AN ABSTRACT OF THE THESIS OF

Karl Jensen for the degree of Masters of Science in Mechanical Engineering

presented on June 10, 2010.

Title: Aerodynamic Undertray Design for Formula SAE

Abstract Approved:

Robert Paasch

Aerodynamic improvements in automotive racing can have a significant effect on vehicle performance. Recent developments in Formula SAE (Society of Automotive Engineers) have included the design and implementation of aerodynamic devices such as inverted wings and undertrays to improve performance. In this work the literature of undertray technology is presented and a design of an undertray for the Global Formula Racing car is developed. Computational Fluid Dynamics simulations are used to iterate the design and discover the effect on the downforce developed of various vehicle parameters such as speed, ride height and roll.

Predicted performance is then tested using on-track data and statistical analysis is preformed on lap times from a back-to-back comparison to identify the gain of the undertray. The comparison shows a 31% error from predicted to measured downforce, with a statistically significant 1% improvement in lap times.

Aerodynamic Undertray Design for Formula SAE

by
Karl Jensen

A THESIS

submitted to

Oregon State University

in partial fulfillment of
the requirements for the
degree of

Masters of Science

Presented June 10, 2010
Commencement June 2011

Masters of Science thesis of Karl Jensen presented on June 10, 2010.

APPROVED:

Major Professor, representing Mechanical Engineering

Head of the School of Mechanical, Industrial and Manufacturing Engineering

Dean of the Graduate School

I understand that my thesis will become part of the permanent collection of Oregon State University libraries. My signature below authorizes release of my thesis to any reader upon request.

Karl Jensen, Author

ACKNOWLEDGEMENTS

I would like to thank Dr. Robert Paasch for his support and advice throughout this work as well as the Global Formula Racing team for their support of the project. I would also like to thank my committee members for their time and expertise.

TABLE OF CONTENTS

Page

LIST OF FIGURES

1. INTRODUCTION

Formula SAE is a collegiate design competition organized by the Society of Automotive Engineers (SAE) in which student engineers design, build, test and race an open wheeled formula style race car. Since the competitions inception in 1981, the cars have been evolving and changing and there has been no single design that stands out as "the best." Wordley et al. [1] explains the lack of design convergence well and cites a few reasons such as lack of information management and careful rules planning that limit design convergence.

One development that seems to be more common of late is the use of downforce producing aerodynamic elements [1-6]. Downforce is the vertical force that is produced from aerodynamic loads instead of mass. A tires coefficient of friction will decrease with added vertical force. This means that a lightweight car will be able to make more efficient use of its tires than a heavier car and will be able to accelerate faster in any direction. Aerodynamic elements, however, produce vertical load on the tires with very little added mass, giving the tires more grip and allowing the car higher acceleration [7]. These elements come in many forms, but the major contributors to downforce are inverted wings and underbody diffusers.

Design of aerodynamic elements for race cars is complex due to the body interactions between the elements and the car, wheels, etc, and has in the past been mostly an experimental science [7, 8]. Recent developments in computational fluid dynamics (CFD) and also computer technology have allowed the simulation of aerodynamics to accurately predict the downforce, flow patterns and many other features of the air flow around the vehicle. This simulation can greatly reduce the cost and time needed to test aerodynamic elements.

In this work the design of a Formula SAE undertray is developed using CFD and verified with on-track testing to determine actual vehicle performance increase.

2. CURRENT UNDERTRAY TECHNOLOGY

The idea behind an undertray is to use the close proximity of the vehicle to the ground, termed ground effect, to cause a venturi-like effect under the vehicle [9-11]. Like a venturi there is a nozzle that increases the velocity of the air underneath the vehicle, a throat where the maximum velocity is reached and a diffuser where the air is slowed back down to free stream velocity. Bernoulli's Equation shows us that as the local velocity increases relative to the free stream velocity the local pressure is decreased. Using this lower pressure under the vehicle and the higher pressure on top, downforce can be created.

Like a venturi, the efficiency of an undertray is only as good as the efficiency of the diffuser section [12]. Due to its high visibility relative to the rest of the undertray, there are some common misconceptions in the race car industry to how a diffuser works [10]. First is that the diffuser is what actually creates all of the downforce of the undertray and second is that the diffuser expands the air under the vehicle causing lowered pressure. Both of these concepts are false since the role of the diffuser is to slow the air under the vehicle back down to free stream to reduce the drag and increase the overall undertray efficiency, and as it is an open system with gaps around the edges it is unable to expand the air to cause a density change. With these things in mind, it is the diffuser angle and entrance location that drives the undertray performance.

The location of the entrance of the diffuser greatly affects where the low pressure occurs on the vehicle undertray. Data presented in Katz et al. [9] shows that there is a low pressure peak at the entrance location. To move the center of pressure of the undertray or the balance, the low pressure concentration can be moved by changing the location of the diffuser entrance more forward or rearward. For a race car, balance is critical to vehicle performance due to its effects on understeer and oversteer characteristics [9-11].

The angle of the diffuser relative to the ground affects the magnitude of downforce that is created [7, 9-12]. In general it is desired to have the highest angle without flow separation to generate maximum downforce. Once separation occurs the downforce is reduced and drag is greatly increased [10]. Two-dimensional simulation of diffuser angle shows maximum downforce is reached with an angle of only 5° [10]. However in experiments and 3-dimensional simulation there is another effect that is occurring that changes this. Starting at the diffuser entrance there is a vortex that forms that travels down the length of the diffuser. A vortex adds a rotational component to the velocity decreasing the pressure along its length. This vortex flow also adds energy to the flow and will delay separation allowing larger diffuser angles [7, 9-13]. Vortices can also be used on other parts of the undertray. Large vortex generators can be placed at the entrance of the undertray so that the vortices travel along the length of the vehicle, reducing the pressure and increasing downforce [7-12, 14]. These vortices can also be used along the sides of the undertray creating a "false seal" that also increases downforce [9].

All of these ideas can be used together to create an effective undertray that will produce large amounts of downforce with a relatively small increase to drag. The problem that occurs however is that there are complex interactions between all parts of the undertray as well as the car body, making design an uncertain area. Also, since racing is a competitive sport, most of the specific information about undertray design is not published.

3. CFD SIMULATION OF VEHICLE AERODYNAMICS

To help clear up the uncertainty that occurs from the interactions, CFD can be used to simulate the flow around the vehicle. The solution to the simulation can be used to observe pressure, velocity, downforce, drag and any other fluid properties of interest. Little work has been published using CFD to help design a Formula SAE car [2, 4-6, 15], and most of the work has been in 2-dimension simulation of airfoils.

There is, however, work published by other areas of the automotive industry, including motorsport, that has information on using CFD for aerodynamic design.

When setting up the CFD model there is usually geometry from a Computer Aided Design (CAD) package that is imported to use in the simulation. The complexity of the CAD geometry depends on what is of interest from the data and how much computing power is available. For external aerodynamics there is a "wind tunnel" box that is placed around the model. The entrance to the wind tunnel is placed a few car lengths ahead of the geometry and is considered a velocity inlet [16-19]. The exit to the wind tunnel is then placed many car lengths behind the geometry and is considered a pressure outlet [16-19]. Since the simulation will be of an open wheeled car the tires should be rotating and the ground set to a moving ground or frictionless. From the literature review it was found that the simulation of the tires was important to the accuracy of the rest of the model [5, 22-25]. For the CFD model it is critical that the tires are rotating and that the mesh will capture the behavior of the flow around them.

Since the airflow around a race car is very turbulent, a model needs to be selected for simulation of the turbulent flow. There appears to be four major turbulence models that are used in the automotive industries: k-ε, k-ω, Lattice-Boltzmann and Large Eddy Simulation (LES). Of these models the k-ε and k-ω are most widely used [2, 5-7, 17-21] with the k-ε said to be the most stable [21]. Direct Numerical Simulation (DNS) has also appeared in the automotive industry, however, it requires very large mesh numbers that take too much computing power and time for a traditional design turn around.

Mesh numbers vary widely depending on the simulation being done and the computational power available. For external aerodynamics it was found that a mesh on the order of a million cells is enough to predict aerodynamic forces [21]. For this size of mesh the lift coefficient can be predicted to within 5% of the true value while the drag seems have more error and can be as much as 20% different from the true value [5, 21].

These simulations can be very beneficial to the designer as they can give visual aids and data of the interactions that are occurring as well as flow trends that were not thought about before. Using this data the design can be iterated to conform to the designers' requirements.

Simulations were conducted with the goal of producing useful results that could predict the downforce within 10%, the drag within 20% and the center of pressure within 5%. These numbers were chosen based from the literature of what the CFD simulations would predict with the resources available. Turn-around time also needed to be relatively quick, on the order of a few days, so that the design could progress and be finalized in the typical three month design cycle of Oregon State University Formula SAE.

CFD simulations were set up using the commercial program Star-CCM+. Imported CAD geometry was used for all surfaces. A box was made around the geometry to serve as boundary conditions that included extra space 1 car length in front, 3 lengths rear and 1 length to the side, as seen in Figure 1. This extra room was to insure that the boundary conditions could be met with the geometry of the vehicle included. To reduce the total cell count, and therefore computing time, a symmetry plane was used down the center of the vehicle.

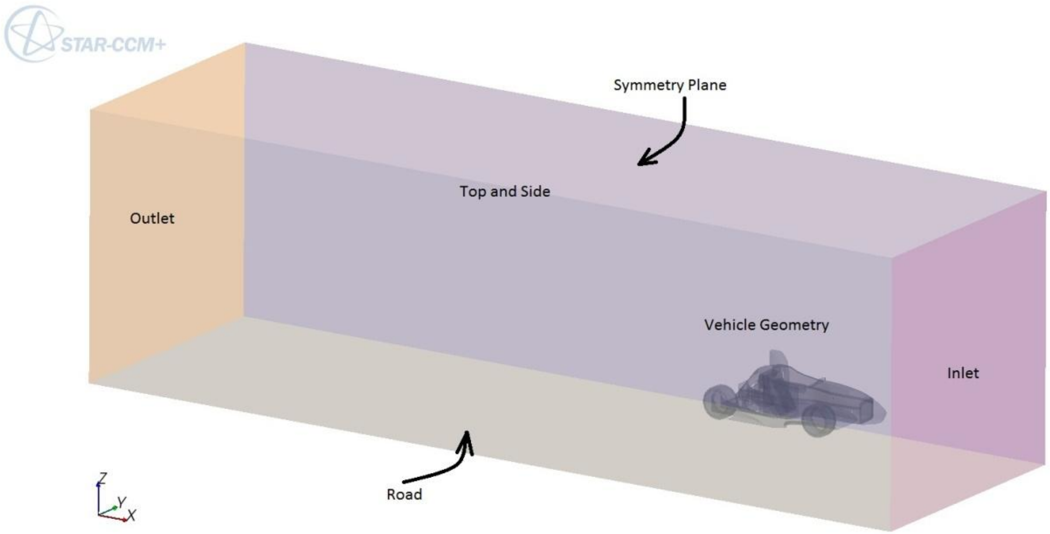


Figure 1 Simulation geometry with labels.

The boundary conditions were set as follows for the simulations unless otherwise noted:

Inlet: Velocity Inlet = 40 Miles per Hour

Outlet: Pressure Outlet = 0 Gauge pressure

Top and Side: Free Stream

Road: Moving no-slip wall

Also included in the simulation is rotating tires. As seen in the literature, rotating tires have a significant effect on the flow field around the vehicle.

Mesh size was chosen such that all detail of the vehicle was accurately captured and overall cell count was on the order of a million. A prism layer mesh of five layers was included to capture near-wall effects, this detail can be seen in Figure 2. The mesh was allowed to grow to large sizes away from the vehicle geometry to reduce cell count where the detailed solution was not important.

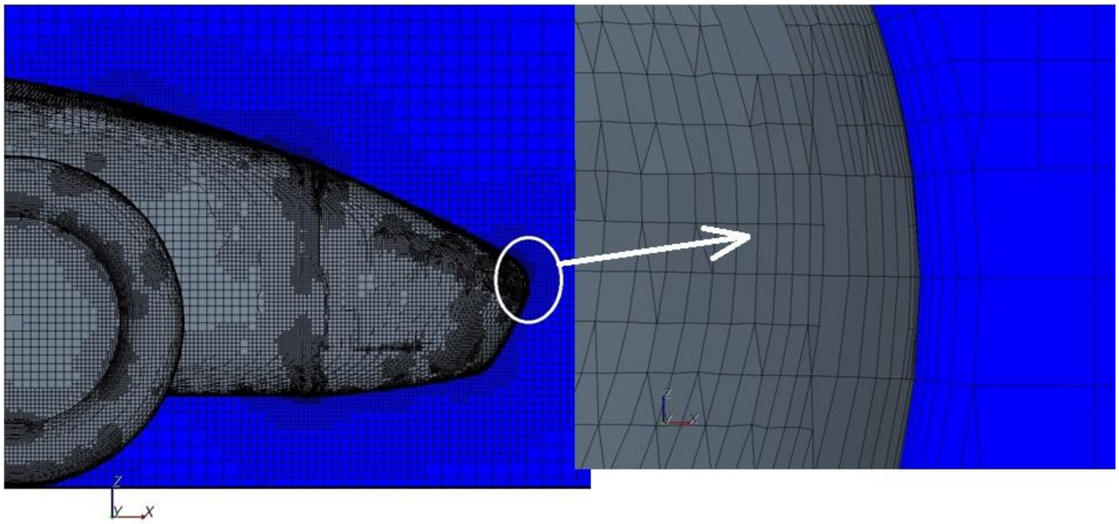


Figure 2 Prism layer mesh detail.

The k-ε turbulence model was selected based on the literature review. It has been shown to produce the accuracy of results desired with minimal computing power required and good stability.

Once the undertray design was close to being finalized, simulations at different speeds, 5° yaw, roll, and ground clearance were tested to see the effects on aerodynamic performance.

4. TESTING AND VALIDATION OF CFD SIMULATIONS

As with any simulation, the aerodynamic model is only useful when it is verified by physical testing. Once it is verified the designers have more confidence in using the simulation as a tool. For automotive external aerodynamics there are two major areas of testing: wind tunnel and on track [2-5, 9-11, 26-31]. Wind tunnel testing is broken down into two areas of full-scale testing and model testing. Full-scale testing uses a full-scale model of the car, or the car itself, to test lift and drag of the vehicle [2, 5, 27, 28]. Model testing, usually around 1/5 scale or larger, has its advantages and disadvantages. Scale models are usually cheaper to create parts for than a full-scale car and are easier to handle and store. The disadvantages are that you have to correct for the scale of the vehicle and that often the design used on the scale model will be conservative for the full scale car, meaning that it can be improved more [9]. Both wind tunnel methods have the advantage of doing flow visualization such as smoke, oil streaking and yarn tufts [5, 9-11] and also that consecutive runs can be completed quickly. Since the underbody of the car is of most interest, some sort of boundary layer control needs to be used. Katz et al. [9] gives the different methods of boundary control as: elevated ground plane, suction ahead of the model, suction plate under the model, tangential blowing, symmetry and moving ground plane or "rolling road."

On-track testing can be the most accurate and useful testing for aerodynamics. The first option would be to just test lap times on a closed course [3]. This would give the representative gain or loss of the aerodynamic changes to the vehicle. The down-side is the factor of the human driver that can skew the data, as well as the time on the track is usually expensive in terms of labor, track use and also wear on the car. Flow visualization for on track testing is also limited and can generally only be oil streaking or yarn tufts [10]. In order to properly capture the data the car must be equipped with data acquisition and the proper sensors to detect lift, drag and balance [9].

Using the sensors installed on the car straight line testing was conducted with the undertray. These runs were conducted to test aerodynamic effects at different speeds of the undertray, and also closely matched simulation testing for verification of the model.

For back-to-back comparisons with and without the undertray an asymmetric oval course was set up. The asymmetric oval consists of a 152 foot large diameter corner and a 30 foot small diameter corner with their centers placed 164 feet apart. A straight section connects one side and a three-cone slalom, spaced 39 feet apart, connects the other. A track map extracted from GPS data can be seen in Figure 3.

This track represents aspects of an endurance course, but with greatly reduced lap times so that the number of samples, or laps, can be increased.

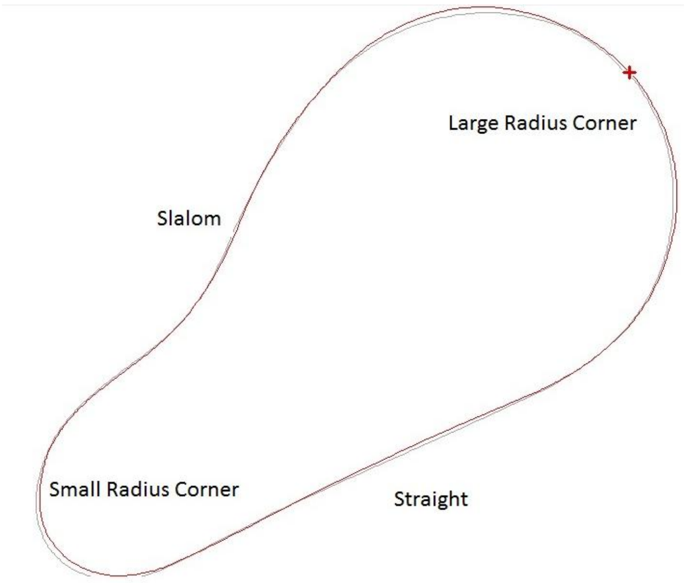


Figure 3 Track map of test asymmetric oval test track

5. RESULTS

5.1 CFD Simulation Results

A simulation of the vehicle was done to get a starting point to work from and establish baseline values of lift, drag and center of pressure location with no aerodynamic elements included. It was found that the vehicle had zero lift, 9 pounds of drag and a center of pressure located 56% rearward. This means that the vehicle is very balanced and will not produce a significant amount of lift or downforce with speed. Using this as a starting point, the undertray design was iterated to come to a final design. After 40 design iterations the final undertray can be seen in Figure 4.

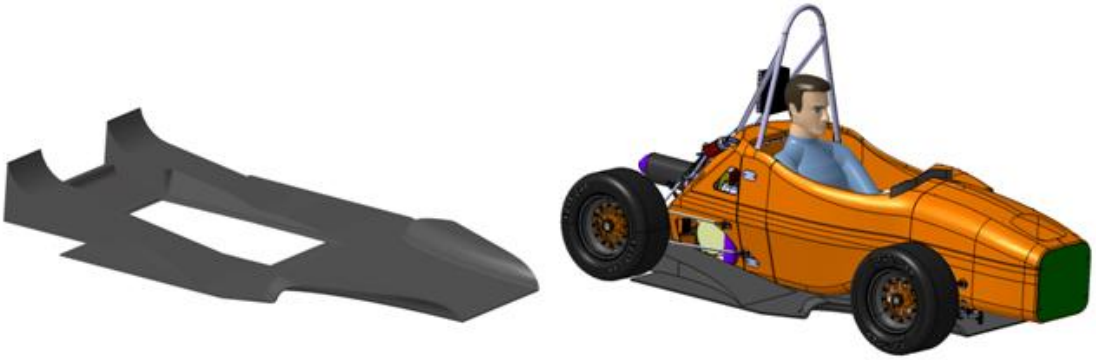


Figure 4 Final undertray design part alone and part on the car.

In combination with a 1.5" windshield gurney, it was predicted that the car with the undertray would produce 50 pounds of downforce, 11 pounds of drag and a center of pressure 55% rearward. Pictures from the simulation can be seen in Figure 5 and Figure 6. Figure 5 shows pressure contours of the full car as well as streamlines that travel under it. The effect of the windshield gurney can also be seen in the figure as a higher pressure zone in front of the gurney on the chassis. Figure 6 shows the bottom view of the undertray with pressure contours and on-surface stream lines, also known as “streak lines.” In the figure the vortex flow in the main tunnels can be seen as areas of low pressure inward of the rear wheels.

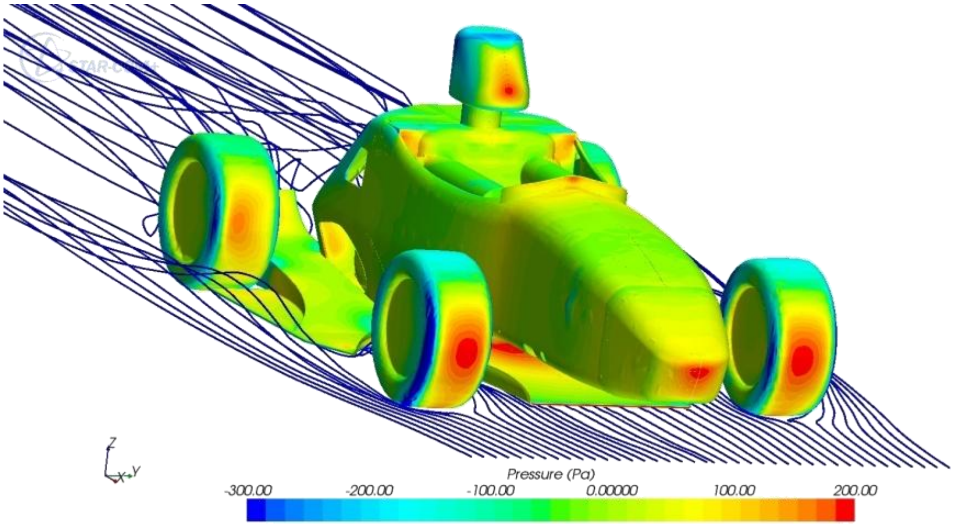


Figure 5 Full car with pressure contours and stream lines.

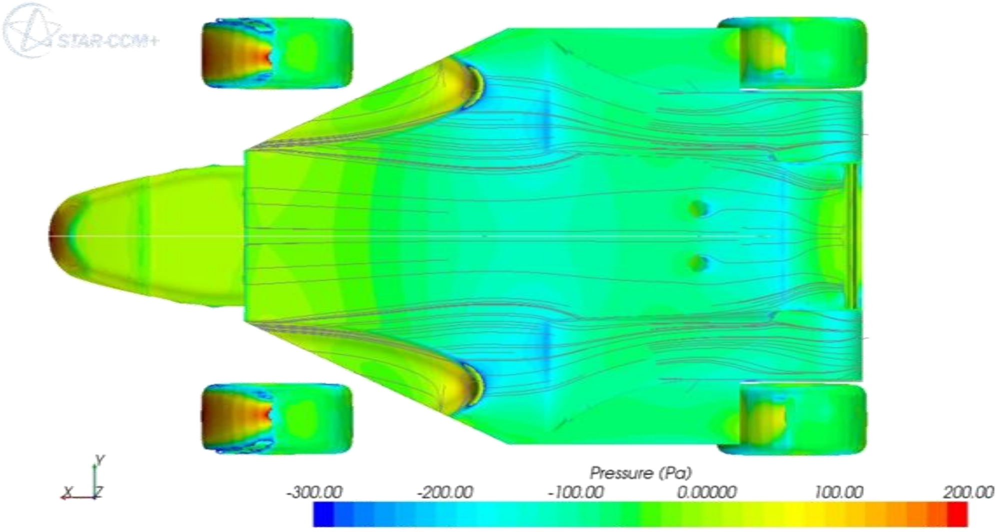


Figure 6 Bottom view of undertray with pressure contours and steak lines.

Figure 7 shows the simulated aerodynamic behavior of the vehicle through speeds that are expected to see during the Formula SAE competition. It can be seen that the downforce roughly follows the squared velocity relationship while the drag seems to not. This could be from the simulations lack to accurately model drag. It can also be seen that the center of pressure migrates forward with speed. This means that the car could become unstable at higher speeds and the center of pressure may need to be adjusted with add-ons such as gurney flaps on the rear of the diffusers or dive plates in the front. Also included in the figure are the baseline values of the vehicle without any aerodynamic elements. The simulations show that there is a significant increase in downforce with only a small increase in drag.

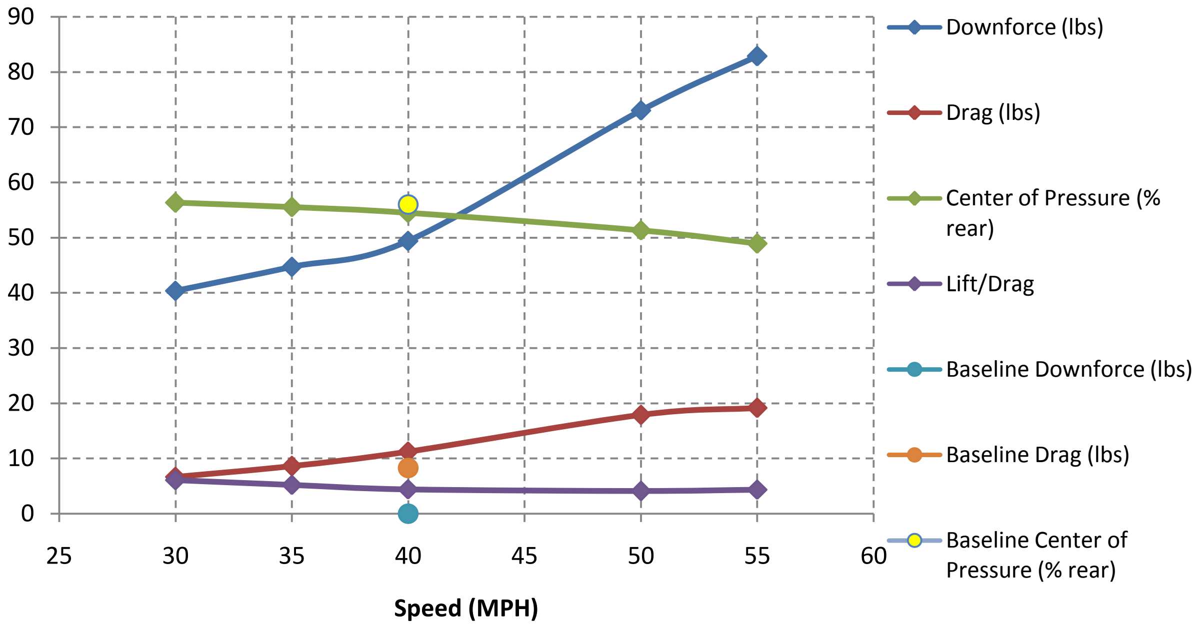


Figure 7 Graph of aerodynamic behavior at different speeds.

The effect of vehicle ride height can be seen in Figure 8. It can be seen that with small changes in ride height there are large changes in aerodynamic loads. Since the car is limited by rules to have 1" minimum static ground clearance, it will be set to the minimum in order to get the most out of the undertray.

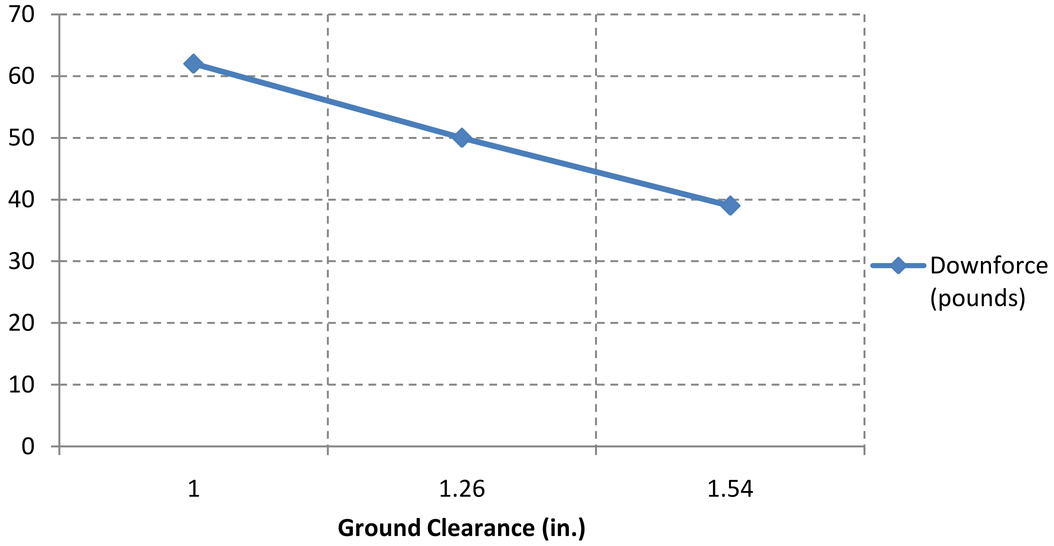


Figure 8 Downforce with change in ground clearance.

When the vehicle is cornering, the body will have a certain yaw angle and will also have some roll. Simulations were done at 5° yaw with no roll and also at 5° yaw with 1° roll. Figure 9 shows the bottom view of the pressure contours and streak lines of the 5° yaw simulation with the car turning to the right. For this case the downforce increased to 62 pounds and it can also be seen in the figure that it will cause a roll moment on the car that will want to roll the car into the corner. This is beneficial to the vehicle performance as it will distribute more load to the inside tires, increasing traction. To test the effect of roll during cornering, a yaw simulation with 1° body roll was completed. This can be seen in Figure 10. From this it was found that 1° of roll reduced the downforce by 6%. This means that from an aerodynamic standpoint, less roll is desired to maximize downforce.

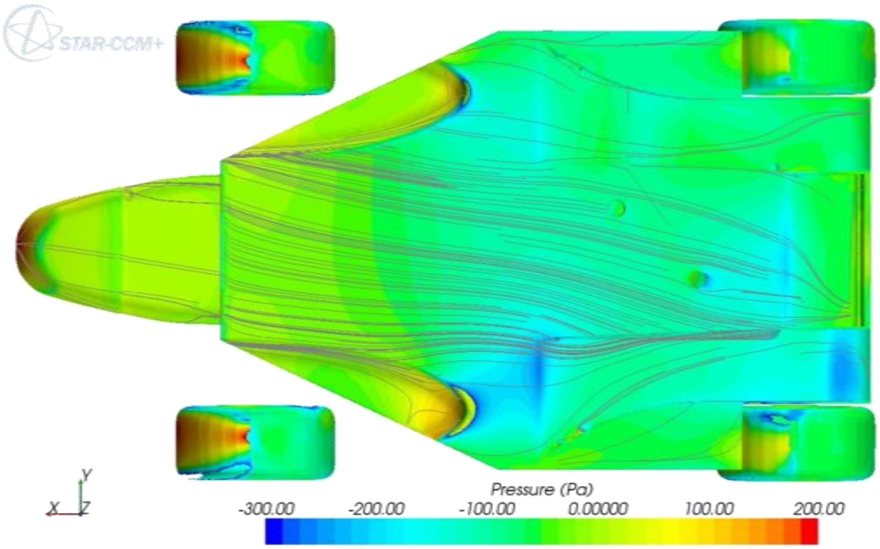


Figure 9 Simulation of 5 degree yaw with no roll including pressure contours and streak lines.

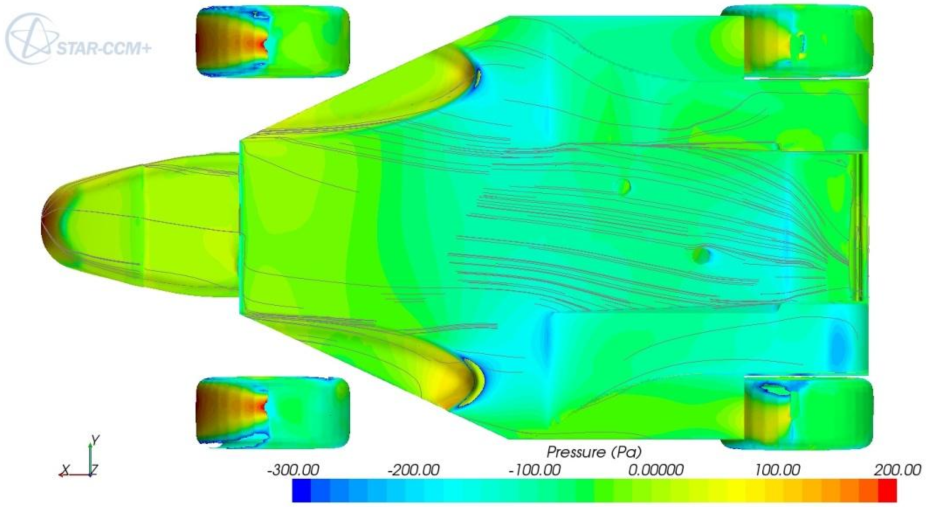


Figure 10 Simulation of 5 degree yaw with 1 deg roll with pressure contours and streak lines

To match the setup that occurred on the test day, a final simulation was conducted without a windshield gurney. This resulted in 29 pounds of downforce, 11.2 pounds of drag and 54% rearward location of the center of pressure.

5.2 On-Track Testing Results

Data from the straight line tests can be seen in the figures below. Full traces of the data can be seen in Figure 11 with total spring force shown as the red trace in the third graph down and vehicle speed as the fourth graph down. The spring force was calculated from linear potentiometer data and known spring rates. Total spring force does indicate that downforce is being produced with speed, however the data gathered is also very noisy. This noise could be produced from vibrations such as the engine or simply road noise from the track. A smaller portion of the data can be seen in Figure 12 where the two runs at 40 mph were conducted. In this graph you can see that the total spring force varies around 22 pounds of downforce for the given speed. Comparing the first and second traces of Figure 11 and Figure 12 shows the downforce distribution front to rear, or center of pressure location. According to the data, the majority of the downforce seems to be produced at the rear. This can lead to a stabilizing effect at higher speeds but also means that the car will become less responsive and have a tendency to understeer.

A plot of the downforce with speed can be seen in Figure 13. This plot was extracted from the data in Appendix A.1. Also included is the predicted downforce of the undertray. The error of the predicted versus the measured at 40 mph comes out to 31%, which is higher than the 10% that was desired for the simulations.

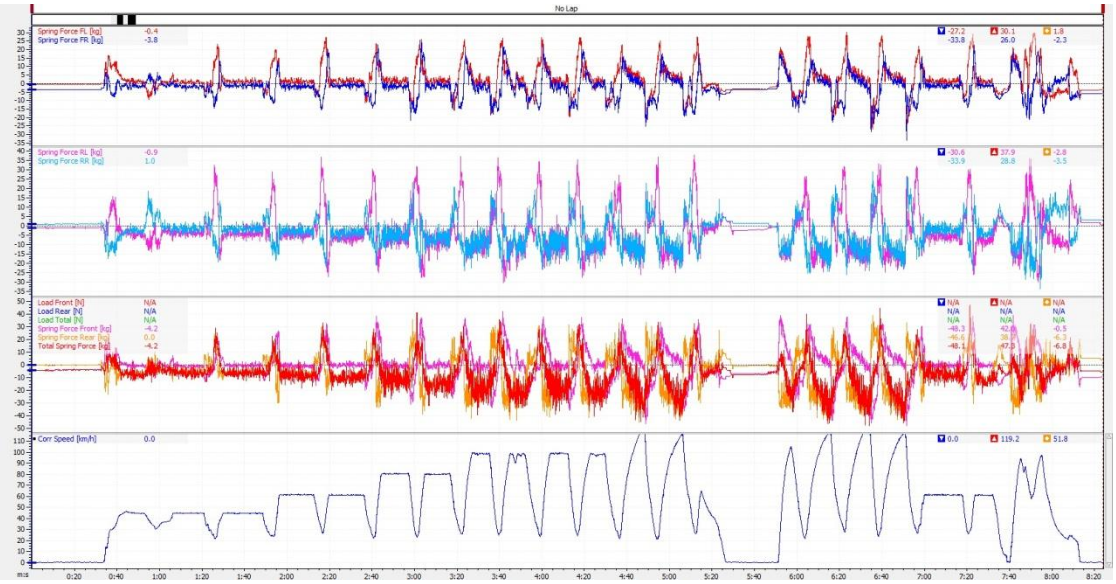


Figure 11 Full data from straight line test including spring force and speed.

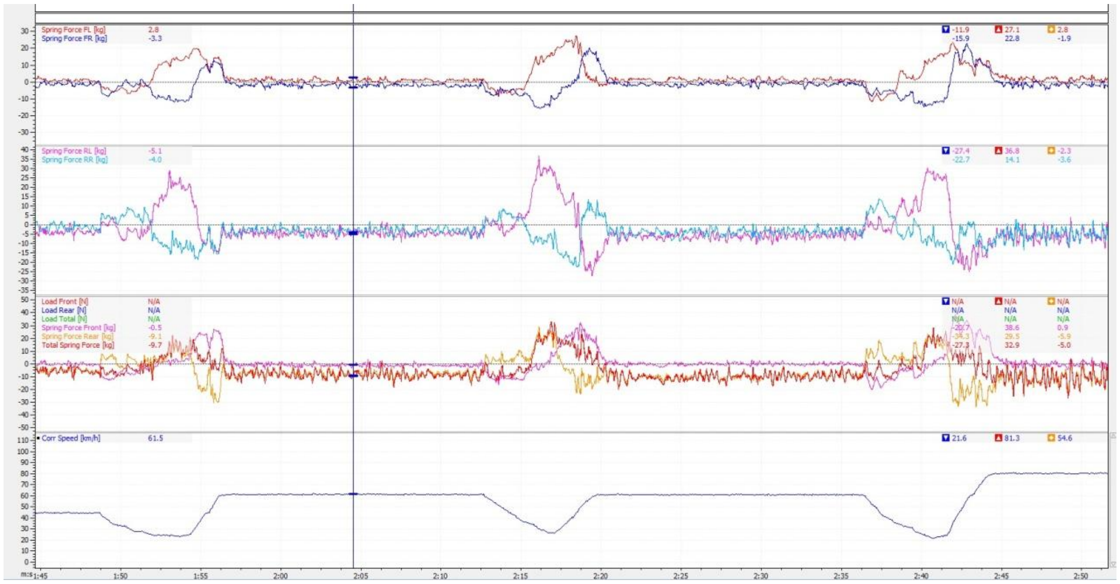


Figure 12 Data traces for 40mph straight line test.

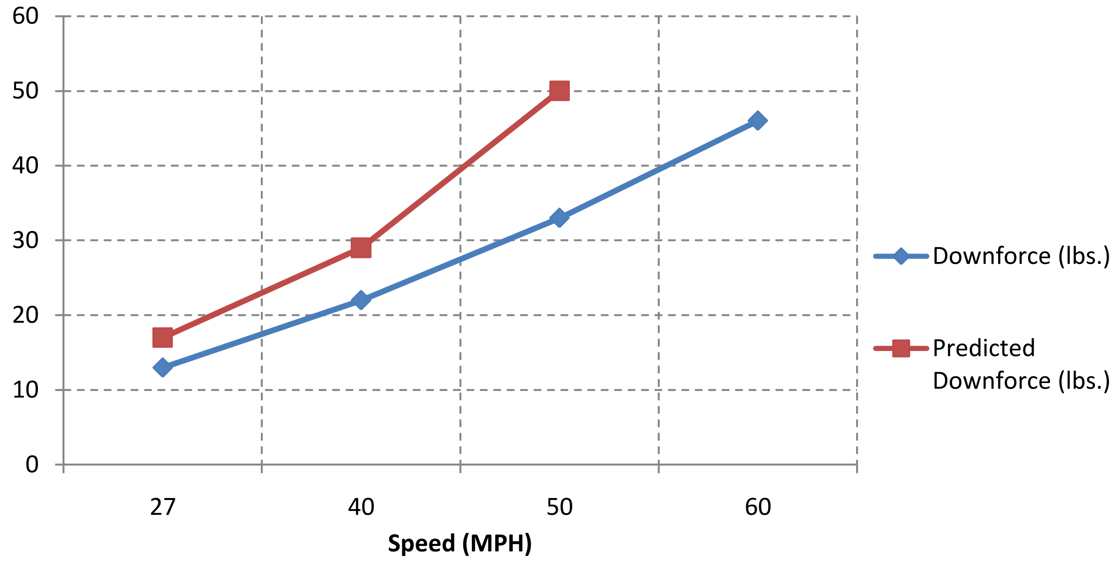
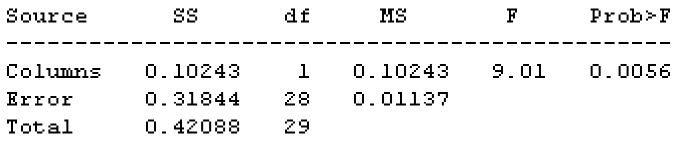


Figure 13 Extracted downforce from straight line test.

Lap times for the asymmetric oval tests are included in Appendix A.2. These lap times include 20 laps for each run, however, for statistical analysis the first 5 laps were removed for driver warm-up. Analysis was run using MATLAB's one-way analysis of variance (ANOVA) for a 99% confidence level. The test calculated a p-value of 0.0056, seen below in Table 1, which is less than 0.01. This means that we reject the null hypothesis and conclude that the lap times are statistically different between the two tests. From Figure 14 it can be seen that the lap times with the undertray are faster than without. The difference between the means of the two tests is 0.1169, meaning that the undertray improved the performance by 1%.

Table 1 ANOVA table for lap time analysis



```
Source        SS       df      MS        F      Prob>F
------------------------------------------------------
Columns    0.10243      1   0.10243    9.01    0.0056
Error      0.31844     28   0.01137
Total      0.42088     29
```

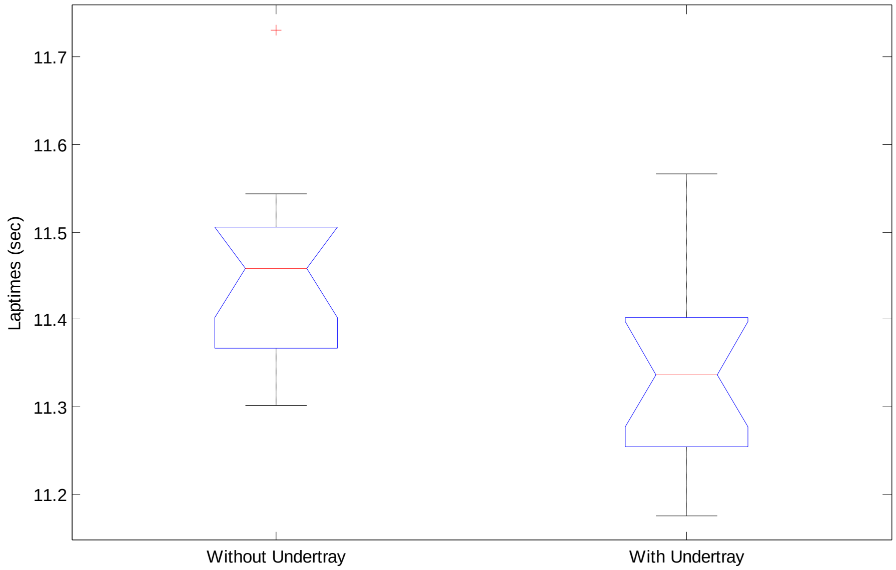


Figure 14 Box plot from ANOVA analysis

6. DISCUSSION

The CFD simulations presented here give a good starting point for future work. While a 31% error from predicted to measured data is not ideal, on-track testing can be difficult to get a controlled test. Small variances in things like wind speed, mounting location and vibration can skew measurements. I suggest that the next step should be to perform a more thorough mesh analysis and validate it with scale model wind tunnel tests.

One aspect of automotive racing is that it is an extremely competitive sport where the difference of a few tenths of a second can be the difference between first and third place. While a 1% improvement in lap time may not sound like that much of an improvement at first glance, when looking at a 60 second lap it is a 0.6 second lap time improvement. For this case the 1% improvement that we want is right on the edge of what is measurable, making these improvements difficult to prove.

7. CONCLUSION

Aerodynamic improvements have become increasingly popular to the Formula SAE audience. These improvements allow the vehicles more traction while adding little weight to the system. Given that automotive racing is a competitive sport, exact details on the dynamics and designs of an undertray are not well documented. Through research a basic shape for an undertray can be constructed that can then be iterated to generate more downforce using CFD simulation as a tool. On-track testing with back-to-back comparisons and vehicle sensors can verify the improvement of the vehicle track performance and also the downforce produced. While there is a 31% error between the predicted downforce and the measured, the improvement of 1% in lap time for the vehicle is a significant gain and could provide the edge over the competition.

BIBLIOGRAPHY

1. Wordley, S.J., and Saunders, J.W., Aerodynamics for Formula SAE: Initial Design and Performance Prediction, SAE Paper 2006-01-0806, 2006.

2. Wordley, S.J., and Saunders, J.W., Aerodynamics for Formula SAE: A Numerical, Wind Tunnel and On-Track Study, SAE Paper 2006-01-0808, 2006.

3. Wordley, S.J., Pettigrew, J., and Saunders, J.W., Aerodynamics for Formula SAE: On-Track Performance Evaluation, SAE Paper 2007-01-0897, 2007.

4. Mckay, N.J., and Gopalarathnam, A., The Effects of Wing Aerodynamics on Race Vehicle Performance, SAE Paper 2002-01-3294, 2002.

5. Desai, S., Leylek, E., Betty Lo, C., Doddegowda, P., Bychkovsky, A., and George, A.R., Experimental and CFD Comparative Case Studies of Aerodynamics of Race Car Wings, Underbodies with Wheels, and Motorcycle Flows, SAE Paper 2008-01-2997, 2008.

6. Doddegowda, P., Bychkovsky, A.L., and George, A.R., Use of computational Fluid Dynamics for the Design of Formula SAE Race Car Aerodynamics, SAE Paper 2006-01-0807, 2006.

7. Zhang, X., Toet, W., and Zerihan, J., Ground Effect Aerodynamics of Race Cars, *Applied Mechanics Reviews*, 2006.

8. Katz, J., and Garcia, D., Aerodynamic Effects of Indy Car Components, SAE Paper 2002-01-3311, 2002.

9. Katz, J., “Race Car Aerodynamics”, Bentley Publishers, Massachusetts, ISBN 978-0-8376-0142-7, 2006.

10. McBeath, S., “Competition Car Aerodynamics”, Haynes Publishing, California, ISBN 978-1-84425-230-5, 2006.

11. Haney, P., and Braun, J., “Inside Racing Technology: Discussions of Racing Technical Topics”, TV Motorsports, California, ISBN 0-9646414-0-2, 1995.

12. Ruhrmann, A., and Zhang, X., Influence of Diffuser Angle on a Bluff Body in Ground Effect, *Transactions of the ASME*, 2003, doi: 10.1115/1.1537252.

13. Mahon, S., Zhang, X., and Gage, C., The Evolution of Edge Vortices Underneath a Diffuser Equipped Bluff body, University of Southampton.

14. Katz, J., and Morey, F., Aerodynamics of Large-Scale Vortex Generator in Ground Effect, *Journal of Fluids Engineering* 071101-1, 2008, doi: 10.1115/1.2948361.

15. Wakabayashi, M., Komuro, K., Pedoussaut, O., Oguri, Y., and Suzuki, T., Improving Vehicle Performance by Aerodynamic Devices in Formula SAE, SAE Paper 2009-32-7103, 2009.

16. Brzustowicz, J.P., Lounsberry, T.H., and La Rode, J.E., Experimental & Computational Simulations Utilized During the Aerodynamic Development of the Dodge Intrepid R/T Race Car, SAE Paper 2002-01-3334, 2002.

17. Jiang, L., Liu, H., Chai, G., Jiang, G., and Lin, W., Effect of Automotive Headlamp Modeling on Automotive Aerodynamic Drag, *IEEE* 978-1-4244-3291-2, 2008.

18. Muyl, F., Dumas, L., Herbert, V., Hybrid method for aerodynamic shape optimization in automotive industry, *Computers & Fluids* 33 (2004) 849-858, 2004.

19. Huminic, A., and Huminic, G., On the Aerodynamics of the Racing Car***s***, SAE Paper 2008-01-0099, 2008.

20. Wang, D., and Yang, Z., CFD Predictions of Minivan's Front-end Flow Performance, *IEEE* 1-4244-0759-1, 2006.

21. Dhaubhadel, M.N., Review: CFD Applications in the Automotive Industry, *Journal of Fluids Engineering*, 1996.

22. Mears, A.P., and Dominy, R.G., Racing Car Wheel Aerodynamics- Comparisons between Experimental and CFD derived Flow-Field Data, SAE Paper 2004-01-3555, 2004.

23. Mears, A.P., Dominy, R.G., and Sims-Williams, D.B., The Air Flow About an Exposed Racing Wheel, SAE Paper 2002-01-3290, 2002.

24. Knowles, R., Saddington, A., and Knowles, K., On the Near Wake of Rotating, 40%-Scale Champ Car Wheels, SAE Paper 2002-01-3293, 2002.

25. Kellar, W.P., Pearse, S.R.G., and Savill, A.M., Formula 1 Car Wheel Aerodynamics, *Sports Engineering* 203-212, 1999.

26. Lee, S., Landman, D., Jordan, J., Watkins, A., Leighty, B., Oglesby, D., and Ingram, J., Perfomance Automotive Applications of Pressure-Sensitive Paint in the Langley Full Scale Tunnel, SAE Paper 2002-01-3291, 2002.

27. Landman, D., and Koster, E., A Full-Scale Wind Tunnel Test of a Short Track Race Car, SAE Paper 2000-01-3548, 2000.

28. Landman, D., Simpson, J., Hall, B., and Sumner, T., Use of Designed Experiments in Wind Tunnel Testing of Performance Automobiles, SAE Paper 2002-01-3313, 2002.

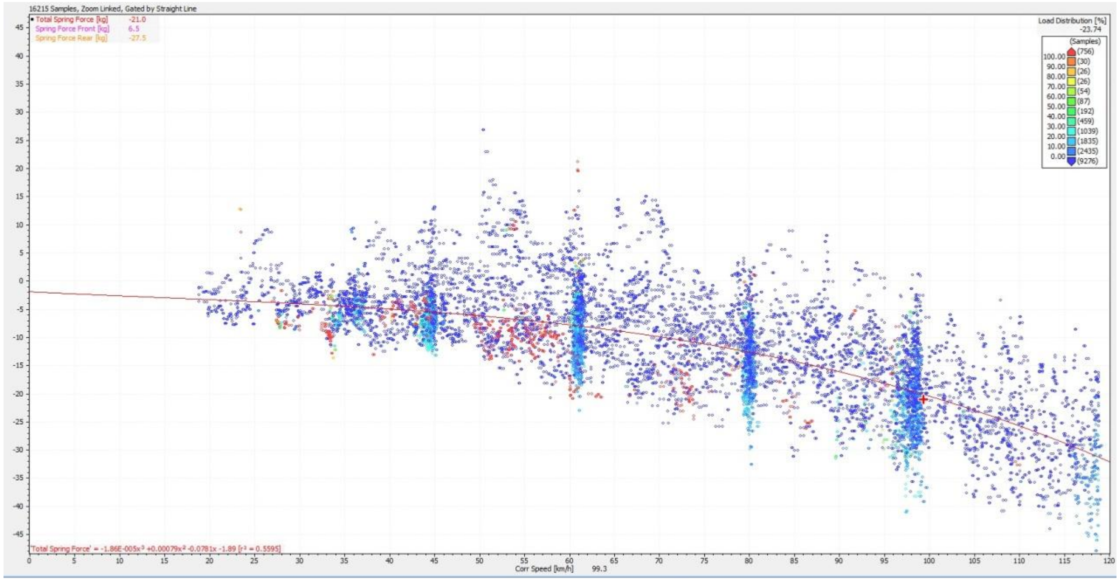
29. Duell, E., Conner, C., Martindale, B., Walter, J., and Arnette, S., Advantages of Adaptive Wall Wind Tunnel Technology: A CFD Study for Testing Open Wheel Race Cars, SAE Paper 2007-01-1048, 2007.

30. Candelpergher, A., Gadola, M., and Adami, R., The Impact of Non-Linear Aerodynamics on Racecar Behavior and Lap Time Simulation, SAE Paper 2002-01-3332, 2002.

31. Buckley, F.T., ABCD – An Improved Coast Down Test and Analysis Method, SAE Paper 950626, 1995.

A APPENDIX

A.1 Downforce versus Speed Data



A.2 Asymmetric Oval Lap Times

Without Undertray	With Undertray
11.949	11.922
11.907	11.665
11.627	11.85
11.516	11.827
11.461	11.744
11.355	11.566
11.467	11.484
11.731	11.411
11.401	11.331
11.512	11.337
11.484	11.307
11.449	11.419
11.329	11.216
11.341	11.339
11.544	11.37
11.481	11.262
11.301	11.226
11.431	11.175
11.458	11.252
11.541	11.377