techniques are expected to be able to comply with these requirements. Two major stages, influencing the total accuracy, time and cost of numerical simulations, are those related to grid generation and processing time. Optimizing at least one of them would result in a considerable reduction of the simulation time and cost, and would increase the simulations credibility. To this purpose, automatic grid generators appear as a viable alternative, since they are capable to deal with complex geometries, produce relatively less elements compared with structured grids, and produce reliable results.

Therefore, the present dissertation is aimed at studying the flow over a race-car rear wing using a methodology that is able to accurately capture the flow physics and produce rapid results by reducing the grid generation and processing time. Focusing to decrease the overall cost of the simulation, it is proposed to employ an open-source code so as to generate the grid and solve the simulation.

1.2 Overview

Basic definitions are presented in chapter 2 in order to familiarize the reader with the terminology used along the following chapters. Besides, it is presented a short literature review about the main parameters that govern the flow over wings as well as CFD analysis.

Chapter 3 presents the numerical models employed to simulate the flow over the rear wing. The governing equations are defined as well as the hypothesis and simplifications that were made in order to facilitate the analysis. Also, some features of the grid generator along with the CFD code are described. In order to evaluate the features of the grids generated automatically, chapter 4 presents the validation of the numerical method by comparing 2D simulations with experimental values and validated numerical data. The Tyrell airfoil data is the basis of this validation.

Chapter 5 presents the results of the three-dimensional simulation and discusses them based on the flow visualization and the calculation of the aerodynamic coefficients. Finally, the conclusions drawn from the study along with recommendations for future works are presented in chapter 6 .