

Turbulence Models and Model Closure Coefficients Sensitivity of NASCAR Racecar RANS CFD Aerodynamic Predictions

Chen Fu, Mesbah Uddin, and Clay Robinson
 UNC Charlotte Motorsports Engineering

Arturo Guzman and David Bailey
 FCA US LLC

ABSTRACT

Cost benefit and teraflop restrictions imposed by racing sanctioning bodies make steady-state RANS CFD simulation a widely accepted first approximation tool for aerodynamics evaluations in motorsports, in spite of its limitations. Research involving generic and simplified vehicle bodies has shown that the veracity of aerodynamic CFD predictions strongly depends on the turbulence model being used. Also, the ability of a turbulence model to accurately predict aerodynamic characteristics can be vehicle shape dependent as well. Modifications to the turbulence model coefficients in some of the models have the potential to improve the predictive capability for a particular vehicle shape. This paper presents a systematic study of turbulence modeling effects on the prediction of aerodynamic characteristics of a NASCAR Gen-6 Cup racecar. Steady-state RANS simulations are completed using a commercial CFD package, STAR-CCM+, from CD-Adapco. Three turbulence models are included in this study: the realizable and AKN variants of the $k - \epsilon$ model, and the SST $k - \omega$ model. The a_1 coefficient for the SST $k - \omega$ model is slightly modified in order to investigate the sensitivity of this modeling constant on the veracity of predictions. The aerodynamic characteristics investigated include the force and moment coefficients, and front-to-rear downforce balance. In addition, the turbulence model dependence of the flow around the vehicle, particularly at some key locations in the wake region, will be analyzed. Although the flow features in the wake of this racecar show a discernable dependence on the choice of the turbulence model, this difference did not translate into a significant difference in the prediction of force coefficients.

CITATION: Fu, C., Uddin, M., Robinson, C., Guzman, A. et al., "Turbulence Models and Model Closure Coefficients Sensitivity of NASCAR Racecar RANS CFD Aerodynamic Predictions," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1547.

INTRODUCTION

Stock car racing is the most popular motorsport in North America, especially in the United States. The National Association of Stock Car Racing (NASCAR) is the world's largest sanctioning body for stock car races, and its Sprint Cup series is the highest level professional racing series in North America. Since their introduction, the current Sprint Cup racecars, popularly known as the Cup cars, have evolved through 6 generations. The Generation 6 car, shortened to Gen-6, is the common name for the Sprint Cup car that has been used since 2013. Although the new generation Cup cars are designed to make their body style close to the manufacturers' street legal versions, and the races are still run mostly on traditional oval tracks, like all other professional races, aerodynamics is very important in the stock car racing industry as well.

During recent years, with a rapid development of computational resources, Computational Fluid Dynamics (CFD) has become an indispensable tool for racing aerodynamicists. Compared to wind tunnel testing, CFD simulation has its own advantage of providing a thorough description of the flow field interacting with a racecar, instead of delivering only the force and moment coefficients. With

proper modeling, CFD now can produce results that are comparable with the accuracy and repeatability of wind tunnel tests with significantly more details of the overall flow-field. However, accurately simulating air flow by solving differential equations that govern the flow around an object requires time and huge computational resources. Because of the fast working pace and tight racing schedule, many race teams devote much effort to the development of accurate CFD methods with faster turnaround times.

The Reynolds-averaged Navier-Stokes equation (RANS) approach is among one of the most popular and efficient CFD approaches for ground vehicle simulations, because of its low computational overhead and the large potential benefits from quick turnaround times. However, since the RANS equations only govern the mean flow, the effects of the nonlinearity and the fluctuations from the turbulence in reality must be modeled [1]. Turbulence modeling has been thoroughly investigated with both internal and external flows in the past, mostly with simple flow structures, such as channel flows[2, 3] and flows past bluff bodies [4], to cite but a few. In terms of automotive applications, a vast majority of the published validation works for turbulence models are confined to simple automotive

models, like Ahmed body[5, 6], or DrivAer model [7]. Very little research has been done that has addressed comparisons among turbulence models directly applied to race vehicles. To the best of the authors' knowledge none such research exists for a Gen 6 NASCAR Cup car at this time.

So far, published numerical studies focusing on the aerodynamics of NASCAR Cup cars are extremely limited and outdated, see reference [8, 9, 10]. The last reported CFD work is presented by Singh [11] using a Car of Tomorrow (COT) model, which has been long replaced on track by the Gen-6 car. Moreover, the main objectives of those studies are to evaluate the car performance under different conditions, such as different designs or different ground simulations, using computational modeling as a tool. None of them investigated the CFD tool itself, for example, which turbulence modeling has the best correlation with the real world test data, or in which areas these RANS based turbulence models needed improvements for better predictions, or how the predicted flow field changes with the selection of turbulence models, and so on.

This paper presents a systematic study on the effects of different turbulence models on the CFD predictions of the aerodynamic characteristics of a full-size detailed Gen-6 Sprint Cup car using a commercial CFD code (STAR-CCM+ version 11.04.010 from CD-Adapco). The simulations are run in steady-state using unstructured hexahedral cells. The prediction veracity of three commonly used two-equation turbulence models, realizable and AKN $k - \varepsilon$, and SST $k - \omega$ are investigated. The predicted results are compared against the test data from Aerodyn Wind Tunnel for drag and lift coefficients, front-to-rear downforce distribution and downforce to drag ratio. As described later, Aerodyn is a non-rolling road open return type wind tunnel housed in a large isolated room, specifically designed to test stock-car type race cars. Accumulated drag and lift coefficients in the longitudinal direction from these turbulence models are compared. In particular, the flow fields in the rear portion of the race car are analyzed from multiple perspectives in order to discern how these models differ in elucidating the separated flows in the wake region.

COMPUTATIONAL METHOD & SIMULATION SETUP

To obtain the steady-state continuity and Reynolds-Averaged Navier-Stokes (RANS) equations for incompressible turbulent flows, the Navier-Stokes equations for instantaneous velocity and pressure fields are decomposed into mean and fluctuating components using the Reynolds decomposition. The resulting mass and momentum conservation equations, known as the continuity and momentum (Navier-Stokes) equations, can be represented by Equations 1 and 2 below, in Einstein indicial notation where repeated indices are summed over the possible values:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} [-\bar{p} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j}] \quad (2)$$

These equations for the mean quantities are essentially identical to the original instantaneous equations, except for the additional term $-\rho \overline{u'_i u'_j}$ in the momentum transport equation. These terms constitute a tensor, known as the Reynolds stress tensor. It has a total of nine components, six of which are independent. These six additional stress terms make the number of unknowns larger than the number of available mathematical equations to solve them. This gives rise to what is known as the turbulence closure problem. To account for this, fluid-dynamicists resort to what is traditionally known as turbulence modeling. Broadly speaking, two basic approaches can be used to model the Reynolds stresses in terms of mean flow quantities and to provide closure of the governing equations: (a) eddy viscosity models, and (b) the Reynolds stress transport models. Eddy viscosity models use the concept of a turbulent viscosity μ_p , and model the turbulent stresses in a way analogous to laminar flows. Most of the turbulence models currently used by CFD engineers in engineering applications are based off of the eddy viscosity models, of which, $k - \varepsilon$ and $k - \omega$ variants are the most widely used ones.

Turbulence Models

Realizable Two-Layer $k - \varepsilon$

Compared to the original standard $k - \varepsilon$ model proposed by Launder and Spalding [12], the realizable $k - \varepsilon$ model of Shih et al. [13] introduced a new realizable eddy viscosity formulation. The eddy viscosity formulation of Launder and Spalding is shown in Equation 3:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

While the original model treats the closure coefficient C_μ in Equation 3 as a constant throughout the flow field, this coefficient is modeled in the realizable formulation as a function of the flow characteristics as given by Equation 4.

$$C_\mu = \frac{1}{A_0 + A_s \frac{\kappa U^*}{\varepsilon}} \quad (4)$$

In Equation 4, $A_0 = 4.04$, $A_s = \sqrt{6} \cos \phi$, $\phi = \frac{1}{3} \cos^{-1} \sqrt{6} W$, $W = (S_{ij} S_{ji} S_{k\ell}) / \bar{S}$ and $\bar{S} = \sqrt{S_{ij} S_{ji}}$, where S_{ij} is the mean rate of strain tensor. The transport equation for the turbulent kinetic energy k is given by,

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (5)$$

The dissipation rate equation, which is also different from the standard $k - \varepsilon$ formulation, as given in Equation 6, is based on the dynamic equation of the mean-square vorticity fluctuation at large turbulence Reynolds numbers.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = C_1 \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (6)$$

In addition to the above changes, the Shih et al. [13] formulation incorporates the so called realizability criterion, hence the name. The realizability criterion prevents a turbulence model from generating non-physical results by ensuring that the normal Reynolds stresses must be non-negative, and the Schwarz inequality must be satisfied between the fluctuating quantities. According to Shih et al. [13], the realizable $k - \varepsilon$ model performs better with flows involving complex geometries or bluff bodies where strong flow separation and recirculation occur. The realizable two-layer $k - \varepsilon$ model used in this study combines realizable $k - \varepsilon$ model with two-layer wall modelling approach, which allows the $k - \varepsilon$ model to be applied in the viscous sublayer [14].

Abe-Kondoh-Nagano (AKN) $k - \varepsilon$ Low-Re

The AKN $k - \varepsilon$ turbulence model of Abe-Kondoh-Nagano [16] is based off of the Nagano-Tagawa $k - \varepsilon$ model [15]. The main improvement of the AKN model is the usage of the Kolmogorov velocity scale u_η , $u_\eta \equiv (\nu \varepsilon)^{1/4}$, instead of the friction velocity u_τ to account for the near-wall effects in attached and detached flows [15]. Damping functions are introduced in the dissipation equation and in the definition of eddy viscosity ν_t .

$$\nu_t = C_\mu f_\mu \frac{k^2}{\varepsilon} \quad (7)$$

$$f_\mu = (1 - e^{\frac{y^+}{14}})^2 \left[1 + \frac{5}{3} e^{-\left(\frac{Re_t}{200}\right)^2} \right] \quad (8)$$

With reevaluated model coefficients, including $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k , and σ_ε , the AKN model is shown to successfully predict the separating and reattaching flows downstream of a backward-facing step, which involve most of the essential physics of complex turbulent flows [15]. These coefficients are related to each other satisfying the relations:

$$C_{\varepsilon 1} = C_{\varepsilon 2} - \frac{\kappa^2}{\sigma_\varepsilon \sqrt{C_\mu}} \quad (9)$$

$$\sigma_k \approx \sigma_\varepsilon \quad (10)$$

A summary of the model constants for realizable $k - \varepsilon$ and AKN $k - \varepsilon$ turbulence models is presented in Table 1.

The low-Re approach for wall modeling used in this study resolves the viscous sublayer by applying damping functions to the model coefficients. These damping functions modulate the coefficients as functions of the turbulence Reynolds number, and incorporates the effect of the wall distance as well.

Table 1. Constants for the $k - \varepsilon$ turbulence models

Model	C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε
RKE	0.09	1.44	1.9	1.0	1.2
AKN	0.09	1.5	1.9	1.4	1.4

The Shear-Stress Transport (SST) $k - \omega$

The $k - \omega$ model replaces the dissipation rate ε with the specific dissipation rate ω , which is defined as $\omega \equiv \bar{\varepsilon}/k$. Compared to the standard $k - \omega$, the SST $k - \omega$ turbulence model, proposed by Menter and his coworkers [17, 18, 19], has an additional non-conservative cross-diffusion term containing $\Delta k \cdot \Delta \omega$ in the ω transport equation. By using a blending function that includes the cross-diffusion term far from the wall but not near the wall, the SST model effectively blends a $k - \varepsilon$ model in the far-field free stream while keeping the $k - \omega$ boundary layer calculation advantages in the near wall region. The expressions for the eddy viscosity ν_t and the transport equations are given in Equations 11 to 18.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_i} = \widetilde{P}_k - \beta^* k \omega + \frac{\partial}{\partial x_i} \left[\left(\nu + \sigma_k \nu_t \right) \frac{\partial k}{\partial x_i} \right] \quad (11)$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_i} = \alpha \frac{1}{\mu_t} \widetilde{P}_k - \beta \omega^2 + \frac{\partial}{\partial x_i} \left[\left(\nu + \sigma_\omega \nu_t \right) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (12)$$

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, S F_2)} \quad (13)$$

$$S = \sqrt{2 S_{ij} S_{ij}} \quad (14)$$

$$P_k = \nu_t \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \rightarrow \widetilde{P}_k = \min(P_k, 10 \beta^* k \omega) \quad (15)$$

$$F_1 = \tanh \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 \nu}{y^2 \omega} \right), \frac{4 \rho \sigma_{\omega 2} k}{C D_{k \omega} y^2} \right] \right\} \quad (16)$$

$$F_2 = \tanh \left[\left[\max \left(\frac{2 \sqrt{k}}{\beta^* \omega y}, \frac{500 \nu}{y^2 \omega} \right) \right]^2 \right] \quad (17)$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right) \quad (18)$$

Here, F_1 and F_2 are blending functions; α , β , β^* , σ_k , σ_ω and $\sigma_{\omega 2}$ are closure coefficients computed by a blend of the corresponding constants from the $k-\varepsilon$ and the $k-\omega$ models via $\alpha = \alpha_1 F_1 + \alpha_2 (1 - F_1)$, etc [20]. The α_1 , constant in ν_t is initially set to 0.31 per STAR-CCM+ user manual. A production limiter is used in the SST model to prevent the build-up of turbulence in stagnation regions.

Vehicle Model

As presented in [Figure 1](#), the vehicle model used in this study is a detailed full-scale Gen-6 Cup car with front and rear ride heights of 76.5 mm and 154.5 mm respectively. The front grille opening is fully taped in the CFD model to simulate a closed-grille configuration. The vehicle geometry is cleaned and prepared in another commercial package, ANSA, from BETA CAE, then imported to STAR-CCM+ for volume meshing, simulating, and post-processing operations.

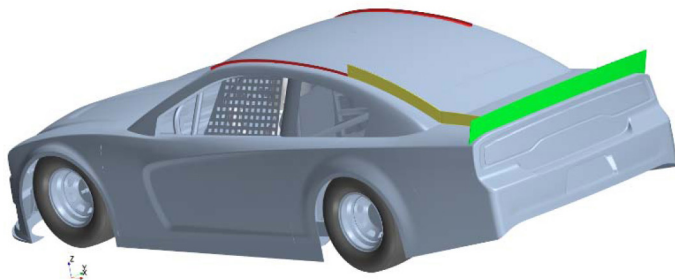


Figure 1. CFD vehicle model.

Simulation Setup

Computational Domain

The computational domain is created to be a free flow test environment with negligible block interference [21]. The rectangular test section is 55 m long, 90 m wide and 45 m high, which results in an approximate blockage ratio of 0.05%.

Mesh

Because of the complexity of the vehicle geometry and the desire for an efficient simulation time, trimmer cells are used for the computational domain discretization. As shown in [Figure 2](#), a slow cell growth rate is chosen for a smoother transition from finer to coarser cells. Prismatic cells with a geometric growth rate are used to resolve the near-wall high gradient flow field over the vehicle surface. The resulting mesh consisted of 113 million total cells. Later simulations show that more than 95% of the surface cells have wall-node $y^+ \leq 1$, and less than 0.25% have $y^+ > 5$, which makes the mesh acceptable even for the turbulence models which require the viscous sublayer to be well resolved.

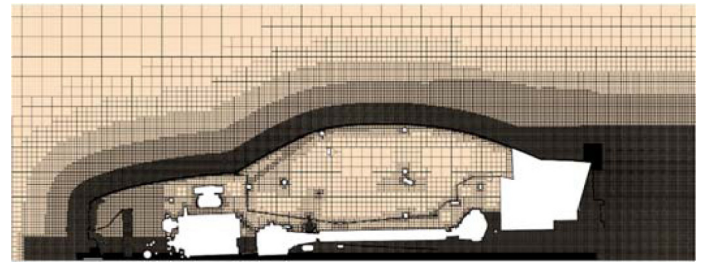


Figure 2. Unstructured cells distribution on center plane.

Wall Treatment

STAR-CCM+ provides three types of wall treatments, the low- y^+ , high- y^+ , and all- y^+ . As an intermediate solution, the all- y^+ wall treatment is selected for this study because it gives reasonable results for either a fine mesh, where the viscous sublayer is well resolved ($y^+ \rightarrow 0$), or a coarse mesh where the near wall region is mostly relying on the wall function ($y^+ \geq 30$). Essentially, the all- y^+ wall treatment increases the flexibility of the grid generation. It should be noted that a wall treatment in STAR-CCM+ is a set of near-wall modeling assumptions, and it is different from the concept of a traditional wall function which refers to only one type of wall treatment. An interested reader is directed to the STAR-CCM+ user manual for additional details.

Initial and Boundary Conditions

A velocity inlet boundary condition is applied to the domain upstream with a constant uniform incoming flow of 58.1 m/s (130 mph), and an inlet turbulence intensity of 0.5%. A tangential velocity of 58.1 m/s, moving along the inlet air, is assigned to the domain floor to mimic the rolling road ground simulation. A pressure outlet boundary condition is used at the domain outlet. The side and top walls are defined as imaginary symmetry plane to eliminate the shear layer growth on those walls and to minimize the blockage influence on the vehicle.

All four wheels, modeled using the local rotation rate, have the same rotation rate of 169.59 rad/s, consistent with the tires traveling at a speed of 58.1 m/s (130 mph). The volume close to the wheels is modeled as moving reference frame (MRF) to simulate the pumping effects of the wheel rotation, which has proven to generate better predictions [22]. An orthotropic porous media is used to model the radiator with actual radiator calorimetry data. The simulations represent a real world driving situation, which is equivalent to running a physical test in a rolling road wind tunnel with a negligible blockage ratio.

Solver and Convergence

The simulations are run with a segregated solver in steady-state. A 2nd order discretization is used for the source and diffusion terms, and a 2nd order upwind scheme for the convection terms of the momentum equations. The algebraic multigrid (AMG) method with a Flex cycle and an under-relaxation factor of 0.7 are used for the velocity solver. The pressure correction equation utilizes a V cycle for the AMG solver with an under-relaxation factor of 0.3. All simulations were

run until the flow reached a full convergence in terms of the force and moment coefficient values. This occurred between 5000 to 6000 iterations, depending on the chosen turbulence model.

RESULTS AND DISCUSSION

Force Coefficients

To protect the confidentiality of the proprietary data, all force coefficients have been normalized by an arbitrary characteristic area. [Table 2](#) summarizes the normalized force coefficients, front to rear lift distribution, and downforce to drag ratio as obtained from wind tunnel test and the CFD analysis. It can be seen in [Table 2](#) that both variants of the $k-\varepsilon$ turbulence model over-predict the vehicle downforce by a similar level of magnitude, but the front to rear distribution is noticeably different. The front downforce prediction from the AKN $k-\varepsilon$ is almost identical to the wind tunnel test data, whilst, the realizable $k-\varepsilon$ model shows a 27-count deficit compared to the wind tunnel data. Although, like the realizable $k-\varepsilon$, the AKN $k-\varepsilon$ also over-predicts the rear downforce, the overall balance from the model is still close to the wind tunnel measurements. The realizable $k-\varepsilon$, on the other hand, has a significantly unbalanced downforce prediction with an unrealistically high clockwise moment towards the rear of the racecar. This severe rear downforce over-prediction from the realizable $k-\varepsilon$ model could be due to the common drawback of the original $k-\varepsilon$ model, which has unrealistic behavior of v_t in the core region of the flow, since the constant $\sigma_k = 1.0$ in both models [23]. By altering σ_k from 1.0 to 1.4, AKN $k-\varepsilon$ model overcomes this defect [15], which leads to a better capturing of the separating and reattaching turbulent flows compared to the other $k-\varepsilon$ models.

Table 2. Summary of force coefficients, front-to-rear balance and downforce/drag ratio

	Wind Tunnel	Realizable $k-\varepsilon$	AKN $k-\varepsilon$	SST $k-\omega$
C_d	0.370	0.355	0.375	0.354
C_l	-0.750	-0.770	-0.773	-0.700
C_{lf}	-0.348	-0.310	-0.347	-0.309
C_{lr}	-0.402	-0.457	-0.424	-0.389
%Front	46.5%	40.3%	44.8%	44.1%
$-C_l/C_d$	2.03	2.17	2.06	1.97

In terms of drag predictions, the AKN $k-\varepsilon$ turbulence model slightly over-predicted the vehicle drag by about 1.3%. The realizable $k-\varepsilon$ model under-predicted the drag value by 4%. Part of this drag difference could be contributed to the larger induced drag component because of the higher downforce prediction from the AKN $k-\varepsilon$ model. To analyze this further,

[Figure 3](#) shows the drag and downforce accumulations along the stream-wise direction from all three turbulent models to identify the disagreement locations. The fact that, for most of the vehicle body, the AKN $k-\varepsilon$ model dominates the downforce accumulation plot provides the proof of more induced drag as a likely cause of the larger drag prediction.

For both drag and downforce accumulations, the force coefficient predictions from all tested turbulence models overlap in the first 1/3 of the racecar. The prediction disagreement starts towards the rear portion of the vehicle body, indicating that the difference between the models mostly came from their differing capabilities in the prediction of shear layer flow and separations. It is a surprise that the force coefficients from the SST $k-\omega$ model have a larger degree of discrepancy with the wind tunnel test data, since this model is known for its proficiency and robustness in boundary layer and free shear layer predictions [17]. It should be noted that the reference wind tunnel in this study is a full-scale closed-jet open return type wind tunnel with contoured walls to mitigate issues related to streamline curvature effects due to lateral and vertical test-section blockages.

The working section of the wind tunnel is shown in [Figure 3](#). The tunnel has the capability of rotating the wheels, and is equipped with a ground plane boundary layer control system on the floor to simulate the actual on-road driving as closely as possible; note that during the time when this racecar was tested, the ground plane simulation was effective for the front half of the car only. However, it is not necessarily a rolling road ground simulation, as the wind tunnel floor in the test section does not move with a tangential velocity while the tires are rotating. With this being considered, the simulations have actual rolling road boundary condition, and the net effect of a rolling road is a faster flow passing through the underbody channel compared to rotating wheels on a static floor, even with the boundary layer removal technology. Faster underbody flow will result in a higher suction force, which will lead to greater downforce. The downforce over-predictions from the $k-\varepsilon$ model support this postulation.



Figure 3. Wind tunnel working section (Courtesy: Aerodyn Technologies)