2 Background

A brief survey about the theory involving the external flow about of solid bodies is presented in this chapter. First, some of the definitions used in the study of streamlined bodies are highlighted, followed by an outline of the physical phenomena involving external flow. Also included is a historical review about the evolution and development of aerodynamic devices in the motorsport industry. Finally, some important features about numerical simulations are highlighted.

2.1 Introduction

In the period between 1912-1918, the analysis of airplane wings took a step forward when Ludwig Prandtl and his colleges at Gottingen Germany, showed that the aerodynamic consideration of wings could be split into two parts: (a) the study of the section of a wing, namely ”airfoil” and (b) the modification of such airfoil properties to account for the complete, finite wing. This approach is still used today, as an example one can see in the literature how most of the CFD simulations are validated through two-dimensional experimental data [17].

The two-dimensional analysis is a very useful tool at the beginning of a project, as it allows rapid cost-effective comparisons of wing profiles regarding its ability to generate lift or in this case downforce. In two-dimensional analysis, the number of independent variables to represent the flow are reduced, decreasing the total time of its processing and analysis, for example, the pressure and velocity field only depend on two variables x, y and also there only exist two sources of drag called skin friction drag and pressure drag.

In order to use the Prandtl's first idea, the experiments carried out to measure the lift, drag and moment coefficients on airfoils are performed at low speeds in a wind tunnel, where the wing spans the entire test section from one side to the other. Thus, it is possible to have a physical condition where the wing does not have tips. This condition is so-called infinite wing condition, in which the wing theoretically stretches to infinity along its span. And because the airfoil section is the same at any spanwise location along the infinite wing, the properties of the airfoil and the infinite wing are identical [17].

On the other hand, a finite wing is a three-dimensional body. The flow has a third component z in the spanwise direction, and therefore, it is expected that the overall wing aerodynamic properties differ from those of its airfoil section. Due to the three-dimensional flow behavior around the finite wing, a phenomenon known as wing tip vortex appears. It takes place when the flow near the wing tips tends to curl around the tips forced by the high pressure difference between the lower and upper surfaces of the wing. Its presence modifies the pressure distribution over the wing producing a net pressure imbalance in the direction of the free-stream velocity. Since this pressure imbalance acts in the free-stream direction, its total effect is to increase the drag. Hence, for subsonic flow the total drag on a finite wing is the sum of the skin friction drag, form drag (pressure drag), and the induced drag. Consequently, in order to accomplish a thorough analysis of the wing features, it is essential to take into account the three-dimensional effects according to Prandtl's second idea. For this purpose, it is necessary to carry out full scale wind tunnel measurements or three-dimensional simulations.

2.2 Airfoil Definitions

In 1929, the National Advisory Committee for Aeronautics (NACA) began to study the characteristics of systematic series of airfoils in an effort to determine and normalize its characteristics. The published data shown the effects of varying the geometry on their aerodynamic characteristics such as lift, drag and moment. As a result, the nomenclature used by NACA was adopted to designate airfoil characteristics.

Based on this nomenclature some of the definitions used in the present study to designate important airfoil geometrical characteristics are mentioned below:

The ***mean camber line*** is the locus of points halfway between the upper and lower surfaces as measured perpendicular to the mean camber line itself. The most forward and rearward points of the mean camber are the ***leading*** and ***trailing*** edges, respectively. The straight line connecting the leading and trailing edge is the ***chord line*** of the airfoil, and the precise distance from the leading to the trailing edge measured along the chord line is simply the ***chord*** of the airfoil. The ***camber*** is the maximum distance between the mean camber line and the chord line, measured perpendicular to the chord line. The ***thickness*** is the distance between the upper and lower surfaces, also measured perpendicular to the chord line. The ***angle of attack***, is the angle measured between the chord line and the free-stream velocity vector [17].

2.3 Aerodynamic Forces

The generation of lift and aerodynamic resistance (drag) involves a complex interaction of the body, the air and in some cases the ground. The resultant forces on the body are due two basic sources: **Pressures distribution** over the body surface and **Shear Stress distribution** over the body surface.

Pressure acts perpendicular to the surface contributing to both lift and drag. When a fluid flows around a body the difference on the pressure distribution is due to the alteration of the surrounding velocity field. Shear stress acts tangential to the surface contributing to drag and lift. Its presence is the result of the friction caused by the interaction of the viscous boundary layer and the solid surface.

Global parameters such as velocity, pressure and forces can be calculated or measured using both experimental and computational methods. Such data are best presented in dimensionless coefficients to condense the information and reduce the complexity of variables which affect a given physical phenomenon, such that trends and rates of change can be observed. The dimensionless coefficients associated to both forces drag and lift are computed using the free stream dynamic pressure as follows:

$$C_L = \frac{L}{\frac{1}{2}\rho U_\infty^2 A} \, , \tag{2.3.1}$$

$$C_D = \frac{D}{\frac{1}{2}\rho U_\infty^2 A} \, , \tag{2.3.2}$$

where the area A is computed by the product of the wing chord c times the wing span s, U_∞ is the free-stream velocity, and ρ is the free-stream fluid density.

2.4 Flow Regimes

The Reynolds number Re is a dimensionless value that gives a measure of the ratio of inertial forces and viscous forces

$$Re = \frac{U_\infty c}{\nu} \, , \tag{2.4.1}$$

where c is the wing chord and ν is the fluid kinematic viscosity.

Since the characteristics of the external flow around solid surfaces clearly depend on the flow conditions and a characteristic length, the Reynolds number is a natural relation to identify the different regimes that appear when flow conditions vary.

There are two different flow regimes corresponding to two different Reynolds numbers, namely laminar and turbulent. A transitional regime appears when laminar regime shifts to turbulent regime.

At laminar regime the viscous forces prevails over the inertial forces. The flow over a solid body is smooth and keeps attach to the surface. The flow is more smooth, producing momentum exchanges only between adjacent layers on a molecular scale, which in turn originates shear stresses entirely due to molecular diffusion. As the inertia forces increase, instability appears and the flow becomes turbulent. The flow in this case is not uniform and highly random. Its motion is three dimensional and unsteady, making it naturally more dissipative than the laminar flow. The velocity fluctuations produced by the random motion increases the momentum exchange between adjacent layers resulting in shearing stresses between them, which added to those produced by molecular diffusion, increases the values of skin friction drag.

2.5 Boundary Layer

The study of external flow has represented a great challenge for scientist and engineers, who in an attempt to compute viscous effects near solid walls have proposed several theories based on empirical relationships obtained from experimental measurements or analytical solutions of the governing equations for simplified geometries. The best approach, proposed by Prandtl in 1904 and still used in present days, consists in divide the flow in two regions, one totally governed by viscous effects and another where the flow can be considered inviscid, so as to simplify the analysis. This theory is known as boundary layer theory and is surveyed below.

Prandtl proposed that the influence of viscosity at high Reynolds numbers is confined to a very thin layer in the immediate neighborhood of a solid wall, inside which the fluid velocity increases from zero at the wall to its full value at the end of the thin layer, which corresponds to external frictionless flow velocity [2]. In other words, the presence of the boundary layer is the result of the friction between the fluid and the surface, which retards the flow in its immediate vicinity, while the fluid away from the surface remains unaffected by these viscous forces.

Provided that the boundary layer remains thin, the flow Reynolds number is sufficiently large, the body is of a streamline form, and its angle of incidence to the flow is moderate, the boundary layer approach enables the possibility of estimating the amount of shear stress at the body surface and the pressure field produced about the body.

The pressure field is computed considering that the flow-field pressure outside the boundary layer, where the flow is essentially inviscid in behavior, prevails throughout the body surface and that the influence of the boundary-layer flow on the pressure is small enough to be considered. On the other hand, wall shear stress is computed solving the equations of viscous motion inside the boundary layer, where appreciable simplifying assumptions enables their solution.

Boundary layers can be laminar or turbulent. As the ratio between inertia and viscous forces changes along the surface, the regime changes from laminar to turbulent. Thus, the boundary layer regime depend on the flow features at a given position on the surface. In the surface of a solid body the transition from laminar to turbulent regime is not abrupt, it always begins as a laminar boundary layer which gradually becomes turbulent as the boundary layer thickens.

In order to gain understanding of the phenomena inside the boundary layer, it can be sub-divided into smaller regions, in which the flow presents different characteristics. To describe these sub-regions, it is practical to use wall units or a non-dimensional parameter known as y^+, which represent the distance from the wall in viscous lengths (*viscous lengths are length scales in the near-wall region which relates the viscosity and the friction velocity*). This distance is considered as a local Reynolds number, so its value relates the importance of viscous and turbulent processes.

It is defined as

$$y^+ = \frac{u_\tau d}{\nu} \ , \tag{2.5.1}$$

where $u_\tau = \sqrt{|\tau_w| / \rho}$, d is the distance to the wall and τ_w is the shear stress at the wall.

Based on y^+ values, two major regions can be identified. One in the inner region of the boundary layer where the viscous effects dominate, and one in the outer region of the boundary layer in which inertial forces dominates and viscosity effects can be neglected.

Inside the inner region, it is possible to identify three additional subregions. The viscous sublayer $y^+ < 5$, in which the turbulence effects are negligible compared with the viscous effects. The buffer layer $5 < y^+ < 30$, in which the viscous and turbulence effects have the same importance. And the log-law region $y^+ > 30$, in which the viscosity has little effects and the turbulence dominates.

2.6 Physical conditions affecting the flow around a wing

Pressure and shear stress distribution over a body surface can vary due to several external and geometric conditions. These conditions influence the flow field by imposing physical constrains that ultimately modify the overall flow behavior. Some of these conditions are: body shape, surface roughness, angle of attack, aspect ratio, free-stream velocity, free-stream turbulence, external pressure gradient, aspect ratio among others.

2.6.1 Effects of Body shape and angle of attack

Body shape and angle of attack modifications varies the total lift and drag. Modifications of the angle of attack, such as in the camber line or the thickness of an airfoil can produce an increase of the lift as well an increase of the drag [17]. For example, in symmetric airfoils at low angles of attack, lift varies linearly with the angle of attack up to a point where flow tends to separate from the top surface of the airfoil creating a large wake behind the airfoil. The consequence separation at higher angles is an abrupt decrease in lift and a large increase in drag; under such conditions the airfoil is said to be stalled.

2.6.2 Effects of free-stream velocity and turbulence

The effect of the Reynolds number on airfoils may be notice as the speed varies. With other parameters fixed, at higher Reynolds number the airfoil performance improves creating more lift, the boundary-layer thins and the friction coefficient usually decrease.

Free-stream turbulence is an important external condition which has to be taken into account in aerodynamic analysis, since it influences the development of boundary layers. The role of the free-stream turbulence is generally ignored, or assumed not to play a significant role in boundary layers, in order to simplify the analysis of the flows under study. However, according to Brand L. et all. [26] free-stream turbulence intensities of 1% or more can produce a more rapid transition from laminar to turbulent boundary layers affecting the complete flow over solid bodies. Moreover, according to Torres-

Nieves S., [31], one of the most significant effects of free stream turbulence is the increase of skin friction, which is directly related to the drag over the wing.

2.6.3 Effects of surface roughness

Surface roughness increases the friction between the air and the solid, adding disturbances in the laminar stream, which combine with those generated by turbulence already present in the boundary layer. If the disturbances generated by roughness are larger than those due to turbulence, it is expected that transition begins earlier [2]. For example, when the roughness elements are very large, transition will occur at the points where they are present, that is the case of the tripping wire normally used in experiments to promote turbulent flows over solids.

According to Myers B., [22], the presence of roughness on wings produces a reduction in maximum lift coefficient and stall angle of attack. Dimitriadis G., [1] agrees with the former statement, but pointed out that the lift is affected only at high angles of attack. Moreover, this author corroborates how surface roughness increases the drag coefficient at all lift coefficient values. In addition, it is important to remark that at high Reynolds numbers roughness promotes boundary-layer transition from laminar to turbulent flow, but at low Reynolds numbers roughness should have little effect.

2.6.4 External pressure gradient

External pressure gradient has a great influence on the stability of the boundary layer, and hence on transition. It can be adverse or favorable. An adverse pressure gradient means that the static pressure of the fluid increases in the direction of the flow, which ultimately induces deceleration of the flow. Conversely, favorable pressure gradients accelerates the flow, which stabilize the boundary layer and in some situations enable the flow relaminarization. On airfoils, external pressure gradients are produced due to the curvature. For example, the nose generates a favorable pressure gradient, while the trailing edge generates an adverse pressure gradient. This can be explained analyzing the velocity field over an airfoil; in the case of the leading edge the velocity reduces to zero in the stagnation point. In the case of the trailing edge, for angles of attack larger that zero, the velocity decrease produces the increase of the pressure at that zone.

2.6.5
Aspect ratio

The aspect ratio is defined as the ratio between the square of the wing span divided by its area [28]. It serves to differentiate between wings whose characteristics are more influenced by tip vortices and wings that are less influence by them. Low aspect ratio wings, for example, are particularly affected by induced effects produced by wing tip vortex: because the downwash induced by the trailing vortex system is quite intensive and increases with wing lift.

Most of aerodynamic research has been carried out to realize how to avoid the influence of tip vortices, which have led to solutions such as the use of wiglets or end plates located at the wing tips to decrease the effects of the pressure difference. In motorsport, for example, the use of low aspect ratio wings in the cars requires the use of end plates to keep the wings performance from the detrimental effects of tip vortices. According to Katz, J.,[28] and Soso, M., et al, [19], the performance of low aspect ratio wings can be maintained only by increasing the height of the end plates, as they can increase the effective aspect ratio of the wings.

2.7
Rear wing

The primary function of a rear wing in a car is to generate aerodynamic downforce, enhancing the performance of dynamic handling and stability of the car, and thus increasing cornering speeds.

A rear wing is composed of a single or multiple wings with end plates at their tips. The wing is composed by one or several section profiles (airfoils), which can be arranged in different setups according to the diverse tracks characteristics, for example, high lift for low velocity circuits or medium lift and low drag for high speed circuits. The end plates most employed in the rear wing are fins or shields. They generate an effective increase in the aspect ratio, preventing air leakage around the wing tips and thus the formation of trailing vortices, which improves the lift and reduces the induced drag. According to Katz J., [28], larger end plates can lead to a values of downforce and drag close to those calculated for the infinite configuration (airfoil).

The first rear wing appeared in 1966, when Jim Hall equipped his Chaparral 2E with a rear wing. From then on, use of wings grew quickly. In Formula 1 for example, wings were first introduced in 1968 at the Belgium grand prix, when Ferrari used full inverted rear wings, and Brabham did likewise, just one day after the Ferrari's wings first appearance.

A new development expected to generate more downforce occurred in the late 1970's when race car engineers paid attention to the well-known fact (within the aeronautical community) that the lift of a wing increases with ground proximity. This effect becomes noticeable when the ground proximity is less than one chord length of an airfoil [28]. As a result, aerodynamic design became as important as mechanical design in vehicle development and currently represents the best way to improve vehicle performance.

A rear wing, for instance, can produce 30-35% of the total downforce of the car and 25-30% of the total drag. To balance the car, the downforce created by the rear wing should be around twice as much as the one created by the front wing [31]. Thus, for a wing to be effective, it should generate maximum downforce and a small quantity of drag. Some possibilities to improve the wing's efficiency could be to increase the area or increase the camber using multi-element wings, which would augment the aspect ratio and camber at the same time.

2.8
Race car wings and CFD development

The first aerodynamic devices employed for motorsport applications were simple wings shaped based on airfoil's experimental data, published by NACA. They were manufactured and suited to race cars depending on the performance requirements. At that time, the effectiveness of these devices were evaluated according to the lap times for different setups, and almost none measurements were carried out to assure that the forces expected were generated.

In the following years, when almost all race cars started to use wings, experimental measurements appeared as an important tool used by teams to test their different designs. Simultaneously, wind tunnels started to be used, and soon they became the appropriate place for testing and developing new aerodynamic devices. However, the competitiveness of this sport led design costs to increase in such a way that spending time in wind tunnels for experimental testing became prohibitive for teams with small budget.

After some time, an alternative approach to reduce cost and improve aerodynamic designs was found in aerodynamic computational simulations; a wise alternative that was already being used by the aerospace industry mainly in aircraft developments and during the design of industrial constructions. Although computers were not as powerful at that time as they are today, it was possible to simulate aerodynamic flows using simple methods based on potential theory (Panel Methods), sometimes coupled to integral boundary-layer solutions. Naturally these simple approaches were restricted to surfaces and were usable only for areas of attached flow.

Posterior developments in discretization techniques and computational resources allowed to solve the inviscid part of the Navier-Stokes equations (Euler equation) using structured volume grids (Finite Volume Methods). It was a better approach, but still not enough to account for drag. The first record of its application to vehicle configurations are from the early 1980s, but for motorsport such an approach did not have any relevance until much later [23]. The reason for this delay may be attributed to the strong limitations in available geometry modeling and computational resources.

Only, in the early 90's after the emergence of unstructured grid methods CFD advanced to a higher stage of evolution. With the new available discretization approaches and more powerful computational resources, it was possible to solve the complete Navier-Stokes equations over complex geometries.

Nowadays, complete Navier-Stokes simulations are possible including heat transfer, compressibility, and multiple rotating reference frames [23]. The reduction of modeling restrictions together with the increase of computational capacity, allowed to change the design approach in the automotive industry, specially in the motorsport area.

2.9
CFD validation and verification

Credibility of simulations is still a main concern among the CFD users. Although significants advances have been made in improving the accuracy of CFD simulations, there is still little agreement on how to assess and report the simulated data. According to Unmmel B. [11], whenever simulations are presented, a statement regarding their credibility is essential, and also it must be shown that the achieved level of credibility is acceptable for the purposes for which simulations are being used.

There are various publications concerning this topic, among which are those published by NASA (National Aeronautics and Space Administration), AIAA (American Institute of Aeronautics and Astronautics) and some others from recognized organizations [6],[32], [24]. All of the authors of the mentioned publications agree that numerical simulations always have to be assessed, but there is not a complete agreement in how to carry out that assessment. Some publications about this topic tend to give guidelines, discussing fundamental concepts and specifying general procedures for assessing numerical simulations. Thus, in this work the criteria described in publications such as [12] and [13] are employed.

Three main concepts for assessing credibility in numerical simulations are: verification, validation and calibration. According to the AIAA report [13], verification is *"the process of determining if a computational simulation accurately represents the conceptual model; but no claim is made of the relationship to the real world"*. Validation is, on the other hand, *the process of determining if a computational simulation represents the real world*.

In essence, what verification does is assessing how accurate the mathematical equations, representing physical problems, are solved. Quoting Gerristma M., [20] verification could also be defined as the process of *'solving the equations right'*. To do so, according to Bienz C., [23], it is necessary to identify and quantify the error in the computational model and its solution.

Verification activities must be accomplished early in the development of the CFD code comparing the computational solution with analytical solutions or benchmark numerical solutions. Nevertheless, verifications such as grid and time-step convergence or iterative convergence are commonly perform when a problem is being solved. Grid and time-step convergence consist in refining the grid size and the time step until a little change in important variables can be observed. Iterative convergence is performed computing a residual error for each iteration, i.e., the magnitude of the disagreement between the left and the right of the difference equation. For instance, to reach iterative convergence the error has to decrease more than four or five orders of magnitude.

Validation, in the other hand, consists in determining how accurate the simulation can represent or predict a real world problem. Quoting Wthrich B., [33] validation could be define as the process of *'solving the right equations'*. To do so, errors and uncertainties in the conceptual and physical model are identified and quantified through systematically comparing CFD simulations with experimental data. Experimental data also have to be assessed to estimate the uncertainty of its values. This guarantee that the numerical simulation uncertainty is not going to be increased due to the comparison. Several approaches can be done in order to validate a complete system.

According to Metha et al., [11], calibration is *"the process of obtaining corrections factors for adjusting simulated results or for adjusting models, respectively"*. This is carried out tunning constants, adding or erasing terms in the equations with the only end of improving agreement of computational results with existing experimental data. Usually, calibration is used when there is a high level of uncertainty in the modeling of complex phenomena, for instance, turbulence, combustion in turbulent flows or multiphase flows are some areas where empirical model are used.

2.10
Grid generation

Grid generation is fundamental in CFD, since the grid properties influence directly the overall accuracy of the results. Complex geometries increases the difficulty to generate the grids, so different approaches have been developed to deal with an increasing variety of problems.

From a general point of view grids can be classified in structured and unstructured. Structured grids have a regular connectivity. Unstructured grids refer to arbitrary distributions of points, where the points are connected by triangles, quadrilaterals or polygons in two dimensions, or by various polyhedrons in three dimensions (tetrahedral, prisms, pyramids, hexahedral or arbitrary polyhedrons) .

Since structured grids are formed by the intersection of continues lines, one for each space dimension, they can be considered as most 'natural' for flow problems, as the flow is generally aligned with the solid bodies and one can imagine the grid lines to follow in some sense the streamlines, at least conceptually, when not possible realistically. According to Hirsch C., [34], since structured grids have a uniform distribution compared to unstructured grids, structured grids are often more efficient from a CFD point of view, in terms of accuracy. One of the reason relies on the fact that arbitrary distribution of elements in unstructured grids is considered as a source of numerical errors.

The reason behind the development of unstructured CFD codes is essentially related to the time required to generate "good" quality grids on complex geometries. For that reason, unstructured grids have progressively become the dominating approach to industrial CFD, due to their advantage in adapting to arbitrary geometries as well as the facility of performing local refinements in a certain regions, without affecting the grid point distribution.

In conclusion, according to Sawley M., [8], a careful choice of meshing technique is crucial for correctly capturing all essential flow field phenomena without creating a mesh of prohibitive size.

2.10.1
Hybrid Grids

Hybrid grids are unstructured grids, in which layers of quadrilaterals or prisms are generated in the near-wall region in order to fit the geometry and reduce the numerical diffusion caused when flow is at an oblique angle to the cell faces. Using hybrid grids is a common practice that have given good results. Most of the automated grid generators have this tool as part of their features, particularly when dealing with complex geometries.

According to Larson et al., [27], using a hybrid meshing strategy based on tetrahedral, hexahedral, and prismatic elements, has proven to be rather efficient. Studying the wake behind a F1 car they found that hexahedral grids can be less dissipative than tetrahedral grids leading to a much better capturing of the stream-wise vortex propagation in wing tip vortex assessment. Furthermore, they demonstrated that it is possible to achieve this enhancement using 25% less hexahedral cells than using tetrahedral elements. Therefore, it is possible to say that hexahedral cells tend to lead to a higher accuracy than tetrahedral cells, for the same level of mesh non-uniformity [34].

2.11 Turbulence Models

Modeling turbulence has been one of the the greatest challenge in fluid mechanics during the last century. Even though the Navier Stokes equations can correctly represent mathematically fluid flows in all regimes (laminar to turbulent), solving them directly, for high Reynolds numbers, requires computational resources that currently are not available. Different approaches have been created to solve for turbulence using less computational resources than those required for direct numerical simulation (DNS).

Those approaches filter the Navier Stokes equations in such a way that part of the total equation is completely computed and the remaining part is modeled. The modeling can take a variety of forms depending of the method used to filtrate the governing equations. Two examples of this are describe in the next paragraphs:

Reynolds Average Navier-Stokes (RANS) is an approach in which the fluctuating part of the turbulence is removed by a temporal filtering process, which introduces a new apparent stresses known as Reynolds stresses. This term is modeled by empirical relationships based on experimental observation or heuristic hypothesis. Time averaging may eliminate important characteristics of the actual turbulent flow, since large and small structures are modeled. However, it is still the cheapest approach to deal with turbulence and its results are considered sufficiently accurate for engineering problems.

Large eddy simulation (LES) is an approach in which the smallest scales of the flow are removed through a spacial filtering operation, and their effect modeled using subgrid scale models. This allows the largest and most important scales of the turbulence to be resolved, while greatly reducing the computational cost incurred by the smallest scales. This method requires greater computational resources than RANS methods, but is far cheaper than DNS. The modeling process is less empiric than those used in RANS, since turbulence at smallest scales is most universal and homogeneous.

These two different approaches are currently used for industrial and academic research. Being the first method the chosen one due to its less computational cost, and the vast research and validation that supports its results.

RANS models can be divided into two broad approaches:

– Boussinesq hypothesis: the Reynolds stress tensor is modeled by and analogy with the Newtonian Constitutive equation for viscous stress, in which the proportionality parameter is the turbulent viscosity. Both algebraic or differential transport equations can be employed for determining the turbulent viscosity. The models available in this approach are often referred to by the number of transport equations associated with the method.

– Reynolds stress model: It is an approach which attempts to actually solve transport equations for the Reynolds stresses. This means introduction of several transport equations for all the Reynolds stresses and hence this approach is much more costly in CPU effort.

According to Ramesh A., [14], one equation and two equation models have presented some success when modeling external flows under adverse pressure gradients at high Reynolds numbers. Two examples of these models are the one equation Spalart Allmaras model and the two equation $k - \omega SST$ model, which are implemented in most of the commercial codes due to its enhanced characteristics for predicting turbulent flows.