Aerodynamic Analysis of F1 in Schools Challenge Car Models

Ryan T. Kell

ZACM 4050 Project, Thesis and Work Experience
University of New South Wales
Australian Defence Force Academy
Canberra, Australia

This report details the background information, project overview and current development of the aerodynamic analysis of models intended for use in the F1 in Schools Challenge. These models were designed in accordance with the 2009 F1 in Schools competition rules, and CFD analysis will be used to evaluate their aerodynamic performance, as well as to identify possible geometric adaptations to improve performance. This report outlines background information on all topics that impact the analysis that will be undertaken. All investigation previously completed on the topic will be explored, as well as an overview of the work procedure and methodology that has been and will be carried out.

Nomenclature

\begin{align*}
    b &= \text{airfoil span} \\
    c &= \text{airfoil chord length} \\
    C_D &= \text{drag coefficient} \\
    d &= \text{distance from leading edge} \\
    D &= \text{total drag force} \\
    D_p &= \text{pressure separation drag} \\
    D_v &= \text{viscous drag} \\
    i &= \text{internal energy} \\
    p &= \text{pressure} \\
    Re &= \text{Reynolds number} \\
    S_M &= \text{momentum source} \\
    V &= \text{velocity} \\
    \Phi &= \text{dissipation function} \\
    \rho &= \text{density} \\
    \tau &= \text{shear stress} \\
    \mu &= \text{dynamic viscosity}
\end{align*}

I. Introduction

The F1 in Schools Challenge is a competition for school students in years 5-12. The challenge requires the students to design and build a model Formula 1 car by using industry standard technologies including Computer Aided Design (CAD), Computational Fluid Dynamics (CFD) and Computer Aided Machining (CAM) products. The vehicle sizes and weights are restricted by a complex set of rules, and the car is powered along a 20m track by a CO₂ canister which, when pierced, provides all thrust. The models used will be those designed by Redline Racing, the 2008 National Champion team from the ACT, for participation in the 2009 World Championships in London, UK. The purpose of this project is to investigate and further develop the aerodynamic features of these models in order to create a model that can achieve the fastest time possible. The steps taken to achieve this will be to run steady state CFD simulations comparing possible models and using wind tunnel tests to validate and sustain confidence in computational results. The final product is to create a fully dynamic CFD simulation of the run using a range of possible models, with experiments carried out to find the time profile of the canister thrust in order to calculate the total 20m run time.
II. Vehicle Aerodynamics

The run time achieved by these models is purely based on the aerodynamic performance of the car. This competition is the only occasion (apart from some race car design) where aerodynamics is the single factor in the shape design of a road vehicle. This section outlines the history of road vehicle aerodynamic design, as well as factors affecting the aerodynamic design of these cars including drag, physical flow characteristics and their impacts on racing and the use of ground effect. Finally, this section looks at the aerodynamic differences between full scale cars and these F1 in Schools models, focusing on Reynolds number and what differences this implies.

A. History

Road vehicles, when viewed aerodynamically, are very complex bluff bodies in ground effect. Their complexity is due to very detailed external geometry, with internal cavities exposed to oncoming flow, and rotating wheels adding to complexity (Hucho, 1993). The flow is fully three dimensional, boundary layers can be turbulent, flow separation occurs, occasionally followed by reattachment. Formation of large streamwise vortices occurs, increasing the already large vehicle wake. The majority of drag is pressure drag, as opposed to aircraft and ships (who both suffer mainly from viscous drag).

To look at the aerodynamics of vehicles today and specifically the F1 model cars that are being looked at, it is of much benefit to explore key developments of vehicles since aerodynamics was first taken into account. Historically car aerodynamics has not been a strong influencing factor in design, due to the number of industrial and commercial requirements including available technology, economic environment, practical design, safety and environmental regulations (Hucho 1993). It was not until after flight technology had made much progress that the idea of applying aerodynamics to road vehicles was explored.

Due to the streamlined air-and-sea craft shapes that had already seen success in practice, the first attempts in modifying car aerodynamics began by using these streamlined shapes. This was of no success for two reasons (Hucho 1993). First, the benefits of aerodynamics were not needed due to minimal engine power and bad roads at the time (which limited speed considerably). Second, that directly transplanting these shapes was not appropriate for the practicality of the vehicle for passenger comfort and component placing (Hucho 1993).

As cars developed over the years, the slow but steady progress of aerodynamic improvement was well documented. Through this development, five main realizations in aerodynamics were noted as decisive events.
1. Recognising that the flow around a body changed when brought close to the ground (see Section II Part D).
2. Truncating the rear of a car body.
3. Introducing the idea of "detail-optimization" into vehicle development (Hucho et al. 1976).
4. Identifying flow characteristics at the rear of a car (Section II Part C).
5. Adding items onto the car body, including underbody air dams, fairings, and wings to passenger cars, trucks, and race cars (Hucho 1993).

Since these achievements, car aerodynamics has been pushed forward due to the ongoing struggle to improve fuel consumption. The two oil crises of the 1970s saw much improvement in car shapes, with Fig. 1 showing the reducing drag coefficients from 0.8 (1920) to 0.28 (1992) over the last century (Hucho 1993).

This historical progression leads into the features used in the design of the F1 in Schools models, most importantly the five notable events listed above.

B. Drag

The nature of the F1 in Schools competition is that drag on the model cars must be as little as possible, because of the limited time (approx 0.3s) that thrust is exerted on the car. After the CO₂ canister is exhausted, the car can only rely on aerodynamics, specifically minimal drag, to maintain as much speed as possible.

Drag is the aerodynamic force exerted on a body that resists movement through the air. The two types of drag found to affect cars are pressure separation drag and viscous drag. Pressure separation drag ($D_p$) is due to
the airflow separating from the boundary surface, creating low pressure along the boundary where separated, and this pressure differential from the opposite side of the body causes a force normal to the boundary. Viscous drag ($D_v$) is the shear force tangential to the boundary (opposing movement of the body) due to the viscous boundary layer effects of the air. The total drag ($D$) experienced by the body is the integral (over all surfaces exposed to the flow) of the pressure separation and viscous drag components, i.e. $D = D_p + D_v$.

Measurement of total drag is fairly easy to obtain by wind tunnel testing, however information on the magnitudes or distributions of the components cannot be found from this. An understanding of the nature of both components can be found by making systematic, parametric changes to the body surfaces; however this can only give understanding into their origin and effect (Hucho 1993). To evaluate $D_v$ quantitatively, we need to know the detailed stress distributions across the entire body surface. This is very difficult to achieve experimentally for simple shapes, even more so on a body as detailed and complex as a car. $D_p$ can be evaluated directly for certain regions, specifically where the surface pressure is almost constant along the surface (after flow separation has occurred for example). This, however, does not give a direct evaluation of the entire pressure drag component. In short, it is very difficult and not practical to try and quantify the separate components of drag for a complex body like a car.

Computational Fluid Dynamics, however, can give us an answer to this problem. CFD codes can specifically output pressure distribution across a surface, making it possible to directly calculate both $D_p$ and $D_v$ (Hucho 1993). CFD will be discussed further in Section II of this report.

Another strong impact on drag is the Reynolds number ($Re$). Reynolds number is a dimensionless parameter that is the most important in terms of low speed incompressible fluid flow (Anderson 2002). The Reynolds number determines boundary layer thickness, at what point the boundary layer becomes turbulent and at what point flow separation occurs, therefore influencing both drag components. The effect of Reynolds number will be looked at in more detail in Section II Part E of this report.

C. Physical Flow Characteristics

Because of the basic similarity in shape between full scale cars and the F1 models, the physical flow will be similar between the two, most notably vortex generation and development into the vehicle wake. In terms of flow characteristics, cars are very detailed and complex, therefore the flow around them is also very detailed and complex. Vehicle drag is mostly caused by pressure separation drag, which results in a turbulent wake behind the vehicle. This wake consists primarily of three elements; the vortices generated across the lateral edges of the roof (explained in detail below), the vortices formed underneath the car body, and the vortices formed along the car side (see Fig. 2, Tsubokura et al. 2009). The strongest of these are the two large streamwise vortices that form along the lateral edges of the roof. The flow over this surface has been closely studied over the last 30 years (Hucho 1993). In modern vehicle design, the slant back angle from the roof down the rear windscreen plays an important factor in reducing drag. The downwash generated from these vortices pushes down the air between them, forcing the flow down the centre of the rear windscreen to reattach, obtaining some pressure recovery and therefore reducing drag (Hucho 1993). If the slant back angle is too much, however, the downwash from these vortices is not enough to reattach the central flow, and the wake is as if the car was truncated from the rear end of the roof.

As previously mentioned, there are many flow factors that contribute to overall pressure drag. For a generic sedan style car, Fig. 2 shows the minor vortices generated by the front pillars, the door mirrors, the front tire house, as well as the large underbody, roof and side vortices all mixing together at the rear end of the vehicle to form the trailing wake. This goes to show that each of these features affect the trailing voices and wake, definitely decreasing the aerodynamic performance of the car (Tsubokra et al. 2009).

D. Racing and Ground Effect

Recent aerodynamic development of road vehicles has predominantly been focussed toward race car performance. This is due to the intense nature of the competition, where aerodynamic design makes a key difference. This design used can, in some instances, be used in design of an F1 model car as race car
aerodynamics are designed to reduce pressure separation drag through shape, which is what we want to achieve in the design of the model cars.

Over the past couple of decades, race car designers have come to the realisation of using an inverted aerofoil to create downforce in order to achieve greater cornering speed. The majority of race car aerodynamic work is on these aerofoils, in the form of front and rear wings. The purpose of these wings is to generate a downforce on the car, which gives the tyres a higher frictional force to oppose sliding, therefore the ability to sustain a higher centrifugal force and a higher cornering speed. This is not what we want to achieve with the model F1 cars. Because the competition track is straight, no cornering traction is required and in fact, traction should be kept to a minimum to try to eliminate tyre friction drag. In order to do this, we could use an (uninverted) aerofoil shape for the front and rear wings to minimise force on the wheels.

The front wing on one of these cars is in close proximity to the ground and is affected by a phenomenon called ground effect. When an inverted aerofoil-shaped body is close to the ground, a venturi tunnel is created between the underside of the body and the ground. Due to the venturi effect, the flow underneath the body will increase speed, therefore decreasing in pressure and creating a suction force on the body. A decreased distance to the ground results in an even higher flow velocity, subsequently increasing the suction force (or downforce) and the associated pressure recovery (Martinez 2004). This ground effect is the same as is experienced by lifting surfaces and as expected, it amplifies lift (Katz 2006). Because of this, we can use ground effect to increase the lift generated by the front wing on the F1 models, in order to minimise tyre friction drag as previously mentioned. The other use of ground effect for these models could be to alter the car body into more of a lifting shape, and have the body of the car become the aerofoil and use the front and rear wings for stabilisation. Ground effect would of course help this as it would amplify the lifting force, therefore giving the possibility of having the car fly throughout most of the race.

E. Small Vehicle Effects

There are a few major differences between the F1 in Schools models and a full scale car. The biggest (and probably the most obvious) is size. Size, however, plays a major factor in the aerodynamics of the vehicle. The most important contributor to aerodynamics is the Reynolds number. This non-dimensional parameter determines characteristics of the flow (Anderson 2002). The Reynolds number is found by the following equation:

\[ \text{Re} = \frac{\rho V d}{\mu} \]  

Using these standard sea level atmospheric values; \( \rho = 1.225 \text{ kg/m}^3 \) and \( \mu = 1.7894 \times 10^{-5} \text{ kg/ms} \); we can evaluate the Reynolds number for a normal size car of velocity \( V = 36 \text{ m/s (~100km/h)} \), length \( d = 5 \text{m} \). This gives us a Reynolds number of \( \text{Re} = 1.23 \times 10^7 \). Doing the same for a model F1 car with maximum velocity \( V = 25 \text{ m/s} \), length \( d = 0.18 \text{m} \) we obtain a Reynolds number of \( \text{Re} = 3.08 \times 10^5 \).

These Reynolds numbers are almost two orders of magnitude apart, which could mean vast differences in the flow characteristics around the vehicle. The boundary layer thickness and turbulence transition as well as the flow separation points are dependent on the Reynolds number, therefore determining the most dominant source of drag. The boundary layer creates viscous drag through shear stress onto the surface, and separation creates pressure drag (Anderson 2002). On a full scale car, the pressure drag is stronger than viscous due to the high Reynolds number and complex shape (Hucho, 1993). On these F1 models, however, due to the much smaller Reynolds number and less restricted and smoother shape, the pressure drag would hopefully be much less, with the viscous drag being of more consequence. If viscous stress does dominate, then total surface area of the car should be minimised (within guidelines). If pressure separation drag dominates however, then the least frontal area and maximum use of streamlined features would be optimum.

In order to calculate a value for flat plate viscous drag approximately the same size as one of the F1 models, we must use the drag formulas found from Anderson (2002) Chap. 4. Before this, we need to find the transition point that the boundary layer becomes turbulent. By rearranging Eq. (1) to solve for \( d \), and given that the transition Reynolds number is \( 5 \times 10^5 \) (Anderson 2002); we find that the transition distance is 0.31m. This distance is further than the entire chord of the flat plate, therefore we can conclude that the boundary layer is fully laminar for the entire length of the plate. The formulae are for the Blasius solution for the drag on a flat plate purely due to skin friction of a fully laminar boundary layer (Anderson 2002). The drag on the plate can be found by the following formula.

\[ D_T = 2b \int_0^b \tau \, dd \]

where \( \tau = \left( \frac{\mu}{\delta y}, \right) \).

Initial Thesis Report 2009, ACME, UNSW@ADFA
which becomes \( \tau = 0.332 \sqrt{\frac{\rho u^2}{\mu}} \) for a flat plate;

\[ D_r = 1.382b \sqrt{cp\mu V^3} \]

This evaluates the friction drag force for a flat plate to be approximately 0.01N for a flat plate of chord 0.18m and width 0.06m (the approximate size of one of the model F1 cars). Now if we assume a pressure drag coefficient of 0.1 (less than half the total drag coefficient of the best cars, and far larger than of some aircraft), we can estimate the pressure drag of some shape with the same chord and span as the flat plate using the following equation (Anderson 2002).

\[ D_p = C_{dp} \times \frac{1}{2} \rho V^2 bc \]  \hspace{1cm} (4)

This gives an estimated pressure drag of 0.41N, substantially larger than the viscous drag. If we change the assumed pressure drag coefficient to 0.01, the total drag coefficient of streamlined aircraft, the estimated pressure drag is still four times higher than the viscous drag for a flat plate. From this we can conclude that even for the vastly different Reynolds numbers, Pressure separation drag is the dominant source of drag on these model cars and therefore all effort must be put into minimising this type of drag.

III. Computational Fluid Dynamics

Computational Fluid Dynamics is a powerful engineering tool that is being used in an ever-widening range of applications. Initially CFD was developed in order to try to give meteorological forecast, after which it was further developed within the aerospace sector (Martinez 2004). This application of CFD has brought with it the development of vastly improved designs for any part of an aircraft regarding the fluid flow around it. This improvement would have been almost impossible without use of CFD, as so much information is made available that was previously too complex to find experimentally or theoretically. This section will explore the fundamental theory behind CFD, followed by its many applications, as well as the accuracy of CFD results and validation methods that can be used to instill confidence in these results.

A. Fundamental Theory

Computational Fluid Dynamics uses the Navier-Stokes equations of fluid mechanics in order to compute any parameter of any fluid at any point in a given flow field (Glowinski et al. 1992). These equations from Versteeg et al. (2007) are shown below. Where \( u, v \) and \( w \) are velocities in the \( x, y \) and \( z \) directions respectively

Continuity

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \]  \hspace{1cm} (5)

\( x \)-momentum

\[ \frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho uV) = -\frac{\partial p}{\partial x} + \nabla \cdot (\mu \nabla u) + S_{mx} \]  \hspace{1cm} (6)

\( y \)-momentum

\[ \frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho vV) = -\frac{\partial p}{\partial y} \]  \hspace{1cm} (7)

\( z \)-momentum

\[ \frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho wV) = \frac{\partial p}{\partial z} + \nabla \cdot (\mu \nabla w) + S_{mz} \]  \hspace{1cm} (8)

Energy

\[ \frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho TV) = -p \nabla \cdot V) + \nabla \cdot (k \nabla T) + \Phi + S_i \]  \hspace{1cm} (9)

Equations of state

\[ p = p(\rho, T) \]  \hspace{1cm} (10)

These equations can be time-discretized to give the Reynolds-Averaged Navier-Stokes (RANS) equations. These equations can be found in Versteeg et al. (2007) Chap 3. This is the basis upon which most engineering flow study is completed. This method focuses on the mean flow and the effects of turbulence on mean flow properties (Versteeg et al. 2007). The reason RANS equations are used so widely is that the computing power required to give moderately accurate flow results is fairly low. Another method used to solve the Navier-Stokes equations is called Large Eddy Simulation (LES). This method uses a different form of turbulence equations that tracks the behaviour of larger eddies by filtering the unsteady Navier-Stokes equations. The computing demands to solve flow using this method are very high, however due to ever-increasing computer power this method is beginning to be effectively used, albeit at a low Reynolds number (see Tsubokura et al. 2009).

On looking at these methods, there are several questions on which method should be used. One of the first questions is how close the equations used can truly simulate the actual physical conditions. Once the equations are selected, the next question is how well different algorithms approximate these equations at the given conditions. Finally, which solution is affordable from the computational power point of view (Katz 2006).

The CFD program Fluent, which will be used extensively throughout this project, uses the RANS equations and as such will give reasonable flow accuracy, perhaps with some difficulty in detailed unsteady flow characteristics (Tsubokura et al. 2009).
B. Application

Due to the extensive amount of literature available publicly on CFD, this section will concentrate on the CFD used in the car design and race car industry. Until recently, CFD was not used extensively for race car aerodynamic design for two main reasons. Firstly, the flow was too complex, and neither the advanced numerical software nor computational power were available. Secondly, CFD had not been proven to give reliable and useful information (Sawley et al. 1997), hence much reluctance to use it as a design tool.

Recently, however, when the above two problems had been rectified, in motorsport the application of CFD has become totally necessary for gaining those tenths of second that distinguish the first from the second (Martinez 2004). The CFD work used on race cars is mostly focused on the front and rear wings, as well as the body undershape, creating a required amount of downforce (see Katz 2006, Martinez 2004). There has also been much work done over a long time on visualization of wake effects of a car. Tsubokura et al. (2009) have performed a LES on a standard shape sedan car (Fig. 2), giving results almost identical to experimentation, which give clear visualization of vortex generation and therefore wake effects. Krajnovic and Davidson (2002) also carried out some similar LES on simpler shapes also giving previously unattainable visualization of unsteady flow. This visualization and made possible by CFD is a very powerful tool for shape design, which is the main purpose for this project.

The process which must be carried out in order to create a successful CFD solution includes a CAD representation of the body under study, a mesh generated to represent the flow field, and finally a solving program that will solve all flow parameters throughout the grid. The programs that will be used to complete the three parts of this process will be CATIA, GAMBIT and Fluent respectively.

1. CAD
   CATIA is one of the leading CAD programs to date. It is a solid modelling program which has good interconnectivity with many grid generation systems, and avoids the interface problems outlined by Kellar et al. (1999) regarding the methods previously used by CAD programs. It has an easy to use system of part design, and then integrating parts together into an assembly. Figure 3 is the basic model of an F1 in Schools car, created by the author.

2. Mesh Generation
   As mentioned, the mesh generation will be carried out in the meshing program GAMBIT. This program is useful due to its easy interface in importing from CATIA, and easily exporting a mesh into Fluent. GAMBIT also has the ability to halve the model down the centre so that computation of only one half of the flow field is possible in order to reduce computation time.

   Mesh generation is a very important part of the CFD process. The fineness or coarseness of a grid determines the accuracy of solution as well as rate of convergence. The finer the grid, the more accurate the result, however the computational time increases significantly with a finer grid.

   There is a variety of meshing methods available including Cartesian, structured, unstructured, and hybrid grids. These different types of grids give different results. For example to calculate accurate boundary layer flow, structured grids are the most useful with rectangular cells, which can be elongated to fit the profile of the shape, but compressed in a normal direction in order to maintain grid fineness to obtain accurate results for boundary layer flow. The weakness of structured grids, however, is that rectangular cells do not easily fit well around complex shapes. To work around both of these problems, a hybrid grid may be used. Hybrid grids of tetrahedral elements used in combination with rectangular elements near surfaces provide a significant reduction in meshing time, and are flexible in dealing with complex geometries (Martinez 2004).

   Another meshing method is called adaptive gridding. This is a method that adapts the grid around steep pressure gradients during simulation, therefore maintaining a high grid resolution where it is most required throughout the flow field.

3. Flow Solving
   The CFD solver used will be Fluent, which has easy interface with GAMBIT and gives fairly accurate results for low speed incompressible flow when used correctly. The problem with CFD is that it can be a great benefit in understanding flow motion; however it can represent a real lack of information if used inadequately, giving appealing pictures but wrong answers. Using CFD requires knowledge and experience of its user. If not, it will lead to incorrect results and wrong flow visualization (Martinez 2004). This is why setting up the solution
is so important. Boundary conditions, as well as flow conditions and properties must be established correctly in order to obtain the most correct results.

**C. Accuracy and Validation**

Accuracy and validation of CFD has been the biggest deterrent from widespread use until recent times. There will always be uncertainty in any CFD solution due to the discretization process. This uncertainty represents the difference between the discretized algorithms used in the solver and the true partial differential equations. As well as this, there is always the error of using unconverged results. This occurs because of an ineffective grid, or not setting a low enough residual minimum. To avoid this, a solution parameter can be observed as the solver works through iterations. Once the parameter stays constant for a long enough time, the solution has converged.

There is a process to validating the use of CFD results that Martinez (2004) describes as necessary for a proper validation analysis. The steps to this process are as follows; examine iterative convergence, examine consistency, examine grid convergence, examine temporal convergence, compare CFD results to experimental data, and examine any model uncertainties. This validation process will be undertaken as results are obtained further into the project.

**IV. Project Overview**

**A. Project Intent**

As the Redline Racing team from the ACT will be travelling to London, UK for the F1 in Schools World Championships, their final car designs will need to be analysed aerodynamically through use of CFD as well as wind tunnel testing. The purpose of this project is to undertake the analysis required, while collaborating with Redline Racing in order to improve the shape design for better aerodynamic performance where possible.

**B. Methodology**

The process required to complete this project is more than just CFD runs and some wind tunnel testing. It involves many different stages, some that can be worked on concurrently, and some that rely on others. The required tasks are as follows.

1. **Project Management Planning**

   The first step is to implement a comprehensive project management plan based upon the client brief agreed upon by the author as project manager and the project supervisors as clients. This plan outlines all tasks to be completed, including estimated dates of completion in order to have the project completed in a timely manner. This plan includes a work breakdown structure, Gantt chart (project schedule), and a milestone chart, all of which can be found in appendix A.

2. **Literature Review**

   After the project management documentation is complete, a review of all relevant information available must be conducted. This review includes finding past work others have done on the topic, critically analysing it with respect to whatever differences there is between that work and the work that is to be completed. The purpose of the review is to explore the fundamentals of road vehicle aerodynamics, as well as to determine whether CFD is the most effective tool to use for the aerodynamic analysis, including results validation.

3. **CAD Modelling**

   The final CAD models will be provided by Redline Racing, however any adaptations and some initial car body shapes will have to be constructed by the author in CATIA. To achieve familiarity with this program, tutorial work will be completed through enrolment in the course *Integrated Design with Vehicle Applications*.

4. **CFD Solution**

   These CAD models will be meshed through GAMBIT and run in Fluent, making sure that the conditions stated under Section III of this report are met. Familiarity with GAMBIT and Fluent, as well as the methodology of setting up a CFD analysis will be obtained through participation in the course *Engineering Applications of CFD*, also by completing tutorials of both programs in allocated thesis work times.

   In order to obtain accurate results, the boundary conditions applied must be closely looked at. Barber (2006) explains the four possibilities of simulating ground in CFD solutions. The image method is the first of these. This method requires setting the lower boundary to be a symmetry condition, in order to obtain no flow through the plane of symmetry. This method, however, would not show any boundary layer across the ground where accelerated flow will occur underneath the car. The second method is to set the lower boundary with a slip condition. This means that there will be exactly zero shear stress at the boundary. This method, however, gives exactly the same problem as the image method; it will not show boundary layer across the ground. The third method uses a fixed ground boundary. This will yield very incorrect results as it will show a boundary layer along the ground in free stream flow, which will not happen in the real situation. The last of the methods is to set the ground plane moving at the inflow velocity. This is the only physically correct solution, and as such it is
the one that will be used for the aerodynamic evaluation CFD runs. This method has been used extensively in the past, by Martinez (2004), Katz (2006), Tsubokura et al. (2009), as well as many others.

There will be two types of CFD runs; the first will be a steady state run with no ground movement and fixed wheels. This run will be used to compare with the wind tunnel results obtained. The reason for a fixed ground and wheels are explained below. The second type of run will be steady state, however with the ground moving and wheels rolling. These runs will be of different possible models to evaluate the aerodynamic performance of each. These will give us the ability to view the true flow field around the model as if it is in the middle of a race.

Because the CFD program Fluent solves the RANS equations, the results obtained will be slightly time averaged. This is the same type of results as seen in Fig. 4, similar results that Katz et al. (1998), Martinez (2004), Kellar et al. (1999) and Barber (2006) have obtained, which give good overall flow visualisation. These results, however, do not show well the instantaneous unsteady flow characteristics as can be seen in Fig. 5 and Fig. 2 that were found by Krajnovic and Davidson (2002) and Tsubokura et al. (2009) respectively by using LES methods. The RANS solutions found by Fluent will however be adequate for the study of vehicle wake that will be used for the remainder of this project.

5. CFD Verification

Verification is necessary in order to have confidence in results obtained. This verification must be done so that the CFD problem is exactly the same as the physical test (Martinez 2004). The verification method used will be simple wind tunnel experiments to evaluate a drag force, and compare this with the CFD results. Hucho (1993) explains the problems encountered when wind tunnel testing a road vehicle, mainly that to obtain a true aerodynamic simulation of a car moving along a road, with wheels moving, there cannot be any boundary layer across the road. He suggests a variety of methods to overcome this problem including boundary layer suction, introducing an accelerated flow across the ground to minimize boundary layer, as well as using a conveyor system to simulate moving ground. The wind tunnel experiments that will be used will be with a static floor, and with fixed wheels. This is because the complexity of using any of the aforementioned systems for boundary layer reduction are too difficult and the wind tunnel validation can still occur accurately if the CFD model experiences the same conditions as the wind tunnel (Lewis et al. 2003).

6. Design Evaluation

From the flow field visualised by the second set of CFD results, we will be able to see which models perform better aerodynamically, and what geometrical features make this so. After identifying successful geometrical shapes, a final car design will be developed by Redline Racing in collaboration with the author, making use of all information gathered by CFD methods.

7. Thrust Canister Experimentation

In creating the fully dynamic simulation of one of the model F1 car runs, the thrust profile over time of the CO₂ canisters must be found so that a correct input force can be established to gain accurate results. Experimental methods will be used to obtain this profile. This experiment will be done using a number of canisters, and the mean values over time will be taken as the thrust profile. The detailed experimental method and procedure will be developed in the near future.

Figure 4. RANS solution streamlines around an Indy-type race car. Reproduced from Katz et al. (1998).

Figure 5. LES of flow around a simplified bus shape. Reproduced from Krajnovic and Davidson (2002).
8. **Dynamic Simulation**

The final outcome of this project is to develop a full dynamic simulation of any given F1 model run. This requires the unsteady CFD analysis of the F1 model over the 20m track distance. This will be done by utilising a moving mesh in Fluent. Source code must be written for use as a user-defined function in Fluent that varies the rate at which the mesh is moving every time step, through calculation of the total force on the car through canister thrust at a given time, as well as the drag calculated by Fluent. So far it seems that using a moving mesh will not make it possible to run this simulation with rolling wheels. This will give incorrect results if not rectified, however the magnitude of this error cannot yet be estimated.

**C. Project Status to Date**

Working through the project methodology stated above, the client brief has been completed and can be found in appendix A. Project management planning has also been completed and documentation can be found in appendix B. A preliminary literature review as outlined above has been conducted and the findings of this review are documented throughout this report. A final literature review will be conducted for inclusion in the final thesis report over the next few months.

Some CAD modelling work has been done in that a basic F1 in Schools body has been constructed by the author (see Fig. 3). This model will be used in preliminary CFD runs while familiarisation of the meshing process takes place.

Most project work thus far has been focussed on development of the source code that will be used as a user-defined function in Fluent. Initially, code was written in MATLAB to simulate the dynamic motion of an F1 model, guessing an arbitrary thrust profile. This code can be found in appendix C. The results of this code give an accurate representation of the velocity and position profile as found by previous accelerometer test results, and so this same method was used in writing the source code for the user-defined function. Writing this source code required the author to learn how to use the C programming language, as well as the complex Fluent specific functions. The source code written can be found at appendix D. This code is written for a two dimensional case, however adaptation to suit a three dimensional case should prove to be minor. At the current stage, the code development has seen significant progress, however it has not been tested as of yet. Testing of this code will take place soon after the May study recess, with development into a 3-D case following soon after.

GAMBIT and Fluent tutorials have been undertaken as part of the course *Engineering Applications of CFD*, as well as Fluent familiarisation work during allocated thesis work times. This has been completed and will be continued so that effective work can be conducted in future, when models are available for testing.

**D. Further Development**

Once the F1 models are completed by Redline Racing, steady state CFD testing can commence, evaluating the aerodynamic performance of each different design, and developing a final most aerodynamically efficient shape. While this is being carried out, preparation work can be done on the wind tunnel testing. Development of the function code will include the adaptation to suit the 3-D case, and once the code is tested and found successful the dynamic simulation can occur, calculating a time taken for the full competition run. By using this dynamic simulation we can compare times of any number of different model shapes to see which shape will be the best to use at the 2009 F1 in Schools World Championships in September. Due to the departure of Redline Racing to London in for the world championships, all computational and design evaluation work should be completed by the end of August, giving ample time to complete the final thesis report.

**V. Summary**

This report has outlined all background aspects of this thesis project, as well as its current progression and future development. Background information on vehicle aerodynamics and CFD have been explored to determine the most effective way to carry out the project, followed by a detailed outline on tasks to be completed and the differences expected from the study others have done in the past. As further progress is made on the project, all milestones and achievements will be documented in construction of the final thesis report.
References


Appendices

A. Client brief
B. Project Management Documentation
   I. Work Breakdown Structure
   II. Gantt Chart/Project Timeline
   III. Milestone Chart
C. MATLAB Code for Vehicle Motion Simulation
D. User-Defined Function Source Code