CFD Study on the Diffuser of a Formula 3 Racecar

Kevin M. Peddie, Luis F. Gonzalez
School of Aerospace, Mechanical
and Mechatronic Engineering
University of Sydney

ABSTRACT

The primary goal of the paper is to examine the aerodynamic behaviour and performance of the diffuser of a Formula 3 racecar within a moving ground plane for critical case situations. The purpose of this investigation was:

- To understand the governing factors and physics which are associated with an automotive diffuser within ground proximity.
- To examine the effects and association between ride height and ramp angle variance.
- To analyse the flow performance associated with the central diffuser section both independently and also as part of the complete diffuser geometry.
- To generate and analyse alternative designs based on the current design flow performance and also alternative category designs.

The paper focuses on the physics primarily associated with the Formula 3 diffuser geometry and to understand how the design of the diffuser and effects of ground clearance on the downforce and drag performance.

Due to the complexity of the complete Formula 3 diffuser, affected by upstream geometry such as the splitter and sidepod design, road surface roughness, downstream obstacles (trailing car), rear wheel location, top surface geometry, ride height and car speed. Therefore a simplified bluff body configuration was considered.

The diffuser study was broken into three stages; a simplified two-dimensional study to provide an understanding of the trend between ride height and associated ramp angle for maximum downforce and reduced drag performance. The central diffuser was considered independently to analyse the current designs suitability for the desired ride height as was identified during cornering. Finally, the complete diffuser geometry was analysed to identify the effects of the central diffuser with the inclusion of the complete diffuser geometry.

The results show that the current diffusers design that is used on the 2004 Dallara Formula 3 model is not optimal for the desired ride heights used by current Australian Formula 3 racing teams. Through CFD testing, it was found that new created designs based on the current pressure performance and within regulation restrictions were able to produce more favourable downforce and drag results.

INTRODUCTION

The world of motorsport is generally regarded as an area of expertise highly reliant on the mechanical ability of the engine performance, the driver's ability, experience and the tactical moves decided by team management. Many people outside the motorsport industry disregard the high importance of aerodynamics on a racecars' performance. The main aspect of both Formula 1 and Formula 3 racecars in reducing lap times is to improve engine output performance, increase downforce to help improve possible cornering speeds and reduce drag forces associated with the racecar.

The main avenues for aerodynamic research in the motorsport industry are wind tunnel testing and Computational Fluid Dynamics (CFD). Increasing concern has been directed at the high costs associated with the running and development of a racing team, and with large costs associated with creating models and the running of the wind tunnel itself, a larger trend has been towards CFD design work with wind tunnel testing used for data verification.

The Formula 3 industry is governed by strict rules and regulations as stated by the Federation Internationale De L'Automobile (FIA) [1], this regulatory body is set-up in order to maintain a standard of safe and fair playing field for the races that are competed in. The Australian Formula 3 Championships for which this study was conducted under, is bound by the rules as stated by Formula 3 Australia, this is similar to the FIA regulations with the exception for classification of class, divisions, safety procedures and general competition procedures. The underbody of the car is no exception to these regulations. Dimensioning of the underbody work is not only restricted by the mechanical components underneath the car such as the differential, but also dimensioning limitations away from the car. A geometric representation of these parameters can be seen in Appendix (1-A) from which the primary restraints as stipulated in the FIA regulations are as follows:

- Width of the bodywork behind the centreline of the rear wheels must not exceed 900mm in width (3.6.1) No component shall be more than 500mm behind the centreline of the rear wheels, with the exception of structure required by article 15.5.1.
- All components lying on the ground reference plane must be symmetric about the longitudinal centreline of the car.
- The diffuser must lie above the lowest ground reference plane, which must be timber structure.
- The step plane must be 50mm above the reference plane.

There are several restraints placed on the diffuser geometry, however regulations help to forward diffuser technology.

The study was originally designed to investigate the performance of the 2004 Dallara Formula 3 diffuser obtain a more detailed understanding of the design factors and features used to optimise the underbody design. A simplified bluff body design was set-up similar to previous studies, to enable ease for comparison of results. To verify the accuracy of solutions in CFD, this was performed in two-dimension.

The three-dimensional studies focused on analysing the suitability of the current design for the design parameters stated by several racing teams as to their measured ride height during cornering, deemed to be the most critical case. The study consisted of investigating the central diffuser as a sole entity and then its effect and the effect of it on the diffusing section from the sidepod.

FORMULA 3 DIFFUSER

The role of the diffuser is to expand the flow from underneath the car to the rear, inturn produce a pressure potential, which will accelerate the flow underneath the car resulting in reduced pressure and as such, a desired increased downforce generation.

The Formula 3 diffuser consists of three main channel sections running underneath the car as seen in Figure 1, from the regulations it may also be noted the underbody ground planes can be based off two levels, the centre section and the two side channels where the sidepods are located above.

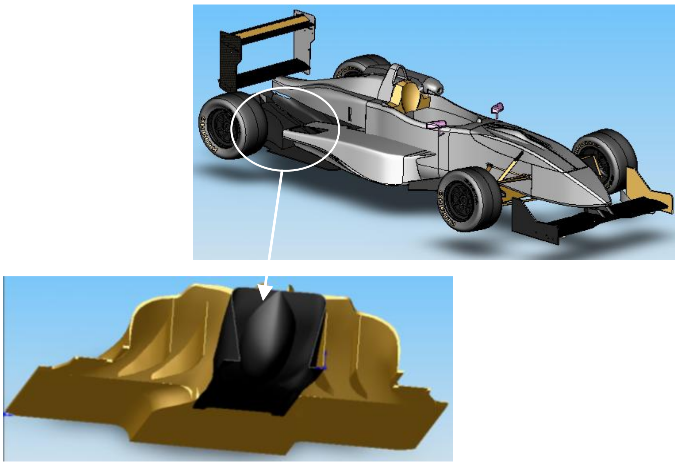

Figure 1: Formula 3 Diffuser

One of the major aspects of the design of the diffuser is the ramp angle or curvature of the diffuser and length. Han [2] investigated the effects of the flow over the rear end of a car using a simple rectangular prism bluff body for comparison and looked into all aspects of the rear end, such as boat angle, ramp angle and backlight angle. He concluded that by comparing each angle separately, that the ideal ramp angle for the diffuser should be 17.8 degrees.

Figure 2 shows the key aspects of the diffuser; ride height, h_1, outlet height, h_2, diffuser length, N, and ramp angle θ.

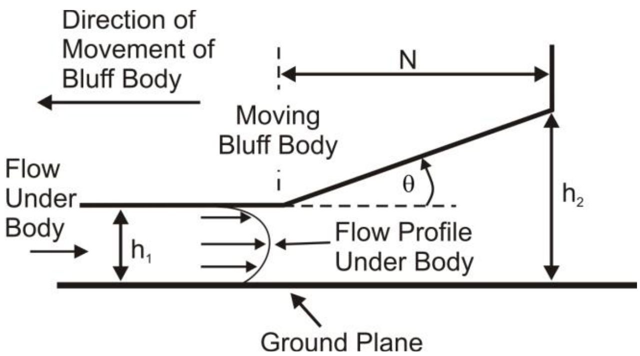

Figure 2: Diffuser Geometry

Cooper [3] has performed two separate investigations on the performance and optimisation of the diffuser of an automotive underbody. Cooper looked primarily into the flow and performance (lift and drag) values produced by the diffuser at varying ride heights and ramp angles for two different diffuser lengths. The simulation was done with the use of a wind tunnel employing a moving belt to simulate the moving ground plane. This design made use of a simplified bluff body design, which has been seen to be an ideal comparison benchmark. The results concluded that from the wind tunnel a ramp angle of 9.64 degrees generated the most downforce, while the CFD results showed the optimal value to lie near 13 degrees which is similar to that used by Indy Light cars.

It was found by Sovran [4] during testing at the Pinninfarina workshop that the diffuser actually acted as a pump to generate downforce over the underbody flow path. This was not deemed to be the only identifiable fluid-mechanical mechanism affecting the flow path around the diffuser. The three main aspects were; '*ground effect*', '*underbody upsweep*' and '*diffuser pumping*'. [4]

Ground Effect plays a role when an object is used in the vicinity of a moving ground plane. Flow asymmetry is developed from the flow accelerating as it travels underneath the body due to ground constraint as a result the static pressure underneath the body is reduced which provides the resulting downforce. This would otherwise increase indefinitely with increased ground proximity if not for that real flows are inviscid. Fluid viscosity is of minimal concern for larger ride heights, however this becomes a dominating factor with reduced ride height due to the restricted area underneath the body for which the flow to travel.

Underbody Upsweep refers to the upsweep of the upsweep at the rear. This is typically cambered in shape, similar to the upper surface of an airfoil. Due to the direction of this camber, a resulting downward directed lift force will result during flow interaction.

Diffuser Pumping refers to the increasing cross-sectional area over the diffuser length, which can be used to increase the flow rate through a system via pressure potential. As the ratio of the inlet to outlet area becomes increasingly greater then unity, this generates greater pressure recovery that, due to the base pressure remaining constant will increasingly depress the base pressure at the inlet. The diffuser acts to reduce the underbody pressure due to the expansion resulting in increased flow rate under the body. This increase results in further decrease in underbody pressure, which produces the 'pumping down' or downforce generated. At very low ride heights, the flow rate under the body is reduced so downforce generated is also restricted.

PERFORMANCE PARAMETERS

There are several key elements of a diffuser geometry, which ascertain the performance that will result. The pressure recovery coefficient ($\overline{C}_p$) is one of these, which relates the pressure at the inlet and outlet of the diffuser section.

$$\overline{C}_p = \frac{(p_2 - p_1)}{\frac{1}{2}\rho \overline{U}_1^2} \qquad (1)$$

where $\overline{U}_1$ is the area averaged inlet velocity, p_1 is the diffuser inlet static pressure and p_2 the static pressure at the diffuser outlet plane. From idealised full expansion of 1-D flow assuming no losses the pressure coefficient is found by:

$$C_{pi} = 1 - \frac{1}{AR^2} \qquad (2)$$

where the area ratio (AR) is a relation between the inlet and outlet heights of the diffuser section. The area ratio for an asymmetric body can therefore be stated as:

$$AR = 1 + \left(\frac{N}{h_1}\right)\tan\theta \qquad (3)$$

where N is the diffuser length, h_1 the ride height and θ the diffuser ramp angle. This shows the relation between geometric parameters of the diffuser and enables a realisation that vehicles with a greater ride height will posses a smaller area ratio for a given diffuser ramp angle compared to that of a lower ride height.

DOWNFORCE MECHANISM

Lift coefficient values are the primary outcome result that will govern the performance improvement of the diffuser along with drag coefficient. Following the expressions for both lift coefficient of the bluff body and the streamwise-distance-averaged, mean-effective pressure coefficients are:

$$C_L = \frac{L}{H}\left[\overline{\overline{C}}_{pl} - \overline{\overline{C}}_{pu}\right] \qquad (4)$$

$$\overline{\overline{C}}_{pi} \equiv \frac{1}{x_i}\int_0^{x_i} C_p(x)dx \qquad (5)$$

where *l* and *u* are the lower and upper surfaces, and L and H the length and height of the body. Therefore it can be noted that the difference between the upper and lower surface pressures is the main concern in which to increase downforce on the body. For all tests cases then, it will be attempted to maintain upper surface pressure values with only variances to lower surface pressure by means of underbody geometry and clearance variance. Since that downforce is denoted as a negative lift coefficient, it is desired that $\overline{\overline{C}}_{pl}$ be made as negative as possible. Furthermore, $\overline{\overline{C}}_{pl}$ can be broken up into two components:

$$\overline{\overline{C}}_{pl} = \left(1 - \frac{N}{L}\right)\overline{\overline{C}}_{pf} + \frac{N}{L}\overline{\overline{C}}_{pd} \qquad (6)$$

where *f* refers to the underbody surface upstream of the diffuser (including frontal radius) and *d* refers to the diffuser length N.

It has been noted that the force behind downforce generation with the diffuser is the pressure recovery performance. The mean effective pressure coefficient can be determined from the fact that the axial pressure distribution in a subsonic diffuser has a characteristic non-linear shape that can be established. The equation for the mean effective pressure ($\overline{C}_{pd}$) for asymmetric, plane-walled, underbody diffusers in viscid, incompressible, one-dimensional flow to be [3]:

$$\overline{C}_{pd} = \frac{(C_{p2} - C_{p1})}{(1 - C_{p1})} \qquad (7)$$

$$C_{p2} = \left(\frac{p_2 - p_\infty}{q_\infty} \right) \qquad (8)$$

where C_{p2} is the pressure coefficient at the diffuser exit.

$$\overline{C}_{p} = \frac{(C_{p2} - C_{p1})}{(1 - C_{p1})} \qquad (9)$$

and $\overline{C}_p$ is the overall pressure recovery coefficient.

It can be deemed that equation (7) is suitable in determining the non-linear behaviour of the pressure distribution in underbody diffusers.

TWO-DIMENSIONAL PERFORMANCE

One way in which diffuser performance can be investigated is through an idealised two-dimensional study as conducted by Sovran & Klomp [4]. Their research mapped the diffuser performance of the pressure recovery of a symmetrical two-dimensional, plane wall, single expansion diffuser as a function of area ratio and non-dimensional diffuser length. A constant diffuser angle can be seen as a straight line radiating from the origin.

An optimal characteristic line can be seen running through the curvature of the contour lines of pressure recovery, this was stated to be the maximum pressure recovery for a given non-dimensional diffuser length and was termed by Sovran & Klomp to be the C_p^* line. Another issue that was investigated by Sovran & Klomp was that of the diffuser inlet velocity profile. It was seen that a non-uniform velocity profile would become largely distorted with the effects of the positive pressure gradient inside the diffuser. A distorted flow profile would block part of the flow cross-section resulting in a reduction in the area ratio and consequently a pressure rise.

CFD THEORY & APPLICATION

There are different methods available to solve more complex aerodynamic problems, the two main prominent methods are the panel method and the use of CFD.

The CFD program used for this work is Fluent™, with the mesh generation program, Gambit™. Fluent uses a controlled volume approach applied to the Reynolds-Averaged Navier-Stokes equations to employ simultaneous analysis of temperature, pressure, velocity and density of the designated volume.

The nature of the flow that is present in and around a diffuser section is turbulent. As such, the mean quantities of the flow are of concern with little reliance on individual particle movement. [5] Numerous studies have been conducted on the performance of these models as turbulent flow solvers. Aroussi [6] stated that through the study of flow in a T-shaped cavity, that the κ-ε model showed a better consistency than a constant viscosity model, however in complex recirculating flows with large pressure fluctuations, the model requires further development to include pressure gradient within wall functions. It is also stated that the variance of the values for ε showed no significant influence on the dissipation rate of the kinetic turbulent energy of the flow structure.

Akanni [7] showed the attraction that CFD offered to the Formula 1 enables aerodynamic research over numerous amounts of design phases without the costly process of physically building models. This also produced considerable time saving to enable more research options to be looked into to help improve racecar performance during each race season. The report looked at the design process that the CFD industry is able to do and follows. CAD modelling is one of the most commonly used forms of design and development in most industries since it enables a visual representation of the structure in mind. CAD models can be imported into CFD software where grid formation and boundary conditions can be set.

DESIGN METHODOLOGY

This research will be used to develop an understanding of the diffuser performance and how these results can be used to help guide future design ideas. Firstly, an understanding of the flow effects surrounding the diffuser is required, this will primarily be focused on the effects on the near moving ground plane surface. It was decided that a detailed study of a two-dimensional diffuser would enable an understanding as to the effects of ramp angle variation and also the effects of ride height to the performance of a diffuser in proximity of a moving ground plane. The 2004 Dallara Formula 3 diffuser will be analysed to observe its performance with varying ride height conditions to determine the effectiveness and suitability of the current design for the ride height settings employed by current Formula 3 racing teams. The study was broken into two stages, the first stage will analyse the central section of the diffuser only and test some current overseas designs from other racing categories and created designs. The second stage involves analysing the complete Dallara Formula 3 diffuser to observe any effects that this may have on the performance of the central section.

<u>Bluff Body Configuration</u> – During standard operation, the diffuser will only interact with the airflow on the lower surface with only some occurring on the sides. This then presents a problem with defining the flow over the front

and upper surfaces of the diffuser. For both the two-dimensional and three-dimensional cases, it was decided that the ideal method to overcome this would be through the use of a bluff body design. The bluff body would consist of a clean surface, primarily an extension of the geometry of the diffuser forwards in the direction of the on going fluid flow. To prevent any discrepancies between solutions, this shape will be maintained for each design as a control for results obtained. The two-dimensional representation is shown in Figure 4 and the three-dimensional is shown in Figure 5.

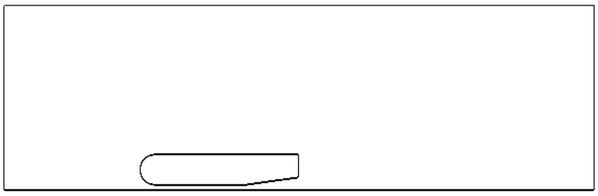

Figure 3: Two-Dimensional Bluff Body

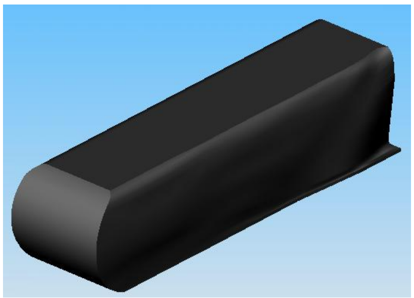

Figure 4: Three-Dimensional Bluff Body

Operating Parameters – The conditions that are encountered over the course of a lap and race tracks will vary considerably, due to road camber altering ride heights, surface conditions will determine turbulence effects created by road roughness, speed and whether accelerating or braking which will affects pitch and finally initial car set-up conditions. All of these effects will alter the performance effects of the diffuser since the airflow that travels under the diffuser due to inlet height and flow structure is not consistent. Therefore what component of a racing lap would the diffuser's performance be of greatest benefit needs to be determined. After discussion with racing team officials, it was decided that during cornering would be most beneficial for improved downforce and as such an averaged cornering speed was obtained to be at 120km/h. The ride height range that the Formula 3 car would run over was also required and based on information from the same source, a minimum ride height of 22mm was required, therefore it was decided that a ride height range of 30mm to 70mm would be used for the two-dimensional study so as to establish values hat could also be verified with previous research data. While a more optimal ride height range of 15mm to 40mm would be used for all three-dimensional cases so as to cover the 'realistic' region of ride heights employed by Formula 3 teams.

TWO-DIMENSIONAL STUDY

The investigation of the diffuser through the initial two-dimensional study would enable an in-depth look into three main areas of interest:

- Effect of diffuser performance within proximity of a moving ground plane.
- Variances in performance with varying ride height.
- Effects of ramp angle on diffuser performance.

Based on the corning speed that the analysis would be conducted over, the Reynolds number was required to be calculated so as to determine the type of flow present. The diffuser length from front to back is 2.1 metres, so the corresponding Reynolds Number is found to be:

$$\mathrm{Re} = \frac{\rho U L}{\mu}$$

$$\mathrm{Re} = 6.93x10^{6}$$

From this it was decided that a turbulence model would be required to calculate the flow, this is due to the fact that laminar flow occurs in flows with Reynolds Numbers below 3 x 10^{5} and anything above this should be considered to be turbulent flow.

The geometry of the two-dimensional study would consist of a simple bluff body configuration with a rounded nose to direct flow around the geometry, with an extension forward of twice the diffuser length, to substantially allow underfloor flow to develop. The height of the diffuser was required to be higher than the largest ramp angle to be used since the underbody flow from the diffuser does not come in contact with the upper surface due to the cars geometry, so a clearance height of 0.2 metres was decided upon. Finally, then length of the diffuser section will be based on that which is used for a 2004 Formula 3 model, of 0.7 metres.

Mesh Sensitivity:

The domain that was used for the study required that the flow was allowed to fully develop and no effects on the results obtained for lift and drag coefficients. As such, an in-depth sensitivity study was conducted on the domain height, downstream length and also the mesh density required. From the results obtained, it was seen that a downstream length of 4 metres showed a stabilising trend for both the lift and drag values. For the domain height, a distance of 3 metres was required before substantial stabilising occurred. The mesh density, since maintain as a single face, was limited to 100,000 cells for the study. The drag coefficient was seen to stabilise after 80,000 cells were employed, however lift coefficient was seen to require more then 100,000 limit. Due to the rate of change at this stage being relatively small and the final accuracy of the two-dimensional study not being the main concern in this case, it was decided that 99,689 cells that was used would be deemed suitable for this case.

The diffuser bluff body was run over three ride height conditions, 30mm, 50mm and 70mm, this would provide a considerable range of performance values. The ramp angle settings will range from 0 degrees to 14 degrees, this was based off the fact that a maximum report ramp angle of 13 degrees was used with Indy lights so beyond this would prove to be pointless.

The analysis of the diffuser took several verification processes in which to establish a reasonably accurate solution without overly expensive computational costs. As was discussed previously, it was found that a residual setting of 5e-6 would establish an accurate solution where there was little or no change to the lift and drag coefficients.

A comparison between the varying ride heights over the varying ramp angles can be seen in Figures 6 and 7. It can be seen here that the drag coefficient over the ramp angle range is relatively consistent at 0.6, however a general trend is that the minimum drag coefficient across the varying ramp angles occurs at a ramp angle of 4 degrees. The reasoning behind this was understood when observation of the pressure contour lines were observed, Figure 7. With the case of the four degree ramp angle, the area underneath the diffuser is increased slower, and as such the pressure change that occurs is smaller than for the ten degree case. The increase in drag for the zero degree case is due to the fact that there is no expansion at the end of the bluff body. Airflow underneath the bluff body is not encouraged to accelerate underneath the bluff body by any pressure potential. The airflow is squeezed at the front of the bluff body so as to direct flow underneath and is only accelerated by the moving underneath by the ground plane.

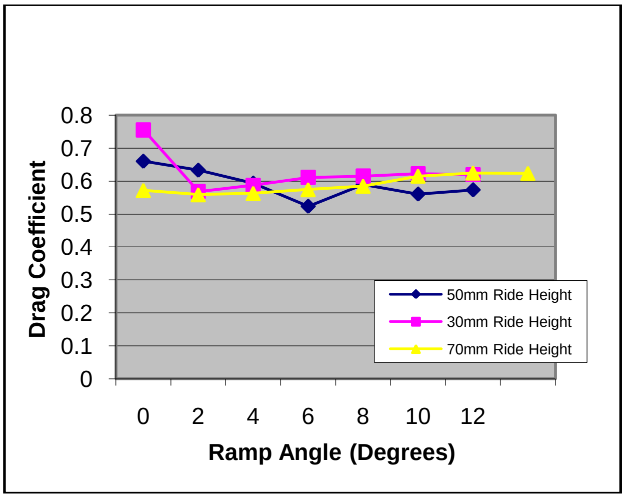

Figure 5: Drag Coefficient for Varying Ramp Angles and Ride Heights

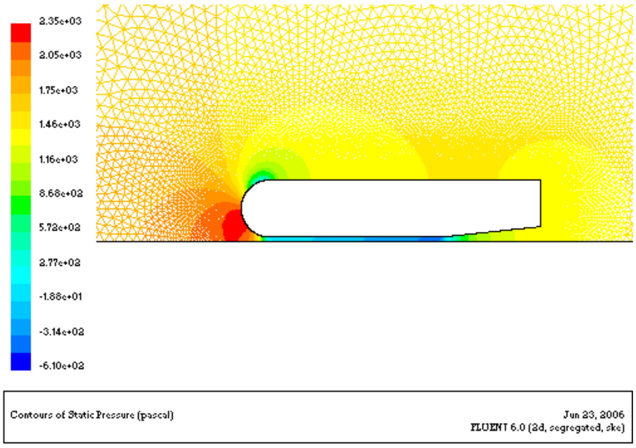

Figure 6: Pressure Contours for 6-Degree Ramp Angle

The lift coefficient is not uniform across the ride heights as seen below in Figure 8. A ramp angle of 9 degrees would seem to produce the largest downforce effect on the bluff body at lift coefficient values of –2.1 and –2.5 for 50mm and 70mm ride heights respectively. The 30mm ride height seems to produce an unusual maximum at 6 degrees for a lift coefficient of –1.83. Initially it was considered that possibly the computational set-up was incorrect which was producing this error. However, a paper by Cooper [3] and Sovran [4] was found to compare the ride height to diffuser exit height and compare the lift and drag coefficients from this for low ride heights. From their analysis on the pressure recovery coefficient for ramp angle and ride heights that were used, it was observed that the results were within 4% of that previously found. Cooper also stated that a correction factor need be applied for blockage correction which resulted in a correction of typically 6% in dynamic pressure. Therefore it can be concluded that the results were reasonable accurate for this study. The difference is results for varying ramp ride height for optimal ramp angle becomes clearer when the values are non-dimensionalised. When this was done, it was found that the maximum negative lift coefficient at 6 degrees for 30mm ride height corresponded to a ride height/diffuser exit height of 0.289, when compared to the 50mm ride height this was seen to correspond to the maximum negative lift coefficient as well.

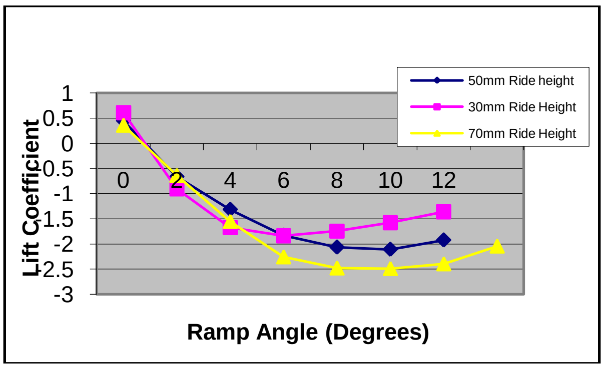

Figure 7: Lift Coefficient for Varying Ramp Angle and Ride Heights

The reduction in pressure underneath the bluff body due to the diffusing section creates a pressure difference which results in an increase airflow velocity underneath the bluff body. The velocity diagram, Figure 8 and the turbulent flow diagram, Figure 9, display the need and use of a curved ramp angle in the diffusing section of the diffuser. From the velocity display, the velocity vectors halfway up the ramp angle are greatly reduced and almost become stagnant at the outlet. This large reduction in velocity is due to the generation of turbulent flow at the outlet as seen in Figure 9. For an ideal diffuser design, this turbulent intensity would be minimal inside the diffuser section, however a large enough expansion rate is required as was seen with the lift coefficient results such that a substantial pressure difference can be created. Therefore a solution to this would be the use of a curved ramp angle, which would enable a more controlled rate of pressure change. This would reduce the amount of turbulent flow and hence reduced performance currently found from a flat ramp angle.

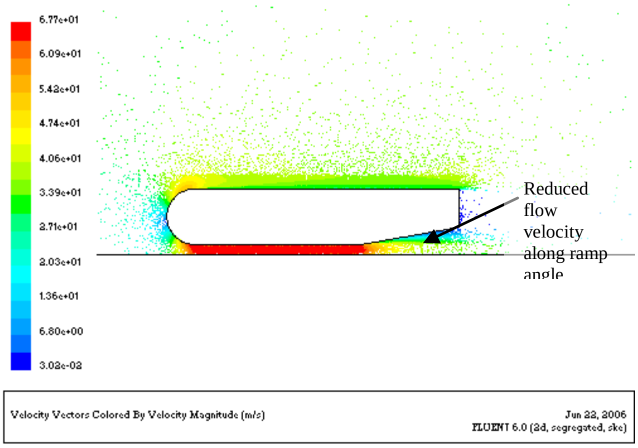

Figure 8: Velocity Vectors at 70mm Ride Height

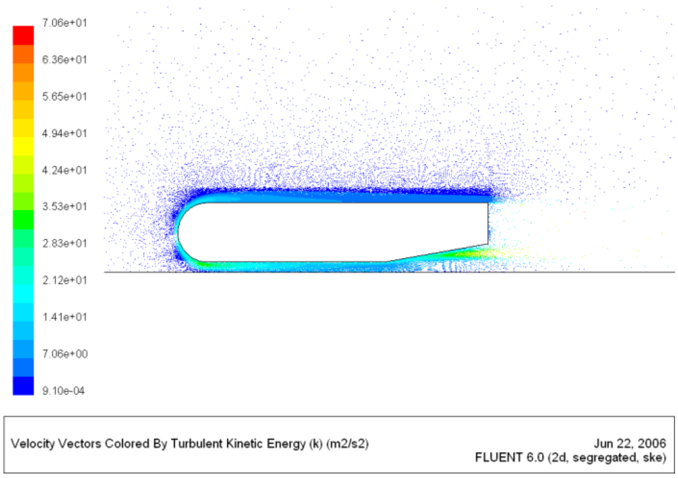

Figure 9: Turbulent Kinetic Energy Display

PRESSURE VARIANCE COMPARISON

The results obtained for the varying lift and drag performance with varying ramp angles is not yet clearly explained as yet, with reasoning seen from a comparison of the pressure contour plots obtained. Initially looking at the zero ramp angle pressure plot, it can be seen that the pressure underneath the bluff body is relatively neutral over the length with some slight drop in pressure towards the outlet. However, when this is compared to all the other ramp angle results, a negative pressure field can be seen underneath the bluff boy, while the pressure above the bluff body is roughly a factor of 10^3 higher in pressure. So this therefore accounts for the negative downforce generated, but not the optimal ramp angle. If special consideration is taken into account of the pressure distribution over the diffuser section, a larger variance can be noted. For the two-degree ramp angle case, it can be noted that the flow has not been expanded enough, therefore a region behind the bluff body where the pressure is still below the ambient air pressure is observed. The reduced pressure underneath the bluff body therefore has not been properly expanded so the lift coefficient is reduced. It can be seen for the optimal six-degree ramp angle in this case, that the pressure has been equalised behind the bluff body and the contours in the diffuser region are seen to be relatively uniform. Although the pressure underneath the bluff body does not reach as low a value, the low pressure cell seen at the top front nose of the bluff body is also reduced in size, so a larger pressure difference between the upper and lower surfaces is achieved, hence a larger pressure difference. The ten-degree ramp angle is the beginning of an over expansion, and is seen by the fact that the pressure contour in the diffuser section is seen to have dipped further under the bluff body along the floor. The comparison between the six degree case and ten degree case are shown in Figures 11 and 12 respectively. This actually produces an increase in turbulent flow behind the diffuser resulting in reduced lift force and some increase in drag forces.

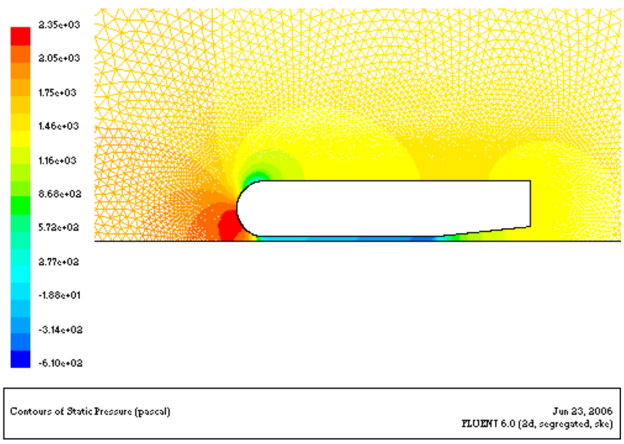

Figure 10: Six Degrees Ramp Angle Pressure Contour Display

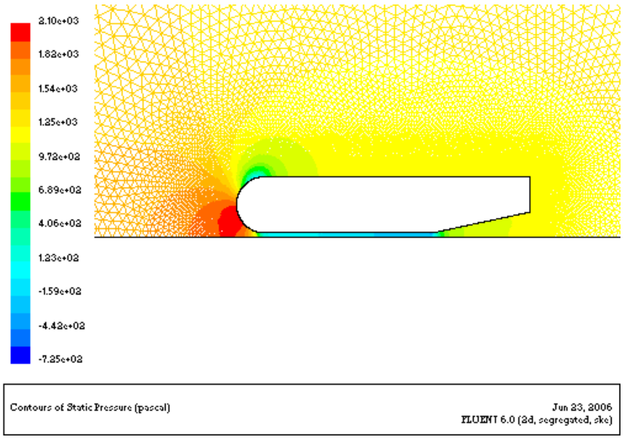

Figure 11: Ten Degrees Ramp Angle Pressure Contour Display

TWO-DIMENSIONAL SUMMARY

The two-dimensional diffuser study used three ride heights, run for a range of ramp angles and it was found that the optimal ramp angle is reliant on the ride height since the diffuser inlet to outlet area ratio determines the pressure recovery coefficient and as such the downforce performance. Therefore the lower the ride height, the slower the initial expansion rate that is required. An under expanded diffuser section saw that although a larger low pressure cell was produced underneath the bluff body, it was noted that the airflow pressure at the outlet was still lower than atmospheric pressure which reduced the pressure difference effect to increase the flow rate underneath the diffuser. This resulted in a reduced pressure region above the bluff body thus reducing the pressure difference between the upper and lower surfaces. An over expansion resulted in a higher pressure cell underneath the bluff body and also the generation of a turbulent flow structure inside the diffuser section which increased drag and reduced downforce. Another occurrence that was noted, is that a flat ramp angle does not enable the velocity profile to be maintained and as such a turbulent flow structure will form along the ramp angle surface, reducing the diffuser performance. Therefore a curved ramp surface would be required so as to increase the rate of expansion with increased section area inside the diffuser.

This information will now be used and compared with the three-dimensional investigations of the central and complete diffuser sections are conducted. The pressure performance and turbulence structure will enable a realisation if the diffuser is performing optimally and/or if design changes may be possible to increase downforce or reduce drag.

THREE-DIMENSIONAL CENTRAL DIFFUSER STUDY

The three-dimensional model of the central Formula 3 diffuser was then considered with the previous results in mind. This will enable analysis of airflow and performance of the current design and determine if the design. The parameters will be based the same as for the two-dimensional study, with airspeed set at 33m/s (120km/h). The central section of the diffuser consists of a removable diverging carbon fibre piece, which fits in just above the central floor plane at the base and comes up to cover the differential at the rear. The three-dimensional case will be examined for variance in ramp angle and curvature with varying ride height and be used to find a supposed idealised curved ramp angle geometry should the current design deem to be inferior.

There are a few geometry restrictions on any possible design modifications that will be attempted so as to maintain within the 2005 FIA regulations which restrict overall length and also having to remain higher off the ground than the lowest timber floor which is located underneath the engine.

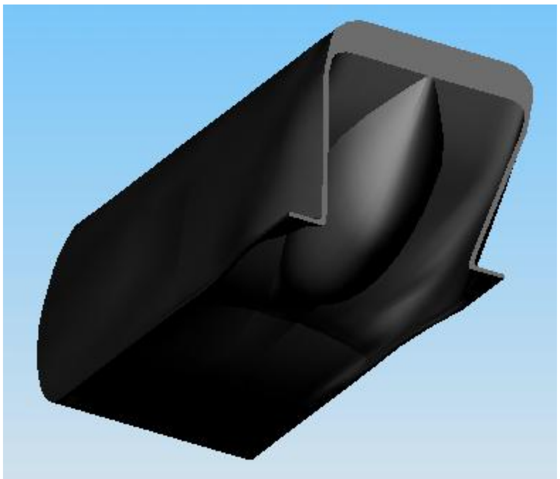

Figure 12: Central Diffuser Geometry

Another restriction is also the location of where the central diffuser is currently fixated to the underbody. The central diffuser is held in place by two screws on either side while at the base of the diffuser are two lugs which are wedged under the timber floor while two rails on the upper surface fit around the outer diffuser section. The diffuser is finally restricted by the location of the differential on the car and is why the need for a bellied section in the middle of the diffuser is required, this however alters expansion rates which will be looked into. As can be seen in Figure 13 above, a simplified bluff body has been used to divert flow around the upper surface which otherwise would not be in contact with airflow.

The domain sizing and mesh sensitivity was a concern, and as such detailed studies of these parameters were conducted. The mesh structure used made use of three primary sizing functions which covered the primary areas of concern were flow would be most complex, this was the bluff body itself in general, the central baseline underneath the diffuser and the base of the bluff body and diffuser to properly predict flow in this region. The mesh generation was created again using *Gambit* and imported into the computation program *Fluent* for analysis. The study will look at several ride heights: 15, 22, 25, 30, 35 and 40 mm. The 22mm ride height may seem somewhat odd value compared to other values used, however this is the exact ride height given by one of the teams, this height occurs during braking and hence its importance.

The lift and drag coefficient performance can be seen in Figures 14 and 15. The lift coefficient result was somewhat unexpected, the downforce generated by the diffuser actually reduces with reduced ride height. The diffuser does not have multiple settings to change its performance for specific ride heights and as can be seen from the lift coefficient curve, the downforce generated at a ride height of 22mm is around three-quarters that generated at 40mm ride height, creating only 71.06N compared to 91.64N. These values may seem only small, but it needs to be considered that only a small portion of the diffuser is being considered so far. The simple solution may seem to be to run the car at a higher ride height since downforce is improved and also drag effects are not altered greatly, however this will also increase the ride height of the car thus increasing the centre of gravity height, this will increase body roll on the car. This will also have an effect on the flow around the front wing and into the sidepods and radiator. So it can be noticed that by optimising one component of a car may in fact destabilise another component and make the overall set-up worse off so there is a trade off.

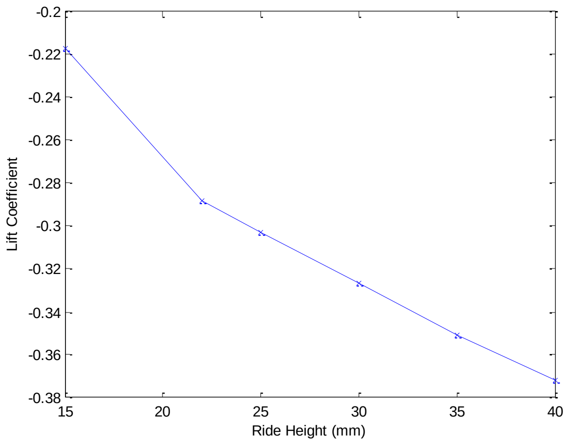

Figure 13: Lift Coefficient for Varying Ride Height

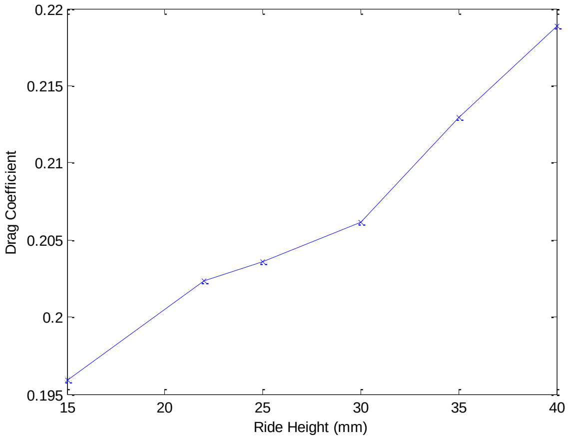

Figure 14: Drag Coefficient for Varying Ride Height

Having found that the diffuser has not been optimally designed for the desired ride height of 22mm, an opportunity for a new design to improve the downforce performance is possible. The design process would look into some possible designs seen in both industry and based on the current flow performance.

DIFFUSER PERFORMANCE PROFILE

The flow pathlines around an object help to discover if the flow is performing in the correct manner as expected and also see what areas in the design are causing any possible downfalls. Figures 16 and 17 show the flow profiles around the central diffuser. It can be seen from this that as the flow first encounters the front of the bluff body, air is pushed either over or under the body, from this, at the under floor inlet, a high pressure region is present causing some of the flow to be squeezed out the sides and join the moving airflow around the body. The airflow travelling under the bluff body was seen to stabilise by the time it reached the diffuser region, which was considered important since this was considered to occur during actual operation. Upon the flow entering the diffuser region, it is seen to have an issue with the location of the elliptical shape covering the differential. An initial expansion occurs before the flow encounters the elliptical shape where the flow is directed around this and expanded at varying rates depending on the cross-sectional location of the flow. This results in the flow being expanded at a much faster rate behind the elliptical shape then compared towards the outer edges. This diffuser design was also seen to cause the pressure inside the diffuser to reach atmospheric pressure prematurely inside the diffuser section. This limits the performance of the diffuser and actually deems the remaining portion behind the contour where atmospheric pressure is reached, to be wasted. This also makes sense as to why the higher rides heights actually produce more downforce. This is similar to the two-dimensional study where the inlet to outlet area ratio determine the diffuser performance, also if the expansion rate of the diffusing section is too quick then the pressure inside will reach the atmospheric pressure too early, when ideally this should occur at the outlet.

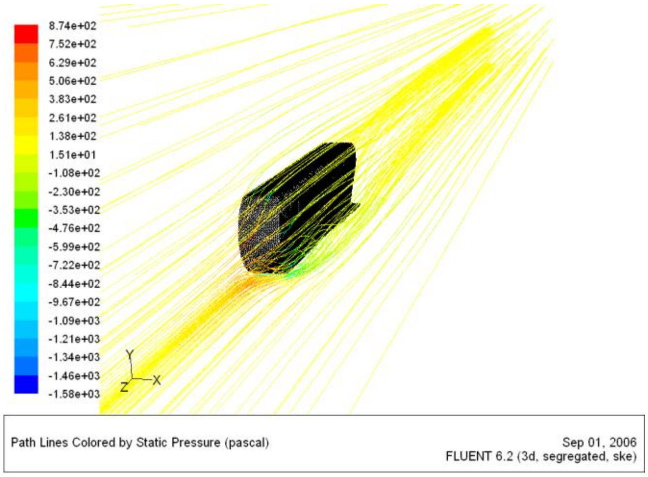

Figure 15: Flow Profile Around the Front of the Tested Bluff Body

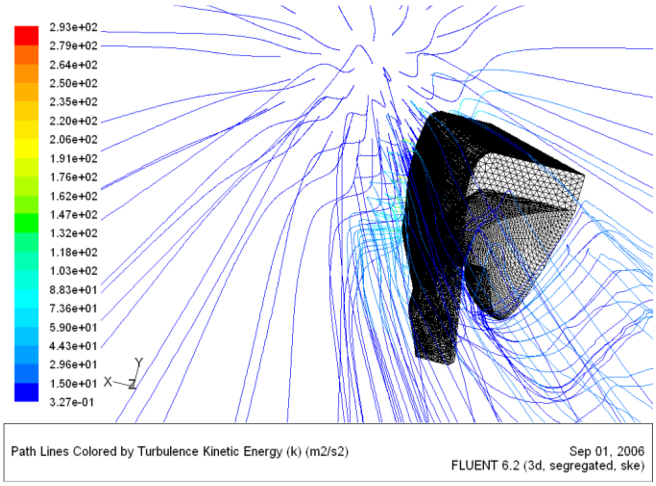

Figure 16: Flow Profile Inside Diffuser Section

SIDEWALL TURBULENT FLOW

One area of concern when analysing the central diffuser section alone is to do with the initial flow profile. Previously it was mentioned that some of the flow trying to travel underneath the bluff body was squeezed out he sides, this process actually slows the flow down. The problem occurs as to where the flow goes after this. In the pressure contours, no real large pressure difference was seen in the diffusing section which was thought to occur, this was seen to be because the flow that was initially squeezed at the inlet is sucked back towards the bluff body side and in underneath the sidewall of the diffuser, and is realised by the turbulence flow generated. This process will actually increase the airflow underneath the bluff body and also increase the pressure inside the diffuser. This will lower the pressure difference between the inlet and outlet of the bluff body and so reduce the airflow velocity underneath the body which inversely also increases the pressure. By increasing the pressure here under the bluff body, the pressure difference between the upper and lower surface is reduced so reduces the downforce value that is generated. This will be reconsidered for the complete diffuser analysis.

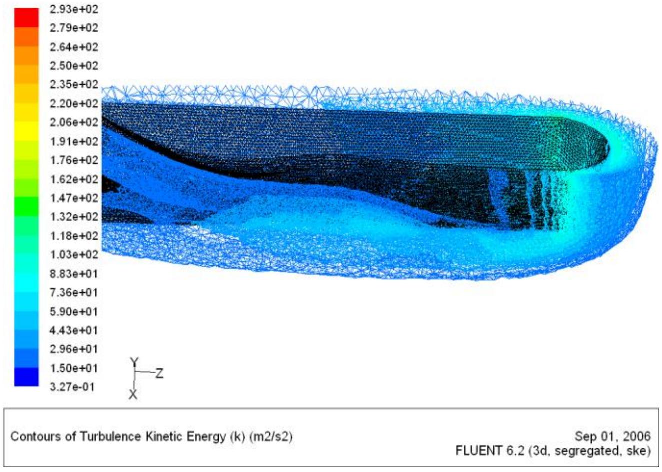

Figure 17: Turbulent Flow Structure Under Sidewall

ALTERNATIVE DIFFUSER DESIGNS

Based on the flow analysis conducted on the current central diffuser design, it was seen that the current design is not optimal for the intended ride height and speeds, which are encountered during cornering. From this five alternative designs were proposed and tested based on designs from other racing codes such as Formula 1 and Indy car, also in conjunction with the Formula 3 Rules and Regulations for bodywork dimensions, five alternate designs were analysed:

1. 200mm extension of the diffuser length
2. 100mm extension of the diffuser length
3. Inclusion of a wedge or aerofoil inside the rear diffuser section.
4. Reduced diffuser outlet area.
5. Alternative diffusion rate.

Extended Designs

From the diffuser performance investigation it was found that one possible cause for the diffuser's under performance at the 22mm ride height was due to an incorrect expansion resulting in the pressure inside the diffuser region reaching the atmospheric pressure at some distance inside. This can be due to two reasons, either the inlet to outlet area ratio was causing expansion of the airflow to the correct pressure to be completed prematurely, or the ramp angle inside the diffuser was too large. The use of extended designs will enable slower diffusion rates while maintaining the advantage of the pressure difference of a large outlet area.

Wedge or Aerofoil Design

This design was based on an old Ferrari Formula 1 diffuser where at the base of the central diffuser, an aerofoil was located. The effects of this on both the airflow characteristics and downforce are unknown, especially since it is placed in a location of expending, slowing airflow. Article 3.11 of the Formula 3 Technical Regulations [1] states, that a maximum of three aerofoil sections may be used behind the front edge of the complete rear wheel assembly. It was decided therefore that a method to possible get this while still producing a similar effect is by inserting a wedge section inside the diffuser section. The angle and location of the wedge was based on an investigation on the flow angle at a similar location to the other aerofoil where the airflow was seen to be travelling at an angle of 32º in this region. As shown in Figure 19, the wedge was then set relative to this such that a negative ten-degree angle of attack was produced so as to generate a downward force.

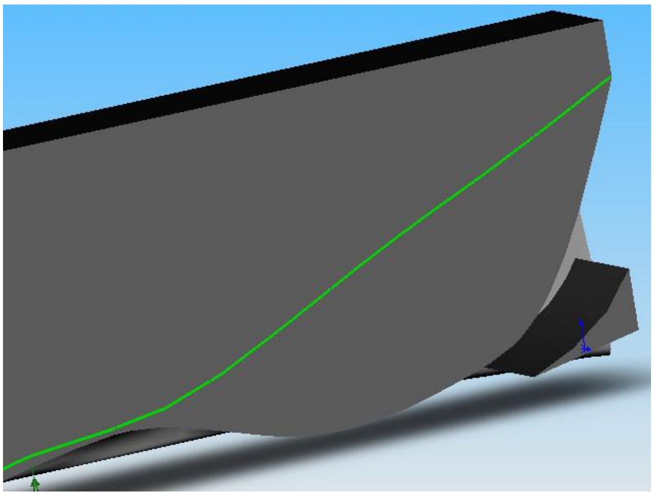

Figure 18: Wedge Design Location

Reduced Outlet Area

Another reason for the diffuser under performance could be due to the possible over-expansion of the side components of the diffuser. This method was to look into reducing the outlet area, so reduce the outlet height. There is a disadvantage to this since the inlet to outlet area ratio will be reduced so the pressure ratio is reduced, this should reduce the flow velocity under the bluff body and theoretically reduce downforce. Although it is considered that the downforce will be reduced, the pressure effects by varying the ramp angle will be more intriguing aspect of this design.

Alternative Diffusion Rate

The final design is the modification of the current ramp angle design. By maintaining the current area ratio, the pressure potential will remain the same, so should underbody flow velocity so the effects of the rate of expansion will be interesting. The design will be based on the two-dimensional results, which saw that for a low ride-height a smaller ramp angle was optimal so as to not over expand the flow while larger ride heights were able to make use of faster expansion rates. So from this, the initial diffusion rate will be reduced compared to the current design, and slower increased towards the outlet.

ALTERNATIVE DESIGN RESULTS

There are several expected results that were seen to occur for the alternative designs. The lift and drag coefficient performance for the designs with varying ride height can be seen in Figures 20 and 21. Adapted Design #1 is the reduced outlet area design, and as was predicted, this resulted in a lower lift coefficient, especially for larger ride heights where the area ratio became even smaller. For this reliance, the reduced outlet area will now no longer be pursued.

The ramp design was thought to have some merit due to its commercial use, however the results showed that the lift coefficient is actually greatly reduced while the drag component is increased. Due to this the pressure profile was further investigated, this showed that the inclusion of a wedge has caused the pressure inside the diffuser to expand correctly however the pressure underneath the bluff body is also increased. The current wedge design has actually acted to reduce the outlet area and thus reducing the pressure potential between the inlet and outlet. Although an ideal wedge shape has not been sought, the restrictions preventing the use of more than three aerofoils could prove to be the downfall of this design.

The alternative ramp angle design #2 has shown some promise with no real major affect to larger ride height lift coefficient but improved downforce for the 30mm ride height case. The problem however is still for the desired 22mm ride height where the design still under performs. The values obtained for the lift coefficient suggest that the alternative ramp angle design is more correctly suited for a 30mm ride height than a 22mm ride height. The region where the downforce has peaked means that the expansion rate is occurring too fast and this is why the lower ride height is not improving. Therefore possible redesign of this may yield more promising results.

The final two designs are the extended versions, these designs are beneficial in slowing down the diffusion rate, and due to a large reference area, theoretically, a larger downforce value. The 200mm design showed very promising results for the drag coefficient, with a 17% reduction in drag coefficient. However the lift coefficient was seen to perform very poorly for the 40mm ride height and only a slight improvement with reduced ride height, this was believed to possibly be due to the expansion rate being reduced too much hence a short 100mm extension was considered. The performance however showed a very similar trend to the adapted ramp angle design except slightly lower values, so possibly the diffuser had reached the optimal length. However, the coefficients were then considered in accordance with their reference areas as listed below:

- Original Design: $0.18648m^2$
- 100mm Extension: $0.20131m^2$
- 200mm Extension: $0.21613m^2$

Based on these values, the actual downforce value for the 22mm ride height was considered and it was found that the force obtained and increased:

- Original Design: 71.698N
- 100mm Extension: 71.712N
- 200mm Extension: 82.061N

So from this, the performance of the 100mm extension design had not changed while the 200mm design had shown an increase in downforce of 14.5% while also a reduction in drag of 17% which is a considerable improvement with regards to racing car designs. The reason for the lack of change in the 100mm design is thought to be possibly due to the curvature used inside the diffuser section and as such will be re-assessed for the complete study. The reason that is was believed to be under performing is due to the pressure expansion inside the diffuser section. For the 100mm case, the diffuser once again reaches atmospheric pressure well before the outlet, this reduces the effect of increasing the diffuser length and as such is why a similar downforce has been obtained.

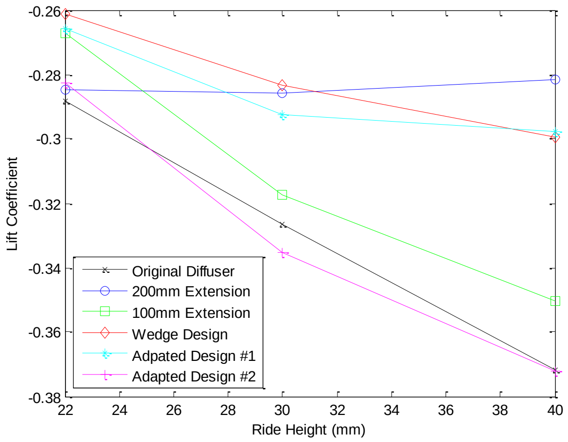

Figure 19: Lift Coefficient For Central Diffuser With Varying Ride Height

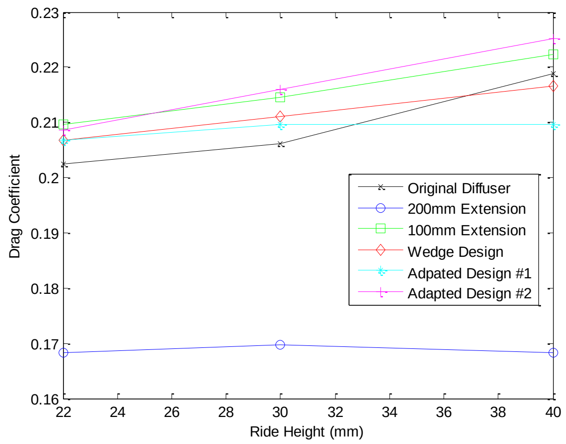

Figure 20: Drag Coefficient For Central Diffuser With Varying Ride Height

COMPLETE DALLARA FORMULA 3 DIFFUSER STUDY

The complete diffuser will now take into account the flow from underneath the sidepods. It was noted from the central diffuser study that the performance of the diffuser may not only be reliant on the flow underneath the bluff body but also on the flow profile along the outside walls. This was noted from the turbulence flow production created by the pressure difference between the outside pressure and inside the diffuser section. The only possible solution for this was the analysis of the complete diffuser section so that the actual pressure values on the outside walls of the central diffuser section could be realised, as such, this may have an effect on the results obtained from the central diffuser study. The main areas that will be addressed for the complete diffuser study will be:

- Effects of ride height on diffuser performance
- Effects of diffused airflow from sidepods on the pressure inside the central diffuser section.
- Flow profile and performance of complete diffuser.

The flow parameters will be maintained the same as for the central diffuser study with a flow speed of 33m/s and same analysis ride height settings. The complete diffuser geometry can be seen below if Figure 22, and as for the central diffuser will be encased into a simplified bluff body configuration of a simple rounded front end to direct flow around the bluff body. The height of the bluff body was a critical decision since the flow interaction between the diffuser flow and that travelling over the top surface of the bluff body is considered to have an affect of the overall performance. Obviously, the design of this aspect of the car geometry is heavily reliant on other aspects of the car design, however for this study due to simplification and shortage of resources, the aspects that could not be considered were:

- Flow effects coming from the rear wing assembly and its affects on the diffuser;
- Flow profile and effects resulting from the location of the rotating rear wheels; and
- Variance in materials used for the underbody (timber and carbon fibre) and road surface.

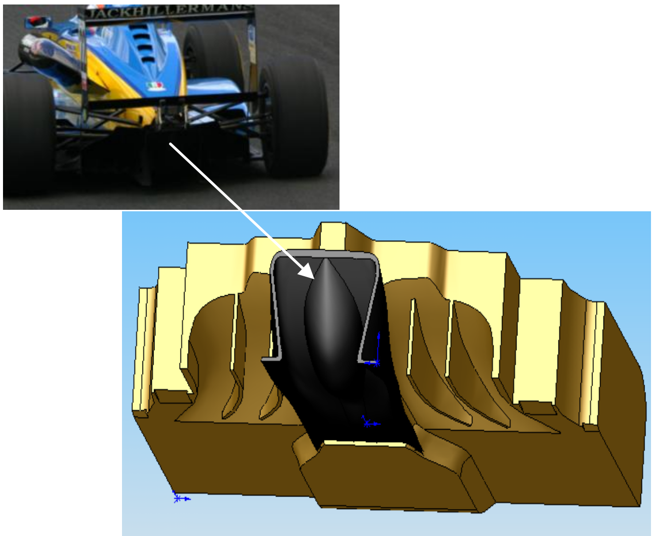

Figure 21: Complete Diffuser Geometry and Location

The geometry was imported into *Gambit* and analysed using *Fluent* as per the central diffuser study. The domain and mesh sensitivity studies were conducted on the geometry to ensure a certain degree of accuracy due to the lack of comparison data that is available. From this it was found that a mesh sizing of around 1.6 million cells would produce relatively accurate results without overly expensive computation cost. This required a run time of 4 days, compared to an increase of half a million cells more resulting in run times of around 16 days.

COMPLETE DIFFUSER RESULTS

The complete diffuser was tested for the same ride heights as used for the central diffuser study of 15, 22, 25, 30, 35 and 40 mm. The overall interest is the lift and drag coefficient performance of the complete diffuser and also if there is any change in the trend line between the complete and the central diffuser geometries.

In Figure 23, the lift coefficient performance for the complete original diffuser can be observed, it is noted straight away that the complete diffuser has not been designed for such a low ride height of 22mm. The change in downforce generated can be seen to be relatively stable over the range of 22 to 35mm. The reason the downforce generation is greatly reduced below this ride height is due to the area underneath the bluff body becoming too small. As this happens the effects of the boundary layer and turbulent flows created reduce the flow speed underneath the diffuser, this in turn increases the pressure underneath the diffuser. This reduction in pressure difference between the upper and lower surfaces of the bluff body is the reason a smaller downforce value is obtained. The reason the lift performance improves with even greater ride heights is due to the expansion rate of the diffuser. The difference in area between the bluff body inlet and the outlet of the diffuser is critical in obtaining an ideal pressure flow. As can be seen from the results, the area ratio that is currently employed is more suitable for a larger ride height and as such means that the airflow from underneath the car is being expanded at too fast a rate for the desired 22mm ride height. The downforce generated at 22mm is equivalent to 196.69N compared to 207.42N for the 40mm ride height case. This is an increase of 5.2% by merely increasing the ride height by 18mm.

The drag coefficient of the diffuser is somewhat unchanged for the varying ride heights although a slight reduction is seen for the lower cases. It can be seen on the scale on the left that a change of 0.05 occurs over the range, which is not even 1N force difference. So the ride height can be realised to not be reliant on the ride height for this design.

Overall, the main issue obtained from these results is that the current design is not optimal for the current desired ride height used. Areas of development and improvement will be based on the information obtained from the central diffuser study and results of the pathlines and pressure contour plots of the current design.

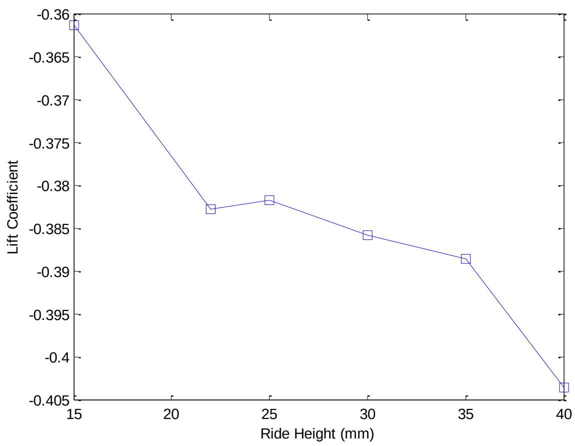

Figure 22: Lift Coefficient of Original Diffuser

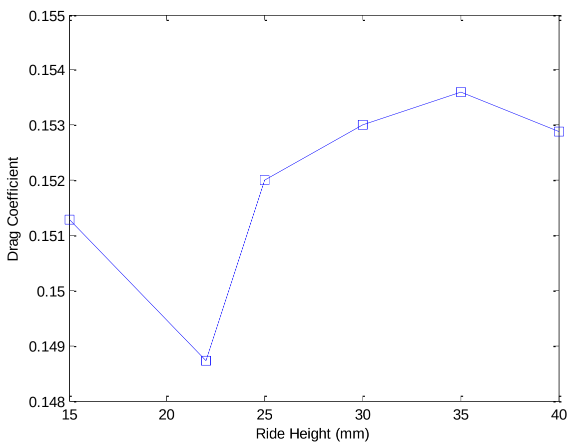

Figure 23: Drag Coefficient of Original Diffuser

FLOW PERFORMANCE PROFILE

It was noted in both the two-dimensional study and the central diffuser study, that the key aspect to the diffuser performance is the resulting pressure contours surrounding the diffuser geometry. Figure 25 shows the underside of the diffuser. A lower pressure region in the front central section of the diffuser is seen. This was expected as the airflow is compressed then sucked and pushed under the bluff body due to pressure differences. This is seen to have dispersed halfway along the floor section. What was not expected was the effect of the step from the central floor section to the underside of the sidepods. The pressure contours are seen to undergo a drastic shape change, denoting a turbulent flow pattern. The other main concern is that the outside pressure reaches a considerable distance inside the central diffuser section, however this does not occur until behind the outer diffuser section. This is due to the variance in diffusion rates and the outer section, although undergoing expansion in the vertical plane, it actually undergoes compression by restrictions in the side width to direct flow around the rear wheels. The reason incomplete diffusion occurs was realised when the turbulent flow generation inside the outer diffusion region was investigated. The turning vanes inside this region are in place to help maintain a relatively consistent flow across the diffuser width, otherwise the directed flow would just be pushed to one wall and greatly reduce the effect of the diffuser and result in large turbulence generation. However it can be seen that the initial angle of the turning vanes is too large to the oncoming flow hence a turbulence region is produced on the inside of the turning vane, hence reducing the effectiveness of the diffusing region resulting in an under-expanded flow.

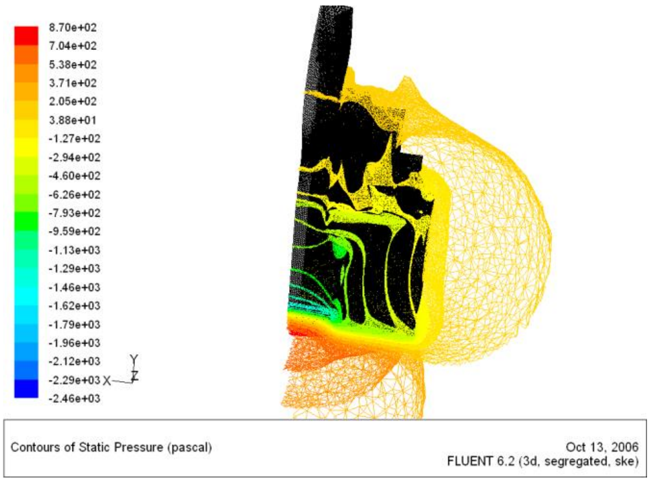

Figure 24: Pressure Contour Display

Another cause of the turbulent region located behind the outer diffusion section is the difference in flow speed from the flow coming out of the diffuser and the faster flow coming from around the side of the bluff body. This is a problem associated with the simplification of the geometry whereby the performance of one aspect of the car is reliant on every other region surrounding it. The flow coming round the side would be interacted with the flow from the rotating rear wheel and hence would not interact with the flow from the outer diffuser section in this manner. What actual effects the rear wheel would have on the diffuser is not part of this study.

NEW DIFFUSER DESIGNS

The alternative central diffuser designs that were investigated varied from extensions, reduced outlet area, wedge implementations and alternative diffusion rates. From this it was seen that the reduced outlet area design posed no real possible improvement, and although the wedge design is plausible, the time and resource required to perform a detailed study to obtain an ideal angle and wedge shape was not possible. It was seen that the diffuser was reliant on a few governing factors, firstly the area ratio is important and hence improvement in the extension designs was observed. The rate of diffusion of the airflow inside the diffuser section is also an important factor. A constant expansion rate does not provide a potential pressure change to draw the flow through the diffuser, also the different sections along the diffuser require different ramp angles due to area ratio changes from increased ride height.

Therefore it was decided that due to their poor performance, some of the designs will not be further studied primarily due to the longer run time associated with the complete diffuser study. The alternative designs that will be considered in the complete diffuser study are:

- 200mm extension of the central diffuser;
- 100mm extension of the central diffuser; and
- Alternative diffusion rate.

The section that was to be altered is still only the central removable section, the results obtained would then be able to be compared to the central diffuser study and any trend change present due to the absence of the turbulent flow from the sidewall would be observed.

The ramp angle design will remain the same as for the central diffuser study for each design with the difference mainly on the upper and side surface. The upper surface of the central diffuser curves over and forms a slight edge before joining to the main diffuser section. The effect o this on the flow is not quite clear since the flow must actually travel from underneath the diffuser, up and over to create any downforce effect on this surface.

NEW DESIGNS RESULTS

Table 1 shows the results of the new designs. These results are somewhat unexpected based on the central diffuser results. The alternative diffusion rate design was somewhat peculiar as it actually changed from having a similar performance to the original design to that of the reduced outlet area results. The only reasoning for this that can be seen is due to the elimination of the turbulent cell underneath the sidewall of the central diffuser, which would have increased the pressure at the start of the diffuser section. The 200mm extension design can be seen to have changed slightly with a positive increase in the trend line so that it has become even less efficient at larger ride heights, while the lift coefficient at the 22mm ride height remains much the same. The 100mm case is the most surprising result, for the central diffuser study it was seen that the lift coefficient showed a similar trendline to the current design only of some magnitude less. However now that the complete diffuser has been taken into account, this design is more optimal for the 22mm ride height with reduced downforce with increasing ride height. This trend is similar to the 200mm extension, the difference between this and the original design is believed to be due to change in diffusion rate of the two extended versions. Both of these designs slowed the initial expansion rate, which is more favourable for a low ride height.

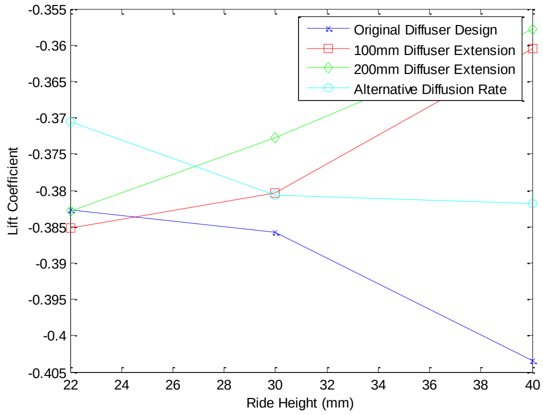

Figure 25: Lift Coefficient for Alternative Diffuser Designs

The critical ride height that has been addressed here is 22mm, however this has been stated to occur under braking, so effects during normal operation concerns ride heights more in the range of 30 to 40mm. Another aspect that has not been considered here is that by increasing the diffuser length also increases the reference area over which the coefficient is acting.

Design	Ride Heights		
	22mm	30mm	40mm
Original Design	196.71 N	198.32 N	207.42 N
100mm Extension	203.87 N	201.32 N	190.78 N
200mm Extension	208.96 N	199.36 N	195.32 N
Alternative Diffusion Rate	194.56 N	195.72 N	196.32 N

Table 1: Downforce for Complete Diffuser Designs

Design	Downforce (N)	%
Original Design	207.42	-
100mm Extension	190.78	-8.022
200mm Extension	195.32	-5.834
Alternative Diffusion Rate	196.32	-5.351

Table 2: 22mm Ride Height Performance

Table 1 shows how the overall performance of the extended designs can be seen to improve the downforce generated by 3.64% for the 100mm case and 6.23% for the 200mm case. These improvements may seem only small but what needs to be realised is that motorsport design changes are based on the search for improvements of even 1 or 2%.

A very important factor is the fact that both the 100mm and 200mm extension designs show an optimal downforce at a ride height of 22mm and reduced downforce with increased ride height. The 22mm case was stated to exist during cornering which is deemed to be the time when maximum downforce is required to increase traction. A higher ride height of up to 40mm would occur during standard straight-line driving such as along straights and accelerating out of corners. The weight of a racecar is critical for increased acceleration due to larger pressure to weight ratio, while increased downforce during cornering would increase traction and enable a faster cornering speed. This is why the optimal downforce performance was sort at the 22mm ride height.

Design	Downforce (N)	%
Original Design	196.71	-
100mm Extension	203.87	3.639
200mm Extension	208.96	6.227
Alternative Diffusion Rate	194.56	-1.092

Table 3: Complete Diffuser Downforce Performance at 40mm Ride Height

Table 3 shows the complete downforce performance at a 40mm ride height. From these results, the 200mm case shows an overall improvement of around 12.06% between additional cornering downforce to reduce standard weight critical for increased acceleration. The 100mm extension design also yielded an 11.66% increase in performance. The alternative diffusion rate may reduce the downforce rate associated whilst under acceleration and straight-line speed, however the cornering downforce has also been reduced which is deemed more critical.

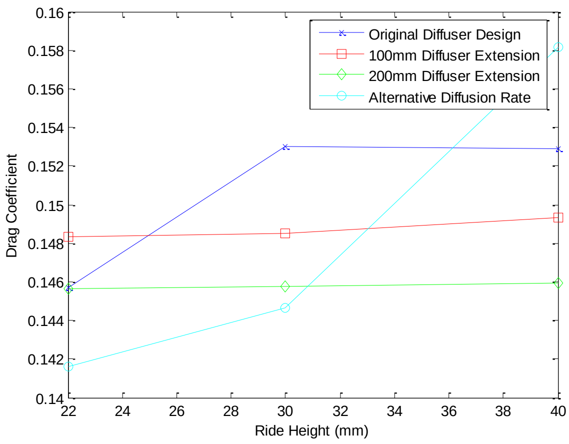

Figure 26: Drag Coefficient for Alternative Diffuser Designs

The drag performance is somewhat different then for the central study. Both the 200mm and 100mm extension design seemed to be relatively consistent for the varied ride heights with no real difference between these and he original design.

NEW DESIGN FLOW PROFILES

The flow structure of the original design had showed an incomplete diffusion and turbulent flow structure being generated. The 200mm design was also seen to reach the atmospheric pressure well inside the central diffuser section, however what was also noted, was the creation of a higher-pressure cell behind the bluff body. The reasoning for the presence of this cell was not able to be determined and as to why this helped increased the downforce was not clear.

When the 100mm design was considered, (Figure 28 and 29) the high-pressure cell behind the bluff body was once again present only slightly smaller. Upon comparison between the 22mm ride height case and 40mm ride height case, the cell was seem to have shrunk for the larger ride height. From this it could be determined that a higher pressure cell was produced due to the flow travelling through the diffuser section was actually being over-expanded since the area ratios are being expanded too quickly for the flow speed. It was noted that for the original design, the flow was being expanded too quickly initially, however a reduced outlet area reduced the pressure difference and hence greatly reduced flow speed under the diffuser which reduced downforce generated. The alternative diffusion rate design showed that altering the diffusion rate is therefore critical while the extended designs a larger distance to diffuse over. The high-pressure cell that is forming behind the diffuser is actually caused by the interaction of the moving ground plane, the slow moving air from the diffuser and the faster moving airflow over the bluff body. This interaction means that the faster moving airflow loses momentum as it makes contact with the slower flow and as such increases the static pressure of the region. This was further supported from the results obtained for the alternative diffusion rate designs, whereby the diffusion rate was initially reduced and increased in rate faster towards the end so as to maintain pressure potential. This is seen to actually push the pressure region further inside the central diffuser and as such a higher-pressure region due to the flow interaction behind the diffuser is once again formed.

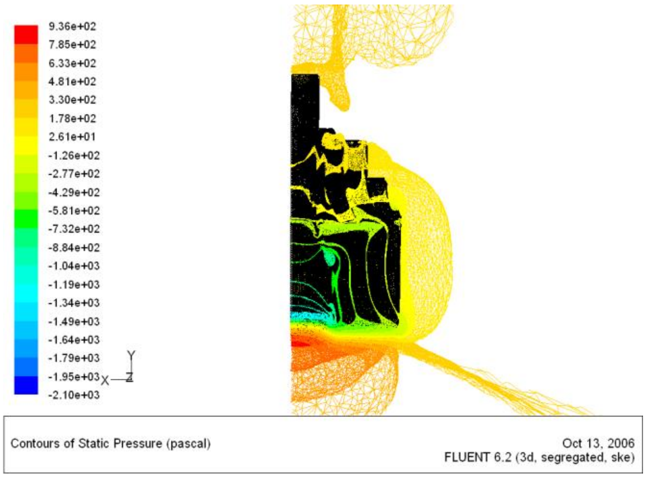

Figure 27: Pressure Contours Under 100mm Design at 22mm Ride Height

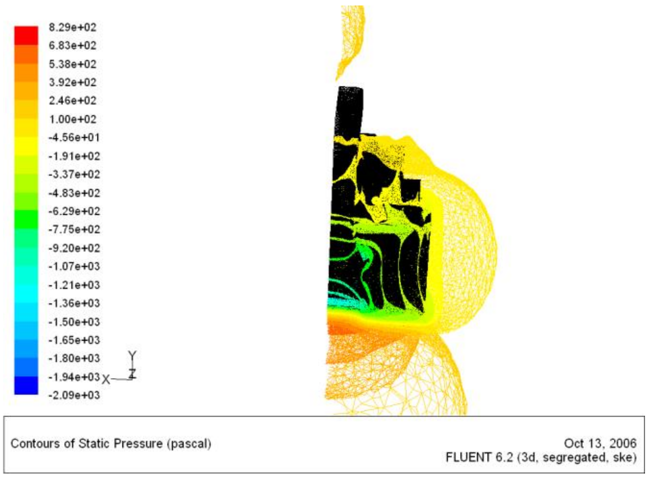

Figure 28: Pressure Contours Under 100mm Design at 40mm Ride Height

CONCLUSIONS

To aim of this study was to investigate the 2004 Dallara Formula 3 diffuser performance under race conditions used and provided to current Formula teams for optimal performance. The design methodology used was based around the understanding of the key factors regarding the diffuser of a Formula 3 racecar and also to designate whether or not the current design was optimal or not for desired cornering conditions.

Two-Dimensional Study

- A two-dimensional study was initially drafted so as to provide a basis for the understanding of the diffuser performance in moving ground proximity. This proved to be comparable to results obtained by Sovran and Klomp [4].
- The two-dimensional geometry was based around the same 700mm diffuser length currently used and floor length the same as for the three-dimensional study. The geometry was tested for a ride height of 30, 50 and 70mm and ramp angle of 0 to 14 degrees.
- The pressure and turbulence performance for varying ride heights and ramp angle designs were used to understand why certain ride heights required certain ramp angles for ideal performance. Whereby a ride height of 30mm required a ramp angle of 6 degrees while the 50 and 70mm ride heights required ramp angles of 9-10 degrees for optimal performance.

Three-Dimensional Central Diffuser

- The critical case study was found from current Australian Formula 3 teams, Protecnica and InsightF3. This was found to be during cornering for which a ride height of 22mm was encountered at an average speed of 120km/h.
- The diffuser was tested for a range of ride heights from 15mm to 40mm to find the trend and the optimal ride height in this range. It was found that the optimal ride height was actually higher than 40mm, which was not expected and as such opened an area for design improvement.
- Five alternative designs were then created based on the pressure performance of the original design and current observed designs from Formula 1 and Indy car. These were a 100 and 200mm extension of the diffuser length, alternative diffusion rate, reduced outlet area and insertion of a wedge or aerofoil inside the diffuser.
- It was found that the wedge and reduced outlet area designs improved pressure performance however also reduced pressure potential so reduced downforce. The 100 and 200mm extension designs showed promising results, especially for the 22mm ride height case, while the alternative diffusion rate showed improved downforce for the 30mm case. These improvements were found to be due to the pressure inside the diffuser reaching atmospheric pressure closer to the outlet.
- A critical flow problem was the presence of a large turbulence-generating vortex entering under the sidewall halfway along the diffuser. Its presence was due to the large pressure difference between the expanding flow inside the diffuser and the increased outside pressure in that area.

Three-Dimensional Complete Diffuser

- The current design was seen to not be designed to produce maximum downforce at the desired ride height providing 196N of downforce.
- The pressure contours showed that the outer diffuser section was under-expanding while the central diffuser had fully expanded by halfway down the diffuser. The reason for the under-expansion was found to be due to turbulent generation resulting for the design of the turning veins. These were actually causing large turbulent structures which carried through the rest of the diffuser length. This along with turbulent flow created by the faster airflow travelling around the sidewall created a large turbulent region behind the outer diffuser section.
- The alternative designs used were the 100 and 200mm extensions of the diffuser length and alternative diffusion rate designs.
- The 100 and 200mm designs were seen to produce results such that the 22mm ride height became the optimal design. For these two cases the downforce generated was seen to increase by 3.6% and 6.2% respectively. This was a downforce of 209N compared to 196N for the original design.
- By the 22mm ride height being the optimal ride height, the reduction in downforce at 40mm which would occur during straight line driving and acceleration was seen to decrease by 8% and 5.8%. The reduction in downforce would result in improved acceleration since reducing downforce here is in a sense reducing the weight so to increase the power-weight ratio.
- The reason for the improved downforce is believed to be due to a large pressure cell that was created behind the diffuser geometry. This cell was formed due to the over expansion of the central diffuser causing slower airflow than previous behind the diffuser. When this flow interacted with that from the top surface, the momentum of the faster moving flow was reduced and hence the increased pressure cell formed. The downforce was increased since this faster moving flow is in fact sucked don over the back edge which actually generates an increased downforce.

ACKNOWLEDGEMENTS

The author would like to thank the help and support from Mr. Matt Thio from Protecnica for fruitful discussions on Formula 3 regulations and diffuser ideas, without his help and opportunity, none of this would have been possible. Also to his family who have always been there and that is eternally grateful.

REFERENCES

1. Appendix J, Article 275 2003 – Formula 3 Technical Regulations, Federation Internationale De L'Automobile

2. Han, T. Hammond Jr., D.C. and Sagi, C.J. Optimisation of Bluff-Body Rear End Shape for Minimum Drag in Ground Proximity, Vehicle Aerodynamics, PT-49, Edited by Dr. V. Sumantran and Dr. G. Sovran, pp-341,

3. Cooper, K. R. Bertenyi, T. Dutil, G. Syms, J. Sovran, G. The Aerodynamic Performance of Automotive Underbody Diffusers, SAE 980030, 1998

4. Sovran, G. Klomp, E.D. – Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular Conical or Annular Cross-Section, Synopsium on Fluid Mechanics of Internal Flow, Elsevier Publishing, Amsterdam, 1967

5. Jimenez, J. Perspective in Fluid Dynamics: A Collective Introduction to Current Research, Chapter 5 'Turbulence', Cambridge University Press, 2003.

6. Aroussi, A. Measurement and Prediction of Recirculating Flows in a "T" Shaped Cavity, Computational Fluid Dynamics, G. de Vahl Davis and C. Fletcher, 1988, p229

7. Saif-Deen A, Journal articles by Fluent Software Users (1999) Running RAMPANT: Computational Fluid Dynamics In Formula Design, http://www.fluent.com/solutions/articles/ja05.pdf,

CONTACT

The author can be contacted +61 (02) 9528-5399, Fax +61 (02) 9589-3061, email kpeddie@windtech.com.au