Journal of Wind Engineering & Industrial Aerodynamics 176 (2018) 98–111

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Turbulence modeling effects on the CFD predictions of flow over a NASCAR Gen 6 racecar

Chen Fu, Mesbah Uddin [*], A. Clay Robinson

North Carolina Motorsports and Automotive Reserach Center, UNC, Charlotte, NC, USA

ARTICLE INFO

Keywords:
Racecar aerodynamics
NASCAR racecar
Road vehicle aerodynamics
External aerodynamics
CFD
RANS
Turbulence modeling

ABSTRACT

Turbulence modeling effects on the RANS CFD simulations of a full-scale NASCAR Gen 6 Cup car are presented in this paper using three commonly used eddy viscosity turbulence models, viz. the realizable and AKN $k-\varepsilon$, and SST $k-\omega$. The simulations were completed using a finite volume code with an unstructured predominantly hexahedral or trimmed mesh of 115 million cells. The prediction disagreements between different turbulence models are highlighted with delta drag and lift force plots along the vehicle model, and generated delta scalar scenes of pressure and velocity fields. The observed differences in the predicted flow-fields were explained in terms of differences in the vortical flow fields educed using the Q-criterion. The simulation results suggest that the turbulence modeling effects are mainly pronounced in the recirculation and separated regions. Compared to the AKN and SST, the realizable $k-\varepsilon$ model showed an inability to properly capture vortex dominated flows, especially towards the rear end of the vehicle, due to under-predictions of the velocity gradients in the separated and wake regions. The SST model predicted vortex shedding similar to the AKN in most part of the vehicle body, except for those on the decklid and off of the spoiler, however, its poor performance in total downforce prediction requires further investigation. Overall, the AKN model appears to be superior to the other two models as the predictions from it best agree with the wind tunnel data in terms of drag, total downforce and front-to-rear vehicle balance.

1. Introduction

Well designed aerodynamic features are crucial for a competitive race vehicle. Aerodynamic effects are just as important to vehicle top speed and handling as available engine power (Singh, 2008). In a 2007 interview, Willy Rampf, the Technical Director of BMW-Sauber Formula F1 Race team at that time, rightfully said, “If you look at all the components that affect the performance of a Formula One car, aerodynamics represent by far the single most important factor” (Grand, 2007), a notion that is supported by the fact that the race teams spend a significant portion of their budget on aerodynamic research and development.

Computational Fluid Dynamics (CFD) simulations and wind tunnel testing are two methods commonly used by the race teams to asses the aerodynamic performance of their cars. Both approaches aim to closely simulate the vehicle running on a racetrack. Wind tunnel tests are often considered as the favored approach due to the use of a real physical model. CFD simulation, however, can provide a more complete understanding of the flow field interacting with the vehicle, and is generally considered as a cost effective companion tool. During the last two decades, CFD capabilities have been greatly improved with the rapid growth of computational power, accompanied by the development of efficient numerical algorithms that can handle complex geometries, and, as well as, the decreasing cost of computing hardware. With appropriate mesh resolution, boundary conditions, and physics models, CFD simulations can now produce results that can rival the accuracy of wind tunnel tests and can provide a significantly more detailed description of the overall flow-field. An aerodynamic evaluation of a race car using CFD simulations includes not only the force and moment data, but also a thorough description of the flow field (Duncan and Golsch, 2004). Additionally, some professional race sanctioning bodies, like the Fédération Internationale de l’Automobile or FIA, the sanctioning body for Formula One (F1), world's highest class of single-seat auto racing, have restrictions on the number of wind-tunnel hours a team can spend in their racecar development. As such, the racing industry has significantly ramped up the use of CFD resources in recent years. However, some professional competitions like F1 even have restrictions on the maximum CPU clock time that a team can use in their aero CFD. Subsequently, the race teams are still vying for the development of reliable CFD methods

---

[*] Corresponding author.
E-mail address: muddin@uncc.edu (M. Uddin).

https://doi.org/10.1016/j.jweia.2018.03.016
Received 22 October 2017; Received in revised form 16 March 2018; Accepted 18 March 2018

with faster turnaround times.

The Reynolds-averaged Navier-Stokes equations (RANS) approach is arguably the most popular and computationally efficient method to model turbulent flows. Although, compared to other turbulence modeling methods, such as the Large Eddy Simulation (LES) and Detached Eddy Simulation (DES), RANS simulations do not have the same level of accuracy and robustness in terms of predicting the turbulence characteristics, research has suggested that RANS modeling is still a very important area due to its low computational expense and the large potential benefits from quick turnaround time (Ashton et al., 2016). For example, The racing community, in general, tends to be very secretive about how they run their business, however, through private communications, the authors understood that in spite of the questionable reliability of the RANS approach, it is still very widely used as the first approximation tool in motorsports because of its computational efficiency and cost effectiveness. Nevertheless, one must be aware that RANS method will have a major drawback in resolving correctly flow inside the wheel-house. It is clearly shown by Krajnović (2016) that this situation warrants an LES; however, for the a NASCAR cup racecar model, this is beyond the capability of current computational resources.

Since the RANS equations only govern the mean flow, the effects of the nonlinearity and the fluctuation from the turbulence must be modeled (Chen, 1997). Turbulence models are classified by the number of additional equations used to model turbulence viscosity transport. Both one-equation (e.g. Spalart-Allmaras) and two-equation turbulence models (viz. $k-\varepsilon$ and $k-\omega$) are widely applied in CFD codes. Each turbulence model has its own advantages and weaknesses in certain areas. For instance, the $k-\varepsilon$ model is good in modeling free shear flows, where the Spalart-Allmaras model is usually considered weaker as it produces more error. Literature on turbulence modeling is abundant, and due diligence will not served by mentioning only a few sources; an interested reader is referred to the book by Wilcox (2006) and to a recently published review paper by Argyropoulos and Markatos (2015).

Most of the turbulence modeling investigations reported in the existing literature involved simplified flows or geometries, such as jet flows (Ghahremanian and Moshfegh, 2014; Heschl et al., 2013; Miltner et al., 2015), channel flows (El-Behery and Hamed, 2011; Gorji et al., 2014), flows past airfoils (Holloway et al., 2008) and flows past bluff bodies (Elkhoury, 2016). Recent turbulence modeling studies in terms of engineering applications includes researches on aerodynamics of trains (Maleki et al., 2017; Munoz-Paniagua et al., 2017; Wang et al., 2017) and civil constructions (Toja-Silva et al., 2015; Mentzoni et al., 2015). In the automotive industry, most of the published validation works are confined to simple automotive models such as the Ahmed body (Guilmineau et al., 2016; Lienhart and Becker, 2003; Guilmineau, 2008; Krastev and Bella, 2011; Wang and Hu, 2012) or DrivAer (Heft et al., 2012) automotive model (Guilmineau, 2014b; a; Ashton et al., 2016); an interested reader is directed to two recent publications by Tunay et al. (2016) and Ashton et al. (2016) for a comprehensive list of notable CFD works on generic/simplified car models. A lesser number of works on the validation of CFD modeling for passenger vehicles can be seen, for example (Jakirlić et al., 2016; Ashton et al., 2016; Buscariolo et al., 2016; Simmonds et al., 2017; Lietz et al., 2017; Guilmineau et al., 2016) to cite but a few. Recently, Mannion et al. (2017), presented CFD and wind-tunnel investigations on tandem cycling aerodynamics with an objective of studying the influence of the CFD grid topology and the turbulence model on the aerodynamic forces on pilot and stoker. However, very few published works address the comparison between the turbulence models directly applied to race vehicles which are drastically different, aerodynamically, as these vehicles include many aero enhancement devices and exhibit much higher drag and lift characteristics, see also Collin et al. (2017). As can be seen in Fig. 1, the aero-enhancement devices attached to this car include a front splitter, a rear spoiler, very low side-skirts, roof-rails and a shark-fin. Additionally, these cars operate at a very low-ground clearance. The drag and lift characteristics of these cars are also very different. A typical passenger vehicle normally produces a small positive lift with a lift to drag ratio of about +0.30 or smaller, whereas, for a better handling, a race-car is required produce down-force (or a negative lift) with a lift to drag ratio of −2.0 or larger.

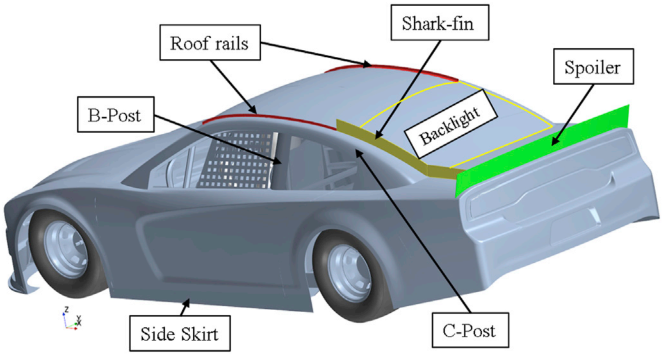

Fig. 1. NASCAR Gen-6 cup racecar model.

In order to strengthen the argument why a race team would still be interested in RANS method in spite of its known limitations, let's consider a recently published study by Collin et al. (2017) who performed Delayed Detached Eddy Simulation (DDES) using an Audi RS5 DTM model in order to investigate the interference of the moving belt geometry on racecar aerodynamics. Their CFD model consists of 90 million cells, and a 512 core cluster took 130 h to complete one simulation. In comparison to that, a 115-million cell RANS simulation of a NASCAR racecar would have taken about 8 h to be completed on the same cluster. Soares et al. (2017) has shown that a suitably designed RANS CFD simulation is simple, cost-effective approach for studying the aerodynamics of a DrivAer car model. As the race-teams in some competitions have the CPU hour restrictions, the successful implementation of a suitable RANS would represent an order of magnitude saving of computational efforts.

The National Association for Stock Car Auto Racing (NASCAR) is the largest sanctioning body for auto racing in North America, and the Monster Energy Cup Series is the premier motorsports series among NASCAR races. To date, the Monster Energy[1] Cup Series cars (Cup cars) have been through an evolution of 6 generations. The Generation 6 car, shortened to Gen-6, is the common name for the car that has been used since 2013. Published numerical studies focusing on the aerodynamics of NASCAR Cup cars are extremely limited and outdated. The latest published CFD work was presented by Singh (2008) using the Gen-5 Car of Tomorrow (COT) model, which has been replaced on track by the Gen-6 cup car with a completely different aero package. In early 2000, General Motors and Daimler Chrysler Corporations also studied the characteristics of the external flow field, using Gen-4 2003 Pontiac NASCAR vehicle (Duncan and Golsch, 2004), and a 2001 Dodge Intrepid R/T (Brzustowicz et al., 2002) racecar. The main purpose of these publications was to understand the car performance with different conditions, such as different designs or different ground simulations, using computational modeling as a tool. Although a very number of papers can be found in the literature, for example TienPhuc et al. (2016), concerning turbulence modeling effects on F1 race-cars, similar in-depth studies involving a stock race-car is absent.

The objective of this paper is to present a comprehensive study of the turbulence modeling effect on CFD predictions of the aerodynamic characteristics of a full-size detailed Gen-6 NASCAR racecar using a commercial CFD package STAR-CCM+ (version 11.04.011). Predictions from various turbulence models are compared against the wind tunnel test data from the Aerodyn Wind Tunnel on drag, lift coefficients and front-to-rear downforce distributions. The computational domain was discretized with hexahedral cells, and the simulations were run at steady state using the RANS approach. Accumulated drag and lift forces along

---

[1] Formerly known as the Sprint Cup Series until 2016.